FIGARO
A general data reduction system
Version 5.6-3
User’s Guide
Abstract

Figaro is a general-purpose data reduction package. The programs it contains can be used to process a wide range of images and spectra.

Figaro can be run in a command-oriented way from the Unix shell or from within ICL. Figaro supports the Starlink NDF data format by default, plus all foreign formats for which conversion utilities exist. These additional formats include Figaro’s old DST format, FITS, and IRAF.

This document offers novice users an easy starting point to using Figaro, and more experienced users a cookbook of a range of problems which can be addressed using the package. It describes version 5.6-3 of Figaro.
# Contents

## 1 Introduction

1.1 What is Figaro? .................................................. 1  
1.2 This document .................................................. 1  
1.3 History .......................................................... 2

## 2 Beginners

2.1 Unix setup ....................................................... 3  
2.2 Starting Figaro .................................................. 4  
  2.2.1 Starting Figaro from the Unix shell ..................... 4  
  2.2.2 Starting Figaro from ICL .............................. 5  
  2.2.3 Starting Figaro from IRAF ............................ 6  
  2.2.4 Inter-operability between Figaro and other packages 7  
2.3 Getting help ................................................... 7  
  2.3.1 Help from within the Unix shell ....................... 7  
  2.3.2 Help from within ICL ................................ 9  
  2.3.3 Help via the WWW ................................... 9  
  2.3.4 Asking for advice .................................. 10  
2.4 Looking at a spectrum ....................................... 10  
2.5 Looking at an image ......................................... 12  
  2.5.1 The ‘image’ command ................................ 12  
  2.5.2 Advanced use of ‘image’ ............................ 13  
  2.5.3 Adding colour ....................................... 16  
2.6 Image display in monochrome ............................... 17  
2.7 Paper copies .................................................. 18  
  2.7.1 Paper copy by cheating ............................. 21  
2.8 Examining data ................................................ 21  
2.9 Doing simple things ........................................ 22

## 3 Advanced users

3.1 Parameters: Controlling commands ......................... 23  
  3.1.1 What is the parameter system? ....................... 23  
  3.1.2 Parameters on the command line ...................... 23  
  3.1.3 Prompted parameters ................................ 24  
  3.1.4 Output parameters .................................. 25  
  3.1.5 Syntax conflicts .................................... 26  
3.2 Data files: Internal details ................................ 27  
  3.2.1 HDS files .............................................. 27  
  3.2.2 NDF format .......................................... 28  
  3.2.3 NDF sections ....................................... 30  
  3.2.4 Foreign formats .................................... 30  
  3.2.5 DST format .......................................... 32  
  3.2.6 Figaro Extension to NDF and DST structures ... 32  
  3.2.7 The Specdre Extension ............................ 33  
  3.2.8 The Contents of the Twodspec Longslit Results Structure . 42  
3.3 Error propagation ............................................. 47  
  3.3.1 How is the error information stored? .............. 47
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.3.2 Getting error information into a file</td>
<td>48</td>
</tr>
<tr>
<td>3.3.3 How are the errors propagated?</td>
<td>48</td>
</tr>
<tr>
<td>3.3.4 Reducing data with error propagation</td>
<td>48</td>
</tr>
<tr>
<td>3.3.5 Flat fielding</td>
<td>48</td>
</tr>
<tr>
<td>3.3.6 Sky subtraction</td>
<td>49</td>
</tr>
<tr>
<td>3.3.7 Optimal extraction</td>
<td>49</td>
</tr>
<tr>
<td>3.3.8 My data are full of cosmic rays—how do I get rid of them?</td>
<td>49</td>
</tr>
<tr>
<td>3.3.9 Zero Variances</td>
<td>49</td>
</tr>
<tr>
<td>3.4 Standard Figaro directories</td>
<td>50</td>
</tr>
<tr>
<td>4 Doing more complex things</td>
<td>51</td>
</tr>
<tr>
<td>4.1 Flats</td>
<td>51</td>
</tr>
<tr>
<td>4.1.1 Image data</td>
<td>51</td>
</tr>
<tr>
<td>4.1.2 Spectral data</td>
<td>52</td>
</tr>
<tr>
<td>4.2 B-Stars</td>
<td>54</td>
</tr>
<tr>
<td>4.3 Filters</td>
<td>54</td>
</tr>
<tr>
<td>4.4 Spiketra</td>
<td>55</td>
</tr>
<tr>
<td>4.5 Flux calibration</td>
<td>56</td>
</tr>
<tr>
<td>4.5.1 Units</td>
<td>56</td>
</tr>
<tr>
<td>4.5.2 Standard files</td>
<td>57</td>
</tr>
<tr>
<td>4.5.3 The final step</td>
<td>58</td>
</tr>
<tr>
<td>4.5.4 Published standards</td>
<td>59</td>
</tr>
<tr>
<td>4.5.5 First step—Turning the table into a spiketrum</td>
<td>59</td>
</tr>
<tr>
<td>4.5.6 Second step—for Oke &amp; Gunn data</td>
<td>60</td>
</tr>
<tr>
<td>4.5.7 Second step—for Filippenko &amp; Greenstein data</td>
<td>60</td>
</tr>
<tr>
<td>4.5.8 An alternative second step—Filippenko &amp; Greenstein data</td>
<td>61</td>
</tr>
<tr>
<td>4.5.9 The final step revisited</td>
<td>62</td>
</tr>
<tr>
<td>4.5.10 AB magnitudes, and a test</td>
<td>62</td>
</tr>
<tr>
<td>4.5.11 Manually setting the exposure time</td>
<td>63</td>
</tr>
<tr>
<td>4.5.12 Summary</td>
<td>63</td>
</tr>
<tr>
<td>4.6 FFT</td>
<td>64</td>
</tr>
<tr>
<td>4.6.1 Complex data structures</td>
<td>64</td>
</tr>
<tr>
<td>4.6.2 Creating a complex data structure</td>
<td>65</td>
</tr>
<tr>
<td>4.6.3 Going smoothly to zero at the ends</td>
<td>65</td>
</tr>
<tr>
<td>4.6.4 Taking the Fourier transform</td>
<td>66</td>
</tr>
<tr>
<td>4.6.5 Extracting the real and imaginary parts</td>
<td>67</td>
</tr>
<tr>
<td>4.6.6 Operations on complex data</td>
<td>67</td>
</tr>
<tr>
<td>4.6.7 Complex filters and data smoothing</td>
<td>67</td>
</tr>
<tr>
<td>4.6.8 Cross correlation, and a sequence something like ‘scross’</td>
<td>69</td>
</tr>
<tr>
<td>4.6.9 Summary of Figaro FFT routines</td>
<td>70</td>
</tr>
<tr>
<td>4.6.10 References</td>
<td>70</td>
</tr>
<tr>
<td>4.7 S-Distortion</td>
<td>71</td>
</tr>
<tr>
<td>4.7.1 Usual sequence</td>
<td>71</td>
</tr>
<tr>
<td>4.7.2 Self-correcting objects</td>
<td>73</td>
</tr>
<tr>
<td>4.7.3 Use of ‘cdist’ parameter ‘rotate’</td>
<td>73</td>
</tr>
<tr>
<td>4.7.4 Making do without an image display</td>
<td>73</td>
</tr>
<tr>
<td>4.8 Wavelength calibration</td>
<td>74</td>
</tr>
<tr>
<td>4.8.1 Discrete and continuous channel numbers</td>
<td>74</td>
</tr>
</tbody>
</table>
4.8.2 Wavelength arrays ............................................ 75
4.8.3 The 'arc' command ........................................... 75
4.8.4 Applying an arc fit to other spectra .................... 76
4.8.5 Linear wavelength scales—scrunching .................. 76
4.8.6 Two-dimensional scrunching ............................... 77
4.8.7 Sequence summary .......................................... 78

4.9 Extinction ....................................................... 79
4.9.1 Reference .................................................... 79

4.10 Arc—A Figaro program for arc wavelength calibration .... 79
4.10.1 Arc line lists ............................................... 80
4.10.2 Other initial prompts ...................................... 80
4.10.3 The line selection process ............................... 81
4.10.4 Subtleties—this bit is worth reading! .................. 82
4.10.5 Moving about the spectrum and other commands .... 82
4.10.6 Fitting and editing—the menu stage .................. 83
4.10.7 The automatic line-finding facility ..................... 84
4.10.8 On the way out of 'arc' .................................... 85
4.10.9 Working with shifted spectra ............................ 85

4.11 Abline—A Figaro program for absorption line analysis .... 86
4.11.1 Introduction ............................................... 86
4.11.2 Running 'abline' .......................................... 86
4.11.3 Line parameter estimation ............................... 91
4.11.4 Conclusion ................................................. 92

4.12 Gauss—A Figaro program for interactive Gaussian fitting ... 92
4.12.1 Introduction ............................................... 92
4.12.2 Running the program ..................................... 92
4.12.3 Useful recipes ............................................. 96
4.12.4 Gauss: Format of data file ............................. 97

4.13 Echelle reduction .............................................. 97
4.13.1 Conventional orientation ................................. 98
4.13.2 Order location ............................................ 98
4.13.3 Order extraction ......................................... 100
4.13.4 Quick-look extraction using a mask .................... 101
4.13.5 Mask creation ........................................... 101
4.13.6 Mask extraction ......................................... 102
4.13.7 Accurate extraction from straightened orders ....... 102
4.13.8 Order straightening ....................................... 103
4.13.9 Order extraction ......................................... 103
4.13.10 Wavelength calibration ................................. 103
4.13.11 Scrunching ............................................ 105
4.13.12 Merging orders ......................................... 105
4.13.13 Summary of the example ............................... 105

Appendices ......................................................... 107

A Classified list of commands ................................. 107
A.1 Data input ................................................... 107
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>A.2 Data output</td>
<td>107</td>
</tr>
<tr>
<td>A.3 Display commands</td>
<td>107</td>
</tr>
<tr>
<td>A.4 Wavelength calibration</td>
<td>107</td>
</tr>
<tr>
<td>A.5 B-star calibration</td>
<td>108</td>
</tr>
<tr>
<td>A.6 Arithmetic operations</td>
<td>108</td>
</tr>
<tr>
<td>A.7 Flat fields</td>
<td>109</td>
</tr>
<tr>
<td>A.8 Data manipulation</td>
<td>109</td>
</tr>
<tr>
<td>A.9 Aperture photometry</td>
<td>110</td>
</tr>
<tr>
<td>A.10 Line analysis</td>
<td>110</td>
</tr>
<tr>
<td>A.11 S-distortion and échelle order straightening</td>
<td>110</td>
</tr>
<tr>
<td>A.12 Fudging data</td>
<td>110</td>
</tr>
<tr>
<td>A.13 Examining data</td>
<td>111</td>
</tr>
<tr>
<td>A.14 Slicing through images and cubes</td>
<td>111</td>
</tr>
<tr>
<td>A.15 Fibre data</td>
<td>111</td>
</tr>
<tr>
<td>A.16 Flux calibration</td>
<td>111</td>
</tr>
<tr>
<td>A.17 Extinction</td>
<td>112</td>
</tr>
<tr>
<td>A.18 Complex data and FFTs</td>
<td>112</td>
</tr>
<tr>
<td>A.19 Infra-red data</td>
<td>112</td>
</tr>
<tr>
<td>A.20 Échelle data</td>
<td>113</td>
</tr>
<tr>
<td>A.21 Spectroscopy Data Reduction (Specdre)</td>
<td>113</td>
</tr>
<tr>
<td>A.22 Spectroscopy Data Reduction (Twodspec)</td>
<td>114</td>
</tr>
<tr>
<td>A.23 Miscellany</td>
<td>115</td>
</tr>
<tr>
<td>B Specdre</td>
<td>116</td>
</tr>
<tr>
<td>B.1 Introduction</td>
<td>116</td>
</tr>
<tr>
<td>B.2 Specdre’s use of parameters</td>
<td>117</td>
</tr>
<tr>
<td>B.3 Graphics</td>
<td>117</td>
</tr>
<tr>
<td>B.4 Slicing data sets</td>
<td>118</td>
</tr>
<tr>
<td>B.5 Cube manipulation</td>
<td>119</td>
</tr>
<tr>
<td>B.6 Spectral fits</td>
<td>120</td>
</tr>
<tr>
<td>B.7 Arc spectrum axis calibration</td>
<td>122</td>
</tr>
<tr>
<td>B.8 References</td>
<td>124</td>
</tr>
<tr>
<td>C Twodspec</td>
<td>125</td>
</tr>
<tr>
<td>C.1 Introduction</td>
<td>125</td>
</tr>
<tr>
<td>C.2 Line Profile Analysis</td>
<td>125</td>
</tr>
<tr>
<td>C.2.1 Introduction</td>
<td>125</td>
</tr>
<tr>
<td>C.2.2 LONGSLIT</td>
<td>126</td>
</tr>
<tr>
<td>C.2.3 FIBDISP—Analyise 3-d Data (spectral)</td>
<td>149</td>
</tr>
<tr>
<td>C.3 ARC2D—Wavelength Calibration</td>
<td>152</td>
</tr>
<tr>
<td>C.3.1 Introduction</td>
<td>152</td>
</tr>
<tr>
<td>C.3.2 Continuity Correction</td>
<td>154</td>
</tr>
<tr>
<td>C.3.3 Evaluating the Dispersion Relation</td>
<td>155</td>
</tr>
<tr>
<td>C.3.4 Summary of Parameters</td>
<td>156</td>
</tr>
<tr>
<td>C.4 COMB and ARCSDI-Correction for Geometrical Distortions</td>
<td>157</td>
</tr>
<tr>
<td>C.4.1 COMB—Correction for S-Distortion</td>
<td>157</td>
</tr>
<tr>
<td>C.4.2 ARCSDI—Correction for Line Curvature</td>
<td>160</td>
</tr>
<tr>
<td>C.5 Display Programs</td>
<td>161</td>
</tr>
</tbody>
</table>
C.5.1 ISCAN—Plot Spectra Extracted From an Image .............................................. 161
C.5.2 HIMAGE—Greyscale Display ........................................................................... 162
C.5.3 CSCAN—Plot Array of Line Profiles from Data “Cube” ................................. 162
C.6 Removing Continua .......................................................................................... 163
C.6.1 FITCONT—Fit Polynomial to Continuum ......................................................... 163
C.6.2 CSUB—Subtract Continuum ............................................................................. 163
C.6.3 CADD—Add Back Continuum Previously Subtracted ................................. 163
C.7 Getting 3-D Data into the Correct Form for FIBDISP or LONGSLIT .............. 163
C.7.1 FIBSEP—Extract Data from Individual Fibres ............................................... 164
C.7.2 FIB2CUBE—Convert Fibre Data to “Cube” .................................................... 164
C.8 Miscellaneous Programs ................................................................................. 165
C.8.1 CUBE2LONG—Extract a Spectrum from a Spectral “Cube” ....................... 165
C.8.2 VIG—Correct For Vignetting ......................................................................... 165
C.9 Note on Menus .................................................................................................. 166
C.10 The Defaults File .............................................................................................. 167
C.11 References ........................................................................................................ 169

D Applications in detail 169
D.1 ABCONV-Convert spectrum from Janskys into AB magnitudes ...................... 169
D.2 ABLINE-Interactive absorption line analysis .................................................... 170
D.3 ADJOIN-Append two spectra (strictly a merge by wavelength value) ......... 172
D.4 ALASIN-Read a spectrum in ALAS (Abs. Line Analysis System) format . . 173
D.5 ALASOUT-Output a spectrum in ALAS (Abs. Line Analysis System) format . 174
D.6 APERTURE-Do simple minded aperture photometry on a series of frames . . 175
D.7 ARC-Interactive manual arc line identification ................................................ 176
D.8 ARC2D-Calibrates distortions in 2D arc line data ............................................. 179
D.9 ARCDISP-Fit polynomial dispersion curve ....................................................... 180
D.10 ARCGENDB-Convert list of laboratory values to feature data base............. 182
D.11 ARCIDENT-Auto-identify located features ..................................................... 184
D.12 ARCLOCAT-Locate line features in a set of spectra ....................................... 187
D.13 ARCSDI-Corrects for arc line curvature ......................................................... 190
D.14 ASCIN-Read a 1-D or N-D data set from an ASCII table ................................ 191
D.15 ASCOUT-Write an NDF to an ASCII table .................................................... 193
D.16 BBODY-Calculate a black body spectrum ...................................................... 194
D.17 BCLEAN-Automatic removal of bad lines and cosmic rays from CCD data . 195
D.18 BFFT-Takes the reverse FFT of a complex data structure ............................. 198
D.19 BSMULT-Atmospheric band removal using a B star calibration spectrum .... 199
D.20 CADD-Add back fitted continuum ................................................................. 200
D.21 CALDIV-Generate calibration spectrum from continuum standard spectra .... 200
D.22 CCURLIN-Applies a linearity correction to AAO CCD data ......................... 201
D.23 CCUR-After SPLOT, uses graphics cursor to indicate data values ............... 202
D.24 CDIST - S-distortion correction using SDIST results ...................................... 203
D.25 CENTERS-Generate file of object centroids from ICUR, IGCUR or IMPOS output 205
D.26 CFIT-Generate a spectrum using the cursor ..................................................... 207
D.27 CHANGED-Indicate fits invalidated due to “cleaning” of an image ............ 207
D.28 CLEAN-Interactive patching of bad lines, bad pixels in an image ............. 208
D.29 CLIP-Clip data above and below a pair of threshold values ....................... 209
D.30 CMPLX2I-Extracts the imaginary part of a complex data structure ............. 209
D.79 FIGINFO-Describes the contents of a Figaro data file .................................................. 266
D.80 FIGS321-Processes a FIGS data cube down to a single spectrum ...................................... 268
D.81 FIGS322-Processes a FIGS data cube down to an image .................................................... 269
D.82 FIGS322-Process a FIGS image-mode hypercube down to an image ........................................ 271
D.83 FIGS323-Process a FIGS image-mode hypercube down to a cube ........................................... 272
D.84 FIGS424-Sort a FIGS image-mode hypercube into wavelength order ...................................... 273
D.85 FIGSEE-Generate a seeing ripple spectrum from a FIGS spectrum ......................................... 274
D.86 FIGSFLUX-Flux calibrates a FIGS spectrum ........................................................................... 274
D.87 FILLCUBE-Copy one NDF into part of another ...................................................................... 275
D.88 FINDSP-Locate fibre spectra in an image ................................................................................. 276
D.89 FITBB-Fit diluted Planck curves to a spectrum ....................................................................... 277
D.90 FITCONT-Fits a Chebyshev polynomial to the continuum for 2D data ...................................... 282
D.91 FITGAUSS-Fit Gauss profiles to a spectrum ............................................................................. 282
D.92 FITPOLY-Fit a polynomial to a spectrum ................................................................................. 286
D.93 FITSET-Set the value of a FITS keyword ................................................................................. 289
D.94 FITSKEYS-List the FITS keywords in a data file .................................................................. 290
D.95 FITTRI-Fit triangular profiles to a spectrum ............................................................................. 291
D.96 FLAG2QUAL-Converts 'flagged' values to produce a quality array ....................................... 295
D.97 FLAIRCOMP-Compresses a FLAIR frame to give a weight vector ..................................... 296
D.98 FLAIREXT-Optimally extracts spectra from a FLAIR NDF to form a new NDF ..................... 297
D.99 FLCONV-Convert a spectrum in Janskys into one in erg/s/cm**2/Angstrom .................. 297
D.100 FOTO-Perform aperture photometry given CENTERS output .......................................... 298
D.101 FSCRUNCH-Rebin data with a disjoint wavelength coverage to a linear one ..................... 299
D.102 FCONV-General unit conversion for spectra ................................................................. 302
D.103 GAUSS-Interactive fit of Gaussians to emission or absorption lines .................................. 302
D.104 GOODVAR-Replace negative, zero and bad variance values ........................................... 306
D.105 GROW-Copy an N-dimensional cube into part of an (N+M)-dimensional one .................... 306
D.106 GROWX-Performs reverse function to that of EXTRACT .................................................. 308
D.107 GROWXY-Copies an image into contiguous XY planes of a cube ..................................... 309
D.108 GROWYT-Copies an image into contiguous YT planes of a cube ..................................... 310
D.109 ICADD-Adds a constant to an image .................................................................................... 324
D.110 ICALOG-Takes the antilog of an image .................................................................................. 321
D.111 ICARC-Given fit to single spectrum, fit all spectra in a 2-D arc ............................................. 321
D.112 ICADD-Adds a constant to an image ..................................................................................... 322
D.113 ICDIV-Divides an image by a constant ................................................................................... 323
D.114 ICMULT-Multiplies an image by a constant .......................................................................... 326
D.115 ICNT-Produces a contour map of an image ......................................................................... 326
D.116 ICOR16-Corrects 16 bit data from signed to unsigned range .............................................. 329
D.127 ICSET-Set a selected region of an image to a constant value .................................................. 329
D.128 ICSUB-Subtracts a constant from an image .............................................................................. 331
D.129 ICUR-Inspect an image with cursor .......................................................................................... 331
D.130 IDEV-Set the device for image display ....................................................................................... 332
D.131 IDIFF-Takes the ‘differential’ of an image .................................................................................. 333
D.132 IDIV-Divides two images (or two spectra) ................................................................................... 333
D.133 GCONV-Convolve an image with a specified filter ..................................................................... 334
D.134 GCUR-Use cursor to show X, Y and data values ..................................................................... 335
D.135 GREY-Produces a grey-scale plot of an image ........................................................................... 336
D.136 LIST-List the data in an image (or spectrum) ............................................................................ 337
D.137 ILOG-Takes the logarithm of an image ....................................................................................... 338
D.138 IMAGE-Display an image ........................................................................................................... 338
D.139 IMPS-Read positions from a file into environment variables ......................................................... 339
D.140 IMULT-Multiplies two images (or two spectra) .......................................................................... 340
D.141 INTERP-Interpolates points of a ‘spiketrum’ to form a spectrum ................................................... 341
D.142 PLOTS-Plots successive cross-sections of an image, several to a page ......................................... 342
D.143 IPWR-Raises an image to a specified power .............................................................................. 343
D.144 IRCONV-Converts data in Janskys to W/m**2/um ............................................................ 344
D.145 IRREVX-Reverse an image (or spectrum) in the X-direction ....................................................... 345
D.146 IRREVY-Reverse an image in the Y-direction .......................................................................... 345
D.147 IRFLAT-Generates a ripple spectrum from an IR spectrum ......................................................... 346
D.148 IRFLUX-Flux calibrates an IR spectrum using a black-body model ......................................... 347
D.149 IROT90-Rotates an image through 90 degrees ......................................................................... 348
D.150 ISCAN-Displays cuts through an IMAGE .................................................................................. 348
D.151 ISCRUNCH-Rebin an image to linear wavelength scale given IARC results ................................ 349
D.152 ISCRUNI-Like ISCRUNCH, but interpolates between two IARC result sets ................................. 351
D.153 ISEDIT- Allows interactive editing of a 1-D or 2-D spectrum ....................................................... 354
D.154 ISHIFT- Applies a linear X and a linear Y shift to an image ......................................................... 354
D.155 ISMEOUH-2-D smooth of image using 9-point smoothing algorithm ............................................. 356
D.156 ISPLOT-Plots successive cross-sections through an image ......................................................... 357
D.157 ISTAT-Provides some statistics about an image (max, min etc.) .................................................. 359
D.158 ISTRETCH-Stretches and shifts an image in X and Y ................................................................ 361
D.159 ISUB-Subtracts two images (or two spectra) ............................................................................ 362
D.160 ISUBSET-Produces a subset of an image ..................................................................................... 363
D.161 ISUPER-Produces a superset of an image .................................................................................... 364
D.162 ISXADD-Adds a spectrum to each X direction X-section of an image ........................................ 365
D.163 ISXDIV-Divides a spectrum into each X direction x-section of an image ..................................... 366
D.164 ISXMUL-Multiplies each X direction image x-sect by a spectrum ................................................ 366
D.165 ISXSUB-Subtracts each X direction image x-sect from a spectrum ............................................. 367
D.166 ISYADD-Adds a spectrum to each Y direction x-section of an image ......................................... 368
D.167 ISYDIV-Divides a spectrum into each Y direction x-section of an image .................................... 369
D.168 SYMUL-Multiplies each Y direction image x-sect by a spectrum ................................................ 370
D.169 SYSUB-Subtracts each Y direction image x-sect from a spectrum ............................................. 371
D.170 XSMOOTH-Smooth in X-direction by gaussian convolution ...................................................... 372
D.171 LINTERP-Interpolates between spiketrum points to form a spectrum ........................................ 373
D.172 LONGLIT-Fits 2D longslit arrays and plots results ...................................................................... 373
D.173 LSPLOT-Hardcopy spectrum plot of specified size (up to 3 metres) ............................................ 376
D.174 LXSET-Set X array of spectrum/image to specified range ............................................................. 377
D.175 LYSET—Set Y array of spectrum/image to specified range .......................... 379
D.176 MASK—Generate a mask spectrum given a spectrum and a mask table ........... 380
D.177 MASKEXT—Extracts echelle orders using a mask created by ECHMASK . .... 380
D.178 MCFIT—Fit a continuum to a spectrum, given a mask spectrum .............. 381
D.179 MEDFILT—Applies a median filter to an image ...................................... 382
D.180 MEDFLTR—Applies a rectangular median filter to an image .................. 383
D.181 MEDSKY—Take the median of a number of images .............................. 384
D.182 MOMENTS—Calculate moments of spectra in a cube ........................... 386
D.183 MOVIE—Browse through slices of a cube ......................................... 387
D.184 MSPLOT—Plots a long spectrum as a series of separate plots ............... 390
D.185 NCSET—Set a region of a spectrum to a constant .............................. 392
D.186 OFFDIST—Applies an offset to an SDIST fit .................................... 392
D.187 OPTEXTRACT—Extracts a long-slit spectrum using Horne’s algorithm .... 394
D.188 OVERPF—Overlays a FINDSP fit on another image ............................. 395
D.189 PEAK—Determines position of highest peak in a spectrum .................... 396
D.190 POLEXT—Extract fibre spectra from an image after a FINDSP analysis .... 396
D.191 POLYSKY—Fits and subtracts sky from a long-slit spectrum ................. 397
D.192 PROFILE—Profiles a long-slit spectrum for use by OPTEXTRACT ......... 398
D.193 Q2BAD—Converts a datafile’s quality into bad values ......................... 399
D.194 QUAL2FLAG—Converts a quality array into ‘flagged’ values ................. 400
D.195 R2CMPLX—Creates a complex data structure from a real data array ....... 400
D.196 RCXS2—Reads UKIRT CGS2 spectrum (also UKT9 and UKT6 CVF) ....... 401
D.197 RDFITS—Read file in AAO de facto ‘Disk FITS’ format .......................... 402
D.198 RDIPSO—Read file in DIPSO/IUEDR/SPECTRUM format ...................... 403
D.199 REMBAD—Removes pixels that have been flagged as bad from data ....... 404
D.200 RENOBJ—Change the name or location of an object within an HDS file .... 404
D.201 RESAMP—Re-sample and average several spectra (was Specdre RESAMPLE) . 405
D.202 RESAMPLE—Rebin an image to different dimensions and/or orientation ... 408
D.203 RESCALE—Rescale using user-defined upper and lower limits ............... 410
D.204 RETYPE—Changes the type of the main data array in a file ................. 410
D.205 ROTX—Rotate data along the X-axis ........................................... 411
D.206 SCLEAN—Interactive patching of images, especially SCUBA data ......... 412
D.207 SCNSKY—Calculates a sky spectrum for a scanned CCD image ........... 415
D.208 CROSS—Cross-correlate two spectra and get relative shift ................. 417
D.209 CRUNCH—Rebin a spectrum to a linear wavelength range ................... 418
D.210 DISIST—Analyse an image containing spectra for S-distortion ............ 421
D.211 SETOBJ—Assign value to an HDS primitive .................................. 423
D.212 SFIT—Fit a polynomial to a spectrum .......................................... 423
D.213 SKYLINER—Removes a sky spectrum normalised the 5577 [OI] line ........ 424
D.214 SLICE—Takes a slice with arbitrary end points through an image ......... 424
D.215 SOFIT—Sets the device/type for terminal graphics ............................ 426
D.216 SPECCONT—Contour a two-dimensional cut .................................... 426
D.217 SPECGRID—Plot spectra on position grid ...................................... 430
D.218 SPECPLOT—Plot a spectrum .................................................... 436
D.219 SPEFLUX—Applies a flux calibration spectrum to an observed spectrum .. 441
D.220 SPED—Interactive spiketrum editor .......................................... 442
D.221 SPFIT—Fits a global polynomial to a spiketrum to form a spectrum .... 443
D.222 SPLOT—Plots a spectrum .................................................... 444
D.223 SQRTERR-Generates an error array as Error = Square Root of (Data/Const)  
D.224 SUBSET-Take a subset of a data set  
D.225 TABLE-List contents of a SPICA memory file  
D.226 TPPEX-Modify individual pixel values with cursor  
D.227 RIMFILE-Creates a copy of an HDS file without unused space  
D.228 WACHEL-Air to vacuum, and/or recession velocity wavelength conversion  
D.229 VIG-Corrects a 2D array for vignetting  
D.230 WDFITS-Writes an image out in the AAO de facto ‘Disk FITS’ format  
D.231 WDIPS0-Writes a file in DIPS0/UEDR/SPECTRUM format  
D.232 XCADD-Adds a constant to the X data in a file  
D.233 XCDIV-Divides the X data in a file by a constant  
D.234 XCMULT-Multiplies the X data in a file by a constant  
D.235 XCOPY-Copy X-info (e.g. wavelengths) into a spectrum  
D.236 XCSUB-Subtracts a constant from the X data in a file  
D.237 XCUR-Uses cursor to delimit part of a spectrum  
D.238 XTPLANE-Adds XT planes of a data cube to form an image  
D.239 XTRACT-Average an N-dimensional cube into an (N-M)-dimensional one.  
D.240 YCADD-Adds a constant to the Y data in a file  
D.241 YCDIV-Divides the Y data in a file by a constant  
D.242 YCMULT-Multiplies the Y data in a file by a constant  
D.243 YCSUB-Subtracts a constant from the Y data in a file  
D.244 YTPLANE-Adds contiguous YT planes of a data cube to form an image  
D.245 YSTRACT-Adds contiguous columns of an image to form a spectrum  
D.246 YTPLOANE-Adds contiguous YT planes of a data cube to form an image  

E  Flux standards  
E.1 Spectrophotometric flux standards converted from SPICA format  
E.2 HST and new Oke spectrophotometric standard star flux tables  
   E.2.1 Oke standards  
   E.2.2 HST standards  

F  Programming  
F.1 Writing your own Figaro-style commands  

G  Release notes  
G.1 Changes from VAX/VMS Figaro 3.0-9 to Portable Figaro 5.1  
   G.1.1 Documentation  
   G.1.2 General  
   G.1.3 Ported applications  
   G.1.4 New applications  
   G.1.5 Un-ported applications  
   G.1.6 Parameters and variables: (F)PAR  
   G.1.7 Data access routines: FDA  
   G.1.8 Old data access routines: DSA and DTA  
   G.1.9 High-level subroutines: FIG  
   G.1.10 The J-library: JTY  
   G.1.11 Low-level subroutines: GEN  

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>G.1.12</td>
<td>Other libraries</td>
<td>488</td>
</tr>
<tr>
<td>G.2</td>
<td>Figaro 3.1.1 release</td>
<td>488</td>
</tr>
<tr>
<td>G.3</td>
<td>Figaro 3.2 release</td>
<td>489</td>
</tr>
<tr>
<td>G.3.1</td>
<td>On-line help topic “News 3.2”</td>
<td>490</td>
</tr>
<tr>
<td>G.4</td>
<td>Figaro 3.2-3 release</td>
<td>491</td>
</tr>
<tr>
<td>G.5</td>
<td>Figaro 5.0 release</td>
<td>491</td>
</tr>
<tr>
<td>G.5.1</td>
<td>Figaro 5.0-1 news item</td>
<td>496</td>
</tr>
<tr>
<td>G.6</td>
<td>Figaro 5.1-3 release</td>
<td>496</td>
</tr>
<tr>
<td>G.7</td>
<td>Figaro 5.1-4 release</td>
<td>499</td>
</tr>
<tr>
<td>G.7.1</td>
<td>Release notes</td>
<td>500</td>
</tr>
<tr>
<td>G.7.2</td>
<td>Figaro tips</td>
<td>501</td>
</tr>
<tr>
<td>G.7.3</td>
<td>New Figaro features</td>
<td>503</td>
</tr>
<tr>
<td>G.8</td>
<td>Figaro 5.1-6 release</td>
<td>505</td>
</tr>
<tr>
<td>G.8.1</td>
<td>Release notes</td>
<td>505</td>
</tr>
<tr>
<td>G.8.2</td>
<td>Figaro tips</td>
<td>506</td>
</tr>
<tr>
<td>G.8.3</td>
<td>Bug fixes</td>
<td>507</td>
</tr>
<tr>
<td>G.9</td>
<td>Figaro 5.2-0 release</td>
<td>507</td>
</tr>
<tr>
<td>G.9.1</td>
<td>Release notes</td>
<td>507</td>
</tr>
<tr>
<td>G.9.2</td>
<td>Error propagation</td>
<td>508</td>
</tr>
<tr>
<td>G.9.3</td>
<td>Bug fixes</td>
<td>508</td>
</tr>
<tr>
<td>G.10</td>
<td>Figaro 5.3-0 release</td>
<td>509</td>
</tr>
<tr>
<td>G.10.1</td>
<td>Incorporation of SPECDRE</td>
<td>509</td>
</tr>
<tr>
<td>G.10.2</td>
<td>Incorporation of IRAFFIG</td>
<td>509</td>
</tr>
<tr>
<td>G.10.3</td>
<td>Minor enhancements</td>
<td>509</td>
</tr>
<tr>
<td>G.11</td>
<td>Figaro 5.4-0 release</td>
<td>510</td>
</tr>
<tr>
<td>G.12</td>
<td>Figaro 5.5-0 release</td>
<td>510</td>
</tr>
<tr>
<td>G.13</td>
<td>Figaro 5.5-1 release</td>
<td>510</td>
</tr>
<tr>
<td>G.14</td>
<td>Figaro 5.5-2 release</td>
<td>510</td>
</tr>
<tr>
<td>G.15</td>
<td>Figaro 5.6-0 release</td>
<td>510</td>
</tr>
<tr>
<td>G.15.1</td>
<td>Error Propagation</td>
<td>511</td>
</tr>
<tr>
<td>G.15.2</td>
<td>Bug Fixes and Minor Enhancements</td>
<td>511</td>
</tr>
<tr>
<td>G.15.3</td>
<td>New Application</td>
<td>511</td>
</tr>
<tr>
<td>G.16</td>
<td>Figaro 5.6-1 release</td>
<td>511</td>
</tr>
<tr>
<td>G.16.1</td>
<td>Bug Fixes and Minor Enhancements</td>
<td>511</td>
</tr>
<tr>
<td>G.16.2</td>
<td>Flux calibration standard stars</td>
<td>511</td>
</tr>
<tr>
<td>G.17</td>
<td>Figaro 5.6-2 release</td>
<td>512</td>
</tr>
<tr>
<td>G.18</td>
<td>Figaro 5.6-3 release</td>
<td>512</td>
</tr>
<tr>
<td>G.18.1</td>
<td>Bug Fixes and Minor Enhancements</td>
<td>512</td>
</tr>
<tr>
<td>G.18.2</td>
<td>Flux Calibration Files</td>
<td>512</td>
</tr>
</tbody>
</table>

**H**  Keyword index | 513
List of Figures

1. Example GWM spectrum .................................................. 11
2. A spectrum in a GWM window ........................................... 12
3. An example of an image displayed in a GWM window ........... 14
4. Advanced use of the IMAGE command .............................. 15
5. Adding color to an image .................................................. 16
6. Using contours ............................................................... 17
7. Monochrome image ......................................................... 18
8. Postscript output ............................................................ 20
9. Spectral data layout ......................................................... 52
10. Conventional orientation for échelle reduction ................... 98
11. Flow chart for reorienting CCD ...................................... 99
12. Flow chart for locating échelle orders .............................. 100
13. Flow chart for extracting orders in échelle spectra ............ 101
14. Wavelength calibrating an échelle spectrum ..................... 104
1 Introduction

1.1 What is Figaro?

Figaro is a general-purpose data reduction package. Many people find it ideal for reducing spectroscopic data, but it also has powerful image and data cube manipulation facilities. The package was developed by Keith Shortridge, originally at Caltech and later at the Anglo-Australian Observatory (AAO). The version described here, Portable Figaro, is released and supported by Starlink.

Portable Figaro can be run in a command-oriented way from the Unix shell (usually a shell similar to the C shell). There is a second command line interface, ICL, from which Figaro can be run. ICL has some advantages over Unix shells, because it has floating point variables and can communicate with the ADAM parameter system that underlies Portable Figaro. ICL also avoids a lot of quirks in command syntax due to Unix shell meta-characters.

By default Figaro accesses data files in Starlink’s NDF data format. However, it can also access a number of different data formats, including Figaro’s old DST format, FITS and IRAF. Most Starlink packages can access the same range of data formats, and consequently Figaro can inter-operate with them. Earlier versions of Figaro supported a different and more restricted range of data formats. The current facilities are available in Portable Figaro version 5.1 and higher.

From version 5.3, Portable Figaro incorporates the Specdre package, which is used for spectroscopy data reduction and analysis. It includes cube manipulation, arc line axis (wavelength) calibration, re-sampling, and spectral fits.

From version 5.5, Portable Figaro incorporates the Twodspec package, which is used for longslit spectroscopy data reduction and analysis. It provides tools for calibration/correction of data and fitting of 2-D longslit arrays, either automatically or as an interactive process. These tools also offer a number of options for generating hard copy output of the resulting fits.

1.2 This document

This text largely dates from a complete re-write of the user documentation of Portable Figaro to accompany version 5.2 of Figaro. The style of this document is now more that of a cook book. It tries to offer novice users an easy entry into using Figaro whilst providing sufficiently detailed information for experienced users.

The main body of the document contains only a few cross-references between sections and references to other documents. References to external documents are provided as pointers to further information and need not be consulted to follow the examples in this manual. There is a keyword index, which should give you a quick way to find the information needed. The index contains references into this, and to other documents.

Section 2 is the starting point for new Figaro users; it outlines the basics, getting started, and gives some simple examples.

Section 3 contains information which can be applied in many Figaro reductions. There are details of parameter usage, the underlying data system, and access to foreign file formats. There is also an introduction to error-propagating tasks in Figaro.
Section 4 gives cookbook style details of a range of problems which can be addressed using Figaro: treatment of flat fields; B-star calibration; handling filters; flux calibration; fast Fourier transforms; removal of S-distortion; wavelength calibration; correction for extinction; and Gaussian fitting.

In Appendix A there is a classified list of Figaro commands.

In Appendix B there is a description of the features available with Specdre.

In Appendix C there is a manual for the Twodspec commands.

1.3 History

This document developed from the documentation for VAX/VMS Figaro, written by Keith Shortridge. It included some separate documents by J.G. Robertson, Jeremy Walsh and William Lupton. Appendices B and C are adapted from the Specdre and Twodspec manuals respectively.

Portable Figaro has evolved from Figaro 3.0 as distributed for VAX/VMS by Starlink, with some minor contributions from Sun Figaro 2.4.5 as produced by Samuel Southard (Caltech).

Figaro 3.0 was a release from AAO, but it had been adapted somewhat by Starlink (Figaro 3.0-3 and 3.0-5). Also some modifications had been made under the term ‘National Figaro’ (3.0-1 through 3.0-6). Porting Figaro 3.0 to Unix was a joint effort by the ‘Figaro Port Group’ Michael Ashley and Brad Carter (UNSW), Stephen Meatheringham MSSSO), Horst Meyerdierks (UoE, Starlink), and Keith Shortridge (AAO).

Specdre was written by Horst Meyerdierks (UoE, Starlink). It was incorporated into Figaro by Anne Charles (RAL, Starlink).

Twodspec was written by Tim Wilkins (Manchester and Cambridge) and Dave Axon (Manchester) with subsequent porting by John Palmer (Manchester) and Anthony Holloway (Manchester).
2 Beginners

2.1 Unix setup

In order to run any Starlink software, your Unix startup must be adapted appropriately. If you work at a Starlink site, chances are that your site manager has already taken care of this.

It is necessary that you use a Unix shell similar to the C shell. A Unix shell is a command line interpreter between you and the Unix operating system. When you log into the Unix system one or another shell will be started for you, namely the one that is your ‘login shell’. The most basic Unix shell is the Bourne shell ‘sh’, but the most common interactive shell is the C shell ‘csh’. Some system administrators may try to educate you to use more fashionable shells like ‘korn’ or ‘bash’.

The Starlink startup scripts all assume that you use the C shell, but a similar, enhanced shell is ‘tcsh’, and this is what probably most users of Starlink software have as their login shell.

When you login, the login shell will execute two scripts in your home directory. One is ‘.cshrc’, or Tc shell users may have a similar file ‘.tcshrc’ instead. The second startup script is ‘.login’.

What you have to do to make Starlink software work, is to include one command in each of these two files. In ‘.cshrc’ or ‘.tcshrc’ you insert the line

```
source /star/etc/cshrc
```

and in ‘.login’ you insert the equivalent

```
source /star/etc/login
```

Figaro and other Starlink packages use a common mechanism for providing input values to, and obtaining output values from, individual applications. This so-called ‘parameter system’ allows values to be passed between related applications, saves un-necessary typing and assists the applications in suggesting sensible default values. The details are not germane here, but files containing the parameter values passed between applications need to be kept somewhere. By default these files reside in a subdirectory of your home directory called (for historical reasons) ‘adam’. This directory is created automatically the first time that you run a Figaro (or other Starlink) application. However, you can create it manually if you prefer. You can choose to use a different directory by adding a command to your ‘.login’ script. Say, you want to use directory ‘parameters’ under your home directory, then add

```
% setenv ADAM_USER $HOME/parameters
```

Similarly, Figaro, and other Starlink packages, use the ‘Applications Graphics Interface’ (AGI) to pass graphical information between related applications, allowing them to inter-operate smoothly. The associated files are created in your home directory by default. However, again you can specify their location by adding a line to your ‘.login’ script. For example

```
setenv AGI_USER $HOME/graphics
```

It is a good idea to delete these files occasionally, since they grow ever bigger and mainly contain old, useless information. The files are called ‘agi_<host>.sdf’ and there is one for each machine you have used for Starlink graphics.
2.2 Starting Figaro

There are three different ways of running Figaro:

- from the Unix shell,
- from ICL (Starlink’s Interactive Command Language),
- from within IRAF.

These alternatives are described individually below. Each has its advantages. If you plan to make only fairly straightforward use of Figaro then it may be simplest to run it from the Unix shell.

2.2.1 Starting Figaro from the Unix shell

To start Figaro from the Unix shell simply type:

```
% figaro
```

and it responds with a message similar to this:

```
---------- Initialising for Figaro ----------
General data reduction
Version 5.6-1 19 September 2002

Type "fighelp figaro" for help
or "fighelp news" for news on changes

Type "showme sun86" to browse HTML documentation

Use "abbrev" and "noabbrev" to turn parameter name abbreviation on and off.
```

Once you have given this Figaro startup command you have about 200 new commands at your disposal, the Figaro commands (or Figaro applications; In Figaro, and other Starlink packages, commands are often referred to as ‘applications’ because each command corresponds to an application program which performs some data reduction task). Each Figaro application can be invoked by two different commands. One is simply a name describing the application, the other the same name prefixed by ‘fig_’. E.g. the commands ‘istat’ and ‘fig_istat’ invoke the same application. The reason for this apparent duplication is that some common command names can also be used in other packages. The ‘fig_’ prefix allows the Figaro commands to be specified unambiguously.

The Unix shell has some disadvantages in running Figaro commands. The most annoying feature is that so-called meta-characters like () [] '" have special meaning to the Unix shell. If you need these characters to pass information in the command line to the Figaro command, then you have to take extra steps to make sure the information makes it through the interpretation by the Unix shell. See Section 3.1.5 for further details.

Of less importance is that the Unix shell does not have floating point variables and arithmetics, and that it has no knowledge of the parameter system used by Figaro commands. Such features are rarely needed; once you get used to handling the shell meta-characters, you should be all right with the Unix shell.
2.2.2 Starting Figaro from ICL

Figaro can also be run from Starlink’s Interactive Command Language (ICL). ICL is a command line interface which has intimate knowledge of the parameter system and of the command line syntax of Figaro commands. When your procedures get really tricky, ICL is probably a better command line interface than the Unix shell. But even in every-day use, ICL saves you masking all those Unix shell meta-characters.

You start ICL with the shell command

% icl

Unlike the Figaro startup command, which only defines new commands for the Unix shell, this starts a new process, a new shell if you like. It is thus no big surprise that the prompt string changes to ‘ICL>’.

From this shell you start Figaro with

ICL> figaro

which will respond with the familiar message

---------- Initialising for Figaro ----------
General data reduction
Version 5.6-1 19 September 2002
Type "help figaro" for help

Type "showme sun86" to browse HTML documentation

Use "abbrev" and "noabbrev" to turn parameter name abbreviation on and off.

The 200-odd Figaro commands are now at your disposal. However, only the short names without the ‘fig_’ prefix are available.

There is a demonstration procedure, which is intended as a test of the installation, but the script itself can give you hints about writing your own ICL procedures and about the syntax for command line parameters. To execute it, you need an X windows display.

To run the demo type:

ICL> load $FIG_DIR/demo

To run the Specdre demo, which demonstrates how the Specdre Extension is used to e.g. gather fit results and pass them on to other applications, type

ICL> load $FIG_DIR/demo_specdre

Once in ICL, you can initialise further packages, such as KAPPA
Any additional initialisation may write a message to the terminal that some ‘key has been redefined’. This message indicates that the same command name is used in different packages and probably for quite different purposes. The latest initialisation overrides previous ones, and if you are using several packages it might be important to initialise them in the correct order.

### 2.2.3 Starting Figaro from IRAF

IRAF is a powerful and comprehensive package for reducing and analysing optical astronomical data. It was developed at the National Optical Astronomy Observatory (NOAO) in Tucson, Arizona and is now in widespread use around the world. Users run IRAF applications from a command line interface called CL. It is also possible to run Figaro from CL. There are (at least) two reasons for wanting to do so:

- it provides a convenient and familiar interface for people who already use IRAF and who want to use some of the Figaro applications,
- it gives Figaro users access to the large number of applications available in IRAF.

The Figaro and standard IRAF applications can inter-operate, so a given dataset can be processed using a mixture of Figaro and standard IRAF applications.

Figaro and IRAF intrinsically use different data formats (the Figaro formats are described in Section 3.2). However, when Figaro is run from the IRAF CL the system is set up so that the Figaro applications automatically convert to and from the IRAF format on input and output respectively. Thus, you will see only data files in the native IRAF format. This conversion happens automatically and invisibly and as a user you will not normally need to be concerned about how it is done. However, one caveat that you should be aware of is that the native Figaro format is more ‘capable’, in the sense that it can contain more auxiliary information such as quality arrays, error arrays, etc. than the IRAF format. Thus, when datasets are written using the IRAF format some of this auxiliary information can be lost.

The procedure to run Figaro from within IRAF is as follows.

1. First start IRAF following the normal procedure at your site. The precise details differ between sites. [SG/12](#sg12) gives some general guidance. However, you should check with your site manager to get the specific instructions for your site.

2. From within IRAF CL simply type:

   `CL> figaro`

   (here ‘CL>’ is the CL prompt, not something that you type in). A welcome message and list of commands will be displayed and the Figaro applications can simply be run.

There are various points which it is useful to be aware of when running Figaro, or other Starlink applications, from within CL. The details are discussed in [SUN/217](#sun217).
2.2.4 Inter-operability between Figaro and other packages

Figaro applications can inter-operate with applications in various other packages. Running Figaro from the IRAF CL (see the previous section) allows Figaro to inter-operate with IRAF applications.

Figaro applications can also inter-operate with the applications in most other Starlink packages. The most widely useful are probably those in the general purpose image processing package KAPPA (see SUN/95). To start both Figaro and KAPPA from the Unix shell simply type:

```
% figaro
% kappa
```

In both cases the start-up messages have been omitted for brevity. The order of the two commands is not important. The applications in both Figaro and KAPPA are now available and to a large extent they inter-operate.

The procedures to run the packages from ICL or to inter-operate with other Starlink packages are similar.

2.3 Getting help

Since you are reading this, you obviously found one way of getting help with Figaro. There are different sorts of help you can get:

- Run-time help can tell you about a command parameter while the command runs and you are being prompted for the parameter value.
- On-line help in the narrow sense is information available via the ‘fighelp’ (Unix shell) or ‘help’ (ICL) commands. Little more than documentation for each command, and a classified list of commands is available.
- The printed documentation, Starlink User Note 86 (usually written SUN/86 for short), complements the on-line help, but does not include it. SUN/86 describes the package in general, but not the commands in detail.
- An extended, hypertext version of SUN/86, which includes all the text of the printed version and the documentation for all the individual Figaro commands, is available via the World Wide Web.

2.3.1 Help from within the Unix shell

From the Unix shell you can get help from the on-line help library. Use the command

```
% fighelp
```

and navigate through the hierarchy of information in the help library. You can also specify topics and sub-topics as command parameters. The command

```
% fighelp istat
```
ISTAT examines an image, or a rectangular subset of an image or spectrum, and outputs a number of statistics about it, such as maximum and minimum value, mean and sigma, etc. The results are used to set Figaro user variables, so can be used by Figaro procedures. There are a number of examples in the documentation.

Additional information available:

PARAMETERS SOURCE_COMMENTS

Often the most extensive and interesting information is in the sub-topic ‘source_comments’.

The command

```
% fighelp istat para image
```

returns

```
ISTAT

PARAMETERS

IMAGE

TYPE FILE
NAME IM(AGE)
OPTIONS INPUT
PROMPT "(IMage) Name of image to examine"
TEXT IMAGE is the name of the image (or spectrum) for which statistics are to be determined.
```

You can also get ‘run-time’ help: When a command prompts you for some additional information and you have no idea what it’s on about, try responding with a question mark. You will then get some help on that parameter of that command, and are re-prompted for the parameter. Observe this:

```
% istat
IMAGE - (IMage) Name of image to examine /@demo/ > ?

ISTAT
PARAMETERS
IMAGE
TYPE FILE
NAME IM(AGE)
OPTIONS INPUT
```
IMAGE is the name of the image (or spectrum) for which statistics are to be determined.

<table>
<thead>
<tr>
<th>Prompt</th>
<th>Text</th>
</tr>
</thead>
<tbody>
<tr>
<td>PROMPT</td>
<td>&quot;(IMage) Name of image to examine&quot;</td>
</tr>
<tr>
<td>TEXT</td>
<td>IMAGE - (IMage) Name of image to examine /@demo/ &gt;</td>
</tr>
<tr>
<td></td>
<td>XSTART - (XStart) First X value to be used /0.5/ &gt;</td>
</tr>
<tr>
<td></td>
<td>XEND - (XEnd) Last X value to be used /2039.5/ &gt;</td>
</tr>
</tbody>
</table>

X-range 1 (0.5000) to 2040 (2039.5)
Total (over 2040 pixels) = 37751.6
Max = 764.51 in pixel (849,1)
Min = -0.12577 in pixel (149,1)
Mean = 18.506
Sigma = 48.3061

2.3.2 Help from within ICL

From ICL you can get help from the on-line help library. Use the command

ICL> help figaro

Help on the commands is also available by giving them as the topic, use e.g. ‘help istat’, not ‘help figaro istat’. Topics other than commands (news, classified list of commands) are available only as follows:

ICL> help figaro
... 
Topic? class exam

Classified

Examination

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>HIST</td>
<td>Produce histogram of data value distribution in an image</td>
</tr>
<tr>
<td>ICUR</td>
<td>Inspect an image with cursor</td>
</tr>
<tr>
<td>IGCUR</td>
<td>Use cursor to show x, y and data values</td>
</tr>
<tr>
<td>ILIST</td>
<td>List the data in an image (or spectrum)</td>
</tr>
<tr>
<td>ISTAT</td>
<td>Provides some statistics about an image (max, min etc.)</td>
</tr>
</tbody>
</table>

Run-time help is available in the same way as from the Unix shell.

2.3.3 Help via the WWW

The most comprehensive help for Figaro is the extended, hypertext version of SUN/86 available via the WWW. To access this document on a Starlink system simply type

% showme sun86

Otherwise use URL
Figaro also has a WWW ‘home page’ at URL

http://www.starlink.ac.uk/figaro/

2.3.4 Asking for advice

If you cannot find the information that you need or if you think that you have found an error in Figaro then you can contact Starlink for advice and assistance. In the first instance you should send an e-mail message to username

starlink@jiscmail.ac.uk

2.4 Looking at a spectrum

Suppose you have a data file ‘a_file.sdf’ with a spectrum, i.e. a one-dimensional data set and want to plot is on an X display. This is probably the most commonly used Figaro command, and it is called ‘splot’. Many command names begin with an ‘s’ for spectrum or with an ‘i’ for image. However, before we can plot anything, we need to tell Figaro what our plotting device is.

Consider the following sequence of commands:

% xdisplay
ICL> soft xw
ICL> splot a_file accept

The first command is necessary if your X display ‘server’ is not the same as the ‘client’ machine running Figaro for you. It passes the remote client the identity of the local server. You have to declare ‘xdisplay’ each time you log onto the remote client.

Conversely, you may also have to reveal the identity of the remote client to the local server, say with an ‘xhost’ command. Otherwise the remote client may not be allowed use the local server as a display.

The second command tells Figaro which graphics device you want to use. ‘xw’ is an abbreviation for ‘xwindows’. Together with the information from ‘xdisplay’ this is sufficient to open the window. You should now get a display window on your screen, and a box with the word ‘PGPLOT’ in the centre is drawn into the window. You need to give the ‘soft’ command only once. Figaro will always remember that you want to use the device ‘xw’.

The third command finally displays the spectrum contained in the file ‘a_file.sdf’. Data files can have names ending with ‘.sdf’ or ‘.dst’. They must not contain any additional periods. The Figaro commands know about this, and must not be given this file name extension.

Don’t worry about the size of this window. By default you get about 700 by 500 pixels, usually big enough to read the axis labels.

The word ‘accept’ looks like a parameter to the ‘splot’ command. Actually you can use it on any command. It prevents the command from prompting you for information that it can guess itself.

Now consider this more complex sequence of commands. It achieves the same thing, basically.
The ‘xdisplay’ command now tells the remote client explicitly what the Internet host name of the local server is. This is necessary when you log on indirectly, via a third host.

There is also an extra ‘xmake’ command to create the display window explicitly. This way, we can tell it the size and shape of the window, and the colours for background and lines. You can make your personal preference permanent by specifying resources for ‘xmake’ in your ‘.Xdefaults’ file. If you look at the paper copy or on a black/white display, you will notice that this is not a good choice of colours, since the grey values for blue and green are rather similar.

‘xmake’ is paired with a command ‘xdestroy’, which you can use to get rid of the graphics window. That is necessary before you ‘xmake’ it with different parameters, or when you have too many windows on your display.
The ‘splot’ command is different in a number of ways. Before, we had given the input data set as the first parameter. When we are not sure about the sequence of parameters, we can specify them by name. For most Figaro commands that work on spectra as opposed to images, the input data are specified in the parameter called ‘spectrum’.

Next, we have specified the ‘whole’ parameter as false, so that this time we can choose only part of the spectrum to be displayed. We also left out the ‘accept’ keyword, that is why the command asks us a number of questions while it runs.

2.5 Looking at an image

2.5.1 The ‘image’ command

Suppose you have a data file ‘a_file.sdf’ with an image, i.e. a two-dimensional data set and want to plot is on an X display. This is one of the most common actions in Figaro. Most users will use the ‘image’ command for this purpose. In general it uses a different display window than where most other plots go. The idea is that most plots are line plots and their device is selected with the command ‘soft’. But ‘image’ should display at least in grey, if not in false colours. So you have to specify the imaging device separately, although it can be the same actual device as the ‘soft’ device.
Consider the following sequence of commands:

```
%xdisplay
ICL> idev xw
ICL> colour grey
ICL> image a_file accept
```

If you are not familiar with the necessities of using X windows over the computer network, see Section 2.4.

The second command tells Figaro which graphics device you want to use. ‘xw’ is an abbreviation for ‘xwindows’. Together with the information from ‘xdisplay’ this is sufficient to open the window. You should now get a display window on your screen, and a box with the words ‘PGPLOT imaging’ in the centre is drawn into the window. You need to give the ‘idev’ command only once. Figaro will always remember that you want to use the device ‘xw’.

The third command changes the display window to show various levels of grey as representation of the image data values. A new display window may have an undefined ‘colour lookup table’, or a different plot command may have changed the lookup table. The ‘colour’ command with parameter ‘grey’ always brings it back to normal.

The fourth command finally displays the image contained in the file ‘a_file.sdf’. Data files can have names ending with ‘.sdf’ or ‘.dst’. They must not contain any additional periods. The Figaro commands know about this, and must not be given this file name extension.

The keyword ‘accept’ prevents the command from prompting you for information that it can guess itself. Unfortunately, you don’t know what it did guess, and you might never learn how you can exert more control over the display.

### 2.5.2 Advanced use of ‘image’

Now consider this more complex sequence of commands.

```
%xdisplay abc.inter.net
%xmake xwindows -g 400x300 -c 128
ICL> image image=a_file prompt
HARDCOPY - Use "hard" devices rather than imaging device /FALSE/ >
IDEV /'xw'/ >
ERASE - Erase screen before display /FALSE/ > t
YSTART - First Y value to be displayed /1/ > min
YEND - Last Y value to be displayed /256/ > max
XSTART - First X value to be displayed /1/ > min
XEND - Last X value to be displayed /256/ > max
LOG - Display using logarithmic scaling /FALSE/ >
OPTIMIZE - Amount of histogram optimisation (0 to 1) /0.5/ > 0
AUTOSCALE - Calculate display limits automatically /TRUE/ >
NEGATIVE - Set limits to give a negative image /FALSE/ > t
XPLACES - Number of sub-displays across screen in X /1/ >
YPLACES - Number of sub-displays across screen in Y /1/ >
ASPECT - Maintain correct aspect ratio for image? /TRUE/ >
IMARRAY /0/ >
```
If you are not familiar with the necessities of using X windows over the computer network, see Section 2.4.

In the first ‘image’ command the keyword ‘prompt’ is used. While ‘accept’ suppresses prompting as far as possible, ‘prompt’ causes any command to ask you everything. This is a good way
of learning the capabilities of commands, but it also causes some spurious prompts like the ones for ‘idev’, ‘imarray’, and ‘imfile’. You should just accept the defaults offered for these parameters.

The first thing we learn through the ‘prompt’ keyword is that ‘image’ could have ‘displayed’ to a printer file instead of a screen window.

We chose to erase the window this time. That gets rid of the remainders of the original plain box. Via ‘ystart’/‘yend’ and ‘xstart’/‘xend’ we can select only part of the image to be displayed. Since we want the whole image and are not sure about the offered default, we use the words ‘min’ and ‘max’. This time, we set ‘negative’ true: The image file contains a negative, negating it during display makes it look positive. With ‘xplaces’/‘yplaces’ we could sub-divide the window into an array of sub-windows and display into one of them. We leave ‘aspect’ true so that image pixels are displayed as squares. Otherwise the display would be stretched horizontally to fill the window.

Having displayed the whole image, we now run ‘image’ again, but display only part of it. We also set ‘xplaces’ zero. That means, we can specify the display area in pixels. Since we do not erase this time, the previous full display remains partially visible.
2.5.3 Adding colour

While reading Section 2.5.1 you may have wondered why the command to choose grey levels is called ‘colour’. Because it can be used to load an existing colour lookup table. This can be done before or after ‘image’. To put some colour into the display we already have:

    ICL> colour contour_lut

![GWM window – xwindows](image)

Figure 5: Adding color to an image.

There are a number of prepared colour lookup tables. They are ‘.sdf’ files, similar to other data files. By convention their names end in ‘_lut.sdf’, and they are stored in a directory pointed to by the Unix shell environment variable ‘FIGARO_PROG_S’. To see a list of colour tables in that directory:

    % ls $FIGARO_PROG_S/*_lut.sdf

Finally, how do you prevent line graphics from resetting your image display? You will recall that the line graphics device is selected with ‘soft’, while the imaging device is selected with ‘idev’. You can simply choose separate windows. Consider this:
% xmake xwindows -c 16
% xmake xwindows2 -c 128
ICL> soft xw
ICL> idev x2w

This will separate the output into two windows. There are four windows possible, but notice the different names in ‘xmake’ and ‘soft’/‘idev’. The ‘-c’ options also reduce the line graphics window to the 16 colours reserved for this, and increase the imaging window to 128 colours.

2.6 Image display in monochrome

If for some reason you cannot use ‘image’ to display images, you can still display them as contour plots or as dithered grey plots. This works even when the display window has only two colours, say when you have a b/w X terminal. Like spectral plots (Section 2.4) these plots go to the ‘soft’ device.

ICL> icont a_file min max min max low=0 high=250 contours=6 accept

Figure 6: Using contours.
If the display window has more than 16 colours, ‘igrey’ will use those resources and display with true grey levels. It will also reset the window colours to grey. Well, strictly speaking the colours are not grey, but go linearly from the background colour to the foreground colour. If you choose pink and green for fore- and background, you can get quite a hideous composition.

2.7 Paper copies

In order to get paper copies instead of screen plots, you will usually use the ‘hard’ device instead of the ‘soft’ device. (Finally you understand why it’s called the ‘soft’ device!) Similar to the commands ‘soft’ and ‘idev’ there is a command ‘hard’ to tell Figaro once and for all what your hard-copy device is:

```
ICL> hard ps_1
```

The main hard-copy devices these days are PostScript files—files, not printers! This means your plots go into a series of files. The first is called ‘gks74.ps’, the second is ‘gks74.ps.1’, the third ends with ‘.2’, and so on.

It is a bit difficult to keep track of these files. E.g. after you created three files you might delete the second. The fourth plot may then use the free name and appear to be the second plot. Also,
if an application erases the graphics device before plotting, you might get two files: If you ‘print’ the first, you may just get a blank sheet of paper from the printer.

You can use ‘ghostscript’ (Unix command ‘gs’; see SUN/197) or, more simply, ‘ghostview’ to view the plots on the screen before you print the wrong one. E.g. type:

```
ghostview gks74.ps
```

There are a number of PostScript devices you can choose from. When ‘hard’ prompts for the device, you can try ‘options’ to get a list of possible replies. The PostScript devices are:

- ps_l: PostScript, grey, A4, landscape
- ps_p: PostScript, grey, A4, portrait
- epsf_l: Encapsulated PostScript, grey, A4, landscape
- epsf_p: Encapsulated PostScript, grey, A4, portrait
- pscol_l: PostScript, colour, A4, landscape
- pscol_p: PostScript, colour, A4, portrait
- epsfcol_l: Encapsulated PostScript, colour, A4, landscape
- epsfcol_p: Encapsulated PostScript, colour, A4, portrait

Before you get enthusiastic about Colour PostScript, it is not possible to use the ‘colour’ application in conjunction with a printer device. This is because ‘colour’ would write its own PostScript file and the information does not go into the PostScript files with the actual display. You need something like KAPPA's command ‘display’, which can load a colour table immediately before displaying the image. In Figaro the only use for Colour PostScript is probably to get coloured line plots where the applications support this.

The difference between the ‘ps’ and ‘epsf’ devices is that the former are complete and intended to be sent to the printer. The latter are incomplete and intended as elements of more elaborate PostScript documents. You could combine several EPSF files into a single PostScript file, or you could include EPSF files as figures in a \TeX or LATE\TeX document. The figures would be included during processing with ‘dvips’.

Consider this example:

```
ICL> hard epsf_p
ICL> igrey image1 17 23 130 125 hardcopy=t
ICL>icont image2 17 23 130 125 hardcopy=t
% psmerge -e -s0.5x0.5 gks74.ps -s0.5x0.5 gks74.ps.1 > hardcopy.eps
```

We choose the portrait orientation for Encapsulated PostScript. This is helpful for later inclusion in a LATE\TeX document, since it is usually printed in portrait orientation as well.

We use ‘igrey’ and ‘icont’ to produce plots of two images that have the same number of pixels per degree on the sky, and we choose the same part of the image. The two applications also use
the same part of the display device. You might be tempted to use this sequence for an overlay on the screen, but ‘icont’ will wipe out the plot that ‘igrey’ made.

We use the ‘hardcopy’ keyword to direct the plots to the ‘hard’ device rather than the ‘soft’ device. The result are two files ‘gks74.ps*’.

Instead of printing each on its own piece of paper, we use the ‘psmerge’ utility (see SUN/164) to combine the two into a single figure. ‘psmerge’ can not only combine several EPSF files, it can also scale, shift and rotate each EPSF graphic individually in the process. The ‘-s’ option scales the graphs to half the size in $x$ and in $y$.

The ‘-e’ option here means that the output is again EPSF rather than PostScript. The idea is that we want to include ‘hardcopy.eps’ in the LATEX version of this document like such:

```latex
...style[11pt,epsf]{article}
...
\begin{figure}[htb]
\begin{center}
\includegraphics{sun86_hard}
\end{center}
\end{figure}
```

We are cheating here, because the GKS EPSF files have an unsuitable ‘BoundingBox’’. You have to edit these files and move one line from the end of the file to the beginning: There are two lines in the EPSF file that begin with ‘%BoundingBox:’. At the top of the file it says ‘(atend)’
2.7.1 Paper copy by cheating

Many line plot commands have a ‘hardcopy’ keyword to choose between the ‘soft’ and ‘hard’ device, or between the ‘idev’ and ‘hard’ device. However, some don’t and others use both devices for different plots.

If you actually tried to enter ‘options’ as hard device you will have noticed that the list contains the screen devices as well. In fact there is no principal difference between all those devices, each can be selected as ‘soft’, ‘hard’, or ‘idev’ device. The three commands to select the three devices just help Figaro to remember more than one device name and to channel the right output to the right device. You can send your hard-copies to the X screen and your screen plots into a PostScript file!

This is a valid option. If there is an application that sends your favourite plot always to the screen, you can

```
ICL> hard ps_l
ICL> soft ps_l draw=f
ICL> idev ps_l draw=f
```

and all your plots go into PostScript files. Note that you should use the ‘draw’ keyword to suppress the creation of an extra page with the frame plotted by ‘soft’ and ‘idev’.

2.8 Examining data

If you have displayed an image with ‘image’ (Section 2.5.1), you can use ‘icur’ to move a cursor over the image. Stop at a point of interest, and you can be told the coordinates and data value of that point.

```
ICL> icur

Use mouse to control cursor.
Press "D" to display pixel data.
Press space bar to record pixel.
Press "Q" to exit.
```

```
76.00000 143.00000 132.3220
198.00000 184.00000 115.9024
214.00000 34.000000 3.863415
```
On the first three positions ‘d’ was used, on the latter three the space bar. ‘Recording’ a pixel does not tell you its brightness, but the position is stored for future use.

If you used ‘igrey’ or ‘icont’ to display, (Section 2.6), then you can use ‘igcur’ in a similar way to find out about pixels.

Another useful inspection command is ‘istat’. It does not need any display, but works on the data directly.

```
ICL> istat a_file accept
Y-range 1 to 256
X-range 1 to 256
Total (over 65536.0 pixels) = 8.4904E+6
Max = 246.29 in pixel (89,47)
Min = 0 in pixel (215,35)
Mean = 129.55
Sigma = 49.3974
```  

2.9 Doing simple things

With Figaro you can not only look at data that happen to be lying around, you can also process data and thus change the contents of data files. The simplest operations are adding or subtracting a constant, or multiplying or dividing a constant into a data set. These four commands are ‘icadd’, ‘icsub’, ‘icmult’, and ‘icdiv’.

```
ICL> icadd
IMAGE - (IMage) Name of image to be added to /@a_file/ > 
FACTOR - (FACtor) Additive constant /1/ > 25
OUTPUT - (OUTput) Name of resulting image > b_file
```

If you compare the statistics of the old ‘a_file.sdf’ and the new ‘b_file.sdf’, you will find that 25 has been added to each pixel value.

What happens if you give for ‘output’ the same file name as for ‘image’? In most cases this is possible. You will then not get a second file, but the first file will be modified to contain the new data.

What if you have a spectrum and not an image? Usually that does not matter. When a command asks for the ‘spectrum’ parameter it usually has to be a spectrum. But when it asks for the ‘image’ parameter you can in most cases give a spectrum instead.

You can also use two images as operands instead of an image and a constant. The commands for this are ‘iadd’, ‘isub’, ‘imult’, and ‘idiv’.

```
ICL> idiv
IMAGE - (IMage) Name of first image /@a_file/ > 
IMAGE1 - (IMAGE1) Name of second image > b_file
OUTPUT - (OUTput) Name of resulting image > a_file
```

The first operand is overwritten with the result.
3 Advanced users

3.1 Parameters: Controlling commands

3.1.1 What is the parameter system?

The applications use the parameter system to get the necessary information from the outside world. The source of information is not always the user’s keyboard. The specification of a parameter on the command line is slightly different from entering the value at the prompt.

A good model to imagine the workings of the parameter system is as follows. The system is a pool of parameter values. On the command line you can pass values to the parameter system (not the application). When the application runs and needs a parameter value, it will ask the parameter system (not the user terminal). For each parameter the system has two sets of rules, one to arrive at a prompt default value and one to work out the value itself. If the value was specified on the command line, the system will just pass that as the value to the application. Otherwise the value is so far unknown to the parameter system and it will construct a prompt default and a value according to the rules. There are several possible sources for these two:

- the last used value of a global parameter (common to more than one application),
- the last used value (as stored on a per-application basis),
- a dynamic default, set by the application at run-time,
- a static default, set in the interface file,
- response to a user prompt.

So asking the user is only one way of getting information from the parameter system. You also see that the defaults offered—or accepted by ‘accept’ on the command line—may be arrived at in a number of ways.

3.1.2 Parameters on the command line

There are three useful keywords the user can give on the command line to control the defaulting and prompting:

- ‘accept’: Don’t prompt, use default values. This is very useful in scripts to avoid the script getting stuck at an unexpected prompt.
- ‘prompt’: Prompt for all parameters. This is a good way to find out the capabilities of a command.
- ‘reset’: For the prompt default use reset values, instead of last used values.
The user is not prompted for a parameter value in all circumstances, but a value can be specified on the command line even if it would not be prompted for. Conversely, if a parameter is given on the command line, then it will not be prompted for.

On the command line, parameters can be specified by position or by name. To specify by position, they must be in the right order and all previous positions must be filled, too. E.g. for a spectrum in ‘a_file.sdf’ the following will work:

```
ICL> istat image=a_file xstart=min xend=max
ICL> istat a_file xstart=min xend=max
ICL> istat a_file min max min max
```

In the third version, the first pair ‘min max’ is for parameters 2 and 3, ‘ystart’ and ‘yend’. Although these are not needed for a spectrum, positions 2 and 3 must be filled in order to use positions 4 and 5 for ‘xstart’ and ‘xend’.

Logical parameters usually do not have positions assigned to them. On the other hand, these can be specified by name and value, or by negated name. Consider the ‘median’ parameter of the command ‘istat’. There are two ways to set it true, and two ways to set it false:

```
ICL> istat median=true
ICL> istat median
ICL> istat median=false
ICL> istat nomedian
```

Furthermore, instead of ‘true’, ‘yes’, ‘t’, or ‘y’ can be used, similarly with ‘false’ and ‘no’.

There are a few vector parameters in Figaro, where the parameter value is not a single number but a vector of numbers. To specify vector values, use square brackets such as

```
ICL> creobj dims=[5,24,3] ...
```

If you set the environment variable ADAM_ABBRV, you can abbreviate the parameter names on your command line to the shortest unambiguous string. Say ‘istat’ has only one parameter beginning with ‘i’. Therefore ‘i=a_file’ is just as well as ‘image=a_file’.

### 3.1.3 Prompted parameters

When a prompt occurs, it consists of the parameter name, a prompt string, and possibly a prompt default. Normally the user responds with the value for the parameter. But other responses are possible:

```
PARNAME - Prompt text /Default/ >
PARNAME - Prompt text /Default/ > \n
PARNAME - Prompt text /Default/ > 123.45
PARNAME - Prompt text /Default/ > a_file
PARNAME - Prompt text /Default/ > '1st_file'
PARNAME - Prompt text /Default/ > @1st_object
PARNAME - Prompt text /Default/ > min
PARNAME - Prompt text /Default/ > max
PARNAME - Prompt text /Default/ > !
PARNAME - Prompt text /Default/ > !
PARNAME - Prompt text /Default/ > ?
```
By entering nothing and just hitting return, the prompt default is accepted as the proper value. Entering the backslash (\) accepts the default for this parameter and the respective defaults for all parameters that would subsequently be prompted for; no further prompting will occur. Sometimes it is necessary to make clear that a value is not a number. When a file name begins with a digit then in may have to be given with single quotes or with an ‘at’ (@).

Numeric parameters have a permitted range. The user can ask for either of the extreme values to be used by entering ‘min’ or ‘max’. In most circumstances the permitted range of ‘xstart’ etc. is the extent of the data in question. The ‘splot’ command is an exception: The permitted range is close to the floating point range of the machine and grossly exceeds the extent of any reasonable spectrum. This is necessary so that you can have a plot that is wider than the data reach. ‘splot’ has the logical parameter ‘whole’ to adjust the plotted range to the data range itself.

Entering an exclamation mark should assign the null value to the parameter, i.e. make the value undefined. In Figaro this has no meaning and the application should abort. A double exclamation mark is the proper signal for the application to abort. The question mark can be used to get ‘run-time’ help, i.e. help on the parameter currently prompted for.

### 3.1.4 Output parameters

Parameters are not only used to pass information to the application, the application may also return information in other parameters. ‘istat’ has a number of output parameters in order that its results can be used in scripts. From ICL, output parameters can be handled quite easily:

```plaintext
ICL> istat a_file stat_mean=(mean) stat_sigma=(var) accept

Y-range 1 to 256
X-range 1 to 256
Total (over 65536.0 pixels) = 53723.8
Max  = 0.90785  in pixel (89,47)
Min  = 0        in pixel (215,35)
Mean = 0.81976
Sigma = 0.065904

ICL> print (mean)

0.81976

ICL> var = var * var
ICL> print 'variance = ', (var)

variance = 0.00434334

ICL> ave = sqrt(var)
ICL> print (ave)

0.0659040

ICL> icdiv a_file (mean) ffield
```

To achieve anything like that from the Unix shell one needs detailed knowledge of the storage of these output parameters in the Unix file system. The Unix shell does not have floating point arithmetics, but one can at least pass the value from one application to the next:
% istat a_file accept

Y-range 1 to 256
X-range 1 to 256
Total (over 65536.0 pixels) = 53723.8
Max  = 0.90785 in pixel (89,47)
Min  = 0 in pixel (215,35)
Mean = 0.81976
Sigma = 0.065904

% icdiv a_file @$HOME/adam/GLOBAL.STAT_MEAN ffield

Here it is assumed that the environment variable ADAM_USER does not exist or points to $HOME/adam.

3.1.5 Syntax conflicts

Some confusion arises from syntax conflicts between the Unix shell and the ADAM parameter system.

An instructive example is where the user wants to find out the value of a certain pixel in an image. In the language of the ADAM parameter system the user is interested in the object ‘a_file.DATA_ARRAY.DATA(5,7)’:

ICL> hdstrace a_file.DATA_ARRAY.DATA(5,7)

A_FILE.DATA_ARRAY.DATA(5,7) <_REAL>
DATA 152.6
End of Trace.

Now try the same from the Unix shell:

% hdstrace a_file.DATA_ARRAY.DATA(5,7)
Badly placed ()’s.

% hdstrace a_file.DATA_ARRAY.DATA\(5,7\)

A_FILE.DATA_ARRAY.DATA(5,7) <_REAL>
DATA 152.6
End of Trace.

As a rule, we have to mask each meta-character with a backslash (\). The backslashes make sure the Unix shell passes the meta-characters on and does not interpret them. They then make it through to the ADAM parameter system, and from there the previous arguments apply.

Some people prefer other schemes such as enclosing the whole object specification in a pair of single quotes or double quotes. The advantage is that you need only two additional characters no matter how many pairs of parentheses are in the object name. The disadvantage is that you need a matching pair of characters, and that sometimes you need to pass quotes to the ADAM parameter system.

The troublesome meta-characters are parentheses ( ), square brackets [ ], quotes ’, double quotes " and the backslash itself. See SC/4 for further details.
3.2 Data files: Internal details

Your run-of-the-mill Figaro application uses the NDF library to access data. But the HDS object manipulators ‘copobj’, ‘creobj’, ‘delobj’, ‘renobj’, ‘setobj’, and ‘trimfile’ use the HDS library to access data files, just like ‘hdstrace’ does. Notice the small difference between accessing data and accessing data files! The point is that two different data formats (NDF and DST) actually use the same file format (HDS). Therefore accessing such files on a low level is different to accessing the data inside the files on a higher level.

3.2.1 HDS files

HDS files play a vital role in Figaro. Two data formats, NDF and DST, are realised in HDS files. In addition, the ADAM parameter system uses HDS files to store its information. HDS stands for Hierarchical Data System, and the main feature of HDS files is that there is no fixed order in which items are arranged within the files. Instead HDS files contain a hierarchy of named objects, all that matters are the hierarchy tree and the names of objects, not their sequence.

The actual information is kept in ‘primitive’ objects. These are numbers or strings, scalar or arrays. The other kind of objects are structures, which can contain further objects. These are needed to build the hierarchy tree. Structures can be arrays, too.

Users can inspect and modify the contents and organisation of HDS files. For this a syntax to specify objects has been developed. To practise the syntax, you can use the ‘hdstrace’ utility (see SUN/102) to list the contents of an HDS file:

```
ICL> hdstrace a_file full
A_FILE  <NDF>
   DATA_ARRAY  <ARRAY> {structure}
      DATA(256,256)  <_REAL> 106.2439,126.5268,116.8683,102.3805,
                          ... 129.4244,134.2537,141.9805,134.2537

End of Trace.
```

This shows that the file contains a top-level structure of type NDF, within it a structure of type ARRAY called ‘DATA_ARRAY’, and within that a primitive array of type _REAL called ‘DATA’. We also see that this latter array contains 256 by 256 floating point numbers.

You do not have to inspect the whole file, but can concentrate on specific objects within the object tree. For this you use the component names and dots between them. To address array elements, use parentheses:

```
ICL> hdstrace @"a_file.sdf".DATA_ARRAY.DATA(2,1)
A_FILE.DATA_ARRAY.DATA(2,1)  <_REAL>
   DATA  126.5268
End of Trace.
```

Here the @ and the double quotes are not necessary, but they show how you can address a DST file instead. The double quotes bind the ‘.sdf’ into the file name so that it does not look like the object ‘SDF’ within the file. The @ makes clear that you talk about an HDS object and not a string or a number. An @ may be useful if you have file names that begin with a digit.
The top-level object is equivalent to the file itself and is not specified. Its name is in fact irrelevant, so long as it is a scalar structure. The other object names are case-insensitive, you can use upper case as in the example, or lower case.

HDS files have names ending with the file name extension `.sdf’, which stands for Starlink Data File. But this does not have to be so. Figaro’s second data format, DST, also uses HDS files, but their names end in `.dst’. You could use any file name extension, `.sdf’ is just the default that is assumed if you don’t specify it, such as:

```
ICL> hdstrace @"a_file.dst".Z.DATA(2,1)
A_FILE.Z.DATA(2,1) <_REAL>
  DATA    126.5268
End of Trace.
```

A remark is necessary about data types here. The primitive data types (those that do not make HDS structures, but scalars, vectors and arrays of numbers, strings, etc.) have names beginning with an underscore. The most common type is `_REAL’, others are `_DOUBLE’, `_INTEGER’, `_BYTE’, `_UBYTE’, and `_CHAR*n’. When an HDS structure manipulator needs to be given a type, then these are to be used. But if a data processing application such as ‘retype’ needs a type specification, then you have to use the traditional Figaro type specifications ‘FLOAT’, ‘DOUBLE’, ‘INT’, ‘BYTE’, and ‘CHAR’.

### 3.2.2 NDF format

NDF, the Extensible N-Dimensional Data Format, is a data format for spectra, images, cubes etc. In the first place this is a specification of an HDS object hierarchy. Although Figaro’s old data access routines (DSA/DTA) did a reasonably good job at implementing the NDF format, Figaro now uses the official implementation in the form of the NDF subroutine library. So the term ‘NDF’ is not only a recipe for an HDS object tree, it is also the name of a subroutine library to create and access such a tree. And it is customary to call a data set in NDF format ‘an NDF’.

The minimum content of an NDF is an object with name ‘DATA_ARRAY’. It must either be a ‘primitive’ array (one that contains numbers etc.), or it must be a structure of type ‘ARRAY’ and contain a primitive array called ‘DATA’. The two variants are called a primitive NDF and a simple NDF. The following two are equivalent, the first is simple, the second is primitive:

```
A_FILE <NDF>
  DATA_ARRAY <ARRAY> {structure}
    DATA(256,256) <_REAL> 106.2439,126.5268,116.8683,102.3805,
      ... 129.4244,134.2537,141.9805,134.2537
End of Trace.
```

The difference can normally be ignored, since the software will sense which variant is present and act accordingly. When you have to specify HDS objects to ‘hdstrace’, ‘creobj’ etc, you must of
course check which variant is present. Simple NDFs are more flexible, and are also the standard variant.

There are a number of further components in addition to the data array that an NDF can have. There may be a variance array of the same shape as the data array, or title, label and unit strings. A relatively complete NDF might look like this:

```
A_FILE <NDF>

TITLE <_CHAR*32> 'krypton singlet'
UNITS <_CHAR*32> 'A/D numbers per exposure'
LABEL <_CHAR*32> 'OBJECT - DARK'

DATA_ARRAY <ARRAY> {structure}
DATA(310,19) <_REAL> 1655.552,1376.111,1385.559,1746.966,
... 1513.654,1465.343,1446.902,0,0,0,0,0
BAD PIXEL <_LOGICAL> FALSE

VARIANCE <ARRAY> {structure}
DATA(310,19) <_REAL> 87.05064,21.38796,1.088109,14.45454,
... 286.3985,132.1791,119.0586,0,0,0,0,0

QUALITY <QUALITY> {structure}
BADBITS <_UBYTE> *
QUALITY(310,19) <_UBYTE> 0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,
... 0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,*,*,*,*,*

AXIS(2) <AXIS> {array of structures}
Contents of AXIS(1)
UNITS <_CHAR*32> 'microns'
LABEL <_CHAR*32> 'Estimated wavelength'

DATA_ARRAY <ARRAY> {structure}
DATA(310) <_REAL> 1.85115,1.852412,1.853675,1.854938,
... 2.237606,2.238869,2.240132,2.241395

Contents of AXIS(2)
DATA_ARRAY <ARRAY> {structure}
DATA(19) <_REAL> 0.5,1.5,2.5,3.5,4.5,5.5,6.5,7.5,8.5,
... 12.5,13.5,14.5,15.5,16.5,17.5,18.5

End of Trace.
```

The most remarkable addition, and quite commonly present, is the AXIS. AXIS is an array of structures, even if the data are one-dimensional. Each element AXIS(1), AXIS(2) ... is thus a structure. In fact, it is very similar to a minimal NDF. In the AXIS each pixel is given its own coordinate value. In general AXIS(i).DATA_ARRAY.DATA need neither be linear nor monotonic. However, most software may assume one or the other.

There are a few cases where Figaro treats NDFs differently from other Starlink packages. The details are as follows.

- In Figaro the variance array must not contain bad values. ‘goodvar’ can be used to clean up an offending data set.
• Figaro ignores an origin different from (1,1,...). The origin is the number of the first pixel. It can specify that the first pixel should be counted as pixel 5, or -10, etc. In simple NDFs the origin can be stored, but Figaro will not use such information. As far as it is concerned all data sets begin with pixel (1,1,...). The \texttt{KAPPA} package has an application \texttt{setorigin} that allows you to adjust the origin.

• Some Figaro applications may assume that an absent AXIS is equivalent with pixel coordinates 1.0, 2.0, 3.0, ... The default pixel coordinates are in fact ORIGIN-0.5, ORIGIN+0.5, ORIGIN+1.5, ... Sometimes this NDF default is manifested as an actual axis data array, in which case applications may no longer recognise it as a default. For the first and second axes you can use \texttt{lxset} and \texttt{lyset} to give the data new pixel coordinates.

3.2.3 NDF sections

A great advantage in accessing data via the NDF library is that you can specify a section of the data rather than the full array. The section can specify a part of the array, or it might only partially overlap with the array, or they might not overlap at all. The section can also have a different dimensionality than the array itself. Here is an example specification of a seven-dimensional section, in each dimension a different feature of the section specification syntax is used:

\begin{verbatim}
% istat file(3:5,21.3:44.5,17,17.0,3:,,:99)
# (3:5, 1st axis: pixels 3 through 5
# 21.3:44.5, 2nd axis: pixel centre values 21.3 through 44.5
# 17, 3rd axis: pixel 17 only
# 17.0, 4th axis: only the pixel with centre = 17.0
# 3:, 5th axis: pixels 3 through end of NDF
# , 6th axis: all pixels
# :99) 7th axis: pixels from start of NDF through to pixel 99
\end{verbatim}

The commas separate the dimensions (axes) from one another. Colons mean that there’s a range rather than a single value. If the start or end of a range is left out, the start or end of the NDF is used. Integer numbers are pixel indices, floating point numbers are pixel centre values (e.g. wavelengths). Instead of using a colon to separate start and end, you can also use a tilde to separate the centre of the section from the size of the section.

You can use NDF sections whenever existing data are accessed. You cannot use NDF sections to specify output data. And you cannot use NDF sections when access is to the HDS file rather than the data inside it.

3.2.4 Foreign formats

Another great advantage of using the NDF library for data access is that data need not be in NDF format. Any foreign data format can be accessed, so long as a conversion to NDF is defined. An easy way to define a large number of format conversions is to start up the CONVERT package in addition to Figaro:

\begin{verbatim}
ICL> convert
CONVERT commands are now available -- (Version 0.6-2, 1996 September)
\end{verbatim}
Defaults for automatic NDF conversion are set.

Type conhelp for help on CONVERT commands.

At the time of writing this is the list of file name extensions and formats. The list is in order of preference of the formats.

- .sdf NDF format
- .fit FITS format (disk FITS, IBM byte order)
- .dst Figaro format
- .imh IRAF format
- .hdr GASP format
- .unf unformatted binary files
- .dat unf0 binary files
- .asc ASCII files
- .txt text files
- .gif GIF images
- .tif TIFF images
- .sdf.Z compressed NDF files

Normally you need not concern yourself with the file name extension. The data access routines will go through the list and give the application the first data set found no matter which original format they are in. But if you want to be specific, you can say

```
ICL> istat file.fit(3:5,21.3:44.5,17,17.0,3:,,:99)
```

if you want to use the disk-FITS file when there is also a ‘file.sdf’ lying around on your disk. As you can see, you can even use NDF sections on foreign data formats.

You can use foreign formats whenever data are accessed, be it input or output. You cannot use foreign formats when access is to an HDS file rather than to data. Things like ‘delobj’ can be used only on HDS files, which restricts it to NDF and DST format.

Whether you can actually store your data in a foreign format, is not guaranteed. Figaro occasionally uses a FITS extension, which you would lose if you store data in GIF. More relevant is that NDF and DST formats allow you to have non-linear pixel centre coordinates, and you are likely to lose such coordinates altogether if you store in FITS or IRAF format.

There is a penalty for using foreign formats rather than NDF. Accessing input and releasing output data takes that bit longer to do the format conversion. And it may take a lot longer if the disk is not local to the computer that does the processing. So if you use only Figaro and other Starlink packages, it is best to stick with NDF format. If every third application you use is from IRAF or MIDAS, you might be better off using disk-FITS.

NDF to ASCII and ASCII to NDF conversions can also be performed using the ex-SPECDRE applications ASCIN and ASCOUT. ASCIN will read axis values, pixel widths, data values, and data errors from an ASCII table into an NDF structure, while ASCOUT will create such an ASCII table from an NDF.

For more information on foreign formats see the Developer’s Guide on adding format conversion to NDF [SSN/20] and the user documentation on CONVERT [SUN/55].
3.2.5 DST format

One of the foreign formats is called the ‘Figaro format’—in fact it is Figaro’s old DST format. These days Figaro uses NDF format, but that was the case only since version 3.0.

Like NDF, DST is a specification of an HDS object hierarchy. Only the hierarchy is different, the same information is in different places.

The NDF shown in Section 3.2.2 would look in DST format like this:

```
A_FILE <FIGARO>
Z <IMAGE> {structure}
UNITS <CHAR*32> 'A/D numbers per exposure'
LABEL <CHAR*32> 'OBJECT - DARK'
FLAGGED <LOGICAL> TRUE
DATA(310,19) <REAL> 1655.552,1376.111,1385.559,1746.966,
... 1513.654,1465.343,1446.902,*,*,*,*
ERRORS(310,19) <REAL> 9.330093,4.624712,1.043125,3.801913,
... 16.92331,11.49692,10.9114,0,0,0,0
X <AXIS> {structure}
LABEL <CHAR*32> 'Estimated wavelength'
UNITS <CHAR*32> 'microns'
DATA(310) <REAL> 1.85115,1.852412,1.853675,1.854938,
... 2.237606,2.238869,2.240132,2.241395
End of Trace.
```

DST stores errors instead of variances. There is no ‘a_file.Y’ structure, since the second axis has default pixel coordinates. In DST format only non-default axes must actually exist.

There is also no quality array, instead the data array has bad values interspersed with good values. This is not a property of the DST format itself, but the way the format conversion treats a quality array.

3.2.6 Figaro Extension to NDF and DST structures

Due to the flexibility of HDS object trees, it is possible to add extra information. Usually such information is recognised only by specific software. In the NDF format, Figaro uses a Figaro extension and a FITS extension. Any extension to an NDF is an object within a ‘MORE’ structure. Since axes are also kind-of NDFs, they can have extensions, too. Figaro is probably the only package to use an extension to axes.

In DST format the information that would be in NDF extensions is not so obvious to locate. The following table translates between objects in an NDF extension and the equivalent in a DST file.
The FITS extensions in the two formats differ internally. In DST each FITS item is an HDS object. But in NDF format the whole FITS header is an array of strings, and each FITS item is a string in this array.

Needless to say, all but the FITS extension go overboard when you use other foreign data formats. You can inspect the FITS extension with ‘fitskeys’ and add or change items with ‘fitset’.

### 3.2.7 The Specdre Extension

The [NDF data access routines](#) – together with the lower-level [HDS routines](#) – allow the construction of extensions to the data format. Extensions unknown to an application package are propagated, and extensions known to an application enable it to provide more functionality.

The Specdre applications use a set of extensions to the basic NDF format to communicate information between themselves. This extension is known (not unreasonably) as the [Specdre Extension](#). It is recognised and understood by all the Specdre applications. Other Figaro applications, and applications in other Starlink packages such as KAPPA, simply propagate the Specdre extension unchanged without modifying (or understanding) it. This behaviour is not always correct, but in general it is the best that can be done. If the dataset is not processed with another Specdre application it does not matter if the Specdre extension and the basic NDF become inconsistent (because other applications will ignore the extension). If a Specdre application is presented with a dataset in which the Specdre extension is obviously inconsistent with the basic NDF it will notice and report an error.

Specdre contains a number of subroutines for easy and consistent access to the Extension. Programmers are advised to contact the Figaro support username (e-mail figaro@star.rl.ac.uk) if they want to make use of the Specdre Extension in their programs.

All Specdre applications support version 0.7 of the Specdre Extension, which was introduced in version 0.7 of Specdre. Some applications support an advanced version 1.1 of the Extension, as introduced in version 1.1 of the package.

**Design** The spectroscopic axis information is not quite complete with only label and units in the axis structure. For a frequency scale you have to know which reference frame it refers to, for wavelengths you might also be worried about the refractive index of the medium in which the wavelength is measured. And for radial velocity scales you need to know the rest frequency of the spectral line. Another thing most useful in spectroscopy would be to store continuum and
line fits along with the data. The concept of extensions to an NDF provides a good means to store such information. The design of Specdre’s own extension is outlined here.

The Specdre Extension is the structure `<myndf>.MORE.SPECDRE` and is of type `SPECDRE_EXT`. `<myndf>` is the creation name of the main NDF, the one with the data, axes etc. `<myndf>.MORE` contains all the extensions to this main NDF. The Specdre Extension may contain other NDFs, thus we have to distinguish between the “main NDF” and the “Extension NDFs”.

Using the Specdre Extension  The demonstration script (`demo_specdre`) shows the use of the Specdre Extension. There is a tool `editext` to look at or manipulate the Extension. `resamp` will usually store some limited amount of information about the interdependence of post-resample pixels in the Extension. If you try to `fitgauss` such re-sampled data, that application will pick up and use the information in the Extension. `fitgauss` or `fitpoly` will store their fit results in the Extension. In conjunction with the NDF sections (see Sections 3.2.3 and B.4) you can work your way through the rows of a CCD long-slit spectrum, store each row’s fit in the Extension and fill the result storage space. The collection of all fits is again an NDF (located in the Extension) and can be fed into other applications like `specplot`, `ascout`, or indeed KAPPA’s `display`.

The Extension also provides a place to store a full-size array for the wavelength or frequency etc. of each pixel. This array of spectroscopic values may be created by `grow`, and is used by `arcdisp` to apply individual dispersion curves for the rows of an image. The image may be a long-slit spectrum, or the rows may be extracted fibre spectra or échelle orders.

There is some potential for confusion here. You may tend to count pixels in your data starting at 1, you may use NDF pixel indices, NDF pixel coordinates, a vector of pixel centre values, or an N-dimensional array of spectroscopic values.

- **Pixel indices** along an axis in an NDF are counted from the lower bound to the upper bound. The lower bound is also known as the origin. It is not necessarily 1. Say, if an NDF has bounds 1 ... N, a section of it may have bounds 5 ... (N-6), or -5 ... (N+6) etc. If the section is made permanent in a new file, it will still have origin other than 1. Usually it is these pixel numbers – or the section bounds – that are used to specify NDF sections in parameter values like `ndf(-25:32,4:)`.

- The centres of pixels in an NDF have pixel coordinates that are the pixel number minus 0.5. So if the bounds are 5 ... 95 then the pixels have centre coordinates 4.5 ... 94.5. The beauty of these coordinates is that, if you give each pixel an extent of 0.5 on either side, then the sequence of pixels 1 .... N cover the coordinate range [0;N]. These are only default coordinates, but they are worth mentioning because they are similar, but not equal, to the pixel indices.

- Each axis of the NDF may have an explicit vector with the coordinates of the pixel centres. These can run non-linear, backwards, or in loops. You may find that some software cannot cope with the weirder of these options. These centres can be used to specify NDF sections. You simply give a floating point bound instead of an integer bound. Note that this works only because each axis has a vector of pixel centres as long as the NDF axis, not an N-dimensional array of pixel centres.
For Specdre it is useful to have in addition an N-dimensional array where for example a wavelength calibration can be stored for each row of an image or a cube individually. For this purpose there may exist an array of spectroscopic values in the Specdre Extension to the main NDF. That array is an NDF with the same bounds as the main NDF. Where the main NDF stores the count rate, brightness etc. of a pixel, the spectroscopic values’ NDF stores the wavelength, frequency, radial velocity etc. for that pixel. This information is recognised only by Specdre. You cannot expect KAPPA’s linplot to use it for axis labelling, Specdre’s specplot does use it of course. You also cannot use this array to specify an NDF section in a parameter value.

There is no rule as to what happens to the centres of the spectroscopic axis when N-D spectroscopic values are created or modified. Both arrays may lead rather independent lives.

Specdre Extension v.0.7 The items of the Extension as introduced with Specdre version 0.7 are described below. All top-level items are optional, but each is either complete or absent.

- **.SPECAXIS** is an _INTEGER scalar which defaults to 1. Its value is the number of the spectroscopic axis. This is the axis along which the one-dimensional ‘spectroscopic subsets’ extend. It must be greater than or equal to 1 and less than or equal to the number of axes in `<myndf>`. A change of specaxis may render other components invalid as regards their values or shapes.

- **.RESTFRAME** is a _CHAR*32 scalar which defaults to ‘unknown’. Its value describes the rest frame used to express the observed frequency, wavelength, radial velocity, or redshift.

- **.INDEXREFR** is a _REAL or _DOUBLE scalar which defaults to 1. Its value is the index of refraction needed to convert frequency into wavelength and vice versa.

- **.FREQREF** is a _REAL or _DOUBLE scalar which defaults to the bad value. Its value is the reference frequency needed to convert between radial velocity and redshift on the one hand and frequency on the other hand. The value is evaluated together with .FREQUNIT:

\[
\nu_0 = \text{freqref} \times 10^{\text{frequnit}} \times \text{Hz}
\]

- **.FREQUNIT** is an _INTEGER scalar which defaults to 0. Its value is the common logarithm of the unit used for .FREQREF divided by Hertz. Note that this item is not of type _CHAR, _REAL or _DOUBLE.

- **.SPECVALS** is an NDF structure with data, label and units components. .SPECVALS.DATA_-ARRAY is a _REAL or _DOUBLE array which has the same shape as `<myndf>`. It defaults to a multiple copy of the centre array of the spectroscopic axis `<myndf>.AXIS(specaxis)` - .DATA_ARRAY. This structure contains spectroscopic axis values (either wavelength, or frequency, or velocity, or redshift) for each pixel of the data cube. Labels and units must be stored with this data structure. Their default values are copies from the spectroscopic axis `<myndf>.AXIS(specaxis)`.LABEL, .UNITS in the main NDF or ‘unknown’ if the axis has no label or unit. A modification of spectroscopic values may render parts of .RESULTS invalid, but no rules are formulated in this respect. This structure must not contain bad values.
- **.SPECWIDS** is an NDF structure with only a data component. **.SPECWIDS.DATA_ARRAY** is a _REAL or _DOUBLE array which has the same shape as `<myndf>`. It defaults to (i) a derivative from **.SPECVALS** in the way prescribed by [SUN/33](#) or (ii) to a multiple copy of the width array of the spectroscopic axis. Just as **.SPECVALS** contains the multidimensional array of spectroscopic values for the pixel centres, this array contains the spectroscopic widths for the pixels. Labels and units are not to be stored with this data structure. This structure is always considered together with **.SPECVALS**. It must not contain bad values.

- **.COVRS** is an NDF-type structure with only a data component. **.COVRS.DATA_ARRAY** is a _REAL or _DOUBLE array which has the same shape as `<myndf>`. **.COVRS** defaults to non-existence. The meaning is as follows: for some reason pixels belonging to the same spectroscopic subset may be interrelated. While no complete covariance matrix is stored, this structure holds the sum over the rows of the covariance matrix (cf. Meyerdierks, 1992). For multi-dimensional data note that this holds information only about the interrelation within any one spectroscopic subset, not between different such subsets. That means we know only about interrelation along the spectroscopic axis (within a spectrum) but not perpendicular to that axis (between spectra).

- **.RESULTS** is an NDF-type structure with data and variance components and a number of extensions in the **.MORE** component. All these extensions are HDS vectors. They have either one element per component or one element per parameter. The shape of the **.RESULTS** structure is defined by (i) the shape of `<myndf>`, (ii) the total number of parameters `tnpar > 0`, (iii) the number of components `ncomp > 0`. Each component has allocated a certain number of parameters `npara(comp) >= 0`. The total number of parameters must be

\[
    tnpar \geq \sum_{comp=1}^{ncomp} npara(comp)
\]

while the parameter index for any component runs from

\[
    para1(comp) = \sum_{i=1}^{comp-1} npara(i) + 1 \quad \text{to} \quad para2(comp) = \sum_{i=1}^{comp} npara(i)
\]

Components are additive, i.e. the combined result is the sum of all those components that can be evaluated.

Note that the concept of a component is different from that of a transition or a kinematic component: a component is a line feature you can discern in the spectrum. Any component can in general be assigned a transition and a radial velocity. So you may have several components belonging to the same transition and several components of similar velocity belonging to different transitions. You may at any time decide that a discernible component has been misidentified and just change its identification. The concept of a component is even more general, in that it can be the continuum, in which case there is in general no laboratory frequency for identification.

---

There is, however, a restriction for data sets with more than one spectrum. Any component may differ from spectrum to spectrum only by the fitted values. The mathematical type and identification of components is common to all spectra.

The results structure in the Specdre Extension v.0.7

- `.RESULTS .DATA_ARRAY` and `.RESULTS .VARIANCE` are _REAL_ or _DOUBLE_ array structures which default to bad values. They have one axis more than <myndf>: the spectroscopic axis is skipped; instead new first and second axes are inserted. The first axis counts the fit parameters up to the maximum (tnpar). The second axis is of length 1 and may be used in future. All further axes are of the same length as the corresponding non-spectroscopic axes in <myndf>.

- `.RESULTS .MORE .LINENAME` is a _CHAR*32_ vector which defaults to ‘unidentified component’. There is one element for each component. Its value is a spectroscopist’s description of the component, such as '[OIII] 5007 v-comp #1’, ‘12CO J=1-0’, ‘nebulium’, ‘5500 K black body’. It is essential that the strings are of length 32.

- `.RESULTS .MORE .LABFREQ` is a _REAL_ or _DOUBLE_ vector which defaults to bad values. There is one element for each component. The value is the laboratory frequency of the transition. The units used are the ones stored in .FREQUNIT. The laboratory frequency is the frequency as observed in the emitter’s rest frame. The meaning of this frequency is similar to that of .FREQREF in that the laboratory frequency of a transition is a useful value for the reference frequency of the velocity or redshift axis. The difference is that each component fitted may or may not have its own laboratory frequency. .FREQREF will usually be a copy of one of the elements of .RESULTS .MORE .LABFREQ.

- `.RESULTS .MORE .COMPTYPE` is a _CHAR*32_ vector which defaults to ‘unknown function’. There is one element for each component. Its value is a mathematician’s description of the component, such as ‘Gauss’, ‘triangle’, ‘Lorentz-Gauss’, ‘Voigt’, ‘polynomial’, ‘Chebyshev series’, ‘sine’. It is essential that the strings are of length 32.

- `.RESULTS .MORE .NPARA` is an _INTEGER_ vector defaulting to INT((tnpar/ncomp)). There is one element for each component. Its value is the number of parameters stored for that component. When more components are added to an existing .RESULTS structure, then the new components are allocated by default INT((tnpar - tnpar_old)/(ncomp - ncomp_old)) parameters. The numbers of parameters must be greater than or equal to zero.

- `.RESULTS .MORE .MASKL` and `.RESULTS .MORE .MASKR` are _REAL_ or _DOUBLE_ vectors which default to bad values; both are of the same type. There is one element in either vector for each component. A component comp is evaluated according to type and parameters in the range of spectroscopic values between maskl(comp) and maskr(comp). The component is assumed to be zero outside this interval. Bad values indicate that the range is not restricted.

- `.RESULTS .MORE .PARATYPE` is a _CHAR*32_ vector which defaults to ‘unknown parameter’. There is one element for each parameter. Its value is a mathematician’s description of the parameter. A Gauss profile might be specified by parameters ‘centre’, ‘peak’, ‘sigma width’, ‘integral’. It is essential that the strings are of length 32.
It should be noted that there exist no rules about how to store certain components, such as ‘A Gauss profile must be called Gauss and have parameters such-and-such’. What is stored should be described by the strings so that a human reader knows what it is all about. This does not prevent pairs of applications from storing and retrieving components if they use the strings in a consistent way. The documentation of any application that writes or reads results should specify what strings are written or recognised.

**Specdre Extension v.1.1** The items of the Extension added with Specdre version 1.1 are described below. All top-level items are optional, but each is either completely or absent.

- `.COORD1` and `.COORD2` are NDF structures each with data, label and units components. `.COORD_i.DATA_ARRAY` are `_REAL` or `_DOUBLE`. Both have the same shape, which is similar to that of `<myndf>`. The only difference is that in `.COORD_i` the spectroscopic axis is degenerate (has only one pixel). Either both or neither of `.COORD1` and `.COORD2` must exist. The data values default to a multiple copy of the pixel centres of the main array along the first and second non-spectroscopic axes. For example, if `.SPECAXIS` is 2, then `.COORD1.DATA_ARRAY` is a multiple copy of `<myndf>.AXIS(1).DATA_ARRAY` and `.COORD2.DATA_ARRAY` is a multiple copy of `<myndf>.AXIS(3).DATA_ARRAY`. In the same example, if `<myndf>` has shape `nx by ny by nz`, then both `.COORD_i` have shape `nx by 1 by nz`. `.COORD_i` store for each spectrum a two-dimensional position. This position could be used by a plot routine to position spectra according to `.COORD_i` on the plot. The values may or may not be sky positions. They could be in any coordinate system and using any units. Labels and units must be stored with both NDF structures. Their default values are copies from the relevant axes of `<myndf>`, or ‘unknown’ if the relevant axis has no `.LABEL` or `.UNITS`. The data components must not contain bad values.

One difference between `.SPECVALS/.SPECWIDS` and `.COORD_i` is important to note. `.SPECVALS` and `.SPECWIDS` are to be used (within Specdre) as replacement of the centres and widths in `<myndf>.AXIS(specaxis)`. But `.COORD_i` are not intended as replacements for the corresponding axis information, these structures are used only in special circumstances. In practice the consequences are as follows.

- If an application requires axis information for any axis, then `.SPECVALS/SPECWIDS` must override `<myndf>.AXIS(specaxis)`, but `.COORD_i` are ignored completely.
- If an application requires information about the more general position information as provided by `.COORD_i`, then these structures are looked for. Only in their absence are the relevant axes in `<myndf>` used to generate the same information.

**An example Specdre Extension** As an example, a sensible Specdre Extension has been added to a three-dimensional NDF, which in fact is a data cube of HI 21 cm spectra taken on a grid of galactic longitudes and latitudes. Here is how `hdstrace` sees the file before the Extension is added:

```bash
LVCHI <NDF>
TITLE <_CHAR*12> 'KAPPA - Cadd'
DATA_ARRAY(32,37,150) <_REAL> -0.1216488,0.1160488,4.3972015E-02,
... 0.4012871,-7.1809769E-02,0.4626274,*
```
Now almost all components that can exist in a Specdre Extension are set, mostly to their default values. The only component missing is the sum of rows of a covariance matrix. This is because that structure usually must not exist: other structures can be assigned 'harmless' values, but the simple existence of .COVRS makes a difference. The Extension was actually made with the editext command, which can also list a summary of the Extension:

List of Specdre Extension (v. 1.1)

Name of NDF: /home/hme/lvchi
Spectroscopic axis: 3
Reference frame: local standard of rest
Refractive index: 1.0000000
Reference frequency: 1420.405751786000 [10**9 Hz]
Spectroscopic values exist and are velocity [km/s].
Spectroscopic widths do exist.
First coordinates exist and are galactic longitude [degree]
Second coordinates exist and are galactic latitude [degree]
Covariance row sums do not exist.
Result structure provides for 4 parameters in 1 component.

# line name lab freq. type npara mask from to
1 HI (21cm) LV component 1420.405751786000 Gauss 4 -.1701412E+39 -.1701412E+39

# parameter type
1 centre
2 peak
3 FWHM
4 integral

End of list.

Using hdstrace to list the NDF now yields:

LVCHI <NDF>
TITLE <_CHAR*12> 'KAPPA - Cadd'
DATA_ARRAY(32,37,150) <_REAL> -0.1216488,0.1160488,4.3972015E-02, ...
AXIS(3) <AXIS> {array of structures}
Contents of AXIS(1)
DATA_ARRAY(32) <_REAL> 130,130.1667,130.3333,130.5,130.6667, ...
Contents of AXIS(2)
DATA_ARRAY(37) <_REAL> 17,17.16667,17.33333,17.5,17.66666, ...
Contents of AXIS(3)
DATA_ARRAY(150) <_REAL> -163.855,-162.565,-161.275,-159.985, ...

DATA_ARRAY(37) <_REAL> 17, 17.16667, 17.33333, 17.5, 17.66666, ...
... 22.49998, 22.66665, 22.83331, 22.99998

Contents of AXIS(3)
DATA_ARRAY(150) <_REAL> -163.855, -162.565, -161.275, -159.985, ...
... 24.48488, 25.77489, 27.06489, 28.35489

MORE <EXT> {structure}
SPECDRE <SPECDRE_EXT> {structure}
SPECAXIS <_INTEGER> 3 (1)
RESTFRAME <_CHAR*32> 'local standard of rest' (2)
INDEXREFR <_REAL> 1 (3)
FREQREF <_DOUBLE> 1420.405751786 (4)
FREQUNIT <_INTEGER> 9 (4)
SPECVALS <NDF> {structure}
DATA_ARRAY <ARRAY> {structure}
DATA(32,37,150) <_REAL> -163.855, -163.855, -163.855, ...
... 28.35489, 28.35489, 28.35489
ORIGIN(3) <_INTEGER> 1, 1, 1

LABEL <_CHAR*64> 'velocity' (6)
UNITS <_CHAR*64> 'km/s' (6)

SPECWIDS <NDF> {structure}
DATA_ARRAY <ARRAY> {structure}
DATA(32,37,150) <_REAL> 1.289993, 1.289993, 1.289993, ...
... 1.290001, 1.290001, 1.290001
ORIGIN(3) <_INTEGER> 1, 1, 1

COORD1 <NDF> {structure}
DATA_ARRAY <ARRAY> {structure}
DATA(32,37,1) <_REAL> 130, 130.1667, 130.3333, 130.5, ...
... 134.8335, 135.0002, 135.1668
ORIGIN(3) <_INTEGER> 1, 1, 1
LABEL <_CHAR*64> 'galactic longitude'
UNITS <_CHAR*64> 'degree'

COORD2 <NDF> {structure}
DATA_ARRAY <ARRAY> {structure}
DATA(32,37,1) <_REAL> 17, 17, 17, 17, 17, 17, 17, ...
... 22.99998, 22.99998, 22.99998
ORIGIN(3) <_INTEGER> 1, 1, 1
LABEL <_CHAR*64> 'galactic longitude'
UNITS <_CHAR*64> 'degree'

RESULTS <NDF> {structure}
DATA_ARRAY <ARRAY> {structure}
DATA(4, 1, 32, 37) <_REAL> *, *, *, *, *, *, *
... *, *, *, *, *, *, *
ORIGIN(4) <_INTEGER> 1, 1, 1, 1

VARIANCE <ARRAY> {structure}
DATA(4, 1, 32, 37) <_REAL> *, *, *, *, *, *, *
... *, *, *, *, *, *, *
ORIGIN(4) <_INTEGER> 1, 1, 1, 1

MORE <EXT> {structure}
LINENAME(1) <_CHAR*32> 'HI (21cm) LV component'
LABFREQ(1) <_DOUBLE> 1420.405751786
COMPTYPE(1) <_CHAR*32> 'Gauss'
NPARA(1) <_INTEGER> 4
MASKL(1) <_REAL> *
(1) The third axis of the main NDF is declared the spectroscopic axis. In fact this is the velocity axis of the data.

(2) The telescope’s local oscillator was controlled such that the spectrometer recorded frequency in the reference frame of the local standard of rest. Alternatively, someone could have re-calibrated the spectroscopic axis at some stage to refer to the LSR.

(3) The refractive index is set to 1, which amounts to ignoring refraction for the purpose of wavelength calibration. Probably no one will ever use wavelengths for these data anyway. If this structure were missing, a value of 1 would be assumed whenever necessary.

(4) The reference frequency is encoded in two numbers. Taken together they say that the velocity axis refers to a laboratory frequency of 1420.405751786 MHz.

(5) Here is a data cube of the same shape as the main data. It stores the velocity axis for each sky position \((l,b)\). Thus for each pixel in the three-dimensional data cube, there is a separate velocity value stored for the pixel centre position. In this case it is a waste of disk space since all positions have the same velocity grid. But a slit spectrum recorded with a CCD may have a shift of wavelength scales from one row to the next.

(6) The cube with the spectroscopic pixel centres should have its own label and unit. As the data reduction and analysis progresses, these may change, while the main NDF’s .AXIS(3) structure will in general remain untouched.

(7) Here is a data cube of the same shape as that with the pixel centres, only that it stores the pixel widths. These arrays of centres and widths may exist, or not, independently of each other. Though usually the width is only useful if there is also an array of centres.

(8) These two structures look similar to the cube of spectroscopic pixel centres. But these ‘cubes’ do not extend along the spectroscopic axis. Each contains one number per spectrum. The two structures combine to give each spectrum a two-dimensional position, which may be the position on the sky, on plotting paper, or both. In this example the first spectrum is at \((130,17)\), the second at \((130.167,17)\) and so on. The positions thus coincide with those according to the first and second axes of the main NDF. In this case it is a waste of disk space. But in general these two structures allow for non-orthogonal and even non-linear projections from NDF axis centre space to \(COORD1/COORD2\) space. A two-dimensional main NDF might actually be an arbitrary sequence of spectra, and still these two structures could help to sort each spectrum into its place on the sky.

(9) Here is a rather complex NDF structure. It is used to store and retrieve fit results. The results can be different from one sky position to the next, but there is only one set of parameters per position. The spectroscopic axis – of length 150 – is scrapped, the two positional axes – lengths 32 and 37 respectively – are retained. Inserted are the first and second axes, here of lengths 4 and 1 respectively. Don’t wonder about the second axis, it is always of length 1. Usually the result structure will be manipulated by applications as they find it necessary to store data in it. But most of the structure can be manipulated explicitly with the command \(\text{editext}\).
(10) This should contain the values of the parameters, by default they take the bad value, which is represented here by asterisks.

(11) This should contain the variances of the parameters as a measure of their uncertainties. Again, by default these values are bad.

(12) The extensions to the result NDF are seven vectors. Most have one element per spectral component – in the example 1. The .PARATYPE vector has four elements, one for each parameter. The sole component provided for in the result NDF is the ‘LV component’ of the 21 cm line. Its laboratory frequency is repeated here, the value is independent of the reference frequency above, but the same unit (MHz) is used. We obviously expect that the spectral line has the shape of a Gauss curve and we want to store four parameters describing that curve. .PARATYPE indicates the meaning of all parameters. No mask is enabled to limit the velocity range where the Gauss curve applies, i.e. .MASKL and .MASKR have the bad value.

We might want to also store the results for a parabolic baseline fit. Then we would add a second spectral component with three parameters. The vectors that are now of length 1 would become of length 2, .PARATYPE would become of length 7. The additional second vector elements would be ‘baseline’, bad value, ‘polynomial of 2nd order’, 3, bad value, bad value. The fifth to seventh element of .PARATYPE could be ‘coeff. 0’, ‘coeff. 1’, ‘coeff. 2’.

3.2.8 The Contents of the Twodspec Longslit Results Structure

The Twodspec applications use a unique results structure to store identifications and parameters of line fits.

COMB, ARCSDI, ARC2D and LONGSLIT create a .RES structure in which to store their results. This structure enables repeat fits etc. to be performed easily, as well as making it unnecessary to do all the fits at one time. Most of this structure is mapped and its arrays thus accessed directly from the programs. Data are mapped by the address of the first element of an array in the file’s being obtained by the program. Fortran passes all arguments to subroutines and functions by giving their address, although for character data this is the address of the descriptor, which includes the address of the data. Therefore, if this address is passed to the subroutine or function—its value not address—then the called routine will treat the data as a normal array. For character data a descriptor must be constructed and passed by address. In the interests of portability it is better to use an array in common to pass the address, the element of the array at that address is passed (even though that is outside the bounds of the array). For character strings the string must be passed as string(start:end). The arrays are in common so that they can be referenced with the same offset from different subroutines.

Since COMB performs fitting of continua rather than lines, the structure for COMB is different from that for the other programs in that the arrays are dimensioned in channels where other programs would dimension them in cross-sections.

The elements of the structure are listed below :-

```plaintext
.RESULTS Results
.DATA_ARRAY[21,5,385,1] Float 10.50 546515.9 6542.3 1.558 165959.0 .... -1.701E+38 -1.701E+38
.VARIANCE[21,5,385,1] Float 100 2.069E+7 0.01732 0.08867 1.29E+8
```
```
MORE Struct
.PARAMS[210] Char Spacel_posBase Centre_1 Width_1 Height_
.REST_WAVE[5] Float 6548.1 6562.8 6583.6 0 0
.TRAML[5] Float 6542.0 6555.0 6575.2 0 0
.TRAMR[5] Float 6553.7 6568.1 6589.4 0 0
.TWODSPEC Struct
.ITALMKS[5,385,1] Short 12 10 12 1 1 12 10 12 1 1 12 10 12 1 1 12 12 12 1 1 12 12 12 1 1 12 12 12 1 1 12
... 12 12 1 1 12 12 12 1 1 12 12 12 1 1
.CONTROL[3,5,385,1] Int 101110.0 11000 2000 101110.0 11000 3000
... 1100 101100.0 11000 1100
.ITERATION Short 12
.FIT_STATUS[3,5,385,1] Int 301114.0 11000 1000 301111.0 11000
... 11000 1000 0 0 0 0 0 0
.SELECT[7,5] Short 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0
... 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0
.TOLS[13] Float 0.01899 3 -3 0.03799 1.5 2 0.1899 5 100000.0 20
... 5 2.000E-3 2.000E-3 3
.OPTSTATE Struct
.PAR_Status[21] Int 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0
... 0 0
.FREE_PARAMETERS[21] Int 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0
... 0 0
.LINK_INDEX[21] Int 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0
... 0 0
.LINK_CONSTANT[21] Double 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0
... 0 0
.LOWER_BOUND[21] Double 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0
... 0 0
.UPPER_BOUND[21] Double 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0
... 0 0
.PERIODIC_PARS[21] Int 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0
... 0 0
.PERIODS[21] Double 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0
... 0 0
.PERIOD_START[21] Double 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0
... 0 0
.TEMPLATE Struct
.DATA_ARRAY[560] Float 0 0 0 0 0 0 0 0 -323.2 8736.9
... 7299.5 7346.9 7426.4
.AXIS[1] Struct
.DATA_ARRAY[560] Float 6513 6513.2 6513.4 6513.6 6513.8
... 6618.8 6619.0 6619.2
.GROUPS Struct
.ALL Struct
.TYPE[20] Char
.NUMBER Short 0
.DATA_ARRAY[5] Short 0 0 0 0 0
.SKY Struct
.TYPE[20] Char
.NUMBER Short 0
.DATA_ARRAY[5] Short 1 1 1 1 1
```
The .TRAML and .TRAMR arrays store the line positions, that is the limits considered for optimisation (assumed at the centre by ARC2D), the .IDS array stores the line identifications and the .REST_WAVE their wavelengths. The .ARC array is used by ARC2D to decide which lines are to be used in the evaluation of the relationship between channel number and wavelength. If the element of .ARC is 0 then the line is included under all circumstances, if it is 1 then it is included for non-‘continuity corrected’ data only, otherwise it is not included for any. A value of 4 indicates that no fits are present, while 10 and 11 are the values if the user manually deletes a line which previously had the value 0 or 1 respectively. If arc is 10 or 11 the fits can be ‘undeleted’.

The TEMPLATE structure keeps a record of the one-dimensional spectrum used for line identification. Since the axis array is also kept, it is possible to CLONE from such an array, even if the main data array has been scrunched.

The DATA_ARRAY array is used to store the results of the Gaussian fitting, and is also used by ARC2D to store the results for the continuity correction. The errors on the results are stored as variances in the VARIANCE array. The .PARAMS structure acts as an index to this (used by the program to determine where to store results). The .CONTROL array gives the fit type to be performed (in the same form as the fit status element of the .RESULTS structure). In the above example, 171 is the number of cross-sections in the image and 10 is the maximum number of lines allowed for in the structure (this can be up to 50). Originally the block number was stored, but this gave an ambiguous way of determining where the block starts and ends. Therefore the starting cross-section of the fit is now stored (it is possible to change the blocking and only alter a small number of fits).

The fit is performed on data extracted from ‘nwindow’ cross-sections starting at ‘first cross-section’.

<table>
<thead>
<tr>
<th>Element</th>
<th>Digit</th>
<th>Number</th>
<th>Name</th>
<th>Refer to as</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>Absorption flag</td>
<td>FIT_ABS</td>
<td>0 - Emission</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1 - Absorption</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>2</td>
<td>Profile Model</td>
<td>FIT_MODEL</td>
<td>0 - none</td>
</tr>
<tr>
<td>3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1 - Gaussian</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>2 - Skew Gaussian</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>3 - Cauchy/Gaussian</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>4 - Centroid</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>5 - Lorentzian</td>
</tr>
<tr>
<td>4</td>
<td>3</td>
<td>3</td>
<td>Fit type</td>
<td>FIT_TYPE</td>
<td>0 - none</td>
</tr>
<tr>
<td>5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1 - single</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>2 - double (fixed separation)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>3 - double (fixed width ratio)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>4 - double (fixed height ratio)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>5 - double (independent)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>6 - Multiple</td>
</tr>
<tr>
<td>6</td>
<td>4</td>
<td>4</td>
<td>Number of Components</td>
<td>FIT_NCMP</td>
<td>or may act as maximum</td>
</tr>
<tr>
<td>7</td>
<td>5</td>
<td>5</td>
<td>Weights Method for Profiles</td>
<td>FIT_WEIGH</td>
<td>0 - Uniform</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1 - Variance</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
| 8 | 6 | Profile Fit Status | FIT_STAT | 0 - No fit  
1 - Success  
2 - Nag error (not serious)  
3 - Nag error (serious)  
4 - Crash  
5 - Failed tols  
6 - Failed tols (non-serious Nag)  
|   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |
| 2 | 1 | Manual guessing flag | FIT_MAN | 0 - No manual guessing  
1 - Manual guessing (between below and fit)  
|   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |
| 2 | 3 | First Guess method | FIT_GUES | 1 - Centroid  
2 - Peak  
3 - Bimodef  
4 - Inherit FORWARD  
5 - Previous answer at this place  
6 - Inherit BACKWARD  
7 - REGION (2d for TAUROUS-to be defined)  
8 - ROBUST estimator  
9 - MODEL (synthetic model eg rotation curve)  
10 - P Cygni  
|   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |
| 4 | 9 | Optimization method | FIT_OPT | Choice of routines  
|   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |
| 5 | 10 | Component number control | FIT_STST | 0 - Fit up to number of components requested  
1 - AIC after fitting  
2 - AIC, before fitting (using guesses)  
|   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |
| 6 | 11 | Constraints Method | FIT_CONSTR | 0 - No Constraints  
1 - Bounds only  
2 - General Constraints (read from constr. struc.)  
3 - EQUATIONS-read from equations Struct  
|   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |
| 7 | 12 | Dynamic weights flag | FIT_DYNWEI | 0 - no dynamic weights  
1 - Use dynamic weights  
|   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |
| 8 | 9 | Fit group | FIT_GROUP | Number of group  
|   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |
| 3 | 1 | Method of Removal | BACK_REMOV | 0 - subtract  
1 - divide  
|   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |
| 4 | 5 | Background Order | BACK_ORDER |   
|   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |
| 6 | 4 | Weight Function to be used | BACK_WEIGH | as for entry 5 above  
|   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |
| 2 | 3 | GLOBAL Background model | BACK_MODEL | 0 - No base  
1 - Flat BAse  

For background model 3 the polynomial is fitted immediately before the line profile fitting is carried out. For background model 4 the polynomial is evaluated using coefficients previously stored in the data file and subtracted before fitting.

The FIT_STATUS array contains information as to the type of fit performed and as to the success or otherwise of the fit (see table 1).

Only LONGSLIT and FIBDISP are able to fit multiple Gaussians, so ARC2D, COMB, and ARCSIDI use a smaller results structure.

The last element in this direction is the density scale factor, used for scaling previous fits for use as first guesses to another fit.

The elements of the CONTROL array are the same as those of the FIT_STATUS array, except that they do not include information on the success of the fit. This array is used to set which type of fit is to be performed.

The .ITMASK array and .ITERATION are used in conjunction to prevent accidental refitting of data. Note that a check is made to prevent a fit being accidently overwritten by a simpler fit. The order of increasing complexity is: single Gaussian, skew Gaussian, Cauchy function, double Gaussian, multiple Gaussian. Both this type of checking and the checking using .MASK and .ITERATION can be overridden in MANUAL mode.

The .TOLS array provides a way of retaining the values for tolerances between successive work on the same file, and also (using the FIGARO function LET, or the CLONE COPY option in LONGSLIT), provides a way of transferring these from one file to another. This retention enables tolerances to be applied in batch, since it is much simpler to specify which tolerances to apply, rather than to list all the values to use. It is also used during multiple fitting in batch to determine the number of components to fit.

The results structure for three–dimensional data arrays used by FIBDISP is similar to the above, but has some extra elements. The VARIANT element gives the form of the relationship between the array elements of the main data array, and their actual positions. For type='HEX' there is a .XDISP array: XDISP[IY] defines the displacement in the X direction of the .Z.DATA[IX,IY], relative to the value given by X.DATA[IX] (that is the element of the data of axis number one). ORDER (at present always ‘SORT’) indicates whether the data are arranged with the first or the last array index varying over wavelength. SORT corresponds to the first axis being wavelength. TOTAL_INTENSITY stores the total intensity integrated along the wavelength axis, and is useful for locating interesting areas of the data.
3.3 Error propagation

From version 5.2-0 onwards Figaro’s error-propagation capabilities have been expanded and enhanced. Consequently many data sets may now be reduced with the error information propagated through to the final result. This section describes the error-propagation features of Figaro.

3.3.1 How is the error information stored?

For NDF format files, the default is to store the error information as an array of variance values (i.e., the uncertainties squared). If error information in your data is not stored as variances, some routines will not propagate the errors properly (this will be true of the old Figaro DST format which uses uncertainties).

Note that in Figaro the variance array must not contain bad values. ‘goodvar’ can be used to clean up an offending data set.

To check that a variance structure exists use the command ‘hdstrace’. If your data already contains a variance array, the output from ‘hdstrace’ will look something like this:

```
NDF <NDF>

DATA_ARRAY <ARRAY> {structure}
DATA(1021,200) <_REAL> 204.8,180.48,225.28,230.4,245.76,
... 586.24,587.52,753.92,710.4,684.8
ORIGIN(2) <_INTEGER> 1,1

TITLE <_CHAR*11> 'PG 1157 +00'
UNITS <_CHAR*9> 'ELECTRONS'
VARIANCE <ARRAY> {structure}
DATA(1021,200) <_REAL> 642.6113,618.2913,663.0913,668.2113,
... 1025.331,1191.731,1148.211,1122.611
ORIGIN(2) <_INTEGER> 1,1

MORE <EXT> {structure}
FITS(174) <_CHAR*80> 'SIMPLE = T', 'BI...
... '15TELE = ...', 'PACKEND', 'END'
CCDPACK <CCDPACK_EXT> {structure}
DEBIAS <_CHAR*24> 'Tue Apr 29 21:11:48 1997'

End of Trace.
```

This trace indicates that there is a 1021 × 200 element object labelled ‘VARIANCE’.

If the output for one of your files does not contain the VARIANCE line, your file contains no usable error information.

Only some Figaro applications propagate error values. The full list of Figaro routines which propagate the variances is:
Note that if you use an application which does not propagate the variances, you will see the following message:

*Warning: The data in the reference SPECT have been re-shaped but the variance array was never updated. It will now be deleted.*

What this means is that since the variance array is now no longer correct, it has been deleted to prevent its use. Running `hdstrace` on the new file shows that the variance structure is gone. The obvious lesson from this is to stick to those Figaro routines which keep the variance structure intact wherever possible. Alternatively, Starlink packages such as KAPPA offer other error-propagating tasks.

### 3.3.2 Getting error information into a file

Raw data from the telescope probably don’t contain a variance array. The recommended way of getting error information into a file is when the data are in its earliest stages of reduction, i.e., when de-biasing. The CCDPACK (see SUN/139) applications `makebias` and `debias` will create a variance array in the de-biased data files which Figaro can use in the remaining stages of reduction.

### 3.3.3 How are the errors propagated?

Variances are propagated using a standard equation (see e.g. equation 3.13 from Bevington and Robinson\(^2\)). Note that covariances are not calculated to save computational time. Thus, calculated variances will not be formal for problems with correlated errors.

### 3.3.4 Reducing data with error propagation

In order to avoid losing your error information during the reduction process, the recipe outlined below describes a recommended reduction path. Note that routines from both CCDPACK and KAPPA are employed. If you are working at a Starlink site, these will already be installed ready for your use. If you are not working from a Starlink site, you might wish to get these packages (e.g. from the [Starlink Software Store](https://www.starlink.ac.uk/software) on the World Wide Web) if you don’t already have them.

### 3.3.5 Flat fielding

CCDPACK’s `makeflat` is the recommended way of producing a master flat-field frame. However, either CCDPACK or Figaro may be used to perform the flat-fielding operation. The recommended Figaro flat-fielding routine is `ff`.

---

3.3.6 Sky subtraction

‘Polysky’ should be used to subtract the sky background from your data. Note that there is no direct way of obtaining the sky variance from the region containing the object spectrum. For this reason it is assumed that the variance in the object region is the average residual to the polynomial fit in the sky regions. Those outlying points not used to calculate the sky fit are not included in the calculation of the average residual.

3.3.7 Optimal extraction

Both the Figaro ‘profile’ and ‘optextract’ routines support error propagation. A normal extraction can be performed with ‘extract’.

3.3.8 My data are full of cosmic rays—how do I get rid of them?

The routines ‘bclean’ and ‘sclean’ now partially support propagation of error information. More specifically, bad pixels are interpolated over as before and the associated variance values are set to zero to indicate the data are ‘fudged’.

3.3.9 Zero Variances

Some Figaro routines use the variance arrays to weight the data values during fitting. When a zero variance is found (e.g. because ‘bclean’, ‘sclean’ and/or ‘cset’ were used), the data will not be given infinite weight! Instead, some routines (such as ‘polysky’) will set the weight of such a point to be zero. Some routines (e.g. ‘ff’) may revert to using uniform weighting in this situation.

To be absolutely sure that the variances are used to weight a fit, one might wish to set zero variances to some high value. (The reason that a large value is not set by default is to prevent numerical overflows in later stages of the reduction process). The way this is achieved is using the KAPPA routines ‘setmagic’ and ‘nomagic’:

% setmagic comp=v in=file1 out=file2 repval=0

This replaces any zeros in the file called ‘file1.sdf’ with a ‘bad’ value and writes the output to ‘file2.sdf’.

% nomagic comp=v in=file2 out=file3 repval=1.0e+20

replaces the ‘bad’ values with a high value (1.0e+20).

Now the fitting routine may be run, giving a very low weight to the previously bad pixels. Finally,

% thresh comp=v in=file3 out=outfile thrlo=0 thrhi=0.99e+20 newlo=0 newhi=0

replaces the high values with a zero value. Any remaining stages of data reduction can now be carried out without the worry of encountering numerical overflow from the variances. Note that any variances which were bad for any reason other than being set as bad by the user will also be returned to zero.
3.4 Standard Figaro directories

Figaro has three directories which are used to keep various data files, such as flux calibration tables and arc line lists. The locations of these directories are not fixed, but rather they are referred to by environment variables. These environment variables are:

<table>
<thead>
<tr>
<th>environment variable</th>
<th>meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>FIGARO_PROG_S</td>
<td>default or standard</td>
</tr>
<tr>
<td>FIGARO_PROG_L</td>
<td>local</td>
</tr>
<tr>
<td>FIGARO_PROG_U</td>
<td>user</td>
</tr>
</tbody>
</table>

FIGARO_PROG_S is a standard directory which is always available and contains the same files in all Figaro installations. On standard Starlink systems it corresponds to directory:

```
/star/etc/figaro
```

FIGARO_PROG_L contains files local to your site and FIGARO_PROG_U your own personal files. They may or not defined at your site. You could define your own Figaro directory by adding a line similar to the following to your `.login` script:

```
% setenv FIGARO_PROG_U $HOME/myfigarodata
```

By default Figaro applications will search for data files in these directories in the following order: first your default directory, then FIGARO_PROG_U, then FIGARO_PROG_L and finally FIGARO_PROG_S.
4 Doing more complex things

4.1 Flats

Flat-field division should be relatively straightforward; you take your raw data and divide it pixel for pixel by your flat-field. In practice, this may be acceptable for image data, but there are additional considerations to be taken into account when the data involved are spectral.

4.1.1 Image data

In principle, you can simply divide your image by the appropriate flat-field and use the result. For example:

ICL> idiv image=mydata image1=ff output=flatdata

In practice, you really don’t usually want to divide your data by the (typically) very large numbers found in flat-fields. Ideally, your flat-field should first be reduced to numbers around unity. The easiest way to do that is to divide the flat-field by its mean value, and the easiest way to find the mean value is with ‘istat’. The command

ICL> istat image=ff reset accept

will give you the mean value over the whole image. If your flat-field has strange effects round the edges, you may prefer to limit the range of x and y values used by ‘istat’. For example, if your image is 800 by 800,

ICL> istat image=ff ystart=100 yend=700 xstart=100 xend=700 accept

will only look at the central 600 by 600 part of the image. ‘istat’ prints out the value of the mean, and you can then divide the flat-field by that. For example, supposing the mean value were 350, the command

ICL> icdiv image=ff factor=350 output=ffdiv

will generate a new flat-field (ffdiv) that will have a mean value around unity, as desired. You may find it convenient to know that ‘istat’ sets its output parameter ‘stat_mean’ to the mean value. So you could use

ICL> istat image=ff ystart=100 yend=700 xstart=100 xend=700 ~
    stat_mean=(x) accept
ICL> icdiv image=ff factor=(x) output=ffdiv
4.1.2 Spectral data

Figaro expects you to have your spectral data oriented so that the x dimension is the wavelength dimension. If this is not the case, use ‘irot90’ to switch the axes. In theory, Figaro does not care whether wavelength increases or decreases with x value, but in practice routines tend to be tested with data whose wavelength increases with x value, and odd bugs may turn up with ‘reversed’ data. You are recommended to get your data into the more common form, if necessary using ‘irevx’. Note that, as always in Figaro, ‘x’ means the horizontal axis when data is displayed using ‘image’.

![Spectral data layout diagram]

The main problem with spectral flat-fields is that the flat-field data will usually vary in x, not because of the instrumental response, but simply because of the spectral response of the flat-field lamp. The usual way of dealing with this is to fit this spectral response—obtained by collapsing the flat-field in y—and then multiplying by the fitted value before dividing by the flat-field. Another way to look at this is to consider the result of dividing the flat-field by the fit to the spectral response. The result would be an image which was essentially flat in X, but which is ‘humped’ in Y—since most flat-fields fall off at the edges in Y due to instrumental vignetting. Dividing by this—in a two-dimensional manner—will give you images where the pixel to pixel variations in the detector have been corrected, along with any spatial vignetting of the instrument, but where the overall wavelength response of the instrument is not corrected.

(Of course, if you think your flat-field lamp is flat—which may be true at high dispersion—you can just divide by the raw flat-field and you’re taking out the instrumental wavelength response as well. However, the better way to deal with that is with your standard stars.)

The only problem, usually, is deciding on what represents an acceptable fit to the collapsed flat-field. One approach is to fit a polynomial to it. This at least has the advantage of being automatic, and if the result is satisfactory this is probably the best thing to do.

So this recipe is as follows:
(1) Collapse the flat-field in Y to give a single spectrum. This is the average spectral response of the flat-field lamp combined with that of the detector.

(2) Fit a low order polynomial to this, giving a smooth spectrum. It may be better to fit to the log of the data if large count numbers are involved, but this is not usually critical.

(3) Take each of the cross-sections of constant Y value in the flat-field and divide them by this smoothed spectrum. The result is your corrected flat-field calibration image.

(4) Divide each pixel of your image to be corrected by the corresponding pixel of the flat-field calibration image.

The individual Figaro commands to do this might be the following

```
ICL> extract image=ff ystart=min yend=max spectrum=spres
ICL> getglobal yend (y)
ICL> icdiv image=spres factor=(y) output=spres
ICL> sfit spectrum=spres order=2 output=spfit logs
ICL> isxdiv image=ff spectrum=spfit output=ffcal
ICL> idiv image=mydata image1=ffcal output=mydataff
```

which, given a flat-field exposure in 'ff', and an image in 'mydata', performs a flat-field calibration as described above and puts the result in 'mydataff'.

This is a sufficiently common way of proceeding that there is a single Figaro command that performs all of this, without the overheads of running four programs and generating the intermediate files. This is the 'ff' command. The command

```
ICL> ff image=mydata flat=ff order=2 output=mydataff
```

is functionally equivalent to the above sequence. However, the advantage of splitting up the process is that you can compare 'spres' and 'spfit' to see how good the polynomial fit to the collapsed data really is. E.g. by

```
ICL> splot spres reset accept
ICL> splot spfit noaxes noerase accept
```

(You also have the advantage that you can control the limits used when collapsing the flat-field; you do not have to use ‘min’ and ‘max’ as the limits, although that is what ‘ff’ does.)

If the fit is not satisfactory, what can you do instead? Well, the problem is essentially that of generating a smoothed spectrum, and Figaro has a number of ways of doing that.

(1) You can actually smooth the spectrum, using ‘ixsmooth’.

(2) You can do it by hand, using ‘cfit’ to generate a spline fit between points indicated interactively using the cursor, having first displayed the spectrum with ‘splot’.

(3) A less interactive way of using spline fits would be to use ‘mcfit’, probably with a zero spectrum for a mask, although you could always use ‘mask’ to generate a suitable mask should there be bad regions in the spectrum. (You can generate a zero mask by multiplying the original spectrum by zero, obviously.)

One of these should enable you to get a satisfactory result, although they all require more effort than does the simple ‘ff’.

---

**SUN/86.21 — Doing more complex things**
4.2 B-Stars

The atmospheric features that mess up the red end of the spectrum may be calibrated out by multiplying by a calibration spectrum obtained from an observation of an object that is featureless in this region. A B star is usually used, hence the term 'B-star calibration'.

The essential Figaro command for this calibration is 'bsmult'. This multiplies the spectrum to be corrected by a B-star calibration spectrum—which is essentially obtained by fitting the continuum of a B star and then dividing that fitted continuum by the B-star observation. 'bsmult' allows for the difference in air masses of the two observations, which is what makes it necessary, rather than, say, the simpler 'imult'.

'bsmult' is a fairly straightforward operation. The problems come in generating the calibration spectrum. At present, the simplest way is to display the B-star spectrum using 'splot' and then generate the continuum by hand using 'cfit'. An alternative is to generate a mask for the lines in the spectrum using ‘mask’ and then fit the masked continuum using ‘mcfit’. The main problem with the latter approach is that the chances are that very little of the spectrum is actually uncontaminated either by stellar lines or by the atmospheric bands.

(The best solution is use an automatic program that fits splines between points on the spectrum that are known to be uncontaminated, but such a program is not available yet—nor is a really good list of such points that will apply for all wavelength ranges and dispersions.)

The calibration spectrum should really be exactly 1.0 at all points not affected by the atmospheric bands, and it is probably worth displaying the calibration spectrum using ‘splot’ and then using ‘cset’ to set such regions to 1.0 interactively. (This is something else that could be made automatic eventually. Fortunately, it isn’t necessary to do this very often.)

Note that ‘bsmult’ requires that both the calibration and the object being corrected have valid air masses. Air masses are stored in NDF data files as ‘file.MORE.FIGARO.SECZ’, and in DST data files as ‘file.OBS.SECZ’. If necessary they may be set by hand using, e.g.

```plaintext
ICL> creobj type=_REAL dims=0 object=file.MORE.FIGARO.SECZ
ICL> setobj object=file.MORE.FIGARO.SECZ value=1.4
```

Also note that the correction applied by ‘bsmult’ is multiplicative—this means that it is not suitable for data that is in logarithmic flux units, such as magnitudes.

4.3 Filters

If the response of a filter has been tabulated, then the table of values may be used to generate a 'spiketrum', which may then be turned into a calibration spectrum. See Section 4.4 for details about these things, which are essentially ways of turning sets of tabulated values into spectra by interpolation.

Say the spectrum to be corrected for a filter response is called ‘spect’. A table for the filter response might look like

```plaintext
* Filter transmission table *
```
This is not a particularly realistic table. A proper table should have enough points to ensure that there are a reasonable number of values tabulated over the region of the spectrum to be corrected. With this table in ‘filter.tab’, a filter calibration spectrum can be produced and applied as follows:

```
ICL> gspike spectrum=spect table=filter.tab spiketrum=filter
ICL> interp spiketrum=filter spectrum=calib
ICL> idiv image=spect image1=calib output=spect
```

Note that ‘idiv’ is used to apply the corrections, which is simply a case of dividing the spectrum to be corrected by the interpolated filter response. The same calibration spectrum may be used for any other spectra that cover the same wavelength range.

### 4.4 Spiketra

A ‘spiketrum’ is a half-way house between a table of values and a spectrum. It has an element for each wavelength value in its range, but only a few of these elements—those that correspond to the table entries—actually have non-zero values. Obviously a spiketrum may be generated very simply given just a table of wavelengths and values at those wavelengths, and a range of wavelength values to be covered. Usually an existing spectrum is used as a template to indicate the wavelength values, and the resulting spiketrum has elements that match those of the template spectrum exactly in wavelength. If a spiketrum is plotted, the result is a set of spikes of varying height—hence the name.

A spiketrum may be turned into a spectrum by interpolating between the spike values. The ‘gspike’ command will generate a spiketrum from a table of wavelengths and data values, and the ‘interp’ command will interpolate between the points to generate a spectrum. ‘gspike’ also records the nearest values that are just outside the wavelength range covered by the template spectrum, so that ‘interp’ may make use of these as well as the actual spiketrum values.

‘gspike’ also allows some control to be exercised over the data structure of the spiketrum—‘SET’ records included in the table file can cause the UNITS or LABEL objects in the structure to be set to appropriate values. As an example, see any of the supplied flux standard table files (files intended for ‘gspike’ usually have a ‘.tab’ extension); these generally set the units of the spiketrum to match those used in the table, so that ‘gspike’ can produce spiketra whose units are AB magnitudes, micro-Janskys, or whatever, simply depending on what is in the table file used.

For more details about ‘gspike’ tables, see Section 4.5.2.

The main reason for the use of spiketra is that they enable what is essentially tabulated data—instrumental response values at certain wavelengths, as calculated by ‘cspike’, for example—to be manipulated using the standard Figaro routines designed to manipulate spectra.
Since spiketra are really just spectra, they can be plotted using ‘splot’. They may be modified, if necessary, using the fudge commands such as ‘tippex’ or ‘setobj’. If ‘spike’ is a spiketrum and ‘smooth’ is the interpolated spectrum generated from it by ‘interp’, the following sequence will generate a plot of the two superimposed—

```
ICL> soft xw draw=false
ICL> splot spike reset accept
ICL> splot smooth noaxes noerase accept
```

Alternative commands to ‘interp’ for interpolating between the points of a spiketrum are ‘spifit’ (which fits a global polynomial to the points) and ‘linterp’ (which uses linear interpolation). The ‘spied’ command is designed to help modify spiketrum points in order to influence the interpolation result—i.e. to fudge the resulting spectrum.

4.5 Flux calibration

4.5.1 Units

First, a brief word about units. Although it may not always be obvious, the underlying philosophy of Figaro is to provide tools which you may use as you wish, and not to force you to reduce your data in a way imposed by the author of the software. It would be in keeping with this for there to be no restrictions on the units that can be used for flux calibration. However, there are practical limitations.

Magnitude units, such as the AB79 system of Oke & Gunn, give results that have a comfortable feel for optical astronomers, and also fall into a numerically convenient range, generally being numbers between 0 and 25. Unfortunately, being logarithmic (not to mention going backwards) they are, while not actually difficult to handle, sufficiently different to linear units to make it impossible to write software that can deal with them other than as a special case.

Rather than do that, Figaro compromises. The flux calibration routines insist on the calibration being performed using linear units. When the flux calibrated spectrum has been produced in such units, it may then be converted to the AB79 scale. This means that rather than have a lot of software that operates on both linear and logarithmic data, we have one routine (‘abconv’) that will perform the conversion.

With that restriction, Figaro will allow you to use whatever linear units you prefer. As will become clearer soon, the units used are determined entirely by entries in the original tables used by the routine ‘gspike’, and if you insist on ‘Joules per minute per square foot per Angstrom’ you are quite at liberty to prepare your own table giving the flux density for your standard at the appropriate wavelengths in these units. You will even find that ‘splot’ will label your axes properly when you plot the data!

As far as linear units go, there are objections to using ‘ergs per sec per square cm per Hz’ and even more so to ‘erg per sec per square cm per Angstrom’ on the grounds that the numbers involved are ridiculously small (and in the latter case can easily go outside the floating point range, which is a serious problem!). So, mainly to be able to avoid having to type commands such as ‘splot high=2.5e-27 low=1.5e-27’, the preferred units for Figaro files are Janskys, mJy, or micro-Janskys.

---

For most of the flux standards for which Figaro supplies tables, two tables are provided: one in AB magnitudes, usually a direct copy of the published data, and one in (milli- or micro-) Janskys, usually the result of a semi-automatic conversion from the former. If you particularly like erg/s/cm\(^2\)/Angstrom, then you can either provide your own flux tables in these units and work from them directly, or you can use the ‘flconv’ command to convert from a spectrum calibrated in Janskys into these units.

### 4.5.2 Standard files

The main Figaro data directory (corresponding to environment variable FIGARO_PROG_S) contains a number of files giving the published flux densities of standard stars, all with a ‘.tab’ extension (see Appendix E.1). In addition Jeremy Walsh has made available the Oke and HST standards and copies can be retrieved by anonymous ftp. See Appendix E.2 for details. For example, the file g158m100.tab begins as follows:

```
* G 1 5 8 - 1 0 0
* Table file for G158-100, faint object flux standard, based on
* Note that these are fitted continuum fluxes, not directly
* measured fluxes, and should be used accordingly. This file is
* designed for use with the Figaro routine GSPIKE. The data here is
* given to 3 decimal places and was supplied directly by Alex
* Filippenko.
*
SET UNITS = "micro-Janskys"
SET LABEL = "Flux"
*  3300  891.251
  3400  990.831
  3500 1135.534
  3600 1282.331
  3700 1432.187
  3800 1599.556
  3900 1770.110
```

The lines beginning with asterisks are treated as comments, and the lines that begin with ‘SET’ are used to set data objects in the file created from this table. An alternative version of this file is ‘g158m100a.tab’, which contains the lines

```
SET UNITS = "AB Magnitudes"
SET LABEL = "Flux"
SET MAGFLAG = 1
*  3300   16.525
  3400   16.410
  3500   16.262
  3600   16.130
  3700   16.010
  3800   15.890
```
The functions of the UNITS and LABEL lines should be fairly obvious. Setting MAGFLAG to 1 in a file indicates to ‘splot’ that the data are in magnitude units and so should be plotted with the flux scale reversed. (Note that most ‘.tab’ files actually used by Figaro in fact use ‘.Z.UNITS’ rather than just ‘UNITS’; these were written for the original version of ‘gspike’ where you were allowed to assume that data units always were held in a file in an item called ‘file.Z.UNITS’. Now that Figaro supports NDF format files, this is no longer the case, and the abstract term UNITS is preferred—however, the original format files still work, since the new ‘gspike’ uses a conversion table to handle these explicitly named items.) Tables based on data from, for example, Oke & Gunn’s 1983 paper will also include the line

```
SET BANDWIDTH = 40
```

to indicate the 40 Angstrom bandwidth used by their data. The Filippenko & Greenstein data represent fitted continuum fluxes, so do not have a bandwidth—a point we shall have to return to very shortly. From the ‘.tab’ files already supplied, it should be possible for you to deduce how to create your own, should that be necessary.

There is a Figaro convention regarding the naming of the table files. The ‘ls’ command may be used to find which files are available as tables. The command

```
% ls $FIGARO_PROG_S/*.tab
```

will list all the table files supplied in the main directory. Note that not all of these are intended for flux calibration; some may be extinction tables, etc. If in doubt, these are all text files, and should have comments at the start describing their function. So, the command

```
% more $FIGARO_PROG_S/file.tab
```

will list the file for you. You should find that most flux files exist in two incarnations, as implied above, one in a Jansky based unit, one in AB magnitudes. The name of the AB magnitude file ends with ‘a’. So, for example, the files ‘l74546.tab’ and ‘l74546a.tab’ both represent the standard star L745-46A, but the former is in Jansky units—and is therefore probably the one you should use—while the latter is in AB magnitudes. The fact that the name of the object itself ends in ‘A’ is an unfortunate complication that may be misleading.

### 4.5.3 The final step

To anticipate for a moment, the final step in the flux calibration process is carried out by the command ‘spflux’. ‘spflux’ multiplies an observed spectrum by a calibration spectrum, to create a flux-calibrated spectrum. Each element of the calibration spectrum contains a value which is effectively the instrumental response of the detector at a given wavelength, the response being in units of ‘Flux density units per count per second per Angstrom’. The ‘Flux density units’ may be any linear units, e.g. mJy. ‘spflux’ assumes that the spectrum to be calibrated is still in counts, and for each element calculates the wavelength range covered by that element, and then combines that with the counts in that element, the value of the calibration spectrum at that element, and the exposure time for the spectrum, to generate a flux density for the central wavelength of the element. The result is, of course, a spectrum in ‘Flux density units’—whatever they happen to be.

‘spflux’ is straightforward enough; the problem, as with similar functions, is to generate the flux calibration spectrum.
4.5.4 Published standards

Published standards fall into two distinct classes; they are similar, but differ sufficiently that Figaro provides different sets of functions for dealing with them.

Older, brighter, standards such as those in Oke & Gunn\textsuperscript{4} are published as tables giving wavelengths and the corresponding flux densities calculated by measuring the observed flux over a range of wavelength centred on the tabulated wavelength. (This is the significance of the .Z.BANDWIDTH value shown in Section 4.5.2.) These tabulated values therefore correspond exactly to what the observed spectrum should look like, even in the presence of absorption features.

More recently, Filippenko & Greenstein\textsuperscript{5} have published tables for fainter stars where they fit a continuum to the observed data and tabulate the value of this continuum at various wavelength values. These values will not therefore represent the actual observed data in regions where there is absorption, and the concept of a ‘bandwidth’ does not apply.

4.5.5 First step—Turning the table into a spiketrum

(You may find it useful to review Section 4.4 before reading on.)

Figaro usually deals with tables, which are tricky to manipulate directly, by turning them into spiketra, which can be manipulated like spectra, although they only have non zero values at points that correspond to those tabulated. No matter which type of published standard is involved, the first step is to generate a spiketrum from the table.

This requires a template spectrum to give the wavelength scale. The one to use is the observed standard spectrum, preferably already scrunched to a linear wavelength scale. (You can use unscrunched data, but it is not advised.) For example, suppose ‘standobs’ is such a spectrum of the standard star HD 84937. There is a table of flux densities for this star (one of the Oke & Gunn standards) in the main Figaro directory, in the file ‘hd84937.tab’. The command

```
ICL> gspike spectrum=standobs table=hd84937 spiketrum=hdspike
```

will generate a spiketrum called ‘hdspike’ that can be used by the subsequent steps. If you want a look at it,

```
ICL> splot hdspike reset accept
```

will show it as a series of vertical spikes, giving the flux density at each of the tabulated points in the wavelength range of the observation.

‘gspike’ will search directories for the table file in the usual Figaro order: first the default directory, then the user’s Figaro directory (environment variable FIGARO_PROG_U), then the local and Figaro directory (FIGARO_PROG_L), and finally the main Figaro directory (FIGARO_PROG_S). So a table file in any of these will be found, should you need to create your own.

4.5.6 Second step—for Oke & Gunn data

The command ‘cspike’ takes this generated spiketrum (hdspike) and combines it with the observation of the standard (standobs) to generate a new spiketrum, whose values are now the instrumental response sampled at the points of the original spiketrum.

```
ICL> cspike spiketrum=hdspike spectrum=standobs output=calspike
```

generates this new spiketrum and calls it ‘calspike’. The values in ‘calspike’ are calculated for each point by summing the counts in the observed spectrum over the appropriate wavelength range, dividing them by the wavelength range and the exposure time, and dividing the result into the flux density value given in ‘hdspike’. The units of ‘calspike’ are therefore ‘units per (count per Angstrom per second)’.

‘calspike’ can be turned into the calibration spectrum required by ‘spflux’ by interpolation. The most direct way to do this is just to use the command ‘interp’.

```
ICL> interp spiketrum=calspike spectrum=calib
```

will produce a new spectrum, ‘calib’, which will be the required calibration spectrum. However, you may find that you are not happy with ‘calib’. It may not be smooth enough. It may be that if there are regions of absorption in the spectrum, poor alignment of the wavelength ranges will result in some spurious values. It may seem to be cheating, but the most direct way to get round this is to edit the spiketrum ‘calspike’ using the ‘spied’ command.

```
ICL> spied spiketrum=calspike output=modspike
```

will display the spiketrum and allow you to delete points, insert new points, and see the results of spline interpolation or global polynomial fitting to the modified points. When you are happy, you can run ‘interp’ on the modified spiketrum to produce a less honest, but more satisfactory, result.

Global polynomial fitting may seem a trifle crude as a way of interpolating between spiketrum points, but it may be that this gives a better result in some circumstances.

```
ICL> spifit spiketrum=calspike order=5 spectrum=calib
```

would be an alternative to the use of ‘interp’ to generate the calibration spectrum.

4.5.7 Second step—for Filippenko & Greenstein data

The most direct way to deal with data where the published values represent fitted continuum values is to fit a continuum to your observed data, interpolate the tabulated data, and divide the two resulting spectra to get a calibration spectrum.

Fitting a continuum is a task that is not easily automated. The simplest way in Figaro is to display the spectrum and then use ‘cfit’. For example:
ICL> splot standobs reset accept
ICL> cfilt output=standfit

will enable you to use the graphics cursor to indicate continuum points on the displayed spectrum and thus generate the continuum spectrum by spline interpolation. A messy job, but not one you have to do often. Remember that the messy parts of the flux calibration are connected with generating the calibration spectrum, and you only have to do that once; applying it is simple.

The published points represent a smooth curve, so just interpolating directly between them should be quite satisfactory. That is, there should be no need to edit the spiketrum generated by 'gspike' prior to using 'interp', although 'spied' is always available if necessary. This time, since HD 84937 is an Oke & Gunn standard, let's use G158-100 instead. So the first two steps will be

ICL> gspike spectrum=standobs table=g158m100 spiketrum=gmspike
ICL> interp spiketrum=gmspike spectrum=gmfit

generates 'gmfit' as the interpolated spectrum from the published points.

Dividing 'gmfit' by 'standfit' will now generate the required calibration spectrum. Note that 'idiv' will not do, since the division has to allow for the wavelength range of each element, and for the exposure time. The command 'caldiv' must be used instead.

ICL> caldiv standard=gmfit spectrum=standfit output=calib

will generate 'calib' as the required calibration spectrum.

4.5.8 An alternative second step—Filippenko & Greenstein data

An alternative way of dealing with this type of data would actually be to treat it as if it were Oke & Gunn data and use 'cspike' and 'interp' as described in Section 4.5.6. This is probably satisfactory so long as the data does not cover a range with significant absorption features. (If it does, you can always remove the bad points with 'spied' before interpolating.)

If you do this, you have to supply a bandwidth, since the table file will not have specified one. You can put one into the spiketrum generated by 'gspike' (Section 4.5.5).

ICL> setobj value=40 object=gmspike.MORE.FIGARO.TABLE.BANDWIDTH

will set the bandwidth to 40 Angstroms. The effect of this is going to be similar to smoothing the observed spectrum with a filter 40 Angstroms wide in order to get the continuum values at the points specified by the table. After that, you just go through the 'cspike', ('spied'), 'interp' sequence as in Section 4.5.6, finally ending up with a calibration spectrum called 'calib'.
4.5.9 The final step revisited

Given ‘calib’ as the calibration spectrum, no matter how it was generated, it is applied as follows:

```
ICL> spflux spectrum=obs calspect=calib output=calobs
```

which calibrates a spectrum called ‘obs’, creating a resulting spectrum called ‘calobs’.

At the risk of being obvious, you should be aware of what you have created here. Each element of ‘calobs’ is a sample at a particular wavelength of the continuous flux density function; it is not a measure of the total flux within a wavelength bin of finite width, which the original spectra were. Apart from anything else, adding two such spectra has the magic (and totally unreasonable) effect of doubling the magnitude of the object! Generally, this is not something to worry too much about, but it is disconcerting to have a spectrum like this generated with a non-linear wavelength scale. (One has a gut feeling that if the bin covers half the wavelength it should have half the value, and that is not true for these spectra.) That is the reason for the warning about not using unscrunched data.

4.5.10 AB magnitudes, and a test

A calibrated spectrum generated in Jansky units (and others, eventually) can be converted to AB magnitudes by ‘abconv’. Our ‘calobs’ spectrum may be converted by

```
ICL> abconv spectrum=calobs output=abobs
```

(‘abconv’ works out the units of the input spectrum by looking at a .UNITS data object in the input file. In some cases, it may not recognise the units—this can happen with spectra that have come from some other system via a translation program. If that happens, and you know that the units are, say, Janskys, you can set the units by hand using

```
ICL> setobj object=calobs.UNITS value=Janskys
```

‘abconv’ will recognise ‘mJy’, or anything that contains ‘Jansky’ and the words ‘milli’ or ‘micro’. The same remarks apply to ‘flconv’.)

An interesting test of the system is to calibrate an object using itself, convert the result into AB magnitudes, and then compare the result with a spiketrum generated from the published AB magnitude tables. For example, remembering that our original spectrum of HD 84937 was ‘standobs’, and given that there is a table called ‘hd84937a.tab’ in the main Figaro directory that has the values in AB magnitudes, we can try

```
ICL> spflux spectrum=standobs calspect=calib output=hdcal
ICL> abconv spectrum=hdcal output=abhdcal
ICL> gspike spectrum=abhdcal table=hd84937a output=abhdspike
ICL> splot spectrum=abhdcal reset accept
ICL> splot spectrum=abhdspike noaxes noerase accept
```
The result should be the spectrum of HD84937 in AB magnitudes, with a series of spikes just touching the spectrum, indicating that (at least at the tabulated wavelengths!) the spectrum has been calibrated to the correct value.

A much tougher test would be to use one standard to calibrate another, and then compare the result with the tabulated values. This does of course require that you really do have spectrophotometric data, with no filter changes, with all of the object in the slit in both cases, and with no clouds in the way.

4.5.11 Manually setting the exposure time

The flux calibration process requires the exposure time of the observation being calibrated. This datum is stored in a standard location in the NDF file holding the spectrum. If your data originates from one of the usual sources, such as the AAT or UKIRT, then the exposure time will probably already be in the correct place in the data structure. However, if your data came from an unusual source then the exposure time may either be present in an unusual location or absent altogether. In these cases you will need to manually add the exposure time to the NDF. The item that has to be added is:

`MORE.FIGARO.TIME`

It should be of type `_REAL` and expressed in units of seconds. Figaro applications ‘creobj’ and ‘setobj’ can be used to respectively create and set the exposure time. E.g. if your data file was called ‘myobs.sdf’ and you wanted to set an exposure time of 90 seconds then you would type:

```
ICL> creobj type=_REAL dims=0 object=myobs.MORE.FIGARO.TIME
ICL> setobj value=90.0 object=myobs.MORE.FIGARO.TIME
```

If the structures ‘MORE’ and ‘FIGARO’ do not already exist in your data file then you will need to create them before creating the exposure time:

```
ICL> creobj type=struct dims=0 object=myobs.MORE
ICL> creobj type=struct dims=0 object=myobs.MORE.FIGARO
```

4.5.12 Summary

For Oke-Gunn type data—

```
ICL> gspike spectrum=standobs table=hd84937 spiketrum=hdspike
ICL> cspike spiketrum=hdspike spectrum=standobs output=calspike
ICL> spied spiketrum=calspike output=calspike { (optional)
ICL> interp spiketrum=calspike spectrum=calib
ICL> spflux spectrum=obs calspect=calib output=calobs
```

For Filippenko-Greenstein type data—

```
ICL> gspike spectrum=standobs table=g158m100 spiketrum=gmspike
ICL> interp spiketrum=gmspike spectrum=gmfit
ICL> splot spectrum=standobs reset accept
ICL> cfit output=standfit
ICL> caldiv standard=gmfit spectrum=standfit output=calib
ICL> spflux spectrum=obs calspect=calib output=calobs
```
4.6 FFT

A number of Figaro functions are available to manipulate complex data, generally with a view to its being used for some process involving Fourier transforms. While there are packaged Figaro routines, such as ‘scross’, which make use of Fourier transforms internally, the functions covered in this section perform the more elementary operations, and can be put together to form a sequence of operations that duplicates the processing performed by, say, ‘scross’, but enabling a finer control to be exercised over the details of the procedure. The general design of this set of routines is based on those provided as part of the SDRSYS system (Straede, 1985). In these notes the term ‘Fourier transform’ is used rather freely; it should be realised that in all cases it is the discrete Fourier transform that is meant.

In Figaro 3.0 the structure of complex data files was changed slightly. Prior to Figaro 3.0, such files contained the modulus of the data as the main data array and held the real and imaginary parts separately. From Figaro 3.0 onwards, the modulus is no longer held as a distinct item, the main data array is the real part of the data, and the imaginary part is still held separately. In practice this should make little difference to the use of these routines, although it does mean that a complex file created by Figaro 2.4 or earlier will not be handled properly by Figaro 3.0. In Figaro 5.1 the format for complex data is changed again slightly. The reason all these changes should make little difference is that complex files are normally intermediate files and are not retained over a long period.

4.6.1 Complex data structures

Figaro defines a ‘complex data structure’ in a fairly precise way. A complex data structure is a structure containing two arrays, one holding the real part of the complex data, and one holding the imaginary part. These arrays have to be the same size and shape, although they may have any number of dimensions. The dimensions of the arrays must be such that the Fast Fourier Transform algorithm (Cooley & Tukey, 1965) may be applied to them. Different implementations of the FFT have different restrictions—many require that the number of elements be a power of 2. The NAG library used by Figaro before version 5.0 required that, in each dimension, the number of elements had to have fewer than 20 prime factors, none greater than 19. The routines used from version 5.0 onwards have no restrictions. They are in the Starlink PDA library, the algorithms come originally from the FFTPACK library.

In a Figaro complex data structure the main data array is the real part of the complex data. This means that most of the Figaro routines may be applied to a complex data structure, and if this is done it will be the real part of the data that they operate on. For example, an easy way to look at complex data is with ‘splot’, which will happily plot the real part of the data, quite ignorant of the fact that the data structure in question is complex. To plot the imaginary part, or the modulus of the data, these will first have to be extracted using ‘cmplx2m’ or ‘cmplx2i’. It is important to note that if an ordinary Figaro function is used to change this data, for example by

\[ \text{ICL> icmult cmplxdata 2 cmplxdata} \]

which multiplies the real array by 2, the imaginary part will be unchanged. Generally, to avoid unnecessary conversions before using the FFT routines, the real and imaginary arrays are held in double precision. However, this is not a strict requirement.
4.6.2 Creating a complex data structure

In most cases, one starts with an ‘ordinary’ (i.e. non-complex) file and wants to produce a complex structure in which this is the real part of the complex data and the imaginary part is zero. This is what the command ‘r2cmplx’ does. For example,

```
ICL> r2cmplx myspect cmplxspect
```

will generate a complex structure with the real part taken from ‘myspect’, and the imaginary part will be set to zero.

In some cases, you may want to set the imaginary part of a complex structure as well as the real part. In this case, you can use ‘i2cmplx’, which takes the data from a non-complex structure and uses it to set the imaginary part of an existing complex structure. This means that the complex structure has to be created initially by ‘r2cmplx’, and then the imaginary part can be set by ‘i2cmplx’. So, if you have two spectra called ‘rspect’ and ‘ispect’ which you want to be the real and imaginary parts of a complex spectrum, the sequence is

```
ICL> r2cmplx rspect cmplxspect
ICL> i2cmplx ispect cmplxspect
```

and the order of operations is important; doing the ‘i2cmplx’ step first will produce a quite different result: the ‘i2cmplx’ will fail, unless it so happens that ‘cmplxspect’ already exists, and even if it succeeds the ‘r2cmplx’ will just produce a new version with a zero imaginary array.

There is no procedure supplied to create a complex structure with the imaginary data taken from a specified file, say ‘ispect’, but with the real part set to zero. It was thought that this would be an unusual requirement. If needed, the following sequence may be used:

```
ICL> icmult ispect 0 scrap
ICL> r2cmplx scrap cmplxspect
ICL> i2cmplx ispect cmplxspect
```

4.6.3 Going smoothly to zero at the ends

The simplest way to ensure that your original data goes smoothly to zero at the ends is to multiply it by a filter that is unity for most of the spectral range but goes down to zero at each end in a smooth manner. The most common form for such a filter is the ‘cosine bell’, and this is what is generated by the Figaro ‘cosbell’ function. (For a detailed discussion, see Brault & White, 1971, and the references they quote).

The only parameter needed by ‘cosbell’ is the percentage of the data that is to be covered by the bell shapes at each end of the data. 10% is a common value to use. ‘cosbell’ uses an input data structure as a template and generates a structure that is the same as the template except for the data itself. Usually, you use the data to which you intend to apply the filter as the template. So, for example, to apply a 10% cosine bell to the data in ‘myspect’,

```
ICL> cosbell myspect 10 bell
ICL> imult myspect bell myspect
```

At present, ‘cosbell’ cannot handle data with more than two dimensions.
4.6.4 Taking the Fourier transform

Actually taking the Fourier transform of a complex data structure is quite straightforward. The forward transform is performed by the Figaro function ‘fft’ and the reverse transform is performed by ‘bfft’. The only parameters for ‘fft’ and ‘bfft’ are the names of the input and output structures.

For example, to calculate the Fourier transform of the spectrum held in the non-complex structure ‘myspect’, it should be enough to do:

```icl
ICL> r2cmplx myspect cmplxspect
ICL> fft cmplxspect cmplxspect
```

which results in ‘cmplxspect’ containing the ‘fft’ of ‘myspect’. If the power spectrum of ‘myspect’ is required it can be obtained by ‘cmplx2m’. For example, it may be plotted using:

```icl
ICL> cmplx2m cmplxspect modulus
ICL> splot modulus reset accept
```

Actually, the modulus is the square root of the power spectrum; the statement that the power spectrum is available represents a slightly cavalier attitude towards the proper usage of terms. However, if the power spectrum is only needed in order to get a feel for the frequency distribution of the original data, then the modulus will generally do just as well.

Often, a power spectrum needs to be plotted logarithmically to produce a sensible plot, so it is quite acceptable to try

```icl
ICL> ilog modulus modulus
ICL> splot modulus reset accept
```

The FFT routines generate data with the low frequency terms in the lowest numbered elements in each dimension of the data. ‘fft’ (the Figaro program) re-orders the transformed data so that the zero frequency term is in the centre of each dimension, and the resulting data now goes from -N to +N (where N is the Nyquist frequency). The plots of the modulus made as described above will show a plot with an axis labeled from -1 to +1, (the unit being the Nyquist frequency), that is symmetrical about the centre and which peaks (for most input data) in the centre. New axis structures are created by ‘fft’ to reflect the new axis values. The reason for this re-ordering of the data is that it seems to be easier to visualise complex filters, particularly in 2-dimensions, when the low frequency terms are collected in the middle of the data rather than scattered into the corners.

The reverse FFT is performed by ‘bfft’. Note that the precise definition of ‘forward transform’ and ‘reverse transform’ differ between FFT implementations. The PDA routines, for example, do not introduce any scaling factors between data that is first forward and then reverse transformed and the original data. This means that the sequence

```icl
ICL> r2cmplx myspect cmplxspect
ICL> fft cmplxspect cmplxspect
ICL> bfft cmplxspect cmplxspect
ICL> cmplx2r cmplxspect newspect
```

should generate a ‘newspect’ that is exactly the same as the original ‘myspect’, except that any axis information will have been lost.
4.6.5 Extracting the real and imaginary parts

The previous section sneaked in a reference to the function ‘cmplx2r’, in the hope that what it did was obvious. As the name is intended to imply, ‘cmplx2r’ is the reverse of ‘r2cmplx’, and generates a non-complex data structure whose data array is taken from the real part of the data in a specified complex data structure.

‘cmplx2r’ does not make any changes to the dimensions of the data array.

Analogous to ‘cmplx2r’ are ‘cmplx2i’ and ‘cmplx2m’, which extract the imaginary part of the data and the modulus of the data. Usually, there will be little point in explicitly extracting the real part, since it is already available to most Figaro functions as described earlier. However, the file generated by ‘cmplx2r’ will be smaller than the original file, since it will no longer contain the imaginary array, and this may be a point in its favour.

4.6.6 Operations on complex data

There isn’t a great deal of point in just transforming data back and forth, unless it’s to get a feel for the errors introduced by such a process. Usually one transforms into the Fourier domain in order to do something useful there. The obvious Fourier domain operations are multiplication (equivalent to a convolution operation in normal space), and multiplication by the complex conjugate (equivalent to a cross correlation in normal space).

Figaro provides a number of functions that operate on complex data to provide a complex result. The operations performed by these should be fairly obvious from their names: ‘cmplxmult’, ‘cmplxdiv’, ‘cmplxadd’, ‘cmplxsub’, and ‘cmplxconj’. The most useful will probably be ‘cmplxconj’, which produces the complex conjugate of a complex data structure, and ‘cmplxmult’, which performs the complex multiplication of two complex data structures.

For example, the cross-correlation function of two spectra may be determined crudely by transforming each, taking the complex conjugate of one, multiplying the two and transforming back. This is crude, because it omits any filtering and preparation of the data, but it will serve as a demonstration of the complex operations involved. If the two spectra in question are ‘spect1’ and ‘spect2’, then

```
ICL> r2cmplx spect1 cspect1
ICL> r2cmplx spect2 cspect2
ICL> fft cspect1 cspect1
ICL> fft cspect2 cspect2
ICL> cmplxconj cspect2 cspect2
ICL> cmplxmult cspect1 cspect2 cspect1
ICL> bfft cspect1 cspect1
ICL> cmplx2r cspect1 corrln
```

will produce a cross correlation function in ‘corrln’.

4.6.7 Complex filters and data smoothing

Because the data in the Fourier domain is in frequency order, it is often simpler to create a filter in the Fourier domain than to produce the corresponding convolution function in normal space and then transform it. In fact, determining the optimum filter for data is often best done by
examining the power spectrum of the data to be filtered and then designing the filter around it. Brault and White (1971) spend some time on this topic.

At present, Figaro provides only one function that generates complex filters, and this is ‘cmplx-filt’. This produces filters of the type described by Hunstead (1980) for use in cross-correlation measurements. These are mid-pass filters formed by taking a Gaussian that peaks at zero frequency and drops to a half height at a specified high cut value, and subtracting a narrower Gaussian that rises from zero reaching its half height at a specified low cut value. That is, the functional form of the data is given by

\[ f(x) = \exp\left(-\frac{x^2}{2v^2}\right) - \exp\left(-\frac{x^2}{2u^2}\right) \]

where \( u \) and \( v \) determine the low frequency and high frequency cutoff values. From this definition it is clear that \( u \) and \( v \) are just the sigma values for the two Gaussians, and that what has been rather loosely termed a ‘half height’ is in fact the point at which the Gaussian has a value of \( \exp(-1/2) \approx 0.6 \). They are specified for ‘cmplx-filt’ in terms of the Nyquist frequency, so a filter might have, say, \( u = 0.1 \) and \( v = 0.3 \). The best way to get a feel for these filters is to generate them and plot them superimposed on the power spectrum of the data to be filtered.

If the low cutoff value is specified as zero, ‘cmplx-filt’ generates a low pass filter whose functional form is just

\[ f(x) = \exp\left(-\frac{x^2}{2v^2}\right) \]

i.e. a Gaussian that drops from unity at zero frequency, having a half width of \( v \). Note that the cyclic nature of the Fourier domain data means that the filter generated is actually symmetrical about the mid point of the data—this is something to be remembered if you want to try to produce other, more specific, filters by generating them yourself. The imaginary parts of the filters generated by ‘cmplx-filt’ are always zero. This means that ‘cmplx-mult’ will have the effect of just multiplying both the real and imaginary parts of the data to be filtered by the real part of the filter (obviously, since \((a + i b) \times (c + i d) = a c + i b c, \text{if } d = 0\)).

‘cmplx-filt’ requires a template complex data structure, which it will use as the basis for the filter it produces. Normally, this will be the data to be filtered, although any data of the same dimensions will do. If the template data is n-dimensional, so will be the resulting filter. The low and high cutoff frequencies will be the same in all dimensions. Since they are specified in terms of the Nyquist frequency, this is usually fairly satisfactory.

So, to take an easy example, an elaborate way of smoothing a spectrum, ‘myspect’ say, by Gaussian convolution, would be

\[ \text{ICL> istat myspect stat_mean=(x) reset accept} \]
\[ \text{ICL> icsub myspect (x) myspect} \]
\[ \text{ICL> cosbell myspect 10 bell} \]
\[ \text{ICL> imult myspect bell myspect} \]
\[ \text{ICL> r2cmplx myspect cmplxspect} \]
\[ \text{ICL> fft cmplxspect cmplxspect} \]
\[ \text{ICL> cmplx-filt cmplxspect 0 0.3 cfilt} \]
\[ \text{ICL> cmplxmult cmplxspect cfilt cmplxspect} \]
ICL> bfft cmplxspect cmplxspect
ICL> cmplx2r cmplxspect myspect
ICL> idiv myspect bell myspect
ICL> icadd myspect (x) myspect

where the 0,0.3 parameters for ‘cmplxfilt’ indicate that a low pass filter is to be created. The 0.3 indicates a cut at around a third of the Nyquist frequency for the data, and the actual value is one best determined by comparison with the power spectrum of ‘myspect’, obtained by performing a ‘splot’ of ‘cmplxspect’ just after the FFT is obtained. Being able to do this is really the main justification for indulging in such a complex procedure when ‘ixsmooth’ could do the same job far faster.

Note that the sequence given above is a pretty complete one. The use of ‘istat’ at the start is to determine the mean level of the data in the spectrum and the result of the ‘icsub’ is to reduce the spectrum to a zero mean level. This reduces the constant component of the power spectrum and should produce a better result. (Note that ‘istat’ sets its output parameter ‘stat_mean’ to the mean value it determines.) The data is apodised by the application of a 10% cosine bell prior to the transform. Both the effects of the cosine bell and the subtraction of the mean level are reversed at the end of the sequence.

4.6.8 Cross correlation, and a sequence something like ‘scross’

As an additional example, consider the following sequence, which attempts to duplicate the functions performed by ‘scross’. It is not an exact duplication, since the internals of ‘scross’ are rather different to those of these FFT routines, and the filters used by ‘scross’ also differ from those generated by ‘cmplxfilt’.

ICL> sfit spect1 4 cont1 log
ICL> isub spect1 cont1 sub1
ICL> sfit spect2 4 cont2 log
ICL> isub spect2 cont2 sub2
ICL> cosbell spect1 10 bell
ICL> imult sub1 bell sub1
ICL> imult sub2 bell sub2
ICL> r2cmplx sub1 cspect1
ICL> r2cmplx sub2 cspect2
ICL> fft cspect1 cspect1
ICL> fft cspect2 cspect2
ICL> cmplxconj cspect2 cspect2
ICL> cmplxmulc cspect1 cspect2 cspect1
ICL> cmplxfilter cspect1 0.1 0.3 cfilt
ICL> cmplxmulc cspect1 cfilt cspect1
ICL> bfft cspect1 cspect1
ICL> cmplx2r cspect1 result
ICL> peak result

‘result’ now contains the cross-correlation of the two spectra, ‘spect1’ and ‘spect2’. ‘peak’ is a utility produced especially for this sort of sequence, listing the position of the highest peak in the spectrum it is given in terms of a shift from the centre of the first element. This is in fact the relative shift of the two spectra ‘spect1’ and ‘spect2’.
Note that the cross-correlation peak will in fact not be central in ‘result’, but will be at one end or the other. It is often easier to handle data like this if the data is rotated so that the peak is roughly central, and this can be done by the function ‘rotx’.

```
ICL> rotx result 765 result
```

where 765 is half the number of pixels in the data (here assumed to be 1530). This has the effect of turning ‘result’ into the cross-correlation spectrum that might have been produced by ‘scross’. Some of the other strange numbers in the sequence may need explanation. The ‘4’s in the ‘sfit’ indicate that the sequence fits a 4th order polynomial to the spectra, and then subtracts away the polynomial fit. This is in fact what ‘scross’ does, unless the fitting is disabled. The 0.1,0.3 parameters in ‘cmplxfilt’ are quite arbitrary, and the optimum values to use are in fact best determined either by looking at the power spectrum of the data to be filtered, or by consideration of the types of features one wants to emphasise in the determination of the correlation (Hunstead, 1980).

### 4.6.9 Summary of Figaro FFT routines

- **bfft**: Takes the reverse ‘fft’ of complex data.
- **cmplx2i**: Extracts the imaginary part of a complex data structure.
- **cmplx2m**: Extracts the modulus of a complex data structure.
- **cmplx2r**: Extracts the real part of a complex data structure.
- **cmplxadd, cmplxdiv, cmplxmult, cmplxsub**: Perform arithmetic operations on two complex data structures giving a third. Cf. ‘iadd’ etc.
- **cmplxconj**: Produces the complex conjugate of a complex data structure.
- **cmplxfilt**: Creates a mid-pass filter for complex data.
- **cosbell**: Creates data that goes to zero at the edges in a cosine bell.
- **fft**: Takes the forward FFT of complex data.
- **i2cmplx**: Copies a data array into the imaginary part of a complex data structure.
- **peak**: Determines position of highest peak in a spectrum.
- **r2cmplx**: Creates a complex data structure from a real data array.
- **rotx**: Rotates a data array along the X axis an integer # of pixels.

### 4.6.10 References

4.7 S-Distortion

The Figaro routines described in this section originated in the need to straighten the distorted spectra produced by image tube detectors. However, even with CCD detectors—which do not suffer the geometrical distortion of the images tubes—some instruments, particularly échelle spectrographs, still produce curved spectra, and the techniques described here can be used to correct these as well. Indeed, nowadays the main application of these routines is to échelle data.

Instruments such as the 2D-Frutti and IPCS that use image intensifiers suffer from various distortions, in particular S-distortion. The effect—and the reason for the name—can be seen clearly by displaying any 2D-Frutti image of a point object, such as a star, on the display. Instead of being a perfectly horizontal line, the spectrum snakes across the image in the shape of a horizontal letter S.

Note that this distortion is actually a two-dimensional distortion—a picture of cartwheel taken through an image tube will show all the spokes bent into S shapes in a radially symmetric manner. However, the difference in pixel scale in the two dimensions for spectral detectors means that—to a first approximation—the distortion can be treated as though it were simply a vertical displacement in the data whose magnitude varies along the spectrum (and to a lesser extent, with position along the slit). In any case, the two dimensional distortion can be corrected by two orthogonal one dimensional corrections: the S-distortion correction described here is one, and the other can be performed as a side effect of a two-dimensional re-binning to a linear wavelength scale. It may be that such full two-dimensional ‘scrunching’ is regarded as overkill—see Section 4.8 for more details.

The process described here is a one-dimensional correction, in which data is re-binned in the direction perpendicular to the dispersion, in such a way as to straighten the spectra.

4.7.1 Usual sequence

The tricky part is to determine the exact shape of the distortion, and in Figaro this is done using the command ‘sdist’. ‘sdist’ requires an image with one or more spectra in it. The usual thing to do is to take an observation of a flat-field through a multi-hole mask, in which case the resulting image will show one spectrum for each hole in the mask, each distorted into an S. The spectra will be roughly parallel, but the two-dimensional nature of the distortion will mean that they are not exactly so; this is the reason for using multiple spectra: it allows the change in distortion with slit position to be determined. The ‘cdist’ command can then be used to apply the results of the distortion analysis performed by ‘sdist’ in order to correct an image.

The usual sequence of operations is as follows: assume ‘holes’ is a multi-hole calibration image, and ‘object’ is an image to be corrected.

(1) Use ‘image’ to display the calibration image (‘holes’) on the image display.

(2) Use ‘icur’ to indicate a starting point for each one of the spectra to be used for the analysis. The starting point should simply be a point, usually near the centre of the spectrum, where the spectrum is strongest. ‘sdist’ will start from these positions as it traces out the spectra.

(3) Run ‘sdist’. This will trace out the indicated spectra and indicate on the display how much of each spectrum it was able to follow. It then fits a polynomial (10th order usually works best) to each and writes the resulting coefficients into a disk file.
(4) Run ‘cdist’ on the image to be corrected. This will read in the results from the previous step and re-bin the image data as indicated by the ‘sdist’ analysis. To determine the distortion values at points between the analysed spectra, for each column in the image ‘cdist’ fits a low order polynomial to positions of the fitted spectra in that column. Obviously, if only one spectrum were used for the analysis ‘cdist’ will have to assume that the distortion is constant along the slit. A good thing to try is to attempt to correct the calibration image. If it doesn’t end up with perfectly horizontal spectra then something has gone wrong.

```
ICL> image holes reset high=xxxx low=xxxx accept
ICL> icur    { using space bar to indicate spectra to be used
ICL> sdist image=holes columns=9 width=2 maxdeg=10
ICL> cdist image=holes ystart=min yend=max output=test maxdegy=5
ICL> image test accept
ICL> cdist image=object ystart=min yend=max output=objc maxdegy=5
```

There are a couple of decisions that have to be made in this sequence. Firstly, there is the question of just how many of the hole spectra should be used. Generally the answer is that all should be used, unless they are going to cause problems—a spectrum that gets too close to the edge of the image will probably not be traced properly, for example. Then there is the use of the ‘columns’ parameter in ‘sdist’ to help control the tracing of the data. In most cases, the main point about the ‘columns’ parameter is that making it larger reduces the number of points to which the polynomials are fitted, and so speeds up the process considerably. However, if the spectra are particularly weak, increasing ‘columns’ will also improve the signal to noise of the cross-section profile of the spectra. (‘sdist’ sums the data over the specified number of columns, then tries to locate the centre of the resulting profile, assuming that the best starting guess for the centre is the value obtained from the previous fit.)

‘sdist’ can use a number of different algorithms for tracing the spectra, selected by the ‘trace’ parameter. The original algorithm used by ‘sdist’ is the G(aussian) option, and this is most suitable if the profile of the summed data is roughly Gaussian (as it usually is for stars observed using a slit). If the profile is roughly ‘top hat’ in profile (if a dekker has been used, for example), then either E(dge) or C(enter of gravity) will probably be better. Edge fitting tends to produce rather jumpy fits, particularly if the edges are very sharp and therefore cannot be located to better than a pixel. The centre of gravity fit (which takes the COG of the data within the edges) is usually smoother, but can be inaccurate if the top of the data is not flat. A ‘ystract’ through the data, followed by a ‘splo’ can give a good feel for the data profiles, and the diagnostic display produced by selecting the ‘softd’ option can be very helpful here. The information produced by specifying the ‘diagnostic’ keyword is really for debugging new trace algorithms and is unlikely to be of general use.

In the end, there is no substitute for watching the results on the display. Remember that the polynomial will be unconstrained outside the range where it could trace the spectrum, and this may well show up in the final results from ‘cdist’. So see if the spectra seem to have been fitted acceptably, and if not, either ignore the bad ones or try to fine tune the fit using the ‘sdist’ parameters. The display that ‘sdist’ can produce on a line graphics device is a useful diagnostic. (In Figaro 3.0 ‘sdist’ could overlay the fit on a previous display done with ‘image’. Sadly this option is not available in the current version.)

‘offdist’ may be useful if there is a slight linear offset in Y between the calibration data and the data to be corrected. This is not usually important if the data is to be corrected by ‘cdist’,
but other routines such as ‘maskext’ can make use of ‘sdist’ results in a way that makes such offsets (often the result of guiding changes) important. ‘offdist’ adds an offset in Y to the results produced by ‘sdist’.

The final step, the correction of the object data, is an automatic process. So if a number of images are to be corrected there is a lot to be said for doing this as a batch job.

### 4.7.2 Self-correcting objects

Image tubes are not always tremendously stable; it may not be reasonable to assume that the details of the distortion do not change with time. This means that if there is only one multi-hole calibration image, probably taken at the start of the night, a distortion analysis based on this single early image may become progressively less correct as images from later and later in the night are processed. And, of course, there is always the possibility that no such image was taken at all.

In these cases, it is always possible to produce a distortion analysis from a spectrum of a single point object. An image of such an object can be used to calibrate itself, from a distortion point of view. The disadvantage of this over the use of a multi-hole exposure is that the correction based on a single object will be exactly correct only at the position of that object in the slit—the sky nearby will not be quite so well corrected.

So the choice of using objects to calibrate themselves, against the use of a multi-hole calibration image, is a trade-off of one source of error against another. The choice has to be yours.

### 4.7.3 Use of ‘cdist’ parameter ‘rotate’

The operation performed by ‘cdist’ involves working on the image column by column. On a virtual memory machine, this is to commit the cardinal sin of accessing an array in the wrong order—against the grain of the virtual memory system. When large images are processed, the result will be excessive page faulting by the process, together with a dramatic increase in the time taken (both CPU time and elapsed time go up), and an even more dramatic increase in the anger levels of the other users of the machine, since the excessive paging will begin to saturate the disk I/O system. This condition is described as ‘thrashing’.

To reduce this faulting, it is possible to rotate the image before it is processed and rotate it back afterwards. ‘cdist’ has a hidden parameter ‘rotate’ which makes it do this automatically. The final results obtained are the same whether or not ‘rotate’ is specified, but the efficiency of the operation can be quite different. Specifying ‘rotate’ adds the overheads of the two rotations, but makes the correction work properly with the virtual memory system. For small images, where the correction algorithm probably does not induce thrashing anyway, use of pre and post rotation will be inefficient since it adds the overheads of the two rotations. However, these overheads are small for small images, and it is probably acceptable to always specify ‘rotate’. This is why ‘rotate’ is a hidden parameter, true by default.

Just what constitutes a ‘small’ or a ‘large’ image—i.e. the size of image at which one starts to gain by specifying ‘rotate’—is hard to say. The best guide is trial and error. As it usually is.

### 4.7.4 Making do without an image display

Firstly, if you cannot display grey-scale or colour, you still can use ‘icont’ on any line graphics terminal. Its display is equivalent to that of ‘igrey’ and can be used by ‘igcur’ (not ‘icur’).
It is—just—possible to make do without any display at all, should one not be available. ‘sdist’
picks up the number of spectra and the pixel coordinates from global parameters ‘npixels’,
‘xpixels’ and ‘ypixels’. You can create these with ‘creobj’, or delete them with ‘delobj’ if necessary.
‘xpixels’ and ‘ypixels’ are vectors, if these are too short, delete and re-create them. Once created
you can assign values to these parameters using ‘setobj’:

```icl
ICL> delobj object=$ADAM_USER/GLOBAL.NPIXELS
ICL> delobj object=$ADAM_USER/GLOBAL.XPIXELS
ICL> delobj object=$ADAM_USER/GLOBAL.YPIXELS
ICL> creobj type=_REAL dims=0 object=$ADAM_USER/GLOBAL.NPIXELS
ICL> creobj type=_REAL dims=8 object=$ADAM_USER/GLOBAL.XPIXELS
ICL> creobj type=_REAL dims=8 object=$ADAM_USER/GLOBAL.YPIXELS
ICL> setobj value=2 object=$ADAM_USER/GLOBAL.NPIXELS
ICL> setobj value=15 object=$ADAM_USER/GLOBAL.XPIXELS(1)
ICL> setobj value=21 object=$ADAM_USER/GLOBAL.YPIXELS(1)
ICL> setobj value=25 object=$ADAM_USER/GLOBAL.XPIXELS(2)
ICL> setobj value=35 object=$ADAM_USER/GLOBAL.YPIXELS(2)
```

(If ‘ADAM_USER’ is not set, use ‘$HOME/adam’ instead.)

4.8 Wavelength calibration

Simple statements such as ‘the wavelength of channel n is so many Angstroms’ are not as
unambiguous as they appear. This statement is really a slightly simplified version of ‘the
wavelength range covered by channel n is from so many Angstroms to so many Angstroms’
but some precision has been lost in the simplification. It presumably means ‘the wavelength
of the centre of the range covered by channel n is so many Angstroms’—or does it mean that ‘the
wavelength of the start of the range covered by channel n is so many Angstroms’?

These notes are intended to explain the conventions used by Figaro. Please note that they are
all somewhat arbitrary and in some cases are historical rather than ideal. However, they are
consistent and they are precisely defined. Hopefully, these notes will also serve as a brief guide
to the use of Figaro for wavelength calibrations. If they seem a trifle pedantic, this is because
experience has shown that the whole subject is a real can of worms.

Arc-fitting is also available with Specdre (Section B.7) which is now part of Figaro. It is reputed
to be far superior to that described in this section. It is currently only useable with Specdre
commands but not with other Figaro commands.

Another alternative for arc-fitting is Molly, a semi-interactive wavelength calibration program
written by Tom Marsh. It allows the first of a series of similar arcs to be calibrated interactively,
then calibrates the rest automatically. In the right circumstances it can be a great time-saver.

4.8.1 Discrete and continuous channel numbers

Most of the problems arise because

1. It is impossible to get away from the fact that data is held in a number of discrete ‘bins’ or
‘channels’, each of which has to be given an integer channel number (to be called ‘n’ in
these notes), and each of which originally covered a range on the detector.
(2) A centre of gravity analysis of an arc line, say, will not in general produce an integer channel number as a result. So there also has to be a ‘continuous channel number’ (called ‘x’ here).

(3) The channels also map onto a wavelength scale, and wavelength scales are most naturally thought of as continuous. It is therefore quite natural to think of a wavelength/channel number relationship of the form lambda(x) = f(x), where f(x) might be a polynomial in x. Note that Figaro does not actually store wavelength information using polynomial coefficients, unlike many systems.

The sad fact is that there is no obviously correct method of mapping the continuous channel numbers (‘x’) onto the integer channel numbers (‘n’). Mainly it’s a question of selecting the zero point. The Figaro convention is that

(1) The centre of channel n = 200 is at x = 200.00, and the channel therefore spans in x from x = 199.5 to x = 200.5.

(2) The ‘wavelength of a channel’ means the wavelength of the centre of that channel, i.e. lambda(n) = f(x) where x = n.

This all sounds mind-blowingly trivial, but it is possible to produce other conventions. In particular, if the left edge of channel 1 were taken as the x = 0.0 point, the centre of channel 200 would be at x = 199.5. This is in fact the convention for the NDF format, used by Figaro. It is of little practical consequence: when pixel coordinates matter, they are never implied from pixel numbers but stored explicitly. The n-th stored value always applies to the centre of the n-th pixel.

4.8.2 Wavelength arrays

Figaro tries to hold wavelength information in as general a way as possible. If a file called, say, ‘spect’ contains a wavelength-calibrated spectrum, then there will be two data arrays in the structure held in the file. The main data array will be an array containing the actual spectrum, and the X-axis array will be an array of related size containing the wavelength information.

Each element of the X-axis array contains the wavelength value for the corresponding element of the spectrum—that is, for the centre of the corresponding data bin. For uncalibrated data, the X-axis array will usually not exist.

The ‘hdstrace’ command may be used to examine elements of either the data or wavelength arrays, and the ‘ilist’ command can be used to tabulate them.

4.8.3 The ‘arc’ command

A single arc spectrum can be generated by summing successive rows of an arc image, using the ‘extract’ command. The wavelength to channel number relationship can then be determined using the ‘arc’ command. This invokes an interactive line identification program which gets the user to select lines in the arc and to specify their wavelengths. ‘arc’ makes full use of interactive graphics and a running fit to make the manual identification process as easy as possible, and it also has an automatic line identification feature that can be used to add lines to a fit. ‘arc’ is described in more detail in Section 4.10.
Once a satisfactory fit has been obtained, ‘arc’ can be used to set the wavelength scale in the arc spectrum being identified. This is the importance of the question ‘make use of this fit?’ that the user is asked at the end of ‘arc’. Once the wavelength scale has been set, subsequent plots of that arc spectrum made by ‘splot’ will be made with a scale in wavelength units.

Note that Figaro programs are all designed to work with data whose X scale either increases or decreases. However, as a practical point, it is regarded as more usual to arrange data so that wavelength increases with pixel number, and there may be bugs still lurking in some routines that will only turn up when used with reversed data. You are therefore strongly recommended to use increasing X scales, if necessary using the ‘irevx’ command to reverse spectra and images.

If ‘arc’ is used to fit a spectrum very similar to the one previously fitted, it is enough to make use of the previous fit (by answering ‘yes’ to the ‘previous’ prompt) and then re-identify a line used in the previous fit. ‘arc’ will ask if the previous identification is to be ignored, or the new identification, and as a final option will assume that there is a shift in the data relative to the identifications and will re-analyse the line centres accordingly. This makes the analysis of a sequence of arcs quite simple.

### 4.8.4 Applying an arc fit to other spectra

Of course, the point of getting such a fit is to apply it to other spectra. The command ‘xcopy’ copies the wavelength scale from one spectrum to another. Normally the spectrum whose wavelength scale is copied is a fitted arc and the spectrum to which it is copied is that of an observed object.

Typically, an observation of an object is bracketed by arc observations. These different arcs will inevitably give slightly different fits, either just because of random variations, or because of some instrumental drift. Particularly if there is drift involved, it may be better to use ‘xcopi’ rather than ‘xcopy’. ‘xcopi’ takes as input two spectra with wavelength scales and generates a new scale that is an interpolation between them. It then applies that interpolated scale to the object spectrum.

### 4.8.5 Linear wavelength scales—scrunching

It is possible to rebin data whose wavelength/channel number relation is known in such a way that the relation is linear. Such an operation is known as ‘scrunching’, and is performed by the ‘scrunch’ command. Scrunching data is generally easier to handle than is un-scrunched. For example, if spectra with slightly differing wavelength scales must be added together they should be scrunched to a common wavelength scale first.

Similarly, there are cases where one needs data rebinned on a logarithmic wavelength scale—cross-correlations to determine redshift need this sort of scale, for example—and this can also be performed using ‘scrunch’.

‘scrunch’ has parameters that specify the start and end wavelengths for the resulting scrunched spectrum, and the number of elements (bins) it should have. Note that the wavelengths are those of the ‘centres’ of the end bins; it is easy to miscalculate the number of bins by 1 when aiming to get a linear dispersion of exactly so many Angstroms per bin. The target wavelength range may also be specified in terms of the start wavelength and the increment, by means of the rather crude convention that if the end wavelength specified is less than the starting wavelength, then that ‘end wavelength’ value will be taken as the wavelength increment. (This behaviour
may be controlled explicitly by two keywords ‘final’ and ‘increment’, which are used to specify the interpretation to be placed on the ‘end wavelength’ value.)

‘scrunch’ has one keyword that can confuse users. If the spectrum being scrunched is in flux units, then the total flux in the spectrum should be conserved by the scrunching process. Note that the distinction is between flux units and flux density units, so ‘raw counts’ are a flux unit. If a spectrum is in flux density units, then scrunching should not change the absolute value of the data. If the scrunching is simply such that it doubles the number of bins in the spectrum, then an element that has a value of, say, 10 mJy should become two elements each with a value of 10 mJy, since Janskys are a flux density unit. Conversely, if the spectrum were still in raw counts, then an element with 10 counts should become two elements each with 5 counts. The default for the ‘flux’ keyword in ‘scrunch’—whose prompt is the perhaps confusing ‘conserve flux (as opposed to mean counts)?’ is ‘yes’, and this is correct for raw counts. To confuse matters further, there is a concurrent parameter ‘mean’ with the opposite meaning to ‘flux’.

4.8.6 Two-dimensional scrunching

The previous sections have all been concerned with single spectra, and this is usually satisfactory where images can easily be reduced to spectra by use of ‘extract’—i.e. where there is no significant change in wavelength/channel relationship across the subset of the image that is being collapsed into a single spectrum. However, there is an alternative approach, which is to scrunch each cross section of an image separately. This may be needed for data on extended objects, or it may simply be regarded as a better (more strictly correct) means of proceeding.

To do this, one needs a good fit to every cross-section of an arc image. This is performed automatically by ‘iarc’, starting off from a manual fit produced by ‘arc’. The usual technique is to extract a spectrum from the centre of the image, probably adding together a few cross-sections to get good signal to noise, and then fit that using ‘arc’. It is important to have as many lines as possible in the fit; it is even worth adding lines in any sparse areas to help lock down the fit, even if their identifications are uncertain—just treat the fitted values as exact. ‘iarc’ does have an automatic search for such ‘lock’ lines, but it is usually better to pick good ones manually.

By default, ‘iarc’ performs its analysis using the same line width parameter, ‘sigma’, as was used in the original fit performed by ‘arc’. However, the algorithm used by ‘iarc’ is quite sensitive to the value of this parameter, and better fits may sometimes be obtained by varying it. For this reason, ‘iarc’ has a hidden parameter ‘rsigma’ which may be used to over-ride the ‘arc’ value. If you have problems with ‘iarc’, there is a hidden keyword, ‘detail’, which causes the details of the fit for each cross-section to be put out. This can be used to see which lines the program is having trouble with. ‘iarc’ will drop a line from its search list if it fails to find it in a a given number of successive cross-sections—this is to prevent it accidentally picking up the wrong line due to distortion moving another line towards the position where it lost the original line. The number of cross-sections is given by the hidden parameter ‘gap’—normally, ‘gap’ is set to 1, meaning that the second time in succession a line is missed, it will be dropped. Setting ‘gap’ sufficiently high will effectively disable this feature—this may be the best way to deal with data where there is little distortion, but where a number of cross-sections may have poor data; fibre data can be of this type. If ‘gap’ is set to zero, a line will be dropped immediately if it cannot be found in a spectrum. In order to pick up as many lines as possible in each cross-section, ‘iarc’ uses a technique known as ‘spreading’—it first looks for the lines using a larger sigma value than that specified, and then refines the fits using the specified sigma value. In some cases this is undesirable, and may be bypassed by the use of the hidden keyword ‘nospread’.
Sets of spectra which have been taken in some long slit mode with a distorting detector, such as an image tube, will differ from spectrum to spectrum but will do so smoothly. Sets of spectra taken with a fibre system may vary discontinuously, with sudden linear shifts between spectra. The ‘xcorr’ keyword in ‘iarc’ is designed to be used with such discontinuous spectra. It causes ‘iarc’ to attempt to determine a linear shift between successive spectra by cross-correlation and to apply that shift to its expected arc line positions before searching for them in the new spectrum. ‘xcorr’ should be used for fibre and similar data, but is probably inappropriate for other data.

Given a fit from ‘iarc’, ‘iscrunch’ can be used to scrunch an image of an object. It is actually a good idea to scrunch the original arc and see what the results look like—lines waving about in some regions show that the fit is badly constrained in those regions. ‘iarc’ only performs a sequence of individual one-dimensional arc fits, rather than a two-dimensional fit to the whole image at once, so there is little constraint that the fits be continuous from one cross section to the next, other than that imposed by the lines themselves.

Note that the ‘iarc’ results can only be used by ‘iscrunch’; they are not written into the data structure in a way that can be used by other routines such as ‘splot’.

For object images bracketed by arc images, ‘isruni’ can be used instead of ‘iscrunch’. This uses wavelength values obtained by interpolation between two different ‘iarc’ fits, in an similar way to ‘xcopi’ as described earlier.

### 4.8.7 Sequence summary

For individual spectra:

- **extract**: get spectra from image. Both arcs and objects.
- **arc**: set wavelength scale on arc spectra.
- **xcopy/xcopi**: copy wavelength scales to objects from arc(s).
- **scrunch**: reduce spectra to linear/log wavelength scales.

For images:

- **extract**: get spectrum from centre of arc image.
- **arc**: set wavelength scale on arc spectra.
- **iarc**: automatic fit to all cross-sections of arc image.
- **iscrunch**: reduce images to linear/log wavelength scale.
- **extract**: get spectra from scrunched images.
4.9 Extinction

The ‘extin’ command is used to correct a spectrum for atmospheric extinction. It requires a ‘coefficient’ spectrum—a spectrum giving the atmospheric extinction coefficient for each element of the spectrum. This coefficient spectrum is normally generated by interpolating a spiketrum formed from a suitable table of extinction coefficients.

One such table is ‘extin’, (actually the file ‘extin.tab’ in the FIGARO_PROG_S directory). This contains the coefficients given for Palomar by Hayes & Latham (1975). So for Palomar data, the following sequence will perform the extinction correction:

```icl
ICL> gspike spectrum=object table=extin spiketrum=extspike
ICL> linterp spiketrum=extspike spectrum=extcal
ICL> extin spectrum=object coeff=extcal output=cobject
```

generating a corrected spectrum, ‘cobject’, from the original uncorrected spectrum, ‘object’. The first two steps do not need to be repeated for subsequent spectra, so long as the wavelength range covered remains unchanged.

(An alternative extinction table is ‘palomar’, which contains the coefficients used by TYB’s Forth system. This differs slightly from ‘extin’, particularly around 9000 Angstroms, where it attempts to correct for some atmospheric features ignored by Hayes & Latham. The differences can be seen by generating and then plotting the spiketra produced by the two tables.)

The ‘aaoext’ table gives the standard AAO extinction coefficients—those used by the SDRSYS data reduction system.

4.9.1 Reference


4.10 Arc—A Figaro program for arc wavelength calibration

This section describes the use of the ‘arc’ program in Figaro. ‘arc’ is an interactive arc fitter, which displays an arc spectrum on the current soft graphics device and gets the user to indicate arc lines and enter their wavelengths. Given sufficient lines, a polynomial fit of wavelength against channel number can be performed, and the results of this fit used to fill the wavelength array (the .AXIS(1).DATA_ARRAY). Section 4.8 describes the way ‘arc’ fits can be used.

‘arc’ is primarily designed as an interactive arc fitter, but it does have an automatic line finding capability. This description will emphasise the interactive aspects of ‘arc’, however, since this automatic capability is intended to help add lines to an already good fit, and so the first and most important thing is to get a good fit manually.

‘arc’ has a number of features that are intended to make it particularly simple to use. These are not always obvious to the user who simply types the command ‘arc’ and waits to see what happens next; hence this description.
4.10.1 Arc line lists

To get the best out of ‘arc’ you need a comprehensive line list for the arc you are trying to identify. Arc line lists are text files containing lists of arc line wavelengths. They have the extension ‘.arc’ and are usually held in the main Figaro directory (FIGARO_PROG_S; see Section 3.4). There may also be files in the local Figaro directory (FIGARO_PROG_L), or you, the user, may have your own in your user Figaro directory (FIGARO_PROG_U), or in the default directory. As an example, the file ‘argon.arc’ in the main Figaro directory, begins:

```
* Argon lines
3243.6887
3249.8003
3281.7016
3307.2283
3350.9243
3376.4359
3388.5309
3454.0952
3476.7474
3478.2324
3480.5055
```

and carries on in the same vein for quite some time. The format is very simple—each line of the file has a wavelength, in free-format, and blank lines and lines beginning ‘*’ are ignored. Note that no line strength information is used. The importance of this line list will become clearer as we go on.

When the ‘arc’ command is given, one of the first prompts asks ‘Type of arc?’ ‘arc’ wants to know which arc line lists to read in. You can specify up to three arc types, although it is unusual to use more than one. If your reply is ‘arc1,arc2,arc3’, then ‘arc’ will look in the various Figaro directories for the files ‘arc1.arc’, ‘arc2.arc’ and ‘arc3.arc’. It will then read in the arc lines from all three. The idea here is that if you have, say, a copper-argon arc, you might give the type as ‘copper,argon’, and ‘arc’ can make use of two separate files, one for each element. In practice, it would be better to have a single file called ‘cuar.arc’, set up specially for the arc you are using, and to reply ‘cuar’. If you have no suitable arc file, reply ‘NONE’.

4.10.2 Other initial prompts

‘arc’ will also prompt for the arc line half width in pixels (the parameter called ‘sigma’), which it uses as a guide when finding the centres of the arc lines, and for the initial order of polynomial to use for the fit. ‘arc’ performs running fits, and unless you are using a previous line list (see below), the first fit will have to be made to a low order, simply because there will not be enough lines for a higher order. Once enough lines have been identified, ‘arc’ will start to use the order you specified initially. You will be able to change the values for both ‘sigma’ and ‘order’ interactively during the fitting process.

‘arc’ will also ask if lines from the previous fit are to be used. During a fit, ‘arc’ keeps writing out to disk the positions and wavelengths of the lines identified so far. (‘arc’ always writes to the file ‘airlines.lis’, thus any such file should be deleted or renamed before you run ‘arc’.) If
you reply ‘yes’ to this prompt, ‘arc’ will read in the file giving the previous list of identified lines. This allows you to start again where you left off the previous time—either because ‘arc’ or the computer crashed (perish the thought!) during the previous fit, or simply because, on reflection, you are unhappy with the fit you obtained and want to experiment with other orders, a different selection, etc. The default for this previous list file is always ‘arlines.lis’, since that is the name of the file that ‘arc’ writes. However, you may specify a different file. You may, for example, have one file that you want to use as the basis for fits to a number of arcs, which you have renamed from ‘arlines.lis’ (the name ‘arc’ will have given it originally when it was produced) to, say, ‘basefit.lis’.

You can also use a previous fit if the arc you are identifying is very similar to the previous arc, but is shifted slightly. This is often the case with arcs taken at intervals throughout a night. ‘arc’ will notice if the file that was used to create the previous fit is different to the file you are fitting. If this is the case, you are prompted for the ‘xcorr’ keyword, which allows you to request that ‘arc’ locate the previous spectrum used and attempt to determine a linear shift by cross-correlating the two arcs and applying the determined shift to the arc line list. (If you are going to use this option, specifying a slightly larger ‘sigma’ than that used for the previous arc will give the line finding algorithm a little more flexibility when it looks for the lines at their shifted positions.) This is a simple operation, but the shift is determined over the whole of the arc. There is an alternative, messier, way to indicate a shift to ‘arc’ on the basis of a single selected line, but this is described later.

4.10.3 The line selection process

‘arc’ will read in the specified spectrum and display a portion of it on the graphics device. Initially, the portion will be 200 channels long; you can change this should you want to. You will be invited to use the cursor to indicate an arc line.

Normally, you move the cursor until it is close to a line whose wavelength you know (you will often find it useful to have a hard plot of the whole arc in front of you as you perform the fit) and select it by hitting the space bar. (Strictly, you can use any key that does not have some specific function, but there are rather a lot that do and the space bar is a safe one to use.) ‘arc’ will then try to find a line close to the point you have indicated, and if it finds one will show you its centre on the display with an arrow. The algorithm used to find a line centre is one described as ‘convolution with the derivative of a Gaussian’, and it incorporates some requirements as to just what constitutes a line, which can lead on rare occasions to its being unable to find a line centred near the position indicated. You can think of the algorithm as a fit to a Gaussian of fixed sigma.

You will be told the channel number of the line, and asked to enter a wavelength. Strictly, you enter a wavelength followed by a type, e.g.

3850 argon

The type should be one of the names specified in response to the ‘What type of Arc’ prompt. ‘arc’ then looks in the table for that type and selects the line whose wavelength is nearest to the one you specified. If you do not specify a type (which is by far the most usual case), ‘arc’ uses the first of the types. Since in most cases only one type was specified, this means that ‘arc’ will normally just look in the one table that it read in. Which is what you would expect. So the response is usually just a number, e.g.
‘arc’ will then tell you what line it assumes you mean:

Wavelength is 3350.924 OK? /YES/ >

If you reply ‘y’, ‘yes’, or just hit the return key, ‘arc’ will use that wavelength. If you reply ‘no’, or ‘n’, it will ask you for the wavelength again. If you just hit return in response to the wavelength prompt, the line will be deleted and you will be back with the cursor looking for another line.

4.10.4 Subtleties—this bit is worth reading!

There are some important alternative ways of specifying the wavelength of the line.

(1) If you know the wavelength exactly, but it is not in the line list, you can specify the type as ‘exact’. So if your response to the wavelength prompt is

3456.789 e

‘e’ being short for ‘exact’—‘arc’ will use that value as the wavelength. If you have specified ‘NONE’ as the arc type, ‘arc’ will assume that all wavelengths given are exact values.

(2) Once two lines have been identified, ‘arc’ is able to estimate the wavelength of a new line by linear interpolation. It will then tell you this interpolated wavelength as well as the channel number. If more than two lines have been identified, the interpolation is based on the two nearest to the new line. You can now use ‘i’ in your response instead of specifying a wavelength—the result is the same as if you had typed in the interpolated wavelength. ‘arc’ will then look for the listed wavelength nearest to the interpolated value.

(3) Similarly, once more than two lines have been identified, ‘arc’ is able to start performing a running fit to the identified lines. The fit is recalculated each time a new line is identified and the RMS of the fit is displayed. So a bad identification will usually show up immediately as a large rise in the RMS figure. ‘arc’ can now display the fitted wavelength as well as the interpolated wavelength when a new line is selected, and you can use ‘f’ for the fitted wavelength in the same way as you can use ‘i’.

What this means in practice, is that with a good line list, you can identify enough lines to tie down the fit fairly well (it helps to do some lines at one end, then at the other, then work in), and then just respond ‘f’ to each wavelength prompt. ‘arc’ will then use the line in the line list closest to the fitted wavelength. This simplifies the process considerably, but it is still under your control, and you can intervene if a new fit shows a considerable increase in RMS.

4.10.5 Moving about the spectrum and other commands

When you are asked to indicate a line using the cursor, you can hit—instead of the space bar, which just selects a line—any one of a number of keys that have specific functions.

For example, ‘n’ moves you on to the Next portion of the arc. ‘b’ moves you Back to the previous section. ‘m’ Moves you to a specific X value. (Note that the value is not always in channels—if
the spectrum already has wavelength data associated with it, the X values will be in Angstroms. This is sometimes a useful feature, sometimes an irritating one.)

This is how you move around from one section to another. ‘l’ lets you change the Length of the portion displayed at any one time.

‘d’ deletes the identified line nearest the cursor. If you see the RMS for the fit shoot up, hitting ‘d’ will delete the last line identified—so long as you haven’t moved the cursor since you selected it.

An important command is ‘q’, which quits the identification process and moves on to the fitting and line editing stage. Note that this does not commit you to anything—you can always come back to the interactive selection stage.

If the normal line centring algorithm cannot find a line, you can use ‘c’ to force a line identification where the line centre is determined by a centre-of-gravity analysis. This is a time-consuming operation, and is not really recommended for normal use.

Strongly not recommended is the use of ‘e’, which allows you to indicate the line centre yourself on an Expanded plot. This command does nothing clever at all—it just takes the position you indicate.

Like most Figaro programs that involve key-selected options, ‘arc’ will respond to ‘h’ or ‘?’ by printing a list of the options it accepts.

4.10.6 Fitting and editing—the menu stage

The ‘q’ command takes you to a more conventional stage where the fit is repeated and the results displayed. As soon as you hit ‘q’ and confirm that you indeed want to move to this next stage, a fit is performed to the lines identified so far and the results are displayed. For each line, the calculated and actual wavelengths are given, together with the discrepancy in Angstroms, and the RMS for the fit that would be obtained if that line were deleted from the list of lines. Finally, the RMS for the current fit is displayed. Remember that the RMS is in terms of Angstroms.

If the fit is bad, a look at the table of residuals output may help to indicate which identifications are at fault. The ‘RMS without this line’ is a particularly useful figure when there are relatively few lines in the fit; it saves you the effort of making a tentative deletion and a refit to see if the result is really an improvement. With a large number of lines, the RMS is less dependent on any one line, and this figure becomes less useful.

You are given a number of options at this point, all of which are listed in a single prompt. This is described as the ‘menu’ prompt. The options are as follows:

- **Fit** — Repeat the fit. This will show you the effects of any changes you have made, either by editing, or by use of the automatic search facility.

- **Disp** — Display the deviation of the fit from a linear fit. This shows the trend of the fit, and the errors in each line, and is a particularly valuable diagnostic. If your graphics device supports colour, those lines fitted automatically are shown in a different colour to those selected manually, and this can help to show if the automatic lines are really helping.

- **Order** — Change the order of the fit. A new fit is performed immediately.
• Edit — Delete or change the wavelength of one or more of the selected lines, without returning to the cursor selection. You have to specify the line(s) by line number, and are then prompted for the wavelength to use. A null wavelength will delete the line (and to get it back you will have to return to cursor selection).

• Reselect — Return to selection using the cursor.

• Print — Prints a copy of the fit (what ‘arlines.lis’ would look like if you were to exit now). Note that this is not quite the same as the display you get by doing a fit—it does not include the ‘RMS if omitted’ figures, for example. Eventually, it will be, but it was easier the way it is.

• Auto — Starting from your current fit and arc line list, ‘arc’ looks for additional lines in the arc at wavelengths given in the line list and adds any it finds to the identified line tables.

• Xauto — Deletes all the lines found by ‘Auto’

• Modify — Explains the function of the Autofit parameters and allows you to change them.

• Quit — Start to exit ‘arc’. See the section called ‘On the way out of ARC’ for a description of what happens next, which is not very much.

• Help — (or ?) Display a summary of this information.

The first letter of each command is sufficient.

4.10.7 The automatic line-finding facility

When you select the ‘Auto’ option in the menu, ‘arc’ tries to use the fit you have obtained already as a starting point and tries to find lines in the arc that match lines in the tables. The algorithm used is very simple, and is based on the principle that the automatic fit should not add lines that will make the fit significantly worse than do the lines you have already got—and are presumably happy with. There are only two parameters (at present) involved and, really, only one is important.

‘arc’ takes each pixel in the spectrum in turn. If that pixel is more than ‘chfact’ times the current sigma value from any line already found, it uses that pixel as the starting point for a line search. This is exactly as if you had selected that pixel with the cursor during the interactive part of the process. If anything resembling a line can be found, it calculates its wavelength and looks in the line tables for a line close to that wavelength.

A line is accepted if the discrepancy between its calculated and tabulated wavelength is less than ‘sigfact’ times the current RMS value. It is this that means that the criterion for accepting new lines is based on how their wavelength discrepancies compare with those for the lines that have already been accepted. ‘sigfact’ is the more important parameter of the two. The default value of 3.0 means that the automatic search can make the overall RMS of the fit somewhat worse, but it will give the program a fair go at finding some lines. Setting ‘sigfact’ to 1 or less, which you may do with the ‘Modify’ menu option, ensures that the automatic search will not make the fit worse, but it will probably not find many lines either.

Just how best to use the automatic line finder is a matter of experience and, probably, opinion. At the moment, it is a relatively new feature and so the experience is lacking, even if the opinion is
not. It does, however, seem fair to say that the better the original fit, the more likely the automatic
fit is to make correct rather than misleading identifications. The ‘Xauto’ menu option, which
causes all automatic line identifications to be deleted, at least means that you can experiment a
little without doing irreparable damage to your fit.

One approach is to let the automatic fit loose, and then tidy up after it, deleting those lines that
it found but that you don’t like the look of. You can do this from the line list produced by the
‘Fit’ menu option, using the ‘Edit’ option to delete the lines affecting the fit the most. Lines
found automatically are flagged in the list by a plus sign (and are shown in ‘arlines.lis’ with a
(A) symbol).

An alternative, and quite a simple operation, is to return to the interactive selection (the ‘Reselect’
menu option) and examine the lines found. The lines found by the Auto option are displayed
with their wavelengths in parentheses, so it is fairly straightforward to run through the spectrum
with the cursor, hitting ‘d’ at every bracketed line that looks like nothing more than a noise
bump.

4.10.8 On the way out of ‘arc’

If you do not return to the interactive stage, you will be asked if you want to make use of this
fit. If you reply ‘yes’, ‘arc’ will generate a set of wavelength values for the spectrum and either
create a new output spectrum with these values in the X data array, or simply put these values
in the X array of the current arc spectrum. See Section 4.8 for details of how you use these values.
If you are going on to ‘iarc’, you do not need to do this. If you are going on to ‘scrunch’, you do
need a spectrum with an array of X wavelengths.

Finally, you have the option of producing a hard plot showing the identifications you have
made. You can also produce a hard-copy of the dispersion curve—the plot produced by the
‘Disp’ menu option. Note that if you produce both plots, they will be produced as separate files,
both of which will have to be output to the hard copy device.

4.10.9 Working with shifted spectra

If you identify one arc, and then move on to another arc which is essentially the same except for
a linear shift, you can make use of the cross-correlation option (the ‘xcorr’ keyword described
earlier), or you can try the following sequence, which gives you a little more control over the
way the shift is determined:

(1) Reply ‘yes’ to the ‘Use results of previous fit?’ prompt.

(2) Reply ‘no’ to the ‘Determine shift by cross-correlation?’ prompt.

(3) The displays will now show the new arc, but with identifications that are offset by the
amount of the shift. The wavelengths will not match the lines properly.

(4) Select a line that was identified in the previous fit. When prompted for the wavelength,
respond with the correct wavelength for the line.

(5) Since this wavelength is that of a line already in the list of identified lines, ‘arc’ will not
accept it. It will ask if you want to delete the previous identification. You do not, since that
identification was correct. It will ask if you want to delete your new identification. You do
not, since it is correct. Answer ‘no’ to both questions.
(6) This leaves ‘arc’ with only one alternative. If both are correct, the arc must be shifted. So it asks if it is to re-identify the lines, assuming the apparent shift. If you reply ‘yes’, it effectively takes all the lines in its list, shifts them over and repeats the centring analysis using the shifted position as the starting point. End of arc identification, in theory.

(7) In practice, especially if the shift is not perfectly linear, some lines may be missed. These are logged, so you can see if this is happening. One trick is to increase the sigma (use the ‘s’ command) before forcing the re-identification. This gives the algorithm a little more scope. You can then reduce the sigma and force another re-fit, by again reselecting a known line. (This time the re-analysis will be assuming a shift of zero.) This trick is effectively that used by ‘iarc’ when working on a set of spectra.

4.11 Abline—A Figaro program for absorption line analysis

This section is more or less the documentation by J.G. Robertson, dated 2 February 1987 as included as on-line document in Figaro 3.0.

4.11.1 Introduction

‘abline’ is an interactive program which runs within the Figaro data reduction system, and whose main purpose is to find wavelengths and equivalent widths of absorption lines. The facility for fitting a polynomial to the continuum may also be of use in other situations. As well as wavelengths and equivalent widths the program estimates width and asymmetry parameters for each line.

The program does not fit Gaussian profiles to lines; in fact this approach is specifically avoided, since it results in model-dependent wavelength and equivalent width values. In practice one has no reason to think lines are even approximately Gaussian, e.g. the Voigt profiles of saturated or damped lines, or the multiple component lines typical of QSOs.

The wavelength estimator used is the median (i.e. the value that equally splits the area of the line). Not only is this more resistant to the effects of low level wings on the line than is the centroid (mean; centre of gravity) but also it can be shown that the median is actually a statistically better (less noisy) estimator than the centroid for locating absorption features. For further discussion of the estimators used for all the parameters see Publ. Astron. Soc. Pacific 98, 1220, (1986).

The program can also be used on emission lines, and will find median wavelength, equivalent width, line width and asymmetry. This version of the program does not calculate uncertainties of the wavelength or equivalent width.

4.11.2 Running ‘abline’

Before starting ‘abline’, make sure the soft and hard plot devices are appropriately specified, using the Figaro commands ‘soft’ and ‘hard’.

When you are in Figaro, just type ‘abline’ to run the program. Only interactive use (not batch) is feasible. ‘abline’ reads data from one-dimensional spectra only. There are three options for handling the continuum fit in the vicinity of the required line:
(1) Make a continuum fit, use it for analysis of one or more lines, but do not save the continuum fit.

(2) Make a continuum fit, use it as above and save the fit to the continuum as a separate spectrum.

(3) Read in continuum fit as written in (2) and use it instead of fitting it again. This mode also allows you to use a continuum fit you have constructed in some other way (check the X axes match!).

‘abline’ needs a graphics terminal for soft plots and cursor input (as specified by Figaro command ‘soft’). It will be useful to have a hard copy plot of the spectrum on hand before entering ‘abline’, so that you can see the approximate wavelengths of lines to be analysed, and reasonable values for the wavelength range over which to fit the continuum. ‘abline’ assumes that the .AXIS(1).DATA_ARRAY structure of the input spectrum gives the wavelength in Angstroms, although other units (e.g. velocity) will work if one suitably re-interprets the wavelength, equivalent width and line width outputs. The .AXIS(1).DATA_ARRAY structure must be linear (i.e. scrunched) because this is assumed in the calculations. Conversion of the .AXIS(1).DATA_ARRAY structure to vacuum heliocentric or LSR can be made using ‘vachel’ (and ‘scrunch’) if necessary, before using ‘abline’. For a noisy spectrum it may be useful to slightly smooth the data used as input to ‘abline’ (this does not bias the wavelength or equivalent width; it obviously does affect the width parameter and to a lesser extent the asymmetry).

‘abline’ has a number of parameters. These may be set in the command line, since they are ordinary Figaro parameters and keywords, but since ‘abline’ is inherently an interactive program it is expected that most users will just type ‘abline’ and allow themselves to be prompted by the program. The prompts in this case are as follows—

- ‘(SPectrum) Spectrum with lines to be fitted’.
  Figaro spectrum to be analysed.

- ‘(OLDcont) Use precomputed continuum?’
  Answer ‘y’ if you already have a continuum fit covering the wavelength region of the (next) line to be analysed, i.e. case (3) above. If you answer ‘n’ a continuum will be fitted, i.e. case (1) or (2) above.

If the continuum is to be fitted in this run, the system next asks:

- ‘(SIG) Multiple of sigma for continuum point rejection?’
  Points in the designated wavelength range(s) for continuum will be iteratively rejected if they are further from the fitted curve than this number of standard deviations. 2.25 seems to work quite well. If the value is too low it may reject valid noise features.

- ‘(ITN) Number of iterations for continuum point rejection’.
  About 4 seems adequate.

- ‘(DEG) Degree of polynomial for continuum fit’.
  The acceptable values are 0–7. Use the lowest degree which will be able to fit the believable trend of the continuum over the range used. Usually 1–3 is plenty. The higher degrees are intended for cases such as trying to fit a ‘continuum’ which includes a broad emission line.
Alternatively, if you elect to read in a pre-computed continuum, the program prompts

- '(C)ontin) File containing precomputed continuum'.
  This should be a normal Figaro one-dimensional spectrum, of the same dimensions as the
  spectrum to be analysed.

At this stage the program examines the .AXIS(1) structure of the input spectrum, and displays
its label and units for the user to check (e.g. they will usually be 'Wavelength' and 'Angstroms').
The first and last values of the .AXIS(1).DATA_ARRAY structure are also displayed for checking.

The next parameter requested from the user controls how the wavelength bounds of the absorp-
tion line will be found:

- '(L)imit) Set line limits at indicated points?'
  'yes' is the more usual answer. It means that you will define the bounds of each absorption
  line using the cursor to indicate the last channel (i.e. furthest from the line centre) on both
  sides which you want to be included in the line. It is not possible to split channels; the
  entire channel indicated by the cursor will be included. If you opt for 'n' then the last
  channel included on each side will be the first whose data value equals or exceeds the
  continuum value. You still have to use the cursor to indicate approximate lower and upper
  bounds of the line: be generous, i.e. go beyond the channels where you can see the data
  exceeds the continuum. If there is no channel whose data exceeds the continuum before
  the cursor specified position is reached, the latter will be used. For emission lines only the
  'y' option will work here.

The next parameter requested is

- '(W)idth) Wavelength range to display at one time'.
  This is the wavelength range which will be plotted on the soft plot. If you are going to
  do a continuum fit to this region, a fairly large range (i.e. going well beyond the desired
  line edges on both sides) is desirable, so that the continuum can be well tied down. (The
  continuum fit cannot extend beyond the range selected when the fit is done.) A range
  of about 200–500 channels (specified as the appropriate value in Angstroms) is usually
  satisfactory. When you come to delimit the line itself with the cursor (see below), you
  can elect to ‘zoom’ this plot by specifying a smaller wavelength range (Width). It is not
  acceptable for this width to be larger than the entire spectrum length, nor can it exceed an
  in-built limit of 2048 channels.

You are now asked whether you want to save the continuum fit(s) to a separate spectrum, and if
so, what the name of this spectrum should be. Normally one does not want to use the continuum
after the present ‘abline’ run; in this case answer ‘no’.

You now reach the loop in which individual lines (all from the same input spectrum) are
analysed. A help facility explains the available commands. The program prompts with

123.4,Sig,Deg,Itn,(No)Limit,Width,Cont,Fit,Recont,Quit,Help
The most usual response is a floating point number: This is the approximate wavelength of the next line to be analysed. After a further question to find the name of this line (which will label the hard and soft plots) a soft plot will be displayed. Its wavelength range will be as specified above for ‘wavelength range...’ and it will be centred at the wavelength specified here, unless this is too close to one end of the spectrum, in which case the plot will extend just to the end of the data. Further progress from this point is discussed below.

Other responses enable you to change the way the program fits the continuum or analyses the line. It is useful to be able to change these parameters here because different lines may require different treatment, and we do not want to have to re-enter the program to specify these. The other possible responses are (where xxx is a floating point number and nnn is an integer):

- **Sig xxx:**
  Set ‘multiple of sigma for continuum point rejection’ to xxx.

- **Deg nnn:**
  Set ‘degree of polynomial fit to continuum’ to nnn.

- **Itn nnn:**
  Set ‘number of iterations for continuum point rejection’ to nnn.

- **Wid xxx:**
  Set ‘wavelength range to display at one time’ to xxx.

- **Lim:**
  To cutoff integrations for equivalent width and wavelength at positions designated by cursor.

- **Nolim:**
  To cutoff at first channel on each side which equals or exceeds continuum

- **Cont:**
  Initiate continuum sub-segment selection and fitting (see below)

- **Fit:**
  Initiate cursor selection of line range and parameter calculation

- **Recont:**
  Recompute continuum from same selection of sub-segments as previous fit; e.g. after alteration of degree of polynomial

- **Quit:**
  Exit ‘abline’

- **Help:**
  Display explanation of the available commands

The changes made will remain in force for all subsequent lines analysed unless they are changed again.

At this stage in analysing a line you have a soft plot of the selected region. You now have to use the cursor to specify the exact channel ranges to be used for fitting the continuum (unless a
pre-computed continuum fit is being used). It is clearly necessary that the continuum fit does
not include the absorption line itself; there may also be other absorption lines or defects nearby
which you want to explicitly exclude from influencing the continuum fit. This is handled by
allowing you to specify up to 10 wavelength sub-segments (within the plotted range) which will
be used as input to the continuum fit. Type ‘cont’ and in answer to the rather extensive prompt
‘Selection of sub-segments for continuum fitting ....’ Do the following:

(1) Move the cursor to the lowest wavelength to be used in the continuum fit, and press any
key except Q, to read the cursor position.

(2) Move the cursor up in wavelength to the end of the first sub-segment you want to include
(e.g. just before an absorption line) and press any key except Q [unless this is the only
sub-segment, in which case press Q].

(3) Repeat for up to 10 sub-segments, specifying both lower and upper boundaries. Use Q to
terminate sub-segment input. Naturally you should choose sub-segments to span both
sides of the line to be fitted, so that the continuum is well tied down.

(4) Notes: If you specify a start position for a sub-segment lower in wavelength than the
end position of the previous one, it will be simply truncated (and a message output).
However if any sub-segment has an end wavelength less than its start wavelength, or if
any sub-segment totally encloses another, you will have to start the whole process again.

Having finished sub-segment selection, a polynomial fit is made to the specified continuum
channels. Channels whose data values deviate from the fitted curve by more than the ‘multiple
of sigma for continuum point rejection’ are rejected and the fit recomputed. This process is
repeated as specified by ‘number of iterations for continuum point rejection’. For the initial fit
and each subsequent iteration the terminal screen displays information about the fit. The right
hand two columns give the number of channels included in the fit on each iteration, and the
number rejected as being too far from the fitted curve. This number usually stabilises after a
few iterations, i.e. the process converges to a limit and further iterations result in no further
change. The first 8 columns of the display are the rms residuals of the (remaining) data points
relative to polynomial fits of degrees 0 to 7. This full table is produced no matter what degree of
polynomial fit you have selected. It enables you to see how low the degree of the polynomial
can be without obtaining a noticeably worse fit to the data.

The continuum fit is drawn on the soft plot; if you have elected to write the continuum to an
output Figaro file, the same data is written into the appropriate wavelength range of that file. It
will overwrite any previous fit in this same ‘abline’ run which included the same wavelength
range; i.e. all continuum fits in the one run go into the same output continuum file.

The next step is to use the cursor to specify the wavelength (channel) limits of the line. How this
is done depends on the response you gave to the Limit parameter. The usual case is ‘limit=yes’:
type ‘fit’ and centre the cursor on the last channel you want included, on each side of the line.
Do the lower wavelength edge first.

The limits of the line are then drawn on the soft plot as vertical lines; for clarity they are drawn
at the outer edges of the last channels included.

If the net area of the selected line is above the continuum rather than below it, the program
reports that this is an emission line.
4.11.3 Line parameter estimation

The program now has both the continuum and line specified, so it calculates and displays the quantities describing the line (for further details see Publ. Astron. Soc. Pacific 98, 1220 (1986)):

1. The median wavelength, as location parameter for the line in the same units as the .AXIS(1) structure of the input spectrum.

2. Equivalent width, in the same units (usually Angstroms). This is calculated using the data as normalised by the continuum, which is the best if the continuum is sloping significantly over the line width: i.e.

\[
EW = \sum_k \left( 1 - \frac{F(k)}{A(k)} \right) / ch
\]

where \( F(k) \) is the data value in channel \( k \) and \( A(k) \) is the value of the fitted continuum there. The sum is over the specified channel range. \( A/ch \) indicates the dispersion in Angstroms per channel.

3. The line width parameter is a measure of the width of the line (in Angstroms, or whatever is in the input axis data) and is calculated as follows. Let \( x(m) \) be the median wavelength, i.e.

\[
\sum_k \left( 1 - \frac{F(k)}{A(k)} \right) = 0.5 \frac{EW}{ch}
\]

Define \( x(l) \) and \( x(h) \) such that

\[
\sum_k \left( 1 - \frac{F(k)}{A(k)} \right) = 0.1587 \frac{EW}{ch}
\]

\[
\sum_k \left( 1 - \frac{F(k)}{A(k)} \right) = 0.8413 \frac{EW}{ch}
\]

(For \( x(m), x(l), x(h) \) linear interpolation is used to find the fractional part of the ‘channel’ satisfying the desired equation.)

The width parameter is

\[
\text{Line width parameter} = 1.1775 \left( x(h) - x(l) \right)
\]

For a line of Gaussian profile this gives exactly the FWHM, since \( x(l) \) and \( x(h) \) are at +/-1 standard deviation. For any reasonable centrally concentrated profile it gives a result very close to the FWHM. This definition is an attempt to obtain a parameter that is useful for a variety of line profiles. Note that the width parameter is considerably influenced by the cursor placement of the line boundaries, if these are at places where \( F(k) \) is not close to \( A(k) \). This is expected, and means that the width of the line as designated is partially under manual control. The line width parameter as defined above is less affected by low level wings far from the line centre than is the calculation of the width from the variance of the data about the mean, i.e. it refers more to the width of the core of the line.
(4) The relative displacements of \( x(l) \) and \( x(h) \) from the median \( x(m) \) carry some information about the asymmetry of the line profile, so an asymmetry parameter is calculated as follows:

\[
\text{asymmetry} = 100 \left( \frac{(x(h) + x(l))}{2} - x(m) \right) / \text{Line width parameter}
\]

The result is a dimensionless parameter (in percent), normalised by the line width. As an extreme case, a triangular line with a vertical low wavelength side has asymmetry of +8.1; if the high wavelength side is vertical, asymmetry -8.1. Higher values may result from deep narrow lines with prominent (noise) wings on one side. In general one expects the asymmetry parameter to be strongly affected by noise, and for many lines of low to moderate signal to noise it will be meaningless. Even the width parameter is rather noisy and should be interpreted carefully.

4.11.4 Conclusion

At this stage there is a short delay while the hard-copy plot is written to the designated file (see above for information about how to plot this after the run). The hard-copy gives a plot of the same wavelength range as selected and plotted on the soft plot; it shows the input data spectrum, the continuum fit, and two vertical lines bracketing the channel range selected for the line itself. A printed description on the same page as the plot lists all four parameters from analysis of the line (wavelength, equivalent width etc.) and also some of the details regarding the origin of the data and the type of continuum fit.

The program then loops back and asks for another centre wavelength for analysis. Give ‘quit’ to terminate if there are no other lines to analyse.

4.12 Gauss—A Figaro program for interactive Gaussian fitting

This section is more or less the documentation by Jeremy Walsh as included as on-line document in Figaro 3.0.

4.12.1 Introduction

‘gauss’ is a Figaro program for interactive fitting of Gaussians to an emission or absorption line spectrum. The program is suitable for handling both low resolution data, where line fluxes of many lines are required, or high resolution data where a multiple Gaussian fit to a single line profile is obtained. The line-free continuum is fitted by a polynomial of desired order with iterative rejection of points greater than a given distance from the fit. The interactive line fit is optimised by minimising the sum of squared residuals between the observed and fitted profiles. The results of the fit are written to a data file. The fitting profile can be saved as a spectrum.

4.12.2 Running the program

‘gauss’ requires an input spectrum in which the data points are equally spaced in wavelength. If you supply a spectrum with unequally spaced points it will abort with an error message. A spectrum with unequally spaced points can be converted to one with equally spaced points using Figaro command ‘scrunch’ (see Section 4.8.5).
Before running ‘gauss’ make sure that a soft and a hard plot device have been allocated using the Figaro commands ‘soft’ and ‘hard’. These are required for graphics (although use of the ‘hard’ device is not mandatory). The program will fail if ‘soft’ has not been specified.

The program will prompt for the following:

- **Name of spectrum file**
  Name of spectrum file to be fitted. Must be 1-d. If no errors are available (.VARIANCE) then a message is written to the terminal to this effect and the goodness of fit will be in terms of rms only. If ‘soft’ and ‘hard’ have not been set, then the program will terminate at this point.

- **Label for plot**
  Label written to top of soft and hard plot, and recorded in the results data file.

- **Are Gaussian fit results to be recorded on a file**
  If ‘yes’ then name of file is prompted for. If ‘no’ results will be written to terminal only.

- **Name of data file for results**
  This is the name of the data file (.dat) to which results are written. If a file of this name already exists then the results are simply appended to this file. If not, then a new file of this name is created.

- **Use whole of spectrum for line analysis**
  If ‘yes’ then all the spectrum is displayed. If ‘no’ the ‘xstart’ and ‘xend’ of the region to be displayed is prompted for (cf. ‘splot’).

- **Scale so all of spectrum fits**
  If ‘yes’ then whole Y extent plotted. If not ‘high’, ‘low’ and ‘bias’ are prompted for (cf. ‘splot’).

The spectrum is then plotted on the soft device in the lower box. If errors are available, then these are displayed in the upper box. The upper box is used for residuals on line and continuum fit. The scalar for the residuals indicates the ratio between the range of spectrum data to the range of residuals data, i.e. smaller errors and residuals have a larger scalar on the residuals box.

The continuum fitting menu next appears. The line or lines are demarcated from the continuum using the cursor, the parameters of the polynomial fitting set, and the fit performed. When satisfactory the fit and residuals are plotted ready for Gaussian fitting.

The options are as follows:

- **CUR**: Use the cursor to indicate the left and right edges of one or more regions of continuum. There is no limit to the number of continuum regions that can be so demarcated. Usually these regions will bracket the lines of interest.

- **ORD**: Order of polynomial fit to continuum points.

- **SIG**: Factor times sigma on last continuum fit such that points whose deviation from the last fit exceed this value are not used for the subsequent fits.
• ERR: Factor times the individual error on a point such that a point whose deviation from the last fit exceeds this value is not used in subsequent fits (only usable if errors are available)

• ITN: Number of iterations performed for rejecting points with deviations exceeding the rejection value

• FIT: Apply the polynomial fit specified by ORD, ITN and SIG/ERR to the continuum. (CUR must have been set for this to occur.) If ORD, SIG, ERR, or ITN have not been set then the default values are 3, 2.0, 1.0, and 1 respectively. For each iteration, the iteration number, the rms on the fit for each order (up to a maximum of 7), the number of rejected points and the number of fitted points are printed to the terminal (cf. `abline`). In the case when errors are available, the mean value of the deviation as a factor times the actual error bar is printed out as a function of the order. The aim is to take the lowest order and minimum number of iterations so that the rms (or mean fractional error) and number of rejected points level out. The values of SIG/ERR, ORD and ITN can be altered until this condition is achieved. The continuum fit highest order, highest iteration number fit is plotted on the ‘soft’ device from the left edge of the left section to the right edge of the right section. The residuals on the continuum fit appear in the residuals box over the extent of the fitted continuum (excluding the line(s)).

• GAU: Continuum fitting complete; move to GAussian fitting.

The Gaussian fitting menu now appears. The line or lines are simulated by one or more Gaussians.

The options are as follows:

• LIM: Left and right edges of observed profile to be fitted are indicated by the cursor. If this option is omitted the profile edges are taken as the nearest edges of the continuum sections, and a warning to this effect is issued.

• SIN: Fit a single Gaussian to a profile. This is a quick way to fit a line by a single Gaussian involving minimum user interaction. With the cursor the peak position of the line is indicated. The peak height of this line is found and the half height points taken to get the line width. These initial values are then used to optimise the fit. If the fit is successful then the results—peak position, peak height, sigma (= FWHM/2.3540), flux in the line, equivalent width of the line and the rms deviation on the fit (mean deviation in terms of the error bars if errors are available) are written to the screen and to the data file if appropriate. See OPT below for details of what happens if the optimisation fails.

• NEW: Introduce a Gaussian at a cursor defined position, of height the same as the data value and width 4 X channels.

• NEX: Introduce another Gaussian to a profile where one or more Gaussians have been fitted. A new Gaussian is introduced at the peak of the residuals (observed — fitted). This can have negative height (absorption line) in an emission profile, or vice versa, if residuals are small, so watch out. At any time at most ten Gaussian lines can be kept, an attempt to specify more lines is rejected.
• INCH: Interactively alter the position, height or width of a Gaussian. The latest Gaussian introduced, or the one SELECTed (see below) is changed thus:

  – P: position
  – H: height; followed by a number to give additive modification
  – W: width
  – S: stop the alteration

For example: ‘P100’ shifts a Gaussian right by the profile extent; ‘H-20’ decreases the Gaussian height by 20% of the peak height of the original line; W10 widens the line by 10% of the total profile extent. At each change the old fit is erased and the new fit is redrawn.

S stops the alteration and re-plots the whole display, including residuals, on the latest fit. The rms (or mean fractional deviation in terms of the error bars) on the fit is written to the terminal.

• LIS: Lists the fitting Gaussians in order of introduction, giving position, height, sigma and flux (proportional to height x FWHM).

• SEL: Set index number to that of the Gaussian whose parameters are to be modified. The list of fitting Gaussians is printed and selection made.

• DEL: Set index number to that of the Gaussian which is to be deleted. The list of fitting Gaussians is printed and selection made. The display is redrawn and the new sum of fitting Gaussians and their residuals are plotted.

• OPT: Optimises the fit. Any of the Gaussian parameters—position, height or width—can be constrained so that the value does not change during the optimisation process. If such single constraints are required then the index number of the Gaussian and the parameter to be constrained are entered in response to the ‘npcon’ prompt. Sets of Gaussians can also be ‘chained’ so that their values vary together: for example the separation of two Gaussians can be kept fixed when fitting a doublet with known separation. The index number of the Gaussians, the parameter to be chained and relation between the parameters is entered in reply to the ‘ichain’, ‘chain’ and ‘rchain’ prompts. Different sets of parameters can be chained, e.g. in a three Gaussian fit the positions of Gaussians 1 and 2 could be chained and the relative heights of Gaussians 2 and 3. However if a parameter of a Gaussian is already constrained it cannot be included in a chain.

The form of weighting for the residuals are prompted. Three options are available:

  – a) no weighting (i.e. unit weights);
  – b) weighting by signal value (without continuum subtraction);
  – c) weighting by inverse square of errors (when available).

By employing weights more emphasis on points with either higher value or lower errors can help to constrain the fit. Method c is analogous to minimising chi-squared. Success cannot be guaranteed with the minimisation routine. In particular if constraints or chaining are employed then the optimisation can be more troublesome.

Usually a small change to the interactive fit will enable a minimum to be found if a failure occurs. However don’t expect miracles. The algorithm only finds a local minimum. If you
don’t believe the fit, try another interactive fit and optimise again. The general rule is to try for a fit by a minimum number of Gaussians and to check that the deviations on the line fit are greater than or equal to the error bars, or the deviations on the continuum fit if errors not available.

If the optimisation is successful or the interactive fit is adopted, then position of peak, peak height, sigma and flux in each fitting Gaussian as well as the equivalent width on the whole profile and the simple (i.e. unweighted) rms deviation on the fit (mean deviation in terms of the error bars if errors are available) are written to the terminal and recorded on the data file (if applicable). On the ‘soft’ display, the fitting Gaussians, their sum and the residuals on the fit are displayed.

- **REC**: Recall a previous Gaussian fit to the profile. The name of the data file on which the fit was recorded is prompted for. The position, height and width of the recalled Gaussians, equivalent width and rms/mean error on the fit are written to the terminal and the individual Gaussians and their sum and residuals displayed. Any of the individual lines can then be altered by SEL and INCH. If any recalled line falls outside the line extent (which must have been set previously) a warning is issued and the line ignored. There is no safeguard against the case where more than ten lines are read from the file. You have to take care that this does not happen, otherwise the internal storage of ‘gauss’ gets messed up in an unpredictable way.

- **HARD**: Plot final fit on the specified hard-copy device. Produces a file which needs to be sent to the appropriate printer.

- **SAV**: Save the continuum and Gaussian fit spectrum. The sum of the Gaussian fit on the continuum is saved as a spectrum file. The name of this file will be prompted for on quitting from the Gaussian fitting menu.

- **CON**: Move on to next section of the spectrum for more line and continuum fitting (returns to point where ‘xstart’ and ‘xend’ are specified).

- **QUIT**: Quit from program (fit finished). If the fit is to be saved then the name of the output spectrum is prompted for:

  – **Fitted spectrum to be saved**
    Name of 1-d spectrum file to be produced. Note that the continuum on this spectrum only extends from the minimum to the maximum X values delimited in the continuum fitting (CUR), but the range of the X values is the same as that of the input spectrum.

### 4.12.3 Useful recipes

a) Low spectral resolution spectra (e.g. line strengths for single emission lines)

```
CUR ORD SIG/ERR ITN FIT GAU
LIM SIN CON
```

b) High spectral resolution spectra (e.g. radial velocity structure of a single profile)

```
CUR ORD SIG/ERR ITN FIT GAU
LIM NEW INCH NEX INCH NEX
INCH SEL INCH OPT HARD QUIT
```
4.12.4 Gauss: Format of data file

DATA ON GAUSSIAN FIT
---------------------
Spectrum file: TEST
Spectrum label:
X and E.W. units are Angstroms; Y units are Counts
Flux units are Counts * Angstroms

Gaussian Fit
Peak Posn. Peak Height Sigma Flux E.W. R.m.s. Mean frac err Order Sigma Rej. Frac err Rej.
6563.336 0.2151E+04 0.2740 0.1476E+04 -8.688 0.7339E+02 1.8157 3 2.000 1.000

The equivalent width tabulated is determined from the observed line profile and the continuum fit; it does not depend on any Gaussians fitted to the lines. The mean fractional deviation is defined as:

\[ \frac{1}{m} \sum_{j=n}^{n+m} \left| \frac{\text{obs} - \text{fit}}{\text{err}_j} \right| \]

4.13 Échelle reduction

The section on échelle reduction is more or less the documentation “UCL Echelle Spectrograph Data Reduction Software” by William Lupton, dated 19 September 1988, as included as on-line document in Figaro 3.0. The appendices are not included here.

This note consists of a simple description of the current state of the échelle reduction software in Figaro. Wherever possible the échelle reduction package kindly provided by Jim McCarthy of Caltech has been used, but in several cases it has been necessary to modify the programs or else to use different approaches.

The software is now nearly in a state where it, together with standard Figaro applications, contains all the facilities necessary for the reduction of échelle data. Further development has been performed by the échelle data reduction programmer at UCL who produced a general purpose échelle data reduction package Echomop.

The diagrams illustrate the recommended processing steps starting from raw CCD or IPCS data. They are accompany a worked example.

Note that no special facilities are provided for bias subtraction, background estimation, sky subtraction or flat-fielding. It is assumed that you will use standard Figaro facilities for these operations.

Most of the rest of this note consists of an annotated example of how to proceed from a raw échelle image to a wavelength calibrated single merged spectrum.

Assume that you have just moved the spectrograph to a new configuration and that you have taken images of a continuum source such as a bright, smooth spectrum star (for tracking the échelle orders), of the Th-Ar lamp (for wavelength calibration) and of your object. In many cases
the object spectra are sufficiently well exposed and free of discontinuities to allow direct order tracking. In such cases there will be no need for a separate continuum spectrum.

These files are in your current directory and are called:

- contraw.sdf - continuum
- arcrew.sdf - arc
- objraw.sdf - object

In what follows it will be useful to refer to the diagrams in the overview section.

### 4.13.1 Conventional orientation

![Diagram of conventional orientation for échelle reduction.](image)

Figure 10: Conventional orientation for échelle reduction.

The first thing to do is to ensure that your data is in the conventional orientation with wavelength increasing from left to right and from bottom to top of the image and thus with order number decreasing from bottom to top of the image.

IPCS data will already be in this orientation but unfortunately CCD data must be rotated and flipped. This is quite time-consuming but is at present necessary for all CCD images. The following commands achieve this for the continuum:

```
ICL> irot90 contraw contrawrot
ICL> irevy contrawrot cont
```

with analogous commands for the arc and the object. The raw files can now be deleted if desired.

In what follows, assume that the IPCS raw data files ‘contraw.sdf’ have been renamed to ‘cont.sdf’ etc.

### 4.13.2 Order location

At present locating the orders is rather an interactive process (there is a program ‘echfind’ that does it automatically but it does not yet work very well in all cases and its use is not recommended).

First decide whether you are going to use the continuum source or the object to locate the orders. It is a good idea first to do a ‘ystract’ / ‘splot’ of the data to get a feel for the width, intensity and profiles of the orders. Sensible commands to use for the GEC chip and the continuum are:
Now display the image using ‘image’:

```
ICL> image cont high=hhhh reset accept
```

The next stage is to use ‘icur’ to define a point somewhere near the peak of each order that you wish to track and extract. You can track orders that are only partially on the image if you wish to but this is not recommended, since it could well affect the wavelength calibration, especially if only a small part of the free spectral range is being covered. It is quite important to choose points close to the peak intensity.

```
ICL> icur
```

Now run ‘sdist’ to track the orders and fit polynomials to them:

```
ICL> sdist image=cont columns=8 trace=G width=3 maxdeg=10 softd=no
```

The two non-obvious parameters are ‘trace’ and ‘width’. Specify ‘Gaussian’ for ‘trace’ if the profiles across the orders are roughly Gaussian and are not cut off by the dekker. Normally specify ‘COG’ otherwise but if there is a noticeable gradient along the profile you can try ‘Edge’ (they are identical in that both locate the rising and falling edges of the orders, but ‘COG’ estimates the centre by calculating the centre of gravity and ‘Edge’ estimates it simply by taking the mean of the edge positions). For ‘width’, specify an estimate of the FWHM for ‘Gaussian’ and an estimate of half the order width for ‘COG’ and ‘Edge’. If anything, underestimate it for ‘Gaussian’ and overestimate it for ‘COG’ and ‘Edge’.

Beware that sky data can confuse ‘sdist’ because it gives rise to profiles that don’t fit any of the trace modes. If this appears to be a problem, use ‘clip’ (which sets all data values below a given
low value to that low value and sets all values above a given high value to that high value) to get rid of the sky data values that are causing the problem.

There is another program that may be useful here if you have moved to a new object that is not quite in the same place on the slit. 'offdist' operates on an ‘sdist.dat’ file and adjusts the constant terms so as to shift the tracked orders up or down by a specified amount.

4.13.3 Order extraction

You now have a choice. For quick-look extraction, you can create a mask image whose data values indicate which order (if any) each pixel belongs to. This mask is created by ‘echmask’ and applied by ‘maskext’. ‘echmask’ allows separate extraction of object and sky but requires the number of rows of object and sky data to be independent of order number. This is not acceptable when (as is usually the case) it is important to minimise the sky noise and to maximise the signal.

For final data reduction or where the use of a mask is not acceptable the orders can be straightened using ‘cdist’ (UCLES is extremely stable and preliminary results indicate that many images can be co-added prior to application of ‘cdist’, so the cost in processing time should be acceptable). Having straightened the orders, ‘echselect’ can be run to identify, for each order, the rows to be used for the object and those to be used for sky.
4.13.4 Quick-look extraction using a mask

As explained above, this method will not maximise signal to noise ratio, but it will do a reasonably good job, especially where the orders in question are not too far from being horizontally aligned on the detector.

4.13.5 Mask creation

‘sdist’ outputs an ‘sdist.dat’ file that contains details of the orders that it has tracked. This file is read by ‘echmask’, which produces a mask image that can be used for fast extraction of orders directly from (in the case of the CCD ‘irot90-ed’ and ‘irevy-d’) raw images. ‘echmask’ can cope with the case where the star / sky periscope is fitted and also allows you to specify the position of object and sky data relative to the centre of the order as determined by the tracking algorithm, but these details will be ignored here.

The normal straightforward behaviour is achieved by specifying ‘periscope’ false and giving zeroes for all widths and offsets. This causes a width derived from the ‘sdist’ fit to be used; if the fit was Gaussian the derived width is just twice the estimate that you gave to ‘sdist’ and if the fit was Edges it is the actual calculated width (the same width is used for each order and the third largest width of all the order widths is used so as to exclude atypical values). If you know better, you can specify your own value for ‘objwidth’—overestimate rather than underestimate so as to prevent noticeable jumps in the extracted data due to the slope and curvature of the orders.

The other thing that ‘echmask’ needs to know is the order numbers corresponding to the orders that it has tracked. The value that you give for ‘mstart’ is the order number corresponding to

The result in both cases is a ‘collapsed échellogram’ where X is wavelength and Y is order number. Your object image will of course give rise to both an object and a sky échellogram.
the first point that you selected with ‘icur’. There should always be an order near to the image
centre. Don’t worry if you get it wrong—you can always adjust the order number by using
‘icadd’ to add or subtract the error from the mask structure. You have to add or subtract ten
counts to adjust by one order, e.g. if the mask contains a data value of 420, this refers to order 42
and adding ten to it to make it 430 causes it to refer to order 43.
This is a typical run of ‘echmask’.

    ICL> echmask cofile=sdist.dat periscope=no objwidth=0 objoffset=0 ~
        s1width=10 s1offset=9 s2width=0 mstart=82 ~
        mdelta=-1 mask=mask
        *** Will use an OBJWIDTH of 6 pixels

When running with the periscope, each order is split up into two parts, each of which looks
rather like an order in its own right. If you are unsure how they are grouped, display an arc
and all will be revealed. When using the periscope it is your responsibility to ensure that the
first and second points selected with ‘icur’ correspond to the two parts of the same order and
similarly for the third and fourth points etc.

4.13.6 Mask extraction

The resulting mask image can be used for fast extraction of orders from (in the case of the CCD
‘irot90-ed’ and ‘irevy-d’) raw images taken at the same spectrograph configuration as it. This is
done using the ‘maskext’ program. ‘maskext’ needs to be told the range of order numbers that
you want to extract and this determines the Y size of the extracted file (referred to as a collapsed
échellogram) and its Y units.

The ‘sub-order’ controls which bits of the order are extracted into the output image. A sub-order
of 0 always extracts object and sky and sub-orders of 1 and 2 can be used to extract object and
sky separately. If the periscope is fitted, sub-order 2 corresponds to the first encountered part
of the order and sub-order 1 corresponds to the second encountered part of the order (so if the
tracked orders went from the bottom upwards the data values in the mask are monotonically
decreasing as you go from bottom to top). If the periscope is not fitted, sub-order 1 always refers
to object and sub-order 2 always refers to sky. This is confusing and will probably change!

    ICL> maskext image=arc mask=mask mlow=68 mhigh=82 subord=0 output=arce

4.13.7 Accurate extraction from straightened orders

The trouble with curved orders is that when the object projects to only one or two pixels on the
detector, the bulk of the signal will sometimes fall into one pixel and sometimes it will be split
between several pixels. This means that there is no single correct number of rows of data to
extract, forcing the extraction of unwanted sky as well as wanted signal.

Obviously it would be possible to provide a program that would make a sensible decision about
how many rows of data to extract at each point along the order, and we intend investigating the
use of some optimal weighted extraction scheme. However, any such scheme needs accurate
knowledge of the noise characteristics of the data and it would take considerable effort to
implement a reliable automatic extraction algorithm.
Accordingly, the recommended approach for accurate extraction is first to use the ‘cdist’ program to re-sample in the Y direction so as to straighten the orders. Experience shows that this program does an excellent job, with little discernible loss of resolution or variation of profile along the order. Once the orders are straightened, the number of rows to extract for object and for sky is merely a function of order number and not of wavelength.

**4.13.8 Order straightening**

The ‘cdist’ program uses the ‘sdist.dat’ file that was written by ‘sdist’. It uses the polynomial coefficients to re-sample in Y so as to straighten the corresponding orders.

```
ICL> cdist image=arc ystart=1 yend=250 output=arcc maxdegy=5
```

**4.13.9 Order extraction**

Having got an image with straight orders, conceptually one wants to take a ‘ystract’ through somewhere near the centre of the orders, display with ‘splot’ and then, for each order, somehow identify which rows are to be used for object and for sky. Having done this, the relevant rows can be extracted into a collapsed échellogram of the same format as that produced by ‘maskext’. ‘echselect’ allows the user to indicate interactively the cross-sections of a corrected échellogram to be used as object and sky for the various orders. It then creates a collapsed échellogram for the object orders, and—optionally—one for the sky orders.

**4.13.10 Wavelength calibration**

We are now at the stage where arc and object have both had their orders extracted into collapsed échellograms. These are two-dimensional images with X still being pixel number and Y being order number. The ‘echarc’ program is used for identifying lines in the arc. This writes wavelength calibration data as a two-dimensional arce.AXIS(1).MORE.FIGARO.DATA_ARRAY.DATA array and the arce.AXIS(1) structure is then copied to the obje.AXIS(1) structure using the ‘xcopy’ program.

‘echarc’ works by first of all doing the equivalent of the one-dimensional ‘arc’ program on a set of three or more orders that you nominate to be fitted interactively. Then it enters an automatic mode where lines are identified in all the other orders. It estimates wavelength in the other orders by fitting lines of constant (order number)*(wavelength) between the interactively fitted orders.

Experience with ‘echarc’ has shown that it is vital to include the extreme orders among those that are interactively fitted, that it may well be worth using four rather than three interactively fitted orders, that a good line list is absolutely vital and that if things start going wrong then they will probably stay wrong (use of Ctrl-C is recommended in this case). Care with the interactively fitted orders is usually rewarded.

The file ‘airlines.ech’ always contains details of identified lines and if a fit fails a good policy is to restart using the results from the previous fit and perhaps selecting slightly different interactively fitted orders, e.g. add the orders that were worst in the automatic phase from the previous run. All the lines previously identified in the orders that are to be interactively fitted are still available so a re-run is not too time-consuming.
Figure 14: Wavelength calibrating an échelle spectrum.

Also note that it is necessary to rename any old files of name ‘arlines.ech’ before running ‘echarc’. The ‘dowaves’ keyword should always be false (if it is true then the wavelength information is written to a separate file and we want it in the input file for the purpose of scrunching). Also note that if the ‘monitor’ keyword is true then a graphical record of how the automatic mode is proceeding will be output to the soft device.

Here is a typical example of ‘echarc’.

ICL> echarc image=arce arctype=thar previous=no interactive=3 ~ 
orders=[82,75,68] orderfit=5 sigma=3 dowaves=no

After a successful arc fit, use ‘xcopy’ to copy the ‘arce.AXIS(1)’ structure to the ‘obje.AXIS(1)’ structure.

ICL> xcopy spectrum=obje arc=arce output=obje
4.13.11 Scrunching

The next step is to scrunch the wavelength calibrated ‘obje’ and ‘arce’ files. Contrary to possible expectation the program used is ‘scrunch’ rather than ‘iscrunch’. Rather than using an ‘airlines.iar’ file, ‘scrunch’ uses the two-dimensional .AXIS(1).MORE.FIGARO.DATA_ARRAY.DATA array produced by ‘echarc’ as its source of wavelength information. You tell it the wavelength range and the number of bins that you want to scrunch into and whether to use a linear or logarithmic wavelength scale and it does the rest.

Since the échelle spectra are such that a pixel corresponds to a fixed velocity interval irrespective of wavelength (e.g., 22 microns = 2.73 km/s), a logarithmic wavelength scale will give bins that each correspond to the same number of pixels, whereas a linear wavelength scale will give bins at the red end that significantly oversample the resolution element relative to bins at the blue end. However, a linear scrunch will still probably be the favoured option in many cases.

It’s probably worth using ‘hdstrace’ to look at the file to find out the minimum and maximum wavelengths before running ‘scrunch’.

```
ICL> scrunch spectrum=obje log=no wstart=4000 wend=0.02 bins=5000 ~
    mean=no quad=yes output=objs
```

Every order will be scrunched into the full wavelength range (4550 to 5000 Angstroms in the above example) and consequently most pixels will be zero.

4.13.12 Merging orders

The final stage is to merge the scrunched orders from possibly several images into a single long spectrum. This is done using the ‘echmerge’ program. If the data is from a region of the field where the full free spectral range is obtained then adjacent orders will overlap and the contributions from the overlapping orders are weighted with estimates of their inverse variances on the assumption of purely Poisson noise. Where one contribution is below a given fraction of the other then it is ignored completely. The variance estimates are based on median filtered versions of the input orders and the median filter box size is controlled by the ‘box’ parameter.

The input images must all have the same X size, units and values and can be one-dimensional or two-dimensional. The output image is always one-dimensional and can be the same as either of the input files, in which case no new image is created. The second input image can be specified as blank in which case it is not required or used.

```
ICL> echmerge image=objs image1='' box=7 cutoff=4 output=objl
```

4.13.13 Summary of the example

Here there is just a reference list of the programs that were run above.

{ conventional orientation (CCD)
  ICL> irot90 contraw contrawrot
  ICL> irevy contrawrot cont
  ICL> irot90 arcraw arcrawrot

ICL> irevy arcrawrot arc
ICL> irot90 objraw objrawrot
ICL> irevy objrawrot obj

{ review of counts, widths and shapes
ICL> ystract cont 185 194 s
ICL> splot s reset accept
ICL> image cont high=hhhh reset accept

{ select points on orders
{ track and fit orders
ICL> icur
ICL> sdist cont .

{ Either ...

{ create mask
ICL> echmask sdist.dat . mask
{ extract orders using mask
ICL> maskext arc mask . arce
ICL> maskext obj mask . obje

{ Or ...

{ straighten orders
ICL> cdist arc . arcc
ICL> cdist obj . objc
{ extract orders
ICL> echselect arcc arce
ICL> echselect objc obje

{ End either.

{ identify arc lines
ICL> echarc arce .
{ copy wavelength info to object
ICL> xcopy obje arce obje

{ scrunch arc and object
ICL> scrunch arce . arcs
ICL> scrunch obje . objs

{ merge orders
ICL> echmerge arcs . arcl
ICL> echmerge objs . objl
A Classified list of commands

A.1 Data input

- **ALASIN** — Read a spectrum in ALAS (Abs. Line Analysis System) format
- **FITSET** — Set the value of a FITS keyword
- **FITSKEYS** — List the FITS keywords in a data file
- **ICOR16** — Corrects 16 bit data from signed to unsigned range
- **RCGS2** — Reads UKIRT CGS2 spectrum (also UKT9 and UKT6 CVF)
- **RDFITS** — Read file in AAO de facto ‘Disk FITS’ format
- **RDIPSO** — Read file in DIPSO/IUEDR/SPECTRUM format
- **TABLE** — List contents of a SPICA memory file

A.2 Data output

- **ALASOUT** — Output a spectrum in ALAS (Abs. Line Analysis System) format
- **WDFITS** — Writes an image out in the AAO de facto ‘Disk FITS’ format
- **WDIPSO** — Writes a file in DIPSO/IUEDR/SPECTRUM format

A.3 Display commands

- **CCUR** — After SPLOT, uses graphics cursor to indicate data values
- **COLOUR** — Set colour table for image display
- **DVDPLOT** — Plot the data in one file against the data in another
- **ELSPLOT** — Produces a long (<3m) error bar plot of a spectrum
- **ESPLOT** — Produces an error bar plot of a spectrum
- **HARD** — Sets the file name for hard copy output
- **HOPT** — Histogram optimisation of an image
- **ICONT** — Produces a contour map of an image
- **ICUR** — Inspect an image with cursor
- **IDEV** — Set the device for image display
- **IGCUR** — Use cursor to show x, y and data values
- **IGREY** — Produces a grey-scale plot of an image
- **IMAGE** — Display an image on the selected image display
- **IPILOTS** — Plots successive cross-sections of an image, several to a page
- **ISPLLOT** — Plots successive cross-sections through an image
- **LSPLLOT** — Hardcopy spectrum plot of specified size (up to 3 metres)
- **MSPLLOT** — Plots a long spectrum as a series of separate plots
- **SOFT** — Sets the device/type for terminal graphics
- **SPLOT** — Plots a spectrum
- **XCUR** — Uses cursor to delimit part of a spectrum

A.4 Wavelength calibration

- **ARC** — Interactive manual arc line identification
- **ECHARC** — Fit an échelle arc
EMLT — Fits gaussians to the strongest lines in a spectrum
FSCRUNCH — Rebin data with a disjoint wavelength coverage to a linear one
IARC — Given fit to single spectrum, fit all spectra in a 2-D arc
ISCRUNCH — Rebin an image to linear wavelength scale given IARC results
ISCRUNI — Like ISCRUNCH, but interpolates between two IARC result sets
LXSET — Set X array of spectrum/image to specified range
SCRUNCH — Rebin a spectrum to a linear wavelength range
VACHEL — Air to vacuum, and/or recession velocity wavelength conversion
XCOPI — Like XCOPY but interpolates X-data from 2 files
XCOPY — Copy X-info (eg wavelengths) into a spectrum

A.5 B-star calibration

BSMULT — Atmospheric band removal using a B-star calibration spectrum
CFIT — Generate a spectrum using the cursor
CSET — Interactively set regions of a spectrum to a constant value
MASK — Generate a mask spectrum given a spectrum and a mask table
MCFIT — Fit a continuum to a spectrum, given a mask spectrum
NCSET — Set a region of a spectrum to a constant

A.6 Arithmetic operations

CLIP — Clip data above and below a pair of threshold values
IADD — Adds two images (or two spectra)
IALOG — Takes the antilog of an image
ICADD — Adds a constant to an image
ICDIV — Divides an image by a constant
ICMULT — Multiplies an image by a constant
ICONV3 — Convolve an image with a 3x3 convolution kernel
ICSUB — Subtracts a constant from an image
IDIFF — Takes the ‘differential’ of an image
IDIV — Divides two images (or two spectra)
IGCONV — Convolve an image with a specified filter
ILOC — Takes the logarithm of an image
IMULT — Multiplies two images (or two spectra)
IPOWER — Raises an image to a specified power
IREVX — Reverse an image (or spectrum) in the X-direction
IREVY — Reverse an image in the Y-direction
ISHIFT — Applies a linear x and a linear y shift to an image
ISMOOTH — 2-D smooth of image using 9-point smoothing algorithm
ISTRETCH — Stretches and shifts an image in X and Y.
ISUB — Subtracts two images (or two spectra)
ISUBSET — Produces a subset of an image
ISUPER — Produces a superset of an image
ISXADD — Adds a spectrum to each X direction x-section of an image
ISXDIV — Divides a spectrum into each X direction x-section of an image
ISXMULT — Multiplies each X direction image x-sect by a spectrum
ISXSUB — Subtracts a spectrum from each X direction x-section of an image
<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ISYADD</td>
<td>Adds a spectrum to each Y direction x-section of an image</td>
</tr>
<tr>
<td>ISYDIV</td>
<td>Divides a spectrum into each Y direction x-section of an image</td>
</tr>
<tr>
<td>ISYMUL</td>
<td>Multiplies each Y direction image x-sect by a spectrum</td>
</tr>
<tr>
<td>ISYSUB</td>
<td>Subtracts a spectrum from each Y direction x-section of an image</td>
</tr>
<tr>
<td>IXSMOOTH</td>
<td>Smooth in x-direction by gaussian convolution</td>
</tr>
<tr>
<td>RESAMPLE</td>
<td>Rebin an image to different dimensions and/or orientation</td>
</tr>
<tr>
<td>RESCALE</td>
<td>Rescale using user-defined upper and lower limits</td>
</tr>
<tr>
<td>ROTX</td>
<td>Rotate data along the X-axis</td>
</tr>
</tbody>
</table>

### A.7 Flat fields

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFIT</td>
<td>Generate a spectrum using the cursor</td>
</tr>
<tr>
<td>FF</td>
<td>Flat field an image (uses JT’s algorithm)</td>
</tr>
<tr>
<td>FFCROSS</td>
<td>Cross-correlate an image and a flat field (mainly IPCS data)</td>
</tr>
<tr>
<td>MASK</td>
<td>Generate a mask spectrum given a spectrum and a mask table</td>
</tr>
<tr>
<td>MCFIT</td>
<td>Fit a continuum to a spectrum, given a mask spectrum</td>
</tr>
<tr>
<td>ISXDIV</td>
<td>Divides a spectrum into each X direction x-section of an image</td>
</tr>
</tbody>
</table>

### A.8 Data manipulation

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ADJOIN</td>
<td>Append two spectra (strictly a merge by wavelength value)</td>
</tr>
<tr>
<td>BCLEAN</td>
<td>Automatic removal of bad lines and cosmic rays from CCD data</td>
</tr>
<tr>
<td>CFIT</td>
<td>Generate a spectrum using the cursor</td>
</tr>
<tr>
<td>CLEAN</td>
<td>Interactive patching of bad lines, bad pixels in an image</td>
</tr>
<tr>
<td>COADD</td>
<td>Form the spectrum which is the mean of the rows in an image</td>
</tr>
<tr>
<td>COMBINE</td>
<td>Combine two spectra, adding with weights according to errors</td>
</tr>
<tr>
<td>COSREJ</td>
<td>Reject cosmic rays from a set of supposedly identical spectra</td>
</tr>
<tr>
<td>CREOBJ</td>
<td>Create a data object or file</td>
</tr>
<tr>
<td>FSCRUNCH</td>
<td>Rebin data with a disjoint wavelength coverage to a linear one</td>
</tr>
<tr>
<td>HCROSS</td>
<td>Cross-correlate two spectra amp get redshift and error</td>
</tr>
<tr>
<td>HIST</td>
<td>Produce histogram of data value distribution in an image</td>
</tr>
<tr>
<td>HOPT</td>
<td>Histogram optimisation of an image</td>
</tr>
<tr>
<td>ICONV3</td>
<td>Convolve an image with a 3x3 convolution kernel</td>
</tr>
<tr>
<td>ICOR16</td>
<td>Corrects 16 bit data from signed to unsigned range</td>
</tr>
<tr>
<td>IDIFF</td>
<td>Takes the ‘differential’ of an image</td>
</tr>
<tr>
<td>IGCONV</td>
<td>Convolve an image with a specified filter</td>
</tr>
<tr>
<td>IREVX</td>
<td>Reverse an image (or spectrum) in the X-direction</td>
</tr>
<tr>
<td>IREVY</td>
<td>Reverse an image in the Y-direction</td>
</tr>
<tr>
<td>IROT90</td>
<td>Rotates an image through 90 degrees</td>
</tr>
<tr>
<td>MEDFILT</td>
<td>Applies a square median filter to an image</td>
</tr>
<tr>
<td>MEDFILTER</td>
<td>Applies a rectangular median filter to an image</td>
</tr>
<tr>
<td>MEDSKY</td>
<td>Take the median of a number of images</td>
</tr>
<tr>
<td>POLSKY</td>
<td>Fits and subtracts sky from a long slit spectrum</td>
</tr>
<tr>
<td>SCLEAN</td>
<td>Interactive patching of images, especially SCUBA data</td>
</tr>
<tr>
<td>SCNSKY</td>
<td>Calculates a sky spectrum for a scanned CCD image</td>
</tr>
<tr>
<td>SCROSS</td>
<td>Cross-correlate two spectra and get relative shift</td>
</tr>
<tr>
<td>SCRUNCH</td>
<td>Rebin a spectrum to a linear wavelength range</td>
</tr>
<tr>
<td>SFIT</td>
<td>Fit a polynomial to a spectrum</td>
</tr>
</tbody>
</table>
A.9 Aperture photometry

**APERTURE** — Do simple minded aperture photometry on a series of frames
**CENTERS** — Generate file of object centroids from ICUR/IGCUR output
**FOTO** — Perform aperture photometry given CENTERS output
**ICUR** — Inspect an image with cursor
**IGCUR** — Use cursor to show x, y and data values

A.10 Line analysis

**ABLINE** — Interactive absorption line analysis
**EMLT** — Fits gaussians to the strongest lines in a spectrum
**GAUSS** — Interactive fit of Gaussians to emission or absorption lines

A.11 S-distortion and échelle order straightening

**CDIST** — S-distortion correction using SDIST results
**FINDSP** — Locate fibre spectra in an image
**ICUR** — Inspect an image with cursor
**IGCUR** — Use cursor to show x, y and data values
**OFFDIST** — Applies an offset to an SDIST fit
**OVERPF** — Overlays a FINDSP fit on another image
**POLEXT** — Extract fibre spectra from an image after a FINDSP analysis
**SDIST** — Analyse an image containing spectra for S-distortion

A.12 Fudging data

**COPOBJ** — Copy an HDS object
**CREOBJ** — Create a data object or file
**CSET** — Interactively set regions of a spectrum to a constant value
**DELOBJ** — Delete a data object or a file
**FLAG2QUAL** — Converts ‘flagged’ values to produce a quality array
**GOODVAR** — Replace negative, zero and bad variance values
**ICSET** — Set a selected region of an image to a constant value
**ISEDIT** — Allows interactive editing of a 1-D or 2-D spectrum
**LXSET** — Set X array of spectrum/image to specified range
**LYSET** — Set Y array of spectrum/image to specified range
**NCSET** — Set a region of a spectrum to a constant
**Q2BAD** — Converts an NDF’s quality into bad values
**QUAL2FLAG** — Converts a quality array into ‘flagged’ values
**REMBAD** — Removes pixels that have been flagged as bad from data
**RENOBJ** — Change the name or location of an object within an HDS file
**SETOBJ** — Assign value to an HDS primitive
**SPIED** — Interactive spiketrum editor
**TIPPEX** — Modify individual pixel values with cursor
**XCADD** — Adds a constant to the X data in a file
**XCDIV** — Divides the X data in a file by a constant
XCMULT — Multiplies the X data in a file by a constant
XCSUB — Subtracts a constant from the X data in a file
YCADD — Adds a constant to the Y data in a file
YCDIV — Divides the Y data in a file by a constant
YCMULT — Multiplies the Y data in a file by a constant
YCSUB — Subtracts a constant from the Y data in a file

A.13 Examining data

HIST — Produce histogram of data value distribution in an image
FIGINFO — Describes the contents of a Figaro data file
FITSKEYS — List the FITS keywords in a data file
ICUR — Inspect an image with cursor
IGCUR — Use cursor to show x, y and data values
ILIST — List the data in an image (or spectrum)
ISTAT — Provides some statistics about an image (max, min etc.)

A.14 Slicing through images and cubes

EXTLIST — Adds a number of non-contiguous lines of an image -> a spectrum
EXTRACT — Adds contiguous lines of an image -> a spectrum
GROWX — Performs reverse function to that of EXTRACT
GROWXT — Copies an image into contiguous XT planes of a cube
GROWXY — Copies an image into contiguous XY planes of a cube
GROWY — Performs reverse function to that of YSTRACT
GROWYT — Copies an image into contiguous YT planes of a cube
OPTEXTRACT — Extracts a long slit spectrum using Horne’s optimal extraction
PROFILE — Determines a long slit spectrum profile for use by OPTEXTRACT
SLICE — Takes a slice with arbitrary end points through an image
XTPLANE — Adds contiguous XT planes of a data cube -> an image
XYPLANE — Adds contiguous XY planes of a data cube -> an image
YSTRACT — Adds contiguous columns of an image -> a spectrum
YTPLANE — Adds contiguous YT planes of a data cube -> an image

A.15 Fibre data

FINDSP — Locate fibre spectra in an image
OVERPF — Overlays a FINDSP fit on another image
POLEXT — Extract fibre spectra from an image after a FINDSP analysis

A.16 Flux calibration

ABCONV — Convert spectrum from Janskys into AB magnitudes
CALDIV — Generate calibration spectrum from continuum standard spectra
CFIT — Generate a spectrum using the cursor
CSE — Interactively set regions of a spectrum to a constant value
CSPIKE — Create calibration spiketrum given spiketrum and standard spectrum
FIGSFLUX — Flux calibrates a FIGS spectrum
FLCONV — Convert a spectrum in Jansky's into one in erg/s/cm**2/Angstrom
FWCONV — General unit conversion for spectra
GSPIKE — Generates a 'spiketrum' from a table of values
INTERP — Interpolates between the points of a 'spiketrum' -> a spectrum
IRFLUX — Flux calibrates an IR spectrum using a black-body model
LINTERP — Linear interpolation between spiketrum points -> spectrum
NCSET — Set a region of a spectrum to a constant
SFIT — Fit a polynomial to a spectrum
SPFLUX — Applies a flux calibration spectrum to an observed spectrum
SPIED — Interactive spiketrum editor
SPIFIT — Fits a global polynomial to a spiketrum -> a spectrum

A.17 Extinction

EXTIN — Correct spectrum for atmospheric extinction
GSPIKE — Generates a 'spiketrum' from a table of values
LINTERP — Linear interpolation between spiketrum points -> spectrum

A.18 Complex data and FFTs

BFFT — Takes the reverse FFT of a complex data structure
CMPLX2I — Extracts the imaginary part of a complex data structure
CMPLX2M — Extracts the modulus of a complex data structure
CMPLX2R — Extracts the real part of a complex data structure
CMPLXADD — Add two complex structures
CMPLXCONJ — Produce the complex conjugate of a complex structure
CMPLXDIV — Divide two complex structures
CMPLXFILT — Create a mid-pass filter for complex data
CMPLXMULT — Multiply two complex structures
CMPLXSUB — Subtract two complex structures
COSBELL — Create data that goes to zero at the edges in a cosine bell
FFT — Takes the forward FFT of a complex data structure
I2CMPLX — Copies an array into the imaginary part of a complex structure
PEAK — Determines position of highest peak in a spectrum
R2CMPLX — Creates a complex data structure from a real data array
ROTX — Rotate data along the X-axis

A.19 Infra-red data

FET321 — Extracts a spectrum from 1 detector from etalon mode FIGS data
FIGS321 — Processes a FIGS data cube down to a single spectrum
FIGS322 — Processes a FIGS data cube down to an image
FIGS422 — Process a FIGS image-mode hypercube down to an image
FIGS423 — Process a FIGS image-mode hypercube down to a cube
FIGS424 — Sort a FIGS image-mode hypercube into wavelength order
FIGSEE — Generate a seeing ripple spectrum from a FIGS spectrum
FIGSFLUX — Flux calibrates a FIGS spectrum
IRCONV — Converts data in Janskys to W/m**2/um
IRFLAT — Generates a ripple spectrum from an IR spectrum
IRFLUX — Flux calibrates an IR spectrum using a black-body model
REMBAD — Removes pixels that have been flagged as bad from data

A.20 Échelle data
CDIST — S-distortion correction using SDIST results
ECHARC — Fit an échelle arc
ECHFIND — Locate spectra in échelle data
ECHMASK — Produce an extraction mask from an SDIST analysis
ECHMERGE — Merge échelle spectra into a single long spectrum
ECHSELECT — Interactive selection of sky and object spectra for an échelle
ICUR — Inspect an image with cursor
IGCUR — Use cursor to show x, y and data values
IMAGE — Display an image on the selected image display
MASKEXT — Extracts échelle orders using a mask created by ECHMASK
OFFDIST — Applies an offset to an SDIST fit
SDIST — Analyse an image containing spectra for S-distortion

A.21 Spectroscopy Data Reduction (Specdre)

Input/output
ASCIN — Read a 1-D or N-D data set from an ASCII table.
ASCOUT — Write an NDF to an ASCII table.

Display
MOVIE — Browse through slices of a cube.
SPECCONT — Contour a two-dimensional cut.
SPECGRID — Plot spectra on position grid.
SPECLOT — Plot a spectrum.

Statistics, fitting
CORREL — Correlate two or three data sets.
EVALFIT — Evaluate fit results.
FITBB — Fit diluted Planck curves to a spectrum.
FITGAUSS — Fit Gauss profiles to a spectrum.
FITPOLY — Fit a polynomial to a spectrum.
FITTRI — Fit triangular profiles to a spectrum.
MOMENTS — Calculate moments of spectra in a cube.

Axis calibration
ARCDISP — Fit polynomial dispersion curve.
ARCGENDB — Convert list of laboratory values to feature data base.
ARCIDENT — Auto-identify located features.
ARCLOCAT — Locate line features in a set of spectra.
Data calibration

**BBODY** — Calculate a black body spectrum.

Convolution, re-sampling, merging

**FILLCUBE** — Copy one NDF into part of another.

**RESAMP** — Re-sample and average several spectra.

Reshaping

**GROW** — Copy an N-dimensional cube into part of an (N+M)-dimensional one.

**SUBSET** — Take a subset of a data set.

**XTRACT** — Average an N-dimensional cube into an (N-M)-dimensional one.

Miscellaneous

**EDITEXT** — Edit the Specdre Extension.

A.22 Spectroscopy Data Reduction (Twodspec)

Display

**ISCAN** — Plots cut through a 2D longslit array.

**HIMAGE** — Plots a greyscale image of a 2D array.

**CSCAN** — Plot array of profiles from a 3D array.

Axis Calibration

**ARC2D** — Calibrates distortions in 2D arc line data.

**COMB** — Corrects for S-distortion using continua.

**ARCSDI** — Corrects for arc line curvature.

Line Profile Analysis

**LONGSLIT** — Fits 2D longslit arrays and plots results.

**FIBDISP** — Fits 3D cubes and plots results.

Data Manipulation

**FITCONT** — Fit continuum for subtraction.

**CSUB** — Subtract fitted continuum.

**CADD** — Add back fitted continuum.

Miscellaneous

**FIBSEP** — Separate spectra in 2D array.

**FIB2CUBE** — Stack LONGSLIT results into a data cube.

**CUBE2LONG** — Extract fits from a cube in LONGSLIT format.

**VIG** — Corrects a 2D array for vignetting.

**CRIGAUSS** — Generates an NDF with a multiple gaussian profile.

**CHANGED** — Lists differences between fits in two files.
A.23 Miscellany

- **CCDLIN** — Applies a linearity correction to AAO CCD data
- **ERRCON** — Converts percentage error values to absolute values
- **FIGHELP** — Browse through the Figaro help library
- **RETYPE** — Changes the type of the main data array in a file
- **SQRTERR** — Generates an error array as Error = Square Root of (Data/Const)
- **TRIMFILE** — Creates a copy of an HDS file without unused space
B Specdre

B.1 Introduction

This appendix contains information on version 1.1 of the Specdre package (for SPEctroscopy Data Reduction) which has been added to Figaro. Sections B.6 and B.7 give examples of using several applications together.

Specdre is a package for spectroscopy data reduction and analysis. Some of the general features of the package are:

- **Hyper-cubes**: The Specdre data set is in general a hyper-cube where each row or hyper-column is a spectrum. Even where a single spectrum is required as input, this can be an appropriate section of the hyper-cube cut out “on the fly” as the application accesses the data.

- **Coherent storage of fit results**: The results of line or continuum fits are stored along with the data. In the case where a hyper-cube is a coherent set of spectra, fit results will also be stored coherently. For example, in a three-dimensional data set the two-dimensional map of line integrals is immediately available to display routines.

- **Bad values and variance**: Bad values (or quality information) are recognised and ignored or propagated, as appropriate. If present, variance information is propagated or used in the processing, e.g. for statistical weights. It can optionally be ignored. Where covariance is created (namely re-sampling), an approximate measure of this is stored along with the data. Other applications (namely fit routines) will use the ordinary variance or the measure of covariance, as appropriate.

The topics addressed by the applications are mainly:

- **ASCII I/O**: The data and errors of hyper-cubes can be written to or read from printable/editable tables. Bad values are converted between the two formats. Single spectra can be read even if the axis data are not linear or monotonic.

- **Graphics**: Display applications allow full control of the plot, including font, colour, line styles, error bars, etc. Overlay on previous plots according to their “world coordinates” is possible. This includes overlays on grey/colour/line plots made by [KAPPA] [Pongo], etc.

- **Cube manipulation**: You can extract averaged hyper-planes from hyper-cubes, assemble hyper-cubes from hyper-planes, or fill in a hyper-cube from several given hyper-cubes.

- **Arc line axis (wavelength) calibration**: While full user interaction via graphics is granted, automatic arc line identification is also possible.

- **Re-sampling**: The application for re-sampling can either re-sample all spectra in a hyper-cube, or re-sample and average into one spectrum any number of input spectra. It allows information about the covariance between pixels to be carried through to a line fit routine.
- **Spectral fits**: You can fit polynomials, blended Gauss or triangle profiles. Fit results are stored along with the data and can be turned into fake data sets for later subtraction, division, etc.

Specdre uses the NDF data access library, which allows you to specify sections rather than the whole data set. Also, for the special requirements of spectroscopy data reduction and analysis, an extension to the NDF format is used which stores additional information with the data, thus allowing much enhanced communication between Specdre applications.

### B.2 Specdre’s use of parameters

Some parameters used by Specdre are common to several commands. The device parameter is sometimes associated with the global parameter `GRAPHICS_DEVICE`. When it is, it usually defaults. And these parameters are really global, in the sense that other packages may use and change them, too.

Where in and/or out are NDFs, they are mostly associated with the global `DATA_ARRAY`. The effect is that the default input is usually the output of the previous command.

`info` and `dialog` are always associated with `SPECDRE_INFO` and `SPECDRE_DIALOG`. These parameters control the amount of information and user interaction of many applications. Once `info` is switched off all applications will become quiet until the parameters are set true again.

`varuse` is a defaulted parameter to many applications, but not associated with a global parameter. By default it is true. Sometimes it has to be set false in order to ignore variance information in the input data.

Other parameters like `start`, `step`, `end` occur naturally in several applications. In some instances they may be scalars, in others vectors. Often their defaults are set by the application with knowledge of the data set at hand.

### B.3 Graphics

The management of graphics output closely follows that of KAPPA. To make full use of the graphics capabilities, you will need to use some KAPPA commands; and you will find Pongo extremely useful to add almost anything to your plots. For normal use you can get along without KAPPA or Pongo. The actual plots are achieved through a combination of AGI, SGS, PGPLOT, and GKS. The graphics is done with PGPLOT, but the device is handled via AGI and SGS, and GKS is the low level package underlying them all. AGI stores information about the graphs produced in a data base. This data base tells co-operating applications where and on which device a plot was made and what its world coordinates were. Other packages use and update the same data base so that a consistent display administration can be achieved, even when different packages are used in turn. The overlay options in Specdre applications use this information and allow you to plot with Specdre in the right place on top of, for example, an image displayed in grey scale or colour with KAPPA.

Usually a command whose main task is to produce a plot, has a parameter `device` which is not prompted for and which is associated with the global parameter `GRAPHICS_DEVICE`. The value of this global parameter will be the name of a graphics device, and the named device will be used for the display. But you can always specify the `device` parameter on the command line, thus overriding the global parameter:
ICL> specplot device=graphon
ICL> specplot device=xw
ICL> specplot device=ps_l

This will also change the global parameter so that next time you use the same device automatically.

There are other Specdre commands that may or may not use a graphics device. These will prompt for the device parameter and offer the null value as default. This can be accepted to avoid using a graphics device. When specifying parameters on the command line, device=! may not always work, but including the accept keyword will have the desired effect.

Some applications not only display graphics, but use a graphics display plus mouse and keyboard to conduct a dialogue with the user. This is usually optional and controlled by the dialog parameter. dialog is a character parameter. It is always allowed to be one of the letters \{Y,N,T,F,y,n,t,f\}. Sometimes it may also be G or g for graphics dialogue. y,t may or may not mean the same as g.

If you specify a printer or PostScript device, this may fail for the graphics dialogue case. But otherwise all plots can be sent to files that you print later. There is one important difference, you have only one screen, but the printer has many sheets of paper. Your plot may be in a number of printer files and each printout starts on a new page. If you have done overlay plots on the screen and want the same on the printer, then you can use as graphics device an Encapsulated PostScript device like epsf_l. You still get a number of files, but they can be merged into one (without form feed) using psmerge(cf. SUN/164). Usually the output is a complete PostScript file ready to be printed.

### B.4 Slicing data sets

Specdre uses the NDF routines to access data in Starlink Data Files (SDF). This allows the user to specify sections of NDFs rather than whole (i.e. base) NDFs. Some applications need one-dimensional input data, but by themselves offer no means to take a subset of a larger or multi-dimensional data set. The desired effect can, however, be achieved by NDF. As an example, fitgauss will fit a spectrum only and rejects two-dimensional input. You still can use a 2-D data set by specifying a section thereof as input to fitgauss:

ICL> fitgauss in=myndf(,5)               { 5-th row of 2-D input
ICL> fitgauss in=myndf(5,)               { 5-th column
ICL> fitgauss in=myndf(20:30,5)          { pixels 20 to 30 of 5-th row
ICL> fitgauss in=myndf(6540.0:,5)        { pixels beyond coordinate 6540.0
ICL> fitgauss in=myndf(6450.0~10.0,5)    { coordinate range 6540.0 +- 5.0

(For fitgauss the sub-setting within the row is not necessary, because it does its own masking of the given 1-D data.) It should be mentioned here that the NDF fed into an application need not be at the top level of its container file. Once your NDF got a Specdre Extension (Section 3.2.7) with fit results in it, you can e.g. plot the second fit parameter versus the row number:

ICL> specplot in=myndf.more.specdre.results(2,1,)
B.5 Cube manipulation

A spectrum may be thought of as a one-dimensional data set. But spectroscopists are also aware of the two-dimensional space of sky position and might use time as an axis in data sets. So the data handling aspects of spectroscopy are more demanding than image processing – Specdre applications may encounter data with any dimensionality. Two-dimensional detectors are often used to take spectra and where observations are not very time-consuming three-dimensional data sets may be quite common.

Specdre can work on data with any dimensionality. The limit is 7-D due to the HDS data access routines. In practice the limit may be 6-D since one structure in the Specdre Extension has one dimension more than the main data array.

However, Specdre is about spectra. An N-D cube is taken as a set of spectra, each spectrum is a row or a column in the cube. A row extends along the first axis of the cube, while a column extends along any axis of the cube. In any cube Specdre assumes that there is exactly one spectroscopic axis, by default this is the first axis. The Specdre Extension specifies which axis is the spectroscopic one.

Specdre’s handling of N-D data falls into three categories.

- Some applications work on one spectrum at a time. They will insist on 1-D input, but you can specify a column to work on. Often the column must extend along the spectroscopic axis. These applications can be used successively on several or all columns of a cube. Insofar as they produce output will they work “in situ” and modify the input file.

- Other applications take on a whole cube at once and work on each row in turn. Usually spectra have to be in rows and the spectroscopic axis must be the first axis. These applications may also refuse to work on only a section of the input.

- Then there are applications that don’t deal with spectra at all but are tools to manipulate cubes. It is important that Specdre has such a set of tools, since the Specdre Extension may have a number of cubes in it as well. The main and extension cubes must be manipulated in a consistent manner, or the Extension becomes invalid. The fact that the cube manipulation routines handle only version 0.7 of the Specdre Extension and not version 1.1, means that the COORD component of the Extension is lost when these routines go about their business.

subset is very similar to KAPPA's ndfcopy or to taking an “on the fly” section as input to an application. The differences are that subset also takes subsets of NDFs in the Specdre Extension (v. 0.7) consistent with the subset of the main NDF, and that it removes “degenerate” axes. Consider the command

```
ICL> subset image(5,1:10) row
```

When subset gets the input it is still an image of size 1 by 10. But in the output the degenerate axis has been removed so that it is also officially 1-D.

Where subset may remove axes, grow deliberately adds new axes – degenerate or genuine ones. So we could reverse the command above with
The zeros as second vector elements just show that the second axis of `image` matches the axis in `row`. `expand` shows which of the output axes are and are not in the input. Normally those new axes will of course not be degenerate, so `size` might be `[5, 0]`. In that case `row` could be copied into any of the output columns, into one of them or repeatedly into a whole range of columns. The main idea of `grow` is that you assemble rows into images, images into cubes etc. So when `new=n` then you will copy into an existing file. The following puts two spectra and one image into a common cube. Whatever part of the cube is not copied to, remains filled with bad values.

If `grow` creates and expands new axes, `xtract` collapses existing axes. This is done by averaging the pixels along each collapsed axis. Note that this is an average and not a sum, which makes a big difference for the use of input variances and the meaning of output variances. Basically an average assumes that all values entering the mean are equal and scatter at random.

grow copies input into output of higher dimensionality, the common dimensions must match. `fillcube` is different. It copies input into output of the same dimensionality. Dimensions need not match, the copy is positioned in output by matching `centre` coordinates. Indeed the copy may not be contiguous in output. The output is an existing file, so you can fill it successively from different input files. This is mosaicing in N-D.

`resample`, too, plays a rôle in cube manipulation, since it can homogenise and linearise the coordinates along the spectroscopic axis. When used in `mode=Cube` it re-samples each row of a cube individually. Afterwards all rows have the same linear coordinate grid as expressed by the new vector of `centres` for the spectroscopic axis. Any spectroscopic values in the Specdre Extension are thus obsolete and removed. Sometimes it is necessary for other applications that grids are linear or that there is no array of spectroscopic values in the Specdre Extension.

### B.6 Spectral fits

Specdre has a number of applications to fit analytical functions to spectral features. Two are line fit routines for Gauss and triangular profiles. These can fit up to six components at a time. The lines can be blended and the line parameters can be free, fixed, or tied to the corresponding (free) parameters of another component. A similar routine fits up to six diluted Planck curves. Finally, a polynomial fit can be performed, the order can be up to 7.

The fit routines can run with a (non-graphic) dialogue or not, they can display data and fit graphically at different stages of the fitting process (masking, guessing, fit residuals).

These fit routines work on one-dimensional data only. But you can pass an NDF section that is (part of) a row or column in an image or cube. For the fit only data inside the union of up to six masking intervals are used. The fit results go first of all to the standard output (the terminal),
but can also be recorded in (appended to) an ASCII file. In addition fit results will always be stored in a results’ NDF in the Specdre Extension. Those results can be used to generate a model data set. You could then subtract that from the original data.

Here is an outline of a complex example showing how you might proceed with a long-slit spectrum (but note that the example will not work as presented since some parameters are omitted for brevity). Also, the telluric correction may not be the correct way of doing things. The purpose of the example is to illustrate how you can inter-operate Specdre and KAPPA applications.

1) For each row of the image use the abscissa ranges [1;2], [3;4], and [5;6] to fit a polynomial continuum. The two intervals excluded here probably contain spectral lines. For each row the parameters describing the fit are stored as the first component in the results’ NDF in the Specdre Extension of the image. You would probably run the fit routine with dialogue and graphics the first time round to play with the mask intervals.

2) For the whole image use the stored result to generate a corresponding data set representing the fitted continuum.

3) Use a KAPPA command to divide the original image by the continuum.

4) For each row of the normalised image use the intervening abscissa ranges and fit three Gauss profiles to them. These are stored as components 2, 3 and 4 in the Extension. Again you would use dialogue to play with masks and parameter guesses in the first use of fitgauss. For subsequent image rows you would try without dialogue and just recall the command line to edit the row number.

5) The second line fitted, the third component stored, is not stellar but terrestrial, subtract it with the KAPPA command sub.

6) Use KAPPA’s cadd to turn the simple telluric lines into proper telluric spectra with continuum at 1.0. Then divide the normalised spectra by the telluric spectra.
B.7 Arc spectrum axis calibration

There are four applications to do a wavelength calibration, or at least a calibration into spectroscopic values anyway. You can use frequencies or photon energies if you feel like it. You can also use nanometre, micrometre or Ångström, as you please.

At the heart of this axis calibration is an algorithm written by Mills (1992) to automatically identify features in an arc spectrum. For this to work, you must have a data base of arc feature identifications rather than just a simple line list. You can use arcgendb to convert a line list (as distributed with Figaro) into a “feature data base”. Unfortunately the data base takes some time to build and is also rather big, 10 to 100 times bigger than the simple list. So there may be a point in taking only the relevant wavelength range from long line lists like Th-Ar and converting it into a feature data base.

With such a data base at your disposal, you still cannot run the auto-identifier. This is because the calibration procedure as performed by Figaro’s arc is broken up into three steps:

- In the first step you go through the arc spectrum and locate features, that is you find out where in pixel coordinates the observed line features are. For this first step you normally use arclocat. Usually this first step will be done in two passes. In the first pass you run arclocat dialog=f so that it tries on its own to find un-blended narrow emission features in the arc spectrum. In a second pass you run arclocat dialog=g and use the graphic dialogue to improve the set of located features. You may add features not found before, or delete features that are blended or known to be absent from the feature data base. In general you should locate as many features as possible, you can always leave some un-identified.

Instead of arclocat you can also use fitgauss or fittri in case that the simplified line fit in arclocat is not good enough. arclocat has modes Gauss and triangle, too, but it fits only one line at a time.

- Only in the second step do you auto-identify the features, that is the Mills algorithm will try to establish for some of the located features what their identification might be in terms of wavelength, frequency or whatever you use in the feature data base. This second step is performed by arcident. The result must be regarded with some scepticism, since there is a small chance that arcident will find a grossly wrong solution or make a slightly unfavourable selection of features to identify. Any such inadvertencies can be corrected in the third step.

- The third step again may or may not be in a graphics dialogue. arcdisp dialog=g will display a plot of wavelength, frequency etc. versus pixel coordinate. It does not show the spectrum. Instead vertical lines indicate unidentified (but located) features, horizontal lines indicate all possible identifications from the feature data base, and crosses indicate identified features. Finally the would-be dispersion curve is displayed. You can now add or remove identifications with mouse and keyboard by clicking on the intersection of the feature location (vertical line) and potential identification (horizontal line).

The improvement of feature identification is one goal of arcdisp. The other is to do a polynomial fit to the pixel-wavelength relation and to convert pixel coordinates into wavelengths using that fit. This latter goal can also be achieved without the graphics dialogue with arcdisp dialog=n. This does not replace the pixel coordinates of the main NDF, but creates a new array of spectroscopic values in the Specdre Extension.
arclocat, arcident and arcdisp in general work not on a spectrum, but on an image or cube that has spectra in its rows (rows, not columns). arclocat dialog=g allows you to switch from one row to another as you please, arclocat dialog=n will scan through all rows in sequence. arcident will do an independent auto-identification on each spectrum, so it is suitable for a collapsed échellogram where successive extracted orders are in the rows of an image.

arcdisp dialog=g will work on each row in sequence, you can work on one row and proceed to the next in your own time. You can quit at any time; this will leave the file without spectroscopic values, but any improved feature identifications are kept. If you step through all rows, then the spectroscopic values will be kept as well. (You can have spectroscopic values in the Specdre Extension either for all rows or none at all!) You may want to set the label and unit for the spectroscopic values after arcdisp with KAPPA's \texttt{setlabel} and \texttt{setunits}.

So far you have only succeeded in producing an array of calibrated spectroscopic values in the Specdre Extension of the \textit{arc spectrum}. You will want to copy that array into the celestial observation you are actually interested in. This can be done with \texttt{ndfcopy}, provided the sky spectrum does have a Specdre Extension and does not have a SPECVALS component in it. That status can be achieved by two \texttt{editext} commands. Finally you may want to re-sample each spectrum (row in a cube) so that all use the same linear grid of spectroscopic values. To that end you use \texttt{resamp mode=cube}. Here is the whole procedure. (The commands are not complete; parameters are missing. Also there may be no Th-Ar line lists available for photon energies in the MeV range.)

\begin{verbatim}
ICL> arcgendb thar.arc thar_arc
ICL> arclocat dialog=n arc1
ICL> arclocat dialog=g arc1
ICL> arcident arc1 arc2 thar_arc
ICL> arcdisp arc2
ICL> setlabel arc2.more.specdre.specvals "particle energy"
ICL> setunits arc2.more.specdre.specvals "log10(10**9*eV)"
ICL> editext "create" sky1
ICL> editext "delete specvals" sky1
ICL> ndfcopy arc2.more.specdre.specvals sky1.more.specdre.specvals
ICL> resamp cube sky1 sky2
\end{verbatim}
B.8 References

- Currie M.J., 1992, KAPPA – Kernel application package, Starlink User Note 95
- Currie M.J., 1994, HDSTRACE – Listing HDS data files, Starlink User Note 102
- Eaton N., 1995, AGI – Application Graphics Interface – A subroutine library for accessing the graphics database, Starlink User Note 48
- Harrison P., Rees P., Draper P., 1994, PONGO – A set of applications for interactive data plotting, Starlink User Note 137
- Shortridge K., 1991, FIGARO – General data reduction and analysis, Starlink Miscellaneous User Document
C Twodspec

C.1 Introduction

Twodspec is a package of programs for the reduction and analysis of long-slit and optical fibre array spectra. It was originally developed as a separate package which complemented, and shared a common data format with, Figaro. For convenience, it has subsequently been incorporated into Figaro. This appendix is more-or-less the final version of the stand-alone Twodspec manual, with only a few minor changes. You should be aware that Twodspec’s style of interaction with the user is somewhat different to that of the basic Figaro applications. The best way to learn to use Twodspec is to find someone who is already familiar with it and persuade them to demonstrate it to you.

LONGSLIT is a program for the analysis of calibrated long-slit spectra. It has, among other things, the capability to fit Gaussians, either manually or automatically—in batch. It can, however, handle data with two spatial dimensions, such as obtained using TAURUS. The program FIBDISP provides further options useful for such data, although this is primarily designed for fibre array data. An extensive range of options is available, especially in terms of output. ARC2D is a two-dimensional arc calibration program. COMB and ARCSDI are for the correction of data for S-distortion and line curvature respectively.

Note that these programs are designed to do as much as possible for the user, to assist quick reduction and analysis of data—LONGSLIT can fit multiple Gaussians to line profiles in batch for example, and will decide how many components to fit. More information on the aims behind the package are given in Wilkins 1988 and Wilkins & Axon 1991.

Users of Twodspec are requested to acknowledge its original authors, T.N. Wilkins and D.J. Axon, in any publications resulting from its use.

C.2 Line Profile Analysis

C.2.1 Introduction

In order to parameterise a long-slit spectrum, it is often convenient to fit a function to the spectral lines. The most commonly used function is a Gaussian, defined as:

\[ f(x) = H e^{-\left(\frac{(x-x_0)^2}{2\sigma^2}\right)} \]  

where \( \sigma \) is the standard deviation of the line, \( x_0 \) is its centre, and \( H \) is its height. If the gas consists of a number of discrete clumps, moving at the same velocity, then a series of Gaussians would normally accurately represent the line profile (although the instrumental profile may alter this).

The Cauchy function may also be used:

\[ f(x) = \frac{H}{1 + \left[2(x-x_0)/w\right]^2} \]  

(2)
The instrumental line profiles from a Fabry-Perot interferometer are approximate Cauchy functions. The precise form (Bland 1986) is:

\[ A(x, y, z) = \left[ 1 + \left( \frac{2N}{\pi} \right)^2 \sin^2 \left( \frac{2\pi \mu l(z) \cos \theta}{\lambda} \right) \right]^{-1} \]  

where \( N \) is the finesse of the etalon, \( \mu \) the refractive index of air, \( l(z) \) the etalon spacing, \( \theta \) the angle to the optical axis, and \( \lambda \) the wavelength of the light concerned.

Frazer and Suzuki (1969) give a function which varies smoothly between a Gaussian and a Cauchy function:

\[ f(x) = \frac{H}{(1 + [2a^2 - 1][2(x - x_0)/w]^2)} \]  

where as \( a \) tends to 0, this function tends to a Gaussian, and for \( a = 1 \), it is a Cauchy function.

The fitting consists of optimising the values of the free parameters, that is \( H, x_0 \), etc. Normally one would also allow the fitting routines to vary the base level.

The programs LONGSLIT and FIBDISP can fit Gaussians (single or multiple), Lorentz profiles and Cauchy functions (actually the function of equation [4]) to line profiles. From the fitting velocity plots may be produced by the programs, as can plots of line width and line flux (not FIBDISP), for each line against cross-section (greyscale in FIBDISP). It is also possible to output greyscale or contour velocity plots. LONGSLIT is designed to operate in batch as much as possible.

### C.2.2 LONGSLIT

LONGSLIT is designed primarily for the analysis of long-slit spectra, although this is mainly a matter of the display options available—the fitting routines, for example, can cope with three-dimensional data.

The basic method of operation of LONGSLIT is as follows:

1. Locate lines. This is done by extracting part (or all) of the image and displaying it. The user is then required to mark to either side of the lines with a cursor and to identify the lines so located. Alternatively CLONE (see section C.2.2) may be used. A great advantage with CLONE here is that, since the data is calibrated, for a lot of data the tram positions will be the same. This makes CLONING in batch very useful, since only one spectrum need be calibrated before submitting a job to calibrate the rest and then perform Gaussian fitting. This part is the same as for ARC2D (see section C.3).

2. Choose the fit type. The type of fit to be performed is entered into the array .RES.CONTROL (see appendix 3.2.8 for a description of the results structure), for the relevant line and cross-section. This defaults to a single Gaussian with a base.

3. Perform the fit. The fit type is read from .RES.CONTROL, the fit is performed, and the results are stored in .RES.DATA.

4. Output results. This can be in the form of plots or tables for a range of variables.
The fact that LONGSLIT uses its own structure in the data file (the .RES structure, see appendix 3.2.8), means that the different stages above can easily be carried out at separate times, without needing to stay in the program between them.

There are two principle modes of operation for line fitting:

**AUTO mode** In this the fits, blocking etc. are defined beforehand, and then performed without any further reference to the user (except for interactive multiple fits). This can be carried out in batch, thus saving time at the terminal.

**MANUAL mode** In this the user can choose the blocking and fit type for each line as the fits are made, or the fit types may be defined in manual mode, but actually performed in AUTO mode, in batch for example. It is also possible to combine any range of cross-sections for a fit. Any previous fits may be displayed (this is the default), and it is possible to scan through the data, either individual cross-sections or with whole window’s being averaged up for each plot.

If you are unfamiliar with LONGSLIT you should first read sections C.2.2, C.2.2 and C.2.2. If you get stuck try looking at section C.2.2.

**Line Identification** Since ARC2D and ARCSDI have much in common with LONGSLIT as regards line selection, we will only describe this part once, noting the differences when they occur.

```%
longslit
(IMage) Name of image for input [1052-5VIG] -
1052-5VIG[2040,171] HH 1-2 HALPHA/[N II]

==========< Arc opts >==========

New : Set up line identifications from scratch
Repeat : Use existing line identifications
Clone : Use line identifications from another file
Fit option [REPEAT] - NEW
```

The options here are:

**NEW** Used on a file LONGSLIT has not been used on before, a .RES structure is created and the user is asked to locate and identify the lines.

**CLONE** As NEW, but used where line identifications are to be copied from another file.

**REPEAT** Used when a .RES structure already exists, that is if LONGSLIT (or ARC2D) has already been run on the file.

NEW mode is needed to set up line identifications.
This only affects the size of the arrays to be used, so you are not committed to 10 lines! You cannot, however, later have more than the number set here.

**MAXGauss**  
Maximum number of Gaussians to allow room for [5] -

Similarly this only sets the maximum allowed number of components in a fit. Note that this is the only “hard” limit to the number of components to be fitted when guesses are made for multiple component fits. A look at the data should allow a sensible value to be selected, five is a reasonable value (data rarely needs more components than this).

**YStart**  
Analysis lower limit [160] -

**YEnd**  
Analysis upper limit [200] -

A one-dimensional spectrum is extracted from the two-dimensional spectrum, and used for line selection. For LONGSLIT it is probably best to include all the data as here (the example data has 171 cross-sections), but for ARC2D and ARCSDI a central portion should be chosen (e.g. cross-sections 70–80 for a total of 171 cross-sections). At this stage the actual lines are located by marking to either side with a cursor. Be sure to include some base to either side, so that the optimisation routines can determine the base level precisely.

Locate Line : 1  
2nd tram line  
Locate Line : 2  
2nd tram line  
Locate Line : 3

A list of cursor options can be obtained by hitting the “?” key (hitting any key will then return to displaying the spectrum). Note that the options change depending upon whether the display is zoomed, and also upon whether any lines are identified (see below). If the key “Y” is pressed, this causes the program to set a new maximum Y value for the display based on the current cursor position. This is used to select weak lines, where the default display is such that they are small on the display, due to the presence nearby of much stronger line(s). The remaining lines are located.

Alternatively we could have used an automatic line-finding algorithm, by hitting key A. This option is available only while no lines have been found, or those found have only been located using this option (in which case the previously identified lines are discarded if the option is selected). This uses a routine adapted from SPICA, the predecessor to FIGARO. The automatic line finding algorithm checks for a minimum acceptable height for a line and for the values of the data dropping off to either side. It then finds the centroids and checks that the possible lines are above a minimum width. A check is also made for the line’s being sufficiently above the noise, and for not being a side-peak to another line.

The lines have now been located, LONGSLIT and ARC2D now need to know the identifications of these lines. Line lists are provided and users may create their own lists. Alternatively the “ID” option in the “Identification Menu” can be used, in which a wavelength and name are used as given. Since ARCSDI only aims to straighten the lines, you would leave ARCSDI here.
The user can create his/her own file for use here. No responsibility is accepted for the wave-
lengths of the lines in the lists supplied. The format is as below, and the files can be created
using an editor. The header consists of a number (as many as you want) of lines with an asterix
in the first column, the only thing that is done with these is that they are written to the terminal
(without the asterix). Blank lines are ignored, and all remaining lines are taken as data lines. The
data lines have the wavelength (read free-format) followed by the (optional) name, the array
for the names is character*10 line_name(number of line slots). These files can be placed in the
default directory (the program will search there first, so they can have the same names as those
supplied). The first few lines of the emission line list supplied are given here:

* Emission line list.
* Current version August 1990
78 lines read from EMISSION

3726.1600 [OII]
3728.9100 [OII]
4101.0000 HDELTA
4276.8300 [FeII]
4287.4000 [FeII]
4319.6200 [FeII]

It is possible to specify the FIGARO.ARC files, as alternatives, but if there is a file included with
Twodspec with the same name, excluding the extension, then the extension must be given (note
that most of these do not include line names, so these will be left blank). When you have read in
all the required line lists, you reply “ok” to the above menu.

The line with some of the surrounding spectrum is displayed so the user can, it is hoped,
recognise the line! If the displayed width is not satisfactory, it can be changed using the WIDTH
option of the menu shown below. A default response is given which is the value of the X array
at the peak in intensity within the line boundaries. If, however, you already have two or more
lines identified, the default will be interpolated or extrapolated from these (again using the positions of the maximum intensities within the line boundaries). It is possible to leave lines unidentified, although at present the only way to get back to the menu is by identifying at least one more line—you can edit the line list, but you then don’t have the advantage of the interpolation to help guessing (the fitting of the dispersion relationship in ARC2D can be used to help line identification). You can keep looping around the line list, identifying the lines in any order.

==========< Identification Menu >==========

[number] : Line wavelength
Width [number] : Redraw with different width
Next : Go to next line
Display : Start/stop displaying tables of wavelengths
Id [wavelength] [name] : Give line ID/wavelength (just taken as given)
Quit : Exit leaving lines unidentified
Identification Menu [ 6548.4] -
6548.1001 [NII]6548 ok? [YES]
Identify line number 2

==========< Identification Menu >==========

[number] : Line wavelength
Width [number] : Redraw with different width
Next : Go to next line
Display : Start/stop displaying tables of wavelengths
Id [wavelength] [name] : Give line ID/wavelength (just taken as given)
Quit : Exit leaving lines unidentified
Identification Menu [ 6563.1] -
6562.8169 HALPHA ok? [YES]

If you enter just a wavelength the nearest line in the lists will be located, and you will be asked if this is what you want (as shown):

==========< Identification Menu >==========

[number] : Line wavelength
Width [number] : Redraw with different width
Next : Go to next line
Display : Start/stop displaying tables of wavelengths
Id [wavelength] [name] : Give line ID/wavelength (just taken as given)
Quit : Exit leaving lines unidentified
Identification Menu [ 6730.5752] -
6730.8501 [SII]6731 ok? [YES]

CURRENT LINE LIST

----------------------------------------
line No identification wavelength
----------------------------------------
1  [NII]6548  6548.1001
2  HALPHA  6562.8169
3  [NII]6584  6583.6001
4  [SII]6717  6716.4702
5  [SII]6731  6730.8501

Edit line list? [NO]
Errors can be corrected using this edit option. The user now reaches the *Main Menu*:

```
Current value of iteration is 1

============= Main menu =============
ADd          : Add more lines
It           : Change iteration
Templates    : Input data for template based fits
AUTO         : Fixed window & fits type menu (use DEFINE to setup models)
Manual       : Interactive setup of window & fits
Define       : Define fits for AUTO
EDit         : Edit/look at results structures
T0ls         : Apply tolerances
Output       : Create output plots, etc.
SKy          : Sky subtraction and data vignetting corrections
SYnthetic    : Generate synthetic spectra from input data and/or fits
EXit         : Leave program
Main menu [MANUAL] -
```

Once you have located and identified lines in one file, this information may be *CLONED* to another file. A powerful feature of ARC2D and LONGSLIT is the ability to *CLONE* a set of line locations from another file, rather than, or in addition to (but preceding), the normal line selection. For this, the two arcs are plotted one above the other and, from three points marked by the user on each arc, the locations of the lines are interpolated for the new arc. The wavelengths and identifications are then also copied over. This means that if two spectra of the same wavelength range are to be used, without co-adding, then the whole process of line identification can be circumvented. If the spectra are very similar, then the information can be copied straight over, even in batch mode. The parts of the images to be extracted for the plots are prompted for.

**MAIN MENU**  This controls the interactive use of LONGSLIT, apart from the initial line identification.

The options are:

**ADD**  Add more lines

**IT**  Change iteration. This allows reduction of iteration, in order to allow previous fits to be over-ridden (they can also be over-ridden in MANUAL mode fitting).

**AUTO**  In this mode one defines fits beforehand, and then goes through the lines and cross-sections in order performing the fits. This is the mode of fitting in batch, and ideally should not be carried out interactively.

**MANUAL**  In this mode one does the fits in whatever order is preferred. This can also be used for checking fits performed in batch mode. The line selected is indicated above the menu. Initially this is line one in the list. MANUAL mode is also driven by a menu:

**Line name**  to go to that line. By stating the line name, or any unique abbreviation, the line will be set to that selected.
LAST Move back to previous line in line list.
NEXT Move on to next line in line list.
TRIM Limit the range of cross-sections. This is useful for altering the block start/end in
window mode (see below)
CHECK This displays 20 profile plots at a time. At present this will start at the current
line, cross-section one, and scan through. When a screen is full, the user is asked
whether he/she wants to see the next plots. This is designed for a quick look, before
more detailed checking and re-fitting, if that is required. It is also useful to provide a
convenient record of the line profiles to refer to, in which case the hardcopy option
should be used. Note that if the “quick plot” option here (which gives plots of similar
quality to softcopy plots) is not used the plots files can be very large.
FIT Fit this window as it stands
ADVANCE Go to next WINDOW
BACK Go to previous window
SEE Look at individual elements of window
WIDTH Change window width
SCAN Scan through windows
OLD Start/stop plotting old fits
DELETE Delete fits in this range
HARD Produce hardcopy of data with fit
EXIT Return to main menu

DEFINE Define fits for AUTO mode, for batch or interactive use. This defines fits, either for all
the lines at a time, or for each line separately. To define fits differently within a line, use
MANUAL mode as described above.

LOOK Look at values of data cube. This just lists the values in the results array.

TOLS Apply tolerances. This will reject fits which have, for example, too great a width.

OUTPUT These options are the main ways of producing output from the program. Selection of
this puts the user into the “OUTPUT MENU”, with the following options:

HARDCOPY Produce plots of profile fits. The line profile is displayed, together with the
fit. It is possible to preview the fits first in softcopy (unless all fits are being plotted,
using the option “plot whole cube”). If the option “plot whole cube?” is not being
used, then the X-axis of the plots can be in km s$^{-1}$ or Ångstroms, otherwise they are
in Ångstroms.

TABLE Print table of profile parameters

PLOT Plot rotation curves. Also width v. cross-section and flux v. cross-section plots
can be made. All these can be made in softcopy or hardcopy. Also a plot of average
velocities v. cross-section will be plotted, if the plot is in hardcopy, and there is more
than one line.

PRINT Print rotation curves

RATIO Evaluate line ratios
GREY  Produce greyscale plot of data. This is for one line at a time, using the wavelength limits for the line from the initial line location. A velocity scale is plotted on the X-axis.

CONTOUR  This is similar to the GREY option, but uses contours rather than a greyscale image.

FULL  This produces a table suitable for input to a FORTRAN program, including the information from both PRINT and PLOT.

CHECK  This produces an array of line profiles to which successful (or NAG error failed fits if allowed) have been made, with the fits superimposed.

SOFT  This says whether plots are in softcopy (as opposed to hardcopy). This only affects the PLOT, CONTOUR and CHECK options.

VELOCITY  Plot with velocity scale for X axis. This affects the PLOT and HARDCOPY options above.

EXIT  Exit and accept current setting of the options. When any selected plots etc. have been carried out, the user is returned to the main menu.

If the PLOT option is selected the user will be presented with the following menu, to select precisely which plots are required:

VELOCITY  Radial velocities
WIDTH  Widths
FLUX  Fluxes
AVERAGE  Average velocities
ALL  Velocities on 1 plot
EXIT  Exit menu and perform plotting

The output options (except GREY and RATIO) can be given in the command line as keywords. An example of the use of the OUTPUT option is given in section C.2.2.

SKY  Not yet implemented

SYNTHETIC  Not yet implemented

EXIT  Leave program

CUBAN  Display array of profiles (3-dimensional data only), and move around under cursor control. This is the same as the option in the main menu of FIBDISP (see below).

Use of LONGSLIT in Auto Mode  LONGSLIT can be used to fit profiles in an automatic mode which saves a great deal of time but still provides the user with many options for this process. To do so the lines must first be located as described in section C.2.2. The desired fit type must then be defined, by selecting the DEFINE option at the main menu.

We will assume that the fit model is to be defined interactively, in which case the user is then asked if the same model is to be used for all lines (if there is more than one line present), and to give the required model(s). In the following example all lines are to be fitted with double Gaussians ("untied" means that the two Gaussians are independent of each other).
Normally you would fit lines with the same model along all of the slit length, but in the following case line 2 is to be fitted with a double Gaussian (unconstrained) for the first 90 cross-sections, and a Cauchy function for the rest. The default fit type is a Gaussian with a base.

**DEFINE** can be used if you want to fit a whole line with a given type of fit e.g. a Lorentzian. The **ADD** option puts you back into the line selection as above, but without the option of automatic line location. The option is given to edit the line list again:

```
Lines identified:
[NII]6584 (6583.6)  [NII]6548 (6548.1)  HALPHA (6562.817)
```

Enter starting line number [1] - 2
(YBlock) Analysis x-sect width [5] - 90
Number of blocks = 2
Enter starting cross-section number [1] -

The line is displayed.

If a previous fit has been made on the same data, this is displayed at this stage (but this can be suppressed).

No previous fit

============ Manual Mode =============

[name] : Line name to go to that line
Last : Move back to previous Line
Next : Move on to next line in list
Trim : LIMIT the range of X-sects
Check : Check previous fits (20 to a screen)
Fit : Fit this window as it stands
Advance : Go to next WINDOW
Back : Go to previous window
See : Look at individual elements of window
Width : Change window width
Scan : Scan through windows
Old : Start/stop plotting old fits
Delete : Delete fits in this range
Hard : Produce hardcopy of data with fit
Exit : Return to main menu
Manual Mode [FIT] -

============ Fit Menu =============

GUess : Alter guessing ............ Multiples - SCRATCH, Singles - MAXIMUM
Base : Alter base model ........................................... CONSTANT
Bimodal : Test for bimodality before fitting
Instant : Define fits only
Absorption : Fit ABSORPTION lines
Tests : Alter tests for auto .................... AIC after fits (max cmps 5)
Nofit : Don't fit this position
Gaussian : Fit Gaussian(s) ........................................... ******
Cauchy : Fit a single Cauchy function
Lorentzian : Fit Lorentzian(s)
Single : Fit single(s) ........................................... ******
Tsep : Fit two Gaussians with fixed separation
TWidth : Fit two Gaussians with fixed width ratio
THeight : Fit two Gaussians with fixed height ratio
Double : Fit double(s)
Manual : Perform manual fitting
Auto : Automatic multiple Gaussian/Lorentzian
Transfer : Start from fit to another line
Exit : Exit menu and perform any fitting
Fit Menu [EXIT] - D

============ Fit Menu =============
The line is displayed.

No previous fit
Cauchy : Fit a single Cauchy function
Lorentzian : Fit Lorentzian(s)
Single : Fit single(s) ........................................... *******
TSep : Fit two Gaussians with fixed separation
TWidth : Fit two Gaussians with fixed width ratio
THeight : Fit two Gaussians with fixed height ratio
Double : Fit double(s)
Manual : Perform manual fitting
Auto : Automatic multiple Gaussian/Lorentzian
Transfer : Start from fit to another line
Exit : Exit menu and perform any fitting
Fit Menu [EXIT] - C

Just reply with “EXIT” from now on to leave the program.

=============< Manual Mode >=============

[name] : Line name to go to that line
Last : Move back to previous Line
Next : Move on to next line in list
Trim : LIMIT the range of X-sects
Check : Check previous fits (20 to a screen)
Fit : Fit this window as it stands
Advance : Go to next WINDOW
Back : Go to previous window
See : Look at individual elements of window
Width : Change window width
Scan : Scan through windows
Old : Start/stop plotting old fits
Delete : Delete fits in this range
Hard : Produce hardcopy of data with fit
Exit : Return to main menu
Manual Mode [FIT] - EXIT
Fits defined = 2
Current value of iteration is 1

=============< Main menu >=============

Add : Add more lines
It : Change iteration
Templates : Input data for template based fits
Auto : Fixed window & fits type menu (use DEFINE to setup models)
Manual : Interactive setup of window & fits
Define : Define fits for AUTO
Edit : Edit/look at results structures
Tols : Apply tolerances
Output : Create output plots, etc.
Sky : Sky subtraction and data vignetting corrections
Synthetic : Generate synthetic spectra from input data and/or fits
Exit : Leave program
Main menu [MANUAL] -
The setting up of lines in ARCSDI is similar to the above, except that the program is exited after the “OK?” question above (before line identification).

The automatic fitting of the lines can then be carried out by selecting the AUTO option in the main menu.

**Fitting Multiple Components in Auto mode**  If LONGSLIT, when in auto mode, comes across a fit defined as multiple, it will attempt to fit it with as many lines as it thinks are needed! It uses the tolerance settings for minimum and maximum width, and minimum height. These are used for the guesses and the final answers, and will probably need some tweaking. Also it will of course still need checking afterwards. It may well have more than one attempt at fitting. This checking is easily carried out using the CHECK option in MANUAL mode—it is best to use this to produce hardcopy plots which can be kept beside the terminal while you go through checking (you may find 20% of the profiles need re-fitting, but this will depend upon the line profiles, as well as the values of the tolerances). You may also need to investigate cross-sections for which no fit is given—this may be because there is no emission at that point, or the fit may have failed for some reason, even with strong emission (it may guess too many components for example, and then crash in the fitting). Note that LONGSLIT outputs to the screen the reason why the next component was not accepted when it searches for components. The display also contain details of any re-fitting. Thus if the fits are not acceptable after the first attempt at using this the user can alter the tolerances accordingly.

The option AIC in the fit defining menu determines whether Akaike’s information criterion (Akaike 1973) is used during multiple fitting to decide how many components to fit. This is true by default. The guesses are made in the same way, whether or not this is true, except that the widths are allowed to be a little larger—in both cases the guesses to the widths are allowed to the value in the tolerances array, to allow for errors in guessing (the fits are only accepted if they are actually within the tolerances), this ratio is larger if AIC is true. The value of the criterion is then calculated, and the best value is selected after progressively removing components to the fit. Thus if the guessing produces three components, the program will also “see what happens” with two components, one component and just a base. The criterion is

\[
C = M \times \ln S + 2 \times N
\]

where \(C\) is Akaike’s information criterion, \(M\) is the number of data points, \(S\) is the weighted sum of squares, and \(N\) is the number of fit parameters. The fit with the lowest value of \(C\) is used.

For fitting the tolerances should be as close as possible to the expected values of the parameters. The user is encouraged to learn to use this option, since it can give considerable time savings.

N.B. To refit a cube which has already been fitted, but the values changed, e.g. flux-calibrated, use the keyword COPY, which will use scaled existing values as first guesses, repeating all the fits in the “cube”. If used with CLONE, it enables similar spectra to be easily fitted.

**Fitting Gaussians etc. Interactively**  The main method of fitting these is the WINDOW option of MANUAL mode. First select MANUAL at the main menu.

========< Main menu >========
ADd : Add more lines
It : Change iteration
Templates : Input data for template based fits
Auto : Fixed window & fits type menu (use DEFINE to setup models)
Manual : Interactive setup of window & fits
Define : Define fits for AUTO
Edit : Edit/look at results structures
TOLS : Apply tolerances
Output : Create output plots, etc.
Sky : Sky subtraction and data vignetting corrections
Synthetic : Generate synthetic spectra from input data and/or fits
Exit : Leave program
Main menu [MANUAL] - MANUAL

Say which line you wish to start working on, give the required blocking (number of cross-sections added together) and the starting cross-section.

Lines identified:
[NII]6584 (6583.6)  [NII]6548 (6548.1)  HALPHA (6562.817)

Enter starting line number [1] - 3
(YBlock) Analysis x-sect width [5] - 10
Number of blocks = 2
Enter starting cross-section number [1] - 110

The line is displayed. This then puts you into MANUAL menu. Select FIT and then select the fit type required. MG allows the user to set the guesses required for optimisation, and to constrain the fits if required. TG allows the ratio of the heights or widths, or the separation of the lines, to be fixed. The other fit types are entirely automatic. Note that the “Cauchy” function is actually a function which varies smoothly between a true Cauchy function and a Gaussian (equation [4]).

No previous fit

============< M a n u a l  M o d e >============

[name] : Line name to go to that line
Last : Move back to previous Line
Next : Move on to next line in list
Trim : LIMIT the range of X-sects
Check : Check previous fits (20 to a screen)
Fit : Fit this window as it stands
Advance : Go to next WINDOW
Back : Go to previous window
See : Look at individual elements of window
Width : Change window width
Scan : Scan through windows
Old : Start/stop plotting old fits
Delete : Delete fits in this range
Hard : Produce hardcopy of data with fit
Exit : Return to main menu
Manual Mode [FIT] -
We choose MG for this example. The program plots the line profile with the current guesses (either taken from a previous fit or "guessed" from scratch).

HALPHA (line = 2)
Previous fit
Single emission Gaussian with base
successful
Will fit a multiple Gaussian
Number of gaussians in previous fit = 2
Guesses:

Component   Centre    fwhm    Height    Base
  1   6564.599   0.8904   255.1     2.934
  2   6561.983   1.485    23.98

The user is free to alter the parameters and to add or delete components. When satisfactory, the parameters are optimised. Hit “F” or click in the rectangle labelled “FIT”.

Background Models  The default model is a flat background, which is quite sufficient for many objects such as many emission nebulae where there is not a strongly varying background. For some objects, however, this will not be sufficient, and allowance must be made for a varying background (e.g. a stellar continuum). LONGSLIT provides three solutions to this problem

1. Fit a Chebyshev polynomial to the base immediately before fitting the line profile. The polynomial it fitted to the whole of the spectrum, except for those parts within the boundaries of the identified lines. The order for the polynomial is stored in the results block, and this is used to re-create the fit as required (e.g. for profile plots). Although this cuts down on storage requirements, it does mean that if you identify any further lines, these will cause the program to make incorrect assumptions. For this the blocking is, of course the same as for the profile fitting. The order can be varied as required for different lines. If this option is used, it is essential that the spectrum is not trimmed (subsetted) too close to the line.

2. Use a cubic spline to interpolate the values in the range of fitting. As with the Chebyshev polynomial the blocking is as for the fit, and the areas within the line boundaries are not used to constrain the splines.

3. Use Chebyshev polynomials, but perform their fitting using FITCONT. LONGSLIT can then pick up the coefficients (from a special coefficient structure), and use them to obtain the base. Since the blocking to be used by LONGSLIT is not known at this stage (by FITCONT at least), the fits are to single cross-sections, and the results are added when required. The areas to be excluded from the fitting are marked with a cursor (this is more versatile than when the fitting is performed by LONGSLIT).

To select a model other than a flat base, use the DEFINE option in the main menu, in which case you will be asked for the background model. Note that once set up, the model remains in force for fitting until the user leaves the program.

If you select option (i) above at the MANUAL menu, LONGSLIT gets you to decide upon the order to fit at that time—various plots are available to help. If you select this from DEFINE, then you be asked to decide upon the order before the first fit, unless you have already given it the order (in WINDOW for example). For this reason, this option cannot yet be used in batch mode.

Producing Output Plots, Tables etc.  When all the desired fits have been done (or earlier to get a check on them), you should used the standard function HARD to set the hardcopy device (like SOFT for a softcopy device), for example:

% hard ps_1
Then enter the program and at the main menu select “OUTPUT”. Then enter the options required e.g.:

......
Output : Create output plots, etc.
Exit : Leave program
Main menu [MANUAL] - OUTPUT

==============< Output Options >===============

Hardcopy : Plots of profile fits ............... F
Table : Table of profile parameters......... T
PLot : Plot rotation curves....................... F
PRint : Print rotation curves............... F
Ratio : Output line ratios (with plots).... F
Grey : Greyscale plot of velocity............. F
COntour : Contour plot of velocity.......... F
Full : Large table.............................. F
CHeck : Profile array........................... F
SOft : Plot in softcopy (* above) .... = T
Velocity : Use velocity scale (+ above) .... = T
Exit : Exit menu
Output Options - TABLE

The selection of options here is to “toggle” the option on or off, so entering an option once will select it, twice will cancel the selection. The options are performed when this menu is left.

==============< Output Options >===============

Hardcopy : Plots of profile fits ............... F
Table : Table of profile parameters......... T
PLot : Plot rotation curves....................... F
PRint : Print rotation curves............... F
Ratio : Output line ratios (with plots).... F
Grey : Greyscale plot of velocity............. F
COntour : Contour plot of velocity.......... F
Full : Large table.............................. F
CHeck : Profile array........................... F
SOft : Plot in softcopy (* above) .... = T
Velocity : Use velocity scale (+ above) .... = T
Exit : Exit menu
Output Options [TABLE] - PLOT

==============< Output Options >===============

Hardcopy : Plots of profile fits ............... F
Table : Table of profile parameters......... T
PLot : Plot rotation curves....................... T
PRint : Print rotation curves............... T
Ratio : Output line ratios (with plots).... F
Grey : Greyscale plot of velocity............. F
COntour : Contour plot of velocity.......... F
Full : Large table......................... F
Check : Profile array..................... F
Soft : Plot in softcopy (* above) ...... = T
Velocity : Use velocity scale (+ above) .... = T
Exit : Exit menu
Output Options [PLOT] - PRINT

========< O u t p u t O p t i o n s >========

Hardcopy : Plots of profile fits......... F
Table : Table of profile parameters...... T
Plot : Plot rotation curves.............. T
Print : Print rotation curves............ T
Ratio : Output line ratios (with plots)... F
Grey : Greyscale plot of velocity....... F
Contour : Contour plot of velocity...... F
Full : Large table....................... F
Check : Profile array.................... F
Soft : Plot in softcopy (* above) ...... = T
Velocity : Use velocity scale (+ above) .... = T
Exit : Exit menu
Output Options [PRINT] - EXIT

========< P l o t O p t i o n s >========

Velocity : Radial velocities .. = T
Width : Widths .............. = T
Flux : Fluxes .............. = T
Average : Average velocities . = T
All : Velocities on 1 plot = T
Exit : Exit menu
Plot Options [] - EXIT
Evaluate correction for Earth motion etc. [YES]

DEC -6 48 0
RA 5 33 56.8

Day of observation? [339] -
Date:- 339/1/1984
UT 17 51 13
Observatory position 149 3 57.91 -31 16 37.34

If the program cannot find information on the observatory, then it will ask. The reply can be the position, or an observatory name (type “HELP”) for help. Likewise the position of the object, date of observation etc. will be prompted for if required. Note that it is satisfactory to give the date as (for example) the 60th day of the year, giving the month as 1, rather than giving the true month, if this is preferred.

The velocity may be corrected to heliocentric, local standard of rest, galactic, or local group. This uses the STARLINK SLA library (Wallace 1990) and is based on the STARLINK program RV (Wallace 1987).

Is longitude ok, taking longitude as +ve west [YES] n
V_HEL=-4.16896E+00 V_LSR= 1.40857E+01
V_GAL= 1.33324E+02 V_LGROUP= 1.38917E+02
Velocity Correction

Hel : Sun's ref frame
Lsr : Local standard of rest
GA1 : Galaxy ref frame
GROUP : Local galactic group ref frame
OTHER : Other - enter from terminal
Velocity Correction [LSR] -
OK? [YES]
Show fits which had NAG errors [NO]
Is data flux calibrated? [NO]
Printing fits
Indicate presence of failed fits [NO]

Flux Marking

No : Don't mark points according to flux
Continuous : Mark using a continuous scale
Three : Mark by grading into 3 divisions
Flux Marking [CONTINUOUS] -
Plotting Velocities
Line number 1
Line number 2
Line number 3
Plotting average velocities
Plot flux v. xsect? [YES] n
Plot width v. xsect? [YES] n
Job 123 entered on queue SYS_LASER
Print of velocities

Unless NOFIT was specified (in which case the user would miss out the main menu entirely), the user is now returned to the main menu.

The option to mark the points can either put the markers so that the most intense 3rd of the points have a filled circle, the next 3rd an open one, and the remainder are not marked, or the area can be proportional to the logarithm of the flux. The latter is recommended.

Note that if NOLABEL is given in the command line the titles will be omitted for many of the output plots.

In some cases LONGSLIT may not have been able to obtain errors on the fits — this involves inverting a matrix. For such cases the errors will be given as zero (obviously not a correct value!) when the fits are listed.

The Keyword FIT This will cause the program to run though the fitting part of the program, by default it is true, so may be ignored. It is, however, possible to specify NOFIT in a command line, to run LONGSLIT in batch without performing fits (or just to skip the middle part of the program when running interactively) e.g.:

% longslit t1 repeat table nofit

This will produce a table of the fit results. It may be used for other output options (except RATIO), e.g.: 
In batch correction may be made for the Earth’s motion by using the parameter VCORR as above, this will be subtracted from the radial velocities calculated.

TRANSFER Option  The OL option in select fit type is for use with the TRANSFER option to transfer fits between different lines. For this to work the fits must be defined using the DEFINE option, before fitting in AUTO mode, preferably in batch. The fits are copied so that one line becomes a copy of another as regards fit type, blocking, etc., but of course the fits are optimised on the data for the current line. If the fits are not re-defined, then other modes will not work for that line (except COPY). TRANSFER does NOT override masking as COPY does, and is selected by specifying OL as the fit type. If this option is used, all the cross-sections of a line should be so defined.

INHERIT Option  This is an option to do any fitting in batch, provided the program has a fit to one block, and the data is fairly continuous. Value -1 to inherit from previous block, +1 to inherit from next. The fit to this block used as the guess for the next fit. Likewise INHERIT must be given on the command line.

Use of Tolerances  The option to apply tolerances interactively is selected at the main menu. First of all the user is asked to confirm (and alter if required) the values of the tolerances to apply, and then the user can select which tolerances to use. It is possible merely to set the values, but not to apply them at the time, for example if you wanted to set them up for use in batch mode.

In the following example the maximum allowed height is set to $10^7$, and then tolerances are applied on height, centre and width.

```
% longslit t1 repeat print table plot hardcopy nofit vcorr=10
```

In the following example the maximum allowed height is set to $10^7$, and then tolerances are applied on height, centre and width.

```
Main menu [MANUAL] - TOLS
Tolerances on position are relative to the line’s rest wavelength

=========< Edit Tolerances >=========

V_Tol %F : Absolute error of line centre .... = 1.8994141E-02
V_MAX %F : Max allowed line centre .......... = 3
V_MIN %F : Min line centre .................. = -3
W_Tol %F : Absolute error of line width ..... = 1.8994141E-02
W_MAX %F : Max allowed line width ........... = 4
W_MIN %F : Min allowed line width ........... = 0.1899414
W_S_n %F : Width signal/noise ............... = 5
H_MAX %F : Max allowed line height .......... = 1000
H_MIN %F : Min allowed line height .......... = 20
H_S_n %F : Height signal/noise .............. = 5
C_tol %F : Absolute error of Cauchy parameter = 2.0000001E-03
S_tol %F : Absolute error of Skew parameter = 2.0000001E-03
H_Tol %F : Absolute error of height ......... = 3
Apply : Apply tolerances now
Exit : Exit but don’t apply tolerances now
Edit Tolerances - H_MAX 1E7

=========< Edit Tolerances >=========
```
V_Tol %F : Absolute error of line centre .... = 1.8994141E-02
V_MAx %F : Max allowed line centre ........... = 3
V_MIn %F : Min line centre .................. = -3
W_Tol %F : Absolute error of line width ...... = 1.8994141E-02
W_MAx %F : Max allowed line width ........... = 4
W_MIn %F : Min allowed line width ........... = 0.1899414
W_S/n %F : Width signal/noise ............... = 5
H_MAx %F : Max allowed line height .......... = 1.E7
H_MIn %F : Min allowed line height .......... = 20
H_S/n %F : Height signal/noise .............. = 5
C_tol %F : Absolute error of Cauchy parameter = 2.0000001E-03
S_tol %F : Absolute error of Skew parameter . = 2.0000001E-03
H_Tol %F : Absolute error of height .......... = 3
Apply : Apply tolerances now
Exit : Exit but don’t apply tolerances now
Edit Tolerances - APPLY

============< Set Rejection Criteria >============

Height : Test on HEIGHT....... = F
Centre : Test on CENTRE....... = F
Width : Test on WIDTH........ = F
Errors : Test on ERRORS....... = F
S/n : Test on S/N.......... = F
SHApe : Test on SHAPE........ = F
SEParations : Test on SEPARATIONS.. = F
Log : Log failures ........ = F
Apply : Apply tolerances now
Set Rejection Criteria - CENTRE

============< Set Rejection Criteria >============

Height : Test on HEIGHT....... = F
Centre : Test on CENTRE....... = T
Width : Test on WIDTH........ = F
Errors : Test on ERRORS....... = F
S/n : Test on S/N.......... = F
SHApe : Test on SHAPE........ = F
SEParations : Test on SEPARATIONS.. = F
Log : Log failures ........ = F
Apply : Apply tolerances now
Set Rejection Criteria - HEIGHT

============< Set Rejection Criteria >============

Height : Test on HEIGHT....... = T
Centre : Test on CENTRE....... = T
Width : Test on WIDTH........ = F
Errors : Test on ERRORS....... = F
S/n : Test on S/N.......... = F
SHApe : Test on SHAPE........ = F
SEParations : Test on SEPARATIONS.. = F
Log : Log failures ........ = F
Apply : Apply tolerances now
Set Rejection Criteria - WIDTH

=========< Set Rejection Criteria >========

Height : Test on HEIGHT....... = T
Centre  : Test on CENTRE....... = T
Width   : Test on WIDTH....... = T
Errors  : Test on ERRORS....... = F
S/n     : Test on S/N.......... = F
Shape   : Test on SHAPE....... = F
SEparations : Test on SEPARATIONS.. = F
Log     : Log failures ........ = F
Apply   : Apply tolerances now
Set Rejection Criteria - APPLY
Testing on: HEIGHT CENTRE WIDTH
75 fits rejected

The option to log the failures lists, for each fit at each cross-section which fails the tolerances, the reason why it failed, and the value of the relevant parameter. For a large data set this can produce a large amount of output! Returned fits will also be listed (if you relaxed your rejection criteria). Otherwise the number of fits rejected is all that is listed. Note that the testing ignores blocking, so each cross-section for each line is counted as a separate fit.

The tolerances are as follows, and apply to each component of the fit unless otherwise stated (i.e. if one component fails, the fit is rejected):

**HEIGHT**  Fits are accepted only if their height is greater than H_MIN and and less than H_MAX.

**CENTRE**  Fits are accepted only if their centre is greater than V_MIN and and less than V_MAX.

**WIDTH**   Fits are accepted only if their width is greater than w_MIN and and less than W_MAX.

**ERRORS**  Fits are accepted only if the error on the height is less than H_TOL, that on width less than W_TOL and that on centre less than V_TOL.

**S/N**     Fits are accepted only if the ratio of Height to Error on height is greater than H_S_N and if the ratio of width to its error is greater than W_S_N.

**SHAPE**   The fit is rejected if the error on a Cauchy fit is greater than C_TOL.

**SEPARATIONS** For the moment this is ignored!

**Summary of Parameters**

**IMAGE** (char) Name of image for input, This is the data and should be a [FIGARO] format data file. This should also have a .X axis array which contains the wavelengths of the lines. For the identification files supplied with the program the units should be Ångstroms. However, if the user supplies his/her own files, this need not apply, although some plots may have the wrong labels.

**ARC_OPTS** (char) Fit option:
NEW (N) Set up a new analysis
REPEAT (R) Iterate on previous analysis
CLONE (C) CLONE an analysis from another file

YStart (int) Analysis lower limit
YEnd (int) Analysis upper limit
YBlock (int) Analysis cross-section width
ITeration (int) New value of iteration
MAXLines (int) Maximum number of lines to allow room for
CLfile (char) Name of image for CLONING from
VCorr (real) Correction to apply to radial velocities
TOls (char) For use in batch only
INherit (int) Number to control inheritance of previous fits:
  0 no inheritance of fits
  1 inheritance from next block
  -1 inheritance from previous block
DEvice (char) Device to use for plotting (greyscale)
FITRat (real) Ratio of widths, heights, or separation, for double fits
CAIrat (int) Ratio of number of iteration to default
WHite (float) Level to plot as white (greyscale option)
BLack (float) Level to plot as black (greyscale option)
MAXGauss (int) Maximum number of Gaussians that can be fitted to a profile.
TStart (int) Analysis lower limit
TEnd (int) Analysis upper limit
TBlock (int) Analysis width in T direction
FIT_MODEL (char) Model for fitting
PLOTLim (float) Limits of plot (world coordinates). This is to allow velocity plots to be forced
to all have the same scale, making comparison easier.

HArdcopy (key) Produce hardcopy plots of fits from cube
TAble (key) Produce table of fits from cube
PLOT (key) Produce plots of rotation curves
PRInt (key) Produce print out of rotation curves
KEEP\_ITT  (key) Keep iteration files. These files contain details of the fitting process. If a fit succeeds the file is always deleted, if it crashed it is always kept, this keyword controls whether it is deleted if a NAG error occurs.

FIT  (key) Perform fitting

COPY  (key) Copy previous fits This will repeat all the fits previously made, which is likely to be of use if data is co-added after one file has been analysed. Also, when used with CLONE the entire .RES structure is copied without any change. For the new fits the previous fits (suitably scaled) are used as first guesses.

ABsorption  (key) Allow fitting of absorption lines. This allows absorption fits to be defined—once defined they can be fitted whatever the value of this.

BOunds  (key) Perform bounded fits to lines (in batch). This only bounds the widths.

LAbel  (key) Put labels on plots. This is true by default, but it may be preferable if plots are to be used in a paper to not have labels.

CONtour  (key) Produce contour plots

GRey  (key) Produce greyscale plots

LOG  (key) Use log scale for greyscale and contour plots

WEIghts  (key) Use weighted fitting (default is true).

PRFits  (key) Print out details of fitting. This is true by default, but if you wish to avoid large log files set it to false.

FULL  (key) Print out full details of fits in table. This provides the information in a form which is easier to read into a FORTRAN program, and includes the information given in PRINT and TABLE.

CHECK  (key) This produces an array of line profiles with the fits superimposed.

The options HARDCOPY, TABLE, PLOT, PRINT, SHAPE and FULL are the same as for the OUTPUT MENU, except that in batch mode HARDCOPY will work on all line profiles/blockings which have a successful fit. CONTOUR and GREYSCALE work on all the lines in batch mode.

Note that the use of TSTART and TEND is with 3-dimensional data files. At present LONGSLIT only partially supports these, but some of the same code is used as in FIBDISP.

The keyword PRFITS is true by default but the user may set it to false to avoid large log files when running in batch.

C.2.3  FIBDISP—Analyise 3-d Data (spectral)

This performs a similar function to LONGSLIT, but is optimised for three-dimensional data arrays. In interactive use a greyscale softcopy device such as a GWM (the X window you get from GKS) is used, and profiles are selected using a cursor. Alternatively an array of profiles may be displayed using line graphics. If run in batch mode the whole data block may be fitted with Gaussians etc. The same fitting options are available as for LONGSLIT (see section C.2.2).
The file should already have a results structure (unlike LONGSLIT, FIBDISP does not create one), this can be created by FIB2CUBE or LONGSLIT.

FIBDISP can handle arrays in which the pixels correspond to a rectangular or hexagonal grid on the sky.

On entry to the program the user is prompted for the name of the data cube and then enters the main menu.

Main Menu  The options in the main menu are as follows:

RESULTS Display a plane of the results block. The plane is accessed by name, and this is one of only two places in the whole of Twodspec that cares about the case of the answer given by the user (the other is MODPARAMS). For example, if you wanted to display the centre of the first component, you would enter “Centre_1”.

DATA Display a plane of the data

PROFILE Examine line profiles. The profiles are selected using a cursor. If more than one is selected they are added together. Gaussians etc. may be fitted to the profile, and the results stored. The fitting is similar to that of LONGSLIT. Hard-copy plots of the profile may also be made.

XCUT Take a cut through the data in the X direction, and display on a greyscale device. The position is marked with a cursor, and the nearest pixels are chosen (i.e. no interpolation).

YOUT Take a cut through the data in the Y direction, and display on a greyscale device. Similar to XCUT.

XOUT Output to file X direction cut through data. This is like XCUT, but the output is to a file, rather than an image display.

YOUT Output to file Y direction cut through data. Similar to YOUT.

IT Reduce iteration

CHECK Display an array of line profiles—this can be in soft or hard-copy

TOTAL Display the summed intensity through the image

LIMIT Limit X and Y range for display. This acts as a toggle, so if you have limited plots and select this option again, you will return to full range.

TOLS Apply/set tolerances

LOOK Look at values in results cube

DELETE Delete fits from results cube

DEFINE Define fits for batch mode. The fits are defined and stored in the control array.

OUTPUT This allows the user to list all the fits onto the line printer, or to produce hardcopy plots of all the fits.

EXIT Leave the program
CUBAN  Cuban-style display/motion. This is better than CHECK for large data-sets (such as from TAUrus). An array of profiles is plotted, with a small greyscale representation of the total intensity in the top right-hand corner—the centre of the array plot is marked on this with a cross. The user has the following options (decided by cursor keys):

- **A**  Add profiles to fit (end with F)
- **C**  Centre here
- **D**  Down
- **E**  Exit
- **F**  Make fit to point
- **H**  Make hardcopy of current plot
- **J**  Jump to new area (using greyscale plot)
- **L**  Left
- **P**  Indicate position
- **R**  Right
- **S**  Set scaling for profile plots
- **U**  Up
- **X**  Erase fit
- **?**  Help

Note that the following options require the use of a greyscale display: RESULTS; DATA; XCUT; YCUT and TOTAL, although PGPLOT is capable of producing a simulated greyscale on line graphics devices, if the array is rectangular.

Summary of Parameters

- **CUbe** (file) Cube for display
- **YStart** (int) analysis lower limit
- **YEnd** (int) analysis upper limit
- **YBlock** (int) Enter analysis x-sect width
- **TStart** (int) analysis lower limit
- **TEnd** (int) analysis upper limit
- **TBlock** (int) Enter analysis blocking width in 3rd dimension
- **DEvice** (char) Device for display
- **IIteration** (short) New value of iteration
- **VCorr** (float) Correction to apply to radial velocities
- **TOls** (char) For use in batch only
FITRat (real) Ratio of widths, heights, or separation, for double fits

CAlrat (int) Ratio of number of iteration to default

OUTPut file: Name for output file

FIT_MODEL (char) Model for fitting

LOW (float) Minimum value for display

HIGH (float) Maximum value for display

Absorption (key) Allow fitting of absorption lines

BOunds (key) Perform bounded fits to lines (in batch)

HArdcopy (key) Produce hardcopy plots of fits from cube

TAble (key) Produce table of fits from cube

PRInt (key) Produce print out of radial velocities

KEEP_ITT (key) Keep iteration files

FIT (key) Perform fitting

WEIghts (key) Use weighted fitting (default is true). This only applies if the file contains an error array.

PRFits (key) Print out details of fitting. When fitting large data-sets in batch this would normally be set to false, otherwise a very large log file will be created.

SHAPE is not yet implemented.

C.3 ARC2D—Wavelength Calibration

C.3.1 Introduction

Once the data has been corrected for distortions or has been “cleaned”, it must be converted to a linear (calibrated) scale in wavelength, or at least the relationship must be defined. In practice it is much easier to handle a re-binned file.

The “traditional” way of doing this is to take one or more spectra of an arc lamp (which has lines of known wavelength), using the same spectrometer arrangement, fairly close in time to the exposures of the objects. This spectrum will contain information on the wavelengths corresponding to different channels. The user will then indicate the locations of the lines in some way to a program, and then this program will find the centroid of each line at each cross-section, or group of cross-sections—a block). These positions are then used to obtain the relationships of the channel number to wavelength, by fitting polynomials to the line positions and wavelengths. The coefficients of these are then used for the re-binning onto a linear scale.

This reduction method has several disadvantages:
• The process of finding centroids is often less accurate than fitting Gaussians, since is is more prone to errors due to poor signal to noise, or a strong base (especially if this is not constant).

• Finding centroids provides no information on the likely error in the centre. Ideally the strongest lines should be given the highest weights. Conventional programs use unweighted polynomial fitting.

• In treating each cross-section or block separately, the fact that the line centres will vary smoothly (for “normal” data) across the image is ignored. This can give rise to steps in what should be smooth lines, for the re-binned image. If the data from successive cross-sections in blocked together then, even if all the centres lie on a smooth curve, steps will occur in the calibrated data.

ARC2D is a two-dimensional arc calibration program. It makes use of a calibration arc as above, but Gaussians are fitted to the arc-lines in order to locate them with maximum precision, as well as to give information on errors. The continuity of the arc lines is taken into account. Rather than locating the line centres immediately prior to fitting the polynomials (to determine the dispersion relationship) whilst travelling up the spectrum, ARC2D finds all the line centres, and the fit to any cross-section can be checked, and poor lines rejected. Then all the polynomials can be fitted and, if satisfactory, written to the output file. Conventional programs require the order of the polynomial and the arc lines to be used to be guessed, before any fits are made (or at best when only one cross-section or block has been fitted)!

The line identification part of ARC2D is the same as for LONGSLIT (see section C.2.2).

The line positions ARC2D has at this stage are approximate since they are only lower and upper limits in channels for the optimisation.

The lines are then accurately located (automatically) at each cross-section by fitting Gaussians to them, assuming a Gaussian can successfully be fitted. For this the data from successive cross-sections can be blocked together, to improve the signal to noise ratio. In our case we typically co-add ten successive cross-sections. This fitting is a least squares optimisation, including a flat base, and is the same as used for LONGSLIT.

ARC2D creates a file for use with ISCRUNCH. Weighted fitting may be used for the polynomials to determine the dispersion relation. In addition polynomials may be fitted to the relation of line centre to cross-section for each line, and the polynomial values from this fitting used instead of the Gaussian centres, to give a smooth variation of the dispersion relation across the spectrum.

To use ARC2D you should do the following:-

(1) Set up tram arrays—to to tell the program where the lines are and to identify them

(2) Fit Gaussians to lines.

(3) Apply continuity correction (this is not essential).

(4) Fit polynomials to define the correction to be applied to “scrunch” the data (you automatically end up here unless you answer “YES” to leave program now...).

(5) If you are satisfied with the fits, create a calibration file and use it as input to ISCRUNCH.
Parts (1) and (2) above are as for LONGSLIT, except that only AUTO mode is available for (2) and the program will follow any curvature of the lines during fitting.

In the main menu LOOK, TOLS, EXIT and ADD are the same as in LONGSLIT, and GAUS is similar to the AUTO option in LONGSLIT. POLY and DISP are described in the next two sections. SOFT and HARD produce plots of use in determining which lines to include in the fitting. These are of line centre against cross-section, line width against centre and error on centre v. height.

### C.3.2 Continuity Correction

In order to avoid “steps” in the scrunched data, it is advisable to use this option for long-slit spectra (of course this isn’t applicable to data such as form fibre arrays where a smooth curve is not to be expected). Select the POLY option in the main menu:

```
=============< Main Menu >============

  Look : Look at values of data cube
  Gaus : Fit Gaussians to line profiles
  Soft : Produce soft-copy plots of diagnostics
  Hard : Produce hard-copy plots of diagnostics
  Tols : Apply tolerances
  Disp : Evaluate dispersion relation
  Add : Add more lines
  Poly : Fit polynomials in X-Sect direction
  Exit : Leave the program
  Main Menu [EXIT] - POLY
  keep fits with nag errors for poly fitting? [NO]
  Weight fits? [YES]
  Performing weighted fit

A plot of the sum of the squares of the residuals against order is displayed.

  Go onto next plot? [YES]
  SEEK MENU

The residuals are plotted against cross-section. When these appear to be due to noise only, answer “NO” to leave the loop, and reply with the order required. A plot of the fit over the centres is given, and similar plots are displayed (two to a page) for the remaining lines, but can be suppressed.

  Y for next plot; N to stop [YES]
  Y for next plot; N to stop [YES]
  (Order) order for polynomial fitting [3] -
  Order returned = 3
  Further plotting may be suppressed by typing ‘‘N’’
  Go onto next plot? [YES]
  Performing weighted fit
  Go onto next plot? [YES]
  Performing weighted fit
  Go onto next plot? [YES]
Performing weighted fit
Go onto next plot? [YES] NO
Performing weighted fit
Produce hardcopies of line centre & residual plots [NO]

=============< C o n t i n u i t y   F i t s >==============

Accept : Accept fits
Retry : Try again
Quit : Give up
Continuity Fits [ACCEPT] -

The results are now stored and the user returned to the main menu.

C.3.3 Evaluating the Dispersion Relation

Select the DISP option at the main menu:

=============< M a i n   M e n u >=============

Look : Look at values of data cube
Soft : Produce soft-copy plots of diagnostics
Hard : Produce hard-copy plots of diagnostics
Tols : Apply tolerances
Exit : Leave the program
Disp : Evaluate dispersion relation
Gaus : Fit Gaussians to line profiles
Add : Add more lines
Poly : Fit polynomials in X-Sect direction
Main Menu [EXIT] - DISP

You now are given a list of the lines on the graphics screen. You can hit “?” to see what the options are. This part of the program is designed to be used with a cursor, rather than using menus. You can edit line lists, set parameters, etc. Typing “f” will then perform a fit and you will be given plots of the fit and residuals.

When you have the fit/residuals plots, the lines used are shown with filled-in markers, and open markers indicate unused lines. Note that, although the lines used are updated on the lower plot, only when you hit “f” again will new fits be performed, so until then the residuals etc. will refer to the old fit.

ARCFIT : Order = 3, cross-section 85

Coefficients of fit are -

8.27062E-16  -3.97077E-12  3.86509E-09  4.17852E-06  0.00000E+00  0.00000E+00

Start wavelength = 4827.310,   End wavelength = 5152.441
Central wavelength = 4989.621,   Mean dispersion (per channel) = 0.1601613
A list of line wavelengths with fitted wavelengths is written to the terminal. If there are any lines not used in the fit, these are still output (after those which are used), so that, if required, estimates of the wavelengths of unidentified lines can be obtained.

The fits should be checked at several positions across the data and, when satisfactory, the tables etc. should be suppressed, and the results written to a file (the “A” option will accept the fits). If “continuity corrected” data is used, then each cross-section is treated separately, otherwise each block is treated as one fit.

Minimum start wavelength = 4824.331, maximum end wavelength = 5153.783
Copy any coefficients from one line to another? [NO]
Fits OK? [YES]

Summary of Image Arc Fit Results -
-----------------------------------

Image dimensions 2040 by 171
Number of rows that could not be fitted = 0
Maximum chi-Squared error = 0.00
Maximum degree polynomial used = 3

Fit results written to matching .iar file

=============< Main Menu >=============

Look : Look at values of data cube
Gaus : Fit Gaussians to line profiles
Soft : Produce soft-copy plots of diagnostics
Hard : Produce hard-copy plots of diagnostics
Tols : Apply tolerances
Disp : Evaluate dispersion relation
Add : Add more lines
Poly : Fit polynomials in X-Sect direction
Exit : Leave the program
Main Menu [EXIT] -

C.3.4 Summary of Parameters

IMage (file) Name of image for input. This should be a file containing an arc spectrum.
ARC_OPTS (char) Enter arc fit option:

- NEW (N) set up a new wavelength calibration
- REPEAT (R) Iterate on previous calibration.
- CLONE (C) Clone a previous calibration.

YStart (int) analysis lower limit. The data between the limits ystart and yend is extracted and
the resultant spectrum is used to locate the lines.

YEnd (int) analysis upper limit. The data between the limits ystart and yend is extracted and
the resultant spectrum is used to locate the lines.

YBlock (int) Enter analysis x-sect width. Each window is of this width (except perhaps the final
one).

Iteration (short) New value of iteration

Order (int) order for polynomial fitting. This is for the continuity correction of the data. Ideally
the arc should have been pre-processed with ARCSDI, so a low order e.g. 2 should be
used.

MAXLines (int) Maximum number of lines to allow room for. This must be greater than or
equal to the number of lines fitted, so room should be allowed in case any more are to be
added later.

CLfile (file) Name of image for cloning from. This should be a file containing an arc spectrum.

TOIs (char) For use in batch only

KEEP_ITT (key) keep iteration files

PRFits (key) Print out details of fitting

C.4 COMB and ARCSDI-Correction for Geometrical Distortions

C.4.1 COMB—Correction for S-Distortion

This program takes a long-slit spectrum with one or more continua, and creates a file which
allows it to apply a correction to further files, such that the continua are made straight. This
performs a similar function to the FIGARO function SDIST, but is more automatic (it may be run
completely in batch if required).

Since the philosophy of all the software we have written is to be as automatic as possible, COMB
will automatically find the “teeth” of the comb, and can be run completely in batch, if so required.
This enables quicker processing of data, since in practice the order required for the Chebyshev
polynomials is found to be 3 for all combs on which the program has been run. It is probably
best, however, to initially locate the teeth interactively (for setting the values in the tram arrays,
which define the edges of the line at the centre of the data), since some experimentation may be
required. The operation of COMB is as follows:
(1) Locate ‘teeth’. A cut is taken from the data, along the slit direction, from the central 20 channels of the data. The algorithm looks for the highest point in this data, and searches outwards to find other teeth, whether a new “tooth” is accepted depends upon the value of the parameter LEVEL, which is the minimum acceptable ratio of an new tooth’s height to that of one already accepted. If too many or too few teeth are found, a different value of LEVEL may be used, or the teeth may be selected manually. To confirm the correct location of teeth, these are output to the user (or batch log file), at the start of the program. Thus a check of the teeth locations may also be made if they are located in batch.

(2) The program then follows the “teeth” along the image (in the channel direction), finding the line centres by fitting Gaussians, or alternatively by finding centroids. Finding centroids is quicker than fitting Gaussians, and may be more suitable for some data. Fitting Gaussians is often more accurate, and gives estimates of the errors. For this the program will co-add a number of channels (say 20) to obtain a higher signal to noise (and reduce the time taken to find the centres). The teeth are located in the centre of the data (in the channel direction), and followed out from there in both directions (they are likely to be strongest in the centre). Since the variation along the image may be considerable, the tram positions (the Y values of the range used for fitting) must be updated using the centre from the last fit, before being used. This enables the program to follow the continua.

(3) The points so obtained are fitted with Chebyshev polynomials (for each tooth). If the parameter ORDER, the order of the polynomial to be fitted is specified in the command line, then it is taken as that. Otherwise the program plots the residuals for each order until told to stop, and then prompts for the order. In batch mode the program will exit at this stage unless ORDER is given in the command line. This is similar to the continuity correction part of ARC2D (see section C.3.2).

(4) The Chebyshev polynomials are then evaluated at each channel for each “tooth”, within the limits used for fitting the polynomials, and the points outside these limits are evaluated using (local) cubic splines (Akima 1972). When a value for the cross-section position of each “tooth” has been obtained, these are used for interpolating (again using cubic splines), the values of the data at the positions at the current channels corresponding to each cross-section at the reference position (the central channel). The correction is such that the data is lined up with this reference channel.

COMB outputs plots of the locations of the teeth at the start (i.e. at the central channel) and a few other points along the data, and also outputs a plot of the positions of all the points on the teeth found by the Gaussian fitting, enabling a check to be made as to whether the results seem reasonable. Plots of the polynomial fits to the teeth and of the residuals on these fits (typically up to about 0.2 cross-sections) are also output. If running interactively these are in soft-copy, otherwise in hard-copy. To apply the correction, use OLD mode, the correction file (i.e. the file with the polynomial coefficients) is COMB.GMC.

This is a program to correct data for S-distortion by moving data in the cross-section direction to line it up for a comb of continua spectra. This correction is then applied to the data itself. A comb dekker is used to produce about ten continuum spectra across an image (this is done at the telescope). This image is then used by the program:- The program locates the teeth and follows them along the image (in the channel direction), finding the line centres by fitting Gaussians. The points so obtained are fitted with Chebyshev polynomials (for each tooth). The intermediate positions are interpolated from these, which are then used to evaluate the required movement.
for each data point. The coefficients are written to a file which may then be read by the program to apply correction to the actual data.

Alternatively, if QUICK is specified, centroids are used rather than the fitting of Gaussians. Since fitting Gaussians is generally quite fast, this is only really likely to be needed when the profiles are significantly different from Gaussian.

Summary of Parameters

IMage (file) Name of image for input. This should be a file containing continua spectra.

ARC_OPTS (char) Enter arc fit option:
- NEW (N) set up a new wavelength calibration
- REPEAT (R) Iterate on previous calibration
- CLONE (C) Clone a previous calibration
- OLD (O) Correct using previous results

OUTPUT (file) Name of output file. File to contain corrected data.

XStart (int) analysis lower limit. The data between the limits xstart and xend is extracted and the resultant spectrum is used to locate the lines.

XEnd (int) analysis upper limit. The data between the limits xstart and xend is extracted and the resultant spectrum is used to locate the lines.

XBlock (int) Enter averaging width in channels. Each window is of this width (except perhaps the final one).

ITERation (short) New value of iteration

LEVEL (float) Level of edge of tooth

ORDER (int) order for polynomial fitting This is for the continuity correction of the data.

MAXLines (int) Maximum number of lines to allow room for. This must be greater than or equal to the number of lines fitted, so room should be allowed in case any more are to be added later.

CLfile (file) Name of image for cloning from. This should be a file containing an comb spectrum.

TOls (char) For use in batch only

KEEP_ITT (key) keep iteration files?

QUick (key) Centroid rather than fit Gaussians?

PRFits (key) Print out details of fitting

PLOtcorr (key) Plot correction?
Main Menu

LOOK  Look at values of data cube

SOFT  Produces soft copy plots of diagnostics

HARD  Produces hard copy plots of diagnostics

TOLS  Apply tolerances

POLY  Fit polynomials to results

CENTRES  Find line centres-takes a long time. This is similar to the GAUS option of ARC2D.

ADD  Add more lines

EXIT  Exit program

C.4.2  ARCSDI—Correction for Line Curvature

This corrects data for line curvature, using an arc spectrum. The function is the same as for comb, but it works in the perpendicular direction (again this is for long-slit spectra).

The aim is to give ARC2D a spectrum with fairly straight arc lines, so it can ‘block’ more cross-sections together to obtain a higher signal to noise. The operation is similar to parts (1) to (3) of ARC2D, except that ARCSDI does not need to know the line identifications. To apply the correction, use OLD mode, the correction file (i.e. the file with the polynomial coefficients) is ARCSDI.GMC. The main menu is the same as for COMB.

The arc lines are located using much of the same code as in ARC2D described below, including the fitting of Gaussians. The reason for doing preliminary correction of this nature, rather than using ARC2D straight away, is that for data which is fairly noisy, it is desirable to give ARC2D as many cross-sections at a time for the fitting. It is intended that ARCSDI be used with a long exposure from a chimney arc. Another problem overcome by this is that of vignetting which occurs in the comparison arc optics of the RGO spectrograph at the AAT. ‘Chimney’ arcs are reflected off the dome, and do not suffer from this problem (the same optical path is used as for the observations). It is often an advantage to use an arc from another wavelength range, if this has more suitable lines, or if they are better distributed along the spectrum.

Once the lines are located, they are treated in much the same way as the positions of continua in COMB. Chebyshev polynomials are fitted to these and the results used to determine the required shifts.

Summary of Parameters

IMAGE  (file) Name of image for input. This should be a file containing an arc spectrum.

ARC_OPTS  (char) Enter arc fit option:

  NEW (N)  set up a new wavelength calibration
  REPEAT (R)  Iterate on previous calibration
  CLONE (C)  CLone a previous calibration
OLD (O)  Correct using previous results

OUTPUT (file) Name of output file File to contain corrected data.

YStart (int) analysis lower limit The data between the limits ystart and yend is extracted and the resultant spectrum is used to locate the lines.

YEnd (int) analysis upper limit The data between the limits ystart and yend is extracted and the resultant spectrum is used to locate the lines.

YBlock (int) Enter analysis x-sect width Each window is of this width (except perhaps the final one).

Iteration (short) New value of iteration

ORDER (int) order for polynomial fitting This is for the continuity correction of the data. Ideally the arc should have been pre-processed with ARCSDI, so a low order e.g. 2 should be used.

MAXLines (int) Maximum number of lines to allow room for This must be greater than or equal to the number of lines fitted, so room should be allowed in case any more are to be added later.

CLfile (file) Name of image for cloning from This should be a file containing an arc spectrum.

TOIs (char) For use in batch only

KEEP_ITT (key) keep iteration files

PRFits (key) Print out details of fitting

PLOtcorr (key) Plot correction? This is used with the OLD option.

C.5  Display Programs

C.5.1  ISCAN—Plot Spectra Extracted From an Image

This produces plots of 1-d spectra which it extracts from a long-slit spectrum (i.e. it acts as if it were a wrap-up of the FIGARO functions EXTRACT and SPLOT). To use ISCAN just set the hard or softcopy device using HARD or SOFT, then type ISCAN, and answer the questions. To use in batch follow the instructions given by the program BATCH.

Summary of Parameters

IMAGE (file) Name of image for input

XStart (float) display lower limit

XEnd (float) display upper limit

YStart (int) display lower limit

YEnd (int) display upper limit
YBlock (int) Enter display x-sect width
SCan (key) Scan through data
HArd (key) use hard graphics device for display

C.5.2 HIMAGE—Greyscale Display

This uses GKS to plot a greyscale image of an image (in the FIGARO sense). It can plot on a GKS greyscale device such as a GWM or Postscript laser printer. The plot can include a key. Unfortunately this program takes a fair time to scale the image, so will "sit there" for a minute or so before anything appears on the screen (this does, of course, depend upon the image size). The parameters are as follows:

Image (file) Name of image to be displayed
YStart (int) First Y value to be displayed
YEnd (Int) Last Y value to be displayed
XStart (int) First X value to be displayed
XEnd (int) Last X value to be displayed
Low (int) Minimum count level for display
High (int) Maximum count level for display
PLOTdev The device to plot on (translated using GNS, so should be "xwindows" etc.).
ASPect This is the aspect ratio (X/Y) of the plot (excluding any key). The default is set to the value which gives square pixels.
SHRINK This gives a wide margin to the plot.
LOG (key) Display using logarithmic scaling
KEY This plots a key giving the data values corresponding to given greyscale values.

C.5.3 CSCAN—Plot Array of Line Profiles from Data “Cube”

This produces an array of line profiles from a 3-d data array, where the first dimension corresponds to wavelength. The data is scaled using the maximum and minimum intensities in the whole data cube, rather than of the individual profiles.

Summary of Parameters

CUBE (file) Name of CUBE for input
YStart (float) display lower limit
YEnd (float) display upper limit
TStart (int) display lower limit
\textbf{TEnd} (int) display upper limit

\textbf{HArd} (key) use hard graphics device for display

As regards the parameters, \textit{Y} is the first spatial dimension and \textit{T} the second.

\subsection*{C.6 Removing Continua}

\subsubsection*{C.6.1 \textbf{FITCONT—Fit Polynomial to Continuum}}

The optimisation in LONGSLIT allows for a flat base. Although this is generally satisfactory, for some data the base varies significantly. The user then has various options: the base can be subtracted from the using CSUB (which subtracts a polynomial fit to the base from the data); the data can be divided by the base; or the base can be left as it is, but a polynomial fit can be subtracted during the actual fitting. The latter option has the advantage that any plots of line profile etc. will still show the base as in the original data. This option is available by the use of the routine FITCONT, or totally within LONGSLIT. In general it is better to use the version in LONGSLIT, but this will not always cope with “difficult” data sets, so FITCONT may be required. The main difference here which allows FITCONT to deal with these cases is that it allows the weighting to be set using a cursor (in LONGSLIT it is done simply by excluding the areas within line boundaries, which is less versatile). This is similar to CSUB in fitting a Chebyshev polynomial to the base. However the data itself is not altered. When LONGSLIT fits a line it can subtract this base. Alternatively the base may be interpolated over the line using cubic splines (FITCONT is not needed for this).

\subsubsection*{Summary of Parameters}

\textbf{IMage} The file with the data in

\textbf{X Sect} The cross-section to use for the first fit

\subsubsection*{C.6.2 \textbf{CSUB—Subtract Continuum}}

This is similar in function to FITCONT, except that is performs the continuum subtraction itself.

\subsubsection*{C.6.3 \textbf{CADD—Add Back Continuum Previously Subtracted}}

This add back a continuum previously subtracted using CSUB, making use of the information CSUB stores in the data file.

\subsection*{C.7 Getting 3-D Data into the Correct Form for FIBDISP or LONGSLIT}

These programs were designed for use with a fibre array on the Manchester Echelle Spectrometer (Meaburn et al 1984). The fibres effectively make it a multi-slit spectrometer, with the fibres gathered into three (for example) rows at the input to the spectrometer. The first task is to extract the data from the individual fibres, the data is then wavelength calibrated and combined into a three-dimensional array, simulating the relative positions of the input ends to the fibres. FIBSEP extracts the data from the individual fibres into what looks like a long-slit spectrum for each row of fibres (at their output end). ARC2D can then be used for the wavelength calibration, and FIB2CUBE will re-arrange them into the format required for FIBDISP (FIB2CUBE also created the results structure).
C.7.1 FIBSEP—Extract Data from Individual Fibres

This separates out the individual fibres from a 2-d spectrum. It is assumed that the data had previously been corrected for S-distortion. This first produces a greyscale plot of the data (in a GWM window for example), and then the user is asked to mark two fibres and the limits (in X) that the program is to consider. If there is more than one “slit”, then each must be dealt with by separate runs of the program. The method is to find the centres of the fibres by fitting Gaussians, and to divide the data half-way between these points. Since the separation of the fibres may vary slightly, the user is recommended to use CSUB on the data first. This method is fairly crude and the user may well wish to improve upon it, but if all that is required is radial velocities this should be adequate.

Summary of Parameters

**Image** (file) Name of image to be displayed

**Output** (file) Name of image to create

**FILE** (char) Name of file for positions of fibres

**YStart** (int) First Y value to be displayed

**YEnd** (int) Last Y value to be displayed

**XStart** (int) First X value to be displayed

**XEnd** (int) Last X value to be displayed

**Low** (int) Minimum count level for display

**High** (int) Maximum count level for display

**DEVICE** (char) Plotting device (normally something like “xwindows”)

**NFIB** (int) Approximate number of fibres expected

**LOG** (key) Display using logarithmic scaling

**OLD** (key) Use previous results to extract data

C.7.2 FIB2CUBE—Convert Fibre Data to “Cube”

This converts calibrated long-slit spectra from a fibre array into a 3-d data cube for use by FIBDISP. Data from separate slits should be in different files, prompted for as IMAGE1, IMAGE2, etc. CUBE is the output file. FILE is a file listing the X, Y positions of each fibre in the output array (default type .DAT), probably produced by ENCODE. Note that, although CSCAN can produce plots from a file produced by FIB2CUBE for types “HEX” and “RECT”, these are only correct for type “RECT”. FIB2CUBE create files with all the structures required by FIBDISP. The emission line is delimited using the same code as used in LONGSLIT. The code only allows for one spectral line per file.

**IMAGE1** (file) Input image 1
IMAGE2 (file) Input image 2
IMAGE3 (file) Input image 3
IMAGE4 (file) Input image 4
IMAGE5 (file) Input image 5
CUBE (file) Output data cube

FILE (char) Name of file with coordinates for output cube. This file must contain the two spatial dimensions of the output cube followed by the number of input images in the first record. After that the X, Y coordinates of the input spectra (a cross-section at a time, starting at 1) in the output cube. After that the contents of the X displacement array (if any). Comments lines start with an exclamation mark.

MAXGAUSS (int) Maximum number of Gaussians that can be fitted to a profile.

C.8 Miscellaneous Programs

C.8.1 CUBE2LONG—Extract a Spectrum from a Spectral “Cube”

This extracts a long-slit-type spectrum from a 3-d data array, which has the dimension corresponding to wavelength last. The values are obtained using local cubic spline interpolation. The program requires a position (xpoint, ypoint) on the “slit”, and an angle. It will then produce a cut through the data, following this “slit” until it reaches the edge of the data in each direction from the given point. Variation of the parameter PIXLEN eases comparison of (for example) TAURUS data with long-slit data. Cubic spline interpolation is used to obtain the data values.

Summary of Parameters

CUBE (file) Name of cube for input.
OUTput (file) This is the name of the resulting image.
YPOINT (int) Y point on slit
XPOINT (int) X point on slit
ANGLE (float) pa of slit
PIXLEN (float) Pixel length

C.8.2 VIG—Correct For Vignetting

This fits a Chebyshev polynomials to templates taken from both directions across an image, and divides the image by the product of the polynomials, normalising so that the data in the centre in unchanged. A file is produced containing the coefficients of this polynomial so that other data files can have the correction applied.
C.9 Note on Menus

Many of the programs use menus to find out what the user requires next. When a menu-type prompt is given, the answer may be abbreviated, as long as a unique match is achieved (in many cases the most abbreviated form is given in brackets). Some menus also allow numbers or additional strings to follow (or even replace) the menu options themselves.

The menu parser can accept the command “%n” to alter the prompting style. At present n can be from 0 (no menu, just menu name) to 3 (full menu, abbreviations given in square brackets). This will then apply to future calls of the menu routine (note that the parameter menus (e.g. ARC_OPTS) are dealt with separately to most of the menus, and only support option 2. Options 0 and 1 are for experienced users, especially if they are working with slow terminal lines (or slow computers!). The formats are as below:

Option 0

```
Edit options [] -
```

Option 1

```
Look, Planes, Ascending, Descending, Exit
Edit options [] -
```

Option 2 (default)

```
=========< Edit options >=========

Look : Look at data values
Planes : Move planes of results around
Ascending : Sort components by wavelength
Descending : Sort components by wavelength
Exit : Return to main menu
Edit options [] -
```

Option 3

```
=========< Edit options >=========

LOOK [L] : Look at data values
PLANES [P] : Move planes of results around
ASCENDING [A] : Sort components by wavelength
DESCENDING [D] : Sort components by wavelength
EXIT [E] : Return to main menu
Edit options [] -
```

For options 1 and 2 the minimum abbreviation is given in upper case.
C.10 The Defaults File

This file (at present called twodspec_defaults.sdf) is used to allow the behaviour of programs to be varied. The elements have the following meanings:

**FIT_HANDLER** If this has the value 1 then a condition handler is used during the fitting (this is in addition to the FIGARO default handler). This condition handler is not available with the Sun version.

**OPT_STEP** This controls the size of jump between iterations in the fitting. The value multiplies the default.

**TOLS** These are the values copied to the TOLS array in the results structure when a new results structure is created. Where appropriate the values will be multiplied by the dispersion.

**USEPEAK** If this is 1 then the single component fitting will default to taking the peak as the centre, for guessing.

**VACUUM** This is set to 1 to indicate wavelengths are values in vacuum.

**AX1_LOG** If this is 1 it indicates that the first axis has a log scale (by default).

**AX1_UNITS** The default units for the first axis.

**RELATIVISTIC** Type integer, if absent, or of value 1, then a full relativistic formula is used for Doppler shifts, otherwise the non-relativistic one is used.

**DEC** FITS keywords to search to find declination of object (not yet used)

**COR_DEC** Multiplier for values from DEC to convert to degrees (not yet used)

**RA** FITS keywords to search to find right ascension of object (not yet used)

**COR_RA** Multiplier for values from RA to convert to hours (not yet used)

**EQUINOX** FITS keywords to search to find equinox

**UT** FITS keywords to search to find UT

**TELESCOP** FITS keywords to search to find telescope name (in form acceptable to SLALIB)

**LONG_OBS** FITS keywords to search to find longitude of observatory

**COR_LONG_OBS** Multiplier for values from LONG_OBS to convert to degrees

**LAT_OBS** FITS keywords to search to find latitude of observatory

**COR_LAT_OBS** Multiplier for values from LAT_OBS to convert to degrees

If the multipliers are absent they are taken as 1.0.

An example of a defaults file is given below (as shown by Figaro exam):
In this case COR_LONG_OBS is used to correct for the value of longitude in the file’s being positive east, whereas we want positive west. Note that you can have several values in each array, which would act as a search list (at present up to 5 values).
C.11 References

- Bailey, J., 1989, ICL — The New ADAM Command Language
- Lawden, M. D., 1989, SG/4 ADAM — The Starlink Software Environment
- Wilkins, T. N., 1988, Ph. D. Thesis, Univ. of Manchester

D Applications in detail

D.1 ABCONV-Convert spectrum from Janskys into AB magnitudes

Description: ABCONV generates a spectrum in AB magnitudes, given a spectrum in some other units. At present, only Janskys, milli-Janskys and micro-Janskys can be handled.

Parameters: SPECTRUM  The name of a spectrum whose units are (currently) either Janskys, milli-Janskys, or micro-Janskys.

OUTPUT  The resulting spectrum, whose units will be AB magnitudes. OUTPUT can be the same as SPECTRUM, in which case the conversion will be performed in situ.
Source comments:

CONV / FLCONV / IRCONV

Converts a spectrum into AB magnitudes (ABCONV) or f-lambda units (erg/s/cm**2/Angstrom) (FLCONV), or W/m**2/um (IRCONV). The original units of the data may be Jy (Janskys), mJy (milli-Janskys), or uJy (micro-Janskys). Other possibilities may be added later.

Command parameters -

SPECTRUM The name of the structure containing the spectrum. currently used for the spectrum. For FLCONV an x-axis data structure giving the wavelengths of the data elements is also required.

OUTPUT The name of the result of the operation. This can be the same as for SPECTRUM. If not, a new structure is created, with everything but the data a direct copy of the input.

Command keywords - None

User variables used - None

KS / CIT 18th May 1984

D.2 ABLINE-Interactive absorption line analysis

Description: ABLINE is an interactive Figaro program whose main purpose is to find wavelengths and equivalent widths of absorption lines. It includes the ability to fit a polynomial to selected regions of the continuum. As well as wavelengths and equivalent widths, the program also estimates width and asymmetry parameters for each line analysed. For a full description, see the TECHNIQUES ABLINE section of the Figaro documentation.

Parameters: SPECTRUM SPECTRUM is the name of the Figaro file containing the spectrum to be analysed. It should have a linear X array, since this is assumed in the calculations, and it is intended that the units be Angstroms, although other units will work (with some interpretation of the results being needed).

OLDCONT ABLINE allows you to fit local polynomials to parts of the spectrum which, together with its discrepant point rejection algorithm, usually allows a satisfactory continuum to be determined. Such a continuum may be written out by ABLINE and then re-read subsequently. Alternatively, the ability to read a pre-computed continuum may be used to read a continuum fitted in some different way, using other more appropriate techniques. If such a continuum is to be read, OLDCONT should be set.

CONTIN If OLDCONT was set, ABLINE will read in a continuum from the file specified by CONTIN. This should be a normal Figaro one-dimensional spectrum, with the same number of channels as the spectrum to be analysed. ABLINE assumes that the continuum channels map directly onto those of the spectrum.

SIG The continuum fitting in the specified regions is an iterative process. A curve is fitted and points that are more than SIG times the standard deviation will be rejected and ignored in the next iteration. 2.25 seems to work quite well.
ITN  ITN is the number of iterations used in determining the continuum. Usually, 4 seems to be adequate.

DEG  DEG is the degree of polynomial to be used when the continuum is fitted. It should be as low as possible while still able to follow the believable trend of the continuum in the area in question. Normal values would be in the range 1 to 3.

LIMIT  It is usual to operate with LIMIT set. In this case, when you delimit a line using the cursor, ABLINE will cutoff its integrations for the equivalent width and wavelength calculation at the channels indicated. If LIMIT is not set, ABLINE will calculate its own limits, taking the channels at which the spectrum first exceeds the continuum (having started from the middle of the indicated range). At present, the algorithm used only works for absorption lines, so LIMIT must be set if emission lines are being analysed.

WIDTH  WIDTH is used to control how much of the spectrum is displayed at one time by ABLINE. Sometimes it helps to use a large width to get a continuum fit and then a small width to home in on a line with the cursor. ABLINE has inbuilt limits to the number of channels it can display at once, and you may find that you exceed these if you make WIDTH too high (in which case you have to select a smaller WIDTH).

NEWCONT  If NEWCONT is set, the continuum produced by ABLINE will be written out as a spectrum. This spectrum can then be used for subsequent ABLINE runs, or for some other purpose altogether.

CONTOUT  If NEWCONT is set, ABLINE writes the continuum out to the file specified by CONTOUT. If this is the same as the input continuum file, the input continuum is over-written. If it is different, a new spectrum is generated with the same structure as the input spectrum and data from the fitted continuum.

CMD  If you enter a number, it will be taken as the central wavelength for a new region to be displayed. The other commands that are accepted are:

SIG x  Set ‘multiple of sigma’ continuum rejection parameter to x. DEG n  Set polynomial degree for continuum fit to n (1..7). ITN n  Set ‘number of iterations’ continuum rejection parameter to n. LIMIT Line will be delimited by the cursor positions indicated. NOLIM Line delimited by points where data becomes < continuum. WIDTH x  Set wavelength range displayed to x. CONT Determine continuum, starting with subsegment selection by cursor. RECONT Repeat continuum fit, using same segments as last time. FIT Analyse a line, delimiting it using the cursor. HELP (or ?) Output this information. QUIT Exit ABLINE.

Commands may be abbreviated. Omitted numeric parameters are prompted for.

Generally, the sequence is: 1) Select center wavelength, 2) CONTinuum, 3) FIT one or more lines. To change degree of continuum fit, use DEG followed by RECONT. Similarly for other continuum parameters.

More details should be available from the printed documentation.

LINENAME  The name of a line to be fitted.

HARDCOPY  Yes if hardcopy of soft plot to be made.

COMMENT  A comment for a hardcopy

Source comments:  A B L I N E

This routine does interactive analysis of absorption lines in spectra.
The user designates a segment of the input spectrum to be analysed in each pass. First a continuum is fitted to this region, using only wavelength subsegments selected graphically by the user (i.e. ignoring the absorption line in question and any other nearby lines or spikes). In addition to this selection, iterative rejection of discrepant points is performed. The functional form of the continuum is a polynomial of degree specified by the user (0 - 7). Alternatively, if a precomputed continuum spectrum is available, it can be used instead.

The user specifies the wavelength limits of the interval containing the line itself: the median wavelength and equivalent width of the absorption line are calculated.

The routine finishes up each segment with a hard copy plot showing the data, continuum and wavelength limits of the line, with a printout of results.

Command line parameters -
SPECTRUM Name of the file containing the spectrum with lines to be fitted CONTIN File containing precomputed continuum, if one is to be used. SIG Multiple of sigma for continuum point rejection ITN Number of iterations for continuum point rejection DEG Degree of polynomial for continuum fit WIDTH Wavelength range to display at one time CONTOUT Output continuum file name, if one is written. If CONTOUT is the same as CONTIN, the new continuum overwrites the old. Otherwise, a new file, the same as SPECTRUM except for the data, is created. CMD The command in the main menu. LINENAME The name of a line to be fitted. COMMENT A comment for a hardcopy

Command keywords -
OLDCONT Set if a precomputed continuum is to be used. LIMIT LIMIT is set if the limits of a line are to be taken as the limits indicated with the cursor. Otherwise, the program will look for the points within the indicated limits where the data drops below the continuum. NEWCONT Set if the continuum constructed during the run is to be written to a file. HARDCOPY Yes if hardcopy of soft plot to be made.

User variables - ("input, "output)
(>) SOFT (Character) Device/type for soft plots (>) HARD (Character) "hard" (<) TVXST (Numeric) ) (<) TVXEN (Numeric) ) Used to set the soft plot (<) TVHIGH (Numeric) ) parameters so that (<) TVLOW (Numeric) ) routines such as CCUR (<) XSTART (Numeric) ) know what the display (<) XEND (Numeric) ) limits for the currently (<) HIGH (Numeric) ) displayed plot have (<) LOW (Numeric) ) been set to. (<) TVFILE (Character)) (<) TVCOLOR (Numeric) ) JGR Jan 1985

D.3 ADJOIN-Append two spectra (strictly a merge by wavelength value)

Description: ADJOIN is a Figaro routine whose primary function is to append one spectrum to another. That is, given two spectra, it produces one output spectrum where the X-axis and data arrays are formed by appending the second spectrum data onto the end of the data from the first.

In detail, ADJOIN is a little more complex, since it produces a spectrum in which the X data increase. This may involve the sorting of the various arrays, so ADJOIN can be regarded as a program that merges two spectra into increasing X order.

If neither spectrum has any X information (i.e. no wavelength array), the sort order will be first and then second. If one or both have X data, the resulting spectrum will be in order of X value.
Parameters: **SPECTRUM** SPECTRUM to be appended to.

**SPECTRUM1** SPECTRUM1 specifies the second of the two spectra. Note that it is the structure of the first spectrum that is copied to the output spectrum. Only data arrays will be copied from the second.

**OUTPUT** OUTPUT is the name of the resulting spectrum. Note that ADJOIN always produces a new file, even if OUTPUT has the same name as one of the input spectra. OUTPUT will have essentially the same structure as the first spectrum, but any array found in the X-axis or data structure of both input spectra, with the same length as the main data array in both spectra, will appear in OUTPUT as an array whose length is the sum of the two arrays.

**Source comments:**

ADJOIN is a Figaro routine whose primary function is to append one spectrum to another. That is, given two spectra, it produces one output spectrum where the .X and .Z arrays are formed by appending the second spectrum data onto the end of the data from the first. In detail, ADJOIN is a little more complex, since it produces a spectrum in which the X data (the contents of the data object .X.DATA) increase. This may involve the sorting of the various arrays, so ADJOIN can be regarded as a program that merges two spectra into increasing X order. The resulting spectrum makes perfect sense if the data represent flux density measurements, but may be misleading if the data represent total flux measured within wavelength bins. The X array may well not represent even a smooth wavelength vs channel relationship, let alone scrunched data. Care should be taken in the use of this routine.

**Command parameters -**

- **SPECTRUM** (Character) The first of the two spectra.
- **SPECTRUM1** (Character) The second of the two spectra.
- **OUTPUT** (Character) The resulting output spectrum.

**Command keywords - None**

D.4 **ALASIN-Read a spectrum in ALAS (Abs. Line Analysis System) format**

**Description:** ALASIN reads a ‘Computed Profile File’ as generated by ALAS (Absorption Line Analysis Software) and writes it to a FIGARO spectrum.

**Parameters:** **ALASFILE** The name of the input ALAS format file. (This is an ASCII file, with one value of radial velocity (km/s) and the corresponding value of residual intensity on each line.) The file type should be given (default is .DAT)

**SPECTRUM** SPECTRUM is the name of the output file. It will have a number of channels equal to the number in the input ALASFILE. The X-axis values are velocities in km/s, as output by ALAS. The data values are the residual intensities as computed by ALAS, i.e. values of intensity normalised by the continuum.
ALASIN reads a 'Computed Profile File' generated by ALAS (Absorption Line Analysis Software) and creates an equivalent FIGARO spectrum.

The ALAS 'Computed Profile' is an ASCII file with each line containing a radial velocity and the corresponding residual intensity value (i.e. normalised by the continuum). This is converted to a standard Figaro spectrum, except that the X values are velocities rather than wavelengths.

**Command Parameters**

- **ALASFILE** Name (including type) of the input ALAS format file
- **SPECTRUM** The output FIGARO format spectrum

---

**D.5 ALASOUT-Output a spectrum in ALAS (Abs. Line Analysis System) format**

**Description:** ALASOUT writes a section of a spectrum to an output file suitable for use as an 'Observed Profile' by ALAS (Absorption Line Analysis Software).

**Parameters:**

- **SPECTRUM** The input spectrum. A section of this spectrum, as specified by XSTART and XEND, will be written to the output file (ALASFILE). Note that ALAS expects the spectrum to have been normalised by the continuum.
- **XSTART** The X value of the first channel which will be written to the output (ALASFILE).
- **XEND** The X value of the last channel which will be written to the output (ALASFILE). The total number of channels converted (XEND - XSTART + 1) may not exceed 1500; ALAS (version as per Starlink LUN/41.1) accepts a maximum of 300 channels in one file.
- **ALASFILE** The name of the output ALAS format file. The file type should not be given: the file type is set internally and is always .ALS. If the type is given it will be ignored.

**Source comments:** ALASOUT takes data from a FIGARO spectrum and writes it to a file in ASCII format, suitable for input to ALAS (Absorption Line Analysis Software) as an 'Observed Profile File'. The user can select the X range to be transferred, i.e. to cover the desired line without too much extra spectrum. This is important since ALAS has a limit of 300 channels for these input files (at least in version as per Starlink LUN/41.1). For each selected channel an output line is written to the file, giving the X value (F9.3) and the data value (F9.5). Note that ALAS expects these data values to have been normalised by the continuum. The file created will have file type .ALS.

**Command parameters**
D.6 APERTURE-Do simple minded aperture photometry on a series of frames

Description: The image name is read and that image is plotted on the current plot device. The user is presented with a menu which allows him/her/it to specify object and sky regions, change the colour levels, change the radius of the aperture, show cuts or quit. Integrations are a simple sum of the values of the pixels within the aperture radius.

Parameters:  
OUTPUT The name of the file where the answers will be written.

IMAGE The name of the file with the frame to be analysed.

OPT The option parameter. Type H for help.

LOW The low data value for 2-D plots.

HIGH The high data value for 2-D plots.

RADIUS The radius of the circle to be used for the integration.

OK Say "yes" when you are happy with the aperture.

Source comments:

Description  
-----------  
Do simple minded aperture photometry on a series of frames  

Scope of program  
-----------------  
- Handles 2-D images only.
- Data array converted to FLOAT.
- Magic values supported.
- Error arrays not supported
- Quality arrays not supported.
- Batch execution not supported.

Parameters  
----------  
OUTPUT Name of output ASCII file for results. The output info goes in the following order: (1) Either 'O' or 'S' for "object" or "sky"; (2) a sequence number; (3) the x,y coords of the cursor; (4) the radius of the aperture; (5) the total flux in pixels which are with a radial distance less than or equal to the aperture radius. No correction is made for partial pixels; (6) the total number of pixels included. (character)(prompted for)

IMAGE Name of individual images in case you’re not using a
list file. (character)(prompted for)

RADIUS  Radius for integration. (real)(prompted for)

LOW     Data value for lowest level in 2-d plot (real)(prompted for)

HIGH    Data value for highest level in 2-d plot (real)(prompted for)

Keywords
--------
OK

Method
------
- The image name is read and that image is plotted on the current plot device.
- The user is presented with a menu which allows him/her/it to specify object and sky regions, change the colour levels, change the radius of the aperture, show cuts or quit.
- Integrations are a simple sum of the values of the pixels within the aperture radius.

Author/s
--------
Jim Lewis RGO (jrl@ast.cam.ac.uk)

D.7 ARC-Interactive manual arc line identification

Description: ARC is a Figaro program that can be used for interactive identification of arc lines.

Each invocation of ARC produces a file arlines.lis in the working directory. This file must be renamed or deleted before re-invoking ARC.

Parameters: SPECTRUM  The arc data. If there is an X-axis data component the information it contains will be used during the program. At the end of the program the X-axis data component can be set to contain the wavelengths determined by the fit.

ARCTYPE  The type of arc that was used - e.g. HELIUM, NEON, etc. ARC will look for a file called ARCTYPE.ARC which should hold the line list for the arc. Can be up to three types, separated by commas.

SIGMA  Arc line half width in pixels.

ORDER  Polynomial order for 1st fit.

PREVIOUS  If set, ARC will read in the line list from the previous fit as a starting point.

ARFILE  The name of the list file from which the previous fit is to be read. Only used if PREVIOUS is set. Note that the output is always written to ARLINES.LIS. Default extension is .LIS

XCORR  If the previous arc fit was to a different arc, then there is the possibility that the current arc is similar to the previous one but has been shifted. If this is the case, ARC can attempt to determine the shift by cross-correlation of the current arc and the previous one, and can then redetermine the arc line centers by looking for the listed lines at their shifted positions.
OUTPUT  If the final fit obtained is to be used, it is used to reset the x-axis structure in the arc spectrum, giving a new output file. OUTPUT is the name of output file, which can be the same as SPECTRUM, in which case the x-axis structure of SPECTRUM is replaced.

DISNCHAN  Length of displayed sections.

MOVETOX  New plot centre x value.

CMD  At this stage in ARC you have the following options available -

- Fit - Repeat the fit.
- Disp - Display the deviation of the fit from a linear fit. This shows the trend of the fit, and the errors in each line.
- Order - Change the order of the fit.
- Edit - Delete or change the wavelength of one or more of the selected lines, without returning to the cursor selection.
- Reselect - Return to selection using the cursor.
- Print - Prints a copy of the fit (what ARLINES.LIS would look like if you were to exit now).
- Auto - Starting from your current fit and arc line list, ARC looks for additional line in the arc at wavelengths given in the line list and adds any it finds to the identified line tables.
- Xauto - Deletes all the lines found by ‘Auto’.
- Modify - Allows you some control over the Autofit parameters.
- Quit - Start to exit ARC.
- Help - (or ?) Display this information.

The first letter of each command is sufficient.

LINENO  Number of line to be edited.

WAVELEN  Wavelength specification.

CHFACT  The autofit algorithm is parameterised as follows.

It takes each pixel in turn. If that pixel is more than CHFACT times the current sigma value from any line already found, it uses that pixel as the starting point for a line search. If anything resembling a line can be found, it calculates its wavelength and looks in the line tables for a line close to that wavelength.

A line is accepted if the discrepancy between calculated and tabulated wavelength is less than SIGFACT times the current RMS value. This means that the criterion for accepting new lines is based on how their wavelength discrepancies compare with those for the lines that have already been accepted.

SIGFACT is the more important parameter.

SIGFACT  The autofit algorithm is parameterised as follows.

It takes each pixel in turn. If that pixel is more than CHFACT times the current sigma value from any line already found, it uses that pixel as the starting point for a line search. If anything resembling a line can be found, it calculates its wavelength and looks in the line tables for a line close to that wavelength.

A line is accepted if the discrepancy between calculated and tabulated wavelength is less than SIGFACT times the current RMS value. This means that the criterion for accepting new lines is based on how their wavelength discrepancies compare with those for the lines that have already been accepted.

SIGFACT is the more important parameter.

WRITEARC  If set, an output spectrum using the arc fit is written.

HARDCRC  If set, the output spectrum is plotted in a hard copy.

HARDISP  If set, the dispersion curve is plotted in a hard copy.

QUITSEL  Used to confirm quitting line selection.
LINEOK Used to confirm a choice of line for deletion, editing etc.
RESOLVE Used to decide what to do if a line is used twice.

Source comments:
Interactively associates lines in an arc spectrum with their wavelengths and performs a fit to these values.

Command parameters -

SPECTRUM The arc data. If there is an x-axis data component the information it contains will be used during the program. At the end of the program the x-axis data component can be set to contain the wavelengths determined by the fit.

ARCTYPE The type of arc that was used - e.g. HELIUM, NEON, etc. ARC will look for a file called ARCTYPE.ARC which should hold the line list for the arc. Can be up to three types, separated by commas.

ORDER The initial order for the polynomial fit.

SIGMA The initial value for the line width.

ARFILE The name of the list file from which the previous fit is to be read. Only used if PREVIOUS is set. Note that the output is always written to ARLINES.LIS. Default extension is .LIS

OUTPUT If the final fit obtained is to be used, it is used to reset the x-axis structure in the arc spectrum, giving a new output file. OUTPUT is the name of output file, which can be the same as SPECTRUM, in which case the x-axis structure of SPECTRUM is replaced.

Command keywords -

PREVIOUS If set, ARC will read in the line list from the previous fit as a starting point.

XCORR If set, and arc is not the same as the arc used to generate the previous line list, a shift between the two will be determined and the line centers reanalyysed.

User variables -

(>) SOFT (Char) The device/type to be used for graphics soft plots. See the SOFT command for details. The device must support a cursor.

(>) HARD (Char) The device/type for graphics hard plots.

Input -

As named May use the lines from a previous run. If so by ARFILE these are read from the previous run’s output file. See below.

Output -
ARLINES.LIS File containing the lines used in the final fit.
Format is as follows -
Number of lines used in fit and file name (I5,23X,A)
1 blank record, then one header record.
Then one record for each line, giving channel number,
wavelength, calculated wavelength and wavelength
discrepancy line number and auto flag (4F13.4,I7,A4)
The auto flag is either " (A)" or is a blank string.
Then one blank record, then a record giving the RMS
error and the value of SIGMA used (12X,F10.2,19X,F5.2)
Then one blank record, then one record giving the
order of fit (i.e. 1 less than number of coefficients)
(15X,I3), then one blank record, then one or more
records giving the coefficients (3D23.16)

KS / CIT 13th June 1984

D.8 ARC2D-Calibrates distortions in 2D arc line data.

Description: This program controls both 1D and 2D wavelength calibration and can operate
either in BATCH or INTERACTIVE modes. The philosophy behind it is somewhat different
to those presented in the existing SPICA/SDRSYS and FIGARO software in many respects.
In particular its exclusive use of gaussian fitting of arclines, its demand for "intellegent"
users, who can decide which lines they want to use initially and then allow them to make
objective assesments of which,if any are erroneous. Typical diagnostic information given
are plots of residuals from the fit versus line width,flux and position. This is all made
possible by the use of the Gaussian fitting. The least squares polynomial fitting allows
weights to be included for each line(again derived from the formal Gaussian fits).Thus
it is possible to constrain the polynomial in difficult regions eg "the 5100 gap" without
distorting the global fit.

Parameters: IMAGE
IMAGE = FILE (Read) Name of image for input This should be a file
containing an arc spectrum.
ARC_OPTS
ARC_OPTS = CHARACTER (Read) Enter arc fit option NEW : set up a new
wavelength calibration REPEAT : Iterate on previous calibration CLONE : CLone a
previous calibration
YSTART
YSTART = INTEGER (Read) analysis lower limit The data between the limits
ystart and yend is extracted and the resultant spectrum is used to locate the lines.
YEND
YEND = INTEGER (Read) analysis upper limit The data between the limits ystart
and yend is extracted and the resultant spectrum is used to locate the lines.
YBLOCK
YBLOCK = INTEGER (Read) Enter analysis x-sect width Each window is of
this width (except perhaps the final one).
ITERATION
ITERATION = INTEGER*2 (Read) New value of iteration
ORDER
ORDER = INTEGER (Read) order for polynomial fitting This is for the continuity
correction of the data. Idealy the arc should have been pre-processed with ARCSDI,
so a low order e.g. 2 should be used.
MAXLINES  MAXLINES = INTEGER (Read) Maximum number of lines to allow room
for This must be greater than or equal to the number of lines fitted, so room should
be allowed in case any more are to be added later.

CLFILE  CLFILE = FILE (Read) Name of image for cloning from This should be a file
containing an arc spectrum.

TOLS  TOLS = CHARACTER (Read) For use in batch only

KEEP_ITT  KEEP_ITT = LOGICAL (Read) keep itteration files’

PRFITS  PRFITS = LOGICAL (Read) Print out details of fitting

Source comments: available

D.9  ARCDISP-Fit polynomial dispersion curve.

Usage:  arcdisp in order

Description:  This routine fits a polynomial dispersion curve to a list of identified arc features
and transforms the NDF pixel coordinates to spectroscopic values. Optionally you can use
a graphical dialogue to improve on the previous feature identification, until you like the
appearance of the dispersion curve.

Parameters:  DIALOG  DIALOG = _CHAR (Read) If this is ‘Y’, ‘T’ or ‘G’, then the graphical di-
alogue is entered before the polynomial dispersion curve for any row is accepted and
applied. If this is ‘N’ or ‘T’ then the dialogue is not entered and separate dispersion
curves are applied to all rows. The string is case-insensitive. ['G']

IN  IN = NDF (Read) The spectrum or set of spectra in which emission features are to be
located. This must be a base NDF, the spectroscopic axis must be the first axis. No
spectroscopic values or widths must exist in the Specdre Extension. The pixel centres
along the first axis must be NDF pixel coordinates. Update access is necessary, the
results structure in the Specdre Extension may be modified, an array of spectroscopic
values will be created in the Specdre Extensions.

ORDER  ORDER = _INTEGER (Read) The polynomial order of dispersion curves. This
cannot be changed during the graphical dialogue. Neither can it differ between rows. [2]

FDB  FDB = NDF (Read) The feature data base. Only the simple list of values FTR_WAVE
is used and only in graphics dialogue. It serves to find the identification for an as yet
unidentified - but located feature.

DEVICE  DEVICE = GRAPHICS (Read) The graphics device to be used. This is unused if
DIALOG is false.

WRANGE  WRANGE( 2 ) = _REAL (Read) In graphical dialogue this parameter is used
repeatedly to get a range of laboratory values. This is used for plotting as well as for
finding identifications in the feature data base.

Source comments: ARCDISP

The input data must be a base NDF. They can be a single spectrum
or a set of spectra. Examples for the latter are a long slit
spectrum, a set of extracted fibre spectra, or a collapsed echellogram (a set of extracted orders from an echelle spectrograph). It is necessary that the spectroscopic axis be the first axis in the data set. It does not matter how many further axes there are, the data will be treated as a linear set of rows with each row a spectrum.

The actual input is the results structure in the Specdre Extension. This must be a set of components of type 'arc feature'. Each must have two parameters 'centre' and 'laboratory value'. These must be corresponding locations one expressed in NDF pixel coordinates, the other in spectroscopic values (wavelength, frequency etc.). The centres must be strictly monotonically increasing, their variances must be available. Laboratory values may be bad values to signify unidentified features.

In the graphical dialogue the results structure may be updated. The locations remain unchanged; all located features form a fixed list of potentially identified features. Identifications may be added, deleted or modified. The user has to work on each row in turn (unless Quit is chosen). When the user switches from one row to the next, the dispersion curve for the finished row is applied and its spectroscopic values in the Specdre Extension are set. When the last row is finished, the application exits; the output of this routine is (i) an updated list of identifications in the results structure of the Specdre Extension and (ii) an array of spectroscopic values according to the dispersion curves for each row, also in the Specdre Extension. At any point the user can quit. In this case the array of spectroscopic values is discarded, but the updated identifications are retained. If run without dialogue, this routine simply performs the polynomial fit of the dispersion curve for each row in turn and works out the array of spectroscopic values. The list of identifications is input only and remains unchanged. If for any row the fit cannot be performed, then the spectroscopic values calculated so far are discarded and the routine aborts.

There must not yet be any spectroscopic value information: There must be no array of spectroscopic values or widths in the Specdre Extension. The pixel centre array for the spectroscopic axis (i.e. the first axis) must be NDF pixel coordinates (usually 0.5, 1.5, ...).

This routine works on each row (spectrum) in turn. It fits a polynomial to the existing identifications. In the optional graphical dialogue two plots are displayed and updated as necessary. The lower panel is a plot of laboratory values (wavelength, frequency etc.) versus pixel coordinate shows

- all possible identifications from the feature data base as horizontal lines,
- all unidentified located features as vertical lines,
- all identified located features as diagonal crosses,
- the dispersion curve.

In the upper panel, a linear function is subtracted so that it displays the higher-order components of the dispersion curve. Crosses indicate the identified located features. Since the scale of this upper panel is bigger, it can be used to spot outlying feature identifications. In the dialogue you can

- R - Switch to next row, accepting the current fit for this row
- X - X-zoom 2x on cursor
- Y - Y-zoom 2x on cursor
- W - Unzoom to show whole row
- N - Pan by 75% of current plot range
- A - Add ID for location nearest to cursor (from FDB)
- S - Set ID for location nearest to cursor (from cursor y pos.)
- D - Delete ID for feature nearest to cursor
- Q - Quit (preserves updated IDs, discards applied fits for all rows)
- ? - Help

Whenever the list of identifications is changed, the dispersion curve is fitted again and re-displayed. If there are too few identifications for the order chosen, then the dialogue will display the maximum order possible. But such an under-order fit cannot be accepted, the R option will result in an error.

The Q option will always result in an error report, formally the routine aborts. After all, it does not achieve the main goal of applying individual dispersion curves to all rows.

On one hand the output of this routine may be an updated list of identifications, which could in principle be used in a future run of this routine. On the other hand this routine will always result in an array of spectroscopic values. The existence of these spectroscopic values prevents using this routine again. Before using this routine again on the same input NDF you have to delete the SPECVALS component in the Specdre Extension.

In order to facilitate repeated use of this routine on the same data, it always uses the Specdre Extension to store spectroscopic values, even if the data are one-dimensional and the first axis centre array would suffice to hold that information. This leaves the first axis centre array at NDF pixel coordinates, as necessary for re-use of this routine.

Notes: This routine recognises the Specdre Extension v. 0.7. This routine works in situ and modifies the input file.

D.10 ARCGENDB-Convert list of laboratory values to feature data base.

Usage: arcgendb in fdb
Description: This routine converts an arc line list - i.e. an ASCII list of laboratory wavelengths or frequencies of known features in an arc spectrum - into a feature data base. That can be used for automatic identification of features in an observed arc spectrum.

Parameters: INFO
INFO = _LOGICAL (Read) If true, informational messages will be issued.

IN
IN = FILENAME (Read) The name of the input ASCII list of wavelengths or frequencies. The list must be strictly monotonically increasing.

FDB
FDB = NDF (Read) The name of the output file to hold the feature data base. This is formally an NDF.

Examples: arcgendb $FIGARO_PROG_S/thar.arc thar_arc
This will convert the Th-Ar list from the Figaro release into a "feature data base" by the name of "thar_arc.sdf".


Source comments: Since generating the feature data base may take some time, you may want to do it once for any line lists you often use, and keep the feature data bases. On the other hand, the feature data bases may be rather big.

This routine reads a list of laboratory values (wavelengths or frequencies). The list must be an unformatted ASCII file. From the beginning of each line one value is read. If this fails, the line is ignored. Comment lines can be inserted by prefixing them with "+", "!" or "#". The value can be followed by any comment, but can be preceded only by blanks. The list must be strictly monotonically increasing.

The list should to some degree match an expected observation. Its spectral extent should be wider than that of an expected observation. But it should not contain a significant number of features that are usually not detected. This is because the automatic identification algorithm uses relative distances between neighbouring features. If most neighbours in the list of laboratory values are not detected in the actual arc observation, then the algorithm may fail to find a solution or may return the wrong solution.

The given list is converted to a feature data base according to Mills (1992). The data base contains information about the distances between neighbours of features. The scope of the feature data base is the number of neighbours about which information is stored. The feature data base is stored in an extension to a dummy NDF. The NDF itself has only the obligatory data array. The data array is one-dimensional with 1 pixel. All the actual information is in an extension with the name "ECHELLE" and of type "ECH_FTRDB". Its HDS components are:
NLINES is the number of features listed in the input file. The scope (=10) controls about how many neighbours information is stored in the data base. The index size is fixed to 5000, which seems sufficient for NLINES = 3500. The size of the FDB is

\[(804 \times \text{NLINES} + 40000) \text{ bytes}\]

plus a small overhead for the HDS structure and the nominal NDF. So it is 10 to 100 times bigger than the original ASCII list. The point about the FDB is the reduced computing time when auto-identifying features in an observed arc spectrum.

**D.11 ARCIDENT-Auto-identify located features.**

**Usage:** arcident in out fdb wrange=?

**Description:** This routine identifies located features in a set of spectra. Auto-identification is done from scratch (without prior identification of any features) with the algorithm by Mills (1992).

**Parameters:**
- **INFO** INFO = _LOGICAL (Read) If false, the routine will issue only error messages and no informational messages. [YES]
- **ECHELLE** ECHELLE = _LOGICAL (Read) If false, the given WRANGE is used for each row, assuming the rows are similar spectra (long slit or fibre). If true, a collapsed echellogram is assumed. In that case each row is an extracted order with different wavelength/frequency range. This routine will divide the given WRANGE into as many sub-ranges as there are rows (orders) in the given input. [NO]
- **IN** IN = NDF (Read) The spectrum or set of spectra in which located features are to be identified. This must be a base NDF, the spectroscopic axis must be the first axis. No spectroscopic values or widths must exist in the Specdre Extension. The pixel centres along the first axis must be NDF pixel coordinates. The input NDF must have a results structure in its Specdre Extension, and the results must contain a number of line components with strictly monotonically increasing position (centre).
- **OUT** OUT = NDF (Read) The output NDF is a copy of the input, except that the results structure holds feature identifications rather than locations (‘peak’ parameters will have been replaced with ‘laboratory value’ parameters).
- **FDB** FDB = NDF (Read) The feature data base. The actual data base is a set of primitive arrays in an extension to this NDF called ECHELLE. A feature data base can be generated from a list of wavelengths or frequencies with ARCGENDB.
WRANGE \ WRANGE(2) = _REAL (Read) The approximate range of wavelengths or frequencies. The narrower this range the faster is the identification algorithm. But if in doubt give a wider range.

DRANGE \ DRANGE(2) = _REAL (Read) The range into which the dispersion in pixels per wavelength or per frequency falls. The narrower this range the faster is the identification algorithm. But if in doubt give a wider range.

STRENGTH \ STRENGTH = _REAL (Read) This specifies the maximum ratio between the strength of features that are to be used initially for identification. If the strongest feature has peak 1000, then the weakest feature used initially has peak greater than 1000/STRENGTH. [50.0]

THRESH \ THRESH = _REAL (Read) This specifies the maximum difference between the ratios of neighbour distances as observed and as found in the feature data base. The difference is evaluated as ABS(1 - ABS(obs/ref)) < THRESH. Values much larger than 0.1 are likely to generate a lot of coincidence matches; values less than 0.01 may well miss ‘good’ matches in less-than-ideal data. You may need to relax this parameter if your arc spectra are very distorted (non-linear dispersion). [0.03]

MAXLOC \ MAXLOC = _INTEGER (Read) This specifies the maximum number of features to be used when generating ratios for initial identification. In general, a good solution can be found using only the strongest 8 to 16 features. The program slowly increases the number of features it uses until an adequate solution if found. However, there may be a large numbers of weak features present which are not in the reference database. This parameter allows the setting of an absolute maximum on the number of features (per row) which are to be considered. If less than MAXLOC features are located in a given row, then the number of identified features is used instead for that row. [30]

MINIDS \ MINIDS = _INTEGER (Read) The minimum number of features that must be identified for the algorithm to be successful. If fewer than MINIDS features are located in a given row, then a smaller number is used instead for that row. [9]

NEIGHB \ NEIGHB(2) = _INTEGER (Read) NEIGHB(1) specifies the starting number of neighbouring features (on each side) to examine when generating ratios for matching. (These are neighbours in the observed spectra, not in the feature data base.) Increasing this will lead to exponential increases in CPU time, so it should be used with caution when all else fails. The default value is 3. Higher values are tried automatically by the program if no solution can be found. The number of neighbours considered is increased until it reaches the maximum of NEIGHB(2), when the program gives up. [3,6]

Source comments:}

The input data must be a base NDF. They can be a single spectrum or a set of spectra. Examples for the latter are a long slit spectrum, a set of extracted fibre spectra, or a collapsed echellogram (a set of extracted orders from an echelle spectrograph). It is necessary that the spectroscopic axis be the first axis in the data set. It does not matter how many further axes there are, the data will be treated as a linear set of rows with each row a spectrum.
The features for which an identification should be attempted must have been located. That is, they must be components of type 'Gauss', 'triangle', 'Gauss feature' or 'triangle feature' in the results structure of the Specdre Extension. Each of these components must have at least a 'centre' and 'peak' parameter. The centres (feature locations) must be a strictly monotonically increasing list. Their variances must be available. The locations (centre parameters) must be in terms of NDF pixel coordinates. The peaks must be positive. They are used as a measure of the importance of a feature.

The coverage in spectroscopic values of all spectra (rows) should either be similar (long slit or fibres) or roughly adjacent (echellogram). There must not yet be any spectroscopic value information: There must be no array of spectroscopic values or widths in the Specdre Extension. The pixel centre array for the spectroscopic axis (i.e. the first axis) must be NDF pixel coordinates (usually 0.5, 1.5, ...). The data must be arranged such that spectroscopic values increase left to right. In the case of rows with adjacent coverage spectroscopic values must also increase with row number. In a collapsed echellogram this usually means that for wavelength calibration the order number must decrease with increasing row number. If this is not the case then it is still possible to work on a collapsed echellogram: You can set ECHELLE false and thus use the full WRANGE for each row, but you must adjust DRANGE to be a more reasonable guess of the dispersion.

Identification is done by comparison with a feature data base according to Mills (1992). The feature data base should to some degree match the observation. Its spectral extent should be wider than that of the observation. But it should not contain a significant number of features that are not located. This is because the automatic identification algorithm uses relative distances between neighbouring features. If most neighbours in the list of laboratory values are not detected in the actual arc observation, then the algorithm may fail to find a solution or may return the wrong solution.

This routine works on each row (spectrum) in turn. It establishes information about relative distances between neighbouring located features and compares this with a feature data base. This serves to identify at least a specified number of features. An auto-identification should always be checked in the process of fitting a polynomial dispersion curve. All located features are retained by this routine, so that further identifications can be added or some identifications can be cancelled.

The result of this routine is a list of feature identifications. All located features are retained, though some will have not been identified. The locations and identifications (pixel coordinates and laboratory values) are stored in the results structure of the Specdre Extension of the input data. This replaces the pre-existing results extension. The locations are strictly
monotonically increasing, as are in all probability the identifications.

The new results structure provides for as many component as the old one had components of any recognised type. Each component has on output the type 'arc feature'. It has two parameters 'centre' and 'laboratory value'. Located but unidentified features will have bad values as laboratory values. The variances of laboratory values are set to zero.

Mills' (1992) algorithm performs only an initial line identification. It is important to verify the returned values by fitting a wavelength or frequency scale (e.g. polynomial or spline fit), and to reject any out-liers. The algorithm should be given the positions of all conceivable features in the spectra. It does not use the fainter ones unless it is unable to identify using only the brightest, but you will get more robust behaviour if you always provide all possible candidate lines for potential identification. The algorithm should not be fed severely blended line positions as the chance of incorrect identifications will be significantly higher (this is the exception to the rule above).

The speed of the algorithm varies approximately linearly with wavelength/frequency range and also with dispersion range so the better constraints you provide the faster it will run. The algorithm takes your constraints as hard limits and it is usually more robust to accept a slightly longer runtime by relaxing the ranges a little.

If the algorithm runs and keeps looping increasing its set of neighbours, then the most likely causes are as follows:

- wavelength/frequency scale does not increase with increasing x (set the CHKRVS parameter true and try again).
- WRANGE or DRANGE are too small (increase them both by a factor of 2 and try again).

Notes: This routine recognises the Specdre Extension v. 0.7.


D.12 ARCLOCAT-Locate line features in a set of spectra.

Usage: arclocat in fwhm thresh

Description: This routine locates narrow features in a set of spectra. Features can be located from scratch automatically. In a different mode, feature locations can be added or deleted in a graphical dialogue. The feature location and peak are determined by a Gauss or triangle line fit.

Parameters: INFO INFO = _LOGICAL (Read) If true, messages about the progress of auto-locating features are issued. [YES]
DIALOG  DIALOG = _CHAR (Read) If this is 'Y', 'T' or 'G', then no auto-locating takes place and the graphics dialogue is entered. If this is 'N' or 'F' then the dialogue is not entered and auto-locating is done instead. The string is case-insensitive. ['G']

MODE  MODE = _CHAR (Read) This can be 'Gauss' or 'triangle' and chooses the line profile to be fitted. This string is case-insensitive and can be abbreviated to one character. ['Gauss']

IN  IN = NDF (Read) The spectrum or set of spectra in which emission features are to be located. This must be a base NDF, the spectroscopic axis must be the first axis. No spectroscopic values or widths must exist in the Specdre Extension. The pixel centres along the first axis must be NDF pixel coordinates. Update access is necessary, the results structure in the Specdre Extension will be modified, possibly re-created.

FWHM  FWHM = _REAL (Read) The guessed full width at half maximum of the features to be located. This is used to estimate the maximum number of features that might be located, to locate baseline ranges next to suspected features, and as a guess for the line fit.

THRESH  THRESH = _REAL (Read) The threshold. While scanning a pixel must exceed this threshold to initiate a line fit. The fitted peak also must exceed the threshold in order that the feature location be accepted. This parameter is significant only for automatic location of features.

DEVICE  DEVICE = GRAPHICS (Read) The graphics device to be used. This is unused if DIALOG is false.

ROWNUM  ROWNUM = _INTEGER (Read) In graphics dialogue this parameter is used to switch to a different row (spectrum).

Source comments: 

The input data must be a base NDF. They can be a single spectrum or a set of spectra. Examples for the latter are a long slit spectrum, a set of extracted fibre spectra, or a collapsed echellogram (a set of extracted orders from an echelle spectrograph). It is necessary that the spectroscopic axis be the first axis in the data set. It does not matter how many further axes there are, the data will be treated as a set of rows with each row a spectrum.

The coverage in spectroscopic values of all spectra (rows) should either be similar (long slit or fibres) or roughly adjacent (echellogram). There must not yet be any spectroscopic value information: There must be no array of spectroscopic values or widths in the Specdre Extension. The pixel centre array for the spectroscopic axis (i.e. the first axis) must be NDF pixel coordinates (usually 0.5, 1.5, ...). The data must be arranged such that spectroscopic values increase left to right. In the case of rows with adjacent coverage spectroscopic values must also increase with row number. In a collapsed echellogram this usually means that for wavelength calibration the order number must decrease with increasing row number.

In automatic mode this routine works on each row (spectrum) in turn. It scans through the spectrum and looks for pixels that
exceed the local background level by at least the given threshold value. When such a pixel is spotted, a single-component line fit is tried no the local profile. The local profile is centred on the pixel suspected to be part of an emission feature. It includes 1.5 times the guessed FWHM on either side and a further 5 baseline pixels on either side. A local linear baseline is subtracted prior to the line fit. In order for the feature to be entered into the list of located features, the fit must succeed, the fitted peak must exceed the threshold, and the fitted peak must exceed the absolute difference of background levels between the left and right.

When run with graphics dialogue this routine works on any choice of rows. It uses a pre-existing list of located features to which can be added or from which features can be deleted. Graphics dialogue can also be used to just check the locations. The graph displays the spectrum currently worked on in bin-style. The current list of located features is indicated by dashed vertical lines. The options in the graphical dialogue are:
- R - Choose different row to work on
- X - X-zoom 2x on cursor
- Y - Y-zoom 2x on cursor
- W - Unzoom to show whole row
- N - Pan left/right by 75% of current x range
- A - Add the feature under cursor to list (subject to line fit)
- S - Add the cursor position as feature to list
- D - Delete the feature nearest cursor from list
- Q - Quit, preserving the updated list of located features
- ? - Help

The difference between the A and S options is that A tries a line fit to the local profile around the cursor, while S accepts the cursor x position as exact centre and the cursor y position as exact peak of a new feature; (the variance of the centre is set to 0.25, the variance of the peak to the bad value).

The result of this routine is a list of Gauss or triangle features. Their locations in NDF pixel coordinates and their peak values are stored in the results structure of the Specdre Extension of the input data. If run in automatic mode, this routine will replace any previously existing results structure. If run with graphics dialogue, this routine will try to work with a pre-existing list of located features. But if the pre-existing results structure does not conform to the required format, then a new results structure is created.

The list of located features (for each row) is always sorted such that the locations are strictly monotonically increasing.

The results structure provides for a certain number of components. These have component type 'Gauss feature' or 'triangle feature'. Each component has two parameters 'centre' and 'peak'. The number of components is determined when the results structure is created, it is derived from the approximate width of features and the
Examples:  arclocat in 4. 20. mode=triangle dialog=f
This will scan through (all rows of) the NDF called "in". It
looks out for features of 4 pixels full width at half maximum
and with a peak value of at least 20 above the local
background. The features are fitted as triangles. The search is
automatic. Thus a new results structure in the input NDF’s
Specdre Extension is created with the locations (centres) and
peaks of located features.

arclocat in 4. mode=Gauss dialog=g rownum=5
This will use the graphic dialogue. Starting with the fifth row
the user can use the mouse cursor to choose features that are
to be deleted from or added to the list of located features.
This can be used to improve on an automatic run, or when no
features have been located so far. If you try to add a feature
to the list, a Gauss fit is tried in the vicinity of the
cursor-selected position.

Notes:  This routine recognises the Specdre Extension v. 0.7.
This routine works in situ and modifies the input file.

D.13 ARCSDI-Corrects for arc line curvature

Description: Program to allow correction of 2-d spectra for S-distortion using an arc - as a
preliminary stage prior to wavelength calibration and scrunching. The lines are located by
fitting gaussians to them. These positions are then used to fit a chebyshev polynomial to
- one for each line. The intermediate positions are interpolated from these. Once this is
done the data are shifted and interpolated in the x-section direction to align them all.

Parameters: IMAGE IMAGE = FILE (Read) Name of image for input This should be a file
containing an arc spectrum.
ARC_OPTS ARC_OPTS = CHARACTER (Read) Enter arc fit option NEW : set up a new
wavelength calibration REPEAT :Iterate on previous calibration. CLONE : CLone a
previous calibration. OLD : Correct using previous results
OUTPUT OUTPUT = FILE (Write) Name of output file File to contain corrected data.
YSTART YSTART = INTEGER (Read) analysis lower limit The data between the limits
ystart and yend is extracted and the resultant spectrum is used to locate the lines.
YEND YEND = INTEGER (Read) analysis upper limit The data between the limits ystart
and yend is extracted and the resultant spectrum is used to locate the lines.
YBLOCK YBLOCK = INTEGER (Read) Enter analysis x-sect width Each window is of
this width (except perhaps the final one).
ITERATION ITERATION = INTEGER*2 (Read) New value of iteration
ORDER ORDER = INTEGER (Read) order for polynomial fitting This is for the continuity
correction of the data. Ideally the arc should have been pre-processed with ARCSDI,
so a low order e.g. 2 should be used.
MAXLINES  MAXLINES = INTEGER (Read) Maximum number of lines to allow room for This must be greater than or equal to the number of lines fitted, so room should be allowed in case any more are to be added later.

CLFILE  CLFILE = FILE (Read) Name of image for cloning from This should be a file containing an arc spectrum.

TOLS  TOLS = CHARACTER (Read) For use in batch only

KEEP_ITT  KEEP_ITT = LOGICAL (Read) keep iteration files’

PRFITS  PRFITS = LOGICAL (Read) Print out details of fitting

PLOTCORR  PLOTCORR = LOGICAL (Read) Plot correction?

Source comments:  available

D.14  ASCIN-Read a 1-D or N-D data set from an ASCII table.

Usage:  ascin in lines colaxes=? coldata=? [start=? step=? end=?] out=?

Description:  This routine reads axis values, pixel widths, data values, and data errors from an ASCII table into an NDF data structure. Most of these items are optional, mandatory are only axis values for each axis and data values. Pixel widths can be read only in the one-dimensional case.

Parameters:  INFO  INFO = _LOGICAL (Read) If false, the routine will issue only error messages and no informational messages. This parameter is of significance only if the output is multi-dimensional. [YES]

TOL  TOL = _REAL (Read) The tolerated fraction of the pixel size by which the table coordinates may deviate from the pixel coordinates. For a line read from the ASCII table, if any one of the axis values deviates by more than TOL times the pixel step, then the information from the table is disregarded. This parameter is of no significance, if the output is one-dimensional, since in that case the axis values found will define the exact (non-linear) grid. [0.2]

BAD  BAD = _REAL (Read) The alternative bad value, i.e. the bad value used in the table. Any data or error value found in the table that is equal to BAD, is replaced by the bad value before insertion into the output. [-999999.]

IN  IN = FILENAME (Read) The file containing the ASCII table.

LINES  LINES( 2 ) = _INTEGER (Read) The line numbers of the first and last lines to be used from the table file. [1,9999]

COLAXES  COLAXES( 7 ) = _INTEGER (Read) The column numbers where the axis values are to be found. All axes must be specified, i.e. at least one. The number of leading non-zero elements defines the number of axes in the output. [1,2]

COLWIDTH  COLWIDTH = _INTEGER (Read) The column numbers where the pixel width values are to be found. This parameter is of significance only if the output is one-dimensional. Enter a 0 if no width information is available. [0]

COLDATA  COLDATA( 2 ) = _INTEGER (Read) The column numbers where the data values (first element) and their associated error values (second element) are to be found. If no error information is available, enter 0 as second element. [3,0]
**START** \( \text{START}(7) = \text{REAL} \) (Read) The coordinates of the first pixel. This parameter is of no significance, if the output is one-dimensional, since in that case the axis values found will define the exact (non-linear) grid.

**STEP** \( \text{STEP}(7) = \text{REAL} \) (Read) The coordinate increments per pixel. This parameter is of no significance, if the output is one-dimensional, since in that case the axis values found will define the exact (non-linear) grid.

**END** \( \text{END}(7) = \text{REAL} \) (Read) The coordinates of the last pixel. This parameter is of no significance, if the output is one-dimensional, since in that case the axis values found will define the exact (non-linear) grid.

**OUT** \( \text{OUT} = \text{NDF} \) (Read) The NDF where to store the data.

**Source comments**: The user specifies in which columns the different items are to be found. A range of line numbers to be used can be specified. Comment lines may be interspersed in this line range, if they are marked by an exclamation mark in the first or second character. All columns leftward of the rightmost used column must be numeric, non-numeric data may follow in further columns. Up to 132 characters are read from table lines. Numbers are read as \text{\_REAL}.

If the result is one-dimensional, the axis values will be taken literally to define a grid, which in general may be non-linear and non-monotonic. If the result is multi-dimensional, the routine will guess from the table a grid that is linear in all directions. The parameter system is consulted to confirm or modify the suggested grid.

The data value read from a line will be stored into exactly one output pixel, if and only if the table coordinates match that pixel’s coordinate to within a specified fraction of the pixel step. Pixels for which no data are in the table are assigned the bad value. Table data equal to a specified “alternative bad value” are replaced by the bad value before insertion into the data set. Where more than one table line corresponds to the same pixel, the pixel is assigned the last value from the table. That is, later specifications of the same pixel override previous ones.

**Examples**: \text{ascin} \( [1,9999] \) \text{colaxes}=[1,2] \text{coldata}=[3,4]

\text{start}=[0,0] \text{end}=[2.5,5] \text{step}=[0.1,1] \text{out} \text{out}

This will read the data from the ASCII file \text{IN}, using line numbers 1 to 9999 (or till end of file if there are less lines in \text{IN}). The 1st axis data are taken from the first column, the 2nd axis data from the second column. The image data are taken from the 3rd column and their errors from the 4th column. The routine tries to store the table data into a grid with the 1st axis running from 0 to 2.5 in steps of 0.1 (26 pixels) and the 2nd axis running from 0 to 5 in steps of 1 (6 pixels). If a coordinate pair from columns 1&2 matches any pixel centre well
enough, the data from columns 4&5 are entered into the corresponding element of the data and errors array. The data file is OUT.

ascin in out [25,39] colaxes=5 coldata=[3,0]
Here the output is one-dimensional and without errors array (thus the zero in COLDATA). Only lines 25 to 39 from IN are used. The axis data are from the 5th column and the spectrum data from the 3rd column. (Note that columns 1, 2 and 4 must contain numeric data.) The axis grid need not be specified. The axis values from the table will be taken literally to form a grid that is in general non-linear and non-monotonic.

Implementation status: It is not possible to read axis values from the table in double precision or create a double precision axis array.

D.15 ASCOUT-Write an NDF to an ASCII table.

Usage: ascout in out

Description: This routine takes an NDF (section) and writes it to an ASCII table.

Parameters: WIDTH
WIDTH = _LOGICAL (Read) True if pixel widths are to be written, too.
[NO]
BAD  BAD = _REAL (Read) The alternative bad value. Where the data or variance array has bad values, BAD is written to the ASCII table.

IN  IN = NDF (Read) The input NDF.
OUT OUT = FILENAME (Read) The ASCII output file.

Source comments: ASCOUT

The first part of the output file is a header giving textual information and a head for the table. These lines start with a blank carriage return control character followed by an exclamation mark as the first printed character. The table itself has to the left all the axis values and optionally the pixel widths, and to the right the data value and its error if known. The spectroscopic axis is written with higher precision (12 significant digits instead of 7) if its storage type is _DOUBLE. The total number of table columns can be 8 at most. All pixel widths are written if and only if requested, regardless of whether there is explicit information in the input file. Each width occupies the column to the right of the corresponding centre value.

Examples: ascout in(1.5:2.5) out
This expects a 1-D data set in IN and will write to the ASCII file OUT the information for axis values between 1.5 and 2.5. Should IN be more than 1-D, the first hyper-row would be used.
ascout in(1.5:2.5,10:15) out
This will accept a 2-D data set in IN and write to OUT the
information for 1st axis coordinate values between 1.5 and 2.5
and for 2nd axis pixel number between 10 and 15. Note that
integers in the section specification are interpreted as pixel
numbers.

Notes: This routine recognises the Specdre Extension v. 0.7.
This routine is available in the Specdre graphical user interface.

D.16 BBODY-Calculate a black body spectrum.


Description: This routine calculates for a given (vacuum) wavelength or frequency axis the
intensity of a black body at given temperature. The intensity is the energy per unit time,
per unit area, per unit solid angle, and per unit frequency (and for all polarisations):

\[
B_\nu = \frac{2 \ h \ \nu^3 \ \Gamma}{c^2 \ \exp(h \ \nu/kT) - 1}
\]

where \(c\) is the velocity of light, and \(h\) and \(k\) are the Planck and Boltzmann constants.

Parameters: LOGAR LOGAR = LOGICAL (Read) True if the common logarithm of intensity is
to be written rather than the intensity itself. [NO]
TEMP TEMP = REAL (Read) The black body temperature in Kelvin.
ERRTEMP ERRTEMP = REAL (Read) The error in the black body temperature in Kelvin.
IN IN = NDF (Read) The file holding axis data to be used. Enter the null value (!) to read
axis data parameters from keyboard.
XSTART XSTART = REAL (Read) The spectroscopic value (pixel centre) for the first
output pixel.
XSTEP XSTEP = REAL (Read) The spectroscopic step (pixel distance) for the output
pixels.
XEND XEND = REAL (Read) The spectroscopic value (pixel centre) for the last output
pixel.
XLABEL XLABEL = CHARACTER (Read) The label for the spectroscopic axis. Allowed
values are "wavelength" and "frequency". [wavelength]
XUNIT XUNIT = CHARACTER (Read) The unit for the spectroscopic axis. If the label
is "wavelength" then the unit can basically be "m" for metre, "um" or "micron" for
micrometre, or "Angstrom" for Angstroem. If the label is "frequency" then the unit
must be basically "Hz" for Hertz. Any of these units may be preceded by a power
of ten, so it could be "10**1*Angstrom" if you want to use nanometre as unit, or
"10**-9*m" to the same effect. The power must be an integer. You can achieve a
logarithmic axis by specifying something like "log10(10**-3*micron)". In this example
the axis values will be the common logarithms of the wavelength in nanometres.
OUT OUT = NDF (Read) The output file.
**Examples:**

```
bbody 5500 in=in out=out
```
This calculates the black-body spectrum for 5500 K. The spectrum is written to file OUT. The routine tries to find all necessary information for the 1st (and only) axis in OUT from the spectroscopic axis of the file IN. Since LOGAR is left at its default value of FALSE, the data are intensity in Jy/sr.

```
bbody 2.7 logar=true in=! xstart=0 xstep=0.05 xend=6
```
This calculates the black-body spectrum for 2.7 K. The spectrum is written to OUT. No input file is specified. The axis contains the logarithms of wavelengths in micron, which run from 0 (1 micron) to 6 (1 metre). Since LOGAR=TRUE, the data are the logarithms of intensity in Jy/sr.

```
bbody 1e6 logar=true in=! xstart=-1 xstep=0.05 xend=2
```
This calculates the black-body spectrum for 1 million K. This time the axis is logarithms of frequency, the units used are $10^{15}$ Hz. The frequency range covered is from $10^{14}$ Hz to $10^{17}$ Hz.

**Notes:** This routine recognises the Specdre Extension v. 0.7.


**D.17 BCLEAN-Automatic removal of bad lines and cosmic rays from CCD data**

**Description:** BCLEAN runs non-interactive cleaning algorithms on a CCD image to detect and remove bad lines and cosmic rays.

**Parameters:**

- **IMAGE** The name of the image to be cleaned of bad rows and cosmic rays. Generally, this will be a CCD image. Note the program assumes the image is oriented so that bad lines are horizontal (i.e., occupy contiguous regions in memory).

- **CRSIG** The cosmic ray search algorithm looks at every pixel in the image and compares its value with the average value of its four nearest neighbours. To be regarded as a cosmic ray, the pixel must exceed that average value by an amount greater than CRSIG times the square root of the average value. Sensible numbers are probably in the range 2.0 to 10.0, but the best way to tune the operation is to try with a few different values, looking at the unfixed results to see which pixels were considered cosmic rays. This constraint is in addition to those enforced by CRFACT and CRMINV. If you set the SHARPNESS keyword, a test of the sharpness of the cosmic ray is also performed.

- **CRFACT** To be regarded as a cosmic ray, a pixel value must exceed the average of its neighbours by an amount that exceeds CRFACT times that average value. This constraint is in addition to those enforced by CRSIG and CRMINV.
CRMINV  To be regarded as a cosmic ray, a pixel value must exceed the average of its neighbours by at least CRMINV. This constraint is in addition to those enforced by CRSIG and CRFACT.

CRSHARPNESS  If you set the SHARPNESS keyword, then a sharpness test is performed on each pixel that passes the constraints of CRSIG, CRMINV, and CRFACT. The sharpness test measures the ratio of the height of the "core" of the cosmic ray to the height of the "wings". Stars, even under conditions of excellent seeing, tend to have wings that are a significant fraction of the peak height, whereas cosmic rays tend to have much sharper edges.

BFACT  The bad row search algorithm looks through an array formed by collapsing the image along the rows, searching for rows that are lower than their neighbours by a value that is greater than BFACT times the median difference between adjacent elements of the array in the local neighbourhood. Sensible values of BFACT are quite low, especially if combined with a large-ish value for BRPASS. BFACT=1.5, BRPASS=4 are reasonable values, and BFACT can be taken lower if necessary.

BRPASS  The bad row searcher makes BRPASS passes through the data, each time taking a different set of columns evenly distributed through the image. A row has to show up as bad in every pass in order to be considered bad.

DEGFIX  Bad data areas are interpolated by fitting local polynomials to the data. DEGFIX is the degree of polynomial to be used. Three seems a reasonable value for most data. DEGFIX is only prompted for if FIX is set.

OUTPUT  The name of the output image to be generated. If OUTPUT is the same IMAGE, the data is modified in situ.

NBROWS  The number of bad rows to be cleaned in the image. Note that a ‘row’ is a horizontal (as seen on the Grinnell) line in the image. Note that NBROWS and BROWS are not prompted for unless AUTOROW is not set.

BROWS  An array giving the numbers of the bad rows to be cleaned in the array. Note that NBROWS and BROWS are not prompted for unless AUTOROW is not set.

AUTOROW  If AUTOROW is set, the program will perform a search for bad rows, using the BFACT and BRPASS parameters. If it is not set, the rows will be prompted for as NBROWS and BROWS. AUTOROW is set by default.

FIX  If FIX is set, the output file has the detected cosmic rays and bad rows corrected by interpolation. If FIX is not set, the data is not corrected, but is written out with the bad areas flagged by being set to a value 1000 less than the previous minimum value in the image. FIX=NO can be used to highlight the areas that will be fixed with the current parameter values. FIX is the default. Note that an image that has been run through BCLEAN with FIX=NO will not subsequently BCLEAN properly, so should usually be output to a scratch file.

WARNING - you should examine the output from BCLEAN carefully to ensure that the parameters that you have chosen are appropriate. FIX=NO and TEXTFILE are useful here.

SHARPNESS  If SHARPNESS is set, the program will perform a test on the sharpness of possible cosmic rays events. This test is in addition to those specified by CRSIG, CRFACT, and CRMINV.
TEXTFILE  If TEXTFILE is set, the program will produce a text file containing the results of the cosmic ray search. This file is very useful when trying to choose appropriate values for CRSIG, CRFACT, CRMINV, and CRSHARPNESS. The file is called BCLEAN.LIS.

DIRECTION  In the zapping of cosmic rays, the pixels are replaced with a value interpolated from the neighboring pixels. This can be done along rows or columns, or one can let the computer decide which one gives smaller residuals (i.e., the best).

See also: FIGARO: CLEAN, COSRE, MEDFILT, MEDSKY, TIPPEX, KAPPA: FFCLEAN, CHIPX, FILLBAD, GLITCH, MEDIAN, MSTATS, ZAPLIN.

Source comments:

This is the non-interactive CCD image cleaning program, which removes bad rows and cosmic rays from images. Note that it expects the data to be orientated so that bad transfer lines in the chip are horizontal - i.e. are rows, rather than columns. The program will detect and blank out the bad data, and optionally fix it up. Running without the fixup allows the user to see what parts of the image will be affected and provides a chance to modify the cleaning parameters accordingly. For details of the cleaning algorithms used, see the comments in the listings of FIG_ABROWS, FIG_ZAPRAYS, FIG_VERTICAL, FIG_FIXAREA. There are four parameters connected with cosmic ray detection, two which affect bad line detection, and one that controls the interpolation used to fix the data.

WARNING: you are strongly advised to examine the effects of this program on your images.

Command line parameters -

IMAGE  (Character) The name of the image to be cleaned.
CRSIG  (Numeric) Cosmic Ray SIGma value. The cosmic ray searcher tests the value of each pixel against the average value of the surrounding pixels. It must exceed the average value by more than CRSIG*(square root of average value).
CRFACT (Numeric) Cosmic Ray FACTor. A cosmic ray must also exceed the average value by CRFACT*(the average value).
CRMINV (Numeric) Cosmic Ray MINimum Value. A cosmic ray must also exceed the average value by CRMINV.
CRSHARPNESS (Numeric) Cosmic Ray SHARPNESS. If the SHARPNESS keyword has been set, then a cosmic ray must also satisfy the sharpness criterion: the height of the cosmic ray above the immediately surrounding sky must exceed the difference between the immediately surrounding sky and the sky a bit further away, by more than a ratio of CRSHARPNESS. Stars tend to have lower values of this ratio than cosmic rays. The default value is 10.
BRFACT (Numeric) The bad row searcher looks through an array formed by collapsing the image along the rows, looking for rows that are lower than their neighbours by a value that is greater than BRFACT*(median difference
between neighbours in the neighbourhood).

**BRPASS**
(Numeric) Bad Row PASS value. The bad row searcher makes BRPASS passes through the data, each time taking a different set of columns evenly distributed through the image. A bad row must show up in all passes.

**DEGFIX**
(Numeric) The degree of polynomials to be used in interpolating over bad data.

**OUTPUT**
(Character) The name of the output image to be generated. If this is the same as IMAGE, the correction will be performed in situ.

**NBROWS**
(Numeric) The number of bad rows to be fixed.

**BROWS**
(Numeric vector) The numbers of the bad rows to be fixed. If NBROWS and BROWS are specified explicitly, then they will be used. Otherwise an automatic bad line search will be performed, unless overridden by the setting of the AUTOROW keyword.

**DIRECTION**
(Numeric) Indicates along which direction on the CCD the cosmic rays will be interpolated across. 1 means columns, -1 means rows, and 0 means let the computer decide which gives smaller residuals.

Command keywords -

**AUTOROW**
If set, an automatic bad row search will be performed. If NBROWS or BROWS are specified explicitly, AUTOROW=NO will be assumed.

**FIX**
If set, the bad data found will be fixed. Otherwise, the output image will simply have the bad pixels flagged by a specific flag value.

**SHARPNESS**
If set, the "sharpness" test for cosmic rays will be performed in addition to the other tests.

**TEXTFILE**
If set, a text file (BCLEAN.LIS) will be produced giving a summary of the cosmic ray test results. This file is useful when deciding on the cosmic ray selection parameters.

KS / CIT 29th June 1984

### D.18 BFFT-Takes the reverse FFT of a complex data structure

**Usage:** bfft frequency_data spatial_data

**Description:** These Figaro functions take the FFT of the data in a file. FFT performs a forward transform, BFFT performs an inverse transform. The input file must contain a complex data structure, i.e. one with IMAGINARY and DATA components.

The data may be multi-dimensional; if it is, a multi-dimensional FFT is performed. Note that the Figaro routine R2CMPLX will turn an existing real data structure into a complex one acceptable to this routine. FFT does not perform any cosine belling or other tapering of the data, nor does it reduce it to a zero mean.

**Parameters:** CDATA CDATA is the name of a complex data structure. Such structures for the spatial domain are most easily produced using the R2CMPLX command. For the
frequency domain, such data were usually created by R2CMPLX and transformed by FFT.

**OUTPUT** OUTPUT is the name of the resulting Fourier transformed data. If OUTPUT is the same as CDATA then the transform is performed in situ; otherwise, a new file is created.

**Notes:** The fourier transform routines available in the various math libraries (NAG, IMSL, etc) all have slightly different characteristics, which show up in the programs that use them. This routine has been written around the NAG library (mainly the routines C06FAF and C06FJF), so many of its characteristics may be deduced by reading the relevant parts of the NAG manuals. In version 5.0 this routine was changed to use the PDA library, effectively FFTPACK routines. The data is re-ordered by FFT after the transform so that the zero frequency component is in the center of the resulting array, and this re-ordering is reversed by BFFT before the transform. This means that after FFT has been run, the various axes all go from -N to +N where N is the Nyquist frequency. New axis data structures that reflect this are created by FFT and will be deleted by BFFT.

**See also:** FIGARO: FFT, COSBELL, CMPLX2I, CMPLX2R, CMPLX2M, I2CMPLX, R2CMPLX, KAPPA: CONVOLVE, LUCY, MEM2D, WIENER.

**Authors:** ks: Keith Shortridge (AAO)
   jm: Jo Murray (RAL, Starlink)
   jms: ??? (AAO)
   hme: Horst Meyerdierks (UoE, Starlink)

### D.19 BSMULT-Atmospheric band removal using a B star calibration spectrum

**Description:** BSMULT applies the calibration spectrum derived from a B-star (for atmospheric bands) to the spectrum of an object.

**Parameters:**
- **SPECTRUM** The spectrum specified by SPECTRUM will be multiplied by the B star calibration spectrum (specified by BSTAR). BSMULT differs from a simple multiplication (e.g. IMULT) in taking account of the differences in air mass of the calibration object and the object being calibrated. So both should have valid values for .OBS.SECZ (the program will complain if they don’t). Note that the calibration is by multiplication, and so is not suitable for data in logarithmic flux units such as magnitudes.

- **BSTAR** BSTAR specifies the B-star calibration spectrum, obtained by dividing a B star (or other suitable calibration spectrum) by its continuum. Any regions where no correction is desired should be set to 1.0.

- **OUTPUT** OUTPUT specifies the name of the calibrated spectrum to be produced by BSMULT. Note that this can be the same as for SPECTRUM, in which case the operation will be performed in situ.

- **BETA** The algorithm used by BSMULT is:
  \[
  \text{OUTPUT} = \text{SPECTRUM} \times (\text{BSTAR}^{(\text{SSECZ} / \text{BSECZ})} \times \text{BETA})
  \]
where SSECZ and BSECZ are the air masses for the spectrum and the B star respectively. BETA should be 0.5 for saturated lines, 1.0 for unsaturated lines, and the generally accepted best compromise value for all lines (determined by P.J. Young) is 0.55.

**Source comments:**
M U L T

Multiplies a spectrum by a B star calibration spectrum - which should be 1.0 except in regions where there are atmospheric bands. This routine differs from an ordinary multiplication (e.g. as in IMULT) since it includes a term for the correction due to the different air masses of the two objects. This means that both the original spectrum and the B star spectrum must have valid OBS.SECZ values. If the spectrum to be multiplied is 2 dimensional, the same operation is repeated for each of the spectra it contains.

**Command parameters -**

- SPECTRUM (Character) The spectrum to be corrected.
- BSTAR (Character) The B star calibration spectrum.
- BETA (Numeric) Power to which relative air mass is to be raised - see FIG_BSMULT.
- OUTPUT (Character) The resulting spectrum.

**Command keywords - None**

**User variables used - None**

KS / CIT 3rd July 1984

**D.20 CADD-Add back fitted continuum**

**Description:** A polynomial previously fitted to the continuum is evaluated and this is added.

**Parameters:**
- IMAGE IMAGE = FILE (Read) Name of image for input
- OUTPUT OUTPUT = FILE (Write) OUTput Name of output file OUTPUT is the name of the resulting spectrum. If OUTPUT is the same as INPUT the data in the input file will be modified in situ. Otherwise a new file will be created.

**Source comments:** None available

**D.21 CALDIV-Generate calibration spectrum from continuum standard spectra**

**Description:** Given a standard spectrum giving the continuum flux densities for each element for a particular standard star (probably generated by GSPIKE followed by INTERP) and the continuum of an observed spectrum of the same object, CALDIV calculates the instrumental response for each element. Note - this routine is intended for standards where the tables give the calculated continuum values at various points (e.g. Filippenko/ Greenstein
standards), rather than the average measured flux density over a given wavelength range centered on each point. CALDIV will not accept a standard spectrum that has data in magnitude units.

**Parameters:**

**STANDARD** A spectrum giving the actual continuum flux density at each point for a standard star. This will probably have been generated by GSPIKE followed by INTERP. The data should not be in magnitude units.

**SPECTRUM** An observed spectrum of the object whose tabulated fluxes were used to generate STANDARD. Note that CALDIV does not insist on the spectrum’s having been scrunched, but things are usually easier if it has been. The wavelength range of SPECTRUM must match that of STANDARD exactly.

**OUTPUT** A spectrum whose values give the instrumental response in "units per (counts per second per angstrom)", where "units" are the units used by the input standard spectrum.

**Source comments:**

Divides the interpolated continuum spectrum of a standard star (e.g. a Filippenko/Greenstein standard, probably created by GSPIKE and INTERP) by the observed continuum of the same star, in order to generate the instrumental response calibration spectrum.

**Command parameters -**

**STANDARD** (Character) The interpolated continuum spectrum of the standard star. Note: This should not be in magnitude units, and should probably not contain a .TABLE.BANDWIDTH data object, since this would indicate that this was generated from published data giving average observed flux density over a wavelength range (e.g. Oke/Gunn data) rather than a fitted continuum.

**SPECTRUM** (Character) The observed continuum spectrum of the standard star. Note that both STANDARD and SPECTRUM should be on the same wavelength scale (given by a .X.DATA array) and ideally this will be a linear scale.

**OUTPUT** (Character) The resulting calibration spectrum.

**Command keywords -** None

**User variables used -** None

KS / CIT 28th May 1984

---

**D.22 CCDLIN- Applies a linearity correction to AAO CCD data**

**Description:** CCDLIN applies a linearity correction to AAO CCD data. The correction applied has the form determined by John Barton (RCA#5 Non-Linearity Correction - AAO internal
document), and has two parameters: a correction factor (alpha), and a bias whose value is assumed when applying the correction.

**Parameters:**
- **IMAGE** The name of the image to be corrected for non-linearity. It should be a raw image obtained with an AAO CCD.
- **OUTPUT** Specifies the resulting corrected image. If this is the same as IMAGE, the image is corrected in situ. Otherwise, a new image is created.
- **ALPHA** The value of the correction factor used. The correction is given by
  \[ M = C(1 + \alpha C) \]
  where \( M \) and \( C \) are the measured and corrected intensities (after bias subtraction) respectively. John Barton’s memo gives a value of \( \alpha = 3.16 \times 10^{-6} \), and this is the reset value.
- **CBIAS** Since the linearity correction is expressed in terms of intensities after bias subtraction, a bias must be assumed during the calculation. The value is not very critical (any error in bias results in a much smaller error in the corrected data), and can safely be assumed to be constant over the image. The reset value is 180.

**Source comments:**
Applies a linearity correction to data from the AAO RCA CCD. The correction applied is that given by John Barton (RCA#5 Non-Linearity Correction, AAO Internal Document).

Command parameters -

- **IMAGE** (Character) The name of the structure containing the image.
- **ALPHA** (Numeric) The value of the constant alpha in the expression giving the linearity correction.
- **CBIAS** (Numeric) The value of the bias level to be applied when making the correction. This is not a particularly critical parameter.
- **OUTPUT** (Character) The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

---

**D.23 CCUR-After SPLIT, uses graphics cursor to indicate data values**

**Description:** If a spectrum has been displayed on the current soft plotting device, CCUR allows the user to move the cursor around the display, indicating the current cursor position and data value each time a terminal key is pressed.

**Parameters:**
- **SCREEN** The screen management routines used by CCUR do not know how to handle all the various types of VDU. If you find that you are having problems getting output from CCUR, try setting SCREEN=NO - this bypasses the fancy screen formatting and puts out the data crudely but directly to the terminal as though it were not a VDU.
Source comments:

Uses the cursor to get information about data as displayed on the soft graphics device.

Command variables - None.

Command keywords - None.

User variables - 

(>) SOFT (Character) Device / type for soft plots. See documentation on 'SOFT' command for details.

(>) TVXST (Numeric) X-start value for current plot.

(>) TVXEN (Numeric) X-end value for current plot.

(>) TVLOW (Numeric) Lowest value of current plot.

(>) TVHIGH (Numeric) Highest value of current plot.

(>) TVFILE (Character) Name of currently displayed data.

KS / CIT 23rd July 1984

D.24 CDIST-S-distortion correction using SDIST results

Description: Applies the distortion correction determined by SDIST to an image.

Parameters: IMAGE The name of the S-distorted image that is to be corrected.

YSTART If the image contains, for example, a single spectrum in the center and a lot of unused background at the edges, it can save processing time if only a part of the Y-range of the data is corrected. YSTART is the first Y value in the range corrected by CDIST.

YEND Specifies the last Y value in the range corrected by CDIST.

OUTPUT Specifies the name of the resulting, corrected, image. If OUTPUT is different to IMAGE, a new image will be created. If it is the same, the correction will be performed in situ.

MAXDEGY If a number of spectra have been used to determine the image distortion, CDIST interpolates between their fitted positions by fitting a low order polynomial to them. MAXDEGY specifies the order of this polynomial.

ROTATE The processing performed by CDIST normally involves accessing the image in a way that causes excessive paging on the VAX. This paging may be considerably reduced by rotating the image (in a relatively efficient way) before and after the correction. Of course, this introduces the overhead of the two rotations. Generally, large images should be rotated, small images should not. The results should be identical in both cases.

Source comments:

Performs an S-distortion correction for an image, using the distortion analysis provided by SDIST. If the SDIST analysis was for a single spectrum, the the correction is '1-dimensional'
in that it consists simply of applying a shift in Y to the
data for each column of the image. The shift is determined
separately for each column from the polynomial fitted by SDIST.
If the SDIST analysis was for more than one spectrum, the
correction is what might be termed '1.5-dimensional' - the
data is only redistributed within columns, but the amount of
shift varies along each column in a manner determined by fitting
a low order polynomial to the results of evaluating the polynomials
that SDIST fitted to each spectrum. The 1.5D algorithm actually
reduces to the 1-D case when there is only one spectrum, but it
simplifies things to think of them as distinct cases.

Command parameters -

IMAGE (Character) The name of the image to be corrected.

YSTART (Numeric) The first Y value to be used.

YEND (Numeric) The last Y value to be used. Using YSTART
and YEND to limit the range of data corrected will
speed up the operation.

OUTPUT (Character) The name of the resulting image. This
can be the same as IMAGE, in which case the correction
will be performed in situ.

MAXDEGY (Numeric) The maximum degree polynomial to fit to the
spectral positions in the case where there is more
than one spectrum covered by the SDIST analysis.

Command keywords -

ROTATE If set, the image to be corrected will be rotated
prior to the application of the correction. This
minimises the number of page faults during the correction,
but at the expense of the overheads of the rotation itself.

User variables - None

Input files -

SDIST.DAT contains the results of the fit(s), as written by
SDIST, in a format treated as follows -

3 header lines, all beginning with ‘*’
One line giving the number of spectra traced, in the
format 20X,I5.
Then, for each spectrum traced, one record giving
the spectrum number, and the leftmost and rightmost
pixels covered by the trace, in format
11X,I5,17X,I5,4X,I5, then 1 record giving the average
Y value in the spectrum, in format 16X,F13.7,
which is followed by 3 records giving the 11
polynomial coefficients for the fit, in 3D23.16.
Coefficients are given constant first, with any unused coefficients set to zero.

KS / CIT 6th Feb 1984

D.25 CENTERS-Generate file of object centroids from ICUR, IGCUR or IMPOS output

Description: Figaro function that takes a list of approximate X,Y positions of objects in a two-dimensional direct image and calculates centroids (that is, accurate positions) for these objects.

The approximate positions input are obtained from environment variables. These variables should be set up prior to running CENTERS, usually by using Figaro functions IGCUR or ICUR. Alternatively, you may enter the positions into a text file and use IMPOS to read this file and copy the values into the environment variables required by CENTERS.

The computed centroids are output to a new file called center.dat.

Parameters:

- **IMAGE** The name of the image containing the objects. This need not be the image used to generate the original list of points - it may be a similar image in a different waveband, with offsets in X and Y with respect to the original image.

- **APERTURE** The aperture used for the centroiding. The aperture actually used is a box APERTURE*2+1 pixels square, which is a rough approximation to a circle of radius APERTURE.

- **XOFF** The offset in X, in pixels.

- **YOFF** The offset in Y, in pixels.

- **PROFBOX** Size of profile box.

- **PROFMIN** Minimum displayed value.

- **PROFMAX** Maximum displayed value.

- **PFILE** If set, a formatted file giving the summed radial profiles will be produced.

- **DISPROF** If set, the profile is displayed.

- **CHGPROF** If set, the profile display can be changed.

Source comments:

Figaro function that takes a list of approximate X,Y positions of objects in a two-dimensional direct image and calculates centroids (that is, accurate positions) for these objects.

The approximate positions input are obtained from environment variables. These variables should be set up prior to running CENTERS, usually by using Figaro functions IGCUR or ICUR. Alternatively, you may enter the positions into a text file and use IMPOS to read this file and copy the values into the environment variables required by CENTERS.

The computed centroids are output to a new file called center.dat.
Command parameters -

**IMAGE** (Character) The name of the image containing the objects. This need not be the image used to generate the original list of points - it may be a similar image in a different waveband, with offsets in X and Y with respect to the original image.

**APERTURE** (Integer) The aperture used for the centroiding. The aperture actually used is a box $APERTURE*2+1$ pixels square, which is a rough approximation to a circle of radius $APERTURE$.

**XOFF** (Real) The offset in X, in pixels.

**YOFF** (Real) The offset in Y, in pixels.

**PROFBOX** (Real) Size of profile box.

**PROFMIN** (Real) Minimum displayed value.

**PROFMAX** (Real) Maximum displayed value.

Command keywords -

**PFILE** If specified, a formatted file giving the summed radial profiles will be produced.

**DISPROF** If specified, the profile is displayed.

**CHGPROF** If specified, the profile display can be changed.

User variables - (”>” input)

(>) **SOFT** (Character) The current device/type for soft plots.

(>) **NPIXELS** (Real) Number of objects for which positions are specified.

(>) **XPIXELS** (Real array) List of approximate X positions of the objects for which the centroids are to be computed (pixels).

(>) **YPIXELS** (Real array) List of approximate Y positions of the objects for which the centroids are to be computed (pixels).

Output -

center.dat contains one record for each point, giving

$XCENT,YCENT,IX,IY,DX,DY,AP$

in the format $2F8.2,2I5,2F8.2,I4$ where

$XCENT,YCENT$ give the position of the centroid

$IX,IY$ are the original pixel position of the point.

$DX,DY$ are the offsets in X and Y, and

$AP$ is the value used for APERTURE.

If the centroid for a point cannot be determined, a record is written giving

‘*** No centroid ’,IX,IY,DX,DY,AP

in the format $A,2I5,2F8.2,I4$.

KS / CIT 29th Sept 1983
D.26 CFIT-Generate a spectrum using the cursor

Description: Figaro function to generate a spectrum by interpolation between points selected interactively using a display device cursor. CFIT assumes that a spectrum has already been displayed by SPLOT, and will generate a new data structure based on the spectrum displayed, with only the data changed.

Parameters: OUTPUT The name of the output file to be created. If this is the same as the displayed spectrum, the data in the displayed spectrum will be modified.

CHANGE Set to modify points.

REDRAW Set to redraw the original spectrum.

Source comments:

Figaro function to generate a spectrum by interpolation between points selected interactively using a display device cursor. CFIT assumes that a spectrum has already been displayed by SPLOT, and will generate a new data structure based on the spectrum displayed, with only the data changed.

Command parameters -

OUTPUT (Character) The name of the output file to be created. If this is the same as the displayed spectrum, the data in the displayed spectrum will be modified.

Command keywords -

CHANGE Set to modify points.

REDRAW Set to redraw the original spectrum.

User variables used - (">" input, "<" output)

(>) TVFILE The name of the displayed spectrum
(>) TVXST The first displayed x-value
(>) TVXEN The last displayed x-value
(>) TVHIGH The maximum displayed y-value
(>) TVLOW The minimum displayed y-value
(>) TVCOLOR The GRPLOT code for the plot colour
(>) SOFT The device/type string defining the display device

KS / CIT 17th May 1983

D.27 CHANGED-Indicate fits invalidated due to "cleaning" of an image

Description: To indicate which fits have changed due to "cleaning" of image during fitting (this is due to bits missed previously). This is set at data being different by more than 1, or 1% of the mean value whichever is larger. This situation can arise with data badly affected with cosmic rays where some are initially missed.
Parameters: IMAGE IMAGE = FILE (Read) Name of image for input

IMAGE2 IMAGE2 = FILE (Read) Name of image for input

Source comments: available

D.28 CLEAN-Interactive patching of bad lines, bad pixels in an image

Description: CLEAN is an interactive routine for fixing bad pixels and rows in CCD data. CLEAN can be used either as a prelude to BCLEAN, in order to see what parameters may be suitable or else it may be used after BCLEAN, to patch up any rows or pixels that BCLEAN missed.

Parameters: IMAGE IMAGE is the image - usually a CCD image - that is to be interactively cleaned of bad rows and bad pixels. Note that the orientation of the image data should be such that bad rows are horizontal (as seen on the Grinnell).

OUTPUT OUTPUT is the name of the image that results from the cleaning process. If OUTPUT is the same as IMAGE (the default value) the correction will be performed in situ.

QUIT Used to confirm quitting the application.

DEG Degree of fit to use for interpolation.

XSIZE Size of deletion box in X.

YSIZE Size of deletion box in Y.

HIGH Highest displayed data value.

LOW Lowest displayed data value.

See also: FIGARO: BCLEAN, COSREJ, MEDFILT, MEDSKY, SCLEAN, TIPPEX, KAPPA, FFCLEAN, CHPIX, FILLBAD, GLITCH, MEDIAN, MSTATS, ZAPLIN.

Source comments: Main routine for the Figaro 'CLEAN' command. Displays a CCD image and then allows the user to move around it with the cursor, selecting rows and columns to be corrected and cosmic rays to be zapped. The idea is that this routine can be used to fix up any areas in an image that were not fixed automatically by the non-interactive version ('BCLEAN'). It may also give a better idea of the best settings for the BCLEAN parameters.

Command parameters -

IMAGE (Character) The name of the image to be displayed.

OUTPUT (Character) The name of the resulting cleaned image. If the same as IMAGE, the image is cleaned in situ.

QUIT (Logical) Used to confirm quitting the application.

DEG (Integer) Degree of fit to use for interpolation.

XSIZE (Integer) Size of deletion box in X.
YSIZE (Integer) Size of deletion box in Y.
HIGH (Real) Highest displayed data value.
LOW (Real) Lowest displayed data value.

User variables - ("<" output, "!" modified)

(!) IMARRAY (Numeric array) Contains current image display parameters.
(<=) IMFILE (Character) Contains name of currently displayed image file.
(>) IDEV (Character) Contains name of display device.

KS / CIT 2nd July 1984

D.29 CLIP-Clip data above and below a pair of threshold values

Description: Clips an image, replacing any elements above a high threshold or below a low threshold with that threshold value.

Parameters: IMAGE The datafile to be threshold clipped.
LOWCLIP Any elements in the image that are less than LOWCLIP are set to LOWCLIP.
HIGHCLIP Any elements in the image that are greater than HIGHCLIP are set to HIGHCLIP.
OUTPUT The name of the resulting image. If OUTPUT is the same as IMAGE the operation will be performed in situ. Otherwise a new file will be created.

See also: FIGARO: IDIFF, RESCALE
KAPPA: THRESH

Source comments:

Clips an image (or spectrum, cube or whatever..). Given a low and a high threshold value, CLIP sets any elements above the high value or below the low value to the appropriate value.

Command parameters -

IMAGE (Character) The name of the structure containing the image.
LOWCLIP (Numeric) The low threshold value
HIGHCLIP (Numeric) The high threshold value
OUTPUT (Character) The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

KS / AAO 22nd July 1985

D.30 CMPLX2I-Extracts the imaginary part of a complex data structure

Description: CMPLX2I creates a data structure, in which the data is taken from the imaginary part of a complex data structure. The resulting structure is not complex (i.e. has only a single data array, with no 'real' and 'imaginary' arrays).
Parameters: **CDATA** The name of an existing complex data structure. The imaginary array from this will become the data array of the resulting structure.

**OUTPUT** The name of the data structure to be created. Its data array will come from the structure specified as CDATA, and it will not have any ‘real’ or ‘imaginary’ arrays. If **OUTPUT** is the same as **CDATA**, **CDATA** will be transformed into a non-complex structure (which means that its real part will be lost); otherwise, a new file is created.

Source comments:

<table>
<thead>
<tr>
<th>Command parameters</th>
<th>Source comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>CDATA (Character)</td>
<td>Creates a real data structure (i.e. one with just a real data array) as opposed to a complex data structure, (in the Figaro sense of a structure with both real and imaginary data arrays) from a complex data structure. In the case of CMPLX2R it is the real part of the complex data that forms the data array in the resulting structure. For CMPLX2I, it is the imaginary part, and for CMPLX2M it is the modulus of the complex data.</td>
</tr>
<tr>
<td>OUTPUT (Character)</td>
<td>The name of the resulting structure. This may be the same as <strong>CDATA</strong>. In either case a new file is created.</td>
</tr>
</tbody>
</table>

Command keywords - None


**D.31 CMPLX2M-Extracts the modulus of a complex data structure**

**Description:** CMPLX2M creates a data structure, in which the data is taken from the modulus of a complex data structure. The resulting structure is not complex (i.e. has only a single data array, with no ‘real’ and ‘imaginary’ arrays).

Parameters: **CDATA** The name of an existing complex data structure. The modulus array from this will become the data array of the resulting structure.

**OUTPUT** The name of the data structure to be created. Its data array will come from the structure specified as **CDATA**, and it will not have any ‘real’ or ‘imaginary’ arrays. If **OUTPUT** is the same as **CDATA**, **CDATA** will be transformed into a non-complex structure (which means that its real and imaginary parts will be lost); otherwise, a new file is created.

Source comments:

<table>
<thead>
<tr>
<th>Command parameters</th>
<th>Source comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>CDATA (Character)</td>
<td>Creates a real data structure (i.e. one with just a real data array) as opposed to a complex data structure, (in the Figaro sense of a structure with both real and imaginary data arrays) from a complex data structure. In the case of CMPLX2R it is the real part of the complex data that forms the data array in the resulting structure. For CMPLX2I, it is the imaginary part, and for CMPLX2M</td>
</tr>
<tr>
<td>OUTPUT (Character)</td>
<td>The name of the resulting structure. This may be the same as <strong>CDATA</strong>. In either case a new file is created.</td>
</tr>
</tbody>
</table>

Command keywords - None

D.32 CMPLX2R—Extracts the real part of a complex data structure

**Description:** CMPLX2R creates a data structure, in which the data is taken from the real part of a complex data structure. The resulting structure is not complex (i.e. has only a single data array, with no ‘real’ and ‘imaginary’ arrays).

**Parameters:**
- **CDATA** (Character) The name of an existing complex data structure. The real data array from this will become the data array of the resulting structure.
- **OUTPUT** (Character) The name of the structure to be created. Its data array will come from the structure specified as CDATA, and it will not have any ‘real’ or ‘imaginary’ arrays. If OUTPUT is the same as CDATA, CDATA will be transformed into a non-complex structure (which means that its imaginary part will be lost); otherwise, a new file is created.

**Source comments:**

CMPLX2R / CMPLX2I / CMPLX2M

Creates a real data structure (i.e. one with just a real data array) as opposed to a complex data structure, (in the Figaro sense of a structure with both real and imaginary data arrays) from a complex data structure.

In the case of CMPLX2R it is the real part of the complex data that forms the data array in the resulting structure.

For CMPLX2I, it is the imaginary part, and for CMPLX2M it is the modulus of the complex data.

**Command parameters:**
- **CDATA** (Character) The name of the input complex structure.
- **OUTPUT** (Character) The name of the resulting structure. This may be the same as CDATA. In either case a new file is created.

**Command keywords:** None


---

D.33 CMPLXADD—Add two complex structures

**Description:** CMPLXADD adds together two complex data structures. These may have any shape and size, so long as they are both the same. A ‘complex data structure’ here
means a structure containing a .Z.REAL, a .Z.IMAGINARY and a .Z.DATA array, all with dimensions that factorise easily so that FFTs can be applied to them.

**Parameters:**

- **CDATA** The first of the images to be summed.
- **CDATA1** The second of the images to be summed.
- **OUTPUT** The name of the resulting structure. This can be the same as CDATA, in which case no new structure is created. Otherwise, OUTPUT will be a new file, with the same structure as CDATA.

**Source comments:**

```plaintext
C M P L X A D D / C M P L X S U B

C M P L X D I V / C M P L X M U L T
```

This routine services the Figaro complex structure arithmetic routines CMPLXADD, CMPLXSUB, CMPLXDIV, and CMPLXMULT. These commands operate on two complex structures, performing element by element arithmetic to produce a new structure. The complex structures may be of any dimensions, so long as they match.

**Command parameters -**

- **CDATA** (Character) The name of the first complex structure.
- **CDATA1** (Character) The name of the second complex structure.
- **OUTPUT** (Character) The name of the resulting structure.

**Command keywords -** None

---

**D.34 CMPLXCONJ- Produce the complex conjugate of a complex structure**

**Description:** CMPLXCONJ takes the complex conjugate of a complex data structure A ‘complex data structure’ here means a structure containing a .Z.REAL, a .Z.IMAGINARY and a .Z.DATA array, all with dimensions that factorise easily so that FFTs can be applied to them.

**Parameters:**

- **CDATA** The name of the complex data structure of which the complex conjugate is to be taken.
- **OUTPUT** The name of the resulting structure. This can be the same as CDATA, in which case no new structure is created. Otherwise, OUTPUT will be a new file, with the same structure as CDATA.

**Source comments:**

```plaintext
C M P L X C O N J
```

This routine produces the complex conjugate of a complex data structure.

**Command parameters -**

- **CDATA** (Character) The name of the first complex structure.
- **OUTPUT** (Character) The name of the resulting structure. This may be the same as CDATA, in which case the operation
D.35 CMPLXDIV - Divide two complex structures

Description: CMPLXDIV divides two complex data structures. These may have any shape and size, so long as they are both the same. A ‘complex data structure’ here means a structure containing a .Z.REAL, a .Z.IMAGINARY and a .Z.DATA array, all with dimensions that factorise easily so that FFTs can be applied to them.

Parameters: CDATA The complex dividend.
           CDATA1 The complex divisor.
           OUTPUT The name of the resulting structure. This can be the same as CDATA, in which case no new structure is created. Otherwise, OUTPUT will be a new file, with the same structure as CDATA.

Source comments:

C M P L X A D D / C M P L X S U B
C M P L X D I V / C M P L X M U L T

This routine services the Figaro complex structure arithmetic routines CMPLXADD, CMPLXSUB, CMPLXDIV, and CMPLXMULT. These commands operate on two complex structures, performing element by element arithmetic to produce a new structure. The complex structures may be of any dimensions, so long as they match.

Command parameters -

CDATA (Character) The name of the first complex structure.
CDATA1 (Character) The name of the second complex structure.
OUTPUT (Character) The name of the resulting structure.

Command keywords - None


D.36 CMPLXFILT - Create a mid-pass filter for complex data

Description: CMPLXFILT generates a complex data structure whose data have the form of a mid-pass filter that can be applied to other complex data of the same size using CMPLXMULT. The filter is specified by a low and high cutoff. If the low cutoff is specified as 0, a low-pass filter is constructed.

Parameters: CDATA CMPLXFILT uses an input complex data structure, specified as CDATA, as a template for the filter it generates. The resulting filter has the same structure (and data dimensions) as the template, only the data arrays differing. If the template data is n-dimensional, an n-dimensional filter will be generated, the cutoff frequencies being the same in each dimension.
LOWCUT The mid-pass filter rises from zero, LOWCUT being the frequency (specified in terms of the Nyquist frequency) at which it reaches a height of \( \exp(-1/2) = 0.6 \). If LOWCUT is specified as zero, a low-pass filter results. The filter is in fact a falling gaussian with a width specified by HICUT which has subtracted from it a rising gaussian with a width specified by LOWCUT.

HICUT HICUT is the element at which the filter drops to a height of \( \exp(-1/2) = 0.6 \). It is specified in terms of the Nyquist frequency.

OUTPUT The name of the resulting filter. If OUTPUT is the same as CDATA, the filter replaces the data originally in CDATA. Otherwise, a new structure is created.

Source comments:

This routine produces a mid-pass complex filter, given a complex structure as a template and low and high cutoff values. The filter is produced by the subtraction of two gaussians. If no low value is specified, the result is a single gaussian low-pass filter.

Command parameters -

CDATA (Character) The name of the template complex structure.
LOWCUT (Numeric) The low cutoff value for the filter. This is specified in terms of the Nyquist frequency. It is the sigma of the low cut gaussian, i.e. the point at which the rising edge of the filter reaches \( \exp(-1/2) \approx 0.6 \).
HICUT (Numeric) The high cutoff value for the filter. This is specified in terms of the Nyquist frequency. It is the sigma of the high cut gaussian, i.e. the point at which the falling edge of the filter reaches \( \exp(-1/2) \approx 0.6 \).
OUTPUT (Character) The name of the resulting structure. This may be the same as CDATA, in which case the operation is performed in situ. Otherwise, a new file is created.

Command keywords - None

Output data -

The resulting complex data have a zero imaginary part, and a real part given by 
\[
F(X) = \exp(-X**2/(2*V**2)) - \exp(-X**2/(2*U**2))
\]
where U and V are the low and high frequency cutoffs respectively. (Note that the actual data generated is symmetrical about the mid point of the data, which is assumed to be the zero frequency - the Figaro function FFT produces data in this form).


D.37 CMPLXMULT-Multiply two complex structures

Description: CMPLXMULT multiplies two complex data structures. These may have any shape and size, so long as they are both the same. A 'complex data structure' here means a structure containing a .Z.REAL, a .Z.IMAGINARY and a .Z.DATA array, all with dimensions that factorise easily so that FFTs can be applied to them.
**Parameters:**

- **CDATA** The first complex data to be multiplied.
- **CDATA1** The second complex data to be multiplied.
- **OUTPUT** The name of the resulting structure. This can be the same as CDATA, in which case no new structure is created. Otherwise, OUTPUT will be a new file, with the same structure as CDATA.

**Source comments:**

This routine services the Figaro complex structure arithmetic routines CMPLXADD, CMPLXSUB, CMPLXDIV, and CMPLXMULT. These commands operate on two complex structures, performing element by element arithmetic to produce a new structure. The complex structures may be of any dimensions, so long as they match.

**Command parameters -**

- **CDATA** (Character) The name of the first complex structure.
- **CDATA1** (Character) The name of the second complex structure.
- **OUTPUT** (Character) The name of the resulting structure.

**Command keywords -** None

---

**D.38 CMPLXSUB-Subtract two complex structures**

**Description:** CMPLXSUB subtracts two complex data structures. These may have any shape and size, so long as they are both the same. A ‘complex data structure’ here means a structure containing a .Z.REAL, a .Z.IMAGINARY and a .Z.DATA array, all with dimensions that factorise easily so that FFTs can be applied to them.

**Parameters:**

- **CDATA** The complex data to be subtracted from.
- **CDATA1** The complex data to subtract from CDATA.
- **OUTPUT** The name of the resulting structure. This can be the same as CDATA, in which case no new structure is created. Otherwise, OUTPUT will be a new file, with the same structure as CDATA.

**Source comments:**

This routine services the Figaro complex structure arithmetic routines CMPLXADD, CMPLXSUB, CMPLXDIV, and CMPLXMULT. These commands operate on two complex structures, performing element by element arithmetic to produce a new structure. The complex structures may be of any dimensions, so long as they match.

**Command parameters -**

- **CDATA** (Character) The name of the first complex structure.
CDATA1 (Character) The name of the second complex structure.
OUTPUT (Character) The name of the resulting structure.

Command keywords - None


D.39 COADD-Form the spectrum which is the mean of the rows in an image

Description: COADD takes a 2-D file as produced by FIGS322 or RCGS2 and combines scans to generate a spectrum with error bars

Parameters: IMAGE The name of a 2-D file (wavelength by scan number) as produced, for example, by FIGS322 or RCGS2.
TSTART The first t-value to be used.
TEND The last t-value to be used.
YSTART The first y-value to be used.
YEND The last y-value to be used.
CUTOFF Values more than CUTOFF times sigma away from the mean value for the spectral point will not be included in the final spectrum.
SPECTRUM The name of the resulting single spectrum produced by collapsing down the image.
NORM NORM=YES causes the data for each cycle to be normalized so that the mean value for each cycle is the same. This gives more reasonable errors when data are taken in the presence of cloud. It should not be used on very faint sources, as the mean level may go negative under these circumstances.

Source comments: A D D

Form a spectrum which is the mean of all the rows in an image or form an image which is the mean of all the planes in a cube. The errors on the result are the standard errors of the mean (i.e. SIGMA/SQRT(N) when N rows or planes are combined). Any error information in the original image or cube is ignored.

An XY image is collapsed along the Y direction to give a spectrum, and an XYT cube is collapsed along the T direction to give an XY image.

Typical uses include the combination of the various cycles of a CGS2 or FIGS observation as output by the FIGS322 or RCGS2 programs, or coadding of CGS4 observations (for this purpose the individual images must be first grown into a cube using GROWXY).

If the NORM keyword is set the errors are calculated after normalizing each row or plane so that the mean value is the same for all rows (planes). This does not effect the output data but generates errors which are determined only by the noise level in the data and are not influenced by any general trend in the data.

If the CUTOFF parameter is specified, points which deviate from the
mean by more than $CUTOFF$ times the standard error for the mean are
excluded from the calculation. The mean is recalculated until no
points exceed the $CUTOFF$ limit. This procedure allows spikes in the
data to be removed.

Command parameters -

'IMAGE' The name of the input 2-D or 3-D file.
'YSTART' (or TSTART) The first Y or T value to use.
'YEND' (or TEND) The last Y or T value to use.
'SPECTRUM' The name of the resulting spectrum or image.
'CUTOFF' The level (in sigma) at which a point will be ignored.

Command keywords -

'NORM' Normalize data to mean level.

Input data -

D.40  COLOUR-Set colour table for image display

Usage:  colour table

Description:  This routine sets the colour table of the image display device. It can either be reset
to a grey scale, or an RGB lookup table from a 3xN image can be used. The lookup table
must have numbers between 0.0 and 1.0.

Parameters:  TABLE  The colour table file to be used. The programme will look for this file in
the default directory and then in the standard Figaro directories. If TABLE is ‘grey’ or
‘gray’ this is trapped and a grey scale table is set up.

IDEV  The name of the imaging device, normally got from a global parameter which was
set with the IDEV command.

Authors:  ks: Keith Shortridge (AAO)
          hme: Horst Meyerdierks (UoE, Starlink)

D.41  COMB-Corrects for S-distortion using continua

Description:  This is a program to correct data for s-distortion by moving data in the cross-
section direction to line it up for a comb of continua spectra. This correction is then applied
to the data itself. A comb dekker is used to produce about ten continuum spectra across
an image (this is done at the telescope). This image is then used by the program: The
program requests two adjacent "teeth" to be marked, it locates the remaining teeth and
follows them along the image (in the channel direction), finding the line centres by fitting
gaussians. The points so obtained are fitted with Chebyshev polynomials (for each tooth),
The intermediate positions are interpolated from these, which are then used to evaluate
the required movement for each data point. The coefficients are written to a file which
may then be read by the program to apply correction to the actual data. Alternatively if QUICK is specified centroids are used rather than Gaussians.

**Parameters:**

- **IMAGE**
  IMAGE = FILE (Read) Name of image for input This should be a file containing continua spectra.

- **ARC_OPTS**
  ARC_OPTS = CHARACTER (Read) Enter arc fit option NEW : set up a new wavelength calibration REPEAT : Iterate on previous calibration. CLONE : CLone a previous calibration. OLD : Correct using previous results

- **OUTPUT**
  OUTPUT = FILE (Write) Name of output file File to contain corrected data.

- **XSTART**
  XSTART = INTEGER (Read) analysis lower limit The data between the limits xstart and xend is extracted and the resultant spectrum is used to locate the lines.

- **XEND**
  XEND = INTEGER (Read) analysis upper limit The data between the limits xstart and xend is extracted and the resultant spectrum is used to locate the lines.

- **XBLOCK**
  XBLOCK = INTEGER (Read) Enter averaging width in channels Each window is of this width (except perhaps the final one).

- **ITERATION**
  ITERATION = INTEGER*2 (Read) New value of iteration

- **LEVEL**
  LEVEL = REAL (Read) Level of edge of tooth

- **ORDER**
  ORDER = INTEGER (Read) order for polynomial fitting This is for the continuity correction of the data. Ideally the arc should have been pre-processed with ARCSDI, so a low order e.g. 2 should be used.

- **MAXLINES**
  MAXLINES = INTEGER (Read) Maximum number of lines to allow room for This must be greater than or equal to the number of lines fitted, so room should be allowed in case any more are to be added later.

- **CLFILE**
  CLFILE = FILE (Read) Name of image for cloning from This should be a file containing an arc spectrum.

- **TOLS**
  TOLS = CHARACTER (Read) For use in batch only

- **KEEP_ITT**
  KEEP_ITT = LOGICAL (Read) keep iteration files?

- **QUICK**
  QUICK = LOGICAL (Read) Centroid rather than fit gaussians?

- **PRFITS**
  PRFITS = LOGICAL (Read) Print out details of fitting

- **PLOTCORR**
  PLOTCORR = LOGICAL (Read) Plot correction?

**Source comments:**
none available

---

**D.42 COMBINE-Combine two spectra, adding with weights according to errors**

**Description:** Combines two spectra, adding them with weights that depend on their errors. Elements with equal errors, for example, will be averaged. COMBINE is intended for use with spectra, but will work on any data that has error values for each data point.

**Parameters:**

- **SPECTRUM**
  The first of the two spectra to be combined.

- **SPECTRUM1**
  The second of the two spectra to be combined.

- **OUTPUT**
  The name of the resulting spectrum. This can be the same as SPECTRUM, in which case no new spectrum will be created.
Source comments:

Combine two spectra or images. If error or variance information is available on the two images a weighted mean of the two is formed. If error information is missing for one or both images a simple average is formed.

Command parameters -

SPECTRUM The name of the structure containing the first spectrum
SPECTRUM1 The name of the structure containing the second spectrum
OUTPUT The name of the result of the operation. This can be the same as for SPECTRUM. If not, a new structure is created, with everything but the data a direct copy of the input.

JAB / AAO 14th July 1986

D.43 COPOBJ-Copy an HDS object

Usage: copobj source object

Description: This routine creates or modifies an object to be a copy of an existing HDS object (in the same or a different container file). The destination object can either be a newly created scalar object or an existing cell of an array. If it is a cell of a structure array, it must be an empty structure.

Parameters: SOURCE The existing HDS object to be copied. Specify beginning with directory and file name in the syntax of the operating system, followed by the dot-separated structure hierarchy. Elements of structure arrays are specified in ordinary brackets ( ).
OBJECT The HDS object to be created or modified. Specify beginning with directory and file name in the syntax of the operating system, followed by the dot-separated structure hierarchy. Elements of structure arrays are specified in ordinary brackets ( ). An array element (cell) cannot be created, but an existing cell can be modified.

Examples: 1. copobj source="file1.dst".Z.DATA object=file2.DATA_ARRAY
Copy the data array from a Figaro DST file into the data array of an NDF. Note that file2.DATA_ARRAY must not exist beforehand, and that file2 without DATA_ARRAY is not a legal NDF. So probably this would be the first action after creation of the empty HDS file "file2".

See also: FIGARO:[CREOBJ],[DELOBJ],[RENOBJ],[SETOBJ],[KAPPA],[ERASE]

Authors: KS: Keith Shortridge (AAO)
HME: Horst Meyerdierks (UoE, Starlink)
JFL: John Lightfoot (ROE)
D.44  CORREL-Correlate two or three data sets.

Usage:  correl inlist out logfil

Description: This routine correlates two or three data sets. Either pair is subjected to a linear fit and the third data set is subjected to a two-parameter linear fit (i.e. regarded as a linear function of the first and second data sets). Each data set may be an NDF section. All must have the same dimensions.

Parameters:  INFO  INFO = _LOGICAL (Read) If false, the routine will issue only error messages and no informational messages. [YES]

VARUSE  VARUSE = _LOGICAL (Read) If false, input variances are ignored. [YES]

INLIST  INLIST = LITERAL (Read) The group of input NDFs. Two or three NDFs must be specified. A complicated INLIST could look something like

\[ M_51(25:35,-23.0,-24.0),M101,^\text{LISTFILE.LIS} \]

This example NDF group specification consists of
- one identified NDF from which a subset is to be taken,
- one identified NDF,
- an indirection to an ASCII file containing more NDF group specifications. That file may have comment lines and in-line comments, which are recognised as beginning with a hash (#).

OUT  OUT = FILENAME (Read) The ASCII output file where the data points are written into a table. A new file will be opened. No file will be opened, if "!" is entered. The table in OUT is without any information else than the values from the 1st, 2nd, 3rd data array and errors from the 1st, 2nd, 3rd variance array in that order. [!]  

LOGFIL  LOGFIL = FILENAME (Read) The ASCII log file where fit results are written to. This will be opened for append, if such a file exists.

D.45  COSBELL-Create data that goes to zero at the edges in a cosine bell

Description: COSBELL generates a spectrum or image which goes to zero at the edges, is unity over its central section, and which rises from 0 to 1 in a cosine bell over a specified outer percentage of its range.

Parameters:  SPECTRUM  A template data file to use when constructing the cosine bell data. The output data will be a copy of the template, except for the main data array. The data may be 1- or 2-dimensional.

BELLPC  The data generated by COSBELL is 1.0 in the centre, 0 at the edges, and rises from 0 to 1 over a range specified as BELLPC % of the total range.

OUTPUT  The name of the resulting datafile. If this is the same as the template SPECTRUM, the data in the template will be lost, being replaced by the cosine bell data. Otherwise, a new file will be created.

Source comments:  B E L L

Given a template data file, COSBELL creates a data file that is the same as the template but in which the data is a cosine bell filter. This can then be applied to the original data (or to
other data with the same dimensions) using IMULT.

Command parameters -

SPECTRUM (Character) The name of the structure containing the template data.

BELLPC (Numeric) The percentage of the data that is to be covered by the rising (or falling) part of the cosine bell.

OUTPUT (Character) The name of the result of the operation. This can be the same as for SPECTRUM. If not, a new structure is created, with everything but the data a direct copy of the input.

D.46 COSREJ-Reject cosmic rays from a set of supposedly identical spectra

Description: Given an image whose cross-sections are a set of spectra of the same object all with exactly the same wavelength to pixel mapping (in other words a set of spectra that should be identical, other than for signal to noise effects and possible differences in their total counts due to differing exposure times etc), COSREJ attempts to remove any cosmic rays or other obvious features that might be contaminating some (but not the majority) of the spectra.

Parameters: IMAGE A 2-D image in which all the cross sections are a set of similar spectra. Note that each pixel of each spectrum is compared with the corresponding pixels in the other spectra, so they should be properly aligned - probably already scrunched.

XSTART COSREJ scales up each spectrum to the same mean value. To do this it calculates the mean for each spectrum between the limits specified for XSTART and XEND (which are given in terms of the X-axis values, not necessarily in pixels). Normally, you would use the whole spectrum, but there may be end effects you’d prefer to avoid.

XEND COSREJ scales up each spectrum to the same mean value. To do this it calculates the mean for each spectrum between the limits specified for XSTART and XEND (which are given in terms of the X-axis values, not necessarily in pixels). Normally, you would use the whole spectrum, but there may be end effects you’d prefer to avoid.

CRSIG Each pixel in each spectrum is compared with the equivalent pixels in the other spectra, and any that differ from the mean of the other pixels by more than a given factor (CRSIG) times the standard deviation of the other pixels are rejected. This is repeated until either only two pixels are left, or until no pixels are rejected in a pass through the remaining pixels. The rejected pixels are set to the mean value of the remaining unrejected pixels. If you’re only trying to get rid of gross effects like cosmic rays, CRSIG values around 10 should be satisfactory. The value is unlikely to be critical.

WMEAN COSREJ can create a ‘spectrum’ with one element for each spectrum in the original image, where each element is the mean value used for that spectrum.
This might possibly be used to scale the resulting image using, for example, ISYDIV. Such a spectrum is produced only if WMEAN is set.

**MSPECT** If WMEAN is set, MSPECT is the name of the ‘mean spectrum’ generated. This is a file containing nothing but a data array containing the mean values for the spectra in the image.

**OUTPUT** The name of the resulting image. The data produced in OUTPUT is the same as that for IMAGE, except that the cosmic rays have been processed out. If OUTPUT and IMAGE are the same, the operation is performed in situ.

**See also:** FIGARO: BCLEAN, CLEAN, MEDFILT, MEDSKY, TIPPEX
KAPPA: FFCLEAN, CHPIX, FILLBAD, GLITCH, MEDIAN, MSTATS, ZAPLIN

**Source comments:** R E J

**Name:** COSREJ

**Function:** Remove cosmic rays from a set of similar spectra.

**Description:**
Given an image whose cross-sections are a set of spectra of the same object all with exactly the same wavelength to pixel mapping (in other words a set of spectra that should be identical, other than for signal to noise effects and possible differences in their total counts due to differing exposure times etc), this routine attempts to remove any cosmic rays or other obvious features that might be contaminating some (but not the majority) of the spectra. First, the mean value for each spectrum over a specified range is calculated and this is used to reduce each spectrum to the same mean value. Each pixel in each spectrum is compared with the equivalent pixels in the other spectra, and any that differ from the mean of the other pixels by more than a given factor (the CRSIG parameter) times the standard deviation of the other pixels are rejected. This is repeated until either only two pixels are left, or until no pixels are rejected in a pass through the remaining pixels. The rejected pixels are set to the mean value of the remaining unrejected pixels. Finally, the spectra are rescaled to their original mean values. If requested, the program can create a ‘spectrum’ with one element for each spectrum in the original image, where each element is the mean value used for that spectrum. This might possibly be used to scale the resulting image using, for example, ISYDIV.

**Command parameters:**

- **IMAGE** The name of the image in which the spectra are held.
- **XSTART** The first x-value of the range to be used to calculate the mean value for each spectrum.
- **XEND** The last x-value of the range to be used to calculate the mean value for each spectrum.
- **CRSIG** The cutoff sigma value to be used.
- **MSPECT** The name of the mean spectrum produced, if WMEAN is yes.
- **OUTPUT** The name of the resulting image with the cosmic rays removed.
Command keywords:

WMEAN Yes if a spectrum of mean values is to be produced.

Error array handling: Ignored.

Data quality / flagged value handling:
Not explicitly performed. Relies on standard DSA processing.

Files used:
BADPIX.DAT Contains a list of the cosmic rays removed from the data.

8th Sept 1987. Original version DJA / AAO

D.47 CREOBJ-Create a data object or file

Usage: creobj type dims object

Description: This routine creates an HDS object, primitive or structure, scalar or array. In theory it is possible to build up a complete NDF or Figaro DST data file. This is not recommended because the risk of creating an illegal hierarchy of HDS structures - i.e. one not accepted by KAPPA, Figaro, etc. - is very high. This routine is intended only for minor repairs to such files, or in emergencies to create a very simple, minimal, data file.

Parameters:

TYPE The HDS type of the object to be created. Figaro users note that this is something like '_REAL', '_DOUBLE', '_INTEGER', '_WORD' etc. Anything which is not such a primitive HDS type will cause a structure to be created. The type specified here will then be used as that structure's type. ['_REAL']

DIMS The dimensions of the object to be created, i.e. the size of the array along each axis. The number of positive integers specified indicates the dimensionality of the object created. To create a scalar object enter zero, a single zero will do. [0]

OBJECT The object to be created. Specify beginning with directory and file name in the syntax of the operating system, followed by the dot-separated structure hierarchy. Elements of structure arrays are specified in ordinary brackets (). An array element cannot be created.

Examples:

1. creobj type=NDF dims=0 object=file
   This will create an empty HDS file. The top level structure is of type "NDF", which has little consequence.

2. creobj type=ARRAY dims=0 object=file.DATA_ARRAY
   This will create the scalar structure DATA_ARRAY in the top level structure of the file "file". The structure type is "ARRAY", which has special meaning in the Starlink Data Format.

3. creobj type=_REAL dims=[20,30] object=file.DATA_ARRAY.DATA
   This will create a two-dimensional array of _REAL numbers called DATA and situated in file.DATA_ARRAY. The size of the new array is 20 by 30 numbers.

4. creobj type=AXIS dims=2 object=file.AXIS
   This will create a one-dimensional array of AXIS structures
called AXIS and situated underneath the top level of "file".

**Authors:** KS: Keith Shortridge (AAO)  
HME: Horst Meyerdierks (UoE, Starlink)

**D.48 CRIGAUSS-**Creates a file with a profile of 1 to 5 gaussians

**Description:** The profiles are evaluated and copied to each cross-section. This is really intended for testing software.

**Parameters:** IMAGE IMAGE = FILE (Read) Name of image to be created  
YDIM YDIM = INTEGER (Read) Y dimension of data  
XDIM XDIM = INTEGER (Read) X dimension of data  
XSTART XSTART = REAL (Read) First X value  
XEND XEND = REAL (Read) Last X value  
WIDTH1 WIDTH1 = REAL (Read) WIDTH1 of gaussian (fwhm)  
CENTRE1 CENTRE1 = REAL (Read) CENTRE1 of gaussian  
HEIGHT1 HEIGHT1 = REAL (Read) HEIGHT1 of gaussian  
WIDTH2 WIDTH2 = REAL (Read) WIDTH2 of gaussian (fwhm)  
CENTRE2 CENTRE2 = REAL (Read) CENTRE2 of gaussian  
HEIGHT2 HEIGHT2 = REAL (Read) HEIGHT2 of gaussian  
WIDTH3 WIDTH3 = REAL (Read) WIDTH3 of gaussian (fwhm)  
CENTRE3 CENTRE3 = REAL (Read) CENTRE3 of gaussian  
HEIGHT3 HEIGHT3 = REAL (Read) HEIGHT3 of gaussian  
WIDTH4 WIDTH4 = REAL (Read) WIDTH4 of gaussian (fwhm)  
CENTRE4 CENTRE4 = REAL (Read) CENTRE4 of gaussian  
HEIGHT4 HEIGHT4 = REAL (Read) HEIGHT4 of gaussian  
WIDTH5 WIDTH5 = REAL (Read) WIDTH5 of gaussian (fwhm)  
CENTRE5 CENTRE5 = REAL (Read) CENTRE5 of gaussian  
HEIGHT5 HEIGHT5 = REAL (Read) HEIGHT5 of gaussian  
BASE BASE = REAL (Read) BASE for gaussian  
NCOMP NCOMP = INTEGER (Read) NCOMP Number of components

**Source comments:** available

**D.49 CSCAN-**Plot array of profiles from a 3D array

**Description:** This displays line profiles from a sorted data cube (i.e. the first dimension is that of wavelength).

**Parameters:** CUBE CUBE = FILE (Read) Name of CUBE for input  
YSTART YSTART = REAL (Read) display lower limit  
YEND YEND = REAL (Read) display upper limit  
TSTART TSTART = INTEGER (Read) display lower limit  
TEND TEND = INTEGER (Read) display upper limit  
HARDCOPY HARDCOPY = LOGICAL (Read) use hard graphics device for display

**Source comments:** available
D.50  CSET-Interactively set regions of a spectrum to a constant value

Description:  CSET is an interactive program that takes a displayed spectrum and allows
the user to select regions that are to be set to constant values. This can be used, for
example, to set regions in a B star calibration spectrum to 1., so that they will have no
effect when used with BSMULT.

Parameters: VALUE  CSET lets you select regions of an already displayed spectrum,
which are then set to a constant value. This value can be changed if required, but
this is the initial setting for it.

OUTPUT  CSET generates an output file that is essentially the data from the dis-
played spectrum, with certain regions set to constant values. OUTPUT is the
name of the resulting spectrum.

QUIT  Used to confirm quitting area selection.

See also: FIGARO: ICSET, NCSET, TIPPEX
          KAPPA: CHPIX, FILLBAD, SEGMENT, NOMAGIC, RIFT, SETMAGIC, ZAPLIN

Source comments:  Figaro function to set large interactively selected regions
of a spectrum to a constant value. This is intended mainly
for use in generating mask spectra, or modifying calibration
spectra such as those used by BSMULT. CSET assumes that a
spectrum has already been displayed by SPLOT, and will generate
a new data structure based on the spectrum displayed, with
only the data changed.

Command parameters -

VALUE  (Numeric) The value to use for the selected regions

OUTPUT  (Character) The name of the output file to
be created. If this is the same as the displayed
spectrum, the data in the displayed spectrum will
be modified.

Command keywords -

QUIT  Used to confirm quitting area selection.

User variables used -  (">" input, "<" output)

(> TVFILE  The name of the displayed spectrum
(> TVXST  The first displayed x-value
(> TXXEN  The last displayed x-value
(> TVHIGH  The maximum displayed y-value
(> TVLOW  The minimum displayed y-value
(> TVCOLOR The GRPLOT code for the plot colour
(> SOFT  The device/type string defining the display device

KS / CIT 11th April 1984
D.51 CSPIKE-Create calibration spiketrum given spiketrum and standard spectrum

Description: Given a ‘spiketrum’ generated by GSPIKE from a table of flux densities at given wavelengths for a particular standard star together with a spectrum of that same object, CSPIKE calculates the instrumental response at the various tabulated points. Note - this routine is intended for standards where the tables give the average measured flux density over a given wavelength range for each point, rather than calculated continuum values. CSPIKE will not accept a spiketrum that has data in magnitude units.

Parameters: SPIKETRUM A spiketrum (see HELP FIGARO TECHNIQUES SPIKETRA for details) generated from a table of measured average flux densities over given wavelength ranges versus the central wavelengths. The data should not be in magnitude units.

SPECTRUM An observed spectrum of the object whose tabulated fluxes were used to generate SPIKETRUM. Note that CSPIKE does not insist on the spectrum’s having been scrunched, but things are usually easier if it has been.

OUTPUT CSPIKE generates a spiketrum whose non-zero values give the instrumental response in "units per counts per second per angstrom", where "units" are the units used by the input spiketrum.

Source comments: I K E

Generates a calibration 'spiketrum', given an observation of a standard star and a spiketrum giving the tabulated flux values for that star. The calibration spiketrum has points giving the instrumental response calculated at the points given by the spikes in the flux spiketrum. A calibration spectrum can then be generated by interpolating between the points of the calibration spectrum.

Command Parameters

SPIKETRUM (Character) The tabulated flux spiketrum. Note: this should include the BANDWIDTH data object - a spiketrum that does not is probably not appropriate for this function. Also note that CSPIKE does not work with data in magnitude units.

SPECTRUM (Character) The observation of the standard star. Note that this should include an exposure time data object. If it does not, a time of 1 sec will be assumed, and the calibration will only be relative. Both SPECTRUM and SPIKETRUM should contain an AXIS(1) data array, giving the wavelength values. These should normally be exactly the same, although this is not essential.

OUTPUT (Character) The resulting spiketrum of calibration points.

Command keywords - None

User variables used - None
D.52 CSUB-Subtracts a continuum from 2 dimensional data

Description: A polynomial is fitted to the continuum and this is subtracted. As with VIG, lines can be excluded from the polynomial fitting. CSUB stores the polynomial fitting coefficients in the actual data file.

Parameters: IMAGE IMAGE = FILE (Read) Name of image for input
OLD OLD = LOGICAL (Read) Old coefficients are to be used for correction
OUTPUT OUTPUT = FILE (Write) Name of output file OUTPUT is the name of the resulting spectrum. If OUTPUT is the same as INPUT the data in the input file will be modified in situ. Otherwise a new file will be created.

Source comments: available

D.53 CUBE2LONG-Takes a longslit spectrum from a non-sorted TAURUS cube

Description: This uses cubic spline interpolation to create a 2-d file from a 3-d file, given a location, angle and length.

Parameters: CUBE CUBE = FILE (Read) Sorted TAURUS cube
XPOINT XPOINT = REAL (Read) X point anywhere on slit
YPOINT YPOINT = REAL (Read) Y point anywhere on slit
ANGLE ANGLE = REAL (Read) Position angle (degrees)
OUTPUT OUTPUT = FILE (Write) Output longslit spectrum

Source comments: available

D.54 DELOBJ-Delete a data object or a file

Usage: delobj object

Description: This routine deletes an HDS object (structure or primitive, scalar or array) in an HDS file.

Parameters: OBJECT The object to be deleted. Specify beginning with directory and file name in the syntax of the operating system, followed by the dot-separated structure hierarchy. Elements of structure arrays are specified in ordinary brackets () . An array element cannot be deleted.

Examples: 1. delobj file.axis(2).units
   The file in question is in the current working directory and has the standard extension ".sdf". The deleted structure is the UNITS string in the 2nd element of the structure array AXIS. Note that it would be impossible to delete AXIS(2), but one could delete AXIS as a whole.

   2. delobj @"/home/resun02/mynname/data/file.dst".z.label
   Here the file is specified with its complete Unix directory and
with its non-standard extension ".dst". The deleted structure is the LABEL within the Z structure.

See also: FIGARO: CREOBJ, COPOBJ, RENOBJ, SETOBJ
KAPPA: ERASE

Authors: KS: Keith Shortridge (AAO)
HME: Horst Meyerdierks (UoE, Starlink)

D.55 DVDPLOT-Plot the data in one file against the data in another

Usage: dvdplot image image2 xlow xhigh low high autoscale

Description: DVDPLOT (Data Versus Data PLOT) plots the data from the main array in one structure against the data in the corresponding elements of the main array in another structure. It was first written to help with linearity tests (plotting data from an image with high count rates against a similar image with lower count rates). The Y value for each point is the pixel value from IMAGE and the X value is the value of the corresponding value from IMAGE2. The images must have identical dimensions.

Parameters: IMAGE Y-value image.
IMAGE2 X-value image.
XLOW XLOW and XHIGH specify the limits in X of the plot. They are values in the same range as the data range as the main array in IMAGE2. If WHOLE is set, XLOW is set to the lowest data value in IMAGE2.
XHIGH XLOW and XHIGH specify the limits in X of the plot. They are values in the same range as the data range as the main array in IMAGE2. If WHOLE is set, XHIGH is set to the highest data value in IMAGE2.
LOW LOW and HIGH specify the limits in Y of the plot. They are values in the same range as the data range as the main array in IMAGE. If AUTOSCALE is set, LOW is set to the lowest data value in IMAGE for which the corresponding IMAGE2 pixel is in the range XLOW..XHIGH.
HIGH LOW and HIGH specify the limits in Y of the plot. They are values in the same range as the data range as the main array in IMAGE. If AUTOSCALE is set, HIGH is set to the highest data value in IMAGE for which the corresponding IMAGE2 pixel is in the range XLOW..XHIGH.
WHOLE WHOLE has the effect of setting XLOW and XHIGH (the x-limits of the plot) to the extreme data values in IMAGE2. The result is that the plot range in X is such as to cover all the pixels in the image.
AUTOSCALE The XLOW and XHIGH parameters (or, alternatively, the WHOLE keyword) select the X-range of the plot and hence the pixels that may be plotted. The LOW and HIGH parameters then limit the Y-range of the plot. The AUTOSCALE keyword has the effect of setting LOW and HIGH to the extreme values of the pixels delimited by XLOW and XHIGH. Which is a complicated way of saying it scales the plot so all the points fit in it.
HARDCOPY  Normally, DVDPLOT produces a plot on the current soft plot device - selected using the SOFT command. If the HARDCOPY keyword is set, the plot is made on the current hard copy plot device - selected using the HARD command. Normally, only a plot file is produced and this must be explicitly sent to the hardcopy device once DVIPLOT has finished.

Source comments:  

Routine name:  
DVDPLOT

Function:  
Plots the data in one data array against the data in another.

Description:  
DVDPLOT (Data Versus Data PLOT) plots the data in the main data array in one Figaro structure against the corresponding data elements of the main data array in another structure. This was originally written to help with linearity tests (where data in an image taken at a low data rate could be plotted against one taken at a higher data rate), but may have other applications.

Usage:  
DVDPLOT IMage IMAGE2 XLow XHigh LOw HIgh AUtoscale

Parameters:  
IMAGE  (Character) The name of the first structure. It is this structure whose data is plotted against the data in IMAGE2, so its data values form the Y values of the plotted points.
IMAGE2  (Character) The name of the second structure. Its data values form the X values of the plotted points.
XLOW  (Numeric) The low end of the data range plotted in X (i.e. the lower limit for the data in IMAGE2).
XHIGH  (Numeric) The high end of the data range plotted in X (i.e. the upper limit for the data in IMAGE2).
LOW  (Numeric) The low end of the data range plotted in Y (i.e. the lower limit for the data in IMAGE).
HIGH  (Numeric) The high end of the data range plotted in Y (i.e. the upper limit for the data in IMAGE).

Keywords:  
WHOLE  If set, XLOW and XHIGH will be set to the limits of the data in IMAGE2.
AUTOSCALE  If set, LOW and HIGH will be set to the limits of the data in IMAGE.
HARDCOPY  If set, a hard copy plot will be produced.

User variables used:  
HARD  (Character) PGPLOT specification for hardcopy plot device
SOFT  (Character) PGPLOT specification for softcopy plot device

Error information:  Ignored.

Quality information:  Handled using flagged values.
D.56  **ECHARC-Wavelength calibrate an echelle arc**

**Description:** Each invocation of ECHARC produces a file arlines.ech in the working directory. This file must be renamed or deleted before re-invoking ECHARC.

**Parameters:**

- **IMAGE** The arc data. This should be a .dst file with a two dimensional .z.data component (pixels,orders). ECHARC assumes there is a .y.data component giving order numbers "m" (such as produced as output from the command ECHTRACT). If there is a .x.data component the information it contains will be used during the program, although usually the .x.data will simply be pixel number.

- **ARCTYPE** The type of arc that was used - e.g. HELIUM, NEON, etc. ARC will look for a file called ARCTYPE.ARC which should hold the line list for the arc.

- **INTERACTIVE** Number of orders to fit interactively.

- **ORDERS** The array of INTERACTIVE order numbers to be fit.

- **SIGMA** Arc line half width in pixels.

- **ORDERFIT** Polynomial order for 1st fit

- **PREVIOUS** If set, ARC will read in the line list from the previous session as a starting point.

- **MONITOR** Monitor ECHARC autofitting on plot device?

- **ARFILE** The name of the list file from which the previous fit is to be read. Only used if PREVIOUS is set, Note that the output is always written to ARLINES.ECH. Default extension is .ECH

- **DOWAVES** Write wavelength information to separate file?

- **WAVES** An output image containing the fitted wavelengths from this ECHARC solution as image data (not as axis data). This is created only if DOWAVES is set.

- **OUTPUT** The name of the output file that combines the input image data with the fitted wavelengths as axis data. This is created only if DOWAVES is NO.

- **CONTINUE** At this stage in ECHARC you can:
  - Continue - go to the next order to be fitted interactively. Repeat - return to working on the order just completed. Start - back to square one. Quit - quit prematurely.

- **ORDPPAG** The number of sub-orders to be plotted per page in the hard copy line atlas.

- **DISNCHAN** Length of displayed sections.

- **MOVETOX** New plot centre x value.

- **CMD** At this stage in ECHARC you have the following options available:
  - Fit - Repeat the fit. O_fit - Change the order of the fit. Disp - Display the deviation of the fit from a linear fit. This shows the trend of the fit, and the errors in each line. Edit - Delete or change the wavelength of one or more of the selected lines, without returning to the cursor selection. Reselect - Return to selection using the cursor. Continue - Start to quit this order. Print - Prints a copy of the fit (what ARLINES.LIS would look like if you were to exit now). Auto - Starting from your current fit and arc line list, ECHARC looks for additional line in the arc
at wavelengths given in the line list and adds any it finds to the identified line tables. Xauto - Deletes all the lines found by 'Auto'. Modify - Allows you some control over the Autofit parameters. Help - (or ?) Display this information. The first letter of each command is sufficient.

**LINENO** Number of line to be edited.

**WAVELEN** Wavelength specification.

**CHFACT** The autofit algorithm is parameterised as follows-
It takes each pixel in turn. If that pixel is more than CHFACT times the current sigma value from any line already found, it uses that pixel as the starting point for a line search. If anything resembling a line can be found, it calculates its wavelength and looks in the line tables for a line close to that wavelength. A line is accepted if the discrepancy between calculated and tabulated wavelength is less than SIGFACT times the current RMS value. This means that the criterion for accepting new lines is based on how their wavelength discrepancies compare with those for the lines that have already been accepted. SIGFACT is the more important parameter.

**SIGFACT** The autofit algorithm is parameterised as follows-
It takes each pixel in turn. If that pixel is more than CHFACT times the current sigma value from any line already found, it uses that pixel as the starting point for a line search. If anything resembling a line can be found, it calculates its wavelength and looks in the line tables for a line close to that wavelength. A line is accepted if the discrepancy between calculated and tabulated wavelength is less than SIGFACT times the current RMS value. This means that the criterion for accepting new lines is based on how their wavelength discrepancies compare with those for the lines that have already been accepted. SIGFACT is the more important parameter.

**HLINEMAP** If set, a map of line locations is plotted as a hard copy.

**HATLAS** If set, an atlas of lines is plotted as a hard copy.

**ANALYSIS** If set, a detailed line-by-line analysis of the arc fit is written to the file "echarc.lis".

**HARDARC** If set, the output spectrum is plotted in a hard copy.

**HARDISP** If set, the dispersion curve is plotted in a hard copy.

**QUITSEL** Used to confirm quitting line selection.

**LINEOK** Used to confirm a choice of line for deletion, editing etc.

**RESOLVE** Used to decide what to do if a line is used twice.

---

Source comments:

A R C

(Version 1.0, 18-NOV-1987 ff.)

(Version 1.5, 08-DEC-1987 ff.)

This substantially revised version of ECHARC performs the 1-D ARC process on 3-30 orders of a collapsed echelle image, and then automatically detects lines and performs fits to all the remaining orders. The output from the program is a complete listing of all lines found (ARLINES.ECH) and an output image with WAVES.Z.DATA containing the fitted wavelengths. One can then use ICMULT and IADD to compute a weighted average of two or more such output fits, and then ECHXREBIN to rebin the data onto a constant "Meff * Lambda" .X.DATA scale.
Command parameters -

**IMAGE**  
The arc data. This should be a two-dimensional image. ECHARC assumes there is a y axis giving order numbers "m". If there is an x axis component the information it contains will be used during the program, although usually the x data will simply be pixel number.

**ARCTYPE**  
The type of arc that was used - e.g. HELIUM, NEON, etc. ARC will look for a file called ARCTYPE.ARC which should hold the line list for the arc.

**INTERACTIVE**  
The number of orders to be fit interactively.

**ORDERS**  
The array of INTERACTIVE order numbers to be fit.

**ORDERFIT**  
The initial order of the polynomial fit.

**SIGMA**  
The initial value for the line width.

**ARFILE**  
The name of the list file from which the previous fit is to be read. Only used if PREVIOUS is set. Note that the output is always written to ARLINES.ECH. Default extension is .ECH

**WAVES**  
An output image containing the fitted wavelengths from this ECHARC solution as image data (not as axis data). This is created only if DOWAVES is set.

**OUTPUT**  
The name of the output file that combines the input image data with the fitted wavelengths as axis data. This is created only if DOWAVES is no.

**CONTINUE**  
Command after an order is done with.

**ORDPPAG**  
The number of sub-orders to be plotted per page in the hard copy line atlas.

**DISNCHAN**  
Length of displayed sections.

**MOVETOX**  
New plot centre x value.

**CMD**  
Command in main menu.

**LINENO**  
Number of line to be edited.

**WAVELEN**  
Wavelength specification.

**CHFACT**

**SIGFACT**

Command keywords -

**PREVIOUS**  
If set, ARC will read in the line list from the previous session as a starting point.

**DOWAVES**  
If set, the fitted wavelengths will be stored as image data in a separate file. Otherwise the fitted wavelengths will be stored as axis data along with the input image.

**HLINEMAP**  
If set, a map of line locations is plotted as a hard copy.

**HATLAS**  
If set, an atlas of lines is plotted as a hard copy.

**ANALYSIS**  
If set, a detailed line-by-line analysis of the arc fit is written to the file "echarc.lis".

**HARDARC**  
If set, the output spectrum is plotted in a hard copy.

**HARDISP**  
If set, the dispersion curve is plotted in a
QUITSEL Used to confirm quitting line selection.
LINEOK Used to confirm a choice of line for deletion, editing etc.
RESOLVE Used to decide what to do if a line is used twice.

User variables -

(>) SOFT (Char) The device/type to be used for graphics soft plots. See the SOFT command for details.
The device must support a cursor.
(>) HARD (Char) The device/type for graphics hard plots.

Input -

As named by ARFILE May use the lines from a previous run. If so these are read from the previous run's output file. See below.

Output -

ARLINES.ECH File containing the lines used in the final fit.
Format is as follows -
Number of lines used in fit (I5)
1 blank record, then one header record.
Then one record for each line, giving order, channel number, wavelength, calculated wavelength, wavelength discrepancy, line number and auto flag (I3,4F13.4,I7,A4)
The auto flag is either " (A)" for a single order Auto fit, " (E)" for complete echelle order auto fit, or is a blank string for lines explicitly identified by user.
Then one blank record, then a record giving the RMS error and the value of SIGMA used (12X,F10.2,19X,F5.2)
Then one blank record, then one record giving the number of coefficients of the fit (15X,I3).

Functions / subroutines used -

ECH_ARINTR (FIGARO) Plots an order section by section and allows user to identify lines, fit a polynomial to those lines, and repeat. When an order is finished, the lines identified within it are written to the file ARLINES.ECH in case of problems during the fit to the next order.

ECH_ARGETL (FIGARO) Reads order, channel, wavelength, etc., information from an existing ARLINES.ECH -format file to use as a starting point for fits to the current order.

Originally ARC : KS / CIT 13 Jun 1984
Stolen & Modified --> ECHARC0: JKM / CIT 9 Dec 1986
Modified: --> ECHARC: v. 1.0 JKM / ESO 18 Nov 1987
Modified: --> ECHARC: v. 1.5 JKM / ESO 8 Dec 1987
D.57  ECHFIND-Locate spectra in echelle data

Description:  Find orders within an echelle image and optionally write a mask image that can be used for quick-look extraction of orders from a raw echelle image.

Parameters:  IMAGE  Name of echelle image in which to search.
  PERISCOPE  Is the periscope fitted?
  YSTART  Y value to start search for orders.
  YEND  Y value to stop search for orders.
  MSTART  Number of the first order in range.
  MDELTA  Order number increment (-1 or +1).
  SDIST  Write output file SDIST.DAT in SDIST format?
  OUTFILE  Name of output report file.
  THRESH  Threshold above which orders are deemed to exist.
  MINHW  Minimum half-width of orders.
  DOMASK  Create an output mask image?
  OUTPUT  Name of mask image showing order positions.

Source comments:

Program name:
  ECHFIND

Function:
  Find orders within an echelle image and optionally write a mask image that can be used for quick-look extraction of orders from a raw echelle image.

Description:
  Note: This program is believed to work, but it has not been as extensively used and tested as has the ICUR/SDIST method. ICUR/SDIST is believed to be a superior if slightly less convenient way of locating and tracking orders. Having said this, you are welcome to try this program!

  The program can be run in several different ways. The SDIST keyword controls whether an SDIST.DAT file (which can later be used by CDIST) is created and the DOMASK keyword controls whether a mask image (that can later be applied by MASKEXT) is created.

  The program locates the orders by taking a vertical cut (i.e. in the cross-dispersion direction) through the data (averaging 7 columns) and then searches for peaks occurring above a user-specified threshold. Unfortunately this threshold has to be a constant and this, plus knowing a sensible value to give for it, is one of the major limitations of the program.

  Having located the orders, they are tracked using a method that is a combination of edge detection and centroiding. Little of the feedback and control that is available with SDIST is available and this is another major problem.
Having tracked the orders, the SDIST.DAT file is written if requested. If an SDIST.DAT file is not required, a more user-readable listing file is written. Finally, the mask image is written if requested. The values in the mask are set to be zero if that pixel in the mask does not lie in an order and to a number derived from the order number otherwise (see below). It is guaranteed that every order is extracted using the same number of rows, but of course the position of these rows may vary along an order so one can expect visible jumps in the extracted data, especially if too few rows are extracted to take all the data from the object.

The PERISCOPE keyword (see below) determines whether each order has two separate parts (corresponding to object and sky and due to the special periscope that samples object and sky at a wide spacing and brings them together on the slit) or one part (corresponding simply to the slit). The data values in the mask are $10 \times (\text{true order number}) + (\text{sub-order number})$ where the sub-order number is 0 if there is no periscope fitted, 1 if this is the first part of an order and 2 if this is the second part of the order. The "first" and "second" parts of an order are defined so that the actual data values in the mask are monotonic along a vertical slice through it, i.e. they might go 412, 411, 402, 401 if the periscope is fitted and they might go 410, 400 if it is not fitted.

If PERISCOPE is NO, then unlike in ECHMASK, the user has no option of splitting the data in an order into object and sky. There is room for enhancement here.

Parameters:

- **IMAGE** (File) The name of the raw echelle image.
- **YSTART** (Integer) The starting and ending Y positions to search for orders. Default entire image.
- **YEND** (Integer) search for orders. Default entire image.
- **PERISCOPE** (Keyword) Whether or not the periscope is fitted. Default YES.
- **MSTART** (Integer) The order number of the first "spectrum" in the coefficient file. Default 1.
- **MDELTA** (Integer) +1 if order numbers increase as "spectrum number" increased, -1 otherwise. Default -1.
- **SDIST** (Keyword) Whether to write an SDIST.DAT file. Default NO.
- **OUTFILE** (Character) If SDIST is NO, the name of the listing file.
- **THRESH** (Real) The threshold above which peaks in the profile across the orders must lie in order to be considered as order peaks. Default 1000.
- **MINHW** (Integer) The half width that is used for the median filter that is passed through the profiles to remove rogue data before looking for orders. Default 5.
- **DOMASK** (Keyword) Whether to write a mask image. Default
AUTHORS: William Lupton, AAO

D.58 ECHMASK-Produce an extraction mask from an SDIST analysis

**Description:** The program reads a file that identifies where the orders are in the image and sets the values in the mask to be zero if that pixel in the mask does not lie in an order and to a number derived from the order number otherwise (see below). It is guaranteed that every order is extracted using the same number of rows, but of course the position of these rows may vary along an order so one can expect visible jumps in the extracted data, especially if too few rows are extracted to take all the data from the object.

The coefficient file will normally have been written by SDIST and if so must have been written by the version of SDIST that was modified to support ECHMASK. The PERISCOPE keyword (see below) determines whether each order has two separate parts (corresponding to object and sky and due to the special periscope that samples object and sky at a wide spacing and brings them together on the slit) or one part corresponding simply to the slit). The data values in the mask are \(10 \times \text{(true order number)} + \text{(sub-order number)}\) where the sub-order number is 0 if there is no periscope fitted, 1 if this is the first part of an order and 2 if this is the second part of the order. The "first" and "second" parts of an order are defined so that the actual data values in the mask are monotonic along a vertical slice through it, i.e. they might go 412, 411, 402, 401 if the periscope is fitted and they might go 410, 400 if it is not fitted.

If PERISCOPE is NO then the user has the option of splitting the data in an order into object and up to two separate regions of sky. The object is assigned sub-order 1 and the sky is assigned sub-order 2. Note that this assignment may differ from when PERISCOPE is yes, since in that case there is no guarantee that sub-order 1 is object - it may be sky! There may be room for enhancement here.

**Parameters:**

- **COFILE** The name of the file that contains details of where the orders lie within the raw input image. It will normally have been created by SDIST. Note that it is necessary to use a version of SDIST that has been specially modified to write details of the dimensions of the input image and of the widths of the orders.

- **PERISCOPE** It is possible to fit a periscope to the spectrograph that collects object and sky light from widely separated areas of the sky and brings them together in
a single order separated by at least one blank row. If the periscope is fitted the
program must be told so that it knows that each order actually looks like two
orders in the raw data. When the periscope is fitted it is the user’s responsibility
to ensure that the first two "orders" listed in the coefficient file are indeed an
object / sky pair. If this is not obvious, look at an arc image.

**OBJWIDTH**  The same number of rows will be extracted from each order. This
is necessary to preserve counts in the extracted echellograms. You can either
specify the number explicitly or else specify zero, in which case a sensible number
will be derived from the information in the coefficient file - assuming that there
are enough orders, the width of the third widest will be used. In this case if a
non-zero value for OBJOFFSET is specified, twice this value will be extracted
from OBJWIDTH so as to ensure that the extracted rows lie within the order.

**OBJOFFSET**  The centre of the order corresponds to the centre of the slit. If the object
is not centred on the slit, you can specify an offset (which is a floating point
number) from the centre. A positive offset corresponds to a higher Y value (i.e. a
position further up the image when displayed in conventional orientation).

**S1WIDTH**  Up to two regions of sky can be defined. Each is defined in a similar
manner to the object except that the offset is an offset from the centre of the
object and not from the centre of the order.

**S1OFFSET**  Up to two regions of sky can be defined. Each is defined in a similar
manner to the object except that the offset is an offset from the centre of the
object and not from the centre of the order.

**S2WIDTH**  Up to two regions of sky can be defined. Each is defined in a similar
manner to the object except that the offset is an offset from the centre of the
object and not from the centre of the order.

**S2OFFSET**  Up to two regions of sky can be defined. Each is defined in a similar
manner to the object except that the offset is an offset from the centre of the
object and not from the centre of the order.

**MSTART**  This is the order number of the first order whose fit is listed in the coeffi-
cient file. Conventionally this should be the order with the smallest Y value and
is the order with the highest order number.

**MDELTA**  This is the increment to the order number as the orders listed in the
coefficient file are processed. Conventionally, if the first order is that with the
lowest Y value, the increment will be -1.

**MASK**  This file will be created and will have the same dimensions as the raw input
image. Areas where there are no orders will have zero data values and areas that
are deemed to lie within an order will have a data value of 10 * (order number)
+ sub-order number) where "sub-order number" is 1 or 2. When the periscope
is fitted the sub-order number is 2 for the first "order" of each pair and 1 for the
second one. Otherwise it is always 1 for object and 2 for sky. The MASKEXT
program can be used to generate an echellogram from this mask and the raw
input image and it permits selection of which sub-orders to extract (one, the other
or both).

**Source comments:**

**Program name:**

ECHMASK
Function:
Write a mask image that can be used for quick-look extraction of orders from a raw echelle image.

Description:
The program reads a file that identifies where the orders are in the image and sets the values in the mask to be zero if that pixel in the mask does not lie in an order and to a number derived from the order number otherwise (see below). It is guaranteed that every order is extracted using the same number of rows, but of course the position of these rows may vary along an order so one can expect visible jumps in the extracted data, especially if too few rows are extracted to take all the data from the object.

The coefficient file will normally have been written by SDIST and if so must have been written by the version of SDIST that was modified to support ECHMASK.

The PERISCOPE keyword (see below) determines whether each order has two separate parts (corresponding to object and sky and due to the special periscope that samples object and sky at a wide spacing and brings them together on the slit) or one part (corresponding simply to the slit). The data values in the mask are $10 \times (\text{true order number}) + (\text{sub-order number})$ where the sub-order number is 0 if there is no periscope fitted, 1 if this is the first part of an order and 2 if this is the second part of the order. The "first" and "second" parts of an order are defined so that the actual data values in the mask are monotonic along a vertical slice through it, i.e. they might go 412, 411, 402, 401 if the periscope is fitted and they might go 410, 400 if it is not fitted.

If PERISCOPE is NO then the user has the option of splitting the data in an order into object and up to two separate regions of sky. The object is assigned sub-order 1 and the sky is assigned sub-order 2. Note that this assignment may differ from when PERISCOPE is yes, since in that case there is no guarantee that sub-order 1 is object - it may be sky! There may be room for enhancement here.

Parameters:

(>) COFILE (Character) The name of the coefficient file. Default SDIST.

(>) PERISCOPE (Keyword) Whether or not the periscope is fitted. Default YES.

(>) OBJWIDTH (Integer) The number of rows to be extracted on behalf of the object per order. If PERISCOPE is YES then object and sky are not distinguished between and this width also applies to the sky. If PERISCOPE is NO then this width applies only to the object and the position of the sky is specified using the S1* and S2* parameters.
If OBJWIDTH is specified as zero then the width information from the coefficient file is used to derive a sensible value. Default 0.

(>) OBJOFFSET  
(Float) The offset of the centre of the object data from the centre of each order. If specified as being non-zero and if OBJWIDTH is zero, the derived width is adjusted to take account of the offset. Default 0.

(>) S1WIDTH  
(Integer) The number of rows to be extracted on behalf of the first region of sky per order. This and the other S* parameters are prompted for only if PERISCOPE is NO. If specified as zero, it is assumed that no sky is to be extracted and the remaining S* parameters are not prompted for. Default 0.

(>) S1OFFSET  
(Float) The offset of the centre of the first region of sky from the centre of the object data (not necessarily from the centre of the order). Default 0.

(>) S2WIDTH  
(Integer) These parameters are the same as S1WIDTH and S2OFFSET but they refer to the second region of sky if any. Defaults 0.

(>) S2OFFSET  
(Float) S1WIDTH and S2OFFSET but they refer to the second region of sky if any. Defaults 0.

(>) MSTART  
(Integer) The order number of the first "spectrum" in the coefficient file. Default 1.

(>) MDELMTA  
(Integer) +1 if order numbers increase as "spectrum number" increased, -1 otherwise. Default -1.

(<) MASK  
(File) The name of the output mask image. This is created with only a .Z.DATA structure. Default MASK.

Language: FORTRAN

External variables used: None

Prior requirements: None

Authors: William Lupton, AAO

D.59 ECHMERGE-Merge echelle spectra into a single long spectrum

Description: The program expects two input files, each of which may be 1-D or 2-D, but both of which must have the same number of pixels in X, must have a recognised wavelength unit as the units of X and must have (near enough) identical .X.DATA arrays. In practice this means that both must have been scrunched on to the same wavelength scale. (The details of this may change when SCRUNCH has been upgraded to write an output file with a 2-D .X.DATA array describing the discontinuous scrunched orders.)
It creates a 1-D output file which consists of all the orders from the input files. Where orders overlap a weighted sum of the overlapping orders is used. The formula used is:

\[
in1weight(i) \times in1(i) + in2weight(i) \times in2(i) = \frac{in1weight(i) \times in1(i) + in2weight(i) \times in2(i)}{in1weight(i) + in2weight(i)}
\]

and the weights are simply the result of median smoothing the data that they weight. This means that more weight is given to stronger signal data, that data where one of the inputs is zero is set to the other of the inputs and that data where both of the inputs are equal is left unaltered. All of these are desirable qualities. There may be less desirable statistical consequences and it is not obvious that signal to noise ratio cannot be degraded although intuitively it will not be since on the assumption of Poisson statistics the weights are essentially just the inverse variances. At low signal, a cutoff applies since the major noise contribution will no longer be Poisson.

The output file can be the same as either of the two input files and the second input file can be given a blank name, in which case it is not required. Often the first run will use a single input file to create the output file and subsequent runs will add in more input files to the existing output file.

**Parameters:**

**IMAGE** ECHMERGE merges one or two images that contain scrunched echelle orders. The images must have the same size in the X direction but each may be 1-D or 2-D.

**IMAGE1** The name of the image that is to be merged with IMAGE. A blank name can be given.

**BOX** The input and output data is smoothed (in workspace) in order to provide slowly varying data from which to derive weights. This is the total width of the median filter used. An even value is rounded down to the next odd number.

**CUTOFF** If the ration (stronger signal)/(weaker signal) exceeds CUTOFF, only the stronger signal will be used. This is an attempt to prevent signal to noise degradation due to adding in of weak signal.

**OUTPUT** The name of the resulting image. The output image is a single long spectrum containing the merged orders from IMAGE and IMAGE1.

**Source comments:**

**Program name:** ECHMERGE

**Function:** Merge scrunched echelle orders into a single long spectrum.

**Description:** The program expects two input files, each of which may be 1-D or 2-D, but both of which must have the same number of pixels in X, must have a recognised wavelength unit as the units of X and must have (near enough) identical .X.DATA arrays. In practice this means that both must have been scrunched on to the same wavelength scale. (The details of this may change when SCRUNCH has been upgraded to write an output file with a 2-D .X.DATA array describing the discontinuous scrunched orders.)

It creates a 1-D output file which consists of all the orders from the input files. Where orders overlap a weighted sum of the
overlapping orders is used. The formula used is:

\[
\text{in1weight}(i) \times \text{in1}(i) + \text{in2weight}(i) \times \text{in2}(i) \\
\text{in1weight}(i) + \text{in2weight}(i)
\]

and the weights are simply the result of median smoothing the data that they weight. This means that more weight is given to stronger signal data, that data where one of the inputs is zero is set to the other of the inputs and that data where both of the inputs are equal is left unaltered. All of these are desirable qualities. There may be less desirable statistical consequences and it is not obvious that signal to noise ratio cannot be degraded although intuitively it will not be since on the assumption of Poisson statistics the weights are essentially just the inverse variances. At low signal, a cutoff applies since the major noise contribution will no longer be Poisson.

The output file can be the same as either of the two input files and the second input file can be given a blank name, in which case it is not required. Often the first run will use a single input file to create the output file and subsequent runs will add in more input files to the existing output file.

Parameters:

(>) IMAGE (File) The name of the first input image. This can be 1-D or 2-D and will normally be the output from SCRUNCH. However it can also be the results of a previous run of this program.

(>) IMAGE1 (File) The name of the second input image. This can be 1-D or 2-D and will normally be the output from SCRUNCH. However it can also be the results of a previous run of this program. It must have the same X size as IMAGE, must agree in X units (which must be a recognised wavelength unit) and must more or less agree in the contents of .X.DATA. If no second input image is required, its name can be specified as blank.

(>) BOX (Integer) The size of the box (in pixels) to be used in calculating the medians. Should be odd; if even, BOX-1 will be used.

(>) CUTOFF (Real) The ratio of higher signal to lower signal at which no contribution from the lower signal will be taken.

(<) OUTPUT (File) The name of the output image. This will be a 1-D image with the same size and X information as a row of either of the input images.

Language:
FORTRAN

External variables used:
None
Prior requirements:
None

Authors: William Lupton, AAO

D.60 ECHSELECT-Interactive selection of sky and object spectra for an echelle

Description: ECHSELECT allows the user to indicate interactively the cross-sections of a corrected echellogram (one straightened by CDIST for example) to be used as object and sky for the various orders. It then creates a collapsed echellogram for the object orders, and (optionally) one for the sky orders.

Parameters: IMAGE IMAGE is used to specify the corrected echellogram to be processed by ECHSELECT. It should have its orders straight and parallel to the x-axis (i.e. as processed by CDIST).

PREVIOUS If PREVIOUS is selected, the results of a previous selection are read from a file called ECHSELECT.LIS in the default directory and may be modified. Otherwise, you start from a clean slate. Note that this allows you to apply a selection based on one image to data in another - for example you can process an arc according to the selection made using an associated object exposure.

WHOLE ECHSELECT gets you to select cross-sections by indicating them on a plot of a 1-D cut through the original image. This cut is orthogonal to the wavelength direction, so the various orders should show up as peaks in the plot. Often, the ends of the image have odd effects and should not be included in this cut. If you want to specify the limits of the plot explicitly, set WHOLE to NO. Otherwise the whole image will be summed to produce the cut.

XSTART If WHOLE is not selected, XSTART gives the x-axis value for the first of the cross-sections to be summed to give the cut through the image

XEND If WHOLE is not selected, XEND gives the x-axis value for the last of the cross-sections to be summed to give the cut through the image

MSTART This is the order number of the first order in IMAGE. Getting this right is unimportant, since you have to option of setting the order numbers explicitly during ECHSELECT.

MDELTA This is the increment to the order number as the orders listed in the coefficient file are processed. Conventionally, if the first order is that with the lowest Y value, the increment will be -1. Note that the order in which the orders appear in the collapsed echellogram depends on the sign of MDELTA.

OBJOUT Once the cross-sections for each order are selected, ECHSELECT sums those for each order to produce the collapsed echellograms. Those selected for the object go into one, whose name is given by OBJOUT, and those for the sky into another. The object cross-sections for each order are simply summed into one cross-section of OBJOUT. The order in which the orders appear in the collapsed echellograms depends on MDELTA - if MDELTA is -1, higher orders are at lower cross-section numbers.

SKYOUT Once the cross-sections for each order are selected, ECHSELECT sums those for each order to produce the collapsed echellograms. Those selected for the object go into one, those for the sky into another whose name is given by
SKYOUT. The sky cross-sections for each order are summed into one cross-section of SKYOUT, but are scaled to match the number of object cross-sections for the same order. This should allow the two collapsed echellograms to be directly subtracted.

DISNCHAN Length of displayed section.
MOVETOX New plot centre x value.
ORDER Next echelle order to work on.
LOW Minimum value to display.
HIGH Maximum value to display.
ADD Used to confirm that more than one cross section are to be used for an order’s object.
CLEAR Used to confirm that all settings for an order to be cleared.
QUITSEL Used to confirm quitting work on an echelle order.

Source comments:

Routine name: ECHSELECT

Function:
Interactive object and sky selection from echellogram.

Description:
This application takes a corrected echellogram (one that has had the orders straightened, probably by CDIST), and generates a number of collapsed echellograms (images where each cross-section is a separate echelle spectrum). To determine which cross-sections should be added to produce the sky and object spectra it gets the user to indicate the relevant limits on a plot of the corrected echellogram collapsed in the spectra direction. For each order the user can select a range of cross-sections to be used for the object, and a number of ranges of cross-sections to be used for the sky. For each order, the object cross-sections are added and the sky cross-sections are added and then scaled by the factor (number of object cross-sections / number of sky cross sections). The object cross-sections are formed into one collapsed echellogram and the sky cross-sections are formed into another collapsed echellogram. Optionally, a straightened arc can also be processed using the same object and sky cross-section information, in this case producing two collapsed arc echellograms, one for the cross-sections designated as object, one for those designated as sky, but in this case without any scaling being applied.

Command parameters:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>IMAGE</td>
<td>(File) The name of the corrected echellogram.</td>
</tr>
<tr>
<td>XSTART</td>
<td>(Numeric) The first x-value to be used when collapsing the image along the wavelength direction.</td>
</tr>
<tr>
<td>XEND</td>
<td>(Numeric) The last x-value to be used when collapsing the image along the wavelength direction.</td>
</tr>
<tr>
<td>MSTART</td>
<td>(Integer) The order number for the first order.</td>
</tr>
<tr>
<td>MDELTA</td>
<td>(Integer) 1 if order numbers increase with cross-section, -1 if they decrease.</td>
</tr>
</tbody>
</table>
OBJOUT (File) The name of the object collapsed echellogram.
SKYOUT (File) The name of the sky collapsed echellogram.
DISNCHAN (Integer) Length of displayed section.
MOVETOX (Numeric) New plot centre x value.
ORDER (Integer) Next echelle order to work on.
LOW (Numeric) Minimum value to display.
HIGH (Numeric) Maximum value to display.

Command keywords:
WHOLE Yes if all of spectral range is to be used
PREVIOUS Yes if previous selection is to be used as a starting point for the interactive selection.
ADD Used to confirm that more than one cross section are to be used for an order’s object.
CLEAR Used to confirm that all settings for an order to be cleared.
QUITSEL Used to confirm quitting work on an echelle order.

Data quality information: Ignored.
Error data: Ignored.

Authors: Keith Shortridge, AAO

Files:
Selections made by ECHSELECT are written to ECHSELECT.LIS in the default directory. This begins with a number of comment lines - all of which have an ‘*’ in the first column, and is then followed by a single line for each cross-section which has been selected, giving cross-section number and order number, in format 2I10. If the cross-section is part of the sky selected for that order, the order number given is the negative of the actual order number. (Note that this program cannot cope with zeroth order!). This file is also that read if PREVIOUS is set.

D.61 EDITEXT-Edit the Specdre Extension.

Usage: editext request in

Description: This routine allows the user to modify the Specdre Extension. See the topic "Requests" for details. Users should also consult the description of the Specdre Extension in SUN/140.

Parameters: REQUEST REQUEST = _CHAR (Read) The action required. This consists of blank separated words. The following is a brief reminder of the syntax and permissible requests. For the full details refer to the "Requests" topic.
- LIST - CREATE - CREATE RESULTS type1 type2 type3 int1 int2 - DELETE - DELETE struct - SET ndf-struct - SET SPECVALS.comp value - SET COORD1.comp value - SET COORD2.comp value - SET scalar value - SET vector element value - TYPE scalar type - TYPE ndf-struct type - TYPE RESULTS type1 type2 type3 - SHAPE RESULTS int1 int2
IN IN = NDF (Read) The NDF the Specdre Extension of which is to be modified. The modification is done in situ, i.e. there is no separate output NDF. In most modes,
the routine requires update access. Only in list mode is read access sufficient.

**LOGFIL**

LOGFIL = FILENAME (Read) The filename for the ASCII output file in list mode. If this file exists, it is opened for append access. A null value for this parameter will signal that no file is to be used. The output will then be directed to the standard output device (the user’s screen).

**Examples:**

```plaintext
editext list in accept
This will look for the Specdre Extension to the main NDF called IN and list the Extension’s contents to the default output device (usually the user’s screen). Some character strings that may be up to 32 characters long are truncated to 16 characters in order to fit on the screen.

editext list in logfil=out
This will look for the Specdre Extension to the main NDF called IN and list the Extension’s contents to the ASCII file out. This happens without string truncation.

editext delete in
This will look for the Specdre Extension to the main NDF called IN and delete the Extension.

editext "set restframe heliocentric" in
This will access the main NDF called IN, find or create its Specdre Extension, find or create the RESTFRAME structure in the Extension, and put the string "heliocentric" into the RESTFRAME structure.

editext "set frequnit 6"
This will access the main NDF called IN, find or create its Specdre Extension, find or create the FREQUNIT structure in the Extension, and put the value 6 into the FREQUNIT structure. This is to mean that reference and laboratory frequencies will be expressed in MHz ($10^6$ Hz).

editext "set labfreq 5 1420" in
This will access the main NDF called IN, find its Specdre Extension and find the RESULTS structure in the Extension (which is an NDF). If this is successful the routine will find the LABFREQ extension of the result NDF and set its fifth element to 1420. This is the laboratory frequency of the fifth spectral component. In conjunction with a FREQUNIT of 6, this is (very roughly) the frequency of the 21 cm ground state hyperfine transition of neutral atomic hydrogen.

editext "set npara 5 3" in
This will access the main NDF called IN, find its Specdre Extension and find the RESULTS structure in the Extension (which is an NDF). If this is successful the routine will find the NPARA extension of the result NDF and set its fifth element to 3. This is to mean that the fifth spectral component is allocated space for three parameters in the result NDF. Changing this number may require to increase the total number of parameters which in turn affects the shape of the result NDF and of the
PARATYPE extension to the result NDF. Changing NPARA(5) also makes it necessary to shift information in the result NDF's data and variance structures as well as in the PARATYPE extension to the result NDF. All this is handled consistently by this routine.

editext "shape results 6 20" in
This will access the main NDF called IN, find or create its Specdre Extension, find or create the RESULTS structure in the Extension, and shape it to provide for six spectral components and a total of 20 parameters. If results existed before, it will be expanded or contracted "at the end". That is, existing components 1 to 6 and parameter 1 to 20 would be retained.

Notes: This routine recognises the Specdre Extension v. 1.1.
This routine works in situ and modifies the input file.

Requests: The request or action required consists of blank-separated words. The first word is a verb specifying the kind of action. The verb can be LIST, CREATE, DELETE, SET, TYPE or SHAPE. The verb is case-insensitive. The length of the request is restricted to 130 characters.

There may or may not follow a second word specifying the structure affected. This can be any of the scalar structures in the Specdre Extension, i.e. SPECAXIS, RESTFRAME, INDEXREFR, FREQREF, FREQUNIT. It can also be any of the NDF-type structures in the Specdre Extension, i.e. SPECVALS, SPECWIDS, COVRS, RESULTS. Finally it can be any structure which is an extension to the (NDF-)structure RESULTS. These latter structures are all HDS vectors, their names are LINENAME, LABFREQ, COMPTYPE, NPARA, MASKL, MASKR, PARATYPE. The structure specification is case-insensitive. Further words contain parameter values, usually one word per parameter. But if the last parameter is a string, it may consist of several words. No quotes are necessary.

There is only one LIST request, namely the sole word LIST. This will cause the complete Specdre Extension - except the contents of NDF arrays - to be listed to the log file or to the screen.

There are two possible CREATE requests.
- "CREATE" on its own will create an empty Specdre Extension, or fail if a Specdre Extension already exists.
- "CREATE RESULTS type1 type2 type3 int1 int2" needs five parameters. Three parameters are case-insensitive HDS data types. These are either _DOUBLE or assumed to be _REAL. The result structure is an NDF-type structure and the different type specifications apply to (i) the data and variance structures of the NDF, (ii) the laboratory frequency extension to the result NDF, (iii) the left and right mask extensions to the result NDF. All extensions to the result NDF are HDS vectors. Some of these have one element for each spectral component, their created length is specified by the fourth (last but one) request parameter, i.e. the sixth word. This word must convert to an integer greater than zero. Other HDS vectors in the extension to the result NDF have one element for each result parameter, their created length is specified by the fifth (last) request parameter, i.e. the seventh word. This word must convert to an integer greater than zero. "CREATE RESULTS" fails if the result NDF already exists.
"DELETE" on its own will delete the whole Specdre Extension. "DELETE struct" will delete the specified structure. This can be any of the NDF-type structures SPECVALS, SPECWIDS, COORD, COVRS, RESULTS. Deleting a structure does not include deleting the whole Extension, even if it becomes empty.

All SET request will create the Specdre Extension, even if the request is not recognised as a valid one.

"SET ndf-struct", where the second word specifies an NDF-type structure, will set the values of the specified structure to the default values. This does not work for COVRS, since it defaults to non-existence. The structure is created if it does not already exist. For SPECVALS and SPECWIDS only the NDF’s data structure is affected. For RESULTS the NDF’s data and variance structures are set to bad values, but all the vectors in the result NDF’s extension remain unchanged.

- "SET SPECVALS" will set the values in the data array of spectroscopic values to the default values. These are copies of the spectroscopic axis centres in the main NDF.
- "SET SPECWIDS" will set the values in the data array of spectroscopic widths to the default values. These are copies of the spectroscopic axis widths in the main NDF.
- "SET COORD" will set the values in the data array of COORD1 and COORD2 to the default values. These are copies axis centres for the first and second non-spectroscopic axes in the main NDF.
- "SET RESULTS" will set the values in the data and variance arrays of the result NDF to bad values.

"SET SPECVALS.comp value" can be used to set the label and unit components of the spectroscopic values’ NDF.
- "SET SPECVALS.LABEL label" will set the value of the label of the spectroscopic values’ NDF.
- "SET SPECVALS.UNITS unit" will set the value of the unit of the spectroscopic values’ NDF.
- "SET COORD1.LABEL label1" will set the value of the label of the COORD1 NDF. Similarly for COORD2.
- "SET COORD1.UNITS unit1" will set the value of the units of the COORD1 NDF. Similarly for COORD2.

"SET scalar value" will convert the third word to a value and put it in the scalar structure specified by the second word.
- "SET SPECAXIS int" will try to convert the third word into an integer. It must be between 1 and the number of axes in the NDF to which this Specdre Extension is an extension. If the value is actually changed, then this command will also delete the NDF-type structures SPECVALS, COVRS and RESULTS. This is because the contents of those structures depends on the choice of spectroscopic axis and become invalid when the value is changed. This command will also create the Specdre Extension and spectroscopic axis structure if they do not yet exist.
- "SET RESTFRAME more words" will put the third and following words (case-sensitive) into the reference frame structure. This command will also create the Specdre Extension and reference frame structure if they do not yet exist.
- "SET INDEXREFR value" will try to convert the third word into a real or double value, depending on the current type of the refractive index structure. This command will also create the Specdre Extension and refractive index structure if they do not yet exist.
- "SET FREQREF value" will try to convert the third word into a real or double value, depending on the current type of the reference frequency structure. This command will also create the Specdre Extension and reference frequency structure if they do not yet exist.
- "SET FREQUNIT int" will try to convert the third word into an integer. This command will also create the Specdre Extension and frequency unit structure if they do not yet exist.
- "SET vector element value" will change the value of the specified element in the specified vector. The vector must be one of the extensions of the result NDF. The result NDF must exist beforehand, which implies the existence of the vector. The vector must also be long enough to contain the element specified and the element number must be integer and greater than zero. There are two kinds of vectors, those indexed by spectral component and those indexed by result parameter.
- "SET LINENAME comp more words" will put the forth and following words (case-sensitive) into the comp-th element of the line name structure.
- "SET LABFREQ comp value" will try to convert the fourth word into a real or double value, depending on the current type of the laboratory frequency structure. It will then put the value into the comp-th element of the laboratory frequency structure.
- "SET COMPTYPE comp more words" will put the forth and following words (case-sensitive) into the comp-th element of the component type structure.
- "SET NPARA comp npara" will try to convert the fourth word into an integer greater than or equal to zero. This is the new number of parameters allocated to the comp-th component. Changing this value will affect several parts of the result structure both in their shapes and values. If the comp-th spectral component is allocated more parameters than before, then it may be necessary to provide for a higher total number of parameters, which implies increasing the size of .MORE.SPECDRE.RESULTS.DATA_ARRAY and VARIANCE and of .MORE.SPECDRE.RESULTS.MORE.PARATYPE. At any rate, the information about spectral components with indices higher than comp must be relocated within those arrays.
- "SET MASKL comp value" and "SET MASKR comp value" will try to convert the fourth word into a real or double value, depending on the current type of the mask structures. It will then put the value into the comp-th element of the relevant mask structure.
- "SET PARATYPE para more words" will put the forth and following words (case-sensitive) into the para-th element of the parameter type structure. A TYPE request can be applied to _REAL or _DOUBLE structures, and of these to scalars and NDF-type structures. Changing the type(s) of the result NDF needs specification of three separate types.
- "TYPE scalar type" can be applied to INDEXREFR and FREQREF. The type specification is case-insensitive. If it is not _DOUBLE, then _REAL is assumed.
- "TYPE ndf-struct type", will change the type of the specified NDF. The type specification is case-insensitive. It must be _DOUBLE or is assumed to be _REAL. This command can be applied to SPECVALS, SPECWIDS, COORD, and COVRS. SPECVALS, SPECWIDS, COORD1 and COORD2 will be created if necessary, COVRS will not be created.
- "TYPE RESULTS type1 type2 type3" will change the types of (i) the NDF’s data and variance, (ii) the NDF’s laboratory frequency extension, (iii) the NDF’s mask
extensions. The parameters are case-insensitive. They must be _DOUBLE or are assumed to be _REAL. This command includes creation of the result structure if necessary.

"SHAPE RESULTS int1 int2" will change the shape of the result structure. The two command parameters must convert to integers greater than zero. The first is the number of spectral components to be provided for, the second is the total number of parameters. If the result structure does not exist, then it is created. If it exists, then existing values are retained unless they were stored outside the new bounds.

D.62 ELSPLOT-Produces a long (<3m) error bar plot of a spectrum

Description: ELSPLOT produces an error bar plot of a spectrum with a physical size that can be specified (in metres) by the user. It will allow plots up to the maximum size allowed by the GKS driver being used - in some cases this means that a non-standard device name must be specified in order to allow a larger maximum size than usual. ELSPLOT is very similar to ESPLOT, except that it has plot dimension parameters and does not support build plots.

Parameters: SPECTRUM The name of the spectrum to be plotted by ELSPLOT It should be a 1-dimensional array.

XSIZE The length of the plot in metres. ELSPLOT can produce plots up to 10 metres in length.

YSIZE The height of the plot in metres. The reset value is the full page height for the device.

WHOLE If WHOLE is set, the whole of the spectrum is plotted. Otherwise, the limits plotted are determined by the values of XSTART and XEND, which you will be prompted for if they were not specified in the command string.

AUTOSCALE If AUTOSCALE is set, the plot is scaled so that all of the data to be plotted just fits on the display. Otherwise, the scale of the plot is determined by the values of HIGH and LOW, which you will be prompted for if they were not specified in the command string.

XSTART Specifies the first X value to be plotted, in the units used by the data - angstroms, for example, if the data is wavelength calibrated. XSTART can be set before the start of the data, if necessary. RESET will set XSTART to the first X value in the data.

XEND Specifies the last X value to be plotted, in the units used by the data - angstroms, for example, if the data is wavelength calibrated. XEND can be set after the end of the data, if necessary. RESET will set XEND to the last X value in the data.

HIGH The maximum data value to be plotted - i.e. the top Y axis value for the plot.

LOW The minimum data value to be plotted - i.e. the bottom Y axis value for the plot.

BIAS A bias value applied to the data, usually to bias up a plot which is to be superimposed upon a previous plot of almost identical data values. This makes the comparison easier. BIAS N is essentially equivalent to setting HIGH and LOW down by an amount N, so can result in unexpected axis values.

LABEL The label that will appear at the top of the plot.
These are versions of SPlot and EPlot that allow the size of the plot to be specified. LPlot produces a plot of a single spectrum, while EPlot produces an error bar plot of a spectrum which has error information.

Command parameters -

XSIZE The size of the plot in X, in metres.
YSIZE The size of the plot in Y, in metres.
SPECTRUM The data to be plotted. If there is an x-axis data component this will be used to give the x-axis. If not, the x-axis will just have to be the numbers from 1 to n.
XSTART The x-value at which plotting is to start.
XEND The x-value at which plotting is to end. (XSTART and XEND are not required if the WHOLE keyword is set.)
HIGH The maximum value to be used for the plot.
LOW The minimum value to be used for the plot.
BIAS A value used to displace the plot - BIAS is effectively a value added to the data before it is plotted. (It is implemented as a value subtracted from both HIGH and LOW.) (HIGH,LOW and BIAS are not required if the AUTOSCALE keyword is set.)
LABEL A label for the plot.

Command keywords -

AUTOSCALE The program is to work out the values for HIGH and LOW, using the maximum and minimum values in the data over the specified range.
WHOLE The program is to display all of the spectrum.
LINES The plot is not done as a histogram, but as a 'join the dots' line plot. (LPlot only)

User variables used:

HARD (Character) The device used for HARD plots.

Note:

The original version of LPlot used GKS 6.2 and the DIAGRAM package. This has now been discontinued, and some of the functionality of DIAGRAM (the ability to specify the size of the plot in physical units) has appeared in PGPLOT. This new version uses PGPLOT. It can produce a plot of the specified size, but only if that size is SMALLER than the default size for the device. In practice, this means that it can only work in the way it was intended with 'unusual' devices that have particularly large default plot sizes (which often need to be set up specially for the purpose).
D.63 **EMLT**-Fits gaussians to the strongest lines in a spectrum

**Description:** Analyses an emission line spectrum (typically an arc), and produces a list of the strongest lines, giving their widths and strengths. Can also produce a synthetic spectrum from the positions, widths and strengths of the fitted lines.

**Parameters:**

- **SPECTRUM** Specifies the spectrum whose strongest emission lines are to be fitted. If the data is more than one-dimensional, the analysis is repeated for each cross-section (i.e. spectrum) in the data.
- **XSTART** Specifies the X-value (wavelength for calibrated data, or pixel number for uncalibrated data) at which the analysis is to start.
- **XEND** Specifies the X-value (wavelength for calibrated data, or pixel number for uncalibrated data) at which the analysis is to end.
- **LINES** If LINES is non-zero, it specifies that only the indicated number of strongest lines are to be included in the analysis. If zero, all lines for which a reasonable fit can be obtained are included.
- **FWHM** If FWHM is zero, the fits performed are unconstrained; that is, the fit determines full width at half maximum for each line independently. If FWHM is non-zero, it specifies a full width at half maximum (in pixels) for each line in the spectrum, and the fit is constrained accordingly.
- **MOMENTS** If MOMENTS is set, a center of moment analysis is performed (in addition to the gaussian fit) for each line and the results included in the output. This analysis is performed for each line found, not just the LINES strongest lines.
- **SYNTH** If SYNTH is set, a synthetic spectrum based on the fitted positions, strengths and widths of the lines is generated.
- **OUTPUT** The name of the resulting synthetic spectrum, if one is to be created.

**Source comments:**

Figaro version of the original SDRSYS routine EMLT, which analyses emission lines in a spectrum, fitting gaussians to the strongest lines and logging their positions, widths and centers. Optionally, it will also give line centers using a centre of moment analysis, and can also produce a synthetic spectrum generated from the positions and widths of the located lines. Note: Figaro and SDRSYS differ in their pixel numbering, Figaro counting from 1 and SDRSYS counting from 0, so there will be a discrepancy of 1 between the output from the two versions for any pixel number values; wavelength values produced by the two should be the same.

**Parameters** -

- **SPECTRUM** (Character) The name of the spectrum to be analysed.
- **XSTART** (Numeric) The first X-value to be used for the analysis.
- **XEND** (Numeric) The last X-value to be used for the analysis.
- **LINES** (Numeric) If LINES is zero, all lines that can be fitted are listed. Otherwise, it gives the number of lines to be included in the analysis, starting with the strongest and cutting off the weaker lines.
FWHM (Numeric) If non-zero, all lines fitted are constrained to a full width at half maximum of this value - in pixels.

OUTPUT (Character) The name of any synthetic spectrum to be generated.

Keywords -

MOMENTS If set, a center of moment analysis is also performed on all lines found.

SYNTH If set, a synthetic spectrum is generated.

User variables - (">") input, "<" output)

(<) EMLT_LINES (Real) Number of lines found.
(<) EMLT_BIN (Real array) List of line centres (pixels).
(<) EMLT_POS (Real array) List of line centres (wavelength units).
(<) EMLT_FWHM_BIN (Real array) List of FWHM (pixels).
(<) EMLT_FWHM_ANG (Real array) List of FWHM (wavelength units).
(<) EMLT_STREN (Real array) List of line strengths.
(<) EMLT_PEAK (Real array) List of peak heights.

D.64 ERRCON-Converts percentage error values to absolute values

Description: At one stage in their development, Figaro routines held error data as percentage values. This was a bad idea, and all the routines were converted to use absolute error values. ERRCON converts a file with percentage errors into one with absolute errors. It should only be needed for old data files written by the old (%) versions of the various Figaro routines.

Parameters: SPECTRUM The name of a file that contains an error array whose values are expressed as a percentage of the data values.

OUTPUT The name of the resulting file, with the error array containing absolute error values. If OUTPUT is the same as SPECTRUM (the default) the operation will be performed in situ. Otherwise a new file will be created.

Source comments:

Converts a Figaro file that has an error array containing percentage errors into one that has absolute values in the error array. This is needed because of the ill-thought-out use of percentage errors at one stage in Figaro.

Command parameters -

SPECTRUM (Character) The name of the file to be converted. This will usually be a spectrum, but data of any dimension will be accepted.

OUTPUT (Character) The name of the resulting file. This can be the same as for SPECTRUM. If not, a new structure is created, with everything but the error array a direct copy of the input.
D.65  **ESPLOT-Produces an error bar plot of a spectrum**

**Description:** The ESPLOT command will plot a spectrum on the current hard or soft graphics device, producing an error bar plot.

**Parameters:**
- **SPECTRUM** The name of the spectrum to be plotted by ESPLOT. It should be a 1-dimensional array.
- **HARDCOPY** If HARDCOPY is set, the plot is written to the device defined as the current hardcopy device. Generally, this is a disk file which will then have to printed. If HARDCOPY is not set, the plot will go to the current soft copy device. The hard and soft copy devices are specified using the HARD and SOFT commands respectively. e.g. SOFT /VT
- **WHOLE** If WHOLE is set, the whole of the spectrum is plotted. Otherwise, the limits plotted are determined by the values of XSTART and XEND, which you will be prompted for if they were not specified in the command string.
- **AUTOSCALE** If AUTOSCALE is set, the plot is scaled so that all of the data to be plotted just fits on the display. Otherwise, the scale of the plot is determined by the values of HIGH and LOW, which you will be prompted for if they were not specified in the command string.
- **XSTART** Specifies the first X value to be plotted, in the units used by the data - angstroms, for example, if the data is wavelength calibrated. XSTART can be set before the start of the data, if necessary. RESET will set XSTART to the first X value in the data.
- **XEND** XEND specifies the last X value to be plotted, in the units used by the data - angstroms, for example, if the data is wavelength calibrated. XEND can be set after the end of the data, if necessary. RESET will set XEND to the last X value in the data.
- **HIGH** The maximum data value to be plotted, i.e. the top Y axis value for the plot.
- **LOW** The minimum data value to be plotted, i.e. the bottom Y axis value for the plot.
- **BIAS** A bias value applied to the data, usually to bias up a plot which is to be superimposed upon a previous plot of almost identical data values. This makes the comparison easier. BIAS N is essentially equivalent to setting HIGH and LOW down by an amount N, so can result in unexpected axis values if it is not accompanied by AXES=NO.
- **LABEL** The label that will appear at the top of the plot.
- **ERASE** Specifies that the screen is to be erased before the plot is made. Usually ERASE and AXES will not be set when a plot is superimposed on a previous one.
- **AXES** Specifies that the axes for the plot are to be drawn. These should be omitted if the plot is being superimposed on a previous one, or sometimes just to save plotting time.
- **COLOUR** Used to specify the colour for the data to be plotted in. The axes are always plotted in white. The colours allowed are Blue, White, Red, Green, Black, Cyan, Magenta, Yellow. Using Black will have the effect of erasing anything where the data is plotted. This only works on the Grinnell.
THICKNESS  Only used for 'build' or 'hard' plots. It is used to increase the thickness of the lines plotted in order to increase legibility, particularly on the Versatec. Generally 1 or 3 is reasonable for the Versatec - depending on how well set up it is at the present, and 1 should be used for other devices.

See also: FIGARO: IPLOTS, MSPLOT, SPLOT.
KAPPA: LINPLOT, MLINPLOT.

Source comments:

Produces a plot of a spectrum. The plot is directed to the device defined by the user variables 'SOFT' and 'HARD', and by the value of the command keyword 'HARDCOPY', so will appear immediately if these specify a video device (VT125, Args, etc.). If a hardcopy device is specified, the file for that device will be produced, but SPLOT does not attempt to spool it off for printing.

ESPLOT is similar to SPLOT, but plots error bars based on the errors in the data.

Command parameters -

SPECTRUM  The data to be plotted. If this contains X-axis information, this will be used. If not, the X-axis will just have to be the numbers from 1 to n.
XSTART  The x-value at which plotting is to start.
XEND  The x-value at which plotting is to end. (XSTART and XEND are not required if WHOLE is set.)
HIGH  The maximum value to be used for the plot.
LOW  The minimum value to be used for the plot.
BIAS  A value used to displace the plot - BIAS is effectively a value added to the data before it is plotted. (It is implemented as a value subtracted from both HIGH and LOW.) (HIGH, LOW and BIAS are not required if AUTOSCALE is set.)
LABEL  A label for the plot.
COLOUR  The colour for the plot, assuming the display device supports it. The axes are always white.
THICKNESS  The width of the lines used for the plot. This is only used for 'hard' & 'build' plots, and should really be 1 for anything other than a high-resolution device like a Versatec or a laser printer.

Command keywords -

AUTOSCALE  The program is to work out the values for HIGH and LOW, using the maximum and minimum values in the data over the specified range.
WHOLE  The program is to display all of the spectrum.
HARDCOPY  The plot is to produce a hard copy.
AXES  Axes will be plotted.
ERASE  The screen will be erased before the plot.
LINES The plot is not done as a histogram, but as a 'join the dots' line plot. (Only applies to SPLIT.)

User variables - (">" input, "<" output)

(>) SOFT Specifies the device and type to be used for soft plots. See the SOFT command for more details.
(>) HARD Specifies the device and type to be used for hard plots. See the HARD command for more details.
(<) TVXST is set to the starting x-value for the plot.
(<) TVXEN Is set to the final x-value for the plot.
(<) TVHIGH Is set to the same value as HIGH.
(<) TVLOW Is set to the same value as LOW.
(<) TVFILE Is set to the value of SPECTRUM.
(<) TVCOLOR Is set to the GRPCKG code for the plot colour. (The TV.. variables are intended for use by cursor routines, and reflect the settings for the last plot made, even if XSTART etc are changed.)

(Other user variables may be set by the command processor, in connection with the parameter values.)

KS / CIT 30th April 1984

D.66 EVALFIT-Evaluate fit results.

Usage: evalfit in out comp=?

Description: This routine turns components in the result structure of the Specdre Extension into a fake data set representing those results. Such a data set is necessary to perform arithmetic operations between the result (otherwise expressed only as a set of parameters) and the original data.

Parameters: INFO INFO = _LOGICAL (Read) If false, this routine will issue only error messages and no informational message. [YES]
DIALOG DIALOG = _CHAR (Read) If 'T', the routine can evaluate several sets of components. After a set of components has been evaluated, the user will be asked whether she wants to specify another set. ['T']
IN IN = NDF (Read) The input NDF. This must be a base NDF. If you need only a section of an NDF, you use SUBSET first to create the section permanently.
OUT OUT = NDF (Read) The output NDF.
COMP COMP = _INTEGER (Read) The numbers of up to 6 components to be added into the output data component. If you are not sure which component is which, you should inspect the result structure of the data first with EDITEXT.
REPLY REPLY = _LOGICAL (Read) Set true to work on another set of components. This parameter is relevant only if DIALOG is true. [NO]

Examples: evalfit in out comp=[2,5,1,2] accept
This will take the input NDF IN and create an equally shaped NDF called OUT. The specified components stored in IN’s (and OUT’s) Specdre Extension are evaluated and added up to make up
the main data in OUT. Note that component no. 2 is added twice.

Source comments: EVALFIT

The routine takes as input a base NDF (a section is not acceptable). The output is a copy of the input, except for the main NDF data and variance components. These are re-calculated from certain components in the result structure of the Specdre Extension. Thus the output contains the fit results both in the result structure and in the main NDF. The main NDF can then be compared pixel by pixel with the original data.

If the input main NDF has a variance component, the output variances will be set to zero.

This routine recognises result components created by FITCHEBY (the precursor of FITPOLY), FITGAUSS, FITPOLY, or FITTRI. Unrecognised components are ignored, i.e. not added into the data. A warning to that effect is given.

If a component in any particular position has bad values as parameters, then that component is ignored on that position. No warning to this effect is given.

A component is accepted as 7th order series of Chebyshev polynomials if the component type is 'Chebyshev series' and it has 11 parameters. These are assumed to be order, xmin, xmax, coeff0, ... coeff7.

A component is accepted as 7th order polynomial if the component type is 'polynomial' and it has 9 parameters. These are assumed to be order, coeff0, ... coeff7.

A component is accepted as Gauss or triangle if the component type is 'Gauss' or 'triangle' and it has 4 parameters. The first three are assumed to be centre, peak, FWHM.

The string comparison to check the component type is case-insensitive.

Notes: This routine recognises the Specdre Extension v. 0.7.

D.67 EXAM-Display the contents/structure of data file

Notes: The EXAM application has been removed. Please use the Starlink utility HD-STRACE (cf. SUN/102). For example, if you want a listing of the contents of a data file use:

% hdstrace datafile full

D.68 EXTIN-Correct spectrum for atmospheric extinction

Description: Given a spectrum and a calibration spectrum whose elements give the extinction coefficients at the wavelengths of the spectrum, corrects that spectrum for
atmospheric extinction.

**Parameters:**
- **SPECTRUM** The spectrum specified by SPECTRUM will be corrected for extinction, given the extinction coefficients in the coefficient spectrum, and taking account of the air mass of the observation. So SPECTRUM should have a valid value for .OBS.SECZ (the program will complain if it doesn’t). The correction algorithm used allows for the data being calibrated in magnitudes.
- **COEFF** A spectrum whose elements give the extinction coefficients applicable for the observation. COEFF will normally have been prepared by first generating a spiketrum from a table of coefficients using GSPIKE and then using LINTERP to linearly interpolate between them.
- **OUTPUT** Specifies the name of the calibrated spectrum to be produced by EXTIN. Note that this can be the same as for SPECTRUM, in which case the operation will be performed in situ.

**Source comments:**
Corrects a spectrum for extinction, given a coefficient spectrum which gives the interpolated extinction coefficients over the wavelength range of the spectrum. The spectrum must have a valid .OBS.SECZ value.

**Command parameters -**
- **SPECTRUM** (Character) The spectrum to be corrected.
- **COEFF** (Character) The coefficient spectrum.
- **OUTPUT** (Character) The resulting spectrum.

**Command keywords -** None

**User variables used -** None

---

**D.69 EXTLIST-Adds non-contiguous lines of an image to form a spectrum**

**Description:** The EXTLIST command extracts specific cross-sections of an image, adding them together to give one spectrum. The cross-section numbers are specified by an array of section numbers - this allows up to 40 non-contiguous cross-sections to be specified, rather than the single range allowed by EXTRACT.

**Parameters:**
- **IMAGE** The name of a 2-dimensional image. A number of rows (cross-sections) will be extracted from this image and added to form a single spectrum.
- **NROWS** Used to specify the number of cross-sections (rows) to be added together to form the single spectrum. It has to be specified here because the defaulting of values for the list makes it impossible to know otherwise how many of the values you actually intend to be used.
- **SECTIONS** The cross-sections of IMAGE specified by the array of row numbers given in SECTIONS will be added together.
- **SPECTRUM** Used to specify the name of the resulting 1-dimensional array. A new file will be created.
**Source comments: LIST**

Adds the rows from IMAGE specified by the array of row numbers in SECTIONS and produces a 1-D data object called SPECTRUM.

Command parameters -

'IMAGE' The name of the image from which the rows are to be taken.

'NROWS' The number of rows to be added.

'SECTIONS' The array of row numbers.

'SPECTRUM' The name of the resulting data.

Output data -

SPECTRUM is created with the same structure as IMAGE, except that data array will only have one dimension, and if IMAGE has Y-axis information, this will be omitted. Any X-axis information will be copied unchanged.

DJA / AAO 10th July 1987

**D.70 EXTRACT-Adds contiguous lines of an image to form a spectrum**

**Description:** Adds a number of consecutive rows from an image to produce a 1-D data object. (A ‘row’ is all the pixels with a given Y-value.)

**Parameters:**

- **IMAGE** Name of the image from which to extract data.
- **YSTART** First y-value to be used.
- **YEND** Last y-value to be used.
- **SPECTRUM** Name of output spectrum.

**Source comments: RACT**

Adds a number of consecutive rows from an image to produce a 1-D data object. (A ‘row’ is all the pixels with a given y-value.)

Command parameters -

'IMAGE' The name of the image from which the rows are to be taken.

'YSTART' The Y-value of the first row to be used.
If IMAGE has a Y axis structure, the data from this is used. If not, the row numbers are used, starting from 1.

'YEND' The Y-value of the last row to be used.

'SPECTRUM' The name of the resulting data.
Output data -

SPECTRUM is created with the same structure as IMAGE, except that the data will only have one dimension, and if IMAGE had a Y axis structure, this will be omitted. Any X axis structure will be copied unchanged.

KS / CIT 29th June 1984

D.71 FET321-Extracts a spectrum from 1 detector from etalon mode FIGS data

Description: FET321 takes a FIGS data cube, as produced by the FIGS data acquisition system running in etalon mode, and reduces it to a single spectrum, summing up the various cycles and performing the beamswitch and chopping subtractions. Data from only one detector is extracted.

Parameters: CUBE The name of a data cube produced by the FIGS data acquisition system. That is is should have the dimensions (wavelength steps,8,beamswitch cycles) The cube should have been taken in Etalon mode.

DETECTOR In etalon mode, each of the FIGS detectors produces data over a different wavelength range. Rather than produce a single spectrum covering disjoint wavelength ranges, FET321 uses DETECTOR to specify a single detector to be used to produce the spectrum.

SPECTRUM The name of the resulting single spectrum produced by collapsing down the FIGS data cube.

CUTOFF Values more than CUTOFF times sigma away from the mean value for the spectral point will not be included in the final spectrum.

ADD Disables the subtraction of the background beamswitch and chopping data. This is unusual (ADD=NO is the default) and is generally only required for test data.

BACK Returns the background spectrum only, rather than the background subtracted source data.

NORM Causes the data for each cycle to be normalized so that the mean value for each cycle is the same. This gives more reasonable errors when data are taken in the presence of cloud. It should not be used on very faint sources, as the mean level may go negative under these circumstances

Source comments: 3 2 1

Given a FIGS data cube as produced by the FIGS data acquisition system running in one of the etalon modes, processes it to produce a single spectrum, for one of the detectors only.

Command parameters -

'CUBE' The name of the cube from which the planes are to be taken. This should be a raw FIGS data cube.

'DETECTOR' The number of the detector to be used.

'SPECTRUM' The name of the resulting spectrum.
‘CUTOFF’  The level (in sigma) at which a point will be ignored (FIGS321 only)

Command keywords -

‘ADD’  Add the data together rather than subtracting the beamswitch and chop backgrounds

‘BACK’  Return the background spectrum only

‘NORM’  Normalize data to mean level of each cycle.

Input data -

CUBE is assumed to have a structure with the actual cube data in CUBE.Z.DATA

This routine assumes that the first axis of the cube data represents wavelength, that the second represents spectral scans in the order A1a,A1b,B1a,B1b,B2a,B2b,A2a,A2b, where A1,A2,B1,B2 represent the parts of the beamswitch ABBA cycle and a and b represent the signal and background chop positions respectively. In etalon mode 2, there are no chop positions, and the second axis is just A1,B1,A2,B2. This means that the second dimension of the cube has to be either 4 or 8. The cube third axis represents beamswitch cycle number.

The data is normalized to give a figure in detected photons per second. Along the wavelength axis, the data is assumed to be in order of etalon position, each etalon position having n values where n is the number of detectors used.

Output data -

IMAGE is created with the same structure as CUBE except that .Z.DATA will only have 1 dimension, and any .Y or .T sub-structures that CUBE has will be deleted.

If a spectrum is produced the errors (derived from the cycle to cycle statistics) are placed in the .Z.ERRORS component


D.72  FF-Flat field an image (uses Jon Tonry’s algorithm)

Description:  Flat fields an image using Jon Tonry’s algorithm.

Parameters:  IMAGE  Name of data to be flat fielded.

          FLAT  Name of flat field to be used.

          ORDER  Order for flat field profile fit.

          OUTPUT  Name of result of operation.

Source comments:

Appplies a flat field correction to an image.
D.73  **FFCROSS-Cross-correlate an image and a flat field (mainly IPCS data)**

**Description:** This is for use with some flat fields (notably IPCS) where there may be a bodily shift between the flat field and the data. For each cross-section in a given range, this routine calculates the cross-correlation between the flat field and the data. It then calculates the average shift for each cross-section, as determined from the individual cross-correlation. It also sums the individual cross-correlations, and calculates the shift given by that summed cross-correlation. The idea is that the shift determined in this way can then be applied using ISHIFT.

**Parameters:**

- **IMAGE** Name of image.
- **FLAT** Name of flat field.
- **YSTART** First Y value to be used.
- **YEND** Last Y value to be used.
- **XSTART** First X value to be used.
- **XEND** Last X value to be used.
- **RECORD** Create file to record cross-correlation?
- **CROSS** Name of cross-correlation data?
- **LOG** Log individual cross-section shifts?

**Source comments:**

Main body of the Figaro FFCROSS function. This is for use with some flat fields (notably IPCS) where there may be a bodily shift between the flat field and the data. For each cross-section in a given range, this routine calculates the cross-correlation between the flat field and the data. It then calculates the average shift for each cross-section, as determined from the individual cross-correlation. It also sums the individual cross-correlations, and calculates the shift given by that summed cross-correlation. The idea is that the shift determined in this way can then be applied using ISHIFT.

Command parameters -
IMAGE  (Character) The IMAGE to be compared with the flat field.
FLAT   (Character) The FLAT field to be used.
The FLAT and IMAGE data arrays should have the same dimensions.
YSTART (Numeric) The first cross-section to be used.
YEND   (Numeric) The last cross-section to be used.
XSTART (Numeric) Data with an AXIS(1) value less than XSTART will be ignored in the cross-correlation.
XEND   (Numeric) Data with an AXIS(1) value greater than XEND will also be ignored. Note that these values are used to determine the channel numbers to be used for IMAGE and the same ones will be used for FLAT even if FLAT has a different AXIS(1) structure.
CROSS  (Character) the name of the data structure to hold the cross-correlation, if it is to be saved.
The file created will be cross.dst, and will look like an ordinary spectrum - i.e. can be plotted by SPLOT, etc. CROSS is ignored if RECORD is not set.

Command keywords -

RECORD  If set, the summed cross-correlation of the two images will be recorded as a new data structure.
LOG     If set, the individual shifts for each cross-section will be logged as they are calculated.

User variables used -

SHIFT   (Numeric) The relative shift of the two images as determined from the summed cross-correlation.
AVSHIFT (Numeric) The average shift of the individual cross-sections.

KS / CIT 5th Oct 1983

D.74 FFT-Takes the forward FFT of a complex data structure

Usage: fft spatial_data frequency_data

Description: These Figaro functions take the FFT of the data in a file. FFT performs a forward transform, BFFT performs an inverse transform. The input file must contain a complex data structure, i.e. one with IMAGINARY and DATA components. The data may be multi-dimensional; if it is, a multi-dimensional FFT is performed. Note that the Figaro routine R2CMPLX will turn an existing real data structure into a complex one acceptable to this routine. FFT does not perform any cosine belling or other tapering of the data, nor does it reduce it to a zero mean.

Parameters: CDATA The name of a complex data structure. Such structures for the spatial domain are most easily produced using the R2CMPLX command. For the frequency domain, such data were usually created by R2CMPLX and transformed by FFT.
**OUTPUT** The name of the resulting Fourier transformed data. If OUTPUT is the same as CDATA then the transform is performed in situ; otherwise, a new file is created.

**Notes:** The fourier transform routines available in the various math libraries (NAG, IMSL, etc) all have slightly different characteristics, which show up in the programs that use them. This routine has been written around the NAG library (mainly the routines C06FAF and C06FJF), so many of its characteristics may be deduced by reading the relevant parts of the NAG manuals. In version 5.0 this routine was changed to use the PDA library, effectively FFTPACK routines. The data is re-ordered by FFT after the transform so that the zero frequency component is in the center of the resulting array, and this re-ordering is reversed by BFFT before the transform. This means that after FFT has been run, the various axes all go from -N to +N where N is the Nyquist frequency. New axis data structures that reflect this are created by FFT and will be deleted by BFFT.

**See also:** FIGARO: COSBELL, BFFT, CMPLX2I, CMPLX2R, CMPLX2M, I2CMPLX, R2CMPLX, KAPPA, CONVOLVE, LUCY, MEM2D, WIENER.

**Authors:**
ks: Keith Shortridge (AAO)
jm: Jo Murray (RAL, Starlink)
jms: ??? (AAO)
hme: Horst Meyerdierks (UoE, Starlink)

**D.75 FIB2CUBE—Arranges fibre output into 3-d data file**

**Description:** To convert a longslit spectrum of fibre spectra to a cube in an arbitrary manner. The output cube is "SORTED" in the TAURUS sense.

**Parameters:**
- **IMAGE1** IMAGE1, IMAGE2 etc. = FILE (Read) Input images
- **CUBE** CUBE = FILE (Write) Output cube
- **FILE** FILE = CHARACTER (Read) File with relationships defined

**Source comments:** available

**D.76 FIBDISP—Fits 3D cubes and plots the results**

**Description:** This cube should have been created using FIB2CUBE. Options available include displaying planes of the cube and profiles and fitting Gaussians etc. to these profiles.

**Parameters:**
- **CUBE** CUBE = FILE (Read) Cube for display This should be a file produced by FIB2CUBE, containing a .FIBRE structure.
- **YSTART** YSTART = REAL (Read) analysis lower limit The data between the limits ystart and yend is extracted and the resultant spectrum is used to locate the lines.
- **YEND** YEND = REAL (Read) analysis upper limit The data between the limits ystart and yend is extracted and the resultant spectrum is used to locate the lines.
- **YBLOCK** YBLOCK = REAL (Read) Enter analysis x-sect width Each window is of this width (except perhaps the final one).
### Parameters:

- **TSTART** = REAL (Read) analysis lower limit The data between the limits tstart and tend is extracted and the resultant spectrum is used to locate the lines.
- **TEND** = REAL (Read) analysis upper limit The data between the limits tstart and tend is extracted and the resultant spectrum is used to locate the lines.
- **TBLOCK** = REAL (Read) Enter analysis blocking width in 3rd dimension Each window is of this width (except perhaps the final one).
- **DEVICE** = CHARACTER (Read) Device for display
- **ITERATION** = INTEGER*2 (Read) New value of iteration
- **OUTABLE** = FILE (Write) Name for EXTATIC file
- **VCORR** = REAL (Read) correction to apply to radial velocities
- **TOLS** = CHARACTER (Read) For use in batch only
- **FITRAT** = REAL (Read) Ratio of widths, heights, or separation, for double fits
- **CALRAT** = INTEGER (Read) Ratio of number of iteration to default
- **OUTPUT** = FILE (Write) Name for output file
- **FIT_MODEL** = CHARACTER (Read) Model of fit to perform
- **LOW** = REAL (Read) Minimum value for display
- **HIGH** = REAL (Read) Maximum value for display
- **ABSORPTION** = LOGICAL (Read) Allow fitting of absorption lines
- **BOUNDS** = LOGICAL (Read) Perform bounded fits to lines (in batch)
- **HARDCOPY** = LOGICAL (Read) produce hardcopy plots of fits from cube
- **TABLE** = LOGICAL (Read) produce table of fits from cube
- **PRINT** = LOGICAL (Read) Produce print out of rotation curves
- **SHAPE** = LOGICAL (Read) Carry out shape analysis
- **KEEP_ITT** = LOGICAL (Read) Keep iteration files'
- **FIT** = LOGICAL (Read) perform fitting
- **AIC** = LOGICAL (Read) Use Akiakes information criterion for fitting
- **WEIGHTS** = LOGICAL (Read) Use weights for fitting
- **PRFITS** = LOGICAL (Read) Print out details of fitting
- **FULL** = LOGICAL (Read) Print out full details of fits in table

Source comments available

### D.77 FIBSEP-Separate spectra in 2D array

**Description:** This cube should have been created using FIB2CUBE. Options available include displaying planes of the cube and profiles and fitting Gaussians etc. to these profiles.

**Parameters:**
- **CUBE** = FILE (Read) Cube for display This should be a file produced by FIB2CUBE, containing a .FIBRE structure.
- **YSTART** = REAL (Read) analysis lower limit The data between the limits ystart and yend is extracted and the resultant spectrum is used to locate the lines.
YEND  YEND = REAL (Read) analysis upper limit The data between the limits ystart and yend is extracted and the resultant spectrum is used to locate the lines.

YBLOCK  YBLOCK = REAL (Read) Enter analysis x-sect width Each window is of this width (except perhaps the final one).

TSTART  TSTART = REAL (Read) analysis lower limit The data between the limits tstart and tend is extracted and the resultant spectrum is used to locate the lines.

TEND  TEND = REAL (Read) analysis upper limit The data between the limits tstart and tend is extracted and the resultant spectrum is used to locate the lines.

TBLOCK  TBLOCK = REAL (Read) Enter analysis blocking width in 3rd dimension Each window is of this width (except perhaps the final one).

DEVICE  DEVICE = CHARACTER (Read) Device for display

ITERATION  ITERATION = INTEGER*2 (Read) New value of iteration

OUTABLE  OUTABLE = FILE (Write) Name for EXTATIC file

VCORR  VCORR = REAL (Read) correction to apply to radial velocities

TOLS  TOLS = CHARACTER (Read) For use in batch only

FITRAT  FITRAT = REAL (Read) Ratio of widths, heights, or separation, for double fits

CALRAT  CALRAT = INTEGER (Read) Ratio of number of iteration to default

OUTPUT  OUTPUT = FILE (Write) Name for output file

FIT_MODEL  FIT_MODEL = CHARACTER (Read) Model of fit to perform

LOW  LOW = REAL (Read) Minimum value for display

HIGH  HIGH = REAL (Read) Maximum value for display

ABSORPTION  ABSORPTION = LOGICAL (Read) Allow fitting of absorption lines

BOUNDS  BOUNDS = LOGICAL (Read) Perform bounded fits to lines (in batch)

HARDCOPY  HARDCOPY = LOGICAL (Read) produce hardcopy plots of fits from cube

TABLE  TABLE = LOGICAL (Read) produce table of fits from cube

PRINT  PRINT = LOGICAL (Read) Produce print out of rotation curves

SHAPE  SHAPE = LOGICAL (Read) Carry out shape analysis

KEEP_ITT  KEEP_ITT = LOGICAL (Read) Keep iteration files’

FIT  FIT = LOGICAL (Read) perform fitting

AIC  AIC = LOGICAL (Read) Use Akiakes information criterion for fitting

WEIGHTS  WEIGHTS = LOGICAL (Read) Use weights for fitting

PRFITS  PRFITS = LOGICAL (Read) Print out details of fitting

FULL  FULL = LOGICAL (Read) Print out full details of fits in table

Source comments: available

D.78 FIGHELP-Provide Figaro on-line help

Usage:  fighelp [topic]

Description:  This routine interfaces the portable help library for the Figaro package with a terminal. The ADAM parameter TOPIC is used only for the initial entry into the help
The user can then navigate through the library with the following responses to the prompt:
- A blank response gets you one level up in the topic hierarchy.
- A question mark (?) re-displays the current topic.
- An end-of-file character exits fighelp. Note that this is Ctrl-z under VMS but usually Ctrl-d under Unix.
- Any normal text specifies (sub-) topics to look for.
- Each blank-separated word stands for one topic in the hierarchy. E.g. three blank-separated words go down three levels in the hierarchy.
- Each underscore-separated word stands for an underscore-separated word in a single topic
- Words (whether separated by blanks or underscores) that are not unique topics or include wild card characters are expanded and help is given on all matching topics. Wild card characters are % for a single character and * for any number of characters including none. In the word expansion A_G_N would match active_galactic_nuclei, which is one topic. The same is true for A*_G*_N* or active or active*.

When the help text to be printed is longer than the terminal page, then the user is asked to press the Return key before proceeding with output. At this point, too, can an end-of-file character be given to exit fighelp immediately.

**Parameters:**
- **PAGE** The number of lines that are a screen-full of information on the terminal. This is used so that FIGHELP knows when to wait for the reader to hit the return key. To turn paging off set this parameter zero. If this is not given, then the routine will try to find out about the terminal on its own.
- **WIDTH** The number of columns on the screen. If this is not given, then the routine will try to find out about the terminal on its own.
- **LIBRARY** The full file name of the library to be enquired. If this is not given, then the translation of the environment variable FIG_HELP will be used.
- **TOPIC** A initial topic to be looked for in the library. If this is not given, the top level of the library will be presented.

**Notes:** This routine is available only under Unix from a Unix shell.

**Authors:** hme: Horst Meyerdierks (UoE, Starlink)

### D.79 FIGINFO-Describes the contents of a Figaro data file

**Description:** FIGINFO provides a way of looking at the contents of a Figaro data file through Figaro’s eyes. You can do an hdstrace of file, but this doesn’t necessarily tell you how Figaro will interpret what you find there, particularly in the case of awkward things like the flag that indicates whether or not the file’s main data array may contain flagged data values (which doesn’t necessarily mean that it does, just that it might). This particular flag can be a problem for Figaro files, partly because the default rules - how you interpret it’s absence - is different in the two data formats, .SDF and .DST. If it is set when the file does not in fact contain flagged values then processing the file can be inefficient, particularly for large files. If the file does contain flagged data values but the flag is not set, then very odd results can be obtained when the file is processed. FIGINFO uses the same file access routines as a normal Figaro program to interpret the file contents. It also provides a couple of options for manipulating the ‘may contain flagged data values’ flag, should it be mis-set.
Parameters: INPUT The name of a datafile. FIGINFO will list its contents - whether it has error or quality information, whether it has a set of FITS keywords, etc. Most of this can be gleaned from doing an hdstrace on the file, but hdstrace just shows the file contents - FIGINFO explains how they are interpreted by FIGARO.

CHECK_FLAGS If CHECK_FLAGS is set, and the file has the flag set to indicate that it MAY have flagged data values in the main data array, then FIGINFO will read the array and see if there are in fact any flagged data values there. If there are not, it clears the 'may have flagged values' flag. This will speed up the processing of the file by Figaro programs that do not handle flagged values themselves. There is the overhead of the check, of course, which can be large for very large data arrays, but FIGINFO does this as efficiently as possible. (The same effect can be obtained by doing an ISTAT on the data, but this is much less efficient)

CLEAR_FLAG If the file has the flag set to indicate that it MAY have flagged data values in the main data array, but the option to check the data array values is not taken - presumably on the grounds that the overhead is not warranted - then the CLEAR_FLAG option may be set to indicate that there are definitely no flagged data values in the array and the file should be modified to show this. This is a DANGEROUS option to use. It should only be taken if the overhead of checking the data array is too large - and that implies a huge data file! - and if the user is CERTAIN that there really are no flagged data values in the data. Use of this option is not recommended.

SET_FLAG If the file does not have the flag set to indicate that it MAY have flagged data values in the main data array, but nevertheless does have such values, then a number of programs will have problems handling the file. The SET_FLAG option allows this flag to be set. This is a safe option - setting it unnecessarily does no harm, but it does make for rather inefficient processing of the file. However, this really shouldn’t be necessary - except, perhaps to correct for a mistaken use of the CLEAR_FLAG option!

Source comments: FIGINFO provides a way of looking at the contents of a Figaro data file through Figaro's eyes. You can do an hdstrace of file, but this doesn’t necessarily tell you how Figaro will interpret what you find there, particularly in the case of awkward things like the flag that indicates whether or not the file's main data array may contain flagged data values (which doesn’t necessarily mean that it does, just that it might). This particular flag can be a problem for Figaro files, partly because the default rules - how you interpret it’s absence - is different in the two data formats, .SDF and .DST. If it is set when the file does not in fact contain flagged values then processing the file can be inefficient, particularly for large files. If the file does contain flagged data values but the flag is not set, then very odd results can be obtained when the file is processed. FIGINFO uses the same file access routines as a normal Figaro program to interpret the file contents. It also provides a couple of options for manipulating the 'may contain flagged data values' flag, should it be mis-set.
D.80 FIGS321-Processes a FIGS data cube down to a single spectrum

Description: FIGS321 takes a FIGS data cube, as produced by the FIGS data acquisition system, and reduces it to a single spectrum, summing up the various cycles and performing the beamswitch and chopping subtractions.

Parameters: CUBE The name of a data cube produced by the FIGS data acquisition system. That is is should have the dimensions (wavelength steps,8,beamswitch cycles)

SPECTRUM The name of the resulting single spectrum produced by collapsing down the FIGS data cube.

CUTOFF Values more than CUTOFF times sigma away from the mean value for the spectral point will not be included in the final spectrum.

ADD Disables the subtraction of the background beamswitch and chopping data. This is unusual (ADD=NO is the default) and is generally only required for test data.

BACK Returns the background spectrum only, rather than the background subtracted source data.

NORM Causes the data for each cycle to be normalized so that the mean value for each cycle is the same. This gives more reasonable errors when data are taken in the presence of cloud. It should not be used on very faint sources, as the mean level may go negative under these circumstances.

Source comments: S 3 2 2 , F I G S 3 2 1

Given a FIGS data cube as produced by the FIGS data acquisition system, processes it to produce either an image of wavelength against cycle number (FIGS322) or a single spectrum (FIGS321).
'CUBE' The name of the cube from which the planes are to be taken. This should be a raw FIGS data cube.

'IMAGE' The name of the resulting image (FIGS322)

'SPECTRUM' The name of the resulting spectrum (FIGS321)

'CUTOFF' The level (in sigma) at which a point will be ignored (FIGS321 only)

Command keywords -

'ADD' Add the data together rather than subtracting the beamswitch and chop backgrounds

'BACK' Return the background spectrum only

'NORM' Normalize data to mean level of each cycle. (FIGS321 only.)

Input data -

This routine assumes that the first axis of the cube data represents wavelength, that the second represents spectral scans in the order A1a,A1b,B1a,B1b,B2a,B2b,A2a,A2b, where A1,A2,B1,B2 represent the parts of the beamswitch ABBA cycle and a and b represent the signal and background chop positions respectively. In grating mode 2, there are no chop positions, and the second axis is just A1,B1,A2,B2. Grating mode 3 data is modified by the on-line acquisition software so that it has the same format as grating mode 1 data. This means that the second dimension of the cube has to be either 4 or 8. The cube third axis represents beamswitch cycle number. The data is sorted into wavelength order using the various grating parameters read from the .FITS sub-structure of CUBE. The data is normalized to give a figure in detected photons per second.

Output data -

IMAGE is created with the same structure as CUBE except that main data array will only have 1 or 2 dimensions, and any AXIS sub-structures that CUBE has will be deleted/renamed as appropriate. If a spectrum is produced the errors (derived from the cycle to cycle statistics) are generated.

KS / AAO 8th June 1985

D.81 FIGS322-Processes a FIGS data cube down to an image

Description: FIGS321 takes a FIGS data cube, as produced by the FIGS data acquisition system, and reduces it to a single image, subtracting off the various beamswitch and chopping backgrounds.

Parameters: CUBE The name of a data cube produced by the FIGS data acquisition
system. That is is should have the dimensions (wavelength steps, beamswitch cycles).

**IMAGE** The name of the resulting single image produced by collapsing down the FIGS data cube.

**ADD** Disables the subtraction of the background beamswitch and chopping data. This is unusual (ADD=NO is the default) and is generally only required for test data.

**BACK** Returns the background spectrum only, rather than the background subtracted source data.

**Source comments:** S 3 2 2, FIGS 3 2 1

Given a FIGS data cube as produced by the FIGS data acquisition system, processes it to produce either an image of wavelength against cycle number (FIGS322) or a single spectrum (FIGS321).

**Command parameters** -

'CUBE' The name of the cube from which the planes are to be taken. This should be a raw FIGS data cube.

'IMAGE' The name of the resulting image (FIGS322)

'SPECTRUM' The name of the resulting spectrum (FIGS321)

'CUTOFF' The level (in sigma) at which a point will be ignored (FIGS321 only)

**Command keywords** -

'ADD' Add the data together rather than subtracting the beamswitch and chop backgrounds

'BACK' Return the background spectrum only

'NORM' Normalize data to mean level of each cycle. (FIGS321 only.)

**Input data** -

This routine assumes that the first axis of the cube data represents wavelength, that the second represents spectral scans in the order A1a,A1b,B1a,B1b,B2a,B2b,A2a,A2b, where A1,A2,B1,B2 represent the parts of the beamswitch ABBA cycle and a and b represent the signal and background chop positions respectively. In grating mode 2, there are no chop positions, and the second axis is just A1,B1,A2,B2. Grating mode 3 data is modified by the on-line acquisition software so that it has the same format as grating mode 1 data. This means that the second dimension of the cube has to be either 4 or 8. The cube third axis represents beamswitch cycle number. The data is sorted into wavelength order using the various grating parameters read from the .FITS sub-structure of CUBE. The data is normalized to give a figure in detected photons per second.
Output data -

IMAGE is created with the same structure as CUBE except that main data array will only have 1 or 2 dimensions, and any AXIS sub-structures that CUBE has will be deleted/renamed as appropriate. If a spectrum is produced the errors (derived from the cycle to cycle statistics) are generated.

KS / AAO 8th June 1985

D.82 FIGS422-Process a FIGS image-mode hypercube down to an image

Description: Given an image mode FIGS data hypercube, produces an image. It included all the scan cycles in the hypercube, but only those wavelength planes within a specified range. The data hypercube may have been sorted into wavelength order (by the program FIGS424) or it may be raw data as produced by the acquisition system.

Parameters: HCUBE The name of an image mode FIGS data hypercube, as produced by the FIGS data acquisition system or as sorted by FIGS424.

XSTART FIGS422 only includes wavelength planes that fall within the wavelength range specified by XSTART..XEND. The program allows some slop in the specification of the wavelength values, so a single plane can be picked out without having to worry about giving its wavelength exactly as it appears in the cube.

XEND Specifies the end of the wavelength range to be included in the resulting image.

CYSTART The first cycle number to be included in the output data file(s).

CYEND The last cycle number to be included in the output data file(s).

IMAGE The name of the resulting image. FIGS422 always generates a new image file.

SPLIT If SPLIT is set, then FIGS422 will generate a separate file for each of the cycles in the range CYSTART through CYEND. The files will be given the name specified by IMAGE, but with the cycle number appended. If SPLIT is not set, FIGS422 adds all the cycles in the range to produce a single image whose name is that specified by IMAGE.

Source comments: S 4 2 2

Given a FIGS image mode data hypercube, either sorted into wavelength order (e.g. by FIGS424) or not, sums all the cycles and wavelength planes within a specified wavelength range to produce an image.

Command parameters -

‘HCUBE’ (Character) The name of the hypercube to be processed.

‘XSTART’ (Real) The start of the wavelength range to be included.

‘XEND’ (Real) The end of the wavelength range to be included.

‘CYSTART’ (Integer) The first cycle to be included.

‘CYEND’ (Integer) The last cycle to be included.

‘IMAGE’ (Character) The name of the resulting image.
Command keywords -

'SPLIT' If set, FIGS422 will create a number of output files, one for each cycle in the specified range, rather than just one with all the cycles in the range summed. In this case, the output files will be named using the name specified using 'IMAGE', but with the cycle number appended.

Input data -

HCUBE is assumed is the actual hypercube data.

This routine assumes that the first axis of the cube data represents wavelength, that the second and third represent the X and Y dimensions of the image (this is an unfortunate, since it means that the AXIS(1) structure of the hypercube represents wavelength, the AXIS(2) represents the image X axis and so forth) and the fourth axis represents scan cycle number.

Output data -

IMAGE is created with the same structure as HCUBE except that the main data array will only have 2 dimensions, and any AXIS sub-structures that HCUBE has will be deleted/renamed as appropriate.

KS / AAO 6th Jan 1985

D.83 FIGS423-Process a FIGS image-mode hypercube down to a cube

Description: Given an image mode FIGS data hypercube, produces a cube, in which all the data from a selected range of cycles has been summed. The data hypercube may have been sorted into wavelength order (by the program FIGS424) or it may be raw data as produced by the acquisition system. In general, it will be better to sort the data before applying FIGS423.

Parameters: HCUBE The name of an image mode FIGS data hypercube, as produced by the FIGS data acquisition system or as sorted by FIGS424.

CYSTART The first cycle number to be included in the output data cube.

CYEND The last cycle number to be included in the output data cube.

CUBE The name of the resulting cube. FIGS423 always generates a new image file.

Source comments:

Given a FIGS image mode data hypercube, either sorted into wavelength order (e.g. by FIGS424) or not, sums all the cycles and wavelength planes within a specified wavelength range to produce an image. Note that it is probably best to have performed the wavelength sort first.
Input data -

This routine assumes that the first axis of the hcube data represents wavelength, that the second and third represent the X and Y dimensions of the image (this is an unfortunate, since it means that the .X axis of the hypercube represents wavelength, the .Y represents the image X axis and so forth) and the fourth axis represents scan cycle number.

Output data -

CUBE is created with the same structure as HCUBE except that the dta array will only have 3 dimensions, and any AXIS(4) sub-structures that HCUBE has will be deleted.

KS / AAO 19th May 1986

D.84 FIGS424-Sort a FIGS image-mode hypercube into wavelength order

Description: Re-orders a raw FIGS image-mode data hypercube so that the spectral dimension is in ascending order of wavelength, instead of in the order as read by the data acquisition system.

Parameters: HCUBE The name of an image mode FIGS raw data hypercube, as produced by the FIGS data acquisition system.

OUTPUT The name of the resulting hypercube. If this is the same as HCUBE the data is processed in situ; otherwise a new output file will be created.

Source comments: S 4 2 4

Given a FIGS image-mode data hypercube as produced by the FIGS data acquisition system, processes it to produce a hypercube in which the data have been sorted into wavelength order in accordance with the wavelength parameters included in the hypercube.

Command parameters -

'HCUBE' The name of the hypercube to be processed. This should be a raw FIGS data hypercube.

'OUTPUT' The name of the resulting hypercube. If this is the same as HCUBE the data is processed in situ, if not a new output file is produced.

Command keywords - None

Input data -
HCUBE is assumed to have a structure with the actual
cube data in HCUBE.Z.DATA

This routine assumes that the first axis of the cube data
represents wavelength, that the second and third represent the
X and Y dimensions of the image (this is an unfortunate,
since it means that the .X axis of the hypercube represents
wavelength, the .Y represents the image X axis and so forth)
and the fourth axis represents scan cycle number.
The data is sorted into wavelength order using the various
grating parameters read from the .FITS sub-structure of HCUBE.
The data is only re-ordered in the first dimension of the
hypercube.

Output data -

OUTPUT is created with the same structure as HCUBE, but with
a .X structure added to contain the wavelength information.

KS / AAO 25th Nov 1985

D.85 FIGSEE-Generate a seeing ripple spectrum from a FIGS spectrum

Description: FIGSEE generates a seeing ripple spectrum from a FIGS spectrum. In prin-
ciple, the FIGS data can then be divided by this ripple spectrum to take out the effects
of variable seeing during the observation.

Parameters: 

**SPECTRUM** Spectrum to use for seeing data.
**NDET** Number of detectors to use.
**DETECTORS** Detectors to use.
**OUTPUT** Name of resulting seeing ripple data.

Source comments:

Figaro function that attempts to produce a seeing ripple spectrum
from a Figs spectrum, averaging the data from one or more detectors,
normalising the result to unity, and generating a spectrum in
which the normalised data from these detectors (ideally ones not
contaminated by spectral features) are repeated for each detector.

Command parameters -

**SPECTRUM** (Character) The name of the file containing the
spectrum to be used.
**NDET** (Integer) The number of detectros to be used.
**DETECTORS** (Numeric array) The detectors to be used.
**OUTPUT** (Character) The name of the resulting ripple spectrum.

Command keywords - None

KS / AAO 11th Feb 1987

D.86 FIGSFLUX-Flux calibrates a FIGS spectrum

Description: Flux calibrates a FIGS spectrum using a standard spectrum.
Parameters: SPECTRUM  Name of Star spectrum.

STANDARD  Name of Standard spectrum.

KMAG  K magnitude of standard star.

OUTPUT  Name of resulting spectrum.

Source comments: S F L U X

Flux calibrates a FIGS spectrum using a standard spectrum

Command parameters -

SPECTRUM  The name of the structure containing the first image.

STANDARD  The name of the structure containing the second image data.

KMAG  The K magnitude of the standard used

OUTPUT  The name of the result of the operation. This can be the same as for SPECTRUM. If not, a new structure is created, with everything but the data a direct copy of the input.

JAB / AAO  14th June 1985

D.87  FILLCUBE-Copy one NDF into part of another.

Usage: fillcube in out

Description: This routine copies data, variance etc. from one NDF into another existing NDF. By successive calls the output NDF can be filled with data from a number of input NDFs. The target area in the output is identified by matching axis data (not pixel indices). Data are copied from input to output only if the input data value is not bad, apart from that existing data in the output are overwritten.

Parameters: INFO  INFO = _LOGICAL (Read) True if informational messages are to be issued.

TOL  TOL = _REAL (Read) The tolerated fraction of the pixel size by which the input coordinates may deviate from the output coordinates. If any one of the axis values deviates more than TOL times the coordinate step, then the input data are ignored and the output data left unchanged. [0.2]

IN  IN = NDF (Read) The input NDF.

OUT  OUT = NDF (Read) The output NDF. This must already exist, update access is required.

Source comments: L L C U B E

This application is more akin to ASCIN than to GROW. The main differences to ASCIN are that FILLCUBE updates an existing output and that its input is an NDF rather than an ASCII table. Its main advantage over GROW is that input and output may (actually must) have the same dimensionality, but any dimensions or axis data can differ. Also it is not necessary that target
pixels form a contiguous subset in the output: The input pixels could match, say, every second or third output pixel. The disadvantages are that results and spectroscopic values in the Specdre Extension are not handled, and that the coordinates along each axis in input and output must be linear.

For each input pixel, FILLCUBE looks for the output pixel that is nearest in the space of axis data coordinates. Data are copied only if the output pixel is hit close to its centre. However, if an axis is degenerate (has only one pixel) in both input and output, then the coordinates are assumed to match.

No indication is given as to how many input pixels did not match any output pixel.

Notes: This routine recognises the Specdre Extension v. 0.7, although it is largely ignored. This routine works in situ on an existing output file.

Spectroscopic values must not exist in the Extension of either the input or the output NDF: A unique coordinate axis is required for all axes, including the spectroscopic one, in order to locate the target pixels by matching coordinates between input and output. If this is inconvenient, GROW may be a more suitable application for your purpose.

Spectroscopic widths must not exist in the Extension of the output NDF and are ignored in the input NDF: This information is likely to be present only when spectroscopic values are present as well.

Covariance row sums must not exist in the Extension of the output NDF: The validity of this information is difficult to assess when only parts of spectra might be copied from one cube to another, and when these parts are contiguous in the input but might not be in the output. Input covariance row sums are ignored.

The results in the input Extension are ignored, and results must not exist in the output Extension.

D.88 FINDSP-Locate fibre spectra in an image

Description: This routine locates spectra in a large fibre frame and produces a polynomial file. The polynomial file has a version 2 format. Version 1 format uses the coefficients of a Chebyshev series, while version 2 format uses ordinary polynomial coefficients.

The technique of this routine is to
1 Compress the data array, 2 Follow ridges from start positions by centroiding, 3 Fit a polynomial Y(X) to the centroids, 4 Write the polynomial coefficients to a text file.

The text file can be read by the applications OVERPF and POLEXT. Those applications will also be able to read text files in version 1 format.

Parameters: IMAGE The fibre frame - one with distorted fibre spectra equally spaced.

BLACK The data value below which the image display is to have the background colour. The display is scaled linearly between the values specified as BLACK and WHITE.

WHITE The data value above which the image display is to have the foreground colour. The display is scaled linearly between the values specified as BLACK and WHITE.
NUMFIB  The total number of fibres used in the observation, including any dud fibres.

NORDER  The order of the polynomial to be fitted along each spectrum. The default is 6 and the maximum order allowed is 10. An even order is suggested by the presence of 'barrel' distortion.

NPTS  The image is compressed in the X (wavelength) direction before the centroids are determined. This parameter fixes the number X-direction bins in the compressed frame. This parameter also by definition is the number of points along the spectrum to be used for fitting the polynomial. Choice of this parameter is a trade-off between having enough points along the spectra that the the curved spectra can be reliably followed and having enough S/N in the compressed image to determine a reliable centroid.

FWCENT  The 'full-width' of the centroiding range in the vertical direction.

CFW  To get the initial centre for the centroiding the program does a linear extrapolation from the last two centroids. The program searches out from the previously determined central centroids. In order to suppress large fluctuations that sometimes occur it is necessary to have damping in the extrapolation. If fact this 'Centroid Weighting Function' parameter is the constant that the true gradient of the linear extrapolation is multiplied by to guess the next centroid. Hence a value of CFW less than 1 damps the extrapolation toward the horizontal.

YFIRST  The position of the centre of the first spectrum. This is expressed as the number of pixels up from the bottom of the image as viewed on the ARGS. Note that the default is only a guess from the size of the image.

YSEP  The average number of pixels separating each spectrum in the input image. Again the default value represents a guess.

PFILE  The file to which the results of the spectrum fitting performed by FINDSP is to be written. If no extension is specified, '.pol' is used.

ADJUST  Used to ask whether centroid start points need adjustment.

CHGPAR  Used to ask whether analysis to be repeated with changed parameters.

REJECT  The number of a fibre to be rejected.

CHGREJ  Used to ask whether the set of fibres to be rejected should be revised.

Authors:  jrl: John Lucey (AAO, Durham)
          hme: Horst Meyerdierks (UoE, Starlink)

D.89  FITBB-Fit diluted Planck curves to a spectrum.


Description:  This routine fits up to six diluted Planck curves to a one-dimensional data set. This can be specified as an NDF section. The data set must extend along the spectroscopic axis. The fit is done on a double logarithmic representation of the data. The axis data must be the common logarithm of frequency in Hertz. The data themselves must be the common logarithm of intensity or flux density in arbitrary units.

Parameters:  INFO  INFO = _LOGICAL (Read) If false, this routine will issue only error messages and no informational message. [YES]
VARUSE VARUSE = _LOGICAL (Read) If false, input variances are ignored. [YES]

DIALOG DIALOG = _CHAR (Read) If 'T', the routine offers in general more options for interaction. The mask or guess can be improved after inspections of a plot. Also, the routine can resolve uncertainties about where to store results by consulting the user. ['T']

IN IN = NDF (Read) The input NDF. This must be a one-dimensional (section of an) NDF. You can specify e.g. an image column as IN(5,) or part of an image row as IN(2:3.3,10). Update access is necessary to store the fit result in the NDF's Specdre Extension.

REPAIR REPAIR = _LOGICAL (Read) If DIALOG is true, REPAIR can be set true in order to change the spectroscopic number axis in the Specdre Extension. [NO]

DEVICE DEVICE = DEVICE (Read) The name of the plot device. Enter the null value (!) to disable plotting. [!]

MASK1 MASK1(6) = _REAL (Read) Lower bounds of mask intervals. The mask is the part(s) of the spectrum that is (are) fitted and plotted. The mask is put together from up to six intervals:

\[ \text{mask} = \{\text{MASK1}(1);\text{MASK2}(1)\} \cup \{\text{MASK1}(2);\text{MASK2}(1)\} \cup \ldots \cup \{\text{MASK1}(\text{MSKUSE});\text{MASK2}(\text{MSKUSE})\} \]

The elements of the MASK parameters are not checked for monotony. Thus intervals may be empty or overlapping. The number of intervals to be used is derived from the number of lower/upper bounds entered. Either MASK1 or MASK2 should be entered with not more numbers than mask intervals required.

MASK2 MASK2(6) = _REAL (Read) Upper bounds of mask intervals. See MASK1.

NCOMP NCOMP = _INTEGER (Read) The number of Planck curves to be fitted. Must be between 1 and 6. [1]

THETA THETA(6) = _REAL (Read) Guess scaling constant for each diluted Planck component.

ALPHA ALPHA(6) = _REAL (Read) Guess emissivity exponent for each diluted Planck component.

LGTEMP LGTEMP(6) = _REAL (Read) Guess common logarithm of colour temperature in Kelvin for each diluted Planck component.

SF SF(6) = _INTEGER (Read) For each component I, a value SF(I)=0 indicates that THETA(I) holds a guess which is free to be fitted. A positive value SF(I)=I indicates that THETA(I) is fixed. A positive value SF(I)=J<I indicates that THETA(I) has to keep a fixed offset from THETA(J).

AF AF(6) = _INTEGER (Read) For each component I, a value AF(I)=0 indicates that ALPHA(I) holds a guess which is free to be fitted. A positive value AF(I)=I indicates that ALPHA(I) is fixed. A positive value AF(I)=J<I indicates that ALPHA(I) has to keep a fixed offset to ALPHA(J).

TF TF(6) = _INTEGER (Read) For each component I, a value TF(I)=0 indicates that LGTEMP(I) holds a guess which is free to be fitted. A positive value TF(I)=I indicates that LGTEMP(I) is fixed. A positive value TF(I)=J<I indicates that LGTEMP(I) has to keep a fixed ratio to LGTEMP(J).

REMASK REMASK = _LOGICAL (Read) Reply YES to have another chance for improving the mask. [NO]

REGUESS REGUESS = _LOGICAL (Read) Reply YES to have another chance for improving the guess and fit. [NO]
FITGOOD  FITGOOD = _LOGICAL (Read) Reply YES to store the result in the Specdre Extension. [YES]

COMP  COMP = _INTEGER (Read) The results are stored in the Specdre Extension of the data. This parameter specifies which existing components are being fitted. You should give NCOMP values, which should all be different and which should be between zero and the number of components that are currently stored in the Extension. Give a zero for a hitherto unknown component. If a COMP element is given as zero or if it specifies a component unfit to store the results of this routine, then a new component will be created in the result storage structure. In any case this routine will report which components were actually used and it will deposit the updated values in the parameter system. [1,2,3,4,5,6]

LOGFIL  LOGFIL = FILENAME (Read) The file name of the log file. Enter the null value (!) to disable logging. The log file is opened for append. [!]

Source comments: T B B

This routine fits up to six diluted Planck curves to a one-dimensional data set. This can be specified as an NDF section. The data set must extend along the spectroscopic axis. The fit is done on a double logarithmic representation of the data. The axis data must be the common logarithm of frequency in Hertz. The data themselves must be the common logarithm of intensity or flux density in arbitrary units.

A diluted Plank component is defined as

$$f_j \Theta_j \alpha_j \nu$$

$$10 = 10^{\frac{(\nu/Hz)(2h/c^2)}{\exp(h\nu/kT_j) - 1}}$$

This assumes that the optical depth is small and the emissivity is proportional to the frequency to the power of alpha. $10^\Theta$ is the hypothetical optical depth at frequency 1 Hz.

If the optical depth is large, a single simple Planck function should be fitted, i.e. only one component with alpha = 0. In this case $10^\Theta$ is the conversion factor from the Planck function in Jy/sr to the (linear) data values. If for example the data are the common logarithm of the calibrated flux density of a source in Jy, then $\Theta$ is the logarithm of the solid angle (in sr) subtended by the source.

The fit is performed in double logarithmic representation, i.e. the fitted function is

$$f = \log[ \sum_j 10^{-f_j} ]$$

The choice of $\Theta$, alpha and $\log(T)$ as fit parameters is intuitive, but makes the fit routine ill-behaved. Very often alpha cannot be fitted at all and must be fixed. Theta and alpha usually anti-correlate completely. Even with fixed alpha do Theta and $\log(T)$ anti-correlate strongly.
Furthermore, Theta is difficult to guess. From any initial guess of Theta one can improve by using Theta plus the average deviation of the data from the guessed spectrum.

After accessing the data and the (optional) plot device, the data will be subjected to a mask that consists of up to six abscissa intervals. These may or may not overlap and need not lie within the range of existing data. The masking will remove data which are bad, have bad variance or have zero variance. The masking will also provide weights for the fit. If the given data have no variances attached, or if the variances are to be ignored, all weights will be equal.

After the data have been masked, guessed values for the fit are required. These are

- the number of components to be fitted,
- the components' guessed scaling constants Theta,
- emissivity exponents alpha and
- common logarithms of colour temperatures in Kelvin. Finally,
- fit flags for each of the parameters are needed.

The fit flags specify whether any parameter is fixed, fitted, or kept at a constant offset to another fitted parameter.

The masked data and parameter guesses are then fed into the fit routine. Single or multiple fits are made. Fit parameters may be free, fixed, or tied to the corresponding parameter of another component fitted at the same time. They are tied by fixing the offset. Up to six components can be fitted simultaneously.

The fit is done by minimising chi-squared (or rms if variances are unavailable or are chosen to be ignored). The covariances between fit parameters - and among these the uncertainties of parameters - are estimated from the curvature of psi-squared. psi-squared is usually the same as chi-squared. If, however, the given data are not independent measurements, a slightly modified function psi-squared should be used, because the curvature of chi-squared gives an overoptimistic estimate of the fit parameter uncertainty. In that function the variances of the given measurements are substituted by the sums over each row of the covariance matrix of the given data. If the data have been re-sampled with a Specdre routine, that routine will have stored the necessary additional information in the Specdre Extension, and this routine will automatically use that information to assess the fit parameter uncertainties. A full account of the psi-squared function is given in Meyerdierks, 1992a/b. But note that these covariance row sums are ignored if the main variance is ignored or unavailable.

If the fit is successful, then the result is reported to
the standard output device and plotted on the graphics device. The final plot view port is saved in the AGI data base and can be used by further applications.

The result is stored in the Specdre Extension of the input NDF. Optionally, the complete description (input NDF name, mask used, result, etc.) is written (appended) to an ASCII log file.

Optionally, the application can interact with the user. In that case, a plot is provided before masking, before guessing and before fitting. After masking, guessing and fitting, a screen report and a plot are provided and the user can improve the parameters. Finally, the result can be accepted or rejected, that is, the user can decide whether to store the result in the Specdre Extension or not.

The screen plot consists of two view ports. The lower one shows the data values (full-drawn bin-style) overlaid with the guess or fit (dashed line-style). The upper box shows the residuals (cross marks) and error bars. The axis scales are arranged such that all masked data can be displayed. The upper box displays a zero-line for reference, which also indicates the mask.

The Extension provides space to store fit results for each non-spectroscopic coordinate. Say, if you have a 2-D image each row being a spectrum, then you can store results for each row. The whole set of results can be filled successively by fitting one row at a time and always using the same component number to store the results for that row. (See also the example.)

The components fitted by this routine are specified as follows:
The line names and laboratory frequencies are the default values and are not checked against any existing information in the input's Specdre Extension. The component types are 'Planck'. The numbers of parameters allocated to each component are 3, the three guessed and fitted parameters. The parameter types are in order of appearance: 'Theta', 'alpha', 'lg(T)'.

Examples: fitbb in device=xw mask1=10.5 mask2=14.5
          ncomp=1 theta=0.5 alpha=0 lgtemp=3.5 sf=0 af=1 tf=0
          comp=1 logfil=planck
This fits a Planck curve to the range of frequencies between about 30 GHz and 3E14 Hz. The temperature is guessed to be 3000 K. The fit result is reported to the text file PLANCK and stored as component number 1 in the input file's Specdre Extension.
Since DIALOG is not turned off, the user will be prompted for improvements of the mask and guess, and will be asked whether the final fit result is to be accepted (stored in the Extension and written to planck).
The xwindows graphics device will display the spectrum before masking, guessing, and fitting. Independent of the DIALOG switch, a plot is produced after fitting.
Notes: This routine recognises the Specdre Extension v. 0.7. This routine works in situ and modifies the input file.

References: Meyerdierks, H., 1992a, Covariance in resampling and model fitting, Starlink, Spectroscopy Special Interest Group

D.90 FITCONT—Fits a Chebyshev polynomial to the continuum for 2D data

Description: As with VIG, lines can be excluded from the polynomial fitting. FITCONT stores the polynomial fitting coefficients in the actual data file, for use by LONGSLIT (the program is specifically for use with LONGSLIT, and of no use otherwise).

Parameters: IMAGE IMAGE = FILE (Read) Input file
XSECT XSECT = INTEGER (Read) Cross-section to take first cut from

Source comments: available

D.91 FITGAUSS—Fit Gauss profiles to a spectrum.


Description: This routine fits up to six Gauss profiles at a time to a one-dimensional data set. This can be specified as an NDF section. The data set must extend along the spectroscopic axis.

Parameters: INFO INFO = _LOGICAL (Read) If false, this routine will issue only error messages and no informational message. [YES]
VARUSE VARUSE = _LOGICAL (Read) If false, input variances are ignored. [YES]
DIALOG DIALOG = _CHAR (Read) If 'T', the routine offers in general more options for interaction. The mask or guess can be improved after inspections of a plot. Also, the routine can resolve uncertainties about where to store results by consulting the user. ['T']
IN IN = NDF (Read) The input NDF. This must be a one-dimensional (section of an) NDF. You can specify e.g. an image column as IN(5,) or part of an image row as IN(2:3,3,10). Update access is necessary to store the fit result in the NDF's Specdre Extension.
REPAIR REPAIR = _LOGICAL (Read) If DIALOG is true, REPAIR can be set true in order to change the spectroscopic number axis in the Specdre Extension. [NO]
DEVICE DEVICE = DEVICE (Read) The name of the plot device. Enter the null value (!) to disable plotting. [!]
MASK1 MASK1( 6 ) = _REAL (Read) Lower bounds of mask intervals. The mask is the part(s) of the spectrum that is (are) fitted and plotted. The mask is put together from up to six intervals:
mask = [MASK1(1);MASK2(1)] U [MASK1(2);MASK1(2)] U ... U [MASK1(MSKUSE);MASK2(MSKUSE)]
The elements of the MASK parameters are not checked for monotony. Thus intervals may be empty or overlapping. The number of intervals to be used is
derived from the number of lower/upper bounds entered. Either MASK1 or MASK2 should be entered with not more numbers than mask intervals required.

**MASK2** MASK2( 6 ) = _REAL (Read) Upper bounds of mask intervals. See MASK1.

**NCOMP** NCOMP = _INTEGER (Read) The number of Gauss profiles to be fitted. Must be between 1 and 6. [1]

**CONT** CONT = _REAL (Read) This value indicates the level of the continuum. Any constant value for CONT is acceptable. [0]

**CENTRE** CENTRE( 6 ) = _REAL (Read) Guess centre position for each Gauss component.

**PEAK** PEAK( 6 ) = _REAL (Read) Guess peak height for each Gauss component.

**FWHM** FWHM( 6 ) = _REAL (Read) Guess full width at half maximum for each Gauss component.

**CF** CF( 6 ) = _INTEGER (Read) For each Gauss component I, a value CF(I)=0 indicates that CENTRE(I) holds a guess which is free to be fitted. A positive value CF(I)=I indicates that CENTRE(I) is fixed. A positive value CF(I)=J<I indicates that CENTRE(I) has to keep a fixed offset from CENTRE(J).

**PF** PF( 6 ) = _INTEGER (Read) For each Gauss component I, a value PF(I)=0 indicates that PEAK(I) holds a guess which is free to be fitted. A positive value PF(I)=I indicates that PEAK(I) is fixed. A positive value PF(I)=J<I indicates that PEAK(I) has to keep a fixed ratio to PEAK(J).

**WF** WF( 6 ) = _INTEGER (Read) For each Gauss component I, a value WF(I)=0 indicates that FWHM(I) holds a guess which is free to be fitted. A positive value WF(I)=I indicates that FWHM(I) is fixed. A positive value WF(I)=J<I indicates that FWHM(I) has to keep a fixed ratio to FWHM(J).

**REMASK** REMASK = _LOGICAL (Read) Reply YES to have another chance for improving the mask. [NO]

**REGUESS** REGUESS = _LOGICAL (Read) Reply YES to have another chance for improving the guess and fit. [NO]

**FITGOOD** FITGOOD = _LOGICAL (Read) Reply YES to store the result in the Specdre Extension. [YES]

**COMP** COMP = _INTEGER (Read) The results are stored in the Specdre Extension of the data. This parameter specifies which existing components are being fitted. You should give NCOMP values, which should all be different and which should be between zero and the number of components that are currently stored in the Extension. Give a zero for a hitherto unknown component. If a COMP element is given as zero or if it specifies a component unfit to store the results of this routine, then a new component will be created in the result storage structure. In any case this routine will report which components were actually used and it will deposit the updated values in the parameter system. [1,2,3,4,5,6]

**LOGFIL** LOGFIL = FILENAME (Read) The file name of the log file. Enter the null value (!) to disable logging. The log file is opened for append. [!]

**FCENTRE** FCENTRE( 6 ) = _REAL (Write) Fitted centre position for each Gauss component.

**FPEAK** FPEAK( 6 ) = _REAL (Write) Fitted peak height for each Gauss component.

**FFWHM** FWHM( 6 ) = _REAL (Write) Fitted full width at half maximum for each Gauss component.
After accessing the data and the (optional) plot device, the data will be subjected to a mask that consists of up to six abscissa intervals. These may or may not overlap and need not lie within the range of existing data. The masking will remove data which are bad, have bad variance or have zero variance. The masking will also provide weights for the fit. If the given data have no variances attached, or if the variances are to be ignored, all weights will be equal.

After the data have been masked, guessed values for the fit are required. These are

- the number of components to be fitted,
- the value of any underlying constant continuum (this must be an a-priori known constant),
- the components’ guessed centre positions,
- peak heights and
- full widths at half maxima. Finally,
- fit flags for each of the Gauss parameters are needed.

The fit flags specify whether any parameter is fixed, fitted, or kept at a constant ratio or offset to another fitted parameter.

The masked data and parameter guesses are then fed into the fit routine. Single or multiple Gauss fits are made to line features. Gauss fit parameters may be free, fixed, or tied to the corresponding parameter of another Gauss component fitted at the same time. Peak and width are tied by fixing the ratios, the centre is tied by fixing the offset. Up to six Gauss components can be fitted simultaneously.

The fit is done by minimising chi-squared (or rms if variances are unavailable or are chosen to be ignored). The covariances between fit parameters - and among these the uncertainties of parameters - are estimated from the curvature of psi-squared. psi-squared is usually the same as chi-squared. If, however, the given data are not independent measurements, a slightly modified function psi-squared should be used, because the curvature of chi-squared gives an overoptimistic estimate of the fit parameter uncertainty. In that function the variances of the given measurements are substituted by the sums over each row of the covariance matrix of the given data. If the data have been re-sampled with a Specdre routine, that routine will have stored the necessary additional information in the Specdre Extension, and this routine will automatically use that information to assess the fit parameter uncertainties. A full account of the psi-squared function is given in Meyerdierks, 1992a/b. But note that these covariance row sums are ignored if the main variance is ignored or unavailable.

If the fit is successful, then the result is reported to the standard output device and plotted on the graphics device. The final plot view port is saved in the AGI data base and can be used by further applications.
The result is stored in the Specdre Extension of the input NDF. Optionally, the complete description (input NDF name, mask used, result, etc.) is written (appended) to an ASCII log file.

Optionally, the application can interact with the user. In that case, a plot is provided before masking, before guessing and before fitting. After masking, guessing and fitting, a screen report and a plot are provided and the user can improve the parameters. Finally, the result can be accepted or rejected, that is, the user can decide whether to store the result in the Specdre Extension or not.

The screen plot consists of two view ports. The lower one shows the data values (full-drawn bin-style) overlaid with the guess or fit (dashed line-style). The upper box shows the residuals (cross marks) and error bars. The axis scales are arranged such that all masked data can be displayed. The upper box displays a zero-line for reference, which also indicates the mask.

The Extension provides space to store fit results for each non-spectroscopic coordinate. Say, if you have a 2-D image each row being a spectrum, then you can store results for each row. The whole set of results can be filled successively by fitting one row at a time and always using the same component number to store the results for that row. (See also the example.)

The components fitted by this routine are specified as follows: The line names and laboratory frequencies are the default values and are not checked against any existing information in the input’s Specdre Extension. The component types are ‘Gauss’. The numbers of parameters allocated to each component are 4, the three guessed and fitted parameters and the line integral. The parameter types are in order of appearance: ‘centre’, ‘peak’, ‘FWHM’, ‘integral’.

**Examples:** fitgauss in device=xw mask1=-1.5 mask2=2.5
  ncomp=1 cont=1.0 centre=0.5 peak=-0.5 fwhm=1.5 cf=0 pf=0 wf=0
  comp=1 logfil=line

This fits a single Gauss profile to the x range [-1.5,2.5]. The continuum is assumed to be constant at 1.0. The Gauss is guessed to be centred at 0.5 with width 1.5. It is guessed to be an absorption line with an amplitude of -0.5. All Gauss parameters are free to be fitted. The fit result is reported to the text file line and stored as component number 1 in the input file’s Specdre Extension. Since DIALOG is not turned off, the user will be prompted for improvements of the mask and guess, and will be asked whether the final fit result is to be accepted (stored in the Extension and written to line).

The xwindows graphics device will display the spectrum before masking, guessing, and fitting. Independent of the DIALOG switch, a plot is produced after fitting.

`fitgauss in(,5) device=! mask1=-1.5 mask2=2.5`
This fits a single Gauss profile to the x range [-1.5,2.5] of the 5th row in the 2-D image IN. The baseline is assumed to be constant at 0.0. The Gauss is guessed to be centred at 0.5 with width 1.5. It is guessed to be an emission line with an amplitude of 13. Centre position and peak height are free to be fitted, but the width is fixed to 1.5. User interaction (DIALOG) and plotting (DEVICE) are de-selected. There is also no log file where to the results are written. If INFO were also switched off, no report whatsoever would be made. However, the results are stored as a new component (COMP=0) in the Specdre Extension of the input file.

Notes: This routine recognises the Specdre Extension v. 0.7.
This routine works in situ and modifies the input file.

References: Meyerdierks, H., 1992a, Covariance in resampling and model fitting, Starlink, Spectroscopy Special Interest Group

D.92 FITPOLY-Fit a polynomial to a spectrum.


Description: This routine fits a polynomial to a one-dimensional data set. This can be specified as an NDF section. The data set must extend along the spectroscopic axis.

Parameters: INFO INFO = _LOGICAL (Read) If false, the routine will issue only error messages and no informational messages. [YES]
VARUSE VARUSE = _LOGICAL (Read) If false, input variances are ignored. [YES]
DIALOG DIALOG = _CHAR (Read) If 'T', the routine offers in general more options for interaction. The mask or guess can be improved after inspections of a plot. Also, the routine can resolve uncertainties about where to store results by consulting the user. ['T']
IN IN = NDF (Read) The input NDF. This must be a one-dimensional (section of an) NDF. You can specify e.g. an image column as IN(5,) or part of an image row as IN(2.2:3.3,10). Update access is necessary to store the fit result in the NDF’s Specdre Extension.
REPAIR REPAIR = _LOGICAL (Read) If DIALOG is true, REPAIR can be set true in order to change the spectroscopic number axis in the Specdre Extension. [NO]
DEVICE DEVICE = DEVICE (Read) The name of the plot device. Enter the null value (!) to disable plotting. [!] 
MASK1 MASK1( 6 ) = _REAL (Read) Lower bounds of mask intervals. The mask is the part(s) of the spectrum that is (are) fitted and plotted. The mask is put together from up to six intervals:
mask = [MASK1(1);MASK2(1)] U [MASK1(2);MASK1(2)] U ... U [MASK1(MSKUSE);MASK2(MSKUSE)]
The elements of the MASK parameters are not checked for monotony. Thus intervals may be empty or overlapping. The number of intervals to be used is
derived from the number of lower/upper bounds entered. Either MASK1 or MASK2 should be entered with not more numbers than mask intervals required.

**MASK2**  
MASK2( 6 ) = _REAL (Read) Upper bounds of mask intervals. See MASK1.

**ORDER**  
ORDER = _INTEGER (Read) The polynomial order of the fit. Must be between 0 and 7. [1]

**REMASK**  
REMASK = _LOGICAL (Read) Reply YES to have another chance for improving the mask. [NO]

**REGUESS**  
REGUESS = _LOGICAL (Read) Reply YES to have another chance for improving the guess and fit. [NO]

**FITGOOD**  
FITGOOD = _LOGICAL (Read) Reply YES to store the result in the Specdre Extension. [YES]

**COMP**  
COMP = _INTEGER (Read and Write) The results are stored in the Specdre Extension of the data. This parameter specifies which existing component is being fitted. It should be between zero and the number of components that are currently stored in the Extension. Give zero for a hitherto unknown component. If COMP is given as zero or if it specifies a component unfit to store the results of this routine, then a new component will be created in the result storage structure. In any case this routine will report which component was actually used and it will deposit the updated value in the parameter system. [1]

**LOGFIL**  
LOGFIL = FILENAME (Read) The file name of the log file. Enter the null value (!) to disable logging. The log file is opened for append. [!]

Source comments:  
T P O L Y

After accessing the data and the (optional) plot device, the data will be subjected to a mask that consists of up to six abscissa intervals. These may or may not overlap and need not lie within the range of existing data. The masking will remove data which are bad, have bad variance or have zero variance. The masking will also provide weights for the fit. If the given data have no variances attached, or if the variances are to be ignored, all weights will be equal.

The masked data are then fed into the fit routine. The highest polynomial order possible is 7. The fit weights data points according to their errors. The coefficients reported are those of an ordinary polynomial. Let \((x,y)\) be the measurements, \(y(x)\) be the polynomial of order \(n\) fitting the measurements, \(c_i (i = 1, \ldots, n+1)\) be the fitted coefficients. Then \(y(x)\) can be calculated as

\[ y(x) = c_1 + c_2 x + c_3 x^{*2} + \ldots + c_{(n+1)} x^{*n} \]

If the fit is successful, then the result is reported to the screen and plotted on the graphics device. The final plot view port is saved in the AGI data base and can be used by further applications.

The result is stored in the Specdre Extension of the input NDF. Optionally, the complete description (input NDF name, mask used, result, etc.) is written (appended) to an ASCII log file.
Optionally, the application can interact with the user. In that case, a plot is provided before masking and before specifying the polynomial order. After masking and fitting, a screen report and a plot (optional) are provided and the user can improve the parameters. Finally, the result can be accepted or rejected, that is the user can decide whether to store the result in the Specdre Extension or not.

The screen plot consists of two view ports. The lower one shows the data values (full-drawn bin-style) overlaid with the fit (dashed line-style). The upper box shows the residuals (cross marks) and error bars. The axis scales are arranged such that all masked data can be displayed. The upper box displays a zero-line for reference, which also indicates the mask.

The Extension provides space to store fit results for each non-spectroscopic coordinate. Say, if you have a 2-D image each row being a spectrum, then you can store results for each row. The whole set of results can be filled successively by fitting one row at a time and always using the same component number to store the results for that row. (See also the example.)

The component fitted by this routine is specified as follows: The line name and laboratory frequency are the default values and are not checked against any existing information in the input’s Specdre Extension. The component type is ‘polynomial’. The number of parameters allocated to the component is 9. The parameter types are in order of appearance: ‘order’, ‘coeff0’, ..., ‘coeff7’. Unused coefficient are stored as zero.

Examples:

```
fitpoly in device=! mask1=2.2 mask2=3.3 order=3 comp=1 logfil=! 
IN is a 1-D NDF. A 3rd order fit is made to the abscissa range 
between 2.2 and 3.3. The result is stored in component number 1 
of the result structure in the Specdre Extension of IN. The 
plot device and ASCII log file are de-selected.
```

```
fitpoly in(,15) device=xw mask1=[2.0,2.3,3.4] mask2=[2.1,3.2,4.0] 
order=2 comp=0 logfil=myfil
Here IN is 2-D and the 15th row is selected as the 1-D input 
for the fit. The mask consists of three intervals 
[2.0;2.1] U [2.3;3.2] U [3.4,4.0]. The fit is a parabola. Space 
for a new component is created for storage in the Specdre 
Extension. The plot device is xwindows.
```

```
fitpoly in(,20) device=xw mask1=[2.0,2.3,3.4] mask2=[2.1,3.2,4.0] 
order=4 comp=2 logfil=myfil
In a follow-up from the previous example, now the 20th row is 
fitted with 4th order. If in the previous run the routine told 
us that it had used component number 2, then COMP=2 is what we 
want to use to store a similar fit for a different row. 
The first time round, the description of component 2 was 
created, saying that it is a polynomial with order of 7 
or less etc. And the fit result for the 15th row was stored in 
an array that has space for all rows in the input file.
```
So the second time round, FITPOLY checks whether component 2 is suitable, whether it is a polynomial with maximum order 7. It then stores the new result for the 20th row in the place reserved for this row. Gradually all rows can be fitted and their results stored in the Extension. Possibly this could be automated by writing a looping ICL procedure or shell script. In the end the corresponding results for all rows are stored in one data structure, and could for example be converted into a plot of the n-th parameter value versus row number.

Notes: This routine recognises the Specdre Extension v. 0.7. This routine works in situ and modifies the input file.

D.93 FITSET-Set the value of a FITS keyword

Description: FIGARO program to set (or modify) a FITS keyword in a Figaro file. This program accepts the name of a single FITS keyword and a new value for it. If the keyword already exists it will be changed, unless it is one of the special keywords ("HISTORY", "COMMENT", or 'blank') that can have multiple values, in which case the new value will be added to those already in the file.

Parameters: FILE The name of the Figaro file one of whose keywords is to be changed.

KEYWORD The name of the FITS keyword to be set or modified. If the keyword already exists it will be listed by the program. Note that FITS keyword names should be limited to 8 characters in length, but this is not enforced by this program. Note: if you set the value of one of the standard keywords that can be deduced from the rest of the data in the file, such as 'NAXIS' or 'CRDELTn' or 'BITPIX' you may get confusing results. This is not recommended.

VALUE The new value for the FITS keyword specified. This is specified as a character string, so may need to be given enclosed by quotes. Note that the FITS standard limits character values to upper case, so any lower case characters will be folded to upper case. If the value is to be treated as a logical value, it should be either 'T' or 'F'. Numeric and logical values are treated slightly differently to the way string values are treated. By default, if the value given can be interpreted as a number by FITSET then it will be; otherwise it will be treated as a literal string. This can be overriden by the STRING and LOGICAL keywords.

COMMENT Each FITS keyword has a comment string associated with it. COMMENT supplies the comment string to be associated with the KEYWORD being set. If a keyword already exists, then the existing comment (if any) is used as the default comment.

LOGICAL If LOGICAL is set, then FITSET will treat the supplied value as a logical one. In this case the string supplied for VALUE must have been either 'T' or 'F'. Logical values are stored internally in FITS files slightly differently to the way string values are treated. In most cases, this doesn’t matter much, but systems other than Figaro may be fussy about the types of specific keywords.

STRING If STRING is set, then FITSET will treat the supplied value as a literal character string, even if it can be interpreted as a number. Numeric values are stored internally in FITS files slightly differently to the way string values are
treated. In most cases, this doesn’t matter much, but systems other than Figaro may be fussy about the types of specific keywords.

Source comments:  SET

Function:  Figaro routine to set (or modify) a FITS keyword in a file.

Description:  This routine allows a FITS keyword in a file to be set or, if it already exists, to be modified. This routine is needed mainly because of the difficulty of changing items in a FITS header when the file in question is in NDF format. (A .DST file has the FITS information in separate structure items which can easily be modified using LET, but an NDF format file has all the FITS keywords in a single character array which is not amenable to such changes.)

Invocation:  FITSET file keyword value comment [logical] [string]

Parameters:  
file (Filename) The name of the Figaro format file in which the keyword is to be set.
keyword (Character string) The name of the FITS keyword that is to be set.
value (Character string) The new value of the FITS keyword. If this can be interpreted as a numeric value it will be set as such. Otherwise it will be kept as a character string.
comment (Character string) The comment to be associated with the keyword.
logical (Keyword) If set, forces the value to be treated as a logical value, in which case it must be one of 'T' or 'F' and will be set as such in the output file.
string (Keyword) If set, forces the value to be treated as a literal string; in this case it will not be treated as a number even if it can be.

Keith Shortridge, AAO.

D.94  FITSKEYS-List the FITS keywords in a data file

Description:  FITSKEYS lists all the items in the FITS-specific sub-structure associated with a specified data structure.

Parameters:  INPUT  INPUT can be any Figaro structure. If it has any items in a FITS substructure, these are listed.

See also: FIGARO: RDFITS, WDFITS, KAPPA, FITSDIN, FITSin, FITSHEAD, FITSIMP, FITSLIST

Source comments:  SK E Y S

Name:  FITSKEYS
**Function:**
List the contents of a FITS-specific structure.

**Description:**
This Figaro program lists all the items in the FITS-specific substructure associated with a data structure. It doesn’t give any information that can’t be obtained using `hdstrace`, but it presents it in a more convenient form, especially when the structure contains multiple comment keywords.

**Command Parameters:**

- **INPUT** (Character) Name of structure whose FITS substructure is to be listed.

**Command keywords:** None.

K. Shortridge, AAO

**D.95 FITTRI—Fit triangular profiles to a spectrum.**

**Usage:**

**Description:**
This routine fits up to six triangular profiles at a time to a one-dimensional data set. This can be specified as an NDF section. The data set must extend along the spectroscopic axis.

**Parameters:**

- **INFO**
  INFO = _LOGICAL (Read) If false, this routine will issue only error messages and no informational message. [YES]

- **VARUSE**
  VARUSE = _LOGICAL (Read) If false, input variances are ignored. [YES]

- **DIALOG**
  DIALOG = _CHAR (Read) If ’T’, the routine offers in general more options for interaction. The mask or guess can be improved after inspections of a plot. Also, the routine can resolve uncertainties about where to store results by consulting the user. [’T’]

- **IN**
  IN = NDF (Read) The input NDF. This must be a one-dimensional (section of an) NDF. You can specify e.g. an image column as IN(5,) or part of an image row as IN(2:3,10). **UPDATE** access is necessary to store the fit result in the NDF’s Specdre Extension.

- **REPAIR**
  REPAIR = _LOGICAL (Read) If DIALOG is true, REPAIR can be set true in order to change the spectroscopic number axis in the Specdre Extension. [NO]

- **DEVICE**
  DEVICE = DEVICE (Read) The name of the plot device. Enter the null value (!) to disable plotting. [!]

- **MASK1**
  MASK1(6) = _REAL (Read) Lower bounds of mask intervals. The mask is the part(s) of the spectrum that is (are) fitted and plotted. The mask is put together from up to six intervals:
mask = [MASK1(1);MASK2(1)] U [MASK1(2);MASK1(2)] U ... U [MASK1(MSKUSE);MASK2(MSKUSE)]. The elements of the MASK parameters are not checked for monotony. Thus intervals may be empty or overlapping. The number of intervals to be used is derived from the number of lower/upper bounds entered. Either MASK1 or MASK2 should be entered with not more numbers than mask intervals required.
MASK2  MASK2( 6 ) = _REAL (Read) Upper bounds of mask intervals. See MASK1.
NCOMP  NCOMP = _INTEGER (Read) The number of triangle profiles to be fitted. Must be between 1 and 6. [1]
CONT  CONT = _REAL (Read) This value indicates the level of the continuum. Any constant value for CONT is acceptable. [0]
CENTRE  CENTRE( 6 ) = _REAL (Read) Guess centre position for each triangle component.
PEAK  PEAK( 6 ) = _REAL (Read) Guess peak height for each triangle component.
FWHM  FWHM( 6 ) = _REAL (Read) Guess full width at half maximum for each triangle component.
CF  CF( 6 ) = _INTEGER (Read) For each triangle component I, a value CF(I)=0 indicates that CENTRE(I) holds a guess which is free to be fitted. A positive value CF(I)=I indicates that CENTRE(I) is fixed. A positive value CF(I)=J<I indicates that CENTRE(I) has to keep a fixed offset from CENTRE(J).
PF  PF( 6 ) = _INTEGER (Read) For each triangle component I, a value PF(I)=0 indicates that PEAK(I) holds a guess which is free to be fitted. A positive value PF(I)=I indicates that PEAK(I) is fixed. A positive value PF(I)=J<I indicates that PEAK(I) has to keep a fixed ratio to PEAK(J).
WF  WF( 6 ) = _INTEGER (Read) For each triangle component I, a value WF(I)=0 indicates that FWHM(I) holds a guess which is free to be fitted. A positive value WF(I)=I indicates that FWHM(I) is fixed. A positive value WF(I)=J<I indicates that FWHM(I) has to keep a fixed ratio to FWHM(J).
REMASK  REMASK = _LOGICAL (Read) Reply YES to have another chance for improving the mask. [NO]
REGUESS  REGUESS = _LOGICAL (Read) Reply YES to have another chance for improving the guess and fit. [NO]
FITGOOD  FITGOOD = _LOGICAL (Read) Reply YES to store the result in the Specdre Extension. [YES]
COMP  COMP = _INTEGER (Read) The results are stored in the Specdre Extension of the data. This parameter specifies which existing components are being fitted. You should give NCOMP values, which should all be different and which should be between zero and the number of components that are currently stored in the Extension. Give a zero for a hitherto unknown component. If a COMP element is given as zero or if it specifies a component unfit to store the results of this routine, then a new component will be created in the result storage structure. In any case this routine will report which components were actually used and it will deposit the updated values in the parameter system. [1,2,3,4,5,6]
LOGFIL  LOGFIL = FILENAME (Read) The file name of the log file. Enter the null value (!) to disable logging. The log file is opened for append. [!]
variances attached, or if the variances are to be ignored, all weights will be equal.

After the data have been masked, guessed values for the fit are required. These are
- the number of components to be fitted,
- the value of any underlying constant continuum (this must be an a-priori known constant),
- the components' guessed centre positions,
- peak heights and
- full widths at half maxima. Finally,
- fit flags for each of the triangle parameters are needed.

The fit flags specify whether any parameter is fixed, fitted, or kept at a constant ratio or offset to another fitted parameter.

The masked data and parameter guesses are then fed into the fit routine. Single or multiple triangle fits are made to line features. Triangle fit parameters may be free, fixed, or tied to the corresponding parameter of another triangle component fitted at the same time. Peak and width are tied by fixing the ratios, the centre is tied by fixing the offset. Up to six triangle components can be fitted simultaneously.

The fit is done by minimising chi-squared (or rms if variances are unavailable or are chosen to be ignored). The covariances between fit parameters - and among these the uncertainties of parameters - are estimated from the curvature of psi-squared. psi-squared is usually the same as chi-squared. If, however, the given data are not independent measurements, a slightly modified function psi-squared should be used, because the curvature of chi-squared gives an overoptimistic estimate of the fit parameter uncertainty. In that function the variances of the given measurements are substituted by the sums over each row of the covariance matrix of the given data. If the data have been re-sampled with a Spectre routine, that routine will have stored the necessary additional information in the Spectre Extension, and this routine will automatically use that information to assess the fit parameter uncertainties. A full account of the psi-squared function is given in Meyerdierks, 1992a/b. But note that these covariance row sums are ignored if the main variance is ignored or unavailable.

If the fit is successful, then the result is reported to the standard output device and plotted on the graphics device. The final plot view port is saved in the AGI data base and can be used by further applications.

The result is stored in the Spectre Extension of the input NDF. Optionally, the complete description (input NDF name, mask used, result, etc.) is written (appended) to an ASCII log file.

Optionally, the application can interact with the user. In that case, a plot is provided before masking, before guessing and
before fitting. After masking, guessing and fitting, a screen report and a plot are provided and the user can improve the parameters. Finally, the result can be accepted or rejected, that is, the user can decide whether to store the result in the Specdre Extension or not.

The screen plot consists of two view ports. The lower one shows the data values (full-drawn bin-style) overlaid with the guess or fit (dashed line-style). The upper box shows the residuals (cross marks) and error bars. The axis scales are arranged such that all masked data can be displayed. The upper box displays a zero-line for reference, which also indicates the mask.

The Extension provides space to store fit results for each non-spectroscopic coordinate. Say, if you have a 2-D image each row being a spectrum, then you can store results for each row. The whole set of results can be filled successively by fitting one row at a time and always using the same component number to store the results for that row. (See also the example.)

The components fitted by this routine are specified as follows: The line names and laboratory frequencies are the default values and are not checked against any existing information in the input’s Specdre Extension. The component types are ‘triangle’. The numbers of parameters allocated to each component are 4, the three guessed and fitted parameters and the line integral. The parameter types are in order of appearance: ‘centre’, ‘peak’, ‘FWHM’, ‘integral’.

Examples:

```
fittri in device=xw mask1=-1.5 mask2=2.5
   ncomp=1 cont=1.0 centre=0.5 peak=-0.5 fwhm=1.5 cf=0 pf=0 wf=0
   comp=1 logfil=line
```

This fits a single triangular profile to the x range [-1.5,2.5]. The continuum is assumed to be constant at 1.0. The triangle is guessed to be centred at 0.5 with width 1.5. It is guessed to be an absorption line with an amplitude of -0.5. All triangle parameters are free to be fitted. The fit result is reported to the text file LINE and stored as component number 1 in the input file’s Specdre Extension. Since DIALOG is not turned off, the user will be prompted for improvements of the mask and guess, and will be asked whether the final fit result is to be accepted (stored in the Extension and written to line).

The xwindows graphics device will display the spectrum before masking, guessing, and fitting. Independent of the DIALOG switch, a plot is produced after fitting.

```
fittri in(,5) device=! mask1=-1.5 mask2=2.5
   ncomp=1 cont=0.0 centre=0.5 peak=13.0 fwhm=1.5 cf=0 pf=0 wf=1
   comp=0 logfil=! dialog=f
```

This fits a single triangular profile to the x range [-1.5,2.5] of the 5th row in the 2-D image IN. The baseline is assumed to be constant at 0.0. The triangle is guessed to be centred at 0.5 with width 1.5. It is guessed to be an emission line with an amplitude of 13. Centre position and peak height are free to
be fitted, but the width is fixed to 1.5. User interaction (DIALOG) and plotting (DEVICE) are de-selected. There is also no log file where to the results are written. If INFO were also switched off, no report whatsoever would be made. However, the results are stored as a new component (COMP=0) in the Specdre Extension of the input file.

**Notes:** This routine recognises the Specdre Extension v. 0.7.
This routine works in situ and modifies the input file.

**References:** Meyerdierks, H., 1992a, Covariance in resampling and model fitting, Starlink, Spectroscopy Special Interest Group

**D.96 FLAG2QUAL—Converts ‘flagged’ values to produce a quality array**

**Description:** FLAG2QUAL removes ‘flagged’ (or ‘magic’) data values from the main array of a Figaro file. Any such values are replaced either with a single specified data value, or else FLAG2QUAL replaces them with data values obtained by interpolation between the good pixels on each side. (This is not a particularly sophisticated interpolation, but then, these are data values that shouldn’t ever actually be used, since they are still flagged as bad in the quality array.) When FLAG2QUAL replaces a flagged data value it sets the corresponding element of the associated quality array to indicate that the pixel is not to be trusted. If there was already a quality array associated with the data it is used when deciding which pixels to use for interpolation and is updated to reflect those flagged pixels that have been replaced. If the file did not already have a quality array then one is created. Many Figaro routines prefer to use quality arrays when dealing with data, and operate more efficiently if the data is not flagged. Also, there may be some programs that are confused by the presence in the file of flagged data values. Finally, having both forms of data quality information in a file can be confusing and FLAG2QUAL can tidy things up.

**Parameters:**

**INPUT** The name of a Figaro format file that might contain flagged (or ‘magic’) data values. It may also contain an associated quality array. Whatever it contains, FLAG2QUAL will process it so that the resulting file has an array that does not contain any flagged data values. Any information that was held in the form of such flagged data values will be held in a quality array in the resulting file.

**OUTPUT** The name of the resulting file. This can be the same as the input file, in which case all changes are made in situ. The resulting file will have all its data quality information held in a quality array and will have no flagged data values in its main array.

**FIXED** The flagged data values in the input array have to be replaced. If FIXED is set, they are replaced by a single data value specified in the VALUE parameter. If FIXED is not set, then FLAG2QUAL replaces them using values calculated by interpolation between the previous and next good pixels.

**VALUE** If FIXED is set, then VALUE is used to specify the single data value to be used to replace the flagged values in the main data array. This can be any value - although it would be perverse to use the actual flag value!
Source comments: G 2 Q U A L

Description:
This is a Figaro program that removes any 'flagged' values from the main data array in a Figaro data file. If there are in fact such values in the main data array (many arrays are flagged as 'may contain flagged values', but in fact do not) then this routine sets the equivalent elements of an associated quality array (which it may have to create).

Command parameters:

INPUT (Character) The name of the structure containing the data.

OUTPUT (Character) The name of the result of the operation. This can be the same as for INPUT. If not, a new structure is created, with everything but the data a direct copy of the input.

VALUE (Numeric) If a fixed value is to be used to replace flagged data values, this supplies that value.

Command keywords:

FIXED If set, a fixed value (supplied in VALUE) is used to replace the flagged data values. If not, the program interpolates over them.


D.97 FLAIRCOMP-Compresses a FLAIR frame to give a weight vector.

Description: This application takes a FLAIR frame stored in an NDF and compresses it along the y axis, normalises the compressed array by its mean value, and then finds the minima in the values and set these to the bad value. Thus it provides the weights for an optimal extraction. It reports the number of fibres found in the NDF. This assumes stability (x positions of the fibres do not move), and vertical orientation of the fibres. These are satisfied by FLAIR (Parker, private communication).

Usage: flaircomp in out

Parameters: IN The input two-dimensional NDF. This should be the co-added arc or sky flat-field frames, so that the compressed array gives the instrumental response of the detector system.

OUT The vector of weights to use during optimal extraction.

TITLE Value for the title of the output NDF. A null (!) propagates the title from input NDF to the output. [!]

Source comments: available.
D.98 **FLAIREXT**—Optimally extracts spectra from a FLAIR NDF to form a new NDF.

**Description:** This application takes a FLAIR frame stored in an NDF with the dispersion along the y axis, and extracts the individual spectra using an optimal extraction. It stores the spectra in an output two-dimensional NDF, configured such that the dispersion is along x axis, and wavelength increases with pixel index, and each spectrum occupies one line.

This assumes stability (x positions of the fibres do not move), and vertical orientation of the fibres. These are satisfied by FLAIR (Parker, private communication).

**Usage:** flairext in profile out fibres

**Parameters:**
- **FIBRES** The number of fibres to extract. This must be in the range 1 to 92.
- **IN** A list of the input two-dimensional NDFs containing FLAIR spectra to be extracted.
- **PROFILE** The vector of weights to use during optimal extraction, as derived from FLAIRCOMP.
- **OUT** The list of the two-dimensional NDF containing the extracted FLAIR spectra. There should be as many files in this list as for parameter IN. The nth item in this list will be the extracted spectra for the nth file in IN.
- **TITLE** Value for the title of the output NDF. A null value (!) will cause the title of the input NDF to be used. ![null]

**Source comments:** available.

---

D.99 **FLCONV**—Convert a spectrum in Janskys into one in erg/s/cm**2/Angstrom

**Description:** FLCONV generates a spectrum in erg/s/cm**2/Angstrom, given a spectrum in some other units. At present, only Janskys, milli-Janskys and micro-Janskys can be handled.

**Parameters:**
- **SPECTRUM** A spectrum whose units are (currently) either Janskys, milli-Janskys, or micro-Janskys. It needs to be wavelength calibrated, in Angstroms.
- **OUTPUT** The resulting spectrum, whose units will be ergs/s/cm**2/A. OUTPUT can be the same as SPECTRUM, in which case the conversion will be performed in situ.

**Source comments:**

CONV / FLCONV / IRCONV

Converts a spectrum into AB magnitudes (ABCONV) or f-lambda units (erg/s/cm**2/Angstrom) (FLCONV), or W/m**2/um (IRCONV).

The original units of the data may be Jy (Janskys), mJy (milli-Janskys), or uJy (micro-Janskys). Other possibilities may be added later.

**Command parameters** -

SPECTRUM The name of the structure containing the spectrum. Currently used for the spectrum. For FLCONV
an x-axis data structure giving the wavelengths of the data elements is also required.

**OUTPUT** The name of the result of the operation. This can be the same as for SPECTRUM. If not, a new structure is created, with everything but the data a direct copy of the input.

**Command keywords** - None

**User variables used** - None

 KS / CIT 18th May 1984

D.100 FOTO-Perform aperture photometry given CENTERS output

**Description:** The FOTO command performs aperture photometry on objects in an image.

**Parameters:**
- **IMAGE** The name of the image on which the photometry is to be performed. A file giving the centers of the objects in IMAGE to be measured should already have been created, probably by the sequence CPOS, CENTERS.
- **RA1** The inner radius of the circular aperture to be used to calculate the sky values. It should be large enough to clear the objects being measured.
- **RA2** The outer radius of the circular aperture to be used to calculate the sky values.
- **NRADII** The number of circular apertures around the objects being measured for which magnitudes will be calculated.
- **RADII** The values of the radii of the circular apertures, in pixels, around the objects being measured.
- **BAD** The value below which pixels will be ignored.
- **PHOTONS** The number of photons per output unit - i.e. the number of photons actually represented by each count in the data array.
- **READNS** The expected error in the readout when the data was taken. It is only used in the calculation of the error values.
- **MZERO** The offset that has to be applied to the calculated magnitudes in order to get the actual magnitude values. If you don’t care about absolute values, you can leave this as zero.
- **SKYVAL** User-specified sky value.
- **SIGMA** User-specified sigma.
- **DISHIST** Used to ask whether sky histogram to be displayed.
- **CONFIRM** Used to confirm acceptability of sky value and sigma.

**Source comments:**

Main routine for the Figaro aperture photometry function, FOTO. This gets the required parameter values and then leaves the real work to PHOTSUB, which is a modification for Figaro of a routine originally written at KPNO by Don Wells.

**Command parameters -**

- **IMAGE** (Character) The name of the image being analysed.
RA1 (Numeric) The inner radius in pixels of the sky aperture.
RA2 (Numeric) The outer radius in pixels of the sky aperture.
NRADII (Numeric) The number of apertures to use.
RADII (Numeric array) The values of the radii, in pixels.
BAD (Numeric) Bad pixel rejection threshold.
PHOTONS (Numeric) Number of photons per ADU.
READNS (Numeric) Readout noise.
MZERO (Numeric) Magnitude offset.
SKYVAL (Numeric) User-specified sky value.
SIGMA (Numeric) User-specified sigma.

Command keywords -

DISHIST Used to ask whether sky histogram to be displayed.
CONFIRM Used to confirm acceptability of sky value and sigma.

User variables used - (>" input, "<" output)

(>) SOFT (Character) The soft plot device/type, as required by PGPLOT.

Input -

CENTER.DAT contains one record for each point, giving
XCENT,YCENT,IX,IY,DX,DY,AP
in the format 2F8.2,2I5,2F8.2,I4 where
XCENT,YCENT give the position of the centroid
IX,IY are the original pixel position of the point.
DX,DY are the offsets in X and Y, and
AP is the value used for APERTURE.
If the centroid for a point cannot be determined, a
record is written giving
'*** No centroid ',IX,IY,DX,DY,AP
in the format A,2I5,2F8.2,I4.

Output -

MAGS.DAT lists the magnitudes as determined by the program.

KS / CIT 1st June 1983

D.101 FSCRUNCH-Rebin data with a disjoint wavelength coverage to a linear one

Description: FSCRUNCH rebins a spectrum or a set of spectra so that the resulting data have either a linear wavelength scale or a wavelength scale that is logarithmic (i.e. has a constant velocity step). FSCRUNCH is intended for use where the input data have wavelength bins that are discontinuous, or overlapping, or both.

Parameters: SPECTRUM The spectrum (or spectra) to be scrunched. If SPECTRUM is an image (i.e. 2-D data) it will be treated as a set of spectra all of which will be scrunched individually. In this case, if the wavelength data is a 2-D array each of the data spectra will be scrunched according to the corresponding cross-section
of the wavelength array. If the wavelength array is 1-D, this single array will be used for all the spectra.

**WSTART** The wavelength of the center of the first bin of the resulting output data. Note that FSCRUNCH needs the output data to be in increasing wavelength order.

**WEND** Normally, represents the wavelength of the center of the last bin of the resulting output data. Because this is not always the most convenient value to supply, FSCRUNCH will allow WEND to be used to specify the wavelength increment value (for linear data, this is the constant wavelength difference between bins; for logarithmic data it is the constant velocity step in Km/sec). The way WEND is interpreted may be controlled explicitly by the FINAL and INCREMENT hidden keywords, but by default FSCRUNCH will assume WEND is an incremental value if it is less than WSTART, and a final value if it is greater than WSTART.

**BINS** The number of elements to be used for the resulting spectrum.

**INORDER** The order of fit to be used during the scrunching in order to interpolate between input data points. A value of zero means that all the input bins will be treated as flat. Values of 1 or 2 will cause linear or quadratic interpolation to be used. Generally, the best results are obtained with quadratic interpolation.

**OUTPUT** The name of the resulting data file containing the scrunched spectrum or set of spectra. The structure of OUTPUT will be the same as that of the input file, except that the data array will generally be a different size.

**INCREMENT** If set, FSCRUNCH will assume WEND is an incremental value (a velocity or wavelength step)

**FINAL** If set, SCRUNCH will assume that WEND is the wavelength of the final element of the resulting spectrum.

**LOG** Controls whether or not the data is binned to a linear or a logarithmic wavelength scale.

**DENSITY** If the input data represents data whose units are flux per unit wavelength (AB magnitudes, Janskys, etc) rather than total flux over a wavelength range (photons, for example) then it should be scrunched so as to conserve the mean value of the data, and DENSITY should be set. See HELP FIGARO TECHNIQUES WAVELENGTH for more details on this point.

**Source comments:**

F S C R U N C H

Figaro routine to scrunch a spectrum or set of spectra in which the input wavelength ranges for the various pixels are not necessarily continuous, and may overlap. It can scrunch either into a linear wavelength scale, where the wavelength increment from bin to bin is constant across the spectrum, or into a logarithmic scale, where the increment of the log of the wavelength from bin to bin is constant.

If the input file is 2-D data, then it is treated as a set of 1-D spectra and each is scrunched individually. If the wavelength array (.X.DATA) is 1-D, then this single array will be used for all the spectra. If it is 2-D, then each spectrum will be scrunched according to the corresponding cross-section of the wavelength array.
The routine can either conserve flux or the mean value of the data. Conserving flux is appropriate where the data is actually in flux units (photons/sec, for example), but not when the data is in units of flux per unit wavelength (AB magnitudes, Janskys, etc). Consider the case where each input bin maps to two output bins; if the data is in flux units - photon counts, for example - then flux should be conserved and the mean data level should drop by a factor 2; if the data is in magnitudes, then the rebinning should not change the mean level. The program tries to determine for itself whether the data is in flux or in flux per unit wavelength by looking at the units, but uses a command keyword (DENSITY) to confirm its guess.

Command parameters -

SPECTRUM (Character) The name of the spectrum to be scrunched.
WSTART (Numeric) The wavelength of the CENTER of the first bin of the resulting scrunched spectrum.
WEND (Numeric) The wavelength of the CENTER of the final bin of the resulting scrunched spectrum. If WEND is less than WSTART, then FSCRUNCH assumes that it is the increment rather than the final value that is being specified. If the scrunch is logarithmic and WSTART is greater than WEND, FSCRUNCH assumes that the WEND value represents a velocity in km/sec. These assumptions can be controlled directly by the keywords INCREMENT and FINAL, if they will not give the desired effect.
BINS (Numeric) The number of bins for the resulting spectrum.
INORDER (Numeric) The order of local fit to be used for the input data. Can be 0, 1 or 2.
OUTPUT (Character) The name of the resulting spectrum. Note that FSCRUNCH cannot rebin a spectrum into itself and so will always create a new output file.

Command keywords -

LOG Bin into logarithmic wavelength bins.
DENSITY Treat input data as being in units of flux per unit wavelength.
INCREMENT WEND is an increment value, even though it is > WSTART.
FINAL WEND is a final value, even though it is < WSTART.

User variables -

SCRUCH_INC Is set to the wavelength increment if linear rebinning is used, and to the velocity increment if log rebinning is used.
SCRUNCH_END Is set to the final wavelength value. (This is for those cases where the WEND value represents an increment.)

Input data -
The input file is expected to contain a data array giving the wavelengths of the centres of the data elements, and an width specification which can be either a single number, a 1-D array, or a 2-D array, giving the wavelength range covered by each of the input data elements. If an error array exists this will be used as well.

KS / AAO 17th June 1986

D.102 FWCONV-General unit conversion for spectra

Description: FWCONV converts the flux units of a spectrum from AB magnitudes, Janskys, milli-Janskys, micro-Janskys or erg/s/cm**2/Angstrom to any of these.

Parameters: SPECTRUM A spectrum whose units are either Janskys, milli-Janskys, micro-Janskys, erg/s/cm**2/Angstrom or AB magnitudes.
FLUX_UNIT Units of either Janskys, milli-Janskys, micro-Janskys, erg/s/cm**2/Angstrom or AB magnitudes.
OUTPUT The resulting spectrum, whose units will be either Janskys, milli-Janskys, micro-Janskys, erg/s/cm**2/Angstrom or AB magnitudes. OUTPUT can be the same as SPECTRUM, in which case the conversion will be performed in situ.

Source comments:

Converts flux units of a spectrum from/to either F_lambda (ergs/sec/cm**2/A), AB magnitudes or F_nu (Janskys).

Command parameters -

SPECTRUM The name of the structure containing the spectrum. An x-axis data structure giving the wavelengths of the data elements is also required.

OUTPUT The name of the result of the operation. This can be the same as for SPECTRUM. If not, a new structure is created, with everything but the data a direct copy of the input.

Command keywords - None

User variables used - None

4th Feb 1991. SMH/ AAO. Original version, based on ABCONV code,

D.103 GAUSS-Interactive fit of Gaussians to emission or absorption lines

Description: The GAUSS command will plot a spectrum on the current soft graphics device and allow interactive Gaussian fitting of upto ten Gaussians on a polynomial continuum.

Parameters: TOL The tolerance on the value of the optimised function required by the NAG routine E04JBF.
SPECTRUM The name of the spectrum to be plotted and analysed. It should be a 1-dimensional array
FILE  Yes if results of the continuum and Gaussian fitting are to be written to a data file
FILNAM The name for the data file to which results of the continuum and Gaussian fit are written
WHOLE  If set, the whole of the spectrum is plotted. Otherwise, the limits plotted are determined by the values of XSTART and XEND, which you will be prompted for if they were not specified in the command string.
AUTOSCALE If set, the plot is scaled so that all of the data to be plotted just fits on the display. Otherwise, the scale of the plot is determined by the values of HIGH and LOW, which you will be prompted for if they were not specified in the command string.
XSTART Specifies the first X value to be plotted, in the units used by the data - angstroms, for example, if the data is wavelength calibrated. XSTART can be set before the start of the data, if necessary. RESET will set XSTART to the first X value in the data.
XEND Specifies the last X value to be plotted, in the units used by the data - angstroms, for example, if the data is wavelength calibrated. XEND can be set after the end of the data, if necessary. RESET will set XEND to the last X value in the data.
HIGH The maximum data value to be plotted - i.e. the top Y axis value for the plot.
LOW  The minimum data value to be plotted - i.e. the bottom Y axis value for the plot.
BIAS A bias value applied to the data, usually to bias up a plot which is to be superimposed upon a previous plot of almost identical data values. This makes the comparison easier. BIAS N is essentially equivalent to setting HIGH and LOW down by an amount N, so can result in unexpected axis values if it is not accompanied by AXES=NO.
LABEL The label that will appear at the top of the plot and the results data file.
ORD The order of the least squares polynomial fit to the continuum (maximum = 7).
SIG The factor * the sigma on the last polynomial continuum fit such that a point with a deviation greater than SIG * sigma from the last fitted continuum will not be included in the next fit.
ERR The factor * the error on a point such that a point with a deviation greater than ERR * error from the fitted continuum will not be included in the next fit.
ITN The number of iterations for performing the polynomial fit to the continuum with rejection of points with large deviations from the fit (maximum = 10).
INDEX The index number ( as listed ) of the Gaussian whose parameters are to be altered.
DELE The index number ( as listed ) of the Gaussian which is to be deleted.
Pn_Hn_Wn_S P followed by number -100 to 100 to alter position. ( 1 unit = line extent/100 ) H followed by number -100 to 100 to alter height. ( 1 unit = line height/100 ) W followed by number -100 to 100 to increase width. ( 1 unit = line extent/100 ) S to plot modified line and return to menu.
CONSTR The fitted Gaussian parameters (peak position, peak height or line width) can be singly constrained or chained to vary together in groups.
SINCON  The fit parameters (peak position, peak height and width) of any of the
Gaussian components can be constrained so as not to vary in the optimization
process.

NPCON  The Gaussian number n can be constrained to the manually fitted value of
peak position (P), height (H) or width (W). A value of -1 indicates the end of the
single constraints.

MULTCON  Sets of Gaussians can have their fit parameters constrained so that the
values are “chained” to vary together. The line separations of Gaussians can be
set; the ratio of their peak heights can be fixed and the ratio of their line widths
can be fixed.

NCHAIN  NCHAIN is the number of Gaussians to chain together so that line sepa-
ration, height or width varies together. Must be less than or equal to the number of
fitted Gaussians. A value of -1 indicates that there are no more chains to input.

CHAIN  Gaussian parameter to be chained: - P to keep Gaussian line separa-
tions fixed; H to keep peak heights in specific ratio; W to keep Gaussian widths varying
in ratio.

ICHAIN  ICHAIN is the index number of the Gaussian which is to be chained.

RCHAIN  RCHAIN is the value of the seperation between two chained Gaussians,
the ratio of the heights of two chained Gaussians or the ratio of their line widths.

WGHT  Weighting of the residuals to be used in the optimization.
    Options are: N for no weighting V for weighting by Y value E for weighting by
1/error**2

MAN  If set, the results of the manual fit will be written to the terminal and the
results file (if open). If not set, then a return to the menu is made so the fit can be
altered.

RECNAME  The name of the data file produced by GAUSS from which a fit is to be
recalled and plotted for subsequent manipulation.

GAUFIT  The name of the continuum and Gaussian fit spectrum file to be written. It
has the same dimension as the input spectrum.

CCMD  Continuum fitting menu entry. Alternatives are:
    CUR - indicate continuum regions by cursor. ORD - order of polynomial fit. SIG
    - factor*sigma on fit for continuum point rejection. ERR - factor*error on point
    for rejection from continuum fit. ITN - number of iterations for continuum point
    rejection. FIT - perform the polynomial fitting. GAU - proceed to Gaussian fitting
    (continuum set).

GCMD  Gaussian fitting menu entry. Alternatives are:
    LIM _ delimit edges of line (default is adjacent continuum edges). SIN - fit a single
    Gaussian to an indicated line. NEW - introduce a Gaussian at the cursor defined
    position. NEX - introduce a new Gaussian at the peak. INCH - interactively
    alter peak position, height or width. LIS - list the Gaussians fitted. SEL - select
    a line to be modified. DEL - delete a selected Gaussian. OPT - optimize the fit.
    RECAL - read a previous fit data file and plot this fit. HARD - plot results of
    fit for hardcopy device. SAVE - save the Gaussian fit spectrum as a file (name
    prompted for on quitting). CON - move to another section of continuum for
    more fitting. QUIT - quit from program (spectrum analysis complete).

Source comments:  S S
Interactively fits Gaussians to an emission or absorption feature in a spectrum. A spectrum, whose length is prompted for, is plotted on the selected 'SOFT' device together with a residual plot above. Regions of continuum are demarcated with the cursor. Continuum is fitted by a polynomial with rejection of points a selected sigma from this continuum or a fraction of the error bar (if available). Single or multiple Gaussian fits are made to the emission/absorption feature. In the case of multiple fits interactive fitting of the profile by Gaussians whose parameters are altered from the terminal occurs. The residuals on the fit (observed - fitted) are plotted in the residuals box above the main plot. If error bars are available for the spectrum they are plotted in the residuals box. The fit may be optimized and any single Gaussian parameter can be constrained or its value chained to another. The residuals on the fit for the purposes of the optimization can be weighted either by value or error. Data on the fit - central wavelength, height and sigma - are reported for each Gaussian and r.m.s. and mean fractional error (if errors available) on fit. The results may be recorded on a data file. The final fit spectrum can be saved as a file for subsequent analysis.

Command parameters -

SPECTRUM The data to be fitted. This should be a 1-d .dst file with a .z.data component. If there is a .x.data component this will be used to give the x-axis. If not, the x-axis will just have to be the numbers from 1 to n.

XSTART The x-value at which plotting is to start.

XEND The x-value at which plotting is to end. (XSTART and XEND are not required if the WHOLE keyword is set.)

HIGH The maximum value to be used for the plot.

LOW The minimum value to be used for the plot.

BIAS Value to be added to zero for the plot.

LABEL A label for the plot and the data file.

Command keywords -

AUTOSCALE The program is to work out the values for HIGH and LOW, using the maximum and minimum values in the data over the specified range.

WHOLE The program is to display all of the spectrum.

User variables - ("<" input, ">" output)

(>) SOFT Specifies the device and type to be used for soft plots. See the SOFT command for more details.

(>) HARD Specifies the device and type to be used for hard plots. See the HARD command for more details.

(<) TVXST is set to the starting x-value for the plot.

(<) TVXEN Is set to the final x-value for the plot.

(<) TVHIGH Is set to the same value as HIGH.
(<> TVLOW Is set to the same value as LOW.
(<> TVFILE Is set to the value of SPECTRUM.

Subroutines called:
MAPSPEC - unmaps the X and Z from virtual memory into arrays in the program
MAPSPECE - unmaps the X, Z and error data from virtual memory into arrays in the program
MAPSPECG - maps the output fitting spectrum Z component into virtual memory
GAUS_XZPLOT - handles all the PGPLOT plotting of spectrum, continuum fit, Gaussian fit, residuals and error bars
CONMENU - the continuum fitting menu for polynomial continuum fitting
GAUMENU - the Gaussian fitting menu for all Gaussian fitting

D.104 GOODVAR—Replace negative, zero and bad variance values

Usage: goodvar in out bad neg zero

Description: This routine checks the variance component of a datafile for values that are bad, negative, or zero and replaces them by values specified by the user. The specified value can be the null value ("!") which is translated into the bad value.

Parameters: IN The input datafile.
OUT The output datafile.
BAD The value which replaces bad values. Enter an exclamation mark to keep bad values. Bad values in VARIANCE or ERRORS are not allowed by Figaro. If DSA has to convert these arrays while mapping them, floating overflows or square roots of negative numbers may occur, and the application is liable to crash. [!] NEG The value which replaces negative values. Enter an exclamation mark to replace negative values with the bad value. Negative errors or variances are nonsense. Negative variances often will cause an application to crash because it takes the square root to calculate the error. [!] ZERO The value which replaces zeroes. Enter an exclamation mark to replace zeroes with the bad value. Errors of zero sometimes are reasonable or necessary for error propagation. In other instances they cause problems, because statistical weights often are the reciprocal of the variance. [!] 

Authors: HME: Horst Meyerdierks (UoE, Starlink)

D.105 GROW—Copy an N-dimensional cube into part of an (N+M)-dimensional one.

Usage: grow in expand stapix endpix size=? out=?

Description: This routine increases the number of axes of a data set by duplicating pixels along some axes while retaining other axes. A simple and common example is turning a single row into a set of identical rows or a set of identical columns. This routine
copies an N-dimensional cube into (part of) an (N+M)-dimensional one. The input cube is in general copied several times into the output, but need not fill the output cube. If the output file is new, its size has to be given. If it is an existing file, it cannot be reshaped, the axes of input and output have to be consistent.

**Parameters:**

- **INFO**
  - INFO = _LOGICAL (Read)
  - If false, the routine will issue only error messages and no informational messages. [YES]

- **NEW**
  - NEW = _LOGICAL (Read)
  - True if a new output file is to be created. [NO]

- **IN**
  - IN = NDF (Read)
  - Input NDF.

- **EXPAND**
  - EXPAND( 7 ) = _INTEGER (Read)
  - For each axis in OUT a 0 indicates that this is an axis with a correspondent in IN. A 1 indicates that it is a new (or expanded axis without correspondent in IN.

- **STAPIX**
  - STAPIX( 7 ) = _INTEGER (Read)
  - There is an EXPAND vector parameter that indicates which axes in OUT are new or have a corresponding axis in IN. Here, for each axis in OUT the value indicates where the multiple copy of input should start. Only the values for new axes in OUT are relevant, but a value for each axis in OUT must be supplied. The number of STAPIX elements given must match the number of axes in OUT.

- **ENDPIX**
  - ENDPIX( 7 ) = _INTEGER (Read)
  - There is an EXPAND vector parameter that indicates which axes in OUT are new or have a corresponding axis in IN. Here, for each axis in OUT the value indicates where the multiple copy of input should end. Only the values for new axes in OUT are relevant, but a value for each axis in OUT must be supplied. The number of ENDPIX elements given must match the number of axes in OUT.

- **SIZE**
  - SIZE( 7 ) = _INTEGER (Read)
  - For each axis in OUT a 0 indicates that the axis is to be taken from IN, an integer greater than 1 indicates that the axis is a new one and that the SIZE value is to be the length of that axis. The number of SIZE elements given must match the number of axes in OUT. The number of zeros given must be the number of axes in IN.

- **OUT**
  - OUT = NDF (Read)
  - Output NDF, containing the expanded data set.

**Examples:**

```fortran
grow spectrum [0,1] [0,1] [0,5] size=[0,5] out=image new=t info=f
Grows a spectrum into an image of 5 identical rows. It forces
the creation of a new output file even if IMAGE exists.
Informational messages are suppressed.
```

```fortran
grow spectrum [1,0] [2,0] [4,0] out=image
Grows a spectrum into an image of 3 identical columns. Column 1
and columns beyond 4 in IMAGE remain unchanged. Since NEW is
not specified, IMAGE must already exist. Its second axis must
match the first axis of SPECTRUM, and its first axis must be
at least 4 pixels long.
```

```fortran
grow spectrum [0,1,1] [0,1,1] [0,2,4] out=cube size=[0,4,8] new=t
Grow the spectrum into a cube with the spectral axis the 1st
cube axis.
```

```fortran
grow spectrum [1,0,1] [1,0,1] [2,0,4] out=cube size=[4,0,8] new=t
Grow the spectrum into a cube with the spectral axis the 2nd
cube axis.
```
grow spectrum [1,0,1] [1,1,0] [2,4,0] out=cube size=[4,8,0] new=t
Grow the spectrum into a cube with the spectral axis the 3rd
cube axis.

grow image [0,0,1] [0,0,1] [0,0,5] out=cube size=[0,0,5] new=t
Grow an image into a cube, using the image as an xy-plane.

grow image [0,1,0] [0,1,0] [0,5,0] out=cube size=[0,5,0] new=t
Grow an image into a cube, using the image as an xt-plane.

grow image [1,0,0] [1,0,0] [5,0,0] out=cube size=[5,0,0] new=t
Grow an image into a cube, using the image as a yt-plane.

Notes: This routine recognises the Specdre Extension v. 0.7. This routine does not propa-
gate any other extensions even when a new output file is created.
This routine may work in situ on an existing output file.
When IN is given as a subset of lower actual dimensionality than its base NDF, the
dimensionality will formally be the same as that of the base NDF with interspersed
dimensions (axis lengths) of 1. If this is inconvenient, use the application SUBSET to
create the subset in advance and without degenerate axes.

D.106  GROWX—Performs reverse function to that of EXTRACT

Description: Replicates a spectrum to form an image. The source spectrum runs along
the X-axis.

Parameters: SPECTRUM  Spectrum to be ‘grown’ into image.
                IMAGE   Image to grow spectrum into.
                YSTART  First cross-section to copy spectrum into.
                YEND    Last cross-section to copy spectrum into.
                NEW     Force creation of a new image?
                YSIZE   Y-dimension of new image.

Source comments: This is the main routine for the Figaro commands GROWX
and GROWY. These both copy a spectrum into one or more
cross-sections of an image, GROWX copying into cross-
sections of constant AXIS(2), and GROWY copying into cross-
sections of constant AXIS(1). The operation performed by GROWX
is the inverse of that performed by EXTRACT, and similarly
for GROWY and YTRACT.

Command parameters:

SPECTRUM   (Character) The name of the input spectrum file.

IMAGE      (Character) The name of the image into which the
            spectrum is to be copied. If the image file does
            not exist, or if NEW is set a new file is created
            with all other data elements set to zero.

XSTART     (Numeric) The number of the first cross-section
or into which the spectrum is to be copied. XSTART
YSTART is used by GROWY, YSTART by GROWX.

XEND (Numeric) The number of the last cross-section
or into which the spectrum is to be copied. XEND
YEND is used by GROWY, YEND by GROWX.

XSIZE (Numeric) If a new image has to be created, one
or of its dimensions will be that of the spectrum, but
YSIZE the other is unknown. This has to be specified as
XSIZE (for GROWY) or YSIZE for GROWX.

Command Keywords:

NEW Used to force the creation of a new image, even if
such an image exists already.

User variables used: None

Error information: Ignored.

Data quality information: Handled using flagged data values.

KS / CIT 19th Sept 1983

D.107  GROWXT-Copies an image into contiguous XT planes of a cube

Description: GROWXT copies an image into successive planes of a cube (strictly a cuboid)
for which the Y value is constant. This is the reverse of the XTPLANE function, except
that all the planes grown into the cube are the same.

Parameters: IMAGE The name of the image to be used.
CUBE The name of the data cube into which the image is to be grown. Successive ‘Y’
planes (planes for which the second array index is constant) will be set the same
as IMAGE. If CUBE does not exist, one will be created. If it does exist, its first
and third dimensions must match the dimensions of IMAGE.
YSTART The number of the first plane in the cube involved in this operation.
YEND The number of the last plane in the cube involved in this operation.
NEW If not set, and a cube of the specified name exists, then IMAGE will be grown
into that existing cube. If NEW is set, a new cube will always be created.
YSIZE If a new cube has to be created, its second dimension is unknown and has to
specified as YSIZE. The other two dimensions will be those of the image.

Source comments: W 3

This is the main routine for the Figaro commands GROWXY, GROWXT
and GROWYT. These both copy an cube into one or more
planes of a cuboid, GROWXY copying into planes of constant T,
GROWYT into planes of constant X, and GROWXT into planes of
constant Y. The operations performed by GROWXY, GROWXT and
GROWYT are the inverses of XYPLANE, XTPLANE and YTPLANE
respectively.
Command parameters -

IMAGE (Character) The name of the input image file. This will actually be image.dst.

CUBE (Character) The name of the cube into which the image is to be copied. If the cube file does not exist, or if NEW is set, a new file is created with all other data elements set to zero.

XSTART, (Numeric) The number of the first plane into which the image is to be copied. XSTART is used by GROWYT, YSTART by GROWXT, TSTART by GROWXY.

YSTART or TSTART

XEND, (Numeric) The number of the last plane into which the image is to be copied. XEND is used by GROWYT, YEND by GROWXT, TEND by GROWXY.

YEND, or TEND

XSIZE, (Numeric) If a new cube has to be created, two of its dimensions will be those of the image, but the other is unknown. This has to be specified as XSIZE, YSIZE or TSIZE for GROWYT, GROWXT, GROWXY respectively.

YSIZE, or TSIZE

Command Keywords

NEW Used to force the creation of a new cube, even if such an cube exists already.

User variables used - None

D.108 GROWXY-Copies an image into contiguous XY planes of a cube

Description: GROWXY copies an image into successive planes of a cube (strictly a cuboid) for which the T value is constant. This is the reverse of the XYPLANE function, except that all the planes grown into the cube are the same.

Parameters:

IMAGE The name of the image to be used.

CUBE The name of the data cube into which the image is to be grown. Successive ‘T’ planes (planes for which the third array index is constant) will be set the same as IMAGE. If CUBE does not exist, one will be created. If it does exist, its first and second dimensions must match the dimensions of IMAGE.

TSTART The number of the first plane in the cube involved in this operation.

TEND The number of the last plane in the cube involved in this operation.

NEW If not set, and a cube of the specified name exists, then IMAGE will be grown into that existing cube. If NEW is set, a new cube will always be created.

TSIZE If a new cube has to be created, its third dimension is unknown and has to specified as TSIZE. The other two dimensions will be those of the image.

Source comments:

This is the main routine for the Figaro commands GROWXY, GROWXT
and GROWYT. These both copy an cube into one or more
planes of a cuboid, GROWXY copying into planes of constant T,
GROWYT into planes of constant X, and GROWXT into planes of
constant Y. The operations performed by GROWXY, GROWXT and
GROWYT are the inverses of XYPLANE, XTPLANE and YTPLANE
respectively.

Command parameters -

IMAGE (Character) The name of the input image file.
This will actually be image.dst.

CUBE (Character) The name of the cube into which the
image is to be copied. If the cube file does
not exist, or if NEW is set, a new file is
created with all other data elements set to zero.

XSTART, (Numeric) The number of the first plane
YSTART or into which the image is to be copied. XSTART
TSTART is used by GROWYT, YSTART by GROWXT, TSTART by GROWXY.

XEND, (Numeric) The number of the last plane
YEND, or into which the image is to be copied. XEND is
TEND used by GROWYT, YEND by GROWXT, TEND by GROWXY.

XSIZE, (Numeric) If a new cube has to be created, two
YSIZE, or of its dimensions will be those of the image, but
TSIZE the other is unknown. This has to be specified as
XSIZE, YSIZE or TSIZE for GROWYT, GROWXT, GROWXY
respectively.

Command Keywords

NEW Used to force the creation of a new cube, even if
such an cube exists already.

User variables used - None

KS / AAO 15th April 1985

D.109 GROWY-Performs reverse function to that of YSTRACT

Description: Replicates a spectrum to form an image. The source spectrum runs along
the Y-axis.

Parameters: SPECTRUM Spectrum to be ‘grown’ into image.

IMAGE Image to grow spectrum into.

XSTART First cross-section to copy spectrum into.

XEND Last cross-section to copy spectrum into.

NEW Force creation of a new image?

XSIZE X-dimension of new image.

Source comments
This is the main routine for the Figaro commands GROWX and GROWY. These both copy a spectrum into one or more cross-sections of an image, GROWX copying into cross-sections of constant AXIS(2), and GROWY copying into cross-sections of constant AXIS(1). The operation performed by GROWX is the inverse of that performed by EXTRACT, and similarly for GROWY and YSTRACT.

Command parameters:

**SPECTRUM** (Character) The name of the input spectrum file.

**IMAGE** (Character) The name of the image into which the spectrum is to be copied. If the image file does not exist, or if the 'NEW' keyword is specified, a new file is created with all other data elements set to zero.

**XSTART** (Numeric) The number of the first cross-section or into which the spectrum is to be copied. XSTART is used by GROWY, YSTART by GROWX.

**XEND** (Numeric) The number of the last cross-section or into which the spectrum is to be copied. XEND is used by GROWY, YEND by GROWX.

**XSIZE** (Numeric) If a new image has to be created, one of its dimensions will be that of the spectrum, but the other is unknown. This has to be specified as XSIZE (for GROWY) or YSIZE for GROWX.

Command Keywords:

**NEW** Used to force the creation of a new image, even if such an image exists already.

User variables used: None

Error information: Ignored.

Data quality information: Handled using flagged data values.

KS / CIT 19th Sept 1983

D.110 GROWYT-Copies an image into contiguous YT planes of a cube

**Description**: GROWYT copies an image into successive planes of a cube (strictly a cuboid) for which the X value is constant. This is the reverse of the YTPLANE function, except that all the planes grown into the cube are the same.

**Parameters**: IMAGE The name of the image to be used.

CUBE The name of the data cube into which the image is to be grown. Successive 'X' planes (planes for which the first array index is constant) will be set the same as
IMAGE. If CUBE does not exist, one will be created. If it does exist, its second
and third dimensions must match the dimensions of IMAGE.

**XSTART**  The number of the first plane in the cube involved in this operation.

**XEND**  The number of the last plane in the cube involved in this operation.

**NEW**  If not set, and a cube of the specified name exists, then IMAGE will be grown
into that existing cube. If NEW is set, a new cube will always be created.

**XSIZE**  If a new cube has to be created, its first dimension is unknown and has to
specified as XSIZE. The other two dimensions will be those of the image.

### Source comments:

This is the main routine for the Figaro commands GROWXY, GROWXT
and GROWYT. These both copy an cube into one or more
planes of a cuboid, GROWXY copying into planes of constant T,
GROWYT into planes of constant X, and GROWXT into planes of
constant Y. The operations performed by GROWXY, GROWXT and
GROWYT are the inverses of XYPLANE, XTPLANE and YTPLANE
respectively.

### Command parameters -

- **IMAGE**  (Character) The name of the input image file.        
  This will actually be image.dst.

- **CUBE**  (Character) The name of the cube into which the
  image is to be copied. If the cube file does
  not exist, or if NEW is set, a new file is created
  with all other data elements set to zero.

- **XSTART**, **YSTART** or **TSTART**  (Numeric) The number of the first plane
  into which the image is to be copied. XSTART
  is used by GROWYT, YSTART by GROWXT, TSTART by GROWXY.

- **XEND**, **YEND**, or **TEND**  (Numeric) The number of the last plane
  into which the image is to be copied. XEND is
  used by GROWYT, YEND by GROWXT, TEND by GROWXY.

- **XSIZE**, **YSIZE**, or **TSIZE**  (Numeric) If a new cube has to be created, two
  of its dimensions will be those of the image, but
  the other is unknown. This has to be specified as
  XSIZE, YSIZE or TSIZE for GROWYT, GROWXT, GROWXY
  respectively.

### Command Keywords

- **NEW**  Used to force the creation of a new cube, even if
  such an cube exists already.

### User variables used - None

KS / AAO 15th April 1985
D.111  GSPIKE-Generates a ‘spiketrum’ from a table of values

Description: Given a template spectrum - from which it gets the wavelength range required - and a table file giving wavelength and data values, GSPIKE generates a ‘spiketrum’; a spectrum where all the elements except for those at the tabulated values are zero.

Parameters:  
- **SPECTRUM** A template spectrum to determine the wavelength range over which to generate the ‘spiketrum’ from tabulated values. Must have a proper array of X values.
- **TABLE** The name of the table file holding the pairs of values (wavelength and data) for the points to be used to generate the ‘spiketrum’. If no extension is specified, .TAB will be used. The table may be in any of the standard Figaro directories, or in the current default directory. Or it may be specified in a form that includes the directory name.
- **SPIKETRUM** The name of the resulting ‘spiked spectrum’ (hence ‘spiketrum’) produced by GSPIKE. It will always be a new file.

Source comments:  
Generates a 'spiketrum' from a table of X and Z values, given a spectrum to use as a template for the X range to be used. The resulting spiketrum will be a spectrum with the same .X structure as the template spectrum, and a .Z structure that has zeros everywhere except at the points given in the table. The table file can include SET commands that set individual item values in the resulting file, but the item names need to have been defined in the file SPIKETRUM.DEF.

Command parameters -

- **SPECTRUM** (Character) The name of the template spectrum.
- **TABLE** (Character) The name of the file containing the table of X and Z values. If TABLE contains no extension then '.TAB' will be assumed.
- **SPIKETRUM** (Character) The name of the spiketrum to be created. Note that this will always be a new file.

Command keywords - None

User variables used - None

KS / CIT 7th May 1984

D.112  HARD-Sets the file name for hard copy output

Description: The HARD command is used to set the current device and type for all hardcopy graphics output.

Parameters:  
- **HARDDEV** Used to specify the device to which all hardcopy graphics output is to be sent. Normally, a device name as recognised by SGS should be used - that is, the device specified should be one of those listed in response to the OPTIONS keyword - i.e. by the command HARD OPTIONS=TRUE. If necessary, a
device type and connection ID can be used, by specifying e.g. HARDDEV="827,1" and by setting FORCE=YES.

Options

If set, a list of all the device names recognised by the SGS system is output.

force Normally, HARD will reject a device specification that is not known to SGS. However, if FORCE is set, it will use whatever is specified for HARDDEV unquestioningly. If it fails, that’s your problem.

Draw If set, HARD will output a test plot to the specified device. This acts as a test that the specification was correct.

Source Comments

SOFT/HARD/IDEV

SOFT is used to set the user variable (SOFT) that controls the soft copy graphics output of Figaro programs. HARD is used to set the user variable (HARD) that controls the hardcopy graphics output. IDEV is used to set the user variable (IDEV) that controls the image display.

Command Parameters -

HARDDEV (Character string) A device name recognised by or the GKS version of PGPLOT. Normally, what should be used is one of the device names recognised by or GNS. HARDDEV and SOFTDEV are used by HARD and SOFT respectively.

Command Keywords -

OPTIONS Causes a list of the various acceptable SGS device names to be output.

force Forces the device specification to be accepted even if it does not match one of the acceptable SGS names.

Draw Draws a test diagram on the screen. (DRAW is the default for SOFT and IDEV, DRAW=NO is the default for HARD)

KS / AAO 16th March 1988

D.113 Hcrossover-Cross-correlate two spectra and get redshift and error

Description: This computes the cross-correlation of two spectra and the location of the central peak of the cross-correlation. It can be used to determine a relative shift between two spectra. Routines added to calculate the confidence levels and improved error estimates (CCFPAR) according to analysis in Heavens A.F.,1993, MNRAS, 263, 735. The cross correlation function can also be saved in a disk structure.

Parameters: SPECTRUM Name of spectrum.

TEMPLATE Name of template spectrum.

VELOCITIES If yes, redshift is in km/S rather than Z.

BASERED Redshift of template spectrum.
**BASERR** Redshift error of template spectrum.

**BASEVEL** Velocity (km/s) of template spectrum.

**BASEVERR** Velocity error (km/s) of template spectrum.

**XSTART** First X value to be used.

**XEND** Last X value to be used.

**FITCONT** Fit continuum?

**CBPC** Percentage of spectrum covered by cosine bell.

**RECORD** Create file to record cross-correlation?

**CROSS** Name of cross-correlation data?

**Source comments:**

Main body of the Figaro HCROSS function. This computes the cross-correlation of two spectra and the location of the central peak of the cross-correlation. It can be used to determine a relative shift between two spectra. Routines added to calculate the confidence levels and improved error estimates (CCFPAR) according to analysis in Heavens A.F., 1993, MNRAS, 263, 735. Routine added to calculate the redshift and error in redshift directly (GET_LOGSTEP and CALCRS). The cross correlation function can also be saved in a disk structure.

Command parameters -

**SPECTRUM** (Character) The spectrum to be compared with a template spectrum.

**TEMPLATE** (Character) The template spectrum to be used. The two spectra should be the same length.

**BASERED** (Numeric) Redshift of template spectrum. Negative=blueshift. Required if VELOCITIES=NO

**BASERR** (Numeric) Redshift error of template. Required if VELOCITIES=NO

**BASEVEL** (Numeric) Recession velocity in km/s (see keyword VELOCITIES below) of template spectrum. Negative=blueshift. Required if VELOCITIES=YES

**BASEVERR** (Numeric) Velocity error of template (km/s). Required if VELOCITIES=YES

**XSTART** (Numeric) Data with an axis data value less than XSTART will be ignored in the cross-correlation.

**XEND** (Numeric) Data with an axis data value greater than XEND will also be ignored. Note that these values are used to determine the channel numbers to be used for SPECTRUM, and the same ones will be used for TEMPLATE, even if TEMPLATE has a different axis structure.

**CBPC** (Numeric) Percentage of spectrum covered by a cosine bell prior to application of the FFT.

**CROSS** (Character) the name of the data structure to hold the cross-correlation, if it is to be saved. The file created will be cross.dst, and will look like an ordinary spectrum - i.e. can be plotted by SPLT, etc. CROSS is ignored if RECORD is not set.
Command keywords -

FITCONT If set, a continuum fit is performed on the two spectra prior to application of the cosine bell.
RECORD If set, the cross-correlation of the two spectra will be recorded as a new data structure.
VELOCITIES If yes, template redshift and error are entered in km/s (Negative=blueshift), and results are printed in km/s. Recommended option, and default, is NO i.e. a redshift z is expected and returned. Velocity option only makes sense if v<<c. \( v=cz \) and is accurate to \( O(z) \)

User variables used -

SHIFT (Numeric) The relative shift of the two spectra.
REDSHIFT (Numeric) The redshift of the spectrum.
REDERR (Numeric) Redshift error of spectrum.
VELOCITY (Numeric) Recession velocity of spectrum (km/s): Only meaningful if v<<c.
VELERR (Numeric) Recession velocity error of spectrum (km/s)
CONF (Numeric) The confidence in the redshift of the spectrum
WARN (Numeric) =1 if routine detects badly-matched spectra. =0 otherwise (not a guarantee of good spectra)

5th Oct. 1993. AFH, NEJ / UoE.

D.114 HIMAGE-Creates greyscale plot of image

Description: Handles an ‘HIMAGE’ command, producing an image on any suitable device. Originally based on the routine IMAGE.

Parameters: IMAGE IMAGE = FILE (Read) The name of the first image to be displayed.
YSTART YSTART = REAL (Read) The first Y value to be displayed.
YEND YEND = REAL (Read) The last Y value to be displayed.
XSTART XSTART = REAL (Read) The first X value to be displayed.
XEND XEND = REAL (Read) The last X value to be displayed. X and Y are the horizontal and vertical directions - as displayed on the Grinnell - respectively. The first value for each is 1.
LOW LOW = REAL (Read) The minimum count level for the display, for the first image.
HIGH HIGH = REAL (Read) The maximum count level for the display, for the first image.
PLOTDEV PLOTDEV = CHARACTER (Read) The name of the plotting device
ASPECT ASPECT = REAL (Read) The aspect ratio of the plot
SHRINK SHRINK = LOGICAL (Read) If to shrink image to leave margin all round
LOG LOG = LOGICAL (Read) To take logarithms of the image to display it

Source comments: available
D.115  HIST-Produce histogram of data value distribution in an image

Description:  HIST produces a histogram showing the distribution of data values in a Figaro data file. The histogram is produced in the form of a one dimensional data structure, which can be plotted using SPLOT.

Parameters:  

- **IMAGE**  The name of the file containing the data to be histogrammed. The data may have any dimensions.
- **MINVAL**  The value of the center of the first bin of the histogram. Data less than MINVAL is not included in the histogram, so if delta is the bin width \((\text{Delta}=(\text{Maxval-Minval})/(\text{Bins}-1))\), the first bin of the result gives the number of pixels between Minval and \((\text{Minval+delta}/2)\), while the bin centered on value will give the number of pixels between value-delta/2 and value+delta/2.
- **MAXVAL**  The value of the center of the last bin of the histogram. Data greater than MAXVAL is not included in the histogram.
- **BINS**  The number of bins to be used in the histogram.
- **SPECTRUM**  The histogram produced by HIST is in the form of a one dimensional data array, i.e. a sort of spectrum which may, for example, be plotted by SPLOT.

Source comments:  

Creates a one dimensional data structure containing a histogram of the data distribution in another structure.

Command parameters -

- **IMAGE**  (Character)  The name of the data structure (not necessarily two dimensional) to be histogrammed.
- **MINVAL**  (Numeric)  The minimum value for the histogram.
- **MAXVAL**  (Numeric)  The maximum value for the histogram.
- **BINS**  (Numeric)  The number of bins to be used for the histogram.
- **SPECTRUM**  (Character)  The histogram produced from the input data.

Command keywords -  None

KS / CIT 29th June 1984

D.116  HOPT-Histogram optimization of an image

Description:  Performs a histogram equalisation on an image in an attempt to enhance the contrast. See, for example, Gonzalez and Wintz, 'Digital Image Processing', Addison-Wesley).

Parameters:  

- **IMAGE**  Name of image to be equalised.
- **MINVAL**  Minimum data value to be used.
- **MAXVAL**  Maximum data value to be used.
- **EQUALISE**  Generate equalised output data?
- **OUTPUT**  Name of resulting image.

See also:  KAPPA, LAPLACE, LUTABLE, SHADOW, THRESH
Source comments:

Performs a histogram equalisation on an image in an attempt to enhance the contrast. See, for example, Gonzalez and Wintz, 'Digital Image Processing', Addison-Wesley).

Command parameters -

IMAGE The name of the structure containing the image.

MINVAL The minimum value to be used in generating the histogram.

MAXVAL The maximum value to be used in generating the histogram.

OUTPUT The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

Command keywords -

EQUALISE Actually perform an equalisation. If this is no (most likely if EQUALISE is not set) then all that happens is that the distribution histogram is generated and displayed.

User variables -

SOFT (Character) The device/type to be used for terminal graphics.

KS / CIT 27th Jan 1983

D.117 I2CMPLX-Copies an array into the imaginary part of a complex structure

Description: I2CMPLX copies the data from a specified file (usually not a complex one) into the imaginary part of an existing complex data file, usually one created by R2CMPLX.

Parameters: IDATA The name of an existing file that contains a data array that is to become the imaginary part of an existing complex data structure.

CDATA The name of the existing complex data structure whose imaginary part is to be set. CDATA may be the same as IDATA (which will have to be a complex structure), in which case one gets the odd effect of setting the imaginary part equal to the previous value of the modulus. Normally, CDATA is a separate file recently created by R2CMPLX.

Source comments: M P L X

Sets the imaginary part of an existing complex data structure. The complex data structure will probably have been created in the
first instance by R2CMPLX, which leaves the imaginary part zero. I2CMPLX takes the data array from a specified file (usually not a complex one - note that if it is complex, it is the modulus that is used) and uses it to form the imaginary part of the complex data.

Command parameters -

IDATA (Character) The data structure containing the array that is to form the imaginary part of the complex data. That is, the data array of IDATA becomes the imaginary data array of the output complex structure.

CDATA (Character) The resulting complex data structure. Note that this is an already existing structure.

KS / AA0 8th Sept 1986

D.118 IADD-Adds two images (or two spectra)

Description: IADD adds together two images of identical dimensions. The number of dimensions does not have to be 2, so IADD can add both spectra and data cubes as well as images.

Parameters: IMAGE The first of the two images to be added.

IMAGE1 The second of the two images to be added.

OUTPUT The name of the resulting image. This can be the same as IMAGE, in which case no new image will be created.

See also: FIGARO: ISUB, IMULT, IDIV, ICADD, ICSUB, ICMULT, ICDIV

KAPPA: CADD, CDIV, CMULT, CSUB, MATHS, ADD, SUB, MULT, DIV

Source comments: S U B

Adds, multiplies, divides or subtracts two images.

Command parameters -

IMAGE The name of the structure containing the first image.

IMAGE1 The name of the structure containing the second image data.

OUTPUT The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data (and any error or data quality information) a direct copy of the first image.

The command itself (IADD, IMULT, IDIV or ISUB) is used to differentiate between the two operations.

KS / CIT 26th Sept 1983
D.119  IALOG-Takes the antilog of an image

Description: IALOG takes the base 10 antilog of each pixel in an image. Each pixel in the output image takes the value 10 to the power of the corresponding pixel in the input image.

Parameters: IMAGE The datafile to be processed.
OUTPUT The name of the resulting image. If this is the same as IMAGE (the default) the operation will be performed in situ. Otherwise a new image will be created.

See also: FIGARO: ILOG
          KAPPA: LOG10, LOGAR, LOGE.

Source comments: G G

Takes the base 10 antilog of an image.

Command parameters -

IMAGE The name of the structure containing the image.

OUTPUT The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

Command keywords - None

User variables used - None  

KS / CIT 22nd April 1984

D.120  IARC-Given fit to single spectrum, fit all spectra in a 2-D arc

Description: The manual arc line fitter, ARC, can be used to fit a single spectrum obtained by summing successive cross-sections taken from the center of a 2-D arc. IARC will then apply that fit to all the rows in the 2-D arc, starting from the center and working out in both directions.

Parameters: IMAGE EXTRACT can be used to sum cross-sections from the center of a 2-D arc - an image containing a set of arc spectra (a long-slit arc). ARC can then be used to fit them, producing a list of identified arc lines, channel number & wavelengths. IARC can then use this fit - in the file ARLINES.LIS - as a starting point for a fit to each of the cross-sections in the 2-D arc in the file whose name is given by IMAGE.

RSTART Normally, IARC starts in the center - where the initial fit should be best - and works its way out. RSTART is the cross-section (or row) at which it starts, and should normally be the central cross-section of those summed to produce the arc spectrum used in the initial ARC fit.

RWIDTH IARC can sum consecutive rows of a 2-D arc, instead of fitting each row individually. Increasing RWIDTH will speed up the fitting, since it reduced the number of spectra fitted, and may reduce the number of lines that get rejected,
since the signal to noise ratio in the summed rows will be improved. However, it will degrade the smooth progression from one row to the next, and may make the lines broader.

FILE  The results of the 2-D fit are written to a file in a form suitable for use by the image scrunchers, ISCRUNCH and ISCRUNI. For ISCRUNCH the default name is usually satisfactory. For ISCRUNI you may want to exercise more control over the process and the files used.

RSIGMA Normally, you should use the same sigma (the line width parameter) for IARC as was used for the original ARC fit, and this is always the default. However, IARC is a little more tolerant in searching for lines than is ARC, and there may sometimes be a case for using a slightly lower value of sigma, especially if the arc lines are very close together.

SPREAD IARC normally searches for lines in a new cross-section in two steps, once with a larger sigma than that specified in order to make it less likely to loose lines that may be a little further away than expected, and then with the specified sigma in order to tighten up the centering. Setting SPREAD=NO will disable this feature and the search will be performed once, with the specified sigma.

GAP If IARC fails to find a line, it will use the position it had previously for the line. If the line is missing a specified number of times, however, it is deleted from the line list. This maximum number of times is specified by GAP. The correct setting for GAP depends on the nature of the arc - an arc with high distortion should have a low GAP value to avoid lines becomming confused. If distortion is low but several cross-sections are missing or have poor data, GAP should be higher to let IARC 'jump' over them.

SIGMIN SIGMIN is a measure of the strength of an arc line. When the program searches for lines to use as 'lock' lines, it evaluates their sigma - here the line height as a multiple of the square of the local continuum. Any line for which this value is less than SIGMIN will not be used.

XCORR If XCORR is set, IARC will attempt to determine a linear shift between successive spectra using cross-correlation and will apply this to the arc lines found in the previous spectrum before looking for them in the new spectrum. This is particularly applicable to fibre data, where such linear shifts often occur. It is probably not useful for cases such as image tube distortion, where the spectra should change in a constant manner.

DETAIL If DETAIL is set, IARC will list the details line position, wavelength, fitted wavelength, error) for each line fitted. This is a lot of information, and is not normally useful.

LOCK If the original fit has regions where there are strongish lines that have not been identified, it may help to use their positions to lock down the fit in those regions. This does not improve the fit, since the wavelengths are simply assumed to be those given by the original fit, but the automatic fits will not be so free to wiggle in these regions. However, you have to be careful not to get a large number of weak lines that dominate the fit at the expense of stronger lines that were positively identified.

WEIGHT Indicates whether the least-squares fit is to be weighted using the peak intensity of each line (so that more weight is given to strong lines).
Performs a 2-D fit to an arc spectrum, given an initial fit to a single arc spectrum as a starting point. Generally, the starting spectrum will have been extracted from the center of the 2-D arc. IARC then starts at a suitable cross-section in the 2-D arc - usually the central one - and works out from there, fitting each cross-section individually, looking for the lines in the starting spectrum. For each line, the program looks for a peak close to where it was found in the previous fit. If a line does not show up in one cross-section, the previous position will be used, but if it fails to show up in the next cross-section, it will be dropped from the search list. In many cases, there may be strong lines which were not identified; these cannot be used to improve the fit, but they can be used to 'lock' it down in the regions where there are few or no identified lines.

**Command parameters -**

**IMAGE** (Character) The name of the image containing the 2-D arc.

**RSTART** (Numeric) The starting cross-section to be used.

**RWIDTH** (Numeric) The number of cross-sections to be added together for each fit - if the arc is weak, this will need to be increased.

**RSIGMA** (Numeric) Normally, the sigma value from the arc line file is used, but this can be overridden by RSIGMA.

**GAP** (Numeric) Number of cross sections over which IARC can fail to find a line before deleting it from the lists.

**FILE** (Character) The name of the file to which the results are to be written. If an extension is not given, .IAR will be used.

**SIGMIN** (Numeric) The minimum acceptable value for the sigma of an arc line found in the locking process. Sigma here is the height of the line relative to the square root of the continuum. Only used if LOCK set.

**Command keywords -**

**LOCK** Indicates that a search is to be made for lines to 'lock' the fit.

**SPREAD** Indicates that IARC looks for lines first with an increased sigma, then with the specified sigma in order to refine the fit. If SPREAD is not set, the search is just with the specified sigma value.

**DETAIL** Indicates that full details of the fits are to be output (This is mainly a diagnostic tool.)

**XCORR** If set, IARC will attempt to determine a linear shift between successive spectra using cross-correlation. This is particularly applicable to fibre data, where such linear shifts may occur. It is probably not useful for cases such as image tube distortion, where the spectra should change in a constant manner.

**WEIGHT** Indicates whether the least-squares fit is to be weighted using the peak intensity of each line (so that more weight
is given to strong lines).

User variables used -

- IARC_WMAX (Numeric) Maximum wavelength for any of the spectra.
- IARC_WMIN (Numeric) Minimum wavelength.
- NOFITS (Numeric) The number of rows that could not be fitted.
- ORDER (Numeric) The order of the original fit.
- RMSMAX (Numeric) Maximum RMS error from the fits.

Input files -

- ARLINES.LIS Contains the details of the starting fit. For format details see comments for subroutine ARGETL, or the ARC command.

Output files -

As named by the FILE parameter.

Dimensions, NX, NY (17X,I5,4X,I5).
# rows not fitted properly, (42X,I5).
Maximum RMS error, (20X,F10.2).
Maximum order used, - ORDER - (33X,I3).
Then, for each row, row number and ORDER+1 polynomial coefficients, the constant term being the last non-zero term. (I14,10X,2D24.16,3(/3D24.16)).

D.121 ICADD-Adds a constant to an image

**Description:** A constant value FACTOR is added to each pixel in the dataset IMAGE.

**Parameters:**
- **IMAGE** Name of image to be added to.
- **FACTOR** Additive constant.
- **OUTPUT** Name of resulting image.

**See also:** FIGARO: ICSUB, ICMULT, ICDIV, IADD, ISUB, IMULT, IDIV, KAPPA: CADD, CDIV, CMULT, CSUB, MATHS, ADD, SUB, MULT, DIV

**Source comments:**

This routine is the main body of ICMULT, ICDIV, ICADD and ICSUB, and of XCMULT, XCDIV, XCADD and XCSUB. The Ixxxx routines operate on the data in an image, the Xxxx routines operate on the data in the X array of the input file.

ICMULT multiplies an image by a constant. Since the constant can be less than 1., this function will also divide an image by a constant, but ICDIV saves the caller from having to calculate a reciprocal. ICADD adds a constant to an image and ICSUB subtracts a constant from an image.

**Command parameters -**
D.122  ICDIV—Divides an image by a constant

**Description:** Each pixel in the dataset IMAGE is divided by the constant value FACTOR.

**Parameters:**
- **IMAGE**  Name of image to be divided.
- **FACTOR**  Value of divisor.
- **OUTPUT**  Name of resulting image.

**See also:** FIGARO: ICSUB, ICMULT, ICADD, IADD, ISUB, IMULT, IDIV. KAPPA: CADD, CDIV, CMULT, CSUB, MATHS, ADD, SUB, MULT, DIV.

**Source comments:**

This routine is the main body of ICMULT, ICDIV, ICADD and ICSUB, and of XCMULT, XCDIV, XCADD and XCSUB. The Ixxxx routines operate on the data in an image, the Xxxx routines operate on the data in the X array of the input file.

ICMULT multiplies an image by a constant. Since the constant can be less than 1., this function will also divide an image by a constant, but ICDIV saves the caller from having to calculate a reciprocal. ICADD adds a constant to an image and ICSUB subtracts a constant from an image.

**Command parameters** -

- **IMAGE**  (Character) The name of the structure containing the image. Uses main data array, or the x-axis data for the Xxxxx routines.
- **FACTOR**  (Numeric) The value of the constant factor.
- **OUTPUT**  (Character) The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

The command name is used to distinguish between the possible operations.
D.123  ICMULT-Multiplies an image by a constant

Description: Each pixel in the dataset IMAGE is multiplied by the constant value FACTOR.

Parameters: IMAGE  Name of image to be multiplied.
               FACTOR  Multiplicative constant.
               OUTPUT  Name of resulting image.

See also: FIGARO: ICSUB, ICADD, ICDIV, IADD, ISUB, IMULT, IDIV.
          KAPPA: CADD, CDIV, CMULT, CSUB, MATHS, ADD, SUB, MULT, DIV.

Source comments:

This routine is the main body of ICMULT,ICDIV,ICADD and ICSUB, and of XCMULT, XCDIV, XCADD and XCSUB. The Ixxxx routines operate on the data in an image, the Xxxx routines operate on the data in the X array of the input file.
ICMULT multiplies an image by a constant. Since the constant can be less than 1., this function will also divide an image by a constant, but ICDIV saves the caller from having to calculate a reciprocal. ICADD adds a constant to an image and ICSUB subtracts a constant from an image.

Command parameters -

IMAGE  (Character) The name of the structure containing the image.
       Uses main data array, or the x-axis data for the XCxxx routines.

FACTOR (Numeric) The value of the constant factor.

OUTPUT (Character) The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

The command name is used to distinguish between the possible operations.

KS / CIT 12th June 1984

D.124  ICONT-Produces a contour map of an image

Description: Produces a contour map of an image.

Parameters: IMAGE  Name of image to be contoured.
             YSTART  First Y value to be displayed.
             YEND    Last Y value to be displayed.
             XSTART  First X value to be displayed.
             XEND    Last X value to be displayed.
             LOW     Lowest contour level.
             HIGH    Highest contour level.
             CONTOURS Number of contour levels.
LABEL  Label for plot.
THICKNESS  Width of plotted lines.
LEVELS  Explicit contour levels to use.
ADJUST  Adjust scales so as to fill display?
BYVALUE  Specify contour level array explicitly?
HARDCOPY  Generate a hardcopy plot?

See also: KAPPA: CONTOUR.

Source comments:

ICONT / IGREY
Handles an 'ICONT' command, producing a contour plot of an image on either the current hard or soft graphics device, or an 'IGREY' command, which produces a grey-scale plot.

Command parameters -

IMAGE  (Character) The name of the image to be contoured.
YSTART  (Numeric) The first Y value to be displayed.
YEND  (Numeric) The last Y value to be displayed.
XSTART  (Numeric) The first X value to be displayed.
XEND  (Numeric) The last X value to be displayed.
Note that this initial version only accepts these values as pixel numbers.
LOW  (Numeric) The minimum contour level (ICONT) or the black level (IGREY).
HIGH  (Numeric) The maximum contour level (ICONT) or the white level (IGREY).
CONTOURS  (Numeric) The number of contours displayed - these will be divided evenly between HIGH and LOW, unless explicitly specified as BYVALUE. (ICONT only).
LABEL  (Character) A label for the plot.
THICKNESS  (Numeric) Thickness to use for lines (only used if the HARDCOPY parameter is specified, and ICONT only)
LEVELS  (Numeric array) The contour levels to use (ICONT only, and only if the BYVALUE keyword is specified).

Command keywords -

HARDCOPY  Output the plot to the current hard graphics device.
ADJUST  Adjust scales so as to fill screen.
BYVALUE  For ICONT, indicates that explicit contour values are specified in the LEVELS parameter.

User variables used -  ("">" input, "<" output)

(> ) SOFT  Current device/type (PGPLOT convention) for soft graphics output.
(> ) HARD  Current device/type (PGPLOT convention) for hardcopy graphics output.
(< ) TVXST  is set to the starting x-value for the plot.
(< ) TVXEN  Is set to the final x-value for the plot.
(< ) TVYST  is set to the starting y-value for the plot.
D.125  ICONV3-Convolve an image with a 3x3 convolution kernel

Description: ICONV3 convolves an image with a 3x3 symmetric convolution kernel, defined by its central value and its edge value. So the image is convolved with a 9 element array that has the 8 edge elements all set to one value and the central element to another. For example, CENTER=-9, EDGE=-1 is a highpass filter, CENTER=0.1, EDGE=0.1 would be a lowpass filter, while CENTER=8, EDGE=-1 would be a Laplacian edge enhancement filter.

Parameters: IMAGE The image to be filtered.
CENTER Central value of 3x3 convolution array.
EDGE Edge value of 3x3 convolution array.
OUTPUT The name of the resulting filtered image. This may be the same as IMAGE, in which case the filtering will be performed in situ, or different, in which case a new file will be created.

See also: FIGARO: ISMOOTH, IXSMOOTH, MEDFILT, KAPPA, CONVOLVE, FFCLEAN, GAUSMOOTH, MEDIAN

Source comments:

Convolves an image with a 3x3 symmetric convolution kernel. This allows a variety of spatial filters to be applied to an image. The 3 by 3 array convolved with the image is defined by two values, the central value and the single value used for the eight edge elements. That is, if C is the central and E the edge value, the kernel array looks like

```
  E E E
  E C E
  E E E
```

Command parameters -

IMAGE  The name of the structure containing the image.
CENTER The value for the kernel central element.
EDGE  The value for the kernel edge element.
OUTPUT The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.
D.126  **ICOR16-Corrects 16 bit data from signed to unsigned range**

**Description:** ICOR16 takes an image that was written as 16 bit unsigned data, but has been read as 16 bit signed data (a particular problem with FITS data) and corrects it.

**Parameters:**
- **IMAGE** The name of an image (or possibly a spectrum) that has been read as signed 16 bit integers, but actually contained unsigned 16 bit integers. The data range in the image will be modified from -32767..32768 to 0..65535 (only the negative values will be changed.)
- **OUTPUT** The name of the resulting image. If this is the same as IMAGE (the default) the operation will be performed in situ. Otherwise a new image will be created.

**Source comments:**

> It is possible to have an image written on tape in what looks like unsigned 16 bit data, when what is intended is actually signed 16 bit data. This is often true of FITS tapes, where 16 bit data is always assumed to be unsigned. ICOR16 corrects such an image.

**Command parameters**

- **IMAGE** The name of the structure containing the image.
- **OUTPUT** The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

**Command keywords** - None

**User variables used** - None

KS / AAO 6th Dec 1984

D.127  **ICSET-Set a selected region of an image to a constant value**

**Description:** ICSET sets a specified region in an image to a constant value. This is a modification of NCSET to operate on a 2-D image, and is a non-interactive version of CSET, which allows the region to be specified precisely in terms of the X-values of the data.

**Parameters:**
- **IMAGE** The name of the image to be modified by NCSET. It should be a 2-dimensional array.
- **YSTART** The first Y value to be set to the constant, in units used by the data - angstroms, for example, if the data is wavelength calibrated. RESET will set YSTART to the first Y value in the data.
- **YEND** The last Y value to be set to the constant, in units used by the data - angstroms, for example, if the data is wavelength calibrated. RESET will set YEND to the last Y value in the data.
- **XSTART** The first X value to be set to the constant, in units used by the data - angstroms, for example, if the data is wavelength calibrated. RESET will set XSTART to the first X value in the data.
XEND  The last X value to be set to the constant, in units used by the data - angstroms, for example, if the data is wavelength calibrated. RESET will set XEND to the last X value in the data.

CONSTANT  The selected region of the image (from XSTART to XEND) will be set to CONSTANT. No noise added on this level.

OUTPUT  NCSET generates an output file that is essentially the data from the input image, with one region set to the constant (user-specified) value. OUTPUT is the name of the resulting spectrum.

See also: FIGARO: CSET, NCSET, TIPPEX, KAPPA, CHPIX, FILLBAD, SEGMENT, NOMAGIC, RIFT, SETMAGIC, ZAPLIN.

Source comments:  E T

Figaro function to set a selected region of an image to a constant value. This is a modification of the standard FIGARO program NCSET to handle 2-D data arrays (which in turn is a non-interactive version of CSET, but has the possible advantage of allowing the region to be specified precisely in terms of the X values of the data).

This program overcomes the limitations of CLEAN or BCLEAN (qv) which inherently assume the data to be modified is in the rows of the input array. This may not be the case for CCD images, and certainly not for 2-D spectra where modification of data over a specified wavelength region is most often required.

USES:

Anticipated uses are patching out of dud columns in a CCD image, or removal of data over a specified wavelength range (whose X limits are supplied by the user). The latter may be used to remove residuals after less-than-perfect sky subtraction.

Command parameters -

IMAGE  (Character) The image to be modified.
YSTART  (Numeric) The Y-value of the start of the region.
YEND  (Numeric) The Y-value of the end of the region.
XSTART  (Numeric) The X-value of the start of the region.
XEND  (Numeric) The X-value of the end of the region.
VALUE  (Numeric) The value to use for the selected region.
OUTPUT  (Character) The name of the output file to be created. If this is the same as the input image, the data will be modified in situ.

Command keywords - None

User variables used - None

KS / CIT 27th March 1985
D.128  ICSUB-Subtracts a constant from an image

**Description:** The constant FACTOR is subtracted from each pixel in IMAGE.

**Parameters:**
- **IMAGE**  Name of image to be subtracted from.
- **FACTOR** Constant to be subtracted from image.
- **OUTPUT** Name of resulting image.

**See also:** FIGARO: ICADD, ICMULT, ICDIV, IADD, ISUB, IMULT, IDIV;
KAPPA: CADD, CDIV, CMULT, CSUB, MATHS, ADD, SUB, MULT, DIV.

**Source comments:**

This routine is the main body of ICMULT,ICDIV,ICADD and ICSUB, and of XCMULT, XCDIV, XCADD and XCSUB. The Ixxxx routines operate on the data in an image, the Xxxx routines operate on the data in the X array of the input file.

ICMULT multiplies an image by a constant. Since the constant can be less than 1., this function will also divide an image by a constant, but ICDIV saves the caller from having to calculate a reciprocal. ICADD adds a constant to an image and ICSUB subtracts a constant from an image.

Command parameters -

- **IMAGE**  (Character) The name of the structure containing the image. Uses main data array, or the x-axis data for the XCxxx routines.

- **FACTOR** (Numeric) The value of the constant factor.

- **OUTPUT** (Character) The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

The command name is used to distinguish between the possible operations.  

KS / CIT 12th June 1984

D.129  ICUR-Inspect an image with cursor

**Description:** This routine displays the position and data value according to the position of the cursor on the image display. Up to 50 pixel positions can be recorded for later use by other applications.

Use the cursor to select a position in the image previously displayed with the application IMAGE. Press one of the following:

D to display coordinates and pixel value, <space> to record pixel position, Q to quit the application.

Any other key is treated like ‘D’, alphabetic keys are case-insensitive.

**Parameters:**
- **IDEV**  The name of the imaging device, normally got from a global parameter which was set with the IDEV command.
IMARRAY  Information about the displayed part of the image and the part of the
display used - set by IMAGE or similar.
IMFILE  File name of the image displayed - set by IMAGE or similar.
XPIXELS  The pixel numbers in X for the points indicated by the cursor.
YPIXELS  The pixel numbers in Y for the points indicated by the cursor.
NPIXELS  The number of points selected by the cursor. Note: if no points are selected,
the values of NPIXELS, XPIXELS, YPIXELS are left unchanged.

See also: FIGARO: ICCUR, KAPPA: CURSOR, PICCUR, GAIA.

Authors:  KS: Keith Shortridge (CIT, AAO)
Implementation status:  This routine does not provide a continuous display of coordinates
and image value.

D.130  IDEV-Set the device for image display

Description:  The IDEV command is used to set the current device and type for all imaging
output.

Parameters:  IMAGDEV  The device to which all imaging output is to be sent. Normally,
a device name as recognised by SGS should be used - that is, the device specified
should be one of those listed in response to the OPTIONS keyword - i.e. by the
command SOFT OPTIONS. If necessary, a device type and connection id can be
used, by specifying e.g. IMAGDEV="827,1" and by setting FORCE=YES.
OPTIONS  If set, a list of all the device names recognised by the SGS system is
output.
FORCE  Normally, SOFT will reject a device specification that is not known to SGS.
However, if FORCE is set, it will use whatever is specified for SOFTDEV unques-
tioningly. If it fails, that’s your problem.
DRAW  If set, SOFT will output a test plot to the specified device. This acts as a test
that the specification was correct.

Source comments:  T / H A R D / I D E V

SOFT is used to set the user variable (SOFT) that controls
the soft copy graphics output of Figaro programs. HARD is
used to set the user variable (HARD) that controls the
hardcopy graphics output. IDEV is used to set the user variable
(IDEV) that controls the image display.

Command parameters -

HARDEV  (Character string) A device name recognised by
or the GKS version of PGPLT. Normally, what should
SOFTWARE be used is one of the device names recognised by
or GNS. HARDEV and SOFTWARE are used by HARD and SOFTWARE
IMAGDEV  respectively.

Command keywords -
OPTIONS  Causes a list of the various acceptable SGS device
names to be output.

FORCE   Forces the device specification to be accepted even
if it does not match one of the acceptable SGS names.

DRAW    Draws a test diagram on the screen. (DRAW is the
default for SOFT and IDEV, NODRAW is the default for HARD)

KS / AAO 16th March 1988

D.131  IDIFF-Takes the ‘differential’ of an image

Description: IDIFF takes an input image and creates a new image where each pixel is the
average absolute difference between the corresponding pixel in the original image
and its nearest neighbours. This emphasises those regions in the image where the
data is changing rapidly.

Parameters: IMAGE  IMAGE must contain two-dimensional data. IDIFF creates a re-
sulting image where each pixel is the average absolute difference between the
corresponding pixel in IMAGE and its nearest neighbours.

OUTPUT   The name of the resulting image. If this is the same as IMAGE (the default)
the operation will be performed in situ. Otherwise a new image will be created.

Source comments  F F

Given an image, creates a new image in which each pixel
is the average absolute difference between the corresponding
pixel in the input image and its immediate neighbours. This
highlights regions in the image where the data is changing
rapidly.

Command parameters -

IMAGE   The name of the structure containing the image.

OUTPUT  The name of the result of the operation. This can
be the same as for IMAGE. If not, a new structure
is created, with everything but the data a direct
copy of the input.

Command keywords - None

User variables used - None

KS / CIT 22nd April 1984

D.132  IDIV-Divides two images (or two spectra)

Description: Each pixel in IMAGE is divided by the corresponding pixel in IMAGE1. The
result is stored in OUTPUT.

Parameters: IMAGE  Name of first image.
**IGCONV-Convolve an image with a specified filter**

**Description:** This Figaro function convolves an image with a box array that is defined in a text file as a set of numbers. This is more flexible - though less efficient - than the simple convolution performed by ICONV3.

**Parameters:**
- **IMAGE** The name of a 1- or 2-dimensional data file that is to be convolved with the data array defined in the file specified by the CONVOL parameter.
- **CONVOL** A text file (default extension .cvf) which contains the definition of the data array that the IMAGE data is to be convolved with. The convolution data often referred to as a filter, since one of the main uses of this routine is to apply sharpening or smoothing filters to images) is specified as a series of numbers in free format (blank lines and lines starting with '*' being ignored). Normally one line of the filter is specified by one line of the file, but a line can be continued by ending it with a 'character.
- **OUTPUT** The name of the resulting data file. This will be a copy of the input file, except that the main data array will have been convolved with the specified array.

**Source comments:**

This is a general-purpose convolution program, able to convolve the main data array in a Figaro file with a rectangular array of any dimensions. This is a more flexible routine than ICONV3, where the convolution is with a 3x3 array specified only by a center and an...
edge value, but the additional flexibility probably makes it noticeably slower, particularly for very large convolution arrays.

Command parameters -

**IMAGE** The name of the structure containing the image.

**CONVOL** This is the name of a text file that contains the definition of the convolution array. See below for the format.

**OUTPUT** The name of the result of the operation. This can be the same as for **IMAGE**. If not, a new structure is created, with everything but the data a direct copy of the input.

File format:
The text file defining the convolution array should simply contain the values for the array in a free format, one line at a time. Lines beginning with a ‘*’ are ignored, as are blank lines. If the data for a line of the array has to be continued onto another line, a ‘\’ should appear at the end of the line to be continued.


**D.134 IGCUR-Use cursor to show X, Y and data values**

**Description:** Interative task to determine pixel values in a dataset. This task can be used to mark points to be used by other tasks.

**Parameters:** No user parameters.

See also: FIGARO: ICUR

KAPPA: CURSOR, PICCUR

GAIA.

**Source comments:** U R

Services an 'IGCUR' command. Displays position and data value on the image display, depending on cursor position.

Command parameters, keywords - None

User variables used - ("">" input, "<" output)

(>)

**IMFILE** (Character) The name of the data currently displayed. Set by IGREY or similar.

(>)

**PGENVARG** (Numeric array) The argument list for PGENV that was used in displaying the image.

(<)

**XPIXELS** (Numeric) The pixel numbers in X for the points indicated by the cursor.

(<)

**YPIXELS** (Numeric) The pixel numbers in Y for the points indicated by the cursor.

(<)

**NPIXELS** (Numeric) The number of points selected by
the cursor. Note: if no points are selected, the values of NPIXELS, XPIXELS and YPIXELS are left unchanged.

KS / CIT 20th March 1984

D.135 IGREY-Produces a grey-scale plot of an image

Description: Plots an image using a monochrome display look-up table.

Parameters: IMAGE Name of image to be grey-scaled.
    YSTART First Y value to be displayed.
    YEND Last Y value to be displayed.
    XSTART First X value to be displayed.
    XEND Last X value to be displayed.
    LOW Lowest data level used (black level).
    HIGH Highest data level used (white level).
    LABEL Label for plot.
    ADJUST Adjust scales so as to fill display?
    HARDCOPY Generate a hardcopy plot?

See also: FIGARO: IMAGE.
          KAPPA: DISPLAY;
          KAPRH: GREYPLOT.

Source comments:

Handles an 'ICONT' command, producing a contour plot of an image on either the current hard or soft graphics device, or an 'IGREY' command, which produces a grey-scale plot.

Command parameters -

IMAGE (Character) The name of the image to be contoured.
    YSTART (Numeric) The first Y value to be displayed.
    YEND (Numeric) The last Y value to be displayed.
    XSTART (Numeric) The first X value to be displayed.
    XEND (Numeric) The last X value to be displayed.
    Note that this initial version only accepts these values as pixel numbers.
    LOW (Numeric) The minimum contour level (ICONT) or the black level (IGREY).
    HIGH (Numeric) The maximum contour level (ICONT) or the white level (IGREY).
    CONTOURS (Numeric) The number of contours displayed - these will be divided evenly between HIGH and LOW, unless explicitly specified as BYVALUE. (ICONT only).
    LABEL (Character) A label for the plot.
    THICKNESS (Numeric) Thickness to use for lines (only used if the HARDCOPY parameter is specified, and ICONT only)
    LEVELS (Numeric array) The contour levels to use (ICONT only,
and only if the BYVALUE keyword is specified).

Command keywords -

HARDCOPY Output the plot to the current hard graphics device.
ADJUST Adjust scales so as to fill screen.
BYVALUE For ICONT, indicates that explicit contour values are specified in the LEVELS parameter.

User variables used - ("">" input, "<" output)

(>) SOFT Current device/type (PGPLOT convention) for soft graphics output.
(>) HARD Current device/type (PGPLOT convention) for hardcopy graphics output.
(<) TVXST is set to the starting x-value for the plot.
(<) TVXEN Is set to the final x-value for the plot.
(<) TVYST is set to the starting y-value for the plot.
(<) TVYEN is set to the final y-value for the plot.
(<) TVHIGH Is set to the same value as HIGH.
(<) TVLOW Is set to the same value as LOW.
(<) PGENVARG Arguments used for PGENV.
(<) IMFILE File containing the displayed image.


D.136  ILIST-List the data in an image (or spectrum)

Description: Displays the values of pixels in a dataset.

Parameters: IMAGE Name of image to be listed.

YSTART First Y value for list.
YEND Last Y value for list.
XSTART First X value for list.
XEND Last X value for list.
HARDCOPY Produce a file containing list?

Source comments S T

Lists the data in a subset of an image.

Command parameters -

IMAGE (Char) The name of the structure containing the image.
YSTART (Numeric) The Y value for the start of the subset.
YEND (Numeric) The Y value for the end of the subset.
XSTART (Numeric) The X value for the start of the subset.
XEND (Numeric) The X value for the end of the subset.

Command keywords -

HARDCOPY List data in a file ready for printing.
D.137  **ILOG-Takes the logarithm of an image**

**Description:** ILOG takes the base 10 log of each pixel in an image. Negative pixels are set to zero in the output image.

**Parameters:** IMAGE The datafile to be processed.

OUTPUT The name of the resulting image. If this is the same as IMAGE (the default) the operation will be performed in situ. Otherwise a new image will be created.

**See also:** FIGARO: [IALOG]

KAPPA: LOG10, LOGAR, LOGE.

**Source comments:**

Takes the base 10 log of an image. Any pixels that are negative will give a zero pixel in the resulting image.

Command parameters -

IMAGE The name of the structure containing the image.

OUTPUT The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

**Command keywords - None**

**User variables used - None**

KS / CIT 22nd April 1984

D.138  **IMAGE-Display an image**

**Usage:** image image ystart yend xstart xend low high xplaces yplaces atx aty xorigin yorigin xpixels ypixels optimize=? [autoscale=? negative=? aspect=? log=? erase=? hardcopy=?]

**Description:** This routine displays an image on the image display. Note that the colour tables are not changed by this command, nor is the device reset. The data can be logarithmised and/or histogram-optimised prior to display.

**Parameters:** IMAGE The image to be displayed.

YSTART The first Y value to be displayed.

YEND The last Y value to be displayed.

XSTART The first X value to be displayed.

XEND The last X value to be displayed.

LOW The lowest data value to be displayed. [0.]

HIGH The highest data value to be displayed. [1000.]

XPLACES If not 0, the number of sub-displays in X. Enter 0 to specify a display region explicitly through X/YORIGIN and X/YPIXELS. [1]

YPLACES If not 0, the number of sub-displays in Y. [1]
ATX  Which sub-display in X to use, counting from 1. [1]
ATY  Which sub-display in Y to use, counting from 1. [1]
XORIGIN  The first pixel in X to be used for display, counting from 0. [0]
YORIGIN  The first pixel in Y to be used for display, counting from 0. [0]
XPIXELS  How many pixels to use in X.
YPIXELS  How many pixels to use in Y.
OPTIMIZE  The degree of histogram optimisation to be applied. 0 for no optimisation, 1 for full optimisation. Optimisation is applied after taking common logarithms if LOG is YES. [0.5]
AUTOSCALE  Yes if the display thresholds are to be the minimum and maximum data values in the subset to be displayed. [T]
NEGATIVE  Yes if the auto-scaling should be reversed. [F]
ASPECT  Yes if data pixels are to be displayed as square pixels. [T]
LOG  Yes if the common logarithm of data is to be displayed rather than the data themselves. [F]
ERASE  Yes if the display is to be erased before display. [F]
HARDCOPY  Yes if the display is to be on the device set by the HARD command. The "idev" device is commonly a screen display, while the "hard" device is commonly a printer. Be wary of the NEGATIVE keyword in conjunction with HARDCOPY: A "positive" display will display the minimum as white and the maximum as black, and it in that sense negative. If you set the NEGATIVE keyword to YES, your hard copy will be "positive".
IDEV  The name of the imaging device, normally got from a global parameter which was set with the IDEV command.
HARD  The name of the "hard" device, normally got from a global parameter which was set with the HARD command.
IMARRAY  Information about the displayed part of the image and the part of the display used.
IMFILE  File name of the image displayed.

See also: FIGARO: IGREY, KAPPA: DISPLAY, GREYPLOT

Authors: KS: Keith Shortridge (AAO)
HME: Horst Meyerdierks (UoE, Starlink)

D.139  IMPOS-read positions from a file into environment variables

Description: Input a list of positions from a file to the environment variables from which CENTERS obtains its input.
CENTERS requires a list of approximate X,Y input positions which it reads from environment variables. Usually this list is created interactively with ICUR or IGCUR.
IMPOS creates the list by reading it from a text file, thus allowing CENTERS to be used non-interactively.
The input file is free-format, with one X,Y position per line. The X and Y values should be separated by one or more spaces and be expressed in pixels. Up to a hundred positions may be included.
Usage: impos file-name

Parameters: INPFLE INPFLE = _CHAR (Read) Name of the input file containing the list of positions.
XPIXELS XPIXELS = _REAL (Write) List of X coordinates (pixels).
YPIXELS YPIXELS = _REAL (Write) List of Y coordinates (pixels).
NPIXELS NPIXELS = _REAL (Write) Number of points in the list.

Source comments: A list of positions from a file to the environment variables from which CENTERS obtains its input.

The contents of an example input file containing positions for three objects might be:

103.4  67.8
231.6  134.5
246.7  89.2

D.140 IMULT-Multiplies two images (or two spectra)

Description: Each pixel in IMAGE is multiplied by the corresponding pixel of IMAGE1. The result is stored in OUTPUT.

Parameters: IMAGE Name of first image.
IMAGE1 Name of second image.
OUTPUT Name of resulting image.

See also: FIGARO: IADD, ISUB, IDIV, ICADD, ICSUB, ICMULT, ICDIV
KAPPA: CADD, CDIV, CMULT, CSUB, MATHS, ADD, SUB, MULT, DIV

Source comments: S U B

Adds, multiplies, divides or subtracts two images.

Command parameters -

IMAGE The name of the structure containing the first image.
IMAGE1 The name of the structure containing the second image data.

OUTPUT The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data (and any error or data quality information) a direct copy of the first image.

The command itself (IADD, IMULT, IDIV or ISUB) is used to differentiate between the two operations.

KS / CIT 26th Sept 1983
### D.141 INTERP-Interpolates points of a ‘spiketrum’ to form a spectrum

**Description:** INTERP interpolates between the points of a ‘spiketrum’ - an array where most of the data is zero except for a few elements whose values are set - to produce a spectrum. Interpolation is by spline fitting, except for those cases where there are not enough points available; in these cases a global polynomial will be used.

**Parameters:**
- **SPIKETRUM** The name of the spiketrum - an array where only a few elements are set to the correct values, the other elements all being zero - which is to be used to produce a spectrum by interpolation between those specified elements. SPIKETRUM has probably been produced by GSPIKE from a table of values.
- **SPECTRUM** The name of the spectrum to be produced by interpolation between the points of the spiketrum.
- **LOG** It doesn’t usually make much difference whether you fit to the log of the data or to the data itself, but if the data is changing very rapidly, particularly at the ends, the fit may be better constrained if the log is used.
- **LINEND** A spline fit tends to go a little wild at the ends of the fitted spectrum - before the first and after the last point defined by the spiketrum. Using linear interpolation in these extreme ranges prevents this, at the expense of the smoothness of the resulting curve. Usually, it doesn’t make much difference.

**Source comments:** INTERP / SPIFIT / LINTERP

Interpolates between the points of a ‘spiketrum’ to generate a spectrum. The INTERP command does this by spline interpolation, the SPIFIT command uses global polynomial fitting, and the LINTERP command uses linear interpolation.

**Command parameters -**
- SPIKETRUM The name of the structure containing the spiketrum
- ORDER The order for the global polynomial to be fitted (SPIFIT only).
- SPECTRUM The name of the result of the operation. This can be the same as for SPIKE. If not, a new structure is created, with everything but the data a direct copy of the input.

**Command keywords -**
- LOG (INTERP & SPIFIT only) Perform the interpolation on the log of the data
- LINEND (INTERP only) Use a linear interpolation for the ends of the data - spline fits can go wild outside the range of the defined points.

**User variables used -** None

KS / CIT 6th July 1984
D.142  IPLOTS-Plots successive cross-sections of an image, several to a page

Description: The command IPLOTS extracts cross-sections from an image and plots them on the current hardcopy device, several to a page.

Parameters: IMAGE Must be either a one- or two-dimensional.

WHOLE If WHOLE is set, the whole of each cross-section is plotted. Otherwise, the limits plotted are determined by the values of XSTART and XEND, which you will be prompted for if they were not specified in the command string. Note that WHOLE does not imply the plotting of EVERY cross-section.

XSTART Start of spectrum to be plotted.

XEND End of spectrum.

YSTART First spectrum to be plotted.

YEND Last spectrum to be plotted.

AUTOSCALE If set, then each spectrum is scaled according to its own minimum and maximum values. Otherwise the collective minimum and maximum of all the spectra is used to scale them all equally.

NSPECT Number of spectra per page.

LABEL The label that will appear at the top of the plot.

HARDCOPY If set, the plot is written to the device defined as the current hardcopy device. Generally, this is a disk file which will then have to be printed. If HARDCOPY is not set, the plot will go to the current softcopy device. The hard- and soft-copy devices are specified using the HARD and SOFT commands respectively.

COLOUR The colour for the data to be plotted in. The axes are always plotted in white. The colours allowed are Blue, White, Red, Green, Black, Cyan, Magenta, Yellow. Using Black will have the effect of erasing anything where the data is plotted. This only works on the Grinnell.

NEXT Used to pause between plots.

See also: FIGARO: ESPLOT, MSPLOT, SPLOT.

KAPPA: LINPLOT, MLINPLOT;
GAIA.

Source comments: O T S

Produces a plot of successive cross-sections of an image, several to a page. The plot is directed to the device defined by the user variable 'HARD', and by the value of the command keyword 'HARDCOPY'.

Command parameters -

IMAGE The data to be plotted. If there is X-axis information then this will be used. If not, the X-axis will just have to be the numbers from 1 to n.

XSTART The x-value at which plotting is to start. XSTART and XEND are only prompted for if WHOLE is not set.

XEND The x-value at which plotting is to end.

YSTART The first cross-section to be plotted.

YEND The last cross-section to be plotted.
LABEL  A label for the plot.
COLOUR The colour for the plot (only meaningful for the
Grinnell - later may be extended to map onto
different line types). The axes are always white.
NSPECT The number of cross-sections to be plotted per sheet.

Command keywords -

WHOLE The program is to display all of each cross-section.
AUTOSCALE If NO (the default), then each spectrum will be scaled to
the extrema of all the sections under consideration. If YES
then all spectra will be autoscaled individually.
HARDCOPY If set then output is sent to the device determined
by HARD. If no device is specified by HARD, or HARDCOPY is
not set, then IPOLOTS will attempt to plot on the current
SOFT device.
NEXT Used to pause between plots.

User variables - ("->" input, "<" output)

-> SOFT Specifies the device and type to be used for soft
plots. See the SOFT command for more details.
-> HARD Specifies the device and type to be used for hard
plots. See the HARD command for more details.

D.143  IPOW - Raises an image to a specified power

Description:  IPOW raises each pixel in an image to a specified power. This can be
used, for example, to take the square root of an image (by using a power of 0.5). Any
pixel for which the operation is not legal - e.g. -ve pixels being raised to a non-integer
power - are set to zero in the output image.

Parameters:  IMAGE Name of the datafile to be processed.

POWER The power to which the input image is to be raised. So setting POWER=0.5
will take the square root of the image, POWER=2.0 will square the image,
POWER=-1 will give the inverse of the image, etc.

OUTPUT The name of the resulting image. If this is the same as IMAGE (the default)
the operation will be performed in situ. Otherwise a new image will be created.

See also: KAPPA, POW

Source comments: W E R

 Raises the data in an image to a power of itself. This can be used
to multiply an image by itself (POWER=2.0) or to take the square
root of an image (POWER=0.5), or may be used with an arbitrary power.
Pixels whose value is such that the operation is illegal will give
a zero result.

Command parameters -
D.144 IRCONV-Converts data in Janskys to W/m**2/um

Description: IRCONV generates a spectrum in W/m**2/um, given a spectrum in some other units. At present, only Jansky, milli-Jansky and micro-Jansky can be handled.

Parameters: SPECTRUM A spectrum whose units are (currently) either Jansky, milli-Jansky, or micro-Jansky. It needs to be wavelength calibrated, in Angstroms or microns.

OUTPUT The resulting spectrum, whose units will be W/m**2/um. It can be the same as SPECTRUM, in which case the conversion will be performed in situ.

Source comments:

Converts a spectrum into AB magnitudes (ABCONV) or f-lambda units (erg/s/cm**2/Angstrom) (FLCONV), or W/m**2/um (IRCONV). The original units of the data may be Jy (Jansky), mJy (milli-Jansky), or uJy (micro-Jansky). Other possibilities may be added later.

Command parameters -

SPECTRUM The name of the structure containing the spectrum. Currently used for the spectrum. For FLCONV an x-axis data structure giving the wavelengths of the data elements is also required.

OUTPUT The name of the result of the operation. This can be the same as for SPECTRUM. If not, a new structure is created, with everything but the data a direct copy of the input.

Command keywords - None

User variables used - None
D.145  IREVX-Reverse an image (or spectrum) in the X-direction

**Description:** The data are reversed in X. The result is an image where the left and right have changed places.

**Parameters:**
- **IMAGE** Name of image to be reversed.
- **OUTPUT** Name of resulting image.

**See also:** FIGARO: [IREVY, IROT90, KAPPA, FLIP, REGRID, ROTATE]

**Source comments:**

Figaro routine to reverse an image (or a spectrum, which is treated as an image with one dimension 1). The reversal is in the X-direction or the Y-direction.

**Command parameters -**

- **IMAGE** (Character) The name of the file containing the image. The data and error components are reversed; the quality component is reversed implicitly by internally using flagged values; the x-axis or y-axis data component is reversed if it contains anything other than just the bin numbers.

- **OUTPUT** (Character) The name of the result of the operation. This can be the same as IMAGE, in which case the operation is performed in situ.

**Command keywords -** None

**User variables -** None

KS / CIT 20th July 1983

D.146  IREVY-Reverse an image in the Y-direction

**Description:** The data are reversed in Y. The result is an image where the top and bottom have changed places.

**Parameters:**
- **IMAGE** Name of image to be reversed.
- **OUTPUT** Name of resulting image.

**See also:** FIGARO: [IREVX, IROT90, KAPPA, FLIP, REGRID, ROTATE]

**Source comments:**

Figaro routine to reverse an image (or a spectrum, which is treated as an image with one dimension 1). The reversal is in the X-direction or the Y-direction.

**Command parameters -**
D.147  IRFLAT-Generates a ripple spectrum from an IR spectrum

**Description:** IRFLAT generates a ripple spectrum from an IR spectrum. In principle, the (raw or ratioed) data can then be divided by this ripple spectrum to take out the effects of different detector responses, which produces a periodic pattern on the continuum of the data (with period equal to the number of detectors).

**Parameters:**

- **SPECTRUM** Spectral data file.
- **CGS2** Flag indicating whether to operate in ‘CGS 2’ mode.
- **PERIOD** Period of the ripple (pixels).
- **OUTPUT** The name of resulting ripple spectrum.
- **XSTART** First X value for region to be used.
- **XEND** Second X value for region to be used.
- **MORE** Include additional ranges? (If TRUE the prompts for XSTART and XEND are repeated for another region.)

**Source comments:**

Figaro function that produces a "flatfield" ripple spectrum from an infrared spectrum, by averaging the data from regions of the spectrum uncontaminated with spectral features (i.e. assumed flat) to determine the relative response of each detector or scan. The output spectrum can be divided into the original spectrum using IDIV to flat field the data.

The program is used to remove two kinds of ripple from spectra. In instruments which interleave a number of scan positions to give a fully sampled spectrum (such as CGS3 and CGS4), the program removes ripple which results from seeing or transparency fluctuations between scan positions. In an instrument such as CGS2 it can remove the ripple which results from the fact that the flatfield (i.e. relative detector responses) is different for extended and point sources. In the case of CGS2 data it makes use of a .MORE.PIXELS extension in the data which specifies the detector and scan position corresponding to each pixel. If this structure is not present it prompts for a period and assumes...
a periodic ripple. The period will normally be the oversampling factor, typically 2 or 3 for CGS4 or CGS3 data.

If the program is run in batch only one region can be specified. Multiple regions can only be specified in interactive mode.

Command parameters -

SPECTRUM (Character) The name of the file containing the spectrum to be used.
CGS2 (Logical) Flag indicating whether to operate in 'CGS 2' mode.
PERIOD (Real) The period of the ripple (in pixels).
OUTPUT (Character) The name of the resulting ripple spectrum.
XSTART (Real) First X value for region to be used.
XEND (Real) Second X value for region to be used.

Command keywords -

MORE If YES, the prompts for XSTART and XEND are repeated for another region.

10th Dec 1990 - JAB / JAC

D.148  IRFLUX-Flux calibrates an IR spectrum using a black-body model

Description: Flux calibrates an IR spectrum using a black body model for a standard star.

Parameters: SPECTRUM Name of Star spectrum.
STANDARD Name of Standard spectrum.
TEMP Temperature of standard.
CALTYPE Type of calibration data.
MAG Magnitude of standard star.
FLUX Flux of standard star at calib wavelength (mJy).
WAVE Calibration wavelength (microns).
OUTPUT Name of resulting spectrum.

Source comments:

Flux calibrates an IR spectrum using a black body model for a standard star

Command parameters -

SPECTRUM The name of the structure containing the first image.
STANDARD The name of the structure containing the second image data.
TEMP The temperature of the black body to be used.
CALTYPE The type of calibration data. A single character as follows: 'J','H','K','L','M' - magnitude in a standard band.
'F' - Flux at specified wavelength

MAG The magnitude of the standard used.

FLUX The flux in mJy of the standard.

WAVE The wavelength at which the flux is specified.

OUTPUT The name of the result of the operation. This can be the same as for SPECTRUM. If not, a new structure is created, with everything but the data a direct copy of the input.

JAB / JAC 19th Sep 1990

D.149 IROT90-Rotates an image through 90 degrees

Description: The datafile is rotated through 90 degrees in a clockwise direction, unless ANTI is specified, in which case the rotation is anti-clockwise.

Parameters: IMAGE The name of the structure containing the image.

OUTPUT The name of the result of the operation. This can be the same as for IMAGE. In any case, a new structure is created, with everything but the data a direct copy of the input.

ANTI If set, the rotation is anti-clockwise.

See also: FIGARO: IREVX, IREVY

KAPPA: FLIP, REGRID, ROTATE

Source comments: T 9 0

Rotates an image through 90 degrees

Command parameters -

IMAGE The name of the structure containing the image.

OUTPUT The name of the result of the operation. This can be the same as for IMAGE. In any case, a new structure is created, with everything but the data a direct copy of the input.

Command keywords -

ANTI If set, the rotation is anti-clockwise.

KS / CIT 7th March 1983

D.150 ISCAN-Displays cuts through an IMAGE

Description: Either the user can go into a loop in which (s)he is prompted for the start and end x-sects to extract, or the program will go through the data, displaying successive cuts.
Parameters: IMAGE  IMAGE = FILE (Read) Image to take cuts from
XSTART  XSTART = REAL (Read) Starting wavelength (or pixel number if not calibrated)
XEND  XEND = REAL (Read) End wavelength etc.
YSTART  YSTART = INTEGER (Read) Starting cross-section
YEND  YEND = INTEGER (Read) End cross-section
YBLOCK  YBLOCK = INTEGER (Read) Width to extract from data in cross-sections (if scanning)
SCAN  SCAN = LOGICAL (Read) If to scan through data
HARDCOPY  HARDCOPY = LOGICAL (Read) If to plot in hard-copy

Source comments: available

D.151  ISCRUNCH-Rebin an image to linear wavelength scale given IARC results

Description: ISCRUNCH rebins an image that contains a set of spectra, so that the resulting data have either a linear wavelength scale or a wavelength scale that is logarithmic (i.e. has a constant velocity step). ISCRUNCH differs from SCRUNCH in that it uses a file that contains a set of polynomial fits to a 2-D arc in order to get the wavelength-channel relationship for each cross-section in the image, while SCRUNCH uses the values in the image’s X-axis array. Note that ISCRUNCH is not suitable for data in which the bins contain data that covers discontinuous or overlapping wavelength ranges (FIGS data, for example).

Parameters: IMAGE  The name of the image to be scrunched. Each cross-section of the image is handled separately, and the channel-wavelength relationship for each cross-section is assumed to be given by a set of polynomial coefficients held in a file. These coefficients will have been determined by an analysis of a 2-D arc, and IMAGE must specify data of the same dimensions as the 2-D arc used.
FILE  ISCRUNCH is driven from the results of a 2-D arc fit - usually performed by IARC - and needs to know where to find the file that contains the polynomial coefficients determined by this fit. Usually, the default value will be the file generated the last time IARC was run. If no extension is specified, .IAR is assumed.
WSTART  The wavelength of the center of the first bin of the resulting output data.
WEND  Normally, represents the wavelength of the center of the last bin of the resulting output data. Because this is not always the most convenient value to supply, ISCRUNCH will allow WEND to be used to specify the wavelength increment value (for linear data, this is the constant wavelength difference between bins; for logarithmic data it is the constant velocity step in Km/sec). The way WEND is interpreted may be controlled explicitly by the FINAL and INCREMENT hidden keywords, but by default ISCRUNCH will assume WEND is an incremental value if it is less than WSTART, and a final value if it is greater than WSTART.
BINS  The number of elements to be used for the wavelength dimension of the resulting image.
OUTPUT The name of the resulting data file containing the scrunched image. The structure of OUTPUT will be the same as that of the input file, except that the data array will generally be a different size, and any error array will have been deleted. (Eventually ISCRUNCH may be able to calculate the errors in the new image.)

INCREMENT If set, ISCRUNCH will assume WEND is an incremental value (a velocity or wavelength step), even if it is greater than WSTART.

FINAL If set, ISCRUNCH will assume that WEND is the wavelength of the final element of the resulting data, even if it is less than WSTART.

LOG Controls whether or not the data is binned to a linear or a logarithmic wavelength scale.

LINEAR Specifies that linear rather than quadratic interpolation is to be used during the scrunching. Usually, better results are obtained with quadratic interpolation. Note that LINEAR is the opposite of QUAD, and NOT of LOG.

QUAD Specifies that quadratic rather than linear interpolation is to be used during the scrunching. Usually, better results are obtained with quadratic interpolation.

DENSITY If the input data represents data whose units are flux per unit wavelength (AB magnitudes, Janskys, etc) rather than total flux over a wavelength range (photons, for example) then it should be scrunched so as to conserve the mean value of the data, and DENSITY should be set. See HELP FIGARO TECHNIQUES WAVELENGTH for more details on this point.

Source comments:

Applies the set of polynomial fits determined by IARC to an image, re-binning each cross-section of the image to either a linear or logarithmic wavelength scale. ISCRUNCH uses the results from a single 2-D arc fit as performed by IARC to get the channel/wavelength relation. ISCRUNI uses two such fits, and performs a linear interpolation between the two.

The routine can either conserve flux or the mean value of the data. Conserving flux is appropriate where the data is actually in flux units (photons/sec, for example), but not when the data is in units of flux per unit wavelength (AB magnitudes, Janskys, etc). Consider the case where each input bin maps to two output bins; if the data is in flux units - photon counts, for example - then flux should be conserved and the mean data level should drop by a factor 2; if the data is in magnitudes, then the rebinning should not change the mean level. The program tries to determine for itself whether the data is in flux or in flux per unit wavelength by looking at the units, but uses a command keyword (DENSITY) to confirm its guess.

Command parameters -

IMAGE (Character) Image to be scrunched.
FILE (Character) File containing results of 2-D arc fit as produced by IARC. If no extension supplied, .IAR is assumed.
FILE2 (Character) ISCRUNI only. Second file containing
IARC results.

**FRACTION** (Numeric) ISCRUNI only. Controls the interpolation between the two sets of wavelength information. Value used=

\[(\text{File value}) + (\text{File2 value} - \text{File value}) \times \text{FRACTION}\]

**WSTART** (Numeric) The wavelength of the CENTER of the first bin of the resulting scrunched spectrum.

**WEND** (Numeric) The wavelength of the CENTER of the final bin of the resulting scrunched spectrum. If WEND is less than WSTART, then SCRUNCH assumes that it is the increment rather than the final value that is being specified. If the scrunch is logarithmic and WSTART is greater than WEND, SCRUNCH assumes that the WEND value represents a velocity in km/sec. These assumptions can be controlled directly by the keywords INCREMENT and FINAL, if they will not give the desired effect.

**BINS** (Numeric) The number of bins for the resulting spectrum.

**OUTPUT** (Character) Name of resulting image. Note that an image cannot be scrunched into itself, so a new output file will always be created.

**Command keywords** -

- **LOG** Bin into logarithmic wavelength bins.
- **DENSITY** Treat input data as being in units of flux per unit wavelength.
- **LINEAR** Use linear interpolation when rebinning.
- **QUAD** Use quadratic interpolation when rebinning.
- **INCREMENT** WEND is an increment value, even though it is > WSTART.
- **FINAL** WEND is a final value, even though it is < WSTART.

**User variables** - None

---

**D.152 ISCRUNI-Like ISCRUNCH, but interpolates between two IARC result sets**

**Description:** ISCRUNI rebins an image that contains a set of spectra, so that the resulting data have either a linear wavelength scale or a wavelength scale that is logarithmic (i.e. has a constant velocity step). ISCRUNI differs from SCRUNCH in that it uses files that contain a set of polynomial fits to a 2-D arc in order to get the wavelength-channel relationship for each cross-section in the image, while SCRUNCH uses the values in the image’s X-axis array. It differs from ISCRUNCH in using 2 such files, and taking values intermediate between the two when calculating its wavelengths. Note that ISCRUNI is not suitable for data in which the bins contain data that covers discontinuous or overlapping wavelength ranges (FIGS data, for example).

**Parameters:** **IMAGE** The name of the image to be scrunched. Each cross-section of the image is handled separately, and the channel-wavelength relationship for each cross-section is assumed to be given by a set of polynomial coefficients held in a
file. These coefficients will have been determined by an analysis of a 2-D arc, and IMAGE must specify data of the same dimensions as the 2-D arc used.

FILE ISCRUNI is driven from the results of a 2-D arc fit - usually performed by IARC - and needs to know where to find the file that contains the polynomial coefficients determined by this fit. Usually, the default value will be the file generated the last time IARC was run. If no extension is specified, .IAR is assumed.

FILE2 The name of the second of the sets of IARC results used by ISCRUNI in calculating the wavelength to channel relation for its input data.

FRACTION When ISCRUNI calculates an input wavelength for a pixel, it calculates it first using the data from FILE, then repeats the calculation using the data from FILE2, and combines them to get an intermediate result using the supplied value of FRACTION. FRACTION=0.5 will give an average of the two, FRACTION=0.0 will effectively just use the values from FILE and FRACTION=1.0 will just use the values from FILE2.

WSTART The wavelength of the center of the first bin of the resulting output data.

WEND Normally, represents the wavelength of the center of the last bin of the resulting output data. Because this is not always the most convenient value to supply, ISCRUNI will allow WEND to be used to specify the wavelength increment value (for linear data, this is the constant wavelength difference between bins; for logarithmic data it is the constant velocity step in Km/sec). The way WEND is interpreted may be controlled explicitly by the FINAL and INCREMENT hidden keywords, but by default ISCRUNI will assume WEND is an incremental value if it is less than WSTART, and a final value if it is greater than WSTART.

BINS The number of elements to be used for the wavelength dimension of the resulting image.

OUTPUT The name of the resulting data file containing the scrunched image. The structure of OUTPUT will be the same as that of the input file, except that the data array will generally be a different size, and any error array will have been deleted. (Eventually ISCRUNI may be able to calculate the errors in the new image.)

INCREMENT If set, ISCRUNI will assume WEND is an incremental value (a velocity or wavelength step), even if it is greater than WSTART.

FINAL If set, ISCRUNI will assume that WEND is the wavelength of the final element of the resulting data, even if it is less than WSTART.

LOG Controls whether or not the data is binned to a linear or a logarithmic wavelength scale.

LINEAR Specifies that linear rather than quadratic interpolation is to be used during the scrunching. Usually, better results are obtained with quadratic interpolation. Note that LINEAR is the opposite of QUAD, and NOT of LOG.

QUAD Specifies that quadratic rather than linear interpolation is to be used during the scrunching. Usually, better results are obtained with quadratic interpolation.

DENSITY If the input data represents data whose units are flux per unit wavelength (AB magnitudes, Janskys, etc) rather than total flux over a wavelength range (photons, for example) then it should be scrunched so as to conserve the mean value of the data, and DENSITY should be set. See HELP FIGARO TECHNIQUES WAVELENGTH for more details on this point.
**Source comments:**

ISCRUNCH / ISCRUNI

Applies the set of polynomial fits determined by IARC to an image, re-binning each cross-section of the image to either a linear or logarithmic wavelength scale. ISCRUNCH uses the results from a single 2-D arc fit as performed by IARC to get the channel/wavelength relation. ISCRUNI uses two such fits, and performs a linear interpolation between the two.

The routine can either conserve flux or the mean value of the data. Conserving flux is appropriate where the data is actually in flux units (photons/sec, for example), but not when the data is in units of flux per unit wavelength (AB magnitudes, Janskys, etc). Consider the case where each input bin maps to two output bins; if the data is in flux units - photon counts, for example - then flux should be conserved and the mean data level should drop by a factor 2; if the data is in magnitudes, then the rebinning should not change the mean level. The program tries to determine for itself whether the data is in flux or in flux per unit wavelength by looking at the units, but uses a command keyword (DENSITY) to confirm its guess.

**Command parameters -**

- **IMAGE** (Character) Image to be scrunched.
- **FILE** (Character) File containing results of 2-D arc fit as produced by IARC. If no extension supplied, .IAR is assumed.
- **FILE2** (Character) ISCRUNI only. Second file containing IARC results.
- **FRACTION** (Numeric) ISCRUNI only. Controls the interpolation between the two sets of wavelength information. Value used = (File value) + (File2 value - File value) * FRACTION
- **WSTART** (Numeric) The wavelength of the CENTER of the first bin of the resulting scrunched spectrum.
- **WEND** (Numeric) The wavelength of the CENTER of the final bin of the resulting scrunched spectrum. If WEND is less than WSTART, then SCRUNCH assumes that it is the increment rather than the final value that is being specified. If the scrunch is logarithmic and WSTART is greater than WEND, SCRUNCH assumes that the WEND value represents a velocity in km/sec. These assumptions can be controlled directly by the keywords INCREMENT and FINAL, if they will not give the desired effect.
- **BINS** (Numeric) The number of bins for the resulting spectrum.
- **OUTPUT** (Character) Name of resulting image. Note that an image cannot be scrunched into itself, so a new output file will always be created.

**Command keywords -**
LOG Bin into logarithmic wavelength bins.
DENSITY Treat input data as being in units of flux per unit wavelength.
LINEAR Use linear interpolation when rebinning.
QUAD Use quadratic interpolation when rebinning.
INCREMENT WEND is an increment value, even though it is > WSTART.
FINAL WEND is a final value, even though it is < WSTART.

User variables - None

KS / CIT 22nd June 1984

D.153 ISEDIT-Allows interactive editing of a 1-D or 2-D spectrum

Description: Interactively edit a 1-D or 2-D spectrum allowing points or complete spectra to be removed.

Parameters: IMAGE IMAGE should be the name of a 1-D or 2-D file.

WHOLE If set, the whole of the spectrum is plotted. Otherwise, the limits plotted are determined by the values of XSTART and XEND, which you will be prompted for if they were not specified in the command string.

XSTART Specifies the first X value to be plotted, in the units used by the data - angstroms, for example, if the data is wavelength calibrated. RESET will set XSTART to the first X value in the data.

XEND Specifies the last X value to be plotted, in the units used by the data - angstroms, for example, if the data is wavelength calibrated. RESET will set XEND to the last X value in the data.

YVALUE The number of the row to be worked on next.

OUTPUT The name of the resulting edited data.

Source comments: D I T

Interactively edit a 1-D or 2-D spectrum allowing points or complete spectra to be removed

Command parameters -

'IMAGE' The name of the input file.

'XSTART' Starting X value to plot

'XEND' Ending X value to plot

'YVALUE' Number of the row to be plotted.

'OUTPUT' The name of the output file.

Command keywords -

'WHOLE' Plot all of spectrum

JAB / JAC 11th Dec 1990

D.154 ISHIFT-Applies a linear X and a linear Y shift to an image

Description: Applies a linear X and Y shift to an image.
The VARIANCE array, if present, is propagated in exactly the same way as the DATA array. This procedure it not formally correct if re-sampling occurs (that is, if either of the shifts is non-integer) and in this case the resulting variance will probably under-estimate the true error.

**Parameters:**

- **IMAGE** Name of image to be shifted.
- **XSHIFT** Shift in X.
- **YSHIFT** Shift in Y.
- **XSPLIT** Subdivision of X pixels.
- **YSPLIT** Subdivision of Y pixels.
- **OUTPUT** Name of resulting image.

**Source comments:**

I F T

Shifts an image in both X and Y, by, in each direction, a constant amount expressed in pixels.

The VARIANCE array, if present, is propagated in exactly the same way as the DATA array. This procedure it not formally correct if re-sampling occurs (that is, if either of the shifts is non-integer) and in this case the resulting variance will probably under-estimate the true error.

Command parameters -

**IMAGE** The name of the structure containing the image.

**XSHIFT** (Numeric) The number of pixels the image is to be shifted in X. A -ve number shifts towards lower numbered pixels.

**YSHIFT** (Numeric) The number of pixels the image is to be shifted in Y. Sense is as for XSHIFT.

**XSPLIT** (Numeric) The number of sub-divisions to be made in each of the original pixels in X.

Note that if both YSHIFT and XSHIFT are integers, the routine will just perform a fast data move, rather than hit the problem with the big hammer of a general rebinning. YSPLIT and XSPLIT will be ignored.

**YSPLIT** (Numeric) The number of sub-divisions to be made in each of the original pixels in Y.

Note that if XSPLIT or YSPLIT are greater than 1, the data will be interpolated using a fit to a 2-D parabola. This increases the accuracy (sometimes) of the rebinning, but results in increased CPU usage.

**OUTPUT** The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.
D.155  ISMOOTH-2-D smooth of image using 9-point smoothing algorithm

Description: ISMOOTH, which smooths a 2-D image using a nine-point 2-D smoothing algorithm. This is a 2-D extension of the 3-point spectrum smoothing algorithm which replaces each pixel with 1/2 its own value plus 1/4 of the value of each of its two neighbours. The routine used here replaces each pixel with a fraction of its own value plus a fraction of the value of its eight nearest neighbours. The fraction from a ‘corner’ neighbour is 1.414 times less than from a ‘side’ neighbour. Repeating the process increases the amount of smoothing applied, and the operation is close in effect to a gaussian convolution.

Parameters: IMAGE Name of image to be smoothed.  
FRACTION Fraction of pixel data for redistribution.  
REPEAT Number of times to repeat basic smoothing.  
OUTPUT Name of resulting image.

See also: FIGARO: [CONV3] [IXSMOOTH] [MEDFILT] [KAPPA] [CONVOLVE] [FFCLEAN] [GAUSMOOTH] [MEDIAN]

Source comments: 
This routine is the main body of ISMOOTH, which smooths a 2-D image using a nine-point 2-D smoothing algorithm. This is a 2-D extension of the 3-point spectrum smoothing algorithm which replaces each pixel with 1/2 its own value plus 1/4 of the value of each of its two neighbours. The routine used here (see GEN_ASMOTH for details) replaces each pixel with a fraction of its own value plus a fraction of the value of its eight nearest neighbours. The fraction from a ‘corner’ neighbour is 1.414 times less than from a ‘side’ neighbour. Repeating the process increases the amount of smoothing applied, and the operation is close in effect to a gaussian convolution.

Command parameters - 

IMAGE The name of the structure containing the image. 
FRACTION (Numeric) The fraction of each pixel’s value to be distributed to its neighbours. The higher this value the greater the smoothing effect. 
REPEAT (Numeric) The number of times the operation is to be repeated. The higher this value, the greater the smoothing effect. 
OUTPUT The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure
is created, with everything but the data a direct copy of the input.

KS / CIT 21st June 1983

D.156  ISPLOT-Plots successive cross-sections through an image

Description: The ISPLOT command will plot a series of cross-sections through an image as hidden line histograms on the current hard or soft graphics device.

Parameters: IMAGE The name of the image whose data is to be plotted by ISPLOT. Successive cross-sections (cuts of constant Y) are plotted as hidden line histograms. ISPLOT is probably most appropriate when IMAGE contains a set of long-slit spectra. If IMAGE is a 1-D array, SPLOT should be used.

HARDCOPY If set, the plot is written to the device defined as the current hardcopy device. Generally, this is a disk file which will then have to printed. If HARDCOPY is not set, the plot will go to the current softcopy device. The hard and soft copy devices are specified using the HARD and SOFT commands respectively.

YSTART ISPLOT plots a series of cross-sections (cuts of constant Y value). YSTART and YEND are used to delimit these. YSTART should be the Y value of the lowest numbered cross-section to be plotted. Note that REVERSE should be used to cause cross-sections to be plotted in reverse order. Attempting to obtain the same effect by specifying YSTART < YEND will not work.

YEND The Y value of the highest numbered cross-section plotted. See the help text for YSTART for more details.

WHOLE If set, the whole of each cross-section is plotted. Otherwise, the limits plotted are determined by the values of XSTART and XEND, which you will be prompted for if they were not specified in the command string. Note that WHOLE does not imply the plotting of EVERY cross-section.

AUTOSCALE If set, the plot is scaled so that all of the data to be plotted just fits on the display. Otherwise, the scale of the plot is determined by the values of HIGH, LOW, and OFFSET, which you will be prompted for if they were not specified in the command string.

XSTART The first X value to be plotted, in the units used by the data - angstroms, for example, if the data is wavelength calibrated. XSTART can be set before the start of the data, if necessary. RESET will set XSTART to the first X value in the data.

XEND The last X value to be plotted, in the units used by the data - angstroms, for example, if the data is wavelength calibrated. XEND can be set after the end of the data, if necessary. RESET will set XEND to the last X value in the data.

REVERSE If set, the cross-sections will be plotted in the reverse of the order implied by the values of YSTART and YEND. Sometimes two plots, one reversed, the other not, are needed to give a proper picture of the data in IMAGE.

HIGH The maximum data value to be plotted - i.e. the top Y axis value for the plot.

LOW The minimum data value to be plotted - i.e. the bottom Y axis value for the plot.
OFFSET  A value added to each cross-section in order to bias it up above the previous cross-section. Offset is normally calculated by the program, but can be overridden - there may be some reason to set it to zero, for example.

SLANT  A number of elements of the data array by which successive cross-sections are to be offset in order to obtain a ‘3-D’ effect for the plot. A positive value causes a slant to the right, a negative value a slant to the left.

LABEL  The label that will appear at the top of the plot.

ERASE  Specifies that the screen is to be erased before the plot is made. Usually ERASE and AXES will not be set when a plot is superimposed on a previous one.

AXES  Specifies that the axes for the plot are to be drawn. These should be omitted if the plot is being superimposed on a previous one, or sometimes just to save plotting time.

COLOUR  The colour for the data to be plotted in. The axes are always plotted in white. The colours allowed are Blue, White, Red, Green, Black, Cyan, Magenta, Yellow. Using Black will have the effect of erasing anything where the data is plotted. This only works on the Grinnell.

Source comments:

S P L O T

Produces a hidden line histogram plot of an image. This routine is really intended for the case where the image is a 2-D set of spectra. The plot may be slanted to give a ‘3-D’ effect if required, and may be in reverse order. The plot is directed to the device defined by the user variables ‘SOFT’ and ‘HARD’, and by the value of the command keyword ‘HARDCOPY’, so will appear immediately if these specify a video device (VT125, Grinnell, etc.). If a hardcopy device is specified, the file for that device will be produced, but S P L O T does not attempt to spool it off for printing.

Command parameters -

IMAGE  The data to be plotted. This should have a two dimensional data structure. If there is an x-axis component this will be used to give the x-axis. If not, the x-axis will just have to be the numbers from 1 to n.

YSTART  The y-value at which plotting is to start.

YEND  The y-value at which plotting is to end.

XSTART  The x-value at which plotting is to start.

XEND  The x-value at which plotting is to end. (XSTART and XEND are not required if the WHOLE keyword is set.)

HIGH  The maximum value to be used for the plot.

LOW  The minimum value to be used for the plot. (HIGH and LOW are not required if the AUTOSCALE keyword is set.)

OFFSET  A bias to be added to each successive cross-section to offset it from the previous one. Normally, this will be allowed to default, unless it is explicitly specified in the command line.

LABEL  A label for the plot.
COLOUR The colour for the plot (only meaningful for the Grinnell - later may be extended to map onto different line types). The axes are always white.

Command keywords -

AUTOSCALE The program is to work out the values for HIGH and LOW, using the maximum and minimum values in the data over the specified range.

WHOLE The program is to plot the whole of each cross-section of the image - note, not all the image, i.e. not every cross-section.

HARDCOPY The plot is to produce a hard copy.

AXES Axes will be plotted.

ERASE The screen will be erased before the plot.

REVERSE The cross-sections will be plotted in reverse order.

User variables - (">" input, "<" output)

(>) SOFT Specifies the device and type to be used for soft plots. See the SOFT command for more details.

(>) HARD Specifies the device and type to be used for hard plots. See the HARD command for more details.

(<) TVDIM is set to 1 if a single spectrum is plotted, to 2 otherwise.

(<) TVYST is set to the starting y-value for the plot.

(<) TVYEN is set to the final y-value for the plot.

(<) TVXST is set to the starting x-value for the plot.

(<) TVXEN Is set to the final x-value for the plot.

(<) TVHIGH Is set to the same value as HIGH.

(<) TVLOW Is set to the same value as LOW.

(<) TVFILE Is set to the value of IMAGE.

(<) TVCOLOR Is set to the GRPCKG code for the plot colour. (The TV.. variables are intended for use by cursor routines, and reflect the settings for the last plot made, even if XSTART etc are changed.)

(Other user variables may be set by the command processor, in connection with the parameter values.)

KS / CIT 5th March 1984

D.157 ISTAT-Provides some statistics about an image (max, min etc.)

Description: ISTAT examines an image, or a rectangular subset of an image or spectrum, and outputs a number of statistics about it, such as maximum and minimum value, mean and sigma, etc. The results are used to set Figaro user variables, so can be used by Figaro procedures. There are a number of examples in the documentation.

Parameters: IMAGE The name of the image (or spectrum) for which statistics are to be determined.

YSTART First Y value to be used.

YEND Last Y value to be used.
**XSTART**  First X value to be used.

**XEND**  Last X value to be used.

**PASS2**  Setting PASS2 makes ISTAT take two passes through the data in order to calculate Sigma more accurately. This accuracy is, of course, gained at the expense of time taken. For cases where fewer than 10,000 pixels are involved, ISTAT always uses two passes, since the other overheads dominate the time taken. If in doubt, try both, and see if the slower algorithm gives significantly different answers to the faster one. If so, use PASS2.

**MEDIAN**  Setting MEDIAN makes ISTAT calculate the median value in the specified range. This can be a time-consuming operation, so is not performed by default.

See also:  [KAPPA], [HISTAT], [HISTOGRAM], [MSTATS], [NUMB], [STATS], [GAIA]

Source comments:

Examines an image (or a subset of an image) and determines a number of useful statistics about the data in it, such as the mean, max and minimum values. For a full list, see below under 'user variables'.

Command parameters -

**IMAGE**  (Char)  The name of the structure containing the image.

**YSTART**  (Numeric)  The Y value for the start of the subset.

**YEND**  (Numeric)  The Y value for the end of the subset.

**XSTART**  (Numeric)  The X value for the start of the subset.

**XEND**  (Numeric)  The X value for the end of the subset.

Command keywords

**PASS2**  If set, the sigma value is calculated using two passes through the data. The one-pass algorithm normally used is prone to rounding error when large numbers of pixels are involved, but is rather faster. If fewer than 10,000 pixels are involved, ISTAT always uses two passes, since the other overheads dominate the time taken.

**MEDIAN**  If set, the median value of the image data is calculated (a possibly time-consuming business).

User variables -  ("<" output)

- `<STAT_TOTAL>`  The sum of the data.
- `<STAT_MIN>`  The minimum data value.
- `<STAT_MAX>`  The maximum data value.
- `<STAT_MEAN>`  The mean data value.
- `<STAT_MEDIAN>`  The median data value (only set if MEDIAN set)
- `<STAT_XMAX>`  The x-value of the pixel where the max was found.
- `<STAT_YMAX>`  The y-value of the pixel where the max was found.
- `<STAT_XMIN>`  The x-value of the pixel where the min was found.
- `<STAT_YMIN>`  The y-value of the pixel where the min was found.
- `<STAT_SIGMA>`  The standard distribution of the data.
- `<STAT_SIZE>`  The number of pixels examined.
D.158 ISTRETCH—Stretches and shifts an image in X and Y.

**Description:** Shifts an image in both X and Y, by, in each direction, a constant amount expressed in pixels and stretches it by a given factor - this is a linear stretch, combined with a shift. Note that this is functionally a superset of ISHIFT, but ISHIFT should be used for cases where the stretch is 1.0 in both X and Y, since it uses a simpler algorithm for very simple cases.

**Parameters:**
- **IMAGE** Name of image to be stretched.
- **XSTRETCH** Stretch factor in X.
- **YSTRETCH** Stretch factor in Y.
- **XSHIFT** Shift in X.
- **YSHIFT** Shift in Y.
- **XSPLIT** Subdivision of X pixels.
- **YSPLIT** Subdivision of Y pixels.
- **OUTPUT** Name of resulting image.

**See also:** KAPPA, BLOCK, COMPAVE, COMPIK, PIXDUPE, REGRID, SQORST.

**Source comments:**

Shifts an image in both X and Y, by, in each direction, a constant amount expressed in pixels and stretches it by a given factor - this is a linear stretch, combined with a shift. Note that this is functionally a superset of ISHIFT, but ISHIFT should be used for cases where the stretch is 1.0 in both X and Y, since it uses a simpler algorithm for very simple cases.

**Command parameters**
- **IMAGE** The name of the structure containing the image.
- **XSTRETCH** (Numeric) The stretch factor to be applied in X.
- **YSTRETCH** (Numeric) The stretch factor to be applied in Y.
- **XSHIFT** (Numeric) The number of pixels the image is to be shifted in X. A -ve number shifts towards lower numbered pixels.
- **YSHIFT** (Numeric) The number of pixels the image is to be shifted in Y. Sense is as for XSHIFT.
- **XSPLIT** (Numeric) The number of sub-divisions to be made in
each of the original pixels in X.

YSPLIT (Numeric) The number of sub-divisions to be made in each of the original pixels in Y.

Note that if XSPLIT or YSPLIT are greater than 1, the data will be interpolated using a fit to a 2-D parabola. This increases the accuracy (sometimes) of the rebinning, but results in increased CPU usage.

OUTPUT The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input. Note that the size of the image is the same as before - so either some pixels will be stretched out of the image, or some pixels will just be zero.

Command keywords - None

KS / CIT 5th Dec 1983

D.159 ISUB-Subtracts two images (or two spectra)

Description: Subtracts from the value of each pixel in an image the value of the corresponding pixel in a second image.

Parameters: IMAGE Name of first image.
IMAGE1 Name of second image.
OUTPUT Name of resulting image.

See also: FIGARO: IADD, IMULT, IDIV, ICADD, ICSUB, ICMULT, ICDIV
          KAPPA: CADD, CDIV, CMULT, CSUB, MATHS, ADD, SUB, MULT, DIV

Source comments: SUB

Adds, multiplies, divides or subtracts two images.

Command parameters -

IMAGE The name of the structure containing the first image.

IMAGE1 The name of the structure containing the second image data.

OUTPUT The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data (and any error or data quality information) a direct copy of the first image.

The command itself (IADD, IMULT, IDIV or ISUB) is used to differentiate between the two operations.
D.160  ISUBSET- Produces a subset of an image

Description: ISUBSET takes a subset of an image. The result is a new image which is a rectangular subset of the original data.

Parameters: IMAGE The name of the image from which a subset is to be taken. IMAGE may be 1 or 2 dimensional.

YSTART The value of the Y-coordinate for the first pixel in the input image to be included in the subset. Note that the subsetting is done on a whole pixel basis, so the resulting subset may not begin exactly on YSTART, but will start with the first pixel that covers that Y value.

YEND The value of the Y-coordinate for the last pixel in the input image to be included in the subset. Note that the subsetting is done on a whole pixel basis, so the resulting subset may not end exactly on YEND, but will end with the first pixel that covers that Y value.

XSTART The value of the X-coordinate for the first pixel in the input image to be included in the subset. Note that the subsetting is done on a whole pixel basis, so the resulting subset may not begin exactly on XSTART, but will start with the first pixel that covers that X value.

XEND The value of the X-coordinate for the last pixel in the input image to be included in the subset. Note that the subsetting is done on a whole pixel basis, so the resulting subset may not end exactly on XEND, but will end with the first pixel that covers that X value.

OUTPUT The name of the resulting subset image. If this is different to IMAGE, then a new file will be created, with everything bar the data a copy of the original image. If OUTPUT is the same as IMAGE, then the input file will be modified to contain just a subset of its original data.

Source comments:

Creates a subset of an image.

Command parameters -

IMAGE (Char) The name of the structure containing the image.
YSTART (Numeric) The AXIS(2) value for the start of the subset.
YEND (Numeric) The AXIS(2) value for the end of the subset.
XSTART (Numeric) The AXIS(1) value for the start of the subset.
XEND (Numeric) The AXIS(1) value for the end of the subset.
OUTPUT The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.
**D.161 ISUPER- Produces a superset of an image**

**Description:** Given an input image, ISUPER creates a similar image whose dimensions are (possibly) different and where there is an overlap between the pixels of the two images. The overlap pixels are copied directly into the output image from the input, and all other pixels of the output image are zeroed. The overlap is specified in terms of the position of the left bottom (1,1) pixel of the input image on the output image. X and Y data arrays will also be supersetted if they contain linear data - otherwise no X and Y sub-structures will be created in the output structure.

**Parameters:**

- **IMAGE** The name of the input image.
- **XSIZE** The number of pixels in the X-direction for the output image. This can be more than, less than, or the same as, the number of X-pixels in the input image.
- **YSIZE** The number of pixels in the Y-direction for the output image. This can be more than, less than, or the same as, the number of Y-pixels in the input image.
- **XPIXEL** The overlap between the input and output images is specified in terms of the position of the bottom left (1,1) pixel of the input image in the output image. If this pixel does not map onto a pixel of the output image, the position should be given as a negative number, or as 0. (0 is the position of an imaginary pixel just outside the image.)
- **YPIXEL** The overlap between the input and output images is specified in terms of the position of the bottom left (1,1) pixel of the input image in the output image. If this pixel does not map onto a pixel of the output image, the position should be given as a negative number, or as 0. (0 is the position of an imaginary pixel just outside the image.)
- **OUTPUT** The name of the resulting superset image.

**Source comments:**

Supersets an image or a spectrum, creating a new image with larger dimensions than the input, and with the input data just a part of the output image. If the AXIS(1) or AXIS(2) arrays contain linear data, then output AXIS(1) and AXIS(2) sub-structures will be created reflecting this data. If they contain non-linear data, ISUPER will not attempt to extrapolate the data values and will omit the AXIS(1) or AXIS(2) sub-structure in question from the output structure.

Note: It is also capable of subsetting.

**Command Parameters -**

- **IMAGE** (Character) The name of the input image.
- **XSIZE** (Numeric) The AXIS(1)-dimension of the output image.
- **YSIZE** (Numeric) The AXIS(2)-dimension of the output image.
- **XPIXEL** (Numeric) The pixel number in AXIS(1) at which the input image is to start.
- **YPIXEL** (Numeric) The pixel number in AXIS(2) at which the input image is to start. That is, pixel (1,1) of the input image maps onto pixel (XPIXEL,YPIXEL) of the output. These values may be such that not all the input image appears in the output - they may even be negative.
- **OUTPUT** (Character) The name of the output image.
D.162  ISXADD-Adds a spectrum to each X direction X-section of an image

Description: A value is added to each pixel of each column in the image. The values to be added come from the corresponding pixels in an input spectrum.

Parameters: IMAGE Name of image.
            SPECTRUM Name of spectrum.
            OUTPUT Name of resulting image.

Source comments:

Performs a number of operations all of the general form

\[ \text{Image} = \text{function(image, spectrum)} \]

Specifically, if an image is of format \((IX, IY)\) then

\[
\begin{align*}
\text{ISXSUB} & \text{ is Result } = \text{IMAGE}(I,N) - \text{SPECTRUM}(I) \quad 1\leq I \leq IX, 1\leq N \leq IY \\
\text{ISXADD} & \text{ " } = \text{IMAGE}(I,N) + \text{SPECTRUM}(I) \\
\text{ISXDIV} & \text{ " } = \text{IMAGE}(I,N) / \text{SPECTRUM}(I) \\
\text{ISXMUL} & \text{ " } = \text{IMAGE}(I,N) \times \text{SPECTRUM}(I) \\
\text{ISYSUB} & \text{ is Result } = \text{IMAGE}(N,I) - \text{SPECTRUM}(I) \quad 1\leq N \leq IX, 1\leq I \leq IY \\
\text{ISYADD} & \text{ " } = \text{IMAGE}(N,I) + \text{SPECTRUM}(I) \\
\text{ISYDIV} & \text{ " } = \text{IMAGE}(N,I) / \text{SPECTRUM}(I) \\
\text{ISYMUL} & \text{ " } = \text{IMAGE}(N,I) \times \text{SPECTRUM}(I)
\end{align*}
\]

So the ISX... operations require a spectrum IX elements long, and the ISY... operations need one IY elements long.

Command parameters -

IMAGE The name of the structure containing the first image.

SPECTRUM The name of the structure containing the second image data.

OUTPUT The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

The command itself (IXSADD, IXSMUL, etc.) is used to differentiate between the possible operations.
D.163 ISXDIV-Divides a spectrum into each X direction x-section of an image

Description: Each pixel of each column in the image is divided by a value. The divisor values come from the corresponding pixels in an input spectrum.

Parameters: IMAGE Name of image.
            SPECTRUM Name of spectrum.
            OUTPUT Name of resulting image.

Source comments

Performs a number of operations all of the general form

\text{Image} = \text{function(image, spectrum)}

Specifically, if an image is of format \((IX, IY)\) then

\begin{align*}
\text{ISXSUB} \text{ is Result} &= \text{IMAGE}(I,N) - \text{SPECTRUM}(I) \quad 1 \leq I \leq IX \quad 1 \leq N \leq IY \\
\text{ISXADD} &= \text{IMAGE}(I,N) + \text{SPECTRUM}(I) \\
\text{ISXDIV} &= \text{IMAGE}(I,N) / \text{SPECTRUM}(I) \\
\text{ISXMUL} &= \text{IMAGE}(I,N) \times \text{SPECTRUM}(I)
\end{align*}

\begin{align*}
\text{ISYSUB} \text{ is Result} &= \text{IMAGE}(N,I) - \text{SPECTRUM}(I) \quad 1 \leq N \leq IX \quad 1 \leq I \leq IY \\
\text{ISYADD} &= \text{IMAGE}(N,I) + \text{SPECTRUM}(I) \\
\text{ISYDIV} &= \text{IMAGE}(N,I) / \text{SPECTRUM}(I) \\
\text{ISYMUL} &= \text{IMAGE}(N,I) \times \text{SPECTRUM}(I)
\end{align*}

So the ISX... operations require a spectrum IX elements long, and the ISY... operations need one IY elements long.

Command parameters -

IMAGE The name of the structure containing the first image.

SPECTRUM The name of the structure containing the second image data.

OUTPUT The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

The command itself (IXSADD, IXSMUL, etc.) is used to differentiate between the possible operations.

KS / CIT 18th Feb 1983

D.164 ISXMUL-Multiplies each X direction image x-sect by a spectrum

Description: Each pixel of each column in the image is multiplied by a value. The multiplier values come from the corresponding pixels in an input spectrum.

Parameters: IMAGE Name of image.
            SPECTRUM Name of spectrum.
            OUTPUT Name of resulting image.
Perform a number of operations all of the general form

\[ \text{Image} = \text{function(image, spectrum)} \]

Specifically, if an image is of format (IX, IY) then

- **ISXSUB**: Result = \( \text{IMAGE}(I,N) - \text{SPECTRUM}(I) \) \( 1 \leq I \leq IX, 1 \leq N \leq IY \)
- **ISXADD**: Result = \( \text{IMAGE}(I,N) + \text{SPECTRUM}(I) \)
- **ISXDIV**: Result = \( \frac{\text{IMAGE}(I,N)}{\text{SPECTRUM}(I)} \)
- **ISXMUL**: Result = \( \text{IMAGE}(I,N) \times \text{SPECTRUM}(I) \)

- **ISYSUB**: Result = \( \text{IMAGE}(N,I) - \text{SPECTRUM}(I) \) \( 1 \leq N \leq IX, 1 \leq I \leq IY \)
- **ISYADD**: Result = \( \text{IMAGE}(N,I) + \text{SPECTRUM}(I) \)
- **ISYDIV**: Result = \( \frac{\text{IMAGE}(N,I)}{\text{SPECTRUM}(I)} \)
- **ISYMUL**: Result = \( \text{IMAGE}(N,I) \times \text{SPECTRUM}(I) \)

So the ISX... operations require a spectrum IX elements long, and the ISY... operations need one IY elements long.

**Command parameters** -

- **IMAGE**: The name of the structure containing the first image.
- **SPECTRUM**: The name of the structure containing the second image data.
- **OUTPUT**: The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

The command itself (IXSADD, IXSMUL, etc.) is used to differentiate between the possible operations.

KS / CIT 18th Feb 1983

**D.165 ISXSUB** — Subtracts each X direction image x-sect from a spectrum

**Description**: A value is subtracted from each pixel of each column in the image. The values to be subtracted come from the corresponding pixels in an input spectrum.

**Parameters**:

- **IMAGE**: Name of image.
- **SPECTRUM**: Name of spectrum.
- **OUTPUT**: Name of resulting image.

**Source comments**

Performs a number of operations all of the general form

\[ \text{Image} = \text{function(image, spectrum)} \]

Specifically, if an image is of format (IX, IY) then
ISXSUB is Result = IMAGE(I,N)-SPECTRUM(I) 1<=I<=IX 1<=N<=IY
ISXADD " " = IMAGE(I,N)+SPECTRUM(I) " "
ISXDIV " " = IMAGE(I,N)/SPECTRUM(I) " "
ISXMUL " " = IMAGE(I,N)*SPECTRUM(I) " "
ISYSUB is Result = IMAGE(N,I)-SPECTRUM(I) 1<=N<=IX 1<=I<=IY
ISYADD " " = IMAGE(N,I)+SPECTRUM(I) " "
ISYDIV " " = IMAGE(N,I)/SPECTRUM(I) " "
ISYMUL " " = IMAGE(N,I)*SPECTRUM(I) " "

So the ISX... operations require a spectrum IX elements long, and the ISY... operations need one IY elements long.

Command parameters -

IMAGE The name of the structure containing the first image.

SPECTRUM The name of the structure containing the second image data.

OUTPUT The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

The command itself (IXSADD,IXSMUL, etc.) is used to differentiate between the possible operations.

KS / CIT 18th Feb 1983

D.166 ISYADD-Adds a spectrum to each Y direction x-section of an image

Description: A value is added to each pixel of each row in the image. The values to be added come from the corresponding pixels in an input spectrum.

Parameters: IMAGE Name of image.
SPECTRUM Name of spectrum.
OUTPUT Name of resulting image.

Source comments -

Performs a number of operations all of the general form

Image=function(image,spectrum)

Specifically, if an image is of format (IX,IY) then

ISXSUB is Result = IMAGE(I,N)-SPECTRUM(I) 1<=I<=IX 1<=N<=IY
ISXADD " " = IMAGE(I,N)+SPECTRUM(I) " "
ISXDIV " " = IMAGE(I,N)/SPECTRUM(I) " "
ISXMUL " " = IMAGE(I,N)*SPECTRUM(I) " "
ISYSUB is Result = IMAGE(N,I)-SPECTRUM(I) 1<=N<=IX 1<=I<=IY
ISYADD " " = IMAGE(N,I)+SPECTRUM(I) " "
ISYDIV " " = IMAGE(N,I)/SPECTRUM(I) " "
ISYMUL " " = IMAGE(N,I)*SPECTRUM(I) " "
So the ISX... operations require a spectrum IX elements long, and the ISY... operations need one IY elements long.

Command parameters -

**IMAGE** The name of the structure containing the first image.

**SPECTRUM** The name of the structure containing the second image data.

**OUTPUT** The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

The command itself (IXSADD, IXSMUL, etc.) is used to differentiate between the possible operations.

KS / CIT 18th Feb 1983

D.167  **ISYDIV**-Divides a spectrum into each Y direction x-section of an image

**Description:** Each pixel of each row in the image is divided by a value. The divisor values come from the corresponding pixels in an input spectrum.

**Parameters:**

- **IMAGE** Name of image.
- **SPECTRUM** Name of spectrum.
- **OUTPUT** Name of resulting image.

**Source comments:** P S

Performs a number of operations all of the general form

Image=function(image, spectrum)

Specifically, if an image is of format (IX,IY) then

- **ISXSUB** is Result = IMAGE(I,N)-SPECTRUM(I) 1<=I<=IX 1<=N<=IY
- **ISXADD** " " = IMAGE(I,N)+SPECTRUM(I) " "
- **ISXDIV** " " = IMAGE(I,N)/SPECTRUM(I) " "
- **ISXMUL** " " = IMAGE(I,N)*SPECTRUM(I) " "
- **ISYSUB** is Result = IMAGE(N,I)-SPECTRUM(I) 1<=N<=IX 1<=I<=IY
- **ISYADD** " " = IMAGE(N,I)+SPECTRUM(I) " "
- **ISYDIV** " " = IMAGE(N,I)/SPECTRUM(I) " "
- **ISYMUL** " " = IMAGE(N,I)*SPECTRUM(I) " "

So the ISX... operations require a spectrum IX elements long, and the ISY... operations need one IY elements long.

Command parameters -

**IMAGE** The name of the structure containing the first image.
SPECTRUM The name of the structure containing the second image data.

OUTPUT The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

The command itself (IXSADD,IXSMUL, etc.) is used to differentiate between the possible operations.

KS / CIT 18th Feb 1983

D.168  ISYMUL-Multiplies each Y direction image x-sect by a spectrum

Description: Each pixel of each row in the image is multiplied by a value. The multiplier values come from the corresponding pixels in an input spectrum.

Parameters: IMAGE Name of image.
SPECTRUM Name of spectrum.
OUTPUT Name of resulting image.

Source comments: P S

Performs a number of operations all of the general form

Image=function(image,spectrum)

Specifically, if an image is of format (IX,IY) then

ISXSUB is Result = IMAGE(I,N)-SPECTRUM(I) 1<=I<=IX 1<=N<=IY
ISXADD " " = IMAGE(I,N)+SPECTRUM(I) " "
ISXDIV " " = IMAGE(I,N)/SPECTRUM(I) " "
ISXMUL " " = IMAGE(I,N)*SPECTRUM(I) " "
ISYSUB is Result = IMAGE(N,I)-SPECTRUM(I) 1<=N<=IX 1<=I<=IY
ISYADD " " = IMAGE(N,I)+SPECTRUM(I) " "
ISYDIV " " = IMAGE(N,I)/SPECTRUM(I) " "
ISYMUL " " = IMAGE(N,I)*SPECTRUM(I) " "

So the ISX... operations require a spectrum IX elements long, and the ISY... operations need one IY elements long.

Command parameters -

IMAGE The name of the structure containing the first image.

SPECTRUM The name of the structure containing the second image data.

OUTPUT The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.
The command itself (IXSADD, IXSMUL, etc.) is used to differentiate between the possible operations.

KS / CIT 18th Feb 1983

D.169 ISYSUB-Subtracts each Y direction image x-sect from a spectrum

Description: A value is subtracted from each pixel of each row in the image. The values to be subtracted come from the corresponding pixels in an input spectrum.

Parameters: IMAGE Name of image.
SPECTRUM Name of spectrum.
OUTPUT Name of resulting image.

Source comments

Perform a number of operations all of the general form

Image=function(image,spectrum)

Specifically, if an image is of format (IX,IY) then

ISXSUB is Result = IMAGE(I,N)-SPECTRUM(I) 1<=I<=IX 1<=N<=IY
ISXADD " " = IMAGE(I,N)+SPECTRUM(I) " "
ISXDIV " " = IMAGE(I,N)/SPECTRUM(I) " "
ISXMUL " " = IMAGE(I,N)*SPECTRUM(I) " "
ISYSUB is Result = IMAGE(N,I)-SPECTRUM(I) 1<=N<=IX 1<=I<=IY
ISYADD " " = IMAGE(N,I)+SPECTRUM(I) " "
ISYDIV " " = IMAGE(N,I)/SPECTRUM(I) " "
ISYMUL " " = IMAGE(N,I)*SPECTRUM(I) " "

So the ISX... operations require a spectrum IX elements long, and the ISY... operations need one IY elements long.

Command parameters -

IMAGE The name of the structure containing the first image.
SPECTRUM The name of the structure containing the second image data.
OUTPUT The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

The command itself (IXSADD, IXSMUL, etc.) is used to differentiate between the possible operations.

KS / CIT 18th Feb 1983
D.170  IXSMOOTH-Smooth in X-direction by gaussian convolution

Description:  IXSMOOTH smooths cross-sections of an array in the X-direction by convolving them with a gaussian of specified sigma and a specified cutoff width. Note that this is not a 2-D smooth, but a series of 1-D smooths; hence the name. The VARIANCE array, if present, is propagated in exactly the same way as the DATA array. This procedure it not formally correct and the computed variance will probably under-estimate the true error.

Parameters:  
- **IMAGE**  Name of data to be smoothed.
- **SIGMA**  Gaussian half width in pixels.
- **WIDTH**  Pixels over which gaussian is calculated.
- **OUTPUT**  Name of resulting data.

See also:  FIGARO: [CONV3][ISMOOTH][MEDFILT][KAPPA][CONVOLVE][FFCLEAN][GAUSMOOTH][MEDIAN]

Source comments:  This routine is the main body of IXSMOOTH, which smooths cross-sections of an array in the X-direction by convolving them with a gaussian of specified sigma and a specified cutoff width. Note that this is not a 2-D smooth, but a series of 1-D smooths; hence the name.

The VARIANCE array, if present, is propagated in exactly the same way as the DATA array. This procedure it not formally correct and the computed variance will probably under-estimate the true error.

Command parameters -

- **IMAGE**  The name of the structure containing the image.
- **SIGMA**  (Numeric) The sigma of the gaussian, expressed in array elements. (This is approximately the half width at half maximum, to within a factor of ~1.17).
- **WIDTH**  (Numeric) The number of array elements over which the gaussian is calculated - i.e. outside this range the gaussian is assumed to be zero. This simplifies the computation and also allows the use of strangely shaped functions such as gaussians with very large sigmas but small widths - which are almost the same as rectangular filters.
- **OUTPUT**  The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

KS / CIT 23rd March 1983
D.171  LINTERP-Interpolates between spiketrum points to form a spectrum

Description: LINTERP interpolates between the points of a 'spiketrum' - an array where most of the data is zero except for a few elements whose values are set - to produce a spectrum. Interpolation is linear between the points of the spiketrum.

Parameters: SPIKETRUM The name of the spiketrum - an array where only a few elements are set to the correct values, the other elements all being zero - which is to be used to produce a spectrum by interpolation between those specified elements. SPIKETRUM has probably been produced by GSPIKE from a table of values.

SPECTRUM The name of the spectrum to be produced by interpolation between the points of the spiketrum.

Source comments: INTERP / SPIFIT / LINTERP

Interpolates between the points of a 'spiketrum' to generate a spectrum. The INTERP command does this by spline interpolation, the SPIFIT command uses global polynomial fitting, and the LINTERP command uses linear interpolation.

Command parameters -

SPIKETRUM The name of the structure containing the spiketrum

ORDER The order for the global polynomial to be fitted (SPIFIT only).

SPECTRUM The name of the result of the operation. This can be the same as for SPIKE. If not, a new structure is created, with everything but the data a direct copy of the input.

Command keywords -

LOG (INTERP & SPIFIT only) Perform the interpolation on the log of the data

LINEND (INTERP only) Use a linear interpolation for the ends of the data - spline fits can go wild outside the range of the defined points.

User variables used - None

KS / CIT 6th July 1984

D.172  LONGSLIT-Fits 2D longslit arrays and plots results

Description: LONGSLIT can fit Gaussians, skew Gaussians, Lorentzians and Cauchy functions to line profiles, and can carry out line profile analysis to produce the Whittle and Heckman asymmetry parameters. Much of this can be carried out in batch mode. This program generates rotation curves from 2D data, for a number of emission/absorption lines. Options are available to automatically fit each line in the spectrum or to do it in an interactive manner allowing unsatisfactory fits to be
rejected. Unsatisfactory fits may then be refitted on a second pass. For each line a
table is created containing all the fit parameters as a function of crossection. If the
data has been block averaged in an range of XSECTS the average XSECT value and
the XSECT range are computed and stored. In manual mode the user can sweep
through the XSECTS choosing different BLOCKINGS and line types repeatedly until
he either accepts the fit or issues an instruction not to fit a given range of XSECTS.
Continuum Fitting is provided in two ways. Firstly a GLOBAL continuum may be
defined for each block of data. This is a continuum which is fitted to all regions of the
spectrum which do NOT contain lines. A variety of models are available. Chebyshev
Polynomials, Splines, (Power Law, Black Body, Black Body with optical depth cuttoff,
Empirical, and Balmer Continuum). Secondly, A local continuum may be applied
to each INDIVIDUAL line. This continuum is applied in addition to that created by
GLOBAL but is confined to the regions of the spectra containing a particular line.
Because most of the allowed line options also fit a FLAT base the combination of
these fits can be made to match even the most complex continua The types of allowed
continuum models for INDIVIDUAL is restricted to Spline, Flat or Chebyshev. It is
possible to Edit the results cube created by LONGSLIT and to create new Synthetic
spectra by doing model -data manipulations or from the models themselves. During
this process several usefull things can be done to the output spectra, notably changing
their Redshift, applying or removing Reddening. A similar approach creating Sky
subtracted data and using sky lines for correcting for instrumental vignetting is also
possible. Rotation curves in individual lines are produced as requested in the PLOT
mode. In addition it is possible to calculate a mean rotation curve. In the current
release the user now has complete control over the plotting parameters and the lines
being plotted.

Parameters: IMAGE  IMAGE = FILE (Read) Name of image for input This is the data.
This should be a .dst file with a .Z.DATA component. This should also have a
.X.DATA array which contains the wavelengths of the lines. For the identification
files supplied with the program the units should be Angstroms, but if the user
supplies his/her own files, this need not apply, although some plots may have
the wrong labels.
ARC_OPTS  ARC_OPTS = CHARACTER (Read) Enter fit option NEW : set up a new
analysis REPEAT : iterate on previous analysis CLONE : Clone an analysis from
another file (line locations etc.)
YSTART  YSTART = INTEGER (Read) analysis lower limit
YEND  YEND = INTEGER (Read) analysis upper limit
YBLOCK  YBLOCK = INTEGER (Read) Enter analysis x-sect width
ITERATION  ITERATION = INTEGER*2 (Read) New value of iteration
MAXLINES  MAXLINES = INTEGER (Read) Maximum number of lines to allow
room for
CLFILE  CLFILE = FILE (Read) Name of image for cloning from
OUTABLE  OUTABLE = FILE (Write) Name for exactic file
VCORR  VCORR = REAL (Read) correction to apply to radial velocities
TOLS  TOLS = CHARACTER (Read) For use in batch only
INHERIT  INHERIT = INTEGER (Read) Number to control inheritance of previous
fits If zero no inheritance of fits If one then inherited from next block If minus
one then inherited from previous block
DEVICE = CHARACTER (Read) Device to use for greyscale plots
FITRAT = REAL (Read) Ratio for double fitting (of widths/heights or separation)
CALRAT = INTEGER (Read) Ratio to multiply default number of iterations in NAG optimisation
WHITE = REAL (Read) White level for greyscale plots
BLACK = REAL (Read) Black level for greyscale plots
MAXGAUSS = INTEGER (Read) Maximum number of Gaussians to allow room for LONGSLIT can fit up to 9 component fits, but the results array for such is quite large. This allows the user to set the maximum number of components he/she is likely to fit, since very little data requires 9 components!
TSTART = REAL (Read) analysis lower limit
TEND = REAL (Read) analysis upper limit
TBLOCK = REAL (Read) Analysis blocking width in T direction
FIT_MODEL = CHARACTER (Read) Model of fit to perform
PLOTLIM = REAL ARRAY (Read) Limits for velocity plots
OUTPUT = FILE (Write) Name of output file
HARDCOPY = LOGICAL (Read) produce hardcopy plots of fits from cube
TABLE = LOGICAL (Read) produce table of fits from cube
PLOT = LOGICAL (Read) produce plots of rotation curves
PRINT = LOGICAL (Read) produce print out of rotation curves
SHAPE = LOGICAL (Read) carry out shape analysis
KEEP_ITT = LOGICAL (Read) keep iteration files
FIT = LOGICAL (Read) perform fitting
COPY = LOGICAL (Read) copy previous fits This will repeat all the fits previously made, which is likely to be of use if data is co-added after one file has been analysed. Also, when used with CLONE the entire .RES structure is copied without any change. For the new fits the previous fits (suitably scaled) are used as first guesses.
ABSORPTION = LOGICAL (Read) Allow fitting of absorption lines
BOUNDS = LOGICAL (Read) Perform bounded fits to lines (in batch)
LABEL = LOGICAL (Read) Put labels on plots
CONTOUR = LOGICAL (Read) Create contour plots
GREY = LOGICAL (Read) Create greyscale plots
LOG = LOGICAL (Read) Use logarithmic scale for CONTOUR and GREY
WEIGHTS = LOGICAL (Read) Use weights for fitting
PRFITS = LOGICAL (Read) Print out details of fitting
FULL = LOGICAL (Read) Print out full details of fits in table
CHECK = LOGICAL (Read) Plot array of line profiles

Source comments: available
D.173 LSPL0T—Hardcopy spectrum plot of specified size (up to 3 metres)

Description: LSPL0T produces a splot of a spectrum, usually on a hard copy device, with a physical size that can be specified (in metres) by the user. The device used is that specified by the HARD command. It will allow plots up to the maximum size allowed by the GKS driver being used - in some cases this means that a non-standard device name must be specified in order to allow a larger maximum size than usual. LSPL0T is very similar to SPLOT, except that it has plot dimension parameters and does not support build plots.

Parameters: SPECTRUM The name of the spectrum to be plotted by LSPL0T. It should be a 1-dimensional array.

XSIZE The length of the plot in metres. LSPL0T can produce plots up to 10 metres in length.

YSIZE The height of the plot in metres. The reset value is the full page height for the device.

WHOLE If set, the whole of the spectrum is plotted. Otherwise, the limits plotted are determined by the values of XSTART and XEND, which you will be prompted for if they were not specified in the command string.

AUTOSCALE If set, the plot is scaled so that all of the data to be plotted just fits on the display. Otherwise, the scale of the plot is determined by the values of HIGH and LOW, which you will be prompted for if they were not specified in the command string.

XSTART Specifies the first X value to be plotted, in the units used by the data - angstroms, for example, if the data is wavelength calibrated. XSTART can be set before the start of the data, if necessary. RESET will set XSTART to the first X value in the data.

XEND Specifies the last X value to be plotted, in the units used by the data - angstroms, for example, if the data is wavelength calibrated. XEND can be set after the end of the data, if necessary. RESET will set XEND to the last X value in the data.

HIGH The maximum data value to be plotted - i.e. the top Y axis value for the plot.

LOW The minimum data value to be plotted - i.e. the bottom Y axis value for the plot.

BIAS A bias value applied to the data, usually to bias up a plot which is to be superimposed upon a previous plot of almost identical data values. This makes the comparison easier. BIAS N is essentially equivalent to setting HIGH and LOW down by an amount N, so can result in unexpected axis values.

LABEL The label that will appear at the top of the plot.

LINES Specifies that the plot is to made as a 'point-plot', in a 'join the dots' fashion. The alternative is a histogram plot. LINES will usually be a little faster, but messier, because the resolution of the plotting device may result in stepped lines.

Source comments: LSPL0T / ESPL0T

These are versions of SPLOT and ESPL0T that allow the size of the plot to be specified. LSPL0T produces a plot of a single spectrum, while ESPL0T produces an error bar plot of a spectrum which has error information.
Command parameters -

XSIZE     The size of the plot in X, in metres.
YSIZE     The size of the plot in Y, in metres.
SPECTRUM  The data to be plotted. If there
          is an x-axis data component this will be used to
give the x-axis. If not, the x-axis will just
          have to be the numbers from 1 to n.
XSTART    The x-value at which plotting is to start.
XEND      The x-value at which plotting is to end.
          (XSTART and XEND are not required if the
          WHOLE keyword is set.)
HIGH      The maximum value to be used for the plot.
LOW       The minimum value to be used for the plot.
BIAS      A value used to displace the plot - BIAS is
effectively a value added to the data before
          it is plotted. (It is implemented as a value
          subtracted from both HIGH and LOW.)
          (HIGH, LOW and BIAS are not required if the
          AUTOSCALE keyword is set.)
LABEL     A label for the plot.

Command keywords -

AUTOSCALE The program is to work out the values for HIGH
          and LOW, using the maximum and minimum values
          in the data over the specified range.
WHOLE     The program is to display all of the spectrum.
LINES     The plot is not done as a histogram, but as
          a 'join the dots' line plot. (LSLLOT only)

User variables used:

HARD      (Character) The device used for HARD plots.

Note:

The original version of LSLOT used GKS 6.2 and the DIAGRAM
package. This has now been discontinued, and some of the
functionality of DIAGRAM (the ability to specify the size of
the plot in physical units) has appeared in PGPLOT. This new
version uses PGPLOT. It can produce a plot of the specified
size, but only if that size is SMALLER than the default size
for the device. In practice, this means that it can only work
in the way it was intended with 'unusual' devices that have
particularly large default plot sizes (which often need to be set
up specially for the purpose).

KS / AAO 30th Jan 1984

D.174  LXSET-Set X array of spectrum/image to specified range

Description:  LXSET is a fudging routine used to set the X values for an image or spectrum,
given a start and end value. The values can be set to vary linearly or logarithmically.
**Parameters:**  

- **IMAGE** The name of the image (or spectrum) whose X values are to be set. If IMAGE already has an X array, this will be modified. If it does not, a new one will be created.

- **WSTART** Specifies the value (usually a wavelength) to be used for the first element of the X array. Note that the value refers to the CENTER of the bin - it’s easy to find yourself one bin out.

- **WEND** Specifies the value (usually a wavelength) to be used for the last element of the X array. Note that the value refers to the CENTER of the bin - it’s easy to find yourself one bin out.

- **OUTPUT** The name of the resulting image (or spectrum). If this is the same as IMAGE, the original file will be modified. Otherwise, a new file will be created.

- **LOG** If set, the X values will vary in a logarithmic manner between WSTART and WEND. The default is for them to vary linearly.

**See also:**  
**FIGARO:** LYSET  
**KAPPA:** SETAXIS, AXCONV, AXLABEL, AXUNITS

**Source comments:**

This routine is the main body of LXSET,LYSET

LXSET is a fudging routine that sets the range of the first axis associated with a spectrum or image, given a start and end wavelength and whether or not the wavelengths are to be scaled logarithmically.

LYSET performs an analogous set for the second axis structure (for images).

**Command parameters** -

- **IMAGE** (Character) The name of the structure containing the image.

- **WSTART** or (Numeric) The value of the CENTER of the first bin of the resulting data. WSTART and VSTART are used by LXSET and LYSET respectively.

- **VSTART** (Numeric) The value of the CENTER of the first bin of the resulting data. WSTART and VSTART are used by LXSET and LYSET respectively.

- **WEND** or (Numeric) The value of the CENTER of the final bin of the resulting data. WEND and VEND are used by LXSET and LYSET respectively.

- **VEND** (Numeric) The value of the CENTER of the final bin of the resulting data. WEND and VEND are used by LXSET and LYSET respectively.

- **OUTPUT** (Character) The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the axis data a direct copy of the input.

**Command keywords** -

- **LOG** Axis values are to increase logarithmically.

KS / AAO 28th March 1985
D.175  **LYSET-Set Y array of spectrum/image to specified range**

**Description:** LYSET is a fudging routine used to set the Y values for an image given a start and end value. The values can be set to vary linearly or logarithmically.

**Parameters:** IMAGE The name of the image whose Y values are to be set. If IMAGE already has an Y array, this will be modified. If it does not, a new one will be created.

VSTART Specifies the value to be used for the first element of the Y array. Note that the value refers to the CENTER of the bin - it’s easy to find yourself one bin out.

VEND Specifies the value to be used for the last element of the Y array. Note that the value refers to the CENTER of the bin - it’s easy to find yourself one bin out.

OUTPUT The name of the resulting image (or spectrum). If this is the same as IMAGE, the original file will be modified. Otherwise, a new file will be created.

LOG If set, the Y values will vary in a logarithmic manner between VSTART and VEND. The default is for them to vary linearly.

See also: FIGARO: LXSET, KAPPA: SETAXIS, AXCONV, AXLABEL, AXUNITS

**Source comments:**

This routine is the main body of LXSET,LYSET

LXSET is a fudging routine that sets the range of the first axis associated with a spectrum or image, given a start and end wavelength and whether or not the wavelengths are to be scaled logarithmically.

LYSET performs an analogous set for the second axis structure (for images).

**Command parameters -**

**IMAGE** (Character) The name of the structure containing the image.

**WSTART or VSTART** (Numeric) The value of the CENTER of the first bin of the resulting data. WSTART and VSTART are used by LXSET and LYSET respectively.

**WEND or VEND** (Numeric) The value of the CENTER of the final bin of the resulting data. WEND and VEND are used by LXSET and LYSET respectively.

**OUTPUT** (Character) The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the axis data a direct copy of the input.

**Command keywords -**

**LOG** Axis values are to increase logarithmically.
D.176 MASK-Generate a mask spectrum given a spectrum and a mask table

Description: Given a spectrum and a table of masked regions, MASK creates a mask spectrum which is the same as the original spectrum in the masked regions, but zero everywhere else. The mask will have the same x-values as the original spectrum.

Parameters: SPECTRUM The name of the original spectrum.
TABLE The name of the file containing the regions to be masked. This is a text file. If no extension is specified, .MSK will be assumed. The table can be one supplied as part of Figaro, or can be one supplied by the user, either in the current default directory or in one with the logical name FIGARO_PROG_U. (There is usually no need to specify the directory, since the program will search for it.)
MASK The name of the mask spectrum generated.

Source comments: K

Generates a spectral mask given a spectrum and a mask table file. The mask table contains a set of central wavelengths and number of anstroms covered, one for each part of the spectrum to be masked. The program generates a spectrum covering the same wavelength range as the original spectrum, with the masked areas set to the same values as the original spectrum in those areas, and the unmasked areas set to zero.

Command parameters -

SPECTRUM (Character) The spectrum to be used
TABLE (Character) The mask table file to be used - if the file has no extension, .MSK will be assumed. The program searches for the mask file in the standard Figaro search path for such files.
MASK (Character) The output mask name.

User variables used - None

KS / CIT 4th April 1984

D.177 MASKEXT-Extracts echelle orders using a mask created by ECHMASK

Description: Extracts spectra from an image using a mask image. The mask pixel values should be the numbers of the spectra that fall in that pixel of the image. This routine is intended for use with echelle data, and the numbers in the image are expected to be based on echelle order numbers. The spectra are extracted into an image whose Y-axis is the order number of the spectra, and whose X-axis is that of the original data. The actual numbers in the image are expected to have the form (echelle order number)*10 + (sub order number), sub orders referring to separate spectra from the same order, as produced, for example, by the UCL echelle at AAO. MASKEXT will extract one specified sub-order from each order, or will add all the sub-orders together for each order.

Parameters: IMAGE Name of image containing spectra.
MASK Name of mask image.
**MLOW**  Lowest order number to extract.

**MHIGH**  Highest order number to extract.

**SUBORD**  Sub-order to extract (0 => all).

**REVERSE**  Create output with order numbers reversed?

**OUTPUT**  Name of resulting set of spectra

Source comments: **K E X T**

Extracts spectra from an image using a mask image. The mask pixel values should be the numbers of the spectra that fall in that pixel of the image. This routine is intended for use with echelle data, and the numbers in the image are expected to be based on echelle order numbers. The spectra are extracted into an image whose Y-axis is the order number of the spectra, and whose X-axis is that of the original data. The actual numbers in the image are expected to have the form (echelle order number)*10 + (sub order number), sub orders referring to separate spectra from the same order, as produced, for example, by the UCL echelle at AAO. **MAKSEXT** will extract one specified sub-order from each order, or will add all the sub-orders together for each order.

Command parameters -

**IMAGE**  (Character) The name of the image containing the spectra to be extracted

**MASK**  (Character) The name of the mask image.

**MLOW**  (Integer) The lowest order number to be extracted.

**MHIGH**  (Integer) The highest order number to be extracted.

**SUBORD**  (Integer) The sub-order number to be extracted. If zero, all sub-orders are added for each order.

**OUTPUT**  (Character) The name of the resulting image.

Command keywords -

**REVERSE**  If set, the image created will have the spectral orders in reverse order - higher order numbers being at the start of the image.


Note -

This version is an interim one - it really needs re-writing using **DSA_RESHAPE_DATA** and **DSA_RESHAPE_AXIS**.

**D.178  MCFIT**  Fit a continuum to a spectrum, given a mask spectrum

**Description:** Performs a masked fit to a spectrum. At a given number of equally separated points in the spectrum, the average of the surrounding points is taken and a new spectrum is formed by interpolating between the resulting values. Any points in spectrum for which the corresponding element of the mask spectrum is non-zero will be ignored.
Parameters: **SPECTRUM**  The spectrum to which the masked fit is to be applied. At a given number of equally separated points in SPECTRUM, the average of the surrounding data is taken and a new spectrum formed by interpolating between the resulting values.

**MASK**  The name of the spectrum to be used as a mask. Any points in SPECTRUM for which the corresponding element in MASK is non-zero will be ignored in the fit.

**POINTS**  The number of spline points to be used in the fit. The actual number used may be less, since the points are equispaced and some may be masked out.)

**OUTPUT**  The name of the resulting fitted spectrum. Note that the default is the same as SPECTRUM - i.e. the fitting will be performed in situ, which is faster but destroys the original data.

Source comments:  

Performs a masked fit to a spectrum. At a given number of equally separated points in the spectrum, the average of the surrounding points is taken and a new spectrum is formed by interpolating between the resulting values. Any points in the spectrum for which the corresponding element of the mask spectrum is non-zero will be ignored.

Command parameters -

- **SPECTRUM** (Character) The spectrum to be fitted
- **MASK** (Character) The mask spectrum to be used
- **POINTS** (Numeric) The number of points to be used
- **OUTPUT** (Character) The name of the resulting spectrum

Command keywords - None

User variables used - None

D.179  **MEDFILT**-Applies a median filter to an image

Description:  The result of this operation is an image in which the value of each pixel is the median value of a rectangular box of pixels centered on the corresponding pixel in the original array. MEDFILT only supports a square box. MEDFILTR allows rectangular boxes to be used.

Parameters: **IMAGE**  Name of image to be smoothed.

- **BOX**  Size of box for median calculations.
- **OUTPUT**  Name of resulting image.

See also:  **FIGARO:** MEDFILTR, BCLEAN, CLEAN, COSREJ, ISMOOTH, XSmoOTH, Kappa, FFclean, GAUSMooTH, MEDIAN

Source comments:  

Figaro routines to median filter an image. The result of this operation is an image in which the value of each pixel
is the median value of a rectangular box of pixels centered on the corresponding pixel in the original array. MEDFILTR allows the box to be specified with different X and Y dimensions, while the original MEDFILT only supported a square box. MEDFILTR is therefore a full superset of MEDFILT, but the older application has to be retained for compatibility reasons.

Command parameters for MEDFILT -

IMAGE The name of the structure containing the image.

BOX (Numeric) The size of the box (in pixels) to be used in calculating the medians. Should be odd; if even, BOX-1 will be used.

OUTPUT The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

KS / CIT 28th Jan 1984

D.180 MEDFILTR-Applies a rectangular median filter to an image

Description: MEDFILTR passes a rectangular median filter through the main data array of an image. The result is an image where each pixel is set to the median value found in the image within a box of the given size centered on that pixel. MEDFILT is a similar program, but it uses a square shaped filter box whereas MEDFILTR allows the box to have different dimensions in X and Y.

Parameters: IMAGE The name of a Figaro format data file. A rectangular median filter of size XBOX by YBOX will be passed through the main data array of the file. The file should normally contain two-dimensional data, but other data can be accommodated - this will require YBOX to be set to 1.

XBOX The X-dimension of the box to be used for the median filter. It is specified in pixels, and ought to be an odd number. If an even number is used, XBOX-1 will be used instead.

YBOX The Y-dimension of the box to be used for the median filter. It is specified in pixels, and ought to be an odd number. If an even number is used, YBOX-1 will be used instead. If the data is not a 2-D image, YBOX will be forced to 1.

OUTPUT The name of the resulting new image file. If it differs from that specified for IMAGE then a new file is created, otherwise the filter is applied in situ.

Source comments: FILT / MEDFILTR

Figaro routines to median filter an image. The result of this operation is an image in which the value of each pixel is the median value of a rectangular box of pixels centered on the corresponding pixel in the original array. MEDFILTR allows the box to be specified with different X and Y dimensions, while the original MEDFILT only supported a square box. MEDFILTR is therefore a full superset of MEDFILT, but the older application has to be
D.181  MEDSKY-Take the median of a number of images

Description:  MEDSKY takes a series of images and produces a single image in which each pixel is the median of the corresponding pixels in the series of images. Note that this is a time-consuming operation. The algorithm used attempts to optimise itself for the amount of memory available and will run much faster if the user has a large working set.

Parameters:  FILES  The name of a text file, each line of which should give the name of an input image. These are the images that will be used to form the output median image.

OUTPUT  The name of the file to be created, in which the value of each pixel is the median value of the corresponding pixels in the input images. Note that OUTPUT must not have the same name as any of the input images - the results will be incorrect if this is the case.

IMGLOG  If LOG is set, IMGLOG supplies the name of the 'log' file produced. This has nothing but a data array containing the numbers from 1 to N of the images that supplied the corresponding median values in the output median image. (Note that if more than one image contained a given median value, the number chosen will be the lowest of the possible values - i.e. the first image has precedence)

SCALED  If set, then MEDSKY calculates the overall median value for each image in turn, and before using the data from an image, multiplies it by a scaling factor given by the median value of the first image divided by the median value of the image being used. This means that all images are conceptually scaled to the median level of the first image before being used. If SCALED is set a file - MEDSKY.LOG - is written giving the file name, median value, and scaling factor used for each image.

LOG  LOG can be set to produce a diagnostic image called a 'log' image as well as the normal output median image. A 'log' image is an integer array of the same
size as the output image, in which each pixel contains the image number in the
order given in the file giving the list of images) of the image that supplied the
demand value used. If LOG is set, the name of the ‘log’ image is supplied by the
IMGLOG parameter.

See also: FIGARO: BCLEAN, CLEAN, COSREJ, MEDFILT
KAPPA: MEDIAN, MSTATS.

Source comments: S K Y

MEDSKY is a program for constructing sky flats for direct imaging,
using an algorithm due to Schneider, Schmidt and Gunn.

A list of images is read in from file, and the medians of
equivalent pixels in all the images are found, e.g. for N images
(of any size), the first pixel (the top-left one) in each image
is fetched: the median of these N pixels is then found, the
result becoming the first pixel in the output image. This process
is repeated for all the pixels in the input images to construct a
complete output image. In practice, more than one pixel is
analysed per pass through the image file list, the number being
determined by the memory available to the program. MEDSKY attempts
to minimise the page faulting which would occur by accessing too
much virtual memory at once by processing fewer pixels per pass,
on the assumption that the extra overheads in doing more passes
will be more than offset by fewer page faults. This approach seems
to have been justified, but performance is very sensitive to
changes in the proportions of memory allocated, and it is possible
that improvements could be made.

Command parameters -

FILES The name of a .DAT file containing a list of names
of images. All these images must have dimensions
equal to those of the first image in the list, FIRST.

OUTPUT The name of the result of the operation. This can
be the same as for FIRST. If not, a new structure
is created, with everything but the data a direct
copy of the input.

IMGLOG Only used if the LOG keyword is set. This supplies
the name of the ‘log’ image created. This will contain
a single integer data array.

Important ~~~~~~~~~
MEDSKY does not check that the output image name is not a member
of the image file list.

Command keywords -

SCALED If set, MEDSKY attempts to compensate for differences
in data scale between the different images. It does this
by conceptually scaling all images so that they have the same median value as the first image.

LOG If set, MEDSKY creates a 'log' image. This is an image with the same dimensions as the output image, with each pixel a number from 1 to N (the number of files) showing the image number which for each output pixel had the data value closest to the median value calculated. This can be used to see if any image dominates the others, or if there is any trend across the image. This is a rather specialised option, and is not expected to be used much.

User variables used - None

D.182 MOMENTS-Calculate moments of spectra in a cube.

Usage: moments in comp

Description: This routine calculates the moments for each spectrum in a cube.

Parameters: INFO INFO = _LOGICAL (Read) If false, this routine will issue only error messages and no informational message. [YES]
VARUSE VARUSE = _LOGICAL (Read) If false, input variances are ignored. [YES]
IN IN = NDF (Read) The input NDF. Update access is necessary to store the results in the Spectre Extension. The NDF can be a section, as in myndf(5:9,). The spectroscopic axis must be the first non-degenerate axis.
COMP COMP = _INTEGER (Read and Write) The component number to be used to store the results. This should be either an existing component of type ‘moments’ or zero. If it is zero, or if the component specified does not exist, or if it is not of type ‘moments’, then a new component will be created in the results structure. In any case, if INFO is true this routine will report which component number has actually been used. [0]
BIAS BIAS = _REAL (Read) y(x) is not itself used as the probability of x, but y(x)-bias. Thus for a spectrum that was normalised to the continuum level, give BIAS=1.0. A bias of zero is suitable for baseline-corrected spectra. [0]

Examples: moments in(-25.:+25.,,) 5
The NDF is probably three-dimensional. Analysis is restricted to pixels between the pixels nearest to x=-25 and +25, according to the AXIS(1) information. If there are five or more components in the results structure and if the fifth is of type 'moments' then it will be used to store the results. Otherwise a new component will be created for storage.

Source comments: M O M E N T S
For most of the calculated moments each spectrum y(x) is regarded as probability distribution of x-values. That is to say that y(x)-b is a proportional to the probability of the value x. The results for each spectrum are stored in the result structure of the Spectre Extension of the input NDF. The component stored is of type ‘moments’.
The numbers calculated are:

- minimum pos.: The smallest x found.
- maximum pos.: The largest x found.
- data minimum: The smallest data minus bias found.
- data maximum: The largest data minus bias found.
- sum of data: The sum of all values of data minus bias. This value is bad if any addend or its variance is bad.
- pos. of minimum: The x value where the minimum data value was found.
- pos. of maximum: The x value where the maximum data value was found.
- median: The median is currently not calculated. The stored value is the bad value.
- centroid: The mean x value. Contrary to the sum of data, this is calculated using only from data points where data and variance are not bad.
- variance: The variance of the x values. This is calculated in a second pass after the centroid is known. An approximate rounding error correction is made according to Press et al. 1992, p. 607.
- mean abs. dev.: The mean absolute deviation of the x values from the centroid. This is calculated in a second pass after the centroid is known.
- skewness: The skewness gives a measure of the asymmetry of the profile of data minus bias versus x. It is positive when the profile has a tail towards large x, negative when the profile has a wing at small x.
- kurtosis: The kurtosis gives a measure of the ‘‘peakedness’’ of the profile. It is zero for a Gaussian profile, positive if the profile peak is more pronounced, negative if the profile is flatter at the centre.
- momentum: If x is radial velocity and data minus bias is a measure of mass, then this is a measure of the radial momentum (inertia). This value is bad if any addend or its variance is bad.
- energy: If x is radial velocity and data minus bias is a measure of mass, then this is a measure of the kinetic energy. This value is bad if any addend or its variance is bad.

Note that the higher moments (variance, skewness, kurtosis) are rather unreliable unless the spectral features are very strong. For further discussion see Press et al. 1992, chapter 14.1.

Notes: This routine recognises the Specdre Extension v. 0.7.

This routine works in situ and modifies the input file.


D.183 MOVIE-Browse through slices of a cube.

Usage: movie in axis low=? high=?
Description: This routine takes a three-dimensional NDF and displays its two-dimensional slices sequentially on a grey or colour graphics device.

Parameters: DIALOG DIALOG = _CHAR (Read) The dialogue mode. If false ('F' or 'N') all frames of the cube will be displayed once in forward order. If true ('T' or 'Y') the routine will not display anything initially, but repeatedly ask for a menu option via the MENU parameter. 'G' for graphic is also permitted, but has the same meaning as 'T'. ['F']

INFO INFO = _LOGICAL (Read) If true, informational messages are given. Such as, which frame is currently displayed. [YES]

MODE MODE = _CHAR (Read) 'Fast', 'Fill', or 'Square' for (i) a tiny but quick display, (ii) to use the whole display area available, (iii) the biggest display with square pixels that is possible in the area available. The mode can be abbreviated to two characters and is case-insensitive. ['Fast']

IN IN = NDF (Read) The input NDF. It must be three-dimensional - not counting degenerate axes.

DEVICE DEVICE = GRAPHICS (Read) The graphics display device. It must be a screen device, not a printer device.

AXIS AXIS = _INTEGER (Read) The number of the movie axis. Of the three axes in the input cube this is the one not visible in the display. This is the axis to count the frames of the movie. [3]

LOW LOW = _REAL (Read) The minimum data value from the cube to be displayed. Values less than this are displayed in the same colour.

HIGH HIGH = _REAL (Read) The maximum data value from the cube to be displayed. Values greater than this are displayed in the same colour.

MENU MENU = _CHAR (Read) The application will ask repeatedly for the menu option, until 'Q' is chosen. These options are also available from the keyboard if dialogue is graphic. ['F'] - F: Display each frame in forward order. - B: Display each frame in backward order. - I: Ask for FRAME parameter and display specified frame. - P: Display previous frame. - N: Display next frame. - Q: Quit. - ?: Help.

FRAME FRAME = _INTEGER (Read) The frame number to be displayed next. Note that frames are counted in NDF pixel indices, i.e. from the NDF’s lower bound to its upper bound.

Source comments: The colour table of the display is unaltered so that a previously loaded colour table will be used. Bad values will be displayed in the display background colour, which in general is distinct from the colour for the lowest (or highest) data value.

This routine is quite primitive. It does not use axis data or spectroscopic values from the Specdre Extension. Pixels and slices in the cube are addressed by their NDF pixel indices, which are integer numbers, usually starting at 1.

The routine also does not pay much attention to the precise timing of the display. The following list gives activities that the routine spends time on and how the user can exert some control over the timing.
- Before a frame can be displayed it must be extracted from the cube. The time taken for this depends greatly on whether the frame counting axis is the first or last axis. Taking slices is fastest if AXIS=3 and can be very slow if AXIS=1, so it may be useful to re-arrange the axes of a cube that will be viewed often with the same frame-counting axis. Another way to reduce the time for taking slices from the cube is to use as small a cube as possible: If it is a-priori known that only a certain range of frames will be looked at, or that only a certain part of all frames is interesting, then the input cube can be given as an appropriate subset of the actual disk file.

- Also before a frame can be displayed it must be converted according to the colour capabilities of the display.

- Each frame needs to be extracted and converted only once and can be viewed several times, converted frames are kept in a workspace until the routine exits.

- In the sequence displays each frame is converted and displayed before the routine goes on to the next frame.

- When a specific frame is requested it is extracted, converted and displayed (unless it has been viewed before).

- When the next or previous frame relative to the displayed one is requested, it is extracted and converted if necessary. Then it is displayed. In anticipation of another request of the same type the next or previous frame is extracted and converted immediately.

- Even if a frame has been converted before, it takes some time to re-sample it from cube pixels to display pixels. This time can be minimised by choosing the fast mode, where a cube pixel is only one display pixel.

- Disruptions occur in the display of a sequence of frames due to the unpredictable need for the machine to page memory.

- Display may be over a network and bandwidth has to be shared with other users. This too causes disruptions of frame sequences.

In summary, it may be best to

- put on your spectacles and settle for the fast (and tiny) display,
- decide which part of the cube is interesting and specify only that sub-cube as input,
- begin the forward sequence to convert the whole input sub-cube,
- have a cup of tea if AXIS=3 and the cube is not small,
- use the options 'I', 'P', 'N' to look at individual frames in your own time.
It is not possible to write the cube as converted for display. Such a cube would be of limited use, since it might contain only part of the input cube and since its scaling depends on the colour capabilities of the display used.

D.184 MSPLOT-Plots a long spectrum as a series of separate plots

Description: The MSPLOT command will plot a spectrum on the current hard or soft graphics device, splitting it into a series of individual plots running down the plotting surface.

Parameters: SPECTRUM SPECTRUM is the name of the spectrum to be plotted by MSPLOT. It should be a 1-dimensional array.

HARDCOPY If set, the plot is written to the device defined as the current hardcopy device. Generally, this is a disk file which will then have to explicitly spooled to the actual plotting device. If HARDCOPY is not set, the plot will go to the current softcopy device. The hard and soft copy devices are specified using the HARD and SOFT commands respectively.

WHOLE If set, the whole of the spectrum is plotted. Otherwise, the limits plotted are determined by the values of XSTART and XEND, which you will be prompted for if they were not specified in the command string.

SAMESCALE MSPLOT always autoscales the data - you do not have the chance to specify the range explicitly. However, there is the option of using the same scale for every spectrum plotted, (which you get if you set SAMESCALE), or of having each individual plot autoscaled.

XSTART The first X value to be plotted, in the units used by the data - angstroms, for example, if the data is wavelength calibrated. XSTART can be set before the start of the data, if necessary. RESET will set XSTART to the first X value in the data.

XEND The last X value to be plotted, in the units used by the data - angstroms, for example, if the data is wavelength calibrated. XEND can be set after the end of the data, if necessary. RESET will set XEND to the last X value in the data.

NSPECT The maximum number of spectra to plot on one page. The sensible maximum depends on the resolution of your output device - a 300dpi laser printer in portrait mode can handle about 10 - after this it starts to get silly.

XRANGE The spectrum is broken up into individual sections, each covering the same axis range. The amount of axis range covered by a single spectrum is specified by XRANGE. So, for example, to split the plot into 5 sections, XRANGE should be (XEND-XSTART)/5. If XRANGE is such that more than NSPECT plots are needed to cover the spectrum, further sets of plots will be drawn. If the soft plotting device is used, you will be prompted at the end of each page of plots.

LABEL The label that will appear at the top of the plot.

COLOUR The colour for the data to be plotted in. The axes are always plotted in white. The colours allowed are Blue, White, Red, Green, Black, Cyan, Magenta, Yellow. Using Black will have the effect of erasing anything where the data is plotted. This only works on the Grinnell.

THICKNESS Only used for ‘build’ or ‘hard’ plots. It is used to increase the thickness of the lines plotted in order to increase legibility, particularly on the Versatec.
Generally 1 or 3 is reasonable for the Versatec - depending on how well set up it is at the present, and 1 should be used for other devices.

NEXT Used to pause before a new page.

Source comments: L O T

Produces a plot of a spectrum, splitting it up into a number of sections, each plotted separately in a series down the plotting surface. The plots are directed to the device defined by the user variables 'SOFT' and 'HARD', and by the value of the command keyword 'HARDCOPY', so will appear immediately if these specify a video device (VT125, Args, etc.). If a hardcopy device is specified, the file for that device will be produced, but MSPLOT does not attempt to spool it off for printing.

Command parameters -

SPECTRUM The data to be plotted. If this contains X-axis information, this will be used. If not, the X-axis will just have to be the numbers from 1 to n.

XSTART The x-value at which plotting is to start.

XEND The x-value at which plotting is to end. (XSTART and XEND are not required if the WHOLE keyword is set.)

LABEL A label for the plot.

COLOUR The colour for the plot, assuming the display device supports it. The axes are always white.

THICKNESS The width of the lines used for the plot. This is only used for 'hard' & 'build' plots, and should really be 1 for anything other than a high-resolution device like a Versatec or a laser printer.

Command keywords -

SAMESCALE The program is to use a the same scale for all the plots (i.e. the scale is to be global). Otherwise the plots will be autoscaled individually. The global scale values are determined from the data range of the whole of the spectrum to be plotted.

SHOWZERO If yes, the autoscale values are constrained to include zero.

WHOLE The program is to display all of the spectrum.

HARDCOPY The plot is to produce a hard copy.

NEXT Used to pause before a new page.

User variables - ("">" input, "<" output)

(>) SOFT Specifies the device and type to be used for soft plots. See the SOFT command for more details.

(>) HARD Specifies the device and type to be used for hard plots. See the HARD command for more details.

(Other user variables may be set by the command processor, in connection with the parameter values.)
D.185  NCSET-Set a region of a spectrum to a constant

Description: NCSET sets a specified region in a spectrum to a constant value. This is a non-interactive version of CSET, which allows the region to be specified precisely in terms of the X-values of the data.

Parameters:  
- SPECTRUM The name of the spectrum to be modified by NCSET. It should be a 1-dimensional array.
- XSTART The first X value to be set to the constant, in units used by the data - angstroms, for example, if the data is wavelength calibrated. RESET will set XSTART to the first X value in the data.
- XEND The last X value to be set to the constant, in units used by the data - angstroms, for example, if the data is wavelength calibrated. RESET will set XEND to the last X value in the data.
- VALUE The selected region of the spectrum (from XSTART to XEND) will be set to VALUE.
- OUTPUT NCSET generates an output file that is essentially the data from the displayed spectrum, with one regions set to the constant value. OUTPUT is the name of the resulting spectrum.

See also: FIGARO: CSET, ICSET, TIPPEX. KAPPA: CHPIX, FILLBAD, SEGMENT, NOMAGIC, RIFT, SETMAGIC, ZAPLIN.

Source comments: E T

Figaro function to set a selected region of a spectrum to a constant value. This is a non-interactive version of CSET, and has the possible advantage of allowing the region to be specified precisely in terms of the X values of the data.

Command parameters -

- SPECTRUM (Character) The spectrum to be modified.
- XSTART (Numeric) The X-value of the start of the region.
- XEND (Numeric) The X-value of the end of the region.
- VALUE (Numeric) The value to use for the selected region.
- OUTPUT (Character) The name of the output file to be created. If this is the same as the input spectrum, the data will be modified in situ.

Command keywords - None

User variables used - None

D.186  OFFDIST-Applies an offset to an SDIST fit

Description: OFFDIST reads the s-distortion fit file produced by SDIST and writes a modified version in which a specified offset in Y has been added to all the fits. This
allows the same distortion analysis to be applied to data which has the same shape but is physically displaced from the data originally analysed.

**Parameters:** 
**INFILE** The name of the distortion file to which an offset is to be added. This will have been produced by SDIST, and so will normally be called SDIST.DAT.

**OFFSET** The offset in Y to be applied to the data is specified by OFFSET. A positive value moves the fits up (in the sense that it will move the lines displayed by ODIST up the screen of an image display). OFFSET is a value in pixels.

**OUTFILE** The name of the distortion file to be created with the offset value applied. A new file is always created, even if the names supplied for OUTFILE and INFILE are the same (a new version is created). Most applications that use these files expect them to be called SDIST.DAT, so if you use anything else some creative RENAMEing may be needed.

**Source comments:**

ODIST

Modifies the output s-distortion file produced by SDIST to add an offset in Y to the fitted positions.

Command parameters -

**INFILE** (Character) The name of the distortion file to be modified.

**OFFSET** (Numeric) The offset in Y to be added to the fits.

**OUTFILE** (Character) The name of the resulting modified file.

Note that many of the applications that use these files assume explicitly that they are called SDIST.DAT. If OUTFILE is the same as INFILE, a new version of the file is produced.

Command keywords - None

Input and output files -

SDIST.DAT contains the results of the fit(s), as written by SDIST, in a format treated as follows -

3 header lines, all beginning with '*'

One line giving the number of spectra traced, in the format 20X,I5.

Then, for each spectrum traced, one record giving the spectrum number, and the leftmost and rightmost pixels covered by the trace, then 1 record including the average Y value in the spectrum, in format 16X,F13.7, which is followed by 4 records giving the 11 polynomial coefficients for the fit. Note that this program only assumes the number of records for each spectrum, and the position of the average Y value. It carefully only modifies the Y value fields, leaving all the rest unchanged.
D.187 OPTEXTRACT—Extracts a long-slit spectrum using Horne’s algorithm

Description: OPTEXTRACT performs optimal extraction of a star spectrum from a 2-D long-slit spectrum using the algorithm of Horne 1986 (PASP 98, 609). A spatial profile image should have previously been determined using the PROFILE command, and the 2-D spectrum to be extracted must be sky subtracted (e.g., using the POLYSKY command).

The input IMAGE array may have error and quality information. An error or variance array is required to perform true optimal extraction as the points should be weighted using the errors on each point as well as the spatial profile information. If an error array is not present (or not all the errors in a column are known) the extraction is performed assuming equal errors on every point in the column. The output spectrum has an error array only if the input image has an error array. The WEIGHT keyword may be set to NO to suppress error weighting even when errors are available.

Parameters:
- **IMAGE** The name of a 2-D long-slit spectrum.
- **PROFILE** The name of the normalized spatial profile image for the object to be extracted. This can be obtained using the PROFILE command.
- **SPECTRUM** The name of the resulting single spectrum.
- **WEIGHT** If WEIGHT is set, the error or variance information associated with the data is used to weight the data. Each point is weighted as 1/variance. If not set, or if no error information is available all points are given equal weight.

Source comments:

OPTEXTRACT performs optimal extraction of a star spectrum from a 2-D long-slit spectrum using the algorithm of Horne 1986 (PASP 98, 609). A spatial profile image should have previously been determined using the PROFILE command, and the 2-D spectrum to be extracted must be sky subtracted (e.g., using the POLYSKY command).

The input IMAGE array may have error and quality information. An error or variance array is required to perform true optimal extraction as the points should be weighted using the errors on each point as well as the spatial profile information. If an error array is not present (or not all the errors in a column are known) the extraction is performed assuming equal errors on every point in the column. The output spectrum has an error array only if the input image has an error array. The WEIGHT keyword may be set to NO to suppress error weighting even when errors are available.

The spatial profile array used by OPTEXTRACT must satisfy the following requirements.

(i) Its quality array must specify a window (of arbitrary shape) in which the extraction will be performed. Points inside the window must have zero quality values, other points must have non-zero quality values.

(ii) The data must be normalized. i.e. The sum along each column must be one.

(iii) The data values may not be negative.

Command parameters -
'IMAGE'  The name of the input 2-D file. This should be a sky subtracted 2-D spectrum from which the spectrum of an object is to be extracted. It may have error and quality arrays.

'PROFILE' The normalized spatial profile image used to perform the extraction. The quality array should specify a window in which the profile is determined. The data within this window is the spatial profile to be used.

'SPECTRUM' The name of the resulting 1-D spectrum. If the input image has errors, so will the output spectrum. The quality will be set to bad only if there is no data in the relevant column.

Command keywords -

'WEIGHT' Use the error or variance array to weight the pixels during extraction.

D.188  OVERPF-Overlays a FINDSP fit on another image

Description: Overlays the Polynomial Fits on an image. Displaying a zoomed part of the image is possible. To be used with FINDSP and POLEXT.

Parameters: IMAGE The fibre frame - one with distorted fibre spectra approximately equally spaced.

  BLACK  The data value below which the image display is to have the background colour. The display is scaled linearly between the data values specified as BLACK and WHITE.

  WHITE  The data value above which the image display is to have the foreground colour. The display is scaled linearly between the data values specified as BLACK and WHITE.

  PFILE  The file containing the polynomial fits. If no extension is specified, '.pol' is used.

  YSTART  First Y value to be displayed.

  YEND  Last Y value to be displayed.

  XSTART  First X value to be displayed.

  XEND  Last X value to be displayed.

  EXTWID  The input may be integer or real. An integer input causes edges of pixels included to be drawn, whereas real input causes trams lines of width EXTWID to be drawn surrounding each polynomial fit.

  REPEAT  Used to ask whether a display with different parameters should be made.

Authors:  jrl: John Lucey (AAO, Durham)

  hme: Horst Meyerdierks (UoE, Starlink)
D.189  PEAK-Determines position of highest peak in a spectrum

Description:  Returns the position and width of the highest peak in a data array. This is intended for the analysis of cross-correlation results.

Parameters:  SPECTRUM  The name of a file containing a 1-D data array, in which PEAK will determine the shift relative to the first element, and the width, of the highest peak.

Source comments:  

Determines the position and width of the largest peak in a data array. The data is treated as circular - i.e. the first element is regarded as adjacent to the last element. This routine is intended for use analysing the results of cross-correlations, and returns the position in terms of a shift relative to the first element. The algorithm used is a two step one: first a parabolic fit to the five points closest to the peak gives a value for the shift and width; the shift is then refined by convolution with the derivative of a gaussian.

Command parameters -

SPECTRUM (Character) The name of the data to to be analysed.

User variables -

(<> SHIFT (Numeric) The shift of the peak relative to the center of the first element, in pixels.

(<> WIDTH (Numeric) The width of the peak, again in pixels.

KS / AAO 29th Sept 1986

D.190  POLEXT-Extract fibre spectra from an image after a FINDSP analysis

Description:  POLEXT extracts a set of fibre spectra from a fibre image, given a file (usually produced by FINDSP) that gives the positions and shapes of the fibre spectra in terms of a set of fitted polynomials

Parameters:  IMAGE  The name of the image that contains the fibre spectra to be extracted.

PFILE  The name of a file (usually produced by FINDSP) that contains the polynomials fitted to the spectra to be extracted. If no extension is specified, .POL is assumed.

DFILE  The name of a file that lists any dud fibres - fibres to be ignored in the extraction process. If no extension is specified, .DUD is assumed. If there are no dud fibres, DFILE can be specified as blank - which may mean specifying it as " " (a blank within quotes).

EXTWIDTH  The polynomial file is used to determine the positions of the centres of all the fibre spectra. EXTWIDTH is used to specify the width of the spectra, and is specified in pixels. Note that this does not have to be an integer.

OUTPUT  The name of the resulting fibre image. Each cross-section of OUTPUT is a single extracted fibre spectrum.
**Source comments**: POLEX

Given an image containing Fibre data, and a polynomial file (produced, probably, by FINDSP) that gives the positions of the centres of the spectra, POLEX extracts the spectra to produce a new image in which each cross-section is one of the fibre spectra.

Command parameters -

IMAGE (Character) The name of the image containing the distorted fibre spectra.

PFILE (Character) The name of the polynomial file that describes the fibre positions. Default extension is .POL

DFILE (Character) The name of an optional dud fibre file that lists the numbers of any dud fibres. If blank, no file is used. Default extension is .DUD. If the file cannot be opened, this is not regarded as a fatal error, so it is possible to specify a non-existent file, such as 'NONE'

EXTWIDTH (Real) The width of the spectra in pixels.

OUTPUT (Character) The name of the resulting image.

Command keywords - None

Original version by John Lucey (JRL), AAO

---

**D.191 POLYSKY-Fits and subtracts sky from a long-slit spectrum**

**Description**: Subtract sky from a long slit spectrum by fitting a polynomial in the spatial direction.

**Parameters**: IMAGE The name of a 2-D file containing a long-slit spectrum with the spectra being along the rows (i.e. the X axis).

YS1 The first Y value to be used for the first sky region.

YE1 The last Y value to be used for the first sky region.

YS2 The first Y value to be used for the second sky region.

YE2 The last Y value to be used for the second sky region.

DEGREE Degree of polynomial to be fitted.

NREJECT Number of points to reject from each column. The NREJECT points which deviate most from the mean will not be included in the polynomial fit.

OUTPUT The name of the resulting sky subtracted data.

WEIGHT If set, the error or variance information associated with the data is used to weight the fit. Each point is weighted as 1/variance. If not set, or if no error information is available all points are given equal weight.

**Source comments**: YSKEY

POLYSKY is used to subtract sky from a long slit spectrum by polynomial fitting in the spatial direction to two regions of sky on either side of an object of interest.
Only the region of the image between the outer edges of the two sky fields is sky subtracted. Data outside this region is unchanged. This enables POLYSKY to be used repeatedly to remove sky from more than one object spectrum on an image.

The input image may optionally have associated error and quality information. If quality is present points with bad quality will be omitted from the fit. If error or variance is present the values may be used to weight the fit.

If a non zero value for NREJECT is specified this number of points will be omitted from the fit to each column. The points chosen for omission will be those which deviate most from the mean for the column.

Command parameters -

'IMAGE' The name of the input file.
'Y1S' Starting Y value to use for first sky field
'Y1E' Ending Y value to use for first sky field
'Y2S' Starting Y value to use for first sky field
'Y2E' Ending Y value to use for first sky field
'DEGREE' Degree of the polynomial fit
'NREJECT' Number of points to reject in each fit
'OUTPUT' The name of the output subtracted file.

Command keywords -

'WEIGHT' Use the error or variance array to weight the polynomial fit.

D.192 PROFILE-Profiles a long-slit spectrum for use by OPTEXTRACT

Description: Determine spatial profile image from a 2-D spectrum using the algorithm of Horne, 1986 (PASP 98, 609).

Parameters: IMAGE The name of a 2-D file.
YSTART Specifies the first Y value to be used.
YEND Specifies the last Y value to be used.
DEGREE Degree of polynomial to be fitted.
NREJECT Number of points to reject in each fit.
PROFILE The name of the resulting profile data.
RESIDUAL The name of an image containing the residuals of the fit.
WEIGHT If set, the error or variance information associated with the data is used to weight the fit. Each point is weighted as 1/variance. If not set, or if no error information is available all points are given equal weight.

Source comments: F I L E

PROFILE determines a spatial profile image for subsequent use
by the optimum extraction program OPTEXTRACT. PROFILE uses the technique described by Horne, 1986 (PASP 98, 609). An initial (noisy) estimate of the spatial profile is made by dividing the values along each column by the sum over that column. A smoothed version of this profile is then constructed by fitting polynomials along each row to account for smooth variations of the spatial profile with wavelength. The technique ensures that the profile is normalized (sums to one along each column) and is everywhere positive. Rejection of the NREJECT worst points in each fit allows the method to be insensitive to cosmic ray hits or other bad data points.

Error and quality information may be present on the input image. Points with bad quality will be ignored in the fit, and the errors may be used to weight the fit.

An image containing the residuals of the fit to the profile is generated and may be used to judge the quality of the fit.

The Horne algorithm is appropriate for the case where there is only a slight tilt or distortion on the spectrum. Where the tilt or distortion is more extreme, such as in a cross-dispersed echelle spectrum, an algorithm such as that of Marsh, 1989 (PASP 101, 1032) should be used to generate the spatial profile.

Command parameters -

`'IMAGE'` The name of the input file.
`'YSTART'` Starting Y value to use.
`'YEND'` Ending Y value to use.
`'DEGREE'` Degree of polynomial to use in fit.
`'NREJECT'` Number of points to reject in each fit.
`'PROFILE'` The name of the output file containing the normalized spatial profile.
`'RESIDUAL'` The name of the data file containing the residuals of the fit.

Command keywords -

`'WEIGHT'` Use the error or variance array to weight the polynomial fit.

D.193  Q2BAD-Converts a datafile’s quality into bad values

Description: The routine converts an datafiles quality information into bad values. There is no QUALITY structure in the output. This is a temporary measure required whilst Figaro cannot handle datafiles with both a QUALITY structure and flagged values.

Parameters: IN  The input datafile.
             OUT  The output datafile.

See also: KAPPA, SETBB
**Authors:** JM: Jo Murray (STARLINK)  
HME: Horst Meyerdierks (UoE, Starlink)

D.194 QUAL2FLAG-Converts a quality array into ‘flagged’ values

**Description:** QUAL2FLAG turns a Figaro file that holds data quality information in a quality array (or as combination of a quality array with flagged data values) into one that holds the data quality information solely in the form of flagged data values. Generally, Figaro programs prefer to work with quality arrays - which allow the original data values to be kept available even though the data element is indicated as being bad through the quality array - but there are some programs that may prefer to work with flagged data values and these will run more efficiently on data that is actually held in that form on disk. Having both forms of data quality information in a file can be confusing and QUAL2FLAG can tidy things up.

**Parameters:**
- **INPUT** The name of a Figaro format file that contains an associated quality array. It may also have some data elements flagged using ‘flagged’ (or ‘magic’) data values. Whatever it contains, QUAL2FLAG will process it so that the resulting file does not have a quality array, and all quality information will be held in the form of flagged data values in the main data array.
- **OUTPUT** The name of the resulting file. This can be the same as the input file, in which case all changes are made in situ. The resulting file will have all its data quality information held as flagged data values in its main array, and will not have a quality array.

**Source comments:** L 2 F L A G

Description:
This is a Figaro program that removes the quality array from a Figaro data file. If the quality array indicated that any of the data array elements were bad, then those elements of the main data array are set to the 'flagged' or 'magic' value. Note that this replaces the previous value in the data array, which is therefore left - so this process can remove information from the file, which is why Figaro tends to prefer the use of quality arrays rather than flagged data arrays.

Command parameters:
- **INPUT** (Character) The name of the structure containing the data.
- **OUTPUT** (Character) The name of the result of the operation. This can be the same as for INPUT. If not, a new structure is created, with everything but the data a direct copy of the input.


D.195 R2CMPLX-Creates a complex data structure from a real data array

**Usage:** r2cmplx rdata cdata
**Description:** Creates a complex data structure from a real data structure. RCMPLX sets the imaginary part of the complex data to zero. It can be set subsequently using the I2CMPLX command.

The output data follows the input in structure, except that the data array is of type DOUBLE. A zero-filled imaginary data array is also created. Any axis structures are retained.

**Parameters:**
- **RDATA**  
  RDATA is the name of an existing file that contains a data array that is to become the real part of the complex data structure that is created by R2CMPLX.
- **CDATA**  
  CDATA is the name of the complex data structure to be created. Its real part will come from the structure specified as RDATA, and its imaginary part will be set to zero. If CDATA is the same as RDATA, RDATA will be transformed into a complex structure; otherwise, a new file is created.

**Authors:** ks: Keith Shortridge (AAO)

**D.196  RCGS2-Reads UKIRT CGS2 spectrum (also UKT9 and UKT6 CVF)

**Description:**  
Reads a UKIRT CGS2 spectrum out of its original container file and create a Figaro spectrum from it. (Also works for UKT9 and UKT6 CVF spectra).

**Parameters:**
- **FILE**  
  The name of the container file to read in.
- **OBS**  
  The observation number in the container file.
- **TWOD**  
  If set, a 2-dimensional file is created giving all scans. Otherwise a individual scan can be specified or the coadded data used in a 1-D file.
- **SCAN**  
  The scan number in the observation.
- **OUTPUT**  
  The name of the Figaro file to be created from the CGS2 Data.

**Source comments:**

Reads a UKIRT CGS2 spectrum out of its original container file and create a Figaro spectrum from it. (Also works for UKT9 and UKT6 CVF spectra).

**Command parameters -**

- **FILE**  
  (Character) The name of the container file
- **OBS**  
  (Numeric) The observation number to be read from the container file.
- **SCAN**  
  (Numeric) The scan number to be read from the observation (Only used if TWOD is not set) use zero to read the coadded data.
- **OUTPUT**  
  (Character) The name of the Figaro file to be created.

**Command keywords -**

- **TWOD**  
  If set create a 2-D array of wavelength by scan number.

User variables used - None

JAB / JAC 25th Feb 1990
D.197 RDFITS-Read file in AAO de facto 'Disk FITS' format

Description: RDFITS reads a disk file containing data in a 'Disk FITS' format and creates a Figaro data file from it. RDFITS was originally intended to handle the AAO 'de facto disk FITS' format, but the addition of the SWAP keyword and code to determine the disk format in use has made it rather more general. Note that RDFITS with SWAP set to NO can read files written in the 'JT disk FITS' format handled by RJT.

Parameters: FILE The name of a disk file containing an image in 'Disk FITS' format - i.e. 2880 byte records looking exactly the same as the records in a FITS tape.

IMAGE The name of the resulting Figaro image. Note that this will contain the image data itself and the 'FITS' header information in a .FITS structure. The conventions used by the header information - e.g. is RA in hours, minutes and seconds or in fractional minutes? - may not be those used by normal Figaro routines, so be wary of just copying them into any standard observation structure.

SWAP FITS data is by definition stored on tape in IBM byte order which is the reverse of internal storage on a VAX. SWAP = YES for de facto 'disk FITS' files such as that from AAO and written by the Figaro function WDFITS. Some other 'disk FITS' formats such as JT's may require SWAP=NO.

FLOAT The default setting for FLOAT is YES, and this causes FITS to generate an output data structure with a main data array whose type will be FLOAT. This is the recommended way to create the file. If the data was originally written with BITPIX=16, then disk space can be saved by setting FLOAT=NO. FLOAT=NO will be overridden if the FITS header specifies BSCALE and BZERO values other than 1.0 and 0.0 respectively, although the special case of FLOAT=NO combined with BSCALE=1.0 BZERO=32768.0, BITPIX=16 is trapped and produces a main data array of unsigned 16bit integers (some FITS writers, notably WDFITS, use this as an efficient way of writing 16bit unsigned integer data).

See also: FIGARO: WDFITS, FITSKEYS, KAPPA: FITSDIN, FITSIN, FITSHEAD, FITSIMP, FITSLIST

Source comments: I T S

Figaro routine to read file in a 'Disk FITS' format, creating a Figaro data structure file that contains all the information from the disk file (although not necessarily in an ideal form, since the program cannot guess at the meaning of all the various FITS keywords). For more details, see the listing for FIG_FITIN.

Command parameters -

FILE (Character) The name of the 'Disk Fits' file.

IMAGE (Character) The name of the Figaro output file.

Command keywords -

SWAP Swap bytes. This should be set to YES if the data have been stored in the proper FITS format (IBM-style)
and the program is running on a non-IBM byte order machine such as a VAX. On a VAX, you should use:
SWAP=YES for AAO de facto 'disk FITS'.
SWAP=NO for WJT 'disk FITS'.

FLOAT Convert the data to floating point numbers. This is normally what will be required, although note that only single precision is supported. If FLOAT=NO is specified, FITS will still convert to floating point if the data have scaling and offset factors that are not 1.0 and 0.0 respectively. The case where FLOAT=NO, BITPIX=16, BSCALE=1.0, BZERO=32768 is treated as a special case and will generate an array of unsigned 16 bit integers. FLOAT=NO is usually only useful in this special case and in the case where BSCALE=1.0, BZERO=0.0, BITPIX=16, where it will create a smaller data file with no loss of precision.

User Variables used - None

Note: Most of the various 'disk FITS' formats differ only in whether or not they swap bytes, and in the details of the way the disk data is organised in records. For example, a VAX VMS file may have a 'FIXED', 'VARIABLE', or 'SEGMENTED' format, whereas UNIX files are generally simpler. Also on a machine that has a record-based file structure (like a VAX) the record lengths may or may not be the 2880 bytes that would match a FITS tape. This program determines the disk format (fixed, variable,segmented) and record length for itself, and uses the SWAP keyword to indicate whether the data bytes are to be treated as swapped or not. It should therefore be able to handle most of the available 'disk FITS' formats.

KS / AAO 17th June 1986

D.198 RDIPSO-Read file in DIPSO/IUEDR/SPECTRUM format

Description: Reads a file in DIPSO/IUEDR format and creates a Figaro file from the data in it. The file can have been written in any of what IUEDR calls SPECTRUM type 0, type 1 or type 2 format.

Parameters: FILE The name of the DIPSO file to be read in. If no extension is specified, .DAT will be used.
CODE There are three SPECTRUM formats used by DIPSO, and RDIPSO can handle any of them. Type 0 is unformatted, and is the most compact and fastest to use, but cannot be read as a text file. Type 1 is a fixed format text file, and type 2 is a free-format text file.
SPECTRUM The name of the Figaro file to be created from the data in the DIPSO format file.

Source comments: P S O  

Reads a file in DIPSO/IUEDR format and creates a Figaro file from the data in it. The file can have been written in any of what IUEDR
calls SPECTRUM type 0, type 1 or type 2 format.

Command parameters -

FILE (Character) The name of the DIPSO format file to be read.
CODE (Numeric) The SPECTRUM type code for the format (0, 1 or 2).
SPECTRUM (Character) The name of the Figaro file to be created.

Command keywords - None

---

D.199 REMBAD - Removes pixels that have been flagged as bad from data

Description: Remove from a spectrum points which are flagged as bad or shown as bad in the quality array. The main purpose of this command is to allow such spectra to be correctly processed by FIGARO commands which do not support data quality.

Parameters: SPECTRUM Spectrum containing bad points.
OUTPUT Name of resulting spectrum.

See also: FIGARO: BCLEAN, CLEAN, COSREJ, MEDFILT, MEDSKY, TIPPEX, KAPPA: CHPIX, FILLBAD, GLITCH, MEDIAN, ZAPLIN.

Source comments: B A D

Remove from a spectrum points which are flagged as bad or shown as bad in the quality array. The main purpose of this command is to allow such spectra to be correctly processed by FIGARO commands which do not support data quality.

Command parameters -

SPECTRUM The spectrum from which bad points will be removed.
OUTPUT The resulting spectrum.

Command keywords -

---

D.200 RENOBJ - Change the name or location of an object within an HDS file

Usage: renobj source=? destin=?

Description: This routine either renames an HDS object in place or moves it to a different place in the structure hierarchy of the same file. It is not possible to reshape objects, i.e. to change their dimensions or dimensionality.

Parameters: SOURCE The existing HDS object to be renamed. Specify beginning with directory and file name in the syntax of the operating system, followed by the dot-separated structure hierarchy. Elements of structure arrays are specified in ordinary brackets (). An array element cannot be renamed.
DESTIN The new name for the HDS object. Specify beginning with directory and file name in the syntax of the operating system, followed by the dot-separated
structure hierarchy. Elements of structure arrays are specified in ordinary brackets (). The destination cannot be an array element. The destination object must not exist beforehand.

Examples:
1. renobj source=file.MORE.FIGARO.OBS.TIME destin=file.MORE.FIGARO.TIME
   This moves the time specification from .MORE.FIGARO.OBS one level up into .MORE.FIGARO.

2. renobj source=file.ERRORS destin=file.VARIANCE
   This renames the object ERRORS into VARIANCE, but the location remains the same. (The contents remain the same anyway.)

See also: FIGARO: CREOBJ, COPOBJ, DELOBJ, SETOBJ.
          KAPPA: ERASE.

Authors: KS: Keith Shortridge (AAO)
          HME: Horst Meyerdierks (UoE, Starlink)

D.201 RESAMP-Re-sample and average several spectra (was Specdre RESAMPLE).

Usage: resamp mode inlist out start step end

Description: (This command was called RESAMPLE in Specdre, but because of a command of the same name in Figaro, it has been renamed.)
Depending on the operation mode this routine either
- takes a list of one-dimensional NDFs as input, re-samples them to a common linear grid of axis values, and averages them into a single one-dimensional NDF, or
- takes a single N-dimensional NDF as input and re-samples each row into a new row of a similar output NDF; re-sampling is along the first axis, all further axes are retained.

Parameters: MODE
MODE = _CHAR (Read) The operating mode. This can be abbreviated to one character, is case-insensitive and must be one of the following:
- ‘SPECTRA’: Average several 1-D input NDFs into a single 1-D output NDF. Re-sample before averaging.
- ‘CUBE’: Accept only one - but N-D - input NDF. Re-sample each row (1-D subset extending along first axis) separately.
Note that a single spectrum could be handled by both modes; it is more effective to treat it as a 1-D cube than as an N=1 average. ['Cube']

INFO
INFO = _LOGICAL (Read) If false, informational and warning messages are suppressed. [YES]

VARUSE
VARUSE = _LOGICAL (Read) If true, input NDFs without variance information are skipped. If false, variance information in the input is ignored. [YES]

INLIST
INLIST = LITERAL (Read) The group of input NDFs. In a complicated case this could be something like
M_51(25:35,-23.0,-24.0),M101,"LISTFILE.LIS"
This NDF group specification consists of
• one identified NDF from which a subset is to be taken,
• one identified NDF,
• an indirection to an ASCII file containing more NDF group specifications.
That file may have comment lines and in-line comments, which are recognised as beginning with a hash (#).

**OUT**  OUT = NDF (Read) The output NDF.

**START**  START = _DOUBLE (Read) The first pixel position in the output NDF. The prompt value is derived from the first valid input NDF.

**STEP**  STEP = _DOUBLE (Read) The pixel position increment in the output NDF. The prompt value is derived as the average increment in the first valid input NDF.

**END**  END = _DOUBLE (Read) The last pixel position in the output NDF. The prompt value is derived from the first valid input NDF.

**Source comments:**

The re-sampling creates an interdependence between pixels of the output NDF. Only limited information on this interdependence is stored in the output and ignored by most applications. Data input to this routine is assumed to have no such interdependence.

The re-sampling algorithm is reintegration (Meyerdierks, 1992a) and it is applied to each input NDF separately. Any re-sampled data value is a weighted sum of the input values. The weights are the normalised overlaps between the output and input pixels. The re-sampled spectra are averaged pixel by pixel.

If input variances are to be ignored it is assumed that the variance is a global constant, i.e. equal in all pixels of all input NDFs. The re-sampling may still result in different weights for different pixels. In the averaging process the global input variance is calculated and reported. The output variance will be derived on a pixel-by-pixel basis from the data scatter in the averaging process.

In any input NDF, this routine recognises axis centres (pixel positions), pixel widths, data values, and data variances. (This routine also recognises the Specdre Extension and will use it where relevant.) Any other information is propagated from the first input NDF.

Labels and units are checked for consistency, but only a warning is given. In interpreting the data all labels and units are assumed to be the same as in the first input NDF.

All input NDFs must have a variance component (unless VARUSE is set false). NDFs without variances are ignored. A warning to that effect is issued. If VARUSE is set false, input NDFs may or may not have variances, such information will be ignored at any rate.

The output NDF is based primarily on the first input NDF. There will be no pixel widths, since the pixel positions are linear and the pixels contiguous. The pixel positions, data values, and data variances will be affected by the re-sampling process. The
output Specdre Extension will be based on the first input NDF or will be created.

The vector of row sums of the covariance matrix (Meyerdierks, 1992a/b) will be created in the Specdre Extension. This is an NDF structure with only a data component of the same shape as the main data array. If such a structure is found in one of the input NDFs, a warning is issued but such information is ignored.

Examples:

```
resample spectra ^inlist out 3.5 0.0254902 10.0
```

The names of input NDFs are read from an ASCII list file called INLIST. The result will be stored in OUT which has 256 pixels covering the coordinates from 3.5 to 10.0

```
resample spectra ^inlist out varuse=false accept
```

The names of input NDFs are read from an ASCII list file called INLIST. The input NDFs either have no variance, or their variance information is to be ignored. The output will be in OUT. The start and end pixel positions for OUT are the same as in the first input NDF. OUT also has the same number of pixels. The pixel spacings are constant in OUT while they may not be in the input NDF.

```
resample cube ^inlist out 3.5 0.0254902 10.0
```

INLIST contains only one NDF probably with more than one dimension. OUT will have the same dimensions except the first, which is the re-sampled one.

Notes: The axis normalisation flag is ignored.

This routine recognises the Specdre Extension v. 0.7.

Pitfalls: This routine uses pixel widths. If there is no width array in the input NDF, the widths default as described in SUN/33. This may have undesired effects on re-sampling spectra that cover several non-adjacent coordinate ranges and where the missing ranges are not covered by bad pixels. Such spectra have highly non-linear pixel positions and the default pixel widths will not be as desired. To illustrate this consider the following spectrum with four pixels, the intended extents of the pixels and the defaulted extents:

```
 x x x x
| 1111111 | 2222222 | 3333333 | 4444444 |
| 1111111 | 33333333333333333333333333333333 | 22222222222222222222222222222222 |
| 4444444 | 1111111 |
```

Since this routine uses the overlap between input and output pixels as weights for re-sampling, non-bad pixels next to such a gap in data will affect too many output pixels with too much weight. Users should be aware that spectra as illustrated here are somewhat pathologic and that they should be given an explicit width array.

The routine accesses one input NDF at a time and needs not hold all input NDFs at the same time. However, The routine needs temporary workspace. If KMAX is the number of pixels in an input NDF and LMAX the number of output pixels, the routine needs - one vector of length LMAX, - one matrix of size KMAX by LMAX, - two matrices of size LMAX by LMAX.
These workspaces are usually of type _REAL. All (!) are of type _DOUBLE if the first valid input NDF has type _DOUBLE for either of the following: - pixel position, - pixel width, - data value, - data variance. In addition one integer vector of length LMAX is needed.

There is an oddity about this routine if only one input NDF is used and its variance array is used and some or all variance values are zero. In this case the output will formally still be an average of input NDFs using 1/variance as weights. Data with zero variance cannot be weighted and are regarded as bad. If this is a problem, users can set VARUSE false to ignore all the input variances. (Note that zero variances always cause that pixel to be ignored by this routine. But where it really calculates an average of two or more spectra, this is considered proper procedure.)

**Timing:** The time used by this routine is about proportional to the number of input NDFs. It is proportional to the square of the number of output pixels. Timing can be optimized, if the input NDFs cover about the same coordinate range as the output NDF rather than include a lot of data irrelevant for the output.

**References:** Meyerdierks, H., 1992a, Covariance in resampling and model fitting, Starlink, Spectroscopy Special Interest Group

### D.202 RESAMPLE-Rebin an image to different dimensions and/or orientation

**Description:** Given an input image, RESAMPLE creates an output image which is the result of rotating/shifting/shearing/scaling the input image. The transformation between the input and output pixel coordinates can be specified by giving a rotation ANGLE and scale factors to be applied to the X and Y axes (XMAG, YMAG), or by explicitly specifying the transformation matrix (using TRANSFORM). The later option allows shears to be specified. The first option automatically selects the output image shift and size so that the entire input image is present. The interpolation used in setting the values of the output pixels can be either ‘nearest pixel’, ‘linear’, or ‘constant noise’.

**Parameters:** IMAGE The name of the input image.
TRANSFORM Allows you to specify the transformation between the input and output pixel coordinates. You can enter a 6-element array, where the elements have the following meanings: 
\[
XIN = C(1)+C(2)*XOUT+C(3)*YOUT \\
YIN = C(4)+C(5)*XOUT+C(6)*YOUT
\]
You can invert the transformation by using the INVERT keyword. If you don’t wish to enter TRANSFORM, respond with a <CR> and you will be prompted for XMAG, YMAG, and ROTATE instead.

XMAG The scale factor to be applied to the X coordinate. For example, if you want the input image blown up by a factor of two in the X direction, specify XMAG=2. The scaling is done before any rotation.

YMAG The scale factor to be applied to the Y coordinate. For example, if you want the input image blown up by a factor of two in the Y direction, specify YMAG=2. The scaling is done before any rotation.
ANGLE The number of degrees you wish to rotate the input image. The angle is measured counterclockwise. Rotation is performed after any scaling you have requested.

INVERT Allows you to invert the transformation that you have specified.

METHOD Allows you to choose the interpolation method used when selecting the output pixel values. The three choices are:
- 1 - Nearest pixel. No interpolation is performed, the value of the nearest pixel is used. This is fast.
- 2 - Linear. A linear interpolation is performed using the values of the 4 nearest pixels.
- 3 - Constant noise. Uses the 9 nearest pixels in some mysterious way.

XSIZE Gives the number of pixels in the X-direction for the output image. The default value is that required to fit the input image after transformation.

YSIZE Gives the number of pixels in the Y-direction for the output image. The default value is that required to fit the input image after transformation.

INVALID The value for a pixel which is assumed to mean that the pixel is invalid (which usually means that it falls off the edge of the image). Such pixels are not used for interpolations purposes.

OUTPUT The name of the resulting resampled image.

Source comments:
Given an input image, RESAMPLE creates an output image which is the result of rotating/shift ing/shearing/scaling the input image. The transformation between the input and output pixel coordinates can be specified by giving a rotation ANGLE and scale factors to be applied to the X and Y axes (XMAG, YMAG), or by explicitly specifying the transformation matrix (using TRANSFORM). The latter option allows shears to be specified. The first option automatically selects the output image shift and size so that the entire input image is present. The interpolation used in setting the values of the output pixels can be either 'nearest pixel', 'linear', or 'constant noise'.

IMAGE (Character) The name of the input image.
TRANSFORM (6 element array) The transformation coefficients.
XMAG (Numeric) The X magnification factor.
YMAG (Numeric) The Y magnification factor.
ANGLE (Numeric) The rotation angle (degrees).
INVERT (Keyword) Whether to invert the transformation.
METHOD (Numeric) The interpolation method (1, 2, or 3).
XSIZE (Numeric) The X-dimension of the output image.
YSIZE (Numeric) The Y-dimension of the output image.
INVALID (Numeric) The value of an invalid pixel.
OUTPUT (Character) The name of the output image.

User variables used - None
D.203  RESCALE-Rescale using user-defined upper and lower limits

Description: This task rescales spectra/images using user-defined upper and lower limits. The scaling is such that the interval [LOWFACT,HIGHFACT] becomes the interval [0,1]. The "limits" are not thresholds, i.e. data values beyond the limits are also scaled.

Parameters: IMAGE Name of image to be rescaled.
            LOWFACT lower limit of rescaling.
            HIGHFACT upper limit of rescaling.
            OUTPUT Name of resulting image.

Source comments: C A L E

This routine rescales spectra/images using user-defined upper and lower limits. The scaling is such that the interval [LOWFACT,HIGHFACT] becomes the interval [0,1]. The "limits" are not thresholds, i.e. data values beyond the limits are also scaled.

Command parameters -

IMAGE (Character) The name of the structure containing the image.
          Uses main data array, or the x-axis data for the XCxxx routines.

LOWFAC (Numeric) The lower limit of the rescaling.

HIGHFAC (Numeric) The upper limit of the rescaling.

OUTPUT (Character) The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.


D.204  RETYPE-Changes the type of the main data array in a file

Description: RETYPE changes the type of the main data array in a Figaro data structure. It converts the data into the new type (warning of any conversion errors that this might generate). Depending on the structure of the file, RETYPE might leave some deadwood in the output file (i.e. it may be larger than it need be). If so, this may be removed using TRIMFILE.

Parameters: INPUT The name of the input data structure in which the type of the main data array is to be changed.
            TYPE The new type for the data array in the data structure. It should be any type recognised by FIGARO (i.e., one of 'FLOAT', 'INT', 'DOUBLE', 'SHORT', 'BYTE', and 'USHORT').
            OUTPUT The name of the resulting data file which will be the same as the original data except for the type of the main data array.

Source comments: Y P E
Name:
RETYPE

Function:
Change the type of the main data array in a Figaro structure.

Description:
RETYPE converts the data in the main array of a Figaro structure from its current type to a specified type. For example, it could convert a USHORT data array created by a data acquisition system into a FLOAT array (which is easier to work with, especially if you contemplate dividing your data by things such as flat fields).

Parameters:
- INPUT (Character) The name of the input data structure.
- TYPE (Character) The type to which the data array is to be converted. This can be any of the types recognised by Figaro.
- OUTPUT (Character) The name of the output data structure.

Keywords: None

Note:
Depending on the structure of the file, RETYPE may end up creating files that are larger than they need to be. In this case, TRIMFILE can be used to cut out the deadwood.

D.205 ROTX-Rotate data along the X-axis

Description: ROTX rotates the data in a file in the X-direction. Data that is shifted out of one end is shifted in at the other end. The data have to be rotated by an integer number of pixels.

Parameters: IMAGE The name of the data to be rotated in the X-direction. The data may have any number of dimensions. Data with more than one dimension is treated as a set of one dimensional arrays, and each is rotated independently.

PIXELS The number of pixels that the data is to be rotated. This must be an integer, and will be truncated if a non-integer value is specified. A positive value indicates a rotation towards higher pixel numbers in X.

OUTPUT The name of the resulting rotated data. If this is the same as IMAGE, (the default), the rotation takes place in situ. Otherwise, a new file is created. Note that the X data structure (if any) is unchanged by ROTX.

Source comments:

Rotates a data array by an integer number of pixels in the X direction. The X data is not changed.

Command parameters -

IMAGE (Character) The name of the structure containing the data to
be rotated.

PIXELS (Numeric) The number of pixels by which the data is to be rotated. A positive number indicates a shift towards higher pixel numbers.

OUTPUT (Character) The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

D.206 SCLEAN-Interactive patching of images, especially SCUBA data.

Description: SCLEAN is an interactive routine for marking and fixing bad pixels, rows and columns in two dimensional images. It has all the facilities of CLEAN, plus some additional ones suitable for use with files generated by the SCUBA instrument. The principal advantages for SCUBA users of SCLEAN over CLEAN are that this application marks bad pixels by setting one of the bits (specified by the parameter BITNUM) of the quality array, rather than simply setting the data value to the magic bad value, and that an additional viewing mode is available which enables a plot of the values for a given bolometer to be seen easily.

In interactive mode, an image is displayed and operations are specified using the cursor and keyboard. In batch mode, the program will read a file of instructions and perform the operations without user intervention. SCLEAN will also optionally write a log file of the operations it has performed in the batch file format, so that a set of operations can be performed interactively, and then reperformed on the same or a different NDF by writing to the log file in the first instance and using that as a batchfile for subsequent runs.

Parameters: IMAGE IMAGE is the image - usually a SCUBA NDF - that is to be cleaned of bad pixels.

OUTPUT OUTPUT is the name of the image that results from the cleaning process.

IDEV IDEV is the name of the image device to use for display. In batch mode (i.e. if BATCHFILE is non-null) it defaults to null, in which case no graphic display is performed. It is an error for IDEV to be null in interactive mode.

QUIT Used to confirm quitting the application.

DEG Degree of fit to use for interpolation.

XSIZE Size of interpolation box in X.

YSIZE Size of interpolation box in Y.

HIGH Highest displayed data value.

LOW Lowest displayed data value.

BITNUM Bit number of quality mask to modify for bad pixels.

ZOOM Minimum pixel magnification factor in Expanded and SCUBA mode.

LOGFILE Name of a file to log to. If null, no logging is performed. The log file begins with comment lines naming the IMAGE and OUTPUT NDFs, and continues with one line per cleaning operation, consisting of the command letter followed by zero, one or two arguments. This log file is understood if read by a subsequent invocation of the program as the BATCHFILE parameter.
**BATCHFILE** Name of a file from which to read batch input. If null (the default) the program will run in interactive mode, but if this parameter has a value the named file will be read to determine the cleaning operations to be performed so that no user intervention is requested.

If a line is encountered which cannot be executed (e.g. it contains an invalid command or refers to a region outside the image) the user is notified and the line is ignored. No other attempt is made to verify that the commands in the batch file are appropriate for the particular image being examined.

Normally the batchfile will have been written by a previous invocation of SCLEAN using the LOGFILE parameter, but it can in principle be modified or written by hand. The format of the file is plain text consisting of one single letter command in column one followed by zero, one or two numeric arguments. Anything after a hash (#), and blank lines, are ignored.

**Commands:** The following keys are recognised -

- **W** - display Whole image
- **E** - Expand image around cursor position
- **B** - SCUBA type display for cursor column
- **R** - delete indicated Row (horizontal line) & fix it
- **C** - delete indicated Column (vertical line) & fix it
- **X** - delete indicated area and fix by interpolation in X direction
- **Y** - like X, but uses vertical interpolation
- **L** - delete indicated line, but don’t fix
- **K** - delete indicated column, but don’t fix
- **A** - delete pixel at cursor, but don’t fix
- **G** - mark as Good pixel at cursor
- **S** - set stretch, ie High & low limits for display
- **D** - set degree of fit used for interpolation
- **N** - set size of deleted area for "X" and "Y"
- **P** - indicate current cursor position
- **T** - test area to see what BCLEAN might find there
- **U** - Undo last operation - replace with original image data
- **Q** - Quit program

**H** - help

A summary of available commands is displayed.

**W** - display Whole image

The whole image is displayed, stretched or compressed in both directions to fit the available plotting surface.

**E** - Expand image around cursor position

A section of the image around the cursor position is displayed with ZOOM * ZOOM display pixels representing each image pixel.

**B** - SCUBA type display for cursor column

A special display mode is used suitable for NDFs from the SCUBA instrument. Each image pixel is at least ZOOM display pixels in X and Y directions, although it will be displayed larger if there is space. Additionally, on one half of the screen is displayed a line plot of one column (the column the cursor was on when ‘B’ was selected). If the cursor is on the graph half of the screen it is considered to be on the column plotted.

**R** - delete indicated Row (horizontal line) & fix it

The row of pixels at the cursor is replaced by interpolation.

**C** - delete indicated Column (vertical line) & fix it

The column of pixels at the cursor is replaced by interpolation.

**X** - delete indicated area and fix by interpolation in X direction

The XSIZE * YSIZE region at the cursor is replaced by a horizontal interpolation using a polynomial fit of degree DEG.
Y Y - delete indicated area and fix by interpolation in Y direction
   The XSIZE * YSIZE region at the cursor is replaced by a vertical interpolation using a
   polynomial fit of degree DEG.
S S - set stretch, ie high & low limits for display
   The user is queried for the parameters HIGH and LOW, the highest and lowest values
   of pixel to be represented in the colour mapping.
L L - delete indicated line, but don’t fix
   The row at the cursor is marked as bad.
K K - delete indicated column, but don’t fix
   The column at the cursor is marked as bad.
A A - delete pixel at cursor, but don’t fix
   The pixel at the cursor is marked as bad.
G G - mark as Good pixel at cursor
   The pixel at the cursor is marked as good (In fact the bit BITNUM in the quality byte
   is set to zero. If other bits in the quality byte are set, or if the data value itself is
   flagged bad, the pixel may still be bad.)
D D - set degree of fit used for interpolation
   The user is prompted for the parameter DEG, the degree of the polynomial used for
   interpolation by the X and Y commands.
N N - set size of deleted area for "X" and "Y"
   The user is prompted for the values of the parameters XSIZE and YSIZE, the dimen-
   sions of the box used for interpolation by the X and Y commands.
P P - indicate current cursor position
   The current cursor position is printed.
T T - test area to see what BCLEAN might find there
   The algorithm used by the BCLEAN application is run and the user is informed what
   spikes it would identify.
U U - Undo last operation - replace with original image data
   The last operation which modified the data is undone. Until the undo buffer fills up
   (usually it does not) all cleaning operations back to the start of the session can be
   undone in a Last In First Out fashion.
Q Q - Quit program
   The program is exited following a prompt for confirmation.

See also: FIGARO: BCLEAN, CLEAN, COSREJ, MEDFILT, MEDSKY, TIPPEX
          KAPPA: FFCLEAN, CHPIX, FILLBAD, GLITCH, MEDIAN, MSTATS, ZAPLIN.

Source comments: 
Main routine for the Figaro 'SCLEAN' command. Displays
a CCD image and then allows the user to move around it with
the cursor, selecting rows and columns to be corrected and
cosmic rays to be zapped. The idea is that this routine can
be used to fix up any areas in an image that were not fixed
automatically by the non-interactive version (‘BCLEAN’). It may also give a better idea of the best settings for the BCLEAN parameters. A mode is provided especially suited for examination of SCUBA data files.

The task works internally with a quality array, setting the bit specified by the parameter BITNUM to mark pixels as bad, but if the input NDF uses flagged bad values and no quality array, the output NDF will represent quality information in the same way.

Variances are propagated, but if any changes are made to the data array, the corresponding element of the variance array is set to zero. This is intended to mark it as a bad value, since Figaro does not support explicitly flagged bad values in the variance array. If this has been done, the user is warned at the end of processing.

Command parameters -

- **IMAGE** (Character) The name of the image to be displayed.
- **OUTPUT** (Character) The name of the resulting cleaned image. If the same as IMAGE, the image is cleaned in situ.
- **QUIT** (Logical) Used to confirm quitting the application.
- **DEG** (Integer) Degree of fit to use for interpolation.
- **XSIZE** (Integer) Size of deletion box in X.
- **YSIZE** (Integer) Size of deletion box in Y.
- **HIGH** (Real) Highest displayed data value.
- **LOW** (Real) Lowest displayed data value.
- **BITNUM** (Integer) Number of quality bit to modify.
- **ZOOM** (Integer) Minimum pixel zoom factor.
- **LOGFILE** (Character) The name of a file to log to. If null, no logging is performed.
- **BATCHFILE** (Character) The name of a file (in SCLEAN log file format) to draw batch input from. If null (default), run in interactive mode.

User variables - ("<" output, "!" modified)

- (!) **IMARRAY** (Numeric array) Contains current image display parameters.
- (<) **IMFILE** (Character) Contains name of currently displayed image file.
- (>) **IDEV** (Character) Contains name of display device.

D.207  SCNSKY-Calculates a sky spectrum for a scanned CCD image

**Description:** Uses the ‘minimum median’ algorithm to create a sky spectrum from an image without any lines clear of stars. Each column of the image is split into sections of a specified length and the median of each section is calculated. The sky element corresponding to the column is set to the value of the minimum median for that column (on the assumption
that this is an uncontaminated value). This algorithm is suitable for exposures taken using a CCD in scanned mode, hence the name of the routine.

**Parameters:**

- **IMAGE** Name of image to obtain sky from.
- **SECTION** Length of each Y-section.
- **SPECTRUM** Name of spectrum to be generated.
- **LIMIT** Ignore pixels below a minimum value?
- **MINIMUM** Minimum valid pixel value

**Source comments:**

S C N S K Y

Uses the 'minimum median' algorithm to create a sky spectrum from an image without any lines clear of stars. Each column of the image is split into sections of a specified length and the median of each section is calculated. The sky element corresponding to the column is set to the value of the minimum median for that column (on the assumption that this is an uncontaminated value). This algorithm is suitable for exposures taken using a CCD in scanned mode, hence the name of the routine.

**Command parameters -**

- **IMAGE** The name of the image
- **SECTION** The number of pixels in each section.
- **MINIMUM** A minimum 'valid' pixel value. A median value will be ignored if the range of pixels over which it was calculated contained pixels below this value.
- **SPECTRUM** The name of the resulting sky spectrum.

**Command keywords -**

- **LIMIT** Use the minimum pixel value to limit the median calculations - this is needed because there is no obviously illegal value for 'MINIMUM' which the program can take as an indication that it is to be ignored. (If 'MINIMUM' is given in the command string explicitly, 'LIMIT' need not be set.)

**Input data -**

- **IMAGE** is assumed to have a 2-D data structure.

**Output data -**

- **SPECTRUM** is created with the same structure as **IMAGE**, except that the data will only have one dimension, and if **IMAGE** has an AXIS(2) structure, this will be omitted. Any AXIS(1) structure will be copied unchanged.

KS / CIT 25th May 1983
D.208  SCROSS-Cross-correlate two spectra and get relative shift

Description: SCROSS cross-correlates two spectra and reports a relative shift in pixels between the two.

Parameters: SPECTRUM  The name of the spectrum whose shift relative to a template spectrum is to be determined.

TEMPLATE  The name of the spectrum to be used as a template - the spectrum relative to which the shift of SPECTRUM is to be determined.

XSTART  The shift determination can be carried out over a restricted number of pixels. XSTART is the X-value of the first pixel to be used. If SPECTRUM and TEMPLATE have different X arrays then that of SPECTRUM is used. I.e, XSTART is converted into a pixel number using the calibration for SPECTRUM, ditto XEND, and this then gives the pixel range used from both SPECTRUM and TEMPLATE.

XEND  The end of the range used. Note that XSTART and XEND effectively specify a spectrum which is a subset of the original spectrum, and it is these subset spectra that are used - i.e. the cosine bell is applied to these subsets, not the whole spectrum, and similarly for the other operations performed.

FITCONT  SCROSS - unlike some other cross-correlators - attempts a continuum fit to the spectra prior to applying a cosine bell and taking the fourier transform. This replaces the reduction to zero mean sometime applied. For some data, this may not be a suitable thing to do, and it can be inhibited by setting FITCONT=NO. However, if you do so, you should have applied your own reduction to zero mean to the data prior to using SCROSS. FITCONT is the recommended option.

CBPC  By default, a cosine bell covering 10% of the spectrum is applied. This is normally OK, but it can be changed by specifying a different value for CBPC.

RECORD  As an option, SCROSS will create a file containing the cross-correlation function. This can be used to see how good the cross-correlation was, or can be used to re-calculate the shift, by applying a different centering algorithm to the central spike. (You can use it as input to ARC, for example.)

CROSS  If RECORD has been set, CROSS gives the name of the cross-correlation spectrum to be created.

Source comments: R. O. S. S.

Main body of the Figaro SCROSS function. This computes the cross-correlation of two spectra and the location of the central peak of the cross-correlation. It can be used to determine a relative shift between two spectra. The cross correlation function can also be saved in a disk structure.

SCROSS uses the Fourier cross-correlation technique, which is described in, for example:


Command parameters -

SPECTRUM (Character) The spectrum to be compared with a template spectrum.

TEMPLATE (Character) The template spectrum to be used. The two spectra should be the same length.

XSTART (Numeric) Data with an axis data value less than XSTART will be ignored in the cross-correlation.

XEND (Numeric) Data with an axis data value greater than XEND will also be ignored. Note that these values are used to determine the channel numbers to be used for SPECTRUM, and the same ones will be used for TEMPLATE, even if TEMPLATE has a different axis structure.

CBPC (Numeric) Percentage of spectrum covered by a cosine bell prior to application of the FFT.

CROSS (Character) the name of the data structure to hold the cross-correlation, if it is to be saved. The file created will be cross.dst, and will look like an ordinary spectrum - i.e. can be plotted by SPLOT, etc. CROSS is ignored if RECORD is not set.

Command keywords -

FITCONT If set, a continuum fit is performed on the two spectra prior to application of the cosine bell.

RECORD If set, the cross-correlation of the two spectra will be recorded as a new data structure.

User variables used -

SHIFT (Numeric) The relative shift of the two spectra.

KS / CIT 3rd Oct 1983

D.209 SCRUNCH-Rebin a spectrum to a linear wavelength range

Description: SCRUNCH rebins a spectrum or a set of spectra so that the resulting data have either a linear wavelength scale or a wavelength scale that is logarithmic (i.e. has a constant velocity step). Note that SCRUNCH is not suitable for data in which the bins contain data that covers discontinuous or overlapping wavelength ranges (FIGS data, for example).

Parameters: SPECTRUM The spectrum (or spectra) to be scrunched. If SPECTRUM is an image (i.e. 2-D data) it will be treated as a set of spectra all of which will be scrunched individually. In this case, if the wavelength data is a 2-D array each of the data spectra will be scrunched according to the corresponding cross-section of the wavelength array. If the wavelength array is 1-D, this single array will be used for all the spectra.

WSTART The wavelength of the center of the first bin of the resulting output data.

WEND Normally, represents the wavelength of the center of the last bin of the resulting output data. Because this is not always the most convenient value to supply,
SCRUNCH will allow WEND to be used to specify the wavelength increment value (for linear data, this is the constant wavelength difference between bins; for logarithmic data it is the constant velocity step in Km/sec). The way WEND is interpreted may be controlled explicitly by the FINAL and INCREMENT hidden keywords, but by default SCRUNCH will assume WEND is an incremental value if it is less than WSTART, and a final value if it is greater than WSTART.

**BINS** The number of elements to be used for the resulting spectrum.

**OUTPUT** The name of the resulting data file containing the scrunched spectrum or set of spectra. The structure of OUTPUT will be the same as that of the input file, except that the data array will generally be a different size, and any error array will have been deleted. Eventually SCRUNCH will be able to calculate the errors in the new spectrum.)

**INCREMENT** If set, SCRUNCH will assume WEND is an incremental value (a velocity or wavelength step), even if it is greater than WSTART.

**FINAL** If set, SCRUNCH will assume that WEND is the wavelength of the final element of the resulting spectrum, even if it is less than WSTART.

**LOG** Controls whether or not the data is binned to a linear or a logarithmic wavelength scale.

**LINEAR** Specifies that linear rather than quadratic interpolation is to be used during the scrunching. Usually, better results are obtained with quadratic interpolation. Note that LINEAR is the opposite of QUAD, and NOT of LOG.

**QUAD** Specifies that quadratic rather than linear interpolation is to be used during the scrunching. Usually, better results are obtained with quadratic interpolation.

**FLUX** If the input data represents flux (photons, for example) rather than flux density (AB magnitudes, Janskys, etc) then it should be scrunched so as to conserve flux, and FLUX should be set. See HELP FIGARO TECHNIQUES WAVELENGTH fo more details on this point.

**MEAN** If the input data represents flux density (AB magnitudes, Janskys, etc) rather than flux (photons, for example) then it should be scrunched so as to conserve the mean value of the data, and MEAN should be set. See HELP FIGARO TECHNIQUES WAVELENGTH fo more details on this point.

**Source comments:**

Figaro routine to scrunch a spectrum or set of spectra. Can scrunch either into a linear wavelength scale, where the wavelength increment from bin to bin is constant across the spectrum, or into a logarithmic scale, where the increment of the log of the wavelength from bin to bin is constant. The operation is performed by the routine FIG_REBIN.

If the input file is 2-D data, then it is treated as a set of 1-D spectra and each is scrunched individually. If the wavelength array is 1-D, then this single array will be used for all the spectra. If it is 2-D, then each spectrum will be scrunched according to the corresponding cross-section of the wavelength array.
The routine can either conserve flux or the mean value of the data. Conserving flux is appropriate where the data is actually in flux units, but not when the data is in magnitude units. (Consider the case where each input bin maps to two output bins; if the data is in flux units - photon counts, for example - then flux should be conserved and the mean data level should drop by a factor 2; if the data is in magnitudes, then the rebinning should not change the mean level.) The program does not try to determine for itself whether the data is in flux or in magnitudes - it uses a command keyword.

Command parameters -

**SPECTRUM** (Character) The name of the spectrum to be scrunched.

**WSTART** (Numeric) The wavelength of the CENTER of the first bin of the resulting scrunched spectrum.

**WEND** (Numeric) The wavelength of the CENTER of the final bin of the resulting scrunched spectrum. If WEND is less than WSTART, then SCRUNCH assumes that it is the increment rather than the final value that is being specified. If the scrunch is logarithmic and WSTART is greater than WEND, SCRUNCH assumes that the WEND value represents a velocity in km/sec. These assumptions can be controlled directly by the keywords INCREMENT and FINAL, if they will not give the desired effect.

**BINS** (Numeric) The number of bins for the resulting spectrum.

**OUTPUT** (Character) The name of the resulting spectrum.

Note that SCRUNCH cannot rebin a spectrum into itself and so will always create a new output file.

Command keywords -

**LOG** Bin into logarithmic wavelength bins.

**MEAN** Conserve mean data level rather than flux.

**FLUX** Conserve flux rather than mean data level.

**LINEAR** Use linear interpolation when rebinning.

**QUAD** Use quadratic interpolation when rebinning.

**INCREMENT** WEND is an increment value, even though it is > WSTART.

**FINAL** WEND is a final value, even though it is < WSTART.

User variables -

**SCRUNCH_INC** Is set to the wavelength increment if linear rebinning is used, and to the velocity increment if log rebinning is used.

**SCRUNCH_END** Is set to the final wavelength value. (This is for those cases where the WEND value represents an increment.)
D.210  SDIST-Analyse an image containing spectra for S-distortion

**Description:** SDIST traces curved continuous spectra in an image. These can be due to image tube S-distortion (the original use of the program), or may perhaps be curved echelle orders. SDIST fits polynomials to the spectra and writes the fits to a file that may be used by other programs such as CDIST. Before using SDIST, starting points for all the spectra to be traced must be indicated, usually by displaying the image using IMAGE and selecting points in the centers of the various spectra using ICUR.

**Parameters:**

**IMAGE** The image that contains the spectra to be traced. Normally, this will be already displayed on the image display, and the centers of the spectra (the places SDIST is to start tracing from - it goes to the edge in both directions out from the selected points) should have been selected using ICUR.

**COLUMNS** SDIST adds the data in several columns at a time to get a cross-section through the spectrum it is tracing. The more columns it averages, the better the signal to noise it obtains, the faster the processing, but the jumpier the fit may become. A few columns is normally enough. It should be increased for noisier data.

**TRACE** SDIST applies one of a selection of algorithms to trace the spectra. Four are available at present:
- **G** (the original option) fits a guassian to the profile of the data. This is suitable for stellar spectra taken with a wide slit, or any data that genuinely is gaussian.
- **E** (Edge) uses an algorithm that looks for the edges in data and takes the average edge position. It is suitable for data that is roughly ‘top-hat’ in shape but tends to give rather quantised results when the edges are very sharp (such edges cannot be determined to better than a pixel). This shows up clearly if the SOFTD option is used.
- **C** (COG) is a compromise option for non-gaussian data that determines the edges, then used the center of gravity of the data between them. B (Balance) is an extremely crude algorithm that determines the centre of gravity over the width specified. It is very robust, but very crude - use only in desperation.

**WIDTH** SDIST needs an indication of the width of the spectra it is to trace. If the trace mode is G, this is the half width of the gaussians it fits to the data, so can be quite critical. Do not make this too large. In particular, the greater SDIST is, the further from the center it looks, and can go off the edge of the image for extreme spectra. For the edge tracing options, the width must not be so great that it encompasses the falling edges of the next spectrum out. Try to give an accurate value for WIDTH.

**MAXDEG** SDIST fits polynomials of degree MAXDEG to the spectra it traces. In general, MAXDEG should be as no higher than is needed to get a satisfactory fit, since this minimises any effects at the edges if the spectra could not be traced all the way to the edges of the image.

**SOFTD** If set, SDIST plots for each spectrum the points traced and the curve fitted to them. This can be particularly useful when trying to decide on the trace mode to use - the quantised effects sometimes found with the E(dge) trace show up very well here.

**DIAGNOSE** If set, the detailed reasons for any failures to trace the various spectra are logged. This is too detailed to be of general interest, being most useful when new trace algorithms are being debugged.

**NEXT** Used to pause between plot pages.
Source comments:

Determines the s-distortion parameters for a star spectrum. This program requires as input an image containing at least one stellar spectrum. For each spectrum it is to examine, it assumes that a point has been indicated somewhere in the center of the spectrum. Starting at that point, it attempts to follow the spectrum and so map out the distortion.

Command parameters -

- **IMAGE** (Character) the image containing the spectra.
- **COLUMNS** (Numeric) The number of columns to be added together when tracing the spectra.
- **TRACE** (Character) Controls the algorithm used to follow the spectra. Only the first character is significant. 'E' (Edges) Indicates that the data profile is a 'top-hat' shape - as you might get from a continuum source through a dekker. If EDGES is set, an edge locating algorithm is used, and the width of the top hat is assumed to be approximately WIDTH*2. 'C' is the same as 'E', except that the center is taken as the center of gravity of the data within the edges, while 'E' takes it as the average of the edges. 'G' (Gaussian) Indicates that the profile is roughly gaussian, of half width WIDTH.
- **WIDTH** (Numeric) The expected half-width of the spectra in pixels.
- **MAXDEG** (Numeric) The maximum degree polynomial to be fitted.

Command keywords -

- **SOFTD** Indicates that the fit results are to be plotted on the current soft graphics device.
- **DIAGNOSE** Requests more detailed diagnostics on the tracing
- **NEXT** Used to pause between plot pages.

User variables - ("reater input, "<" output)

- **IMARRAY** (Numeric array) Contains the parameters that specify the current image display. See IMAGE for full details.
- **NPIXELS** (Numeric) Number of points selected using ICUR
- **XPXELS** (Numeric array) X positions of points selected
- **YPXELS** (Numeric array) Y positions of points selected

Output files -

- **SDIST.DAT** contains the results of the fit(s), in a format that can be treated as follows -

  3 header lines, all beginning with '*'
  One line giving the number of spectra traced, and the
dimensions of the image used, in the format 20X,I5,15X,I8,4X,I8.

Then, for each spectrum traced, one record giving the spectrum number, and the leftmost and rightmost pixels covered by the trace, in format 11X,I5,17X,I5,4X,I5, then 1 record giving the average Y value in the spectrum, and the width of the spectrum, in format 16X,F13.7,10X,F9.2, which is followed by 3 records giving the 11 polynomial coefficients for the fit, in 3D23.16. Coefficients are given constant first, with any unused coefficients set to zero.

KS / CIT 5th Feb 1984

D.211  SETOBJ-Assign value to an HDS primitive

Usage:  setobj value object

Description: This routine assigns a specified (numeric or string) value to an existing HDS primitive. The destination object must exist. It can be a primitive scalar or a cell in a primitive array.

Parameters:  VALUE  The value the scalar primitive HDS object is to assume.

OBJECT  The HDS object to be modified. Specify beginning with directory and file name in the syntax of the operating system, followed by the dot-separated structure hierarchy. Elements of structure arrays are specified in ordinary brackets ()

Examples:  1. setobj value=90. object=file.MORE.FIGARO.TIME
        Store the number or string 90 in the existing scalar HDS object MORE.FIGARO.TIME in the file "file".

See also:  FIGARO:[CREOBJ][COPOBJ][DELOBJ][RENOBJ][KAPPA][ERASE]

Authors:  KS: Keith Shortridge (AAO)

          HME: Horst Meyerdierks (UoE, Starlink)

          JFL: John Lightfoot (ROE)

D.212  SFIT-Fit a polynomial to a spectrum

Description: Task to generate a spectrum by fitting a polynomial to an input spectrum. An orthogonal polynomial of specified order is fitted either to the spectrum or to its logarithm, and the result is used to generate a new spectrum.

Parameters:  SPECTRUM  Name of spectrum to be fitted.

ORDER  Order of fitted polynomial.
LOGS  Fit to logarithms of data.
OUTPUT  Name of resulting spectrum.

Source comments:

Figaro function to generate a spectrum by fitting a polynomial to an input spectrum. An orthogonal polynomial of specified order is fitted either to the spectrum or to its logarithm, and the result is used to generate a new spectrum.

Command parameters -

SPECTRUM  (Character) The name of the spectrum to be fitted.
ORDER  (Numeric) The order to be used for the fit.
OUTPUT  (Character) The name of the output file to be created. If this is the same as the input spectrum, the data in the input spectrum will be modified.

Command keywords -

LOGS  Indicates that the logs of the pixel values of the input spectrum are to be fitted.

KS / CIT 5th June 1983

D.213  SKYLINER-Removes a sky spectrum normalised the 5577 [OI] line.

Description:  Removes a sky spectrum normalised by the height of the 5577 [OI] emission line.
Usage:  skyliner in sky out

Parameters:  IN  The one- or two-dimensional spectrum to be sky-subtracted.
             SKY  The one-dimensional sky spectrum. It must not contain bad pixels as they could affect the estimation of the [OI] line strength.
             OUT  The sky-subtracted one- or two-dimensional spectrum.

Source comments:  available.

D.214  SLICE-Takes a slice with arbitrary end points through an image

Description:  SLICE creates a ‘spectrum’ by taking a slice through a two dimensional image. This particular program is essentially lifted from the AIPS package and uses the scheme described there as ‘Everett Interpolation’. Note that the ‘slice’ through the image is conceptually of zero width - i.e. it is a slice through the interpolated surface represented by the discrete image data.

Parameters:  IMAGE  The image file to be sliced.
YSTART The Y value of the starting end of the slice. Note that SLICE conceptually takes a slice of zero width through the interpolated surface generated from the image data. Therefore, the ends of the slice need not be centered on an image pixel. The convention used is that the X and Y values used for a pixel refer to the CENTER of the pixel. It may help to point out that if IMAGE were a 256 by 256 image, the commands
SLICE IMAGE XST=1. XEN=256. YST=128. YEN=128. ÊL=256 EXTRACT IMAGE YST=128 YEN=128 would be equivalent

YEND The Y value of the terminating end of the slice. Note that YEND may be less than YSTART.

XSTART The X value for the starting end of the slice.

XEND The X value of the terminating end of the slice. Note that XEND may be less than XSTART.

ELEMENTS Since the slice is taken - conceptually - through a continuous surface generated by interpolating between the pixel values in the image, it can have any number of pixels desired.

SPECTRUM The name of the spectrum generated. Any data objects in the original image will be copied over into SPECTRUM, except for the X and Y information. Which is a good reason for using EXTRACT or YSTRACT even in cases where SLICE would generate equivalent data.

Source comments:

SLICE creates a 'spectrum' by taking a slice through a two dimensional image. This particular program is essentially lifted from the AIPS package and uses the scheme described there as 'Everett Interpolation'. For more details see the comments for the various subroutines such as FIG_CSLICE. Note that the 'slice' through the image is conceptually of zero width - i.e. it is a slice through the interpolated surface represented by the discrete image data.

Command parameters -

IMAGE (Character) The image from which the slice is to be taken.
YSTART (Numeric) The AXIS(2) value for the start of the slice.
YEND (Numeric) The AXIS(2) value for the end of the slice.
XSTART (Numeric) The AXIS(1) value for the start of the slice.
XEND (Numeric) The AXIS(1) value for the end of the slice.
Note that YSTART etc can represent any point in the image, and the convention is that axis data values for pixels refer to the CENTERS of those pixels - This means, for example, that if IMAGE is a 256 by 256 image, the commands
SLICE IMAGE XSTART=1.0 XEND=256.0 YSTART=128.0 YEND=128.0 EXTRACT IMAGE,128,128 will have exactly the same results.

ELEMENTS (Numeric) Number of pixels in the resulting spectrum.
SPECTRUM (Character) The resulting spectrum.

KS / CIT 23rd March 1984
D.215 SOFT-Sets the device/type for terminal graphics

Description: The SOFT command is used to set the current device and type for all 'soft-copy' (as opposed to 'hardcopy') graphics output.

Parameters: SOFTDEV The device to which all soft-copy (as opposed to hardcopy) graphics output is to be sent. Normally, a device name as recognised by SGS should be used - that is, the device specified should be one of those listed in response to the OPTIONS keyword - i.e. by the command SOFT OPTIONS. If necessary, a device type and connection id can be used, by specifying e.g. SOFTDEV="827,1" and by setting FORCE=YES.

OPTIONS If set, a list of all the device names recognised by the SGS system is output.

FORCE Normally, SOFT will reject a device specification that is not known to SGS. However, if FORCE is set, it will use whatever is specified for SOFTDEV unquestioningly. If it fails, that's your problem.

DRAW If set, SOFT will output a test plot to the specified device. This acts as a test that the specification was correct.

Source comments:

SOFT / HARD / IDEV

SOFT is used to set the user variable (SOFT) that controls the soft copy graphics output of Figaro programs. HARD is used to set the user variable (HARD) that controls the hardcopy graphics output. IDEV is used to set the user variable (IDEV) that controls the image display.

Command parameters -

HARDDEV (Character string) A device name recognised by or the GKS version of PGPLLOT. Normally, what should be used is one of the device names recognised by or GNS. HARDDEV and SOFTDEV are used by HARD and SOFT respectively.

Command keywords -

OPTIONS Causes a list of the various acceptable SGS device names to be output.

FORCE Forces the device specification to be accepted even if it does not match one of the acceptable SGS names.

DRAW Draws a test diagram on the screen. (DRAW is the default for SOFT and IDEV, DRAW=NO is the default for HARD)

KS / AAO 16th March 1988

D.216 SPECCONT-Contour a two-dimensional cut.

Description: This routine displays a two-dimensional cut as a contour plot. The cut can be an ordinary image, a long-slit spectrum, or any cut through a spectroscopic data cube. The plot area is the current (AGI) picture of the graphics device.

Parameters:

CLEAR  CLEAR = _LOGICAL (Read) If true the plot area will be cleared before plotting. [FALSE]

OVERLAY  OVERLAY = _LOGICAL (Read) If true then the last (AGI) data picture inside the current (AGI) picture is used to define the plot area and its world coordinates. Only that area will be cleared if CLEAR is true. No new labelling of the plot will occur. [FALSE]

IN  IN = NDF (Read) The input NDF. It must be two-dimensional - not counting degenerate axes.

DEVICE  DEVICE = GRAPHICS (Read) The graphics display device.

FILL  FILL = _LOGICAL (Read) If false then the plot window will be adjusted to give the same plot scale horizontally and vertically. If true, scaling is independent in each direction and the plot will fill the area available. This parameter is used only if OVERLAY is false. [FALSE]

ROMAN  ROMAN = _LOGICAL (Read) If true, PGPLOT’s roman font is used for drawing text. If false, the normal (single-stroke) font is used. [FALSE]

HEIGHT  HEIGHT = _REAL (Read) The height of the characters measured in units of PGPLOT default text height, which is approximately 1/40 of the height of the (AGI) base picture (i.e. 1/40 the height of the work station window, screen or paper). HEIGHT will be used for labelling the plot box. The contour labels are always half that size. [1.]

COLOUR  COLOUR = _INTEGER (Read) The PGPLOT colour index to be used for the plot. This can be formally between 0 and 255, but not all devices support all colours. The default colour representation is: - 0: Background, - 1: Foreground (default), - 2: Red, - 3: Green, - 4: Blue, - 5: Cyan, - 6: Magenta, - 7: Yellow, - 8: Orange, - 9: Green/Yellow, - 10: Green/Cyan, - 11: Blue/Cyan, - 12: Blue/Magenta, - 13: Red/Magenta, - 14: Dark grey, - 15: Light grey.

THICK  THICK = _INTEGER (Read) The PGPLOT line thickness. Can be between 1 and 21. [1]


AXES  AXES = _CHAR (Read) Array of switches to turn on or off the drawing of either of the four box sides. The sides are counted from bottom clockwise: bottom, left, top, right. Any switch can be "0" or "+". E.g. ‘00++’ would switch off the bottom and left axes and switch on the top and right axes. [++++]

TICK  TICK = _CHAR (Read) Array of switches to turn on or off the drawing of ticks along either axis. Ticks are drawn only if the corresponding axis is also drawn. The sides are counted from bottom clockwise: bottom, left, top, right. Any switch can be "0", "+" or "-". E.g. ‘00+-’ would switch off the bottom and left ticks and switch on the top and right ticks. The top axis would have ticks outward, the right axis would have ticks inward. Note that with the current plot software, ticks are drawn only if the axis is drawn as well. [‘—’]
NUML  = _CHAR (Read) Array of switches to turn on or off the drawing of numeric labels along either axis. The sides are counted from bottom clockwise: bottom, left, top, right. Any switch can be "0" or "+"; the second and fourth switch can also be ".". E.g. '0+0-' would switch off the bottom and top labels and switch on the left and right labels. The left axis would have labels horizontal (orthogonal), the right axis would have labels vertical (parallel). ['++00']

TEXT  = _CHAR (Read) Array of switches to turn on or off the drawing of text labels along either axis. The sides are counted from bottom clockwise: bottom, left, top, right. Any switch can be "0" or "+". E.g. '0++0' would switch off the bottom and right labels and switch on the left and top labels. ['++++']

MAJOR  = _REAL (Read) The distance in world coordinates between major tick marks. The first element is for the horizontal direction, the second for the vertical direction. This is also the distance along the axis between numeric labels. Values of 0 cause PGPLOT to choose the major tick interval automatically. [0.,0.]

MINOR  = _INTEGER (Read) The number of minor tick intervals per major tick interval. The first element is for the horizontal direction, the second for the vertical direction. Values of 0 for MINOR or MAJOR cause PGPLOT to choose the minor tick interval automatically. [0,0]

BOTTOM = _CHAR (Read) The text label for the bottom axis. Give null to construct the label from the input NDF axis label and unit. Within the string, you can use the following escape sequences:

- \fn Normal (single stroke) font,
- \fr Roman font,
- \fi Italic font,
- \fs Script font,
- \u Superscript (use only paired with \d),
- \d Subscript (use only paired with \u),
- \b Backspace,
- \\Backslash,
- \A Danish umlaut (Angstroem),
- \g Any greek letter.

LEFT  = _CHAR (Read) The text label for the left axis. Give null to construct the label from the input NDF axis label and unit. Within the string, you can use the following escape sequences:

- \fn Normal (single stroke) font,
- \fr Roman font,
- \fi Italic font,
- \fs Script font,
- \u Superscript (use only paired with \d),
- \d Subscript (use only paired with \u),
- \b Backspace,
- \\Backslash,
- \A Danish umlaut (Angstroem),
- \g Any greek letter.
TOP  TOP = _CHAR (Read) The text label for the top axis. Give null to use the title from
the input NDF. Within the string, you can use the following escape sequences:

- \fn Normal (single stroke) font,
- \fr Roman font,
- \fi Italic font,
- \fs Script font,
- \u Superscript (use only paired with \d),
- \d Subscript (use only paired with \u),
- \b Backspace,
- \ Backslash,
- \A Danish umlaut (Angstroem),
- \g Any greek letter.

RIGHT  RIGHT = _CHAR (Read) The text label for the right axis. Give null to construct
the label from the input NDF label and unit. Within the string, you can use the
following escape sequences:

- \fn Normal (single stroke) font,
- \fr Roman font,
- \fi Italic font,
- \fs Script font,
- \u Superscript (use only paired with \d),
- \d Subscript (use only paired with \u),
- \b Backspace,
- \ Backslash,
- \A Danish umlaut (Angstroem),
- \g Any greek letter.

LABSPC  LABSPC( 4 ) = _REAL (Read) If the plot is not an overlay then this specifies the
space left at the bottom, left, top, and right between the plot window and the current
(AGI) picture. The space is measured as a fraction of the current picture. Ticks and
numeric labels are drawn outward from the plot window, but text labels are drawn
inward from the current picture. The labelling space must be between zero and 0.5.

[0.1,0.1,0.05,0.05]

WORLD  WORLD( 4 ) = _REAL (Read) The world coordinates of the plot window. Give
null to use the extent of the input NDF instead. The four elements are the bounds on
the left, right, bottom, and top in that order. Left and right bound must not be equal,
neither must bottom and top. [!]

START  START = _REAL (Read) The first contour level.

STEP  STEP = _REAL (Read) The step between successive major contour levels. If zero is
given then only one contour at value START will be drawn. STEP can be negative.
In addition to the major contours, three minor contours will be drawn between
successive major contours. The major contours are thick and labelled with the
contour value. The minor contours are thin and labelled with an arrow pointing
counter-clockwise around a local maximum.

END  END = _REAL (Read) The last contour level. This may be adjusted slightly so as
to comply with START and STEP. If given equal to START then only one contour at
value START will be drawn. END can be smaller than START.
**Source comments:**

The plot can be an overlay over the most recent data picture inside the current picture, or a new plot inside the current picture. (The current picture after SPECCONT is the same as before.)

The screen contents of the current picture can be erased or not. If the plot is not an overlay, then the space to be left for axis labels as well as the exact labelling can be specified.

The labelling consists of axes, axis ticks, numeric labels at the major ticks, and text labels. The axes are counted from bottom clockwise. Each axis can be drawn or not. Each drawn axis can have ticks or not. Each axis can have numeric labels or not. The left and right axes can have either horizontal (orthogonal) or vertical (parallel) numeric labels. Each axis can have a text label or not.

The kind of labelling is controlled by several 4-character strings. Each character is the switch for axis 1, 2, 3, 4 respectively. "0" turns an option off for that axis, "+" turns it on. For the ticks and for numeric labels of axes 2 and 4, "-" is also allowed. It yields inward ticks and vertical numeric labels.

The attributes of the plot can be selected. These are
- colour
- line thickness
- character height
- simple or roman font
- dash pattern

Most parameters default to the last used value.

**Examples:**

```plaintext
specccont cube(,5,) start=2 step=2 end=10 accept
```

This takes the fifth xz-cut from the input cube. It draws contours at values 2, 4, 6, 8, and 10.

**Notes:**

This routine recognises the Specdre Extension v. 0.7.

This routine recognises and uses coordinate transformations in AGI pictures.

**D.217 SPECGRID-Plot spectra on position grid.**

**Usage:**

```plaintext
```

**Description:**

This routine takes an NDF and displays all the spectra (rows) in it. Each spectrum occupies a cell in the plot which is positioned according to its coordinates. The coordinates for each spectrum are normally stored in the Specdre Extension, but for an NDF of at least three dimensions the first two non-spectroscopic axes can be used instead. The plot area is the current (AGI) picture of the graphics device.
Parameters: CLEAR

CLEAR = _LOGICAL (Read) If true the plot area will be cleared before plotting. When plotting to a printer file, set this false. Otherwise the output may be preceded by an empty page. [FALSE]

OVERLAY

OVERLAY = _LOGICAL (Read) If true then the last (AGI) data picture inside the current (AGI) picture is used to define the plot area and its world coordinates. Only that area will be cleared if CLEAR is true. No new labelling of the plot will occur. [FALSE]

IN

IN = NDF (Read) The input NDF. If it does not have explicit components COORD1 and COORD2 in its Specdre Extension, then the NDF must have at least three dimensions (one spectroscopic and two positional axes). In any case, the spectroscopic axis must be the first non-degenerate axis.

DEVICE

DEVICE = GRAPHICS (Read) The graphics display device.

LIN

LIN = _LOGICAL (Read) If true, the spectral data points will be connected by a line-style polygon. [TRUE]

BIN

BIN = _LOGICAL (Read) If true, the spectral data points will be connected by a bin-style (or histogram-style) polygon. [FALSE]

MARK

MARK = _INTEGER (Read) This parameter determines the kind of marker to be drawn at each spectral data point [0]: - 0: No markers drawn, - 1: Diagonal cross, - 2: Asterisk, - 3: Open circle, - 4: Open square, - 5: Filled circle, - 6: Filled square.

ERROR

ERROR = _LOGICAL (Read) If true and variance information available, error bars will be drawn for each spectral data point. [FALSE]

WIDTH

WIDTH = _LOGICAL (Read) If true, the pixel width will be indicated by horizontal bars for each spectral data point. [FALSE]

FRAME

FRAME = _LOGICAL (Read) If true, each spectral cell gets a plain box drawn around it. [TRUE]

FILL

FILL = _LOGICAL (Read) If false then the plot window will be adjusted to give the same plot scale horizontally and vertically. If true, scaling is independent in each direction and the plot will fill the area available. This parameter is used only if OVERLAY is false. [FALSE]

ROMAN

ROMAN = _LOGICAL (Read) If true, PGPLOT’s roman font is used for drawing text. If false, the normal (single-stroke) font is used. [FALSE]

HEIGHT

HEIGHT = _REAL (Read) The height of the characters measured in units of PGPLOT default text height, which is approximately 1/40 of the height of the (AGI) base picture (i.e. 1/40 the height of the work station window, screen or paper). HEIGHT will be used for labelling the plot box (coordinate space). The legend cell labels are always half that size. [1.]

COLOUR

COLOUR = _INTEGER (Read) The PGPLOT colour index to be used for the plot. This can be formally between 0 and 255, but not all devices support all colours. The default colour representation is: - 0: Background, - 1: Foreground (default), - 2: Red, - 3: Green, - 4: Blue, - 5: Cyan, - 6: Magenta, - 7: Yellow, - 8: Orange, - 9: Green/Yellow, - 10: Green/Cyan, - 11: Blue/Cyan, - 12: Blue/Magenta, - 13: Red/Magenta, - 14: Dark grey, - 15: Light grey.

THICK

THICK = _INTEGER (Read) The PGPLOT line thickness. Can be between 1 and 21. [1]
DASH  DASH = _INTEGER (Read) The PGPLOT dash pattern [1]:
  • 1: Full line,
  • 2: Long dash,
  • 3: Dash-dot-dash-dot,
  • 4: Dotted,
  • 5: Dash-dot-dot-dot.

AXES  AXES = _CHAR (Read) Array of switches to turn on or off the drawing of either of the four box sides. The sides are counted from bottom clockwise: bottom, left, top, right. Any switch can be "0" or "+". E.g. '00++' would switch off the bottom and left axes and switch on the top and right axes. ['+++']

TICK  TICK = _CHAR (Read) Array of switches to turn on or off the drawing of ticks along either axis. Ticks are drawn only if the corresponding axis is also drawn. The sides are counted from bottom clockwise: bottom, left, top, right. Any switch can be "0", "+" or ".". E.g. '00+-' would switch off the bottom and left ticks and switch on the top and right ticks. The top axis would have ticks outward, the right axis would have ticks inward.

NUML  NUML = _CHAR (Read) Array of switches to turn on or off the drawing of numeric labels along either axis. The sides are counted from bottom clockwise: bottom, left, top, right. Any switch can be "0" or "+"; the second and fourth switch can also be ".". E.g. '0+0-' would switch off the bottom and top labels and switch on the left and right labels. The left axis would have labels horizontal (orthogonal), the right axis would have labels vertical (parallel). ['++00']

TEXT  TEXT = _CHAR (Read) Array of switches to turn on or off the drawing of text labels along either axis. The sides are counted from bottom clockwise: bottom, left, top, right. Any switch can be "0" or "+". E.g. '0++0' would switch off the bottom and right labels and switch on the left and top labels. ['++++']

MAJOR  MAJOR = _REAL (Read) The distance in world coordinates between major tick marks. The first element is for the horizontal direction, the second for the vertical direction. This is also the distance along the axis between numeric labels. Values of 0 cause PGPLOT to choose the major tick interval automatically. [0.,0.]

MINOR  MINOR = _INTEGER (Read) The number of minor tick intervals per major tick interval. The first element is for the horizontal direction, the second for the vertical direction. Values of 0 for MINOR or MAJOR cause PGPLOT to choose the minor tick interval automatically. [0,0]

BOTTOM  BOTTOM = _CHAR (Read) The text label for the bottom axis. Give null to construct the label from the input NDF axis label and unit. Within the string, you can use the following escape sequences []:
  • \fn Normal (single stroke) font,
  • \fr Roman font,
  • \fi Italic font,
  • \fs Script font,
  • \u Superscript (use only paired with \d),
  • \d Subscript (use only paired with \u),
  • \b Backspace,
• \\Backslash,
• \A Danish umlaut (Angstroem),
• \g Any greek letter.

**LEFT**  LEFT = _CHAR (Read) The text label for the left axis. Give null to construct the label from the input NDF axis label and unit. Within the string, you can use the following escape sequences [!]:
- \fn Normal (single stroke) font,
- \fr Roman font,
- \fi Italic font,
- \fs Script font,
- \u Superscript (use only paired with \d),
- \d Subscript (use only paired with \u),
- \b Backspace,
- \\Backslash,
- \A Danish umlaut (Angstroem),
- \g Any greek letter.

**TOP**  TOP = _CHAR (Read) The text label for the top axis. Give null to use the title from the input NDF. Within the string, you can use the following escape sequences [!]:
- \fn Normal (single stroke) font,
- \fr Roman font,
- \fi Italic font,
- \fs Script font,
- \u Superscript (use only paired with \d),
- \d Subscript (use only paired with \u),
- \b Backspace,
- \\Backslash,
- \A Danish umlaut (Angstroem),
- \g Any greek letter.

**RIGHT**  RIGHT = _CHAR (Read) The text label for the right axis. Give null to construct the label from the input NDF label and unit. Within the string, you can use the following escape sequences [!]:
- \fn Normal (single stroke) font,
- \fr Roman font,
- \fi Italic font,
- \fs Script font,
- \u Superscript (use only paired with \d),
- \d Subscript (use only paired with \u),
- \b Backspace,
- \\Backslash,
- \A Danish umlaut (Angstroem),
- \g Any greek letter.
LABSPC  LABSPC( 4 ) = _REAL (Read) This is a measure for the distance of the text labels from the coordinate view port. The elements are for the bottom, left, top, and right edge respectively. In the first instance the whole plot is inside the current (AGI) picture and LABSPC specifies the fraction of this view surface to be reserved for labelling. However, if FILL is false, then the view port will shrink further either horizontally or vertically to give equal plot scales. The labelling area will then move inwards as well. The labelling space is measured as a fraction of the current picture. The values must be between zero and 0.5. [0.1,0.1,0.05,0.05]

CWWORLD  CWWORLD( 4 ) = _REAL (Read) The world coordinates of the plot window. Give null to use the coordinate extent of the input NDF instead. The elements are:
- 1: Left bound in coordinate space,
- 2: Right bound in coordinate space,
- 3: Bottom bound in coordinate space,
- 4: Top bound in coordinate space.

Left and right bound must not be equal, neither must bottom and top. [!]

LEGEND  LEGEND( 2 ) = _REAL (Read) The coordinates of the legend cell. Each spectral cell has coordinates according to the NDF’s Specdre Extension (or positional axes). LEGEND is in the same units the position of the empty cell that explains the spectral and intensity coverage of all cells. Give null to avoid the legend cell being drawn. [!] 

CELLSZ  CELLSZ( 2 ) = _REAL (Read) The size of the spectral cells, specified in coordinate units. These must be positive. [1.,1.]

SWORLD  SWORLD( 4 ) = _REAL (Read) The world coordinates within the spectrum cells. Give null to use the spectral extent and data range of the input NDF instead. The elements are:
- 1: Left bound in spectroscopic space,
- 2: Right bound in spectroscopic space,
- 3: Bottom bound in spectroscopic space,
- 4: Top bound in spectroscopic space.

Left and right bound must not be equal, neither must bottom and top. [!]

XLEGEND  XLEGEND = _CHAR (Read) The text label for the bottom axis of the legend cell. Give null to construct the label from the input NDF axis label and unit. Within the string, you can use the following escape sequences [!]:

- \fn Normal (single stroke) font,
- \fr Roman font,
- \fi Italic font,
- \fs Script font,
- \u Superscript (use only paired with \d),
- \d Subscript (use only paired with \u),
- \b Backspace,
- \\ Backslash,
- \A Danish umlaut (Angstroem),
- \g Any greek letter.
YLEGEN\^D YLEGEN\^D D = _CHAR (Read) The text label for the left axis of the legend cell. Give null to construct the label from the input NDF label and unit. Within the string, you can use the following escape sequences [!]:

- \fn Normal (single stroke) font,
- \fr Roman font,
- \fi Italic font,
- \fs Script font,
- \u Superscript (use only paired with \d),
- \d Subscript (use only paired with \u),
- \b Backspace,
- \\ Backslash,
- \A Danish umlaut (Angstroem),
- \g Any greek letter.

Source comments:

The plot can be an overlay in coordinate space to a previous plot, or a new plot. A plot can be combined from spectra in different NDFs, or it can overlay spectra on an image of the same or a similar region in coordinate space.

The previous contents of the plot area can be erased or not. If the plot is not an overlay, then the space to be left for axis labels as well as the exact labelling can be specified.

The labelling (in coordinate space) consists of axes, axis ticks, numeric labels at the major ticks, and text labels. The axes are counted from bottom clockwise. Each axis can be drawn or not. Each drawn axis can have ticks or not. Each axis can have numeric labels or not. The left and right axes can have either horizontal (orthogonal) or vertical (parallel) numeric labels. Each axis can have a text label or not.

The kind of labelling is controlled by several 4-character strings. Each character is the switch for axis 1, 2, 3, 4 respectively. "0" turns an option off for that axis, "+" turns it on. For the ticks and for numeric labels of axes 2 and 4, "-" is also allowed. It yields inward ticks and vertical numeric labels.

Labelling in spectroscopic space (e.g. wavelength-intensity space) is optional. It is in the form of an empty spectrum cell at a specified position in coordinate space. This legend cell is labelled with the ranges in spectroscopic space that is covered by each cell. It also has text labels to indicate what quantities are displayed.

The attributes of the plot can be selected. These are
- colour
- line thickness
- character height
Most parameters default to the last used value.

Examples:

specgrid in accept

Let's assume the given NDF is three-dimensional and has neither a Specdre Extension, nor any axis information. This implies that the first axis is spectroscopic with pixel centres 0.5, 1.5, ..., NX-0.5. The second and third axes will thus be used to position the plots of the individual spectra horizontally and vertically. The positions will also be (0.5,0.5), (1.5,0.5), ..., (NY-0.5,NZ-0.5). Each cell has the default size of 1.0 by 1.0, thus neighbouring pixels in the Y-Z plane of the NDF will be in adjacent cells on the plot. All cells use internally the same scales for the spectroscopic value and the data value. Since these are not specified in parameters, each cell goes from 0.5 to NX-0.5 horizontally and from the minimum data value to the maximum data value vertically. There will be no legend cell, since its position was not given.

Notes: This routine recognises the Specdre Extension v. 1.1.

This routine recognises and uses coordinate transformations in AGI pictures.

D.218 SPEC PLOT-Plot a spectrum.


Description: This routine plots a spectrum (or any one-dimensional NDF section) in the current (AGI) picture of the graphics device.

Parameters: INFO

INFO = _LOGICAL (Read) If false, the routine will issue only error messages and no informational messages. [TRUE]

CLEAR

CLEAR = _LOGICAL (Read) If true, the part of the graphics device corresponding to the current (AGI) picture is erased before the plot is drawn. [FALSE]

OVERLAY

OVERLAY = _LOGICAL (Read) If true, the plot will be an overlay on the most recent (AGI) data picture within the current (AGI) picture. If false, the plot will be user-defined, but the inner box is restricted to the current (AGI) picture.

IN

IN = NDF (Read) The input NDF.

LIN

LIN = _LOGICAL (Read) If true, the data points will be connected by a line-style polygon. [TRUE]

BIN

BIN = _LOGICAL (Read) If true, the data points will be connected by a bin-style (or histogram-style) polygon. [FALSE]

MARK

MARK = _INTEGER (Read) This parameter determines the kind of marker to be drawn at each data point [0]: - 0: No markers drawn, - 1: Diagonal cross, - 2: Asterisk, - 3: Open circle, - 4: Open square, - 5: Filled circle, - 6: Filled square.

ERROR

ERROR = _LOGICAL (Read) If true and variance information available, error bars will be drawn. [FALSE]
WIDTH  WIDTH = _LOGICAL (Read) If true, the pixel width will be indicated by horizontal bars. [FALSE]

ROMAN  ROMAN = _LOGICAL (Read) If true, PGPLOT’s roman font is used for drawing text. If false, the normal (single-stroke) font is used. [FALSE]

HEIGHT  HEIGHT = _REAL (Read) The height of the characters. This also affects the size of the markers. Markers are about half the size of characters. The height is measured in units of PGPLOT default text heights, which is approximately 1/40 of the height of the (AGI) base picture (i.e. 1/40 the height of the workstation window, screen or paper). [1.]

COLOUR  COLOUR = _INTEGER (Read) The PGPLOT colour index to be used for the plot. This can be formally between 0 and 255, but not all devices support all colours. The default colour representation is: - 0: Background, - 1: Foreground (default), - 2: Red, - 3: Green, - 4: Blue, - 5: Cyan, - 6: Magenta, - 7: Yellow, - 8: Orange, - 9: Green/Yellow, - 10: Green/Cyan, - 11: Blue/Cyan, - 12: Blue/Magenta, - 13: Red/Magenta, - 14: Dark grey, - 15: Light grey.

THICK  THICK = _INTEGER (Read) The PGPLOT line thickness. Can be between 1 and 21. [1]


AXES  AXES = _CHAR (Read) Array of switches to turn on or off the drawing of either of the four box sides. The sides are counted from bottom clockwise: bottom, left, top, right. Any switch can be "0" or "+". E.g. ’00++’ would switch off the bottom and left axes and switch on the top and right axes. [’++++’]

TICK  TICK = _CHAR (Read) Array of switches to turn on or off the drawing of ticks along either axis. Ticks are drawn only if the corresponding axis is also drawn. The sides are counted from bottom clockwise: bottom, left, top, right. Any switch can be "0", "+" or "-". E.g. ’00+-+-’ would switch off the bottom and left ticks and switch on the top and right ticks. The top axis would have ticks outward, the right axis would have ticks inward. [’—-’]

NUML  NUML = _CHAR (Read) Array of switches to turn on or off the drawing of numeric labels along either axis. The sides are counted from bottom clockwise: bottom, left, top, right. Any switch can be "0" or "+"; the second and fourth switch can also be "+". E.g. ’00+0’ would switch off the bottom and top labels and switch on the left and right labels. The left axis would have labels horizontal (orthogonal), the right axis would have labels vertical (parallel). [’+00’]

TEXT  TEXT = _CHAR (Read) Array of switches to turn on or off the drawing of text labels along either axis. The sides are counted from bottom clockwise: bottom, left, top, right. Any switch can be "0" or "+". E.g. ’00+0’ would switch off the bottom and right labels and switch on the left and top labels. [’++00’]

NORTHO  NORTHO = _REAL (Read) If orthogonal numeric labels have been selected, you must specify how much space there must be between the axis and the text label, i.e. how long the longest numeric label along the left or right axis will be. The unit is character heights. [1]

MAJOR  MAJOR( 2 ) = _REAL (Read) The distance in world coordinates between major tick marks. The first element is for the horizontal direction, the second for the vertical
direction. This is also the distance along the axis between numeric labels. Values of 0 cause PGPLOT to choose the major tick interval automatically. [0.,0.]

**MINOR** MINOR(2) = _INTEGER (Read) The number of minor tick intervals per major tick interval. The first element is for the horizontal direction, the second for the vertical direction. Values of 0 for MINOR or MAJOR cause PGPLOT to choose the minor tick interval automatically. [0,0]

**BOTTOM** BOTTOM = _CHAR (Read) The text label for the first axis. Within the string, you can use the following escape sequences:

- \fn Normal (single stroke) font,
- \fr Roman font,
- \fi Italic font,
- \fs Script font,
- \u Superscript (use only paired with \d),
- \d Subscript (use only paired with \u),
- \b Backspace,
- \\Backslash,
- \A Danish umlaut (Angstroem),
- \g Any greek letter.

**LEFT** LEFT = _CHAR (Read) The text label for the second axis. Within the string, you can use the following escape sequences:

- \fn Normal (single stroke) font,
- \fr Roman font,
- \fi Italic font,
- \fs Script font,
- \u Superscript (use only paired with \d),
- \d Subscript (use only paired with \u),
- \b Backspace,
- \\Backslash,
- \A Danish umlaut (Angstroem),
- \g Any greek letter.

**TOP** TOP = _CHAR (Read) The text label for the third axis. Within the string, you can use the following escape sequences:

- \fn Normal (single stroke) font,
- \fr Roman font,
- \fi Italic font,
- \fs Script font,
- \u Superscript (use only paired with \d),
- \d Subscript (use only paired with \u),
- \b Backspace,
- \\Backslash,
- \A Danish umlaut (Angstroem),
- \g Any greek letter.
RIGHT  RIGHT = _CHAR (Read) The text label for the fourth axis. Within the string, you can use the following escape sequences:
- \fn Normal (single stroke) font,
- \fr Roman font,
- \fi Italic font,
- \fs Script font,
- \u Superscript (use only paired with \d),
- \d Subscript (use only paired with \u),
- \b Backspace,
- \\Backslash,
- \A Danish umlaut (Angstroem),
- \g Any greek letter.

DEVICE  DEVICE = DEVICE (Read) The graphics device for the plot.

LABSPC  LABSPC( 4 ) = _REAL (Read) The space between outer box (AGI current picture) and inner box measured in units of character heights. The four numbers are for the bottom, left, top, right labelling space in that order. The dynamic default offered is based on the space requirements for the axis labelling, and can in general be accepted.

WORLD  WORLD( 4 ) = _REAL (Read) The world coordinates that the left, right, bottom and top ends of the inner box should correspond to. The dynamic default is based on the coordinates of the first and last pixel of the selected subset and on the extreme data values of the selected subset. Reverse axes can be achieved by giving WORLD(1) > WORLD(2) and/or WORLD(3) > WORLD(4).

Source comments:

Specplot

The plot can basically be an overlay over the most recent data picture inside the current picture, or a new plot inside the current picture. (The current picture after SPECPLT is the same as before.)

The screen contents of the current picture can be erased or not.

The plot location and size is governed by the outer and the inner box. The inner box is the area where data are plotted, the outer box contains the inner box and the plot labels.

In the overlay case the inner box and its world coordinates are identified with the most recent data picture inside the current picture. No labelling is done in the overlay case, so the outer box has no meaning in this case.

In the case of a new plot, the outer box will be identified with the current picture, although the plot labels are allowed to extend beyond this area. Depending on the choice of labelling, a sensible location for the inner box is offered. After the inner box is specified, its world coordinates are enquired. The prompt values correspond to the extreme values found in the data. The location and world coordinates of the inner box are saved as a data picture in the AGI data base.
The labelling consists of axes, axis ticks, numeric labels at the major ticks, and text labels. The axes are counted from bottom clockwise. Each axis can be drawn or not. Each drawn axis can have ticks or not. Each axis can have numeric labels or not. The left and right axes can have either horizontal (orthogonal) or vertical (parallel) numeric labels. Each axis can have a text label or not.

The kind of labelling is controlled by several 4-character strings. Each character is the switch for axis 1, 2, 3, 4 respectively. "0" turns an option off for that axis, "+" turns it on. For the ticks and for numeric labels of axes 2 and 4, "-" is also allowed. It yields inward ticks and vertical numeric labels.

The data can be plotted as a set of markers, as a line-style polygon connecting the data points, or as a bin-style polygon. In addition error bars or pixel width bars can be plotted. Each of the options can be selected independent of the others, i.e. several (or all) options can be selected at the same time. If no variance information is available, error bars are de-selected automatically. Bad data are omitted from the plot. If error bars are selected, bad variances cause the corresponding data also to be omitted.

The attributes of the plot can be selected. These are:
- colour
- line thickness
- character height (equivalent to marker size)
- simple or roman font
- dash pattern for polygon connections

Most parameters default to the last used value.

**Examples:**

specplot spectrum accept
This is the simplest way to plot a 1-D data set in its full length.

specplot imagerow(-100.:50.,15.) accept
This will take a 2-D data set IMAGEROW and plot part of the row specified by the second coordinate being 15. The part of the row plotted corresponds to the first coordinate being between -100 and +50. Note that the decimal point forces use of axis data. Omitting the period would force use of pixel numbers.

specplot imagecol(15.,-100.:50.) accept
This will take a 2-D data set IMAGEROW and plot part of the column specified by the first coordinate being 15. The part of the row plotted corresponds to the second coordinate being between -100 and +50. Note that the decimal point forces use of axis data. Omitting the period would force use of pixel numbers.
specplot spectrum lin=false bin=true accept
   Replace direct connections between data points by bin-style connections.

specplot spectrum mark=1 accept
   Mark each data point by a diagonal cross.

specplot spectrum error=true width=true accept
   Draw an error bar and a pixel width bar for each data point.

specplot spectrum roman=true height=1.5 colour=3 accept
   Draw text with the roman font, draw text and makers 1.5 times their normal size, and plot the whole thing in green colour.

specplot spectrum bottom=Weekday left="Traffic noise [dBA]" accept
   Specify text labels on the command line instead of constructing them from the file's axis and data info.

specplot spectrum overlay=true clear=false accept
   The position and scale of the plot are determined by the previous plot (which might have been produced by a different application).

specplot spectrum world=[0.,1.,-1.,1.] accept
   Use plot limits different from the extreme data values.

Notes: This routine recognises the Specdre Extension v. 0.7.
   This routine recognises and uses coordinate transformations in AGI pictures.

D.219 SPFLUX—Applies a flux calibration spectrum to an observed spectrum

Description: SPFLUX takes a spectrum - ideally a scrunched one - and a calibration spectrum (usually one produced by CSPIKE and then some interpolation routine) and produces a flux calibrated spectrum.

Parameters: SPECTRUM The spectrum to be calibrated. It should be currently in raw counts, preferably scrunched to a linear wavelength scale.

   CALSPECT The calibration spectrum to be applied. Each element of the data should contain the instrumental response, in ‘units’ per count per second per angstrom, where ‘units’ may be any linear unit, but should not be magnitudes. CALSPECT is usually interpolated from a spiketrum generated by CSPIKE.

   OUTPUT The name of the resulting calibrated spectrum.

Source comments: LUX

Applies a flux calibration spectrum (typically generated by the sequence CSPIKE, INTERP) to a spectrum, generating a flux calibrated spectrum.

Command parameters -
D.220  SPIED-Interactive spiketrum editor

Description: SPIED allows the user to edit a spiketrum - deleting points and adding new points - in order to control the spectrum obtained by interpolating between the points (e.g. by INTERP or SPIFIT).

Parameters: SPIKETRUM The spiketrum to be edited (probably generated either by CSPIKE or by GSPIKE). For more details see HELP FIGARO TECHNIQUES SPIKETRA.

OUTPUT The name of the resulting edited spiketrum. It may be the same as SPIKETRUM, in which case the result will replace the input in the same file.

COLOUR The colour for the data to be plotted in. The axes are always plotted in white. The colours allowed are Blue, White, Red, Green, Black, Cyan, Magenta, Yellow. This only works on a colour display, of course. Fitted data will be plotted in the complementary colour

HIGH The maximum Y value for the plot.

LOW The minimum Y value for the plot.

XSTART The maximum X value for the plot.

XEND The minimum X value for the plot.

ORDER The order for the polynomial fit.

QUIT Used to confirm quitting from application.

Source comments: E D

SPIketrum EDitor. Allows the values of a spiketrum to be edited interactively, until the user is satisfied with the data that results from interpolating between them.

Command Parameters -

SPIKETRUM (Character) The name of the spiketrum to be edited.

OUTPUT (Character) The name of the resulting edited spiketrum.

COLOUR (Character) The colour to be used for the data when it is plotted.

HIGH (Real) The maximum Y value for the plot.

LOW (Real) The minimum Y value for the plot.

XSTART (Real) The maximum X value for the plot.
D.221 SPIFIT-Fits a global polynomial to a spiketrum to form a spectrum

Description: SPIFIT interpolates between the points of a ‘spiketrum’ - an array where most of the data is zero except for a few elements whose values are set - to produce a spectrum. Interpolation is by fitting a global polynomial to the data points.

Parameters: SPIKETRUM The name of the spiketrum - an array where only a few elements are set to the correct values, the other elements all being zero - which is to be used to produce a spectrum by interpolation between those specified elements. SPIKETRUM has probably been produced by GSPIKE from a table of values.
ORDER The order of polynomial to be fitted to the points of the spiketrum.
SPECTRUM The name of the spectrum to be produced by interpolation between the points of the spiketrum.
LOG It doesn’t usually make much difference whether you fit to the log of the data or to the data itself, but if the data is changing very rapidly, particularly at the ends, the fit may be better constrained if the log is used.

Source comments: INTERP / SPIFIT / LINTERP

Interpolates between the points of a 'spiketrum' to generate a spectrum. The INTERP command does this by spline interpolation, the SPIFIT command uses global polynomial fitting, and the LINTERP command uses linear interpolation.

Command parameters -
SPIKETRUM The name of the structure containing the spiketrum
ORDER The order for the global polynomial to be fitted (SPIFIT only).
SPECTRUM The name of the result of the operation. This can be the same as for SPIKE. If not, a new structure is created, with everything but the data a direct copy of the input.

Command keywords -
LOG (INTERP & SPIFIT only) Perform the interpolation on the log of the data

LINEND (INTERP only) Use a linear interpolation for the ends of the data - spline fits can go wild outside the range of the defined points.

User variables used - None

D.222 SPLOT - Plots a spectrum

Description: The SPLOT command will plot a spectrum on the current hard or soft graphics device.

Parameters: SPECTRUM The name of the spectrum to be plotted by SPLOT. It should be a 1-dimensional array.

HARDCOPY If set, the plot is written to the device defined as the current hardcopy device. Generally, this is a disk file which will then have to printed. If HARDCOPY is not set, the plot will go to the current softcopy device. The hard and soft copy devices are specified using the HARD and SOFT commands respectively.

WHOLE If set, the whole of the spectrum is plotted. Otherwise, the limits plotted are determined by the values of XSTART and XEND, which you will be prompted for if they were not specified in the command string.

AUTOSCALE If set, the plot is scaled so that all of the data to be plotted just fits on the display. Otherwise, the scale of the plot is determined by the values of HIGH and LOW, which you will be prompted for if they were not specified in the command string.

XSTART The first X value to be plotted, in the units used by the data - angstroms, for example, if the data is wavelength calibrated. XSTART can be set before the start of the data, if necessary. RESET will set XSTART to the first X value in the data.

XEND The last X value to be plotted, in the units used by the data - angstroms, for example, if the data is wavelength calibrated. XEND can be set after the end of the data, if necessary. RESET will set XEND to the last X value in the data.

HIGH The maximum data value to be plotted - i.e. the top Y axis value for the plot.

LOW The minimum data value to be plotted - i.e. the bottom Y axis value for the plot.

BIAS A bias value applied to the data, usually to bias up a plot which is to be superimposed upon a previous plot of almost identical data values. This makes the comparison easier. BIAS N is essentially equivalent to setting HIGH and LOW down by an amount N, so can result in unexpected axis values if it is not accompanied by AXES=NO.

LABEL The label that will appear at the top of the plot.

ERASE Specifies that the screen is to be erased before the plot is made. Usually ERASE and AXES will not be set when a plot is superimposed on a previous one.
AXES  Specifies that the axes for the plot are to be drawn. These should be omitted if the
plot is being superimposed on a previous one, or sometimes just to save plotting
time.

LINES  Specifies that the plot is to be made as a ‘point-plot’, in a ‘join the dots’ fashion.
The alternative is a histogram plot. LINES will usually be a little faster, but messier,
because the resolution of the plotting device will result in stepped lines.

COLOUR  The colour the data are to be plotted in. The axes are always plotted in white.
The colours allowed are Blue, White, Red, Green, Black, Cyan, Magenta, Yellow.
Using Black will have the effect of erasing anything where the data is plotted. This
only works on the Grinnell.

THICKNESS  Only used for ‘build’ or ‘hard’ plots. It is used to increase the thickness of
the lines plotted in order to increase legibility, particularly on the Versatec. Generally
1 or 3 is reasonable for the Versatec - depending on how well set up it is at the present,
and 1 should be used for other devices.

See also: FIGARO: ESPLOT, IPLOTS, MSPLLOT.
KAPPA: LINPLOT, MLINPLOT.

Source comments:

S P L O T  /  E S P L O T

Produces a plot of a spectrum. The plot is directed
to the device defined by the user variables 'SOFT' and
'HARD', and by the value of the command keyword 'HARDCOPY',
so will appear immediately if these specify a video
device (VT125, Args, etc.). If a hardcopy device
is specified, the file for that device will be produced,
but SPLLOT does not attempt to spool it off for printing.

ESPLLOT is similar to SPLOT, but plots error bars based on the
errors in the data.

Command parameters -

SPECTRUM  The data to be plotted. If this contains X-axis
information, this will be used. If not, the X-axis
will just have to be the numbers from 1 to n.
XSTART  The x-value at which plotting is to start.
XEND  The x-value at which plotting is to end.
      (XSTART and XEND are not required if the
      WHOLE keyword is set.)
HIGH  The maximum value to be used for the plot.
LOW  The minimum value to be used for the plot.
BIAS  A value used to displace the plot - BIAS is
effectively a value added to the data before
it is plotted. (It is implemented as a value
subtracted from both HIGH and LOW.)
      (HIGH, LOW and BIAS are not required if the
      AUTOSCALE keyword is set.)
LABEL  A label for the plot.
COLOUR  The colour for the plot, assuming the display device
supports it. The axes are always white.
THICKNESS The width of the lines used for the plot. This is only used for 'hard' & 'build' plots, and should really be 1 for anything other than a high-resolution device like a Versatec or a laser printer.

Command keywords -

AUTOSCALE The program is to work out the values for HIGH and LOW, using the maximum and minimum values in the data over the specified range.

WHOLE The program is to display all of the spectrum.

HARDCOPY The plot is to produce a hard copy.

AXES Axes will be plotted.

ERASE The screen will be erased before the plot.

LINES The plot is not done as a histogram, but as a 'join the dots' line plot. (Only applies to SPLOT.)

User variables - (">" input, "<" output)

(>) SOFT Specifies the device and type to be used for soft plots. See the SOFT command for more details.

(>) HARD Specifies the device and type to be used for hard plots. See the HARD command for more details.

(<) TVXST is set to the starting x-value for the plot.

(<) TVXEN Is set to the final x-value for the plot.

(<) TVHIGH Is set to the same value as HIGH.

(<) TVLOW Is set to the same value as LOW.

(<) TVFILE Is set to the value of SPECTRUM.

(<) TVCOLOR Is set to the GRPCKG code for the plot colour.

(The TV.. variables are intended for use by cursor routines, and reflect the settings for the last plot made, even if XSTART etc are changed.)

(Other user variables may be set by the command processor, in connection with the parameter values.)

KS / CIT 30th April 1984

D.223 SQRTEERR-Generates an error array as Error = Square Root of (Data/Const)

Description: SQRTEERR produces an error array, given a data array and a constant. Each element of the data array is calculated as the square root of the equivalent element of the data array, after division by a specified constant factor.

Parameters: IMAGE The name of the file containing the data for which an error array is to be calculated. The data can have any dimensions. If there is already an error array, its contents are replaced.

FACTOR Calculating the errors as just the square root of the data is OK only for certain types of data, such as raw data from photon counting detectors. For this type of data, FACTOR should be set to 1.0. Some other types of data, such as raw data from CCD detectors, can be handled if the data value is first divided by a constant factor - the
data number per photon value (or ADU/photon value). Other types of data should not be treated using this routine, in general.

**OUTPUT**  The name of the resulting image. This can be the same as the input image, in which case the error array is added to the original image, or different, in which case a new file containing an error array is created.

Source comments:

Sets an error array for which the value of each element is given by the square root of the corresponding data array element. Optionally, the data array element value may be divided by a specified constant before the square root is taken. The case where the constant is 1 is suitable for data such as photon-counting data is being used. For CCDs and similar devices, the constant should be equal to the ADU/photon value.

Command parameters -

**IMAGE** (Character) The name of the structure containing the image.

**FACTOR** (Numeric) The value of the constant by which the data values are divided before the square root is taken.

**OUTPUT** (Character) The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the error array a direct copy of the input.

KS / AAO 8th Jan 1987

**D.224 SUBSET-Take a subset of a data set.**

**Usage:** subset in out

**Description:** Takes a rectangular subset of a data set. The given data set and the resulting subset may have up to seven axes. Axes that become degenerate by sub-setting - i.e. along which only one pixel is chosen - are deleted from the subset. Thus the subset may have smaller dimensionality than the original.

**Parameters:**

**IN**  IN = NDF (Read) The input file.

**OUT**  OUT = NDF (Read) The output file.

**Examples:**

```
subset in(1.5:2.5,10:12) out
```

This takes the data from IN and writes the subset to OUT. The subset is specified as having 1st axis coordinates between 1.5 and 2.5 and 2nd axis pixel numbers between 10 and 12.

**Notes:** This routine recognises the Specdre Extension v. 0.7.
D.225  **TABLE-List contents of a SPICA memory file**

**Description:** This function produces a list of a SPICA memory file contents on the screen, and in a file. If required, the file may then be printed.

**Parameters:**
- **FILE** The name of the SPICA Memory file from which the records are to be listed. The name of the file should be given in full - e.g. ‘ABCD1D.DAT’. The Memory file can be 1-D or 2-D.
- **PRINT** If set, the table will be printed on the default printer queue. Default is PRINT=NO.
- **DELETE** If set, the table will be deleted at the end or after printing. If DELETE is not set, the table will be saved in the file TABLE.LIS in the default directory. Default is DELETE=YES.

**Source comments:**

This function produces a list of a SPICA memory file contents on the screen, and in a file. If required, the file may then be printed. The command is TABLE

With parameter

FILE name of SPICA memory file

With keywords

PRINT=NO specifies that the list is not to be printed (D)
PRINT=YES that the list is to be printed

DELETE=NO that the listing file is to be kept
DELETE=YES that the listing file is to be deleted (D)

**SUBROUTINES AND FUNCTIONS CALLED:**

ICH_LEN - (F) Looks for position of last non space in a string
INDSORT - Sorts index records (from SPSUBS)
PAR_WRUSER - Writes a string to the terminal
PAR_RDCHAR -
PAR_RDKEY -
GEN_FORTERR -
FIG_SETERR -

**VARIABLES AND PARAMETERS:**

DESREC : Desired record
HEADER : Array holding entry header
I, J : Loop counters
LABEL : Label of memory file entry
LISLU : (P) LU on which listing file is opened
MEMLU : (P) LU on which the memory file is opened
MEMORY : Memory file name
NI : Number of channels in the data
NL : Number of X sections in the data
NUMBER : Index entry number
RLVAL : General REAL variable
STATUS : Status return
STRING : Output string
WSTAT : Dummy status for PAR_WRUSER

Programmer : S. A. Morris       Date : 21-APR-1983
Modified to sort the records Alan Bridger 21-SEP-1983
Modified for FIGARO environment P.W.Hill 16-MAR-1986

D.226 TIPPEX-Modify individual pixel values with cursor

Description: Modifies individual pixel values with cursor.

Parameters: OUTPUT The name of the output file to be created. If this is the same as the displayed spectrum, the data in the displayed spectrum will be modified.

CONFIRM Used to confirm change to data.
QUIT Used to confirm quitting application.

See also: FIGARO [CSET][ICSET][NCSET][KAPPA][CHPIX][FILLBAD][SEGMENT][NOMAGIC][SETMAGIC][ZAPLIN]

Source comments: P E X

Figaro function to modify a displayed spectrum using the cursor.

Command parameters -

OUTPUT (Character) The name of the output file to be created. If this is the same as the displayed spectrum, the data in the displayed spectrum will be modified.

Command keywords -

CONFIRM Used to confirm change to data.
QUIT Used to confirm quitting application.

User variables used - (">" input, "<" output)

(> TVFILE The name of the displayed spectrum
(> TVXST The first displayed x-value
(> TVXEN The last displayed x-value
(> TVHIGH The maximum displayed y-value
(> TVLOW The minimum displayed y-value
(> TVCOLOR The GRPLOT code for the plot colour
(> SOFT The device/type string defining the display device

KS / CIT 25th May 1983
**D.227 TRIMFILE-Creates a copy of an HDS file without unused space**

**Usage:** trimfile in out

**Description:** Certain changes to HDS files may cause them to contain unused space, deleted or temporary structures, etc. This routine will create a new copy of the file which will only contain actually used structures. This is in fact only a call to HDS_COPY.

**Parameters:**
- **IN** The HDS file suspected to contain a lot of garbage. The default extension is .sdf. For other extensions use the @-sign and double quotes as in @"file.dst".
- **OUT** The name of the new, clean copy of the HDS file. The default extension is .sdf. For other extensions just specify them as in file2.dst or "file2.dst", but not @"file2.dst".

**Authors:**
- KS: Keith Shortridge (AAO)
- HME: Horst Meyerdierks (UoE, Starlink)

**D.228 VACHEL-Air to vacuum, and/or recession velocity wavelength conversion**

**Description:** VACHEL provides facilities for conversion of the X-array (wave-length) from air to vacuum values, and/or correction of the wavelengths for the effects of a Doppler shift (e.g. to correct from observed to heliocentric velocity frame).

**Parameters:**
- **IMAGE** The name of the image (or spectrum) whose X values are to be converted. IMAGE must already have a valid X-array.
- **OUTPUT** The name of the resulting image (or spectrum). If this is the same as IMAGE, the original file will be modified. Otherwise, a new file will be created.
- **VEL** The recession velocity in Km/s whose effects will be removed by this routine. Use a negative VELOCITY to correct for a velocity of approach.
- **VAC** If VAC is set, the input wavelengths will be assumed to be in air, and will be converted to vacuum values. The input wavelengths must be in Angstroms for this to give the correct result. The resulting wavelength scale will be (slightly) non-linear due to the wavelength dependence of the refractive index of air. Therefore it is generally advisable to rescrunch after use of this correction.
- **DPLR** If DPLR is set, the input wavelengths will be corrected for the effects of a recession velocity of VEL Km/S.

**Source comments:**

VACHEL alters the wavelength (X) structure, to convert it from air to vacuum wavelengths, and to correct for a recession velocity (user supplied value). Usually (but not necessarily) the velocity will be the recession velocity required to correct from observed wavelengths to heliocentric. The velocity correction uses the relativistic formula.

Either or both of the vacuum and recession corrections may be applied in the one run.

The resulting X structure will be (slightly) non-linear, due to the wavelength dependence of the refractive index of air. It will therefore generally be advisable to rescrunch after use of
this routine. An exception is if only a recession correction is made: this leaves a linear scale (if the input is linear).

The input wavelength (X) data must be in Angstroms for the air to vacuum conversion to work correctly.

Command parameters -

**IMAGE** (Character) The name of the structure containing the image.

**VEL** (Float) Recession velocity for which wavelength is to be corrected

**OUTPUT** (Character) The name of the result of the operation. This can be the same as for **IMAGE**. If not, a new structure is created, with everything but the x-data a direct copy of the input.

Command keywords -

**VAC=YES** Correction from air to vacuum is to be made

**VAC=NO** No correction from air to vacuum is to be made

**DPLR=YES** Correction for (+ or -) recession velocity is to be made

**DPLR=NO** No velocity correction

JGR July 1985

D.229 VIG-Corrects a 2D array for vignetting

**Description:** This routine accepts a sky or flat field IMAGE obtained at the same filter for imaging, or at the same grating angle as a series of spectra which need to be corrected for vignetting in 2D. Two orthogonal one dimensional cuts through the 2D IMAGE are formed and fitted with Chebyshev polynomials to form flattening functions on the respective axes. The correction applied to the data is formed from the product of the terms of each of these two series. Because the correction IMAGES may include unwanted signals, such as sky lines the users can specify regions of the data which may be excluded from the fits. In practice rather actually restructure the template cuts formed in the two directions, by deleteing these data this is achieved by ascribing them a very low weight in the fitting process. At the current time it has been found that a weight of 1E-6 is an effective way of eliminating such points.

**Parameters:**

**IMAGE** IMAGE = FILE (Read) Name of image for input

**OLD** OLD = LOGICAL (Read) old coefficients are to be used for correction

**OUTPUT** OUTPUT = FILE (Write) OUTput Name of output file OUTPUT is the name of the resulting image. If OUTPUT is the same as INPUT the data in the input file will be modified in situ. Otherwise a new file will be created.

**YSTART** YSTART = INTEGER (Read) start value to extract in channel direction The data between the limits ystart and yend is extracted and the resultant spectrum is used to find the vignetting in the channel direction.

**YEND** YEND = INTEGER (Read) end value to extract in channel direction The data between the limits ystart and yend is extracted and the resultant spectrum is used to find the vignetting in the channel direction.
XSTART  XSTART = INTEGER (Read) start value to extract in x-sect direction The data between the limits xstart and xend is extracted and the resultant spectrum is used to find the vignetting in the cross-section direction.

XEND  XEND = INTEGER (Read) end value to extract in x-sect direction The data between the limits xstart and xend is extracted and the resultant spectrum is used to find the vignetting in the cross-section direction.

Source comments: none available

D.230  WDFITS—Writes an image out in the AAO de facto ’Disk FITS’ format

Description:  WDFITS writes a Figaro image out in the AAO de facto standard ’Disk FITS’ format. This is used at the AAO for archiving of data, and is a byte-swapped, 2880 byte record, format.

Parameters: IMAGE  The name of the image to be written out to a disk file in the ’Disk FITS’ format. The data written out will be the main image in the file, and any subsidiary information held in a .FITS structure that is not an array. Not all the information held in a general Figaro file can be written out, since the Figaro format is a superset of that used by FITS.

FILE  The name of the ’Disk FITS’ format file that will be created by WDFITS.

BIT16  Normally, WDFITS writes real data (FLOAT or DOUBLE arrays) as 32 bit integers in order to preserve the maximum amount of precision. However, to conserve space, it is possible to specify BIT16, in which case the data will be written as 16 bit integers. In both cases, a BSCALE a BZERO value will be used to scale the real data to make maximum use of all the 32 or 16 bits.

See also:  FIGARO: RDFITS FITSKEYS
          KAPPA: FITSDIN FITSIN FITSHEAD FITSIMP FITSLIST
          CONVERT: NDF2FITS

Source comments: F I T S

Name:  WDFITS

Function:  Writes a Figaro image out to disk in FITS format.

Description:  Writes a Figaro image out to disk in FITS format. Since the Figaro format is a superset of FITS, not all the information held in the Figaro file can be written to tape, so this routine concentrates on the main data array. This means that it is really only suitable for images with no associated calibration data. If the file has axis structures that contain linear data arrays, these will be converted into the appropriate CDLyn, CRPIXn and CRValn keywords. Non-linear axis
data will be ignored. The only other information written into
the FITS header will be taken from the FITS-specific data
structure, should the structure contain one. Any entries in this
structure that can reasonably be output as header quantities (arrays
of data cannot) will be.

If the end of the tape is reached while the data is being written,
the tape is backspaced to the start of the image data and an
end of tape mark is written. The user is then given the option
of mounting a new tape and repeating the output. Note that the
'standard' FITS recipie for handling end of tape - which allows
images to span tapes - is not followed. In batch mode the user
is not given this option.

This program serves for the command WDFITS, which writes
an image out in the AAO de facto 'Disk Fits' format - i.e. to a
disk file whose 2880 byte records are exactly the same as the
records on a FITS tape would be, except that the data is not
byte-swapped.

Parameters:
IMAGE (Character) The file containing the Figaro data
structure to be written to disk.
FILE (Character) The name of the
disk file to which the data is to be written.

Keywords:  
BIT16 Forces real data to be written out as 16 bit rather
than as 32 bit data.

User variables used:  (">" input, "<" output)

Keith Shortridge, AAO.

D.231  WDIPOSOWrites a file in DIPSO/IUEDR/SPECTRUM format

Description:  WDIPOSO creates a copy of a specified Figaro file in the SPECTRUM format used
by DIPSO.

Parameters:  SPECTRUM The name of the Figaro file to be output in a format DIPSO can read.
It should contain one-dimensional data, wavelength calibrated.
CODE There are three SPECTRUM formats used by DIPSO, and WDIPOSO can handle any
of them. Type 0 is unformatted, and is the most compact and fastest to use, but cannot
be read as a text file. Type 1 is a fixed format text file, and type 2 is a free-format text
file. Unless it is important that a readable text file be produced, type 0 is probably
the best format to use. Type 1 output is easier to read that type 2, but more prone to
rounding errors due to the fixed format.
FILE The name of the DIPSO file to be created. If no extension is specified, .DAT will be
used.

Source comments: I P S O
Wrote a copy of a Figaro file in DIPSO/IUEDR format. The file can be written in any of what IUEDR calls SPECTRUM type 0, type 1 or type 2 format.

Command parameters -

SPECTRUM (Character) The name of the Figaro file to be converted.
CODE (Numeric) The type code for the format used.
FILE (Character) The name of the DIPSO format file to be created.

Command keywords - None

KS / AAO 10th Oct 1986

D.232  
XCADD-Adds a constant to the X data in a file

Description: Adds a constant to the X-axis values of the datafile.

Parameters: IMAGE Image with X data to be added to.
FACTOR Additive constant.
OUTPUT Name of resulting image.

Source comments:

This routine is the main body of ICMULT,ICDIV,ICADD and ICSUB, and of XCMULT, XCDIV, XCADD and XCSUB. The Ixxxx routines operate on the data in an image, the Xxxx routines operate on the data in the X array of the input file. ICMULT multiplies an image by a constant. Since the constant can be less than 1., this function will also divide an image by a constant, but ICDIV saves the caller from having to calculate a reciprocal. ICADD adds a constant to an image and ICSUB subtracts a constant from an image.

Command parameters -

IMAGE (Character) The name of the structure containing the image. Uses main data array, or the x-axis data for the XCxxx routines.

FACTOR (Numeric) The value of the constant factor.

OUTPUT (Character) The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

The command name is used to distinguish between the possible operations.

KS / CIT 12th June 1984
D.233  XCDIV—Divides the X data in a file by a constant

**Description:** Divides the X-axis values of the datafile by a constant.

**Parameters:**
- **IMAGE** Image with X data to be divided.
- **FACTOR** Constant.
- **OUTPUT** Name of resulting image.

**Source comments:**
This routine is the main body of ICMULT, ICDIV, ICADD and ICSUB, and of XCMULT, XCDIV, XCADD and XCSUB. The Ixxxx routines operate on the data in an image, the Xxxxx routines operate on the data in the X array of the input file. ICMULT multiplies an image by a constant. Since the constant can be less than 1., this function will also divide an image by a constant, but ICDIV saves the caller from having to calculate a reciprocal. ICADD adds a constant to an image and ICSUB subtracts a constant from an image.

**Command parameters -**

- **IMAGE** (Character) The name of the structure containing the image. Uses main data array, or the x-axis data for the XCxxx routines.

- **FACTOR** (Numeric) The value of the constant factor.

- **OUTPUT** (Character) The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

The command name is used to distinguish between the possible operations.

KS / CIT 12th June 1984

D.234  XCMULT—Multiplies the X data in a file by a constant

**Description:** Multiplies the X-axis values of the datafile by a constant.

**Parameters:**
- **IMAGE** Image with X data to be multiplied.
- **FACTOR** Multiplicative constant.
- **OUTPUT** Name of resulting image.

**Source comments:**
This routine is the main body of ICMULT, ICDIV, ICADD and ICSUB, and of XCMULT, XCDIV, XCADD and XCSUB. The Ixxxx routines operate on the data in an image, the Xxxxx routines operate on the data in the X array of the input file. ICMULT multiplies an image by a constant. Since the constant can be less than 1., this function will also divide an
image by a constant, but ICDIV saves the caller from having to calculate a reciprocal. ICADD adds a constant to an image and ICSUB subtracts a constant from an image.

Command parameters -

IMAGE (Character) The name of the structure containing the image.
   Uses main data array, or the x-axis data for the XCxxx routines.

FACTOR (Numeric) The value of the constant factor.

OUTPUT (Character) The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

The command name is used to distinguish between the possible operations.

KS / CIT 12th June 1984

D.235 XCOPI-Like XCOPY but interpolates X-data from 2 files

Description: The main use of this task is to set the axis data in a data file to values intermediate between those of two other files. Typically, the first file will be a spectrum, the others will be arcs taken either side of the spectrum. The whole axis structure from the first 'arc' file replaces that from the original file, and then the axis data array is modified to give the required intermediate result.

Parameters: SPECTRUM File requiring X data.
   ARC File containing X data.
   ARC2 Second file containing X data.
   FRACTION Fraction in the relationship:
      X data OUTPUT=ARC+(ARC2-ARC1)*FRACTION.
   OUTPUT Name of resulting file.

Source comments:

The main use of this routine is to set the axis data in a data file to values intermediate between those of two other files. Typically, the first file will be a spectrum, the others will be arcs taken either side of the spectrum. The whole axis structure from the first 'arc' file replaces that from the original file, and then the axis data array is modified to give the required intermediate result.

Command parameters -

SPECTRUM (Character) The file name of the spectrum whose data is to be combined with the axis data from the arcs.
**D.236 XCOPY-Copy X-info (e.g. wavelengths) into a spectrum**

**Description:** Copies the axis information from one datafile (typically an arc) into another (typically some other spectrum). The result is either a modified version of the second datafile, or a new datafile.

**Parameters:**
- **SPECTRUM** File requiring X data.
- **ARC** File containing X data.
- **OUTPUT** Name of resulting file.

**Source comments:**

Copies the axis information from one Figaro data structure (typically an arc) into another data structure (typically some other spectrum). The result is either a modified version of the other data structure, or a new structure.

**Command parameters:**

- **SPECTRUM** (Character) The file name of the spectrum whose data is to be combined with the axis data from the arc.
- **ARC** (Character) The file name of the arc spectrum - i.e. the spectrum whose axis information is to be used.
- **OUTPUT** (Character) The file name for the resulting data file.
structure. If this is the same as SPECTRUM, the
axis data in SPECTRUM will be changed in situ.

Command keywords - None

User variables used - None

KS / CIT 7th June 1983

D.237  XCSUB-Subtracts a constant from the X data in a file

Description: Subtracts a constant value from each X-axis value in a datafile.

Parameters: IMAGE  Image with X data to be subtracted from.
            FACTOR  Constant to be subtracted from data.
            OUTPUT Name of resulting image.

Source comments:

This routine is the main body of ICMULT,ICDIV,ICADD and ICSUB, and of XCMULT, XCDIV, XCADD and XCSUB. The Ixxxx routines operate on the data in an image, the Xxxxx routines operate on the data in the X array of the input file. ICMULT multiplies an image by a constant. Since the constant can be less than 1., this function will also divide an image by a constant, but ICDIV saves the caller from having to calculate a reciprocal. ICADD adds a constant to an image and ICSUB subtracts a constant from an image.

Command parameters -

IMAGE  (Character) The name of the structure containing the image. Uses main data array, or the x-axis data for the XCxxx routines.

FACTOR  (Numeric) The value of the constant factor.

OUTPUT  (Character) The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

The command name is used to distinguish between the possible operations.

KS / CIT 12th June 1984

D.238  XCUR-Uses cursor to delimit part of a spectrum

Description: If a spectrum has been displayed on the current soft display device, XCUR allows the user to delimit a range of it in X using the cursor. The sequence ‘SPLOT parms’, ‘XCUR’, ‘SPLOT will result in the display of the delimited part of the spectrum.
**Source comments:**

Uses the graphics cursor to delimit a range in X, setting the user variables XSTART and XEND accordingly.

Command variables / keywords - None.

User variables - (">" input, "<" output)

(>) SOFT (Character) Device / type for soft plots. See documentation on 'SOFT' command for details.

(>) TVXST (Numeric) X-start value for current plot.

(>) TVXEN (Numeric) X-end value for current plot.

(>) TVLOW (Numeric) Lowest value of current plot.

(>) TVHIGH (Numeric) Highest value of current plot.

(<) XSTART (Numeric) X-start for next plot.

(<) XEND (Numeric) X-end for next plot.

KS / CIT 26th July 1982

**D.239 XTPLANE-Adds XT planes of a data cube to form an image**

**Description:** Given a data cube (strictly a cuboid) XTPLANE extracts a 2-D image from it by adding together a number of planes of constant Y.

**Parameters:**

- **CUBE** The name of a 3-dimensional data cuboid, from which XTPLANE is to extract a 2-D image by adding together contiguous planes of constant Y value, the Y dimension being the second dimension of the data.

- **YSTART** The Y value of the first of the planes to be added together to form the extracted image. If the Y axis of the cube is calibrated, YSTART must be specified in the units in which it is calibrated. Otherwise, the element number (1..N) should be used.

- **YEND** The Y value of the last of the planes to be added together to form the extracted image.

- **IMAGE** The name of the 2-D image to be created by XTPLANE.

**Source comments:**

Adds a number of consecutive planes from a cube to produce a 2-D data object. YTPLANE, XTPLANE and XYPLANE take planes whose X,Y or T values respectively are constant over the extracted plane. (Pedantic note: strictly, a cuboid is meant, rather than a cube.)

(Further pedantic note: the X, Y and T axes mentioned in this routine should really be referred to as AXIS(1), AXIS(2) and AXIS(3) respectively. However, explanations become confusing if this is done, so the references to X, Y and T remain. This does not mean that the data is stored in .X .Y and .T structures - JM.)

Command parameters -
Applications in detail

'CUBE' The name of the cube from which the planes are to be taken.

'XSTART' The X, Y, or T-value of the first plane to be used.

'YSTART' If CUBE has a X, Y or T structure, the data from this is used. If not, the plane numbers are used, starting from 1.

'TSTART' The X, Y, or T-value of the last plane to be used.

'XEOMD' 'YEND' 'TEND' 'IMAGE' The name of the resulting data structure.

Input data -

CUBE - 3-D data-array

Output data -

IMAGE is created with the same structure as CUBE except that the data array will only have 2 dimensions, and if CUBE has a X,Y or T structure, this will be omitted - that is, XPLANE will omit any X structure, YPLANE any Y structure and etc.

KS / AAO 8th Jan 1985

D.240 XTRACT-Average an N-dimensional cube into an (N-M)-dimensional one.

Usage: xtract in colaps out

Description: This routine reduces the number of axes of a data set by averaging pixels along some axes while retaining other axes. A simple and common example is averaging all or a certain range of rows (or columns) of an image to create a single row, e.g. an averaged spectrum from a 2-D slit spectrum. Input pixels with bad or zero variance are treated as bad, i.e. disregarded in the averaging (unless NOVAR is true).

Parameters: INFO INFO = _LOGICAL (Read) If false, the routine will issue only error messages and no informational messages. [YES]

VARUSE VARUSE = _LOGICAL (Read) If false, data variance in the input is ignored and output variance is calculated from the scatter of averaged values. If true, data variance in the input is used to weight mean values and to calculate output variance. [YES]

IN IN = NDF (Read) Input file.

COLAPS COLAPS(7) = _INTEGER (Read) For each axis in IN a 0 indicates that the axis is to be retained in OUT, a 1 indicates that along that axis pixels from IN are to be averaged.

OUT OUT = NDF (Read) Output file, containing the extracted data set.
Examples:  
xtract cube(-30.:30.,1.5:2.5,10:20) [0,0,1] xyplane  
This first takes a subset from the 3-D data cube extending  
from -30 to +30, 1.5 to 2.5, 10 to 20 along the 1st, 2nd, 3rd  
axes respectively. (Coordinates are used along the 1st and 2nd  
axes, pixel indices along the 3rd.) From that sub-cube all the  
x-y-planes are averaged to create a 2-D image.  
(E.g. this averages the channel maps between 10 and 20 into an  
integrated map.)

xtract cube(-30.:30.,1.5:2.5,10:20) [1,1,0] spectrum  
This averages each x-y-plane into a single point of the output  
row. The subset used is the same as above. (E.g. this averages  
the cube of channel maps into a mean spectrum.)

xtract image(-30.:30.,1.5:2.5) [0,1] spectrum info=no varuse=no  
This averages all rows between 1.5 and 2.5 into a spectrum. The  
spectrum extends from -30 to +30. Informational messages are  
suppressed, and data variances in the image are ignored. The  
variances in the spectrum are calculated from the row-to-row  
scatter in each column.

Notes: This routine recognises the Specdre Extension v. 0.7. However, no extraction is per-  
formed on NDFs in the input Specdre Extension. If the spectroscopic axis is retained,  
then the scalar components in the Extension are propagated. If the spectroscopic axis is  
collapsed, the Extension is not propagated at all.

D.241  XYPLANE-Adds XY planes of a data cube to form an image

Description: Given a data cube (strictly a cuboid) XYPLANE extracts a 2-D image from it by  
adding together a number of planes of constant T.

Parameters: CUBE  
The name of a 3-dimensional data cuboid, from which XYPLANE is to  
extract a 2-D image by adding together contiguous planes of constant T value, the T  
dimension being the third dimension of the data.

TSTART  
The T value of the first of the planes to be added together to form the extracted  
image. If the T axis of the cube is calibrated, TSTART must be specified in the units in  
in which it is calibrated. Otherwise, the element number (1..N) should be used.

TEND  
The T value of the last of the planes to be added together to form the extracted  
image.

IMAGE  
The name of the 2-D image to be created by XYPLANE.

Source comments:  
Y T P L A N E ,  X T P L A N E ,  X Y P L A N E  
Adds a number of consecutive planes from a cube to  
produce a 2-D data object. YTPLANE, XTPLANE and XYPLANE  
take planes whose X,Y or T values respectively are  
constant over the extracted plane. (Pedantic note:  
strictly, a cuboid is meant, rather than a cube.)  
(Further pedantic note: the X, Y and T axes  
mentioned in this routine should really be referred to  
as AXIS(1), AXIS(2) and AXIS(3) respectively. However,
explanations become confusing if this is done, so the references to X, Y and T remain. This does not mean that the data is stored in .X .Y and .T structures - JM.)

Command parameters -

'CUBE' The name of the cube from which the planes are to be taken.

'XSTART' The X, Y, or T-value of the first plane to be used.

'YSTART' If CUBE has a X, Y or T structure, the data from this is used. If not, the plane numbers are used, starting from 1.

'TSTART' The X, Y, or T-value of the last plane to be used.

'XEND' The X, Y, or T-value of the last plane to be used.

'YEND'

'TEND'

'IMAGE' The name of the resulting data structure.

Input data -

CUBE - 3-D data-array

Output data -

IMAGE is created with the same structure as CUBE except that the data array will only have 2 dimensions, and if CUBE has a X,Y or T structure, this will be omitted - that is, XPLANE will omit any X structure, YPLANE any Y structure and etc.

KS / AAO 8th Jan 1985

D.242 YCADD-Adds a constant to the Y data in a file

Description: Adds a constant value to each Y-axis value in a datafile.

Parameters: IMAGE Image with Y data to be added to.

FACTOR Additive constant.

OUTPUT Name of resulting image.

Source comments:

This routine is the main body of ICMULT,ICDIV,ICADD and ICSUB, and of XCMULT, XCDIV, XCADD and XCSUB. The Ixxxx routines operate on the data in an image, the Xxxx routines operate on the data in the X array of the input file. ICMULT multiplies an image by a constant. Since the constant can be less than 1., this function will also divide an image by a constant, but ICDIV saves the caller from having to calculate a reciprocal. ICADD adds a constant to an image and ICSUB subtracts a constant from an image.
Command parameters -

**IMAGE** (Character) The name of the structure containing the image. Uses main data array, or the x-axis data for the XCxxx routines.

**FACTOR** (Numeric) The value of the constant factor.

**OUTPUT** (Character) The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

The command name is used to distinguish between the possible operations.

KS / CIT 12th June 1984

---

**D.243 YCDIV-Divides the Y data in a file by a constant**

**Description:** Divides each Y-axis value in a datafile by a constant.

**Parameters:**
- **IMAGE** Image with Y data to be divided.
- **FACTOR** Constant.
- **OUTPUT** Name of resulting image.

**Source comments:**
This routine is the main body of ICMULT, ICDIV, ICADD and ICSUB, and of XCMULT, XCDIV, XCADD and XCSUB. The Ixxxx routines operate on the data in an image, the Xxxx routines operate on the data in the X array of the input file.
ICMULT multiplies an image by a constant. Since the constant can be less than 1., this function will also divide an image by a constant, but ICDIV saves the caller from having to calculate a reciprocal. ICADD adds a constant to an image and ICSUB subtracts a constant from an image.

Command parameters -

**IMAGE** (Character) The name of the structure containing the image. Uses main data array, or the x-axis data for the XCxxx routines.

**FACTOR** (Numeric) The value of the constant factor.

**OUTPUT** (Character) The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

The command name is used to distinguish between the possible operations.

KS / CIT 12th June 1984
D.244 YCMULT-Multiplies the Y data in a file by a constant

Description: Multiplies each Y-axis value in a datafile by a constant.

Parameters: IMAGE Image with Y data to be multiplied.
            FACTOR Multiplicative constant.
            OUTPUT Name of resulting image.

Source comments:

This routine is the main body of ICMULT, ICDIV, ICADD and ICSUB, and of XCMULT, XCDIV, XCADD and XCSUB. The Ixxxx routines operate on the data in an image, the Xxxx routines operate on the data in the X array of the input file. ICMULT multiplies an image by a constant. Since the constant can be less than 1., this function will also divide an image by a constant, but ICDIV saves the caller from having to calculate a reciprocal. ICADD adds a constant to an image and ICSUB subtracts a constant from an image.

Command parameters -

IMAGE (Character) The name of the structure containing the image. Uses main data array, or the X-axis data for the XCxxx routines.

FACTOR (Numeric) The value of the constant factor.

OUTPUT (Character) The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

The command name is used to distinguish between the possible operations.

KS / CIT 12th June 1984

D.245 YCSUB-Subtracts a constant from the Y data in a file

Description: Subtracts a constant value from each Y-axis value in a datafile.

Parameters: IMAGE Image with Y data to be subtracted from.
            FACTOR Constant to be subtracted from data.
            OUTPUT Name of resulting image.

Source comments:

This routine is the main body of ICMULT, ICDIV, ICADD and ICSUB, and of XCMULT, XCDIV, XCADD and XCSUB. The Ixxxx routines operate on the data in an image, the Xxxx routines operate on the data in the X array of the input file. ICMULT multiplies an image by a constant. Since the constant can be less than 1., this function will also divide an
image by a constant, but ICDIV saves the caller from having to calculate a reciprocal. ICADD adds a constant to an image and ICSUB subtracts a constant from an image.

Command parameters -

IMAGE (Character) The name of the structure containing the image. Uses main data array, or the x-axis data for the XCxxx routines.

FACTOR (Numeric) The value of the constant factor.

OUTPUT (Character) The name of the result of the operation. This can be the same as for IMAGE. If not, a new structure is created, with everything but the data a direct copy of the input.

The command name is used to distinguish between the possible operations.

KS / CIT 12th June 1984

D.246 YSTRACT-Adds contiguous columns of an image to form a spectrum

Description: Adds a number of consecutive columns from an image to produce a 1-D data object. (A 'column' is all the pixels with a given X-value.)

Parameters: IMAGE Name of image to extract data from.
XSTART First x-value to be used.
XEND Last x-value to be used.
SPECTRUM Name of spectrum to be generated.

Source comments: YSTRACT

Adds a number of consecutive columns from an image to produce a 1-D data object. (A 'column' is all the pixels with a given AXIS(1) value.)

Command parameters -

'IMAGE' The name of the image from which the rows are to be taken.

'XSTART' The AXIS(1)-value of the first row to be used. If IMAGE has an AXIS(1) structure, the data from this is used. If not, the column numbers are used, starting from 1.

'XEND' The AXIS(1)-value of the last column to be used.

'SPECTRUM' The name of the resulting data.

Input data - an IMAGE
Output data -

SPECTRUM is created with the same structure as IMAGE, except that the data-array will only have one dimension, and if IMAGE has an AXIS(1) structure, this will be omitted. Any AXIS(2) structure will be copied unchanged.

KS / CIT 22nd March 1984

D.247 YTPLANE - Adds contiguous YT planes of a data cube to form an image

Description: Given a data cube (strictly a cuboid) YTPLANE extracts a 2-D image from it by adding together a number of planes of constant X.

Parameters: 

CUBE The name of a 3-dimensional data cuboid, from which YTPLANE is to extract a 2-D image by adding together contiguous planes of constant X value, the X dimension being the first dimension of the data.

XSTART The X value of the first of the planes to be added together to form the extracted image. If the X axis of the cube is calibrated, XSTART must be specified in the units in which it is calibrated. Otherwise, the element number (1..N) should be used.

XEND The X value of the last of the planes to be added together to form the extracted image.

IMAGE The name of the 2-D image to be created by YTPLANE.

Source comments: YTPLANE, XTPLANE, XYPLANE

Adds a number of consecutive planes from a cube to produce a 2-D data object. YTPLANE, XTPLANE and XYPLANE take planes whose X, Y or T values respectively are constant over the extracted plane. (Pedantic note: strictly, a cuboid is meant, rather than a cube.) (Further pedantic note: the X, Y and T axes mentioned in this routine should really be referred to as AXIS(1), AXIS(2) and AXIS(3) respectively. However, explanations become confusing if this is done, so the references to X, Y and T remain. This does not mean that the data is stored in .X .Y and .T structures - JM.)

Command parameters -

'CUBE' The name of the cube from which the planes are to be taken.

'XSTART' The X, Y, or T-value of the first plane to be used.

'YSTART' If CUBE has a X, Y or T structure, the data from this is used. If not, the plane numbers are used, starting from 1.

'TSTART'

'XEND' The X, Y, or T-value of the last plane to be used.

'YEND'

'TEND'
'IMAGE' The name of the resulting data structure.

Input data -

CUBE - 3-D data-array

Output data -

IMAGE is created with the same structure as CUBE except that the data array will only have 2 dimensions, and if CUBE has a X,Y or T structure, this will be omitted - that is, XPLANE will omit any X structure, YPLANE any Y structure and etc.

KS / AAO 8th Jan 1985
E  Flux standards

E.1  Spectrophotometric flux standards converted from SPICA format

Figaro includes a large number of standard files supplied by Roderick Johnstone, IoA Cambridge. This section reproduces the document he supplied with those files and which dates from December 1987. The files are kept in the standard Figaro directory FIGARO_PROG_S (see Section 3.4), which on a normal Starlink system corresponds to directory:

```
/star/etc/figaro
```

The files were prepared on a VAX/VMS system, where file names were in upper-case only. Consequently they are listed in upper-case in the note, but following the normal convention the corresponding Unix file names are lower-case. SPICA was a spectroscopic data reduction system, now long-defunct, which was in some ways a precursor of Figaro.

The files listed contain stellar fluxes in milli-Jansky. AB magnitude files have the same name, but with an A before the ‘.’ as in standard Figaro style. I have tried to conform to the systematic naming used by Figaro so files are called after the star with positive and negative signs replaced by P and M respectively. Note that some files have been included more than once under aliases of the star name.

Numerical values of wavelength, flux, bandwidth and order changeover wavelengths have been checked and are believed to be correct. Some errors in the SPICA tables were found. Where existing Figaro files are correct these have not been duplicated.

Note that there are several different magnitude systems which determine the fluxes presented in these tables. These are defined through different calibrations of the primary standard Vega and use different ways of transferring the magnitude system from Vega to the fainter secondary standards. In general the stars listed here are not actual standards but are just good calibrations of non-variable stars. The user is strongly advised to read the paper from which the numbers come (listed at the top of each file) to gain an idea of both the random and the systematic errors in the fluxes as well as the magnitude system from which the fluxes derived.

I am most grateful to Doreen Oliver at RGO who I believe typed in most of the original SPICA files and to Carolin Crawford who gave invaluable help checking the numbers.

<table>
<thead>
<tr>
<th>Star</th>
<th>File</th>
<th>Coverage</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>40 Eri B</td>
<td>40ERIB.TAB</td>
<td>3320–9880</td>
<td></td>
</tr>
<tr>
<td>AM CVn</td>
<td>HZ29.TAB</td>
<td>3340–8720</td>
<td></td>
</tr>
<tr>
<td>BD+25 3941</td>
<td>BDP253941.TAB</td>
<td>3200–8370</td>
<td></td>
</tr>
<tr>
<td>BD+28 4211</td>
<td>BDP284211.TAB</td>
<td>3200–8370</td>
<td></td>
</tr>
<tr>
<td>BD+33 2642</td>
<td>BDP332642.TAB</td>
<td>3200–8370</td>
<td></td>
</tr>
<tr>
<td>BD+40 4032</td>
<td>BDP404032.TAB</td>
<td>3200–8370</td>
<td></td>
</tr>
<tr>
<td>BD+82 2015</td>
<td>BDP82015.TAB</td>
<td>3200–8370</td>
<td></td>
</tr>
<tr>
<td>EG 5</td>
<td>VMA2.TAB</td>
<td>3260–10640</td>
<td>Used 2nd order values at overlap</td>
</tr>
<tr>
<td>EG 11</td>
<td>L870M2.TAB</td>
<td>3210–7220</td>
<td></td>
</tr>
<tr>
<td>EG 28</td>
<td>LB1240.TAB</td>
<td>3210–7220</td>
<td></td>
</tr>
</tbody>
</table>
EG 29  LB227.TAB  3210-7220
EG 31  HZ2.TAB  3210-7220
EG 33  40ERIB.TAB  3320-9880
EG 39  HZ7.TAB  3210-7220
EG 42  HZ14.TAB  3230-7420
EG 50  HE3.TAB  3340-9440
EG 54  L745M46AA.TAB  3320-10520  Cosmetic change to lambda_double
EG 63  LDS235B.TAB  3320-8920  Cosmetic change to lambda_double
EG 67  SA29M130.TAB  3210-10200
EG 76  AL97OM30.TAB  3180-7040  40A bins in 2nd order
EG 76  BL97OM30.TAB  3320-9880  80A bins in 2nd order
EG 77  TDN573.TAB  3340-8160
EG 79  R627.TAB  3160-10840
EG 86  HZ21.TAB  3340-9660
EG 91  HZ29.TAB  3340-8720
EG 98  HZ43.TAB  3320-10520
EG 99  W485A.TAB  3320-9880
EG 102  GRWP705824.TAB  3210-10200
EG 119  ROSS640.TAB  3320-10520
EG 129  GRWP708247.TAB  3340-9200
EG 133  L1573M31.TAB  3336-10073
EG 139  L930M80.TAB  3320-9240  Cosmetic change to lambda_double and correct value at 4510
EG 144  GRWP738031.TAB  3340-9440
EG 148  L1363M3.TAB  3316-10520
EG 149  L930M80.TAB  3320-9240  Cosmetic change to lambda_double and correct value at 4510
EG 162  L1512M34B.TAB  3210-10040
EG 182  G47M18.TAB  3340-9280
EG 184  GD140.TAB  3160-10560
EG 192  AGD185.TAB  3200-7240  80A bins 2nd order
EG 192  BGD185.TAB  5820-9780  360A bins
EG 247  G191B2B.TAB  3320-10520
Feige 110  F110.TAB  3200-8370  _S: file seems to have wrong bandwidth and lambda_double
Feige 15  F15.TAB  3200-8370
Feige 25  F25.TAB  3200-8370
Feige 34  F34.TAB  3200-8370
Feige 56  F56.TAB  3200-8370
Feige 92  F92.TAB  3200-8370
Feige 98  F98.TAB  3200-8370
G47-18  G47M18.TAB  3340-9280
GD 140  GD140.TAB  3160-10560
GD 185  AGD185.TAB  3200-7240  80A bins 2nd order
GD 185  BGD185.TAB  5820-9780  360A bins
GD-248  GDM248.TAB  3300-10000  Not from SPICA
Grw+70  5824  GRWP705824.TAB  3210-10200
Grw+70  8247  GRWP708247.TAB  3340-9200
Grw+73  8031  GRWP738031.TAB  3340-9440
He 3  HE3.TAB  3340-9440
HZ 14  HZ14.TAB  3230-7420
HZ 15  HZ15.TAB  3200-8370
HZ 2  HZ22.TAB  3210-7220
HZ 21  HZ21.TAB  3340-9660
<table>
<thead>
<tr>
<th>Object</th>
<th>File</th>
<th>Freq Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>HZ 29</td>
<td>HZ29.TAB</td>
<td>3340-8720</td>
</tr>
<tr>
<td>HZ 43</td>
<td>HZ43.TAB</td>
<td>3320-10520</td>
</tr>
<tr>
<td>HZ 44</td>
<td>HZ44.TAB</td>
<td>3316-10520</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Ambiguity in bandwidth at changeover to first order</td>
</tr>
<tr>
<td>HZ 7</td>
<td>HZ7.TAB</td>
<td>3210-7220</td>
</tr>
<tr>
<td>Kopff 27</td>
<td>KOP27.TAB</td>
<td>3200-8370</td>
</tr>
<tr>
<td>L1363-3</td>
<td>L1363M3.TAB</td>
<td>3316-10520</td>
</tr>
<tr>
<td>L1403-49</td>
<td>TON573.TAB</td>
<td>3340-8160</td>
</tr>
<tr>
<td>L1512-34B</td>
<td>L1512M34B.TAB</td>
<td>3210-10040</td>
</tr>
<tr>
<td>L1573-31</td>
<td>L1573M31.TAB</td>
<td>3336-10073</td>
</tr>
<tr>
<td>L745-46A</td>
<td>L745M46AA.TAB</td>
<td>3320-10520</td>
</tr>
<tr>
<td>L870-2</td>
<td>L870M2.TAB</td>
<td>3210-9240</td>
</tr>
<tr>
<td>L930-80</td>
<td>L930M80.TAB</td>
<td>3180-7040</td>
</tr>
<tr>
<td>L970-30</td>
<td>AL970M30.TAB</td>
<td>3180-7040</td>
</tr>
<tr>
<td>L970-30</td>
<td>BL970M30.TAB</td>
<td>3320-9880</td>
</tr>
<tr>
<td>LB1240</td>
<td>LB1240.TAB</td>
<td>3210-7220</td>
</tr>
<tr>
<td>LB227</td>
<td>LB227.TAB</td>
<td>3210-7220</td>
</tr>
<tr>
<td>LDS235B</td>
<td>LDS235B.TAB</td>
<td>3320-8920</td>
</tr>
<tr>
<td>R627</td>
<td>R627.TAB</td>
<td>3160-10840</td>
</tr>
<tr>
<td>Ross 640</td>
<td>ROSS640.TAB</td>
<td>3320-10520</td>
</tr>
<tr>
<td>SA29-130</td>
<td>SA29M130.TAB</td>
<td>3210-10200</td>
</tr>
<tr>
<td>Ton573</td>
<td>TON573.TAB</td>
<td>3340-8160</td>
</tr>
<tr>
<td>VMa 2</td>
<td>VMA2.TAB</td>
<td>3260-10640</td>
</tr>
<tr>
<td>W1346</td>
<td>W1346.TAB</td>
<td>3210-10640</td>
</tr>
<tr>
<td>W485A</td>
<td>W485A.TAB</td>
<td>3320-9880</td>
</tr>
</tbody>
</table>

### E.2 HST and new Oke spectrophotometric standard star flux tables

In 1991 Jeremy Walsh provided two new sets of spectrophotometric flux tables for use with Figaro. One comprised a set of stars observed by Oke, the other a set of HST standard stars. Copies of these data can be retrieved by anonymous ftp. The details are as follows.

```
ftp site:    ftp.roe.ac.uk
directory:   /pub/acd/fluxstandards
files:       README.LIS
              hst.tar.Z
              oke.tar.Z
```

The data files are compressed tar archives. Remember to use ftp in binary mode when retrieving them. Once you have retrieved copies of the files they can be decompressed using Unix command `uncompress`, for example:

```
% uncompress oke.tar.Z
```

Then extract the individual files from the tar archive:
Brief details of the two datasets follow. For details of the processing of these files please refer to Jeremy Walsh at the ST-ECF.

E.2.1 Oke standards

These standards comprise 25 stars observed by Oke in the wavelength range 3200 to 9200 Å. Table 2 gives summary details for these stars.

E.2.2 HST standards

These standards comprise 27 stars selected to be standards for the HST. The basic reference is Turnshek et al., but see also Bohlin. The observations cover the approximate wavelength range 800 to 12,000 Å. Table 3 gives summary details for these stars.

---

8R.C. Bohlin, September 1992, STScI Newsletter, 9, No. 2.
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>G158-100</td>
<td>00 33 54.3</td>
<td>-12 07 57</td>
<td>sdG</td>
<td>14.89</td>
<td>14.82</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>HZ 4</td>
<td>03 55 21.7</td>
<td>+09 47 19</td>
<td>DA4</td>
<td>14.52</td>
<td>10.47</td>
<td>•</td>
</tr>
<tr>
<td>3</td>
<td>G191B2B</td>
<td>05 05 30.6</td>
<td>+52 49 54</td>
<td>DAO</td>
<td>11.78</td>
<td>11.72</td>
<td>•</td>
</tr>
<tr>
<td>4</td>
<td>G193-74</td>
<td>07 53 27.4</td>
<td>+52 29 36</td>
<td>DC</td>
<td>15.70</td>
<td>15.58</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>BD+75 325</td>
<td>08 10 49.3</td>
<td>+74 57 57</td>
<td>O5p</td>
<td>9.54</td>
<td>9.52</td>
<td>•</td>
</tr>
<tr>
<td>6</td>
<td>Feige 34</td>
<td>10 39 36.7</td>
<td>+43 06 10</td>
<td>DO</td>
<td>11.18</td>
<td>11.13</td>
<td>•</td>
</tr>
<tr>
<td>7</td>
<td>HD 93521</td>
<td>10 48 23.5</td>
<td>+37 34 13</td>
<td>O9Vp</td>
<td>7.04</td>
<td>6.96</td>
<td>•</td>
</tr>
<tr>
<td>8</td>
<td>HZ 21</td>
<td>12 13 56.4</td>
<td>+32 56 31</td>
<td>DO2</td>
<td>14.68</td>
<td>14.67</td>
<td>•</td>
</tr>
<tr>
<td>9</td>
<td>Feige 66</td>
<td>12 37 23.6</td>
<td>+25 04 00</td>
<td>sdO</td>
<td>10.50</td>
<td>10.43</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>Feige 67</td>
<td>12 41 51.8</td>
<td>+17 31 20</td>
<td>sdO</td>
<td>11.81</td>
<td>11.78</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>G60-54</td>
<td>13 00 09.5</td>
<td>+03 28 56</td>
<td>DC</td>
<td>15.81</td>
<td>15.73</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>HZ 44</td>
<td>13 23 35.4</td>
<td>+36 08 00</td>
<td>sdO</td>
<td>11.66</td>
<td>11.65</td>
<td>•</td>
</tr>
<tr>
<td>13</td>
<td>GRW+70 5824</td>
<td>13 38 51.8</td>
<td>+70 17 08</td>
<td>DA3</td>
<td>12.77</td>
<td>12.72</td>
<td>•</td>
</tr>
<tr>
<td>14</td>
<td>BD+33 2642</td>
<td>15 51 59.9</td>
<td>+32 56 55</td>
<td>B2IV</td>
<td>10.81</td>
<td>10.74</td>
<td>•</td>
</tr>
<tr>
<td>15</td>
<td>G138-31</td>
<td>16 27 53.6</td>
<td>+09 12 24</td>
<td>DA7</td>
<td>16.14</td>
<td>16.07</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>G24-9</td>
<td>20 13 56.0</td>
<td>+06 42 55</td>
<td>DC</td>
<td>15.72</td>
<td>15.77</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>BD+28 4211</td>
<td>21 51 11.1</td>
<td>+28 51 52</td>
<td>Op</td>
<td>10.51</td>
<td>10.47</td>
<td>•</td>
</tr>
<tr>
<td>18</td>
<td>BD+25 4655</td>
<td>21 59 39.3</td>
<td>+26 25 42</td>
<td>DC</td>
<td>9.65</td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>LTT 9491</td>
<td>23 19 35.0</td>
<td>-71 05 30</td>
<td>DC</td>
<td>14.04</td>
<td>14.06</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>Feige 110</td>
<td>23 19 58.4</td>
<td>-05 09 56</td>
<td>DOp</td>
<td>11.82</td>
<td>11.81</td>
<td>•</td>
</tr>
<tr>
<td>21</td>
<td>GD 248</td>
<td>23 26 06.7</td>
<td>+16 00 21</td>
<td>DC</td>
<td>15.09</td>
<td>15.08</td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>GD 50</td>
<td>03 48 50.1</td>
<td>-00 58 30</td>
<td>DA2</td>
<td>14.06</td>
<td>14.06</td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>SA95-42</td>
<td>03 53 43.7</td>
<td>-00 04 33</td>
<td>DA2</td>
<td>15.61</td>
<td>15.60</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>GD 108</td>
<td>10 00 47.3</td>
<td>-07 33 31</td>
<td>sdB?</td>
<td>13.56</td>
<td>13.57</td>
<td>•</td>
</tr>
<tr>
<td>25</td>
<td>NGC 7293</td>
<td>22 29 38.5</td>
<td>-20 50 13</td>
<td>DC</td>
<td>13.51</td>
<td>13.48</td>
<td>•</td>
</tr>
</tbody>
</table>

Table 2: Summary details for the Oke standard stars, from Table 1 of Oke (1990). A bullet (‘•’) in the final column indicates that the star is also included in the companion set of HST spectrophotometric standards.
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>AG+81 266</td>
<td>09 21 19.1801</td>
<td>+81 43 27.642</td>
<td>12.10</td>
<td>B2</td>
<td></td>
</tr>
<tr>
<td>Alpha Lyr</td>
<td>18 36 56.3364</td>
<td>+38 47 01.291</td>
<td>0.03</td>
<td>A0V</td>
<td></td>
</tr>
<tr>
<td>BD+28 4211</td>
<td>21 51 11.0222</td>
<td>+28 51 50.364</td>
<td>10.54</td>
<td>Op</td>
<td></td>
</tr>
<tr>
<td>BD+33 2642</td>
<td>15 51 59.8855</td>
<td>+32 56 54.324</td>
<td>10.81</td>
<td>B2IVp</td>
<td></td>
</tr>
<tr>
<td>BD+75 325</td>
<td>08 10 49.4903</td>
<td>+74 57 57.929</td>
<td>9.50</td>
<td>O5pvar</td>
<td></td>
</tr>
<tr>
<td>BPM 16274</td>
<td>00 50.1</td>
<td>-52.09</td>
<td>14.16</td>
<td>DA...</td>
<td></td>
</tr>
<tr>
<td>Eta UMA</td>
<td>13 47 32.4377</td>
<td>+49 18 47.754</td>
<td>1.852</td>
<td>B3V</td>
<td></td>
</tr>
<tr>
<td>Feige 110</td>
<td>23 19 58.3977</td>
<td>-05 09 56.207</td>
<td>11.4</td>
<td>DA:</td>
<td></td>
</tr>
<tr>
<td>Feige 34</td>
<td>10 39 36.7400</td>
<td>+43 06 09.252</td>
<td>11.0</td>
<td>DA:</td>
<td></td>
</tr>
<tr>
<td>G 191-B2B</td>
<td>05 05 30.6119</td>
<td>+52 49 51.945</td>
<td>11.79</td>
<td>DAw...</td>
<td></td>
</tr>
<tr>
<td>G 93-48</td>
<td>21 52 25.3835</td>
<td>+02 23 19.556</td>
<td>12.73</td>
<td>DA</td>
<td></td>
</tr>
<tr>
<td>Gamma UMa</td>
<td>11 53 49.8475</td>
<td>+53 41 41.136</td>
<td>2.427</td>
<td>A0V</td>
<td></td>
</tr>
<tr>
<td>GD 108</td>
<td>10 00 47</td>
<td>-07 33.5</td>
<td>13.56</td>
<td>B</td>
<td></td>
</tr>
<tr>
<td>GD 50</td>
<td>03 48 50.10</td>
<td>-00 58 28.5</td>
<td>13.98</td>
<td>DAw...</td>
<td></td>
</tr>
<tr>
<td>GRW+70 5824</td>
<td>13 38 50.4743</td>
<td>+70 17 07.619</td>
<td>12.80</td>
<td>DA:</td>
<td></td>
</tr>
<tr>
<td>HD 49798</td>
<td>06 48 04.6995</td>
<td>-44 18 58.431</td>
<td>8.27</td>
<td>sdO6p</td>
<td></td>
</tr>
<tr>
<td>HD 60753</td>
<td>07 33 27.3188</td>
<td>-50 35 03.319</td>
<td>6.682</td>
<td>B2III</td>
<td></td>
</tr>
<tr>
<td>HD 93521</td>
<td>10 48 23.5112</td>
<td>+37 34 13.090</td>
<td>7.06</td>
<td>O9Vp</td>
<td></td>
</tr>
<tr>
<td>HZ 21</td>
<td>12 13 56.27</td>
<td>+32 56 31.4</td>
<td>14.22</td>
<td>DA:</td>
<td></td>
</tr>
<tr>
<td>HZ 2</td>
<td>04 12 43.49</td>
<td>+11 51 50.6</td>
<td>13.86</td>
<td>DA:</td>
<td></td>
</tr>
<tr>
<td>HZ 44</td>
<td>13 23 35.258</td>
<td>+36 07 59.51</td>
<td>11.3</td>
<td>B2</td>
<td></td>
</tr>
<tr>
<td>HZ 4</td>
<td>03 55 21.77</td>
<td>+09 47 18.4</td>
<td>14.47</td>
<td>DA:</td>
<td></td>
</tr>
<tr>
<td>LB 227</td>
<td>04 09 28.50</td>
<td>+17 07 55.2</td>
<td>15.35</td>
<td>DA</td>
<td></td>
</tr>
<tr>
<td>LDS 749B</td>
<td>21 32 16.1</td>
<td>+00 15 15</td>
<td>14.58</td>
<td>DB</td>
<td></td>
</tr>
<tr>
<td>Mu Col</td>
<td>05 45 59.8950</td>
<td>-32 18 23.166</td>
<td>5.148</td>
<td>O9.5V</td>
<td></td>
</tr>
<tr>
<td>NGC 7293</td>
<td>22 29 48.4</td>
<td>-20 49 26</td>
<td>13.54</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Zeta Cas</td>
<td>00 36 58.2846</td>
<td>+53 53 48.874</td>
<td>3.666</td>
<td>B2IV</td>
<td></td>
</tr>
</tbody>
</table>

Table 3: Summary details for the HST standard stars. The data were assembled from SIMBAD. A bullet (•) in the final column indicates that the star is also included in the companion set of Oke spectrophotometric standards.
F Programming

F.1 Writing your own Figaro-style commands

Portable Figaro 5.1 is not intended as a programming environment. It is not recommended to write software that calls the Figaro object libraries. You should use the Starlink and ADAM facilities instead, namely the NDF data access subroutine library and the ADAM parameter system library.

Also consult Section G.1 on changes between the VMS and Unix versions. The “Figaro Programmers’ Guide” of January 1991 is still largely valid, apart from section 12.

However, there exists a large body of code—in a handful of released software items as well as user-written private applications—that relies on calling Figaro subroutines. For this reason the compiled object libraries are included in the release of Figaro.

Let us assume that you have two private applications ‘myapp1’ and ‘myapp2’ that you want to port from VAX/VMS Figaro to Portable Figaro. As you are aware, each application consists of the Fortran source code ‘myapp1.f’ and the interface source code ‘myapp1.con’. The Fortran code needs virtually no change, but the interface must be completely re-written. You can in the first instance do an automatic conversion of the interface, but this is only possible on VAX/VMS:

```fortran
$! Startup for Figaro software
$!  development
$  figaro
$  figdev
$!  Convert .con into .par
$!  Convert .par into .ifl
$  crepar myapp1
$  par2ifl myapp1
```

Now take the two source files per application—‘myapp1.f’ and ‘myapp1.ifl’ to your Unix machine. Both are ASCII files.

In the Fortran code, you have to change any include statements. The include files should be specified only by their file name stem, without path or name extension. And they should be specified in upper case. The most common include statement then becomes

```
INCLUDE 'DYNAMIC_MEMORY'
```

The interface file ‘myapp1.ifl’ can probably be left alone. However, if the application reads or writes what used to be Figaro user variables, then these need entries in the interface file. For a read-only variable called XXXX use this entry:

```
parameter XXXX
  type  ',_CHAR' or ',_REAL'
  access ',READ'
  vpath  ',GLOBAL'
  ppath  ',GLOBAL'
  default ', or 0.
  association '<-GLOBAL.XXXX'
endparameter
```
and for a write-only variable use

```fortran
parameter XXXX
  type 'CHAR' or 'REAL'
  access 'WRITE'
  vpath 'DEFAULT'
  default ' ' or 0.
  association '->GLOBAL.XXXX'
endparameter
```

It is possible within one application to read through the prompt path with e.g. ‘par_rdary’ and later write into the global association with ‘var_setary’.

```fortran
parameter XXXX
  type 'CHAR' or 'REAL'
  vpath 'PROMPT'
  ppath 'GLOBAL,aaaaa,bbbbbb,...'
  association '<->GLOBAL.XXXX'
  prompt 'yyyyyy'
endparameter
```

More elaborate schemes may prove difficult.

If you are not able to do an automatic conversion of the interface on a VMS machine, then you will have to edit your ‘myapp1.con’ into a suitable ‘myapp1.ifl’. Use the interface files of actual Figaro applications for guidance, and consult the documentation on these interface files.

If you are planning to link your application with the original AAO Figaro DSA and DTA data access libraries then the parameters used by ‘dsa_input’, ‘dsa_output’ etc. should have type ‘_CHAR’ or ‘LITERAL’, just like any parameter obtained with ‘par_rdchar’. Alternatively, if you are planning to use the equivalent Portable Figaro FDA library, then the parameters used to access data should have type ‘NDF’. (See Appendices G.1.7 and G.1.8 for details of FDA, DSA and DTA.)

It is recommended to combine all your private applications into a single monolith, in the same way that all Figaro applications are in fact in three monolithic executables. For this you need an additional Fortran source, the monolith routine. If you call your collection of applications ‘mypack’, then you would need a Fortran source ‘mypack.f’ modelled on ‘/star/figaro/figaro1.f’. You have to change the subroutine statement so that the module is called ‘mypack’ and not ‘figaro1’. And you have to change the big if-else-if block where the applications are called, so that your applications are called when their name is detected as the command.

Now you can go about compiling the Fortran code. In order for the include statements to work, you need a symbolic link from the actual include file to the one used in the source code. Figaro’s public include file is in /star/figaro, and has a lower-case name.

```
% ln -s /star/sources/figaro/dynamic_memory DYNAMIC_MEMORY
% f77 -c mypack.f myapp1.f myapp2.f
```

Next you can link your three new object files with the Figaro and Starlink libraries. Be sure that ‘mypack.o’ comes first.
% alink mypack.o myapp[12].o \\
-L/star/sources/figaro -L/star/lib '/star/sources/figaro/figaro_link'

‘figaro_link’ links your programs with the Portable Figaro FDA data access library. If you need to link with the original AAO DSA and DTA libraries instead then make your own copy of file ‘/star/sources/figaro/figaro_link’ and amend it accordingly.

On Solaris, currently, you also have to add an option -lucb. Be aware that in the command the ‘quotes’ are not quotes (’), but back-quotes (‘). After linking you should be left with an executable called ‘mypack’.

If you try to run this executable you will only get an error message. But then, you want to run your two applications, not the monolith as such. In order to run your applications from the Unix shell what you need are symbolic links from the monolith to each application name:

% ln -s mypack myapp1
% ln -s mypack myapp2

Now you can try to run either application under its name. If the application’s interface file is in the same directory, that will even work. Otherwise you get another error message. You can use the source interface files ‘myapp1.ifl’, but it is better to compile them into binary form:

% compifl myapp1
% compifl myapp2

This gives you files ‘myapp1.ifc’ etc. All you need in your executable system is in one directory:

- the monolithic executable,
- one symbolic link for each application from the monolith to the applications’ names,
- the binary interface file ‘myapp1.ifc’ for each application.

The source code (.f and .ifl) can be moved somewhere else, and the object files (.o) can be removed. You can rename the Fortran code to have file names ending in ‘.f’, as is common on Unix systems. Figaro uses ‘.for’ due to an international agreement, there is no rational reason for this.

The extra bit that Figaro does when you give the ‘figaro’ command is to define an alias for each application.

You can also extend your monolith so that you can run it from ICL instead of the Unix shell. For this you need to concatenate all your application interfaces:

% echo 'monolith mypack' > mypack.ifl
% cat myapp[12].ifl >> mypack.ifl
% echo 'endmonolith' >> mypack.ifl

You can compile this as well with ‘compifl’ and place the interface in the same directory as the monolith. In ICL you will have to define the commands in the same way as the Figaro startup script does:
ICL> define myapp1 /my/dir/mypack
ICL> define myapp2 /my/dir/mypack

There is little support for Figaro as a programming environment. In rare cases you may find that your applications call routines that have not been ported from VMS to Unix. The Figaro object libraries exist to serve Figaro applications, they do not pretend to be complete software items in their own right. (You may be lucky and the 'missing' routine has just been renamed.)

That is why you are encouraged to write new software using Starlink libraries, which are designed and supported as software items independent from specific applications.
G Release notes

G.1 Changes from VAX/VMS Figaro 3.0-9 to Portable Figaro 5.1

The main changes are

- Data access is via the NDF library (by means of Figaro’s FDA library). This allows input NDF sections to be specified in the name of the data set. It also allows a variety of foreign formats to be used, if the CONVERT package is initialised in addition to Figaro.

- Access to DST format is as a foreign format. The FDA library does not contain code to access DST format, it relies on conversion programs in the CONVERT package and on the NDF library invoking these as necessary.

- Access to FITS format is available to any data processing application, since disk-FITS is one of the foreign formats supported by the NDF library in conjunction with the CONVERT package.

- Specific conversion from and to FITS within Figaro is supported only from/to disk files. IEEE floating point FITS (BITPIX -32) is not supported. In view of major changes to the FITS standard, Figaro will not contain FITS readers/writers other than the present ‘rdfits’ and ‘wdfits’. Even these will not be upgraded further. Up-to-date FITS support will be a matter for the KAPPA or CONVERT packages.

- The interface files must be ADAM interfaces (’.ifl’ files).

- The executables are a small number of ‘monoliths’, each responsible for a large number of applications. The monoliths are triggered from the Unix shell or ICL by typing the application name.

- ‘par_batch’ works differently: When the monolith is invoked, it checks the environment variable ‘FIGARO_MODE’. If and only if it exists and its value converted to upper case equals ‘BATCH’, then batch mode is assumed. The application can at any time enquire the mode by calling ‘par_batch’.

- There is little support for Figaro software development.

- The object libraries do not contain all routines from Figaro 3.0.

- An application ‘igcur’ was added to examine an image previously displayed with ‘igrey’ or ‘icont’.

- ‘image’, ‘icur’, ‘colour’ and ‘clean’ were re-written or modified to use a graphics device very similar to the ‘soft’ device. The device is chosen with the command ‘idev’, which is a close relative of ‘soft’ and ‘hard’.

- Image pairs cannot be handled, blinking is not possible. This is primarily because modern displays are less than 8 bit deep.
G.1.1 Documentation

The on-line help is mainly what was the ‘commands.hlb’ library. It includes all the new prologues. Also the hierarchy was changed so that it is in line with the (A)PAR parameter system run-time help.

The general help ‘figaro.hlp’ as in Figaro 3.0 was not ported. Much of the information is not exactly applicable to Portable Figaro. The user guide was identical to that help library. The relevant parts were updated and included in this document. That user guide also made references to further documents that existed only on-line. Those were also reviewed and included in this document.

The on-line help is complemented by the printed version of Starlink User Note 86, and both are combined in the Web version of SUN/86.

The Figaro Programmers’ Guide of January 1991 is largely still valid, apart from section 12. A recipe for building VAX/ICL monoliths from Figaro-ish applications was outlined in section 3.3 of SSN/40. However that is not exactly applicable under Unix; refer also to the section for programmers in this document. And note that Figaro is no longer recommended as a programming environment. New software should use Starlink libraries.

G.1.2 General

The port started out from Standard Figaro version 3.0 as modified and extended by Starlink/UK National Figaro version 3.0-6. These are releases for VAX machines. The DTA library had been ported by Keith Shortridge in early 1992 and had been released in Sun Figaro version 2.4.5 patch 5. The DSA library has been ported in August 1992 by Keith Shortridge with contributions from Horst Meyerdierks. A small number of routines that were in VAX Macro or in VAX Pascal have not been taken from Figaro 3.0. Instead their C or Fortran counterparts in Sun Figaro 2.4.5 were used and reviewed. The C code is now ANSI-compliant and its interface to the Fortran code is portable since it uses the ‘F77’ macros.

Portable Figaro 3.1-0 concentrated on providing as many Figaro 3.0 applications as possible in a portable release and on a reasonable time scale. Version 3.1-1 added the most important missing applications, those for image display etc. Version 3.2 was a complete port of Figaro 3.0 in that is also contained disk-FITS readers and writers. Fringe items like ‘dta2hds’ or ‘cabujy’ are not available. Nor is there a software development environment; programmers have to adapt their software to ‘alink’ and ‘compiff’ themselves.

There is only one executable system similar to the monolithic ICL Figaro 3.0. DCL and Callable Figaro are not available. However, Portable Figaro uses the ADAM parameter system in a much improved way; it is superior to ICL Figaro 3.0. The Figaro libraries are available only as object libraries, no sharable libraries exist.

Portable Figaro does not include the TVPCKG or MTPCKG libraries. It also does not include DSK which affects one mode of operation of a small number of line graphics applications. CNV is discontinued, the small number of calls to it could be re-directed to one new routine ‘dsa_fmtcon’. DYN was merged into DSA, VARS into PAR. With TVPCKG gone, DUT and MEM are not needed either. Since Portable Figaro does not support tape handling, there is only a dummy TIO library. The FIT library is complete, in order to support disk-FITS. NAG_FIX was eliminated by changing the out-of-date calls to NAG. FIG and GEN were ported only to the extent that ported applications called these routines. The JTY (formerly JT) library was brought
into line with the other libraries: the routine names and file names now have prefixes JTY and the files contain in general one routine only.

What remains of the Figaro environment—apart from the object libraries—are some environment variables. These are set in the Starlink startup scripts:

```
FIG_DIR /star/bin/figaro
FIG_HELP /star/help/figaro/figaro
FIGARO_PROG_S /star/etc/figaro
FIGARO_PROG_N /dev/null
FIGARO_FORMATS ndf,dst
```

FIGARO_FORMATS is in fact no longer used by Figaro itself. Applications that are linked against DSA/DTA will be subject to your data format choice in FIGARO_FORMATS.

The usual file search path is the same, only that the directory of the executable is not available:

```
./file
$FIGARO_PROG_U/file
$FIGARO_PROG_L/file
$FIGARO_PROG_N/file
$FIGARO_PROG_S/file
```

The ‘_U’ variable is free for the user to set, ‘_L’ is for the site manager to set. They can of course be left un-set. The concept of a national directory is given up, since it did not work out quite as well as one might have thought. With Portable Figaro being a release from Starlink, there is no longer the need for a separate directory for Starlink’s modifications.

Portable Figaro relies on Starlink infrastructure software to a larger extent than Figaro 3.0 did. In general more reliance on Starlink software results in easier maintenance of the resulting Figaro package. Figaro 3.0 did already use

- GKS: Graphics Kernel System (not a Starlink item, but available from RAL),
- SGS: Simple Graphics System,
- GNS: Graphics workstation Name Service,
- GWM: Graphics Window Manager,
- PGPLOT: Graphics Subroutine Library,
- HDS: Hierarchical Data System.

In Portable Figaro the following were added:

- The alink and compifl commands to link and to compile interfaces files.
- (A)PAR: The ADAM parameter system. Used only in a few A-task applications in ‘appstl’ and in FDA’s dsa_axis_range. But also the fundamental basis of (F)PAR.
• SUBPAR: The lower level of the ADAM parameter system. Used only in FDA’s dsa1_rdnam.

• PRIMDAT: Primitive Numerical Data processing. These are used for type conversions, while trapping errors and using bad values.

• PSX: Posix Fortran Interface. Used only in DSA.

• HELP: Interactive Help System.

• NDF: Extensible N-dimensional Data Format access routines. Via Figaro’s FDA library these perform data access to NDF format and indirectly to a multitude of foreign formats, which include DST.

• FIO: Fortran Input/Output routines. These are used to replace the VAX run-time library routines to get and release Fortran I/O units.

• CHR: Character Handling Routines. These are used only in a few A-task applications in ‘appstl’. But also widely used in FDA.

• ERR, EMS, MSG: Starlink error reporting and messaging. These are used in a few A-task applications in ‘appstl’. (F)PAR and FDA use MSG for messaging. DTA and occasionally DSA use EMS to suppress Starlink error reports. FDA uses ERR to report errors and MSG to issue warning messages.

• AGI: Application Graphics Interface. Some applications that are primarily display functions, save their plot in the AGI graphics data base. Other application packages can then pick up that information. The AGI compliance is achieved by calling new routines ‘fig_pgbeg’ and ‘fig_pgend’ instead of ‘pgbegin’ and ‘pgend’. To remove dependency on AGI, these two routines in the FIG library can be changed back to do only the trivial PGPLOT calls.

• PDA: Public Domain Algorithms’ Library. This is a replacement for the—proprietary—NAG library.

Colour tables are no longer un-formatted 2-byte integer arrays in the range 0 to 255 with 3 records of 256 integers. Instead they are 3-by-N images in NDF format. The values range from 0 to 1 now. All existing colour tables happen to be 3 by 256. Colour tables are still stored in the directory ‘$FIGARO_PROG_S’, they have names ‘<table>_lut.sdf’. Not only the traditional tables from Figaro 3.0 were ported, but also some tables from NDPROGS. In addition, KAPPA’s colour tables have the same format and can be used.

G.1.3 Ported applications

There are two directories with application source code, ‘applic’ for the traditional sources and ‘appstl’ for a small set of ‘pure Starlink’ applications. The latter are written in quite different style; they use only Starlink libraries and no Figaro libraries. In addition, ‘appsub’ is used to absorb any subroutines that were split off the otherwise biggest application source files in ‘applic’ (the sources that exceeded 50 kByte).

Most applications were ported straight from VAX Figaro 3.0, either from the Standard release or the National release. The changes made to these applications are mainly hard-wired file names
or parts thereof, which must be lower case now. Also OPEN, CLOSE, INQUIRE statements sometimes used VAX extensions to Fortran 77. Some bugs were fixed (in addition to fixes already in National Figaro 3.0-6).

Some applications use text files with fixed names for output. This may prevent running them again before the output file is deleted or renamed. The problem was serious for ‘echselect’, which uses such a file for input and subsequently for output. ‘echselect’ will now delete any existing file before writing. This is in effect the same as on VAX, only that VAX/VMS would keep old versions of the file.

A small number of applications have been renamed: ‘fitslist’ becomes ‘fitskeys’, ‘rotate’ becomes ‘irot90’, ‘ndfbad’ becomes ‘q2bad’.

‘soft’, ‘hard’, ‘idev’ use GNS instead of SGS to check the device name. This allows device names to be abbreviated. The new imaging routines replace calls to TVPCKG with calls to PGPLOT. ‘findsp’ and ‘overpf’—which used raw GKS before—have been converted to PGPLOT. Thus all graphics is done with PGPLOT.

‘abline’, ‘echarc’, ‘echfind’, ‘echmask’, ‘echmerge’, ‘fet321’ and ‘figs424’ have been converted to use the DSA library rather than DTA. All of Figaro supports DST and NDF formats.

‘abline’, ‘arc’, ‘centers’, ‘cfit’, ‘clean’, ‘cset’, ‘echarc’, ‘echselect’, ‘findsp’, ‘foto’, ‘gauss’, ‘iplots’, ‘isedit’, ‘msplot’, ‘sdist’, ‘spied’ and ‘tippex’ have been modified to avoid calls to ‘par_q*’ or ‘par_rducer’. Instead of asking with a generic parameter name and a dynamic parameter prompt string, these routines now have additional parameters with static prompt strings. This should not affect the use of these applications, the new parameters are not to be given on the command line anyway, and users are used to the questions that are being asked. However, it is now possible to abort the application when it was not possible before.

‘dvdplot’, ‘hopt’, ‘icont’, ‘igrey’, ‘image’, ‘splot’, and ‘esplot’ are passively AGI compliant. Their PGPLOT view port is saved as an AGI picture so that other applications can pick up information about the plot made.

As of version 5.0, NAG is no longer used. Instead the PDA library is used. The resulting changes to applications are in general minor. An exception is ‘gauss’ where a different fit algorithm was adopted. The new algorithm is the same as in the Specdre package.

The ‘straight port’ traditional Figaro routines are:

```
abconv abline adjoin alasin alasout arc
bclean bfft bsmult ccdlin ccur caldiv
cdist centers cfit clean clip cmplx2i
cmplx2m cmplx2r cmplxadd cmplxconj cmplxdiv cmplxxfilt
cmplxmult cmplxsub coadd combine cosbell cosrej
cset capike
echmerge echselect elmt elsplot errcon espplot
extin extract fet321 ff ffcross fft
figs321 figs322 figs422 figs423 figs424 figaee
figsflux findsp fitskeys flconv foto fscrunch/fwconv gauss growx growx
/ growy
/growyt gepike hard hist hoft i2cmp1x
/iadd iarc icadd icdiv icmult icdiv
/icont icor16 icset icsub idiff idiv
/igrey ilist ilog imult interp iplots
```
Apart from ‘clean’, which could be modified to use PGPLOT instead of TVPCKG, the imaging routines have been re-written. ‘idev’ has been added to select the imaging device. ‘image’ and ‘clean’ now support bad values (or quality). ‘igcur’ is an equivalent to ‘icur’, but uses a previous display on the ‘soft’ device made with ‘igrey’ or ‘icont’. The routines re-written in traditional Figaro style are:

```
colour  icur  idev  igcur  image
```

Most of the HDS file structure manipulation routines have been re-written as ‘Starlink-style’ A-tasks and are stored in the ‘appstl’ directory. ‘crobj’ is replaced by ‘creobj’, ‘let’ by ‘copobj’ and ‘setobj’. This group also contains the on-line help program ‘fighelp’, and the two NDF-fixing routines ‘q2bad’ to merge quality into bad values and ‘goodvar’ to replace bad and negative variances.

These applications not being in the traditional Figaro style and not using (F)PAR, the syntax to specify HDS structures is slightly different. Where traditionally array elements were specified in square brackets [ ], you must now use parentheses ( ). Array shapes and sizes are no longer part of a structure specification, but are passed as separate parameters. Furthermore, data types must be HDS types, e.g. ‘Float’ becomes ‘_REAL’.

The A-task applications in ‘appstl’ are:

```
creobj  copobj  delobj  goodvar  q2bad  renobj
setobj  trimfile
```

‘appstl’ contains not only the A-task routines, but also the subroutines called by these A-tasks. Those have names with the FIG prefix; they are kept here to isolate Starlink-ish routines from Figaro-ish ones.

A number of bugs were fixed in various applications since the first version of Portable Figaro (3.1-0).

**G.1.4 New applications**

New applications are:

```
aperture   extlist   hcross   figinfo   fitset   flag2qual
igconv     medfiltr  qual2flag resample ycadd ycdiv
ycmult     ycsup
```
G.1.5 Un-ported applications

A number of applications could not be ported because tape handling is not supported or DSK is not available. Other un-ported applications have to do with image pairs, since they rely on the display device being 8 bit deep. In addition most format conversions were not ported. Furthermore some applications are meaningless in an environment other than DCL Figaro or Callable Figaro. Finally ‘extlist’ never worked, but is now included as a new application. The un-ported applications are:

args    blink    bplot    contract    cpair    cpos
exam    expand    extlist    figset    find    fits
ierase    ikon    image2    imageps    impair    odist
pair    r4s    rbaz    rcshec    rctio    re coff
recon    rew ind    rj ab    rjkm    rj pl    rjt
rlol    rntyb    rpdm    rp fuel i    rshec    rsit
rspdm    rspica    rtyb    rxmlic    sf find    skip
starin    starout    tape    tapeo    vshow    wais
wifits    wjab    wlol    wpdm    wjt    wspica
wvista    zoom

G.1.6 Parameters and variables: (F)PAR

The Figaro PAR and VAR routines have been merged into one library (F)PAR (retaining the original routine prefixes). The routines translate more or less directly into calls to the ADAM parameter system (A)PAR. They emulate the behaviour of the Figaro 3.0 parameter and variable system as best as possible under these circumstances. The @-feature to repeat the application for an edited sequence of parameter settings is not supported.

Variables are still supported, but formally become global parameters and must be declared as such in the interface files.

In some circumstances VAR routines may prompt for global parameters. Say, the ‘soft’ application has never been used to select a plot device. When an application tries to get the value of the global parameter ‘soft’ it will prompt the user. Also if the ‘prompt’ keyword is specified by the user, variables are just normal parameters and will be prompted for. The inconvenience this causes amounts only to hitting the <Return> key in order to accept the default value.

The application interface files ‘<appli c>.ifl’ derive from the Figaro 3.0 binary parameter files ‘<appl ic>.par’. For Figaro 3.0 this conversion had been done automatically with the utility ‘creifl.exe’. Figaro 3.0 (on VAX only!) does contain the utility ‘par2ifl’ for this conversion. See also Section F.1.

No keywords are used in the interfaces. In ICL Figaro 3.0 a shortest abbreviation had to be used. From version 5.1-3 onwards, parameter names can be abbreviated if the environment variable ADAM_ABBRV is set.

In National Figaro 3.0-6 a small number of parameters in a moderate number of ICL Figaro applications (mostly parameters ‘xstart/xend’) had been changed to become global parameters. The only global parameters in Portable Figaro are ‘image/spectrum’ in all applications, and ‘xstart/xend’ in ‘xcur’ and ‘splot/esplot’.

While in Figaro 3.0 all non-logical parameters were of type ‘LITERAL’, numeric parameters are now of type ‘_REAL’. Portable Figaro itself does not use any double precision parameters,
those would be of type ‘_DOUBLE’. Since Figaro 5.1 parameters related to opening data sets via ‘dsa_input’, ‘dsa_output’ etc. are of type ‘NDF’. Their values are obtained within the FDA library by calls to SUBPAR.

Non-global prompt paths are usually ‘CURRENT,DYNAMIC’. So normally the current value is used, but with the ‘reset’ keyword the dynamic value is used.

All interface files connect to the help library ‘FIG_HELP;’. For all parameters the help key ‘*’ is used, equivalent to ‘<applic> Parameters <param>’.

‘par_wruser’ will now strip off a leading ‘$’ or ‘+’ thus disabling the line feed control. The status argument of ‘par_wruser’ is a returned argument; ‘par_wruser’ works even when called with bad status, but it turns the status into ‘OK’.

Null and abort responses given by the user to (A)PAR are detected as non-OK status by (F)PAR. (F)PAR will set the abort flag in its common block. The ‘normal’ (F)PAR routines ‘par_rdchar’, ‘par_rdkey, par_rval’, ‘par_rvald’ and ‘par_rdlary’ will return without action if the abort flag has been set by a previous call to (F)PAR. Applications should test the abort flag and take appropriate action. Most do.

The ‘abnormal’ (F)PAR routines ‘par_qnum’, ‘par_qstr’, ‘par_quest’ and ‘par_rdluser’ do not handle abort requests. Also these routines use dynamic prompt strings. Both these features make them difficult to use on top of (A)PAR. Use of these routines should be avoided. Portable Figaro does avoid them, though they are still included in the object library.

In the ADAM environment PGPLOT is not allowed to communicate with the user when ‘pgadvance’ is called. Thus ideally any call to ‘pgbegin’ should be followed by a ‘CALL PGASK(.FALSE.)’, at least if the application also calls ‘pgadvance’. Where it is necessary for the application to hold before clearing the previous plot, this should be achieved through the parameter system.

G.1.7 Data access routines: FDA

As of version 5.1, Portable Figaro no longer uses the DSA and DTA libraries for data access, but uses instead the FDA library. This provides an interface to the NDF library. FDA tries to be a simple and direct interface to NDF, but emulates DSA at truly as possible. The routine names and argument lists are the same as in DSA/DTA.

Detailed information on the FDA library is beyond the scope of this document.

G.1.8 Old data access routines: DSA and DTA

The need for the VAX error condition handler has been eliminated by more robust coding of some routines and by using PRIMDAT routines. It is however necessary that applications use ‘UPDATE’ access for existing output data that must be read as well a written, and that they use ‘WRITE’ access when output data do not exist. The latter is important, otherwise applications are liable to be aborted by the operating system.

A problem arose with the use of the string ‘\n’ in the buffered output by calls to ‘dsa_wruser’. The backslash is not in the Fortran character set. In Sun Fortran it is interpreted in a similar way as in Unix or C. ‘dsa_wruser’ should no longer be called with the new-line escape sequence ‘\n’. Instead an extra call to ‘dsa_wrflush’ should be made. In general, applications should call
'par_wruser' instead of 'dsa_wruser'. Where a backslash is needed in source code—e.g. in text strings for PGPLOT—'CHAR(92)' should be used.

'dsa_map_data' would un-flag data even when access was 'WRITE'. In that case the mapped array contains garbage. Comparing it against the bad value results in the occasional invalid operand, and will often encounter 'NaN's. The routine now does not un-flag write-accessed data. 'dsa_map_errors' and 'dsa_map_variance' did not zero-initialise an existing write-accessed array; they do now.

One feature of DST-format axes cannot be accommodated in NDF-format data: In DST each axis exists independent of the others. In an NDF either the whole axis structure array is absent, or each axis up to the dimensionality of the main data exists and has a 'DATA_ARRAY'. (By default that data array contains pixel numbers minus 0.5, Figaro uses by default pixel numbers ignoring origin, i.e. numbers 1,2,3,...)

A check and rectification is performed now when a structure or DSA itself closes down: 'dsa_check_structure' now at the very end makes a check of the '.AXIS' structure if the structure to be checked is an NDF. This is done by a call to 'dsa_check_ndf_axis', which is new. Note that this check is made for all DSA structures of NDF type even if they were opened for input and are write-protected. If the rectification fails a message is issued telling whether the whole axis structure was deleted or the structure remained in contravention of the NDF specification.

The present DTA library derives from the one released in Sun Figaro 2.4.5, patch 5. The folding of file names and appending of version numbers on Unix systems via 'dta_filefold' and 'dta_versname' has been discarded.

There was a bug in 'dta_dlvar' and 'dta_rnvar' regarding their use of 'dta_splitn' and 'dta_locate' when an upper level structure was a cell in a structure array. This has been fixed. dta_cyvar’s job is not quite as easy as it was before structure arrays could be encountered. The routine was completely re-written.

An obscure problem occurred whenever a DSA structure reference name was the same as the corresponding parameter name and the parameter also had a global association. This was tracked down to be a problem of conflicting names of HDS locator groups as used by DTA and (A)PAR. DTA now constructs group names by appending and extra 'G'. Group names must be no longer than 15 characters. This now implies that DTA top level names and DSA reference names must be 14 characters or less.

In version 3.1-1 DTA was converted to use 'DAT__SZLOC' instead of the constant 15. On OSF/1, HDS locators are of length 16.

In version 5.0-0 a new routine dta_dfned had to be introduced to fix the N-D axis handling in DSA.

In version 5.1-3 'dsa_input' and 'dsa_named_input' received additional entry points 'dsa_input_update' and 'dsa_named_input_update' respectively. In FDA the routines are distinct, and the entry points are provided to allow modified applications to be linked against DSA/DTA instead of FDA. For the same reason 'dsa_map_complex' was added to DSA in order to map real and imaginary parts of the data in a single call.

G.1.9 High-level subroutines: FIG

Virtually all source files have been split into single-routine files and all externally called routines have now the FIG prefix in their names. Thus ‘wxyfit, seterr, figx_shift’ have changed name.
Some file names are abbreviated routine names in order to be only 12 characters long (excluding the extension ‘.for’). Only ‘fig_eterp.for’ contains subsidiary routines without FIG prefix and all in one file.

While FIG still calls DSA, DSA no longer calls FIG; FIG need not appear twice in the link command.

‘fig_rebin2d’ uses DSA to get a work space needed to call ‘gen_poly2d’. Thus ‘dsa_open’ must have been called before ‘fig_rebin2d’, and ‘dsa_close’ must be called some time after ‘fig_rebin2d’.

Two routines are added in version 5.0. ‘fig_pgbeg’ and ‘fig_pgend’ can be used to replace ‘pgbegin’ and ‘pgend’. The effect is that the last PGPLOT view port is saved as an AGI picture when ‘fig_pgend’ is called.

In version 5.1-3 ‘fig_convchk’ and ‘fig_dtaerr’ have been removed because of their relationship with DTA. Figaro applications no longer need these routines.

G.1.10 The JT-library: JTY

The files that contained the JT routines have been split into single-routine files. Both routine names and file names have the JTY prefix now. In some cases file names are shorter (12 character plus ‘.for’) than routine names.

Some care had to be taken when changing routine names to ‘jty_*’, because some were functions and implicitly typed ‘REAL’. The prefix would imply them to be ‘INTEGER’. Care must be taken wherever the functions ‘evaleg, getrms, poly2d, rmsfilter’ are used. They must be declared properly when changed to ‘jty_*’.

Many routines used ‘PARAMETER’ statements without parentheses around the assignment. This is a VAX extension to Fortran 77; standard Fortran statements are used now. Some routines might have written to Fortran unit 6; these statements have been replaced with calls to ‘par_wruser’.

G.1.11 Low-level subroutines: GEN

There are some C routines. In general an ANSI-compliant C compiler must be used, the makefile uses ‘$(CC)’. Due to alignment problems on Sun4 work-stations, ‘gen_qdisort.c’ was withdrawn and sorting must take place with an equivalent single-precision vector.

Some C routines come in machine-dependent versions. The problem is not the non-portability of the interface to Fortran code—the interface to Fortran is portable by using F77/CNF—but the machine-dependency of binary data. Disk-FITS support requires byte-swapping on Digital hardware, but no swapping on Sun hardware.

The bug in ‘gen_gconv’ causing problems with an un-initialised work array element in the case of even ‘WIDTH’ argument was fixed. ‘gen_poly2d’ was translated to Fortran. It must now be given an extra work space by the calling routine. The most likely such routine is ‘fig_rebin2d’. ‘gen_errmsg’ is no longer available. A new routine ‘gen_astatb’ is like ‘gen_astat’, but recognises bad values. Another routine ‘gen_getcwd’ can be used on Unix platforms to find out the current working directory.
G.1.12 Other libraries

All GKD routines have been ported. ‘gkd_clear_alpha’, ‘gkd_close’, ‘gkd_init’ are dummy routines. The rest are simple calls to (F)PAR. Calls to GKD should be avoided and (F)PAR be called directly.

‘ich_cf’ and ‘ich_cd’ were modified to avoid copying substrings within one string. ‘ich_dfold’ was included to provide for folding to lower case.

Version 3.1 included a library RTL, which was only an interim measure to satisfy numerous calls to VAX/VMS system routines on other platforms. Such calls have been eliminated from all code, the RTL library is withdrawn.

G.2 Figaro 3.1-1 release

The printed documentation for Portable Figaro 3.1 is Starlink User Note 86. The changes between versions 3.1-0 and 3.1-1 have not been included in SUN/86.

In version 3.1-1 the following imaging commands have been added:

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>IMAGE</td>
<td>Display an image</td>
</tr>
<tr>
<td>ICUR</td>
<td>Inspect an image with cursor</td>
</tr>
<tr>
<td>IDEV</td>
<td>Set the device for image display</td>
</tr>
<tr>
<td>COLOUR</td>
<td>Set colour table for image display</td>
</tr>
<tr>
<td>CLEAN</td>
<td>Interactive patching of bad lines, bad pixels in an image</td>
</tr>
</tbody>
</table>

The following three applications are also new and replace two discontinued ones:

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>COPOBJ</td>
<td>Copy an HDS object (replaces LET)</td>
</tr>
<tr>
<td>CREOBJ</td>
<td>Create a data object or file (replaces CROBJ)</td>
</tr>
<tr>
<td>SETOBJ</td>
<td>Assign value to an HDS primitive (replaces LET)</td>
</tr>
</tbody>
</table>

So, whenever SUN/86 tells you to use ‘igrey’ and/or ‘igcur’ you can use ‘image’ and ‘icur’ instead. But note that you must use the matching cursor routine. ‘igcur’ will not work properly on displays made with ‘image’ and ‘icur’ will not work properly on displays made with igrey or icont.

Colour tables are stored as NDFs of size 3 by N. Their names end in ‘_lut’ now. These are stored in $FIGARO_PROG_S, a list can be produced with ‘ls $FIGARO_PROG_S/*_lut.sdf’. When specifying the table to the ‘colour’ application, don’t include ‘$FIGARO_PROG_S’ or ‘.sdf’. As an example, the STANDARD lookup table would appear in the ‘ls’ command as ‘/star/etc/figaro/standard_lut.sdf’, but to load it into the display, just use the command ‘colour standard_lut’.

idev works similar to soft. It will draw the text ‘PGPLOT imaging’ on the device, and it will reset the colour table to grey. Note that a newly created GWM window has arbitrary colours—often all black. If the display generated with ‘image’ looks odd, try to load a grey scale with ‘colour grey’.

The ‘idev’ and ‘soft’ devices can be the same. A device-erasing line plot will, however, reset the colour table. The rationale of separate devices is that you can set the image display to ‘xw’ once and for all, but set the ‘soft’ device to ‘xw’ today, ‘graph_on’ tomorrow, or ‘x2w’ next week.
You can have two separate GWM windows ‘xwindows’ (device ‘xw’) and ‘xwindows2’ (device ‘x2w’) for ‘soft’ plots and image display resp.

GWM windows are usually created with 780x512 pixels and 64 colours, unless your .Xdefaults specifies otherwise. 16 colours are reserved for PGPLOT line graphics, the rest is used for image display. It may be sensible to create two windows explicitly and to use them as ‘soft’ and ‘idev’ devices:

```
% xmake xwindows -colour 16 -geom -0-0
% xmake xwindows2 -colour 128 -geom 512x512-0+0
% figaro
% soft xw
% idev x2w
```

G.3 Figaro 3.2 release

- Contrary to versions 3.1-0 and 3.1-1 this is regarded as a complete package. A new version of the printed User Guide, Starlink User Note 86, version 9, is associated with this release.

- Figaro 3.2 is released for Sun4 (SunOS 4.x), Sun4 (Solaris), DECstation (Ulrix) and Alpha AXP (OSF/1). It has been tested on VAX/VMS (ICL).

- As a service to non-Starlink sites interested in Figaro, the required items from the Starlink Software Collection have been identified and can be shipped as a tar file for any Starlink-supported combination of hardware and operating system. A minimum set is about 10 MByte in size, the full set (including Figaro sources, Figaro object libraries and necessary Starlink libraries) is about 35 MByte.

- Figaro 3.2 includes disk-FITS support. There is no support for tape-FITS, but FITS files can be copied from and to tape with Unix commands.

- Some remaining non-DSA applications were converted to use the DSA library for data access. Thus all applications now support DST and NDF data format. The only exception is that the multi-dimensional axis array needed by SCRUNCH—and created by ECHARC, XCOPI or XCOPY—is not yet supported in NDF format.

- A number of applications were converted to no longer use the general parameters LOGICAL_VALUE etc. Users will hardly notice the difference, all these are prompted parameters that cannot be given on the command line. Users may notice that they now can abort these applications at any prompt.

- A number of bugs have been fixed. Most notably, workspaces used internally were not always released after use (by DSA_UNMAP). MEDSKY in particular did often hit the limit of 32 workspace slots and abort.

- FIGHELP will now find out the terminal size (as set with stty) and page accordingly. Its use is simpler and better documented (try "fighelp fighelp"), you can leave FIGHELP at any prompt including the paging prompt.

To start Figaro on Unix, type:
You should ensure you have a directory $HOME/adam. Help on Figaro may be obtained by typing

```
% fighelp figaro
% fighelp news
% fighelp \
```

Further details can also be found in SUN/86.9.

G.3.1 On-line help topic “News 3.2”

The printed documentation for Portable Figaro 3.2 is Starlink User Note 86.

Figaro 3.2 includes disk-FITS support, i.e. the applications

```
RDFITS Read file in AAO de facto 'Disk FITS' format
WDFITS Writes an image out in the AAO de facto 'Disk FITS' format
```

There is no support for tape-FITS, but FITS files can be copied from and to tape with "dd" using a block size of 2880 byte for the tape. On Unix systems each tape tends to have two devices, e.g. "/dev/rst0" and "/dev/nrst0" for rewound and non-rewound. To skip files etc. use the "mt" command, like:

```
% mt -f /dev/nrst1 eom # forward to end of tape
% dd if=file.fits of=/dev/nrst1 obs=2880 # append FITS file to tape
% mt -f /dev/nrst1 rewind # rewind
% mt -f /dev/nrst1 fsf 5 # skip over five files
% dd of=file.fits if=/dev/nrst1 ibs=2880 # copy 6th file from tape
```

This recipe is offered without warranty. Tapes on Unix systems come with all sorts of peculiarities, and you must try how things work on each tape drive individually.

The following applications were converted from using DTA-only data access to DSA-only data access:

```
ABLINE, ECHARC, ECHFIND, ECHMASK, ECHMERGE, FET321, FIGS424
```

Thus all applications now support both data formats. However the NDF format still forbids N-dimensional axis arrays, so from ECHARC / XCOPY to SCRUNCH you still need to use DST format.

The following applications have been changed in the way they use the Figaro parameter system.

```
ABLINE, ARC, CENTERS, CFIT, CLEAN, CSET, ECHARC, ECHSELECT, FINDSP, FOTO, GAUSS, IPLOTS, ISEDIT, MSLOT, SDIST, SPIED, TIPPEX
```

New parameters have been introduced to replace the general parameters LOGICAL_VALUE etc. Users will hardly notice the difference, all these are prompted parameters that cannot be given on the command line. Users may notice that they now can abort these applications at any prompt.
G.4  **Figaro 3.2-3 release**

- Some bugs have been fixed. This involves arc/echarc, copobj, gspike, interp, xcopi.

- Version 3.2-3 is linked with HDS 4.1, which provides some safeguard against corrupting an input file. Some applications cannot operate in situ on input data, but force a new output file. In earlier versions if the same name was given for output as for input, then the input (and output) file would have been corrupted.

- Version 3.2-3 is linked with PAR 2.0, which supports the use of strings "min" and "max" when prompted for a number. The workaround in previous versions to use ridiculously small and big numbers does no longer work.

- There are new applications HCROSS to calculate redshifts and APERTURE for simple photometry.

- Version 3.2-3 runs under ICL. To start up the package give the ICL command

  
  ICL> load $FIG_DIR/figaro

- The monolith has been split into three separate ones in order that Figaro runs under ICL. The single monolith would have an interface too big for the ADAM parameter system to handle. This does not affect the way the user runs commands.

- The C routine gen_qdisort has been removed. Its maintenance has always been a problem due to different alignment rules for Fortran doubles and C doubles on Sun machines. Figaro’s use of the routine could be diverted to its sister routine gen_qfisort.

- Reading IEEE floating point disk-FITS with RDFITS has been disabled. This did not work on Alpha workstations. The problem was in the interface between routines written in Fortran and C, combined with the different type sizes in different C implementations. IEEE floating point FITS data can be read into NDF format with the FITS readers in the KAPPA package.

- The parameter 16BIT of WDFITS has been changed to BIT16. The parameter name beginning with a digit caused problems in the ADAM parameter system.

- The hints for programmers in SUN/86 are slightly out of date:

  - Object libraries are now e.g. /star/figaro/dsa/libdsa.a. Thus each library is in a different directory. For the time being there are symbolic links for all libraries in /star/figaro/lib, so that the old names should still work.

  - The directory /star/figaro/mono is discontinued. The relevant information is now in /star/figaro. Also, the monolith source code is now in /star/figaro/figaro[123].for.

G.5  **Figaro 5.0 release**

Major changes in this release are:

- ICL is preferred command line interface
- HyperText help
- improved DSA library
- the proprietary NAG library is no longer used
- passive AGI compliance
- GKS is no longer called directly, all plotting via PGPLOT
- bug fixes
- new applications
- figaro_link script to assist linking with Figaro libraries

Various changes have been made to a number of applications, some have been changed in more than one respect. Below is a table stating which changes affect which applications.

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
<th>12</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>abline</td>
<td>arc</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>bclean</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>bfft</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>ccurl</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>cdist</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>cfit</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>cosrej</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>cset</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>dvplot</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>2</td>
<td>3</td>
<td>4</td>
<td>5</td>
<td>6</td>
<td>7</td>
<td>8</td>
<td>9</td>
<td>10</td>
<td>11</td>
<td>12</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>echarc</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>echfind</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>2</td>
<td>3</td>
<td>4</td>
<td>5</td>
<td>6</td>
<td>7</td>
<td>8</td>
<td>9</td>
<td>10</td>
<td>11</td>
<td>12</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>esplot</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>exam</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>extlist</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>figinfo</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>findspl</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>flag2qual</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>2</td>
<td>3</td>
<td>4</td>
<td>5</td>
<td>6</td>
<td>7</td>
<td>8</td>
<td>9</td>
<td>10</td>
<td>11</td>
<td>12</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>gauss</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>growx</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>growy</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>growxt</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>growxy</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>growyt</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>hcross</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>hopt</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>2</td>
<td>3</td>
<td>4</td>
<td>5</td>
<td>6</td>
<td>7</td>
<td>8</td>
<td>9</td>
<td>10</td>
<td>11</td>
<td>12</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>i2cmplx</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>2</td>
<td>3</td>
<td>4</td>
<td>5</td>
<td>6</td>
<td>7</td>
<td>8</td>
<td>9</td>
<td>10</td>
<td>11</td>
<td>12</td>
<td>x</td>
</tr>
</tbody>
</table>
(1) new in 5.0

(2) new, adopted from Standalone Portable Figaro 4.2

(3) new version, adopted from Standalone Portable Figaro 4.2

(4) no longer uses NAG

(5) saves last plot in AGI data base for the benefit of KAPPA and other application packages

(6) no longer uses GKS directly
(7) improved behaviour regarding text output files: arc and echarc will now abort if arlines.* cannot be opened for output; iarc will overwrite the *.iar file if it exists

(8) improved storage of image file information

(9) other improvement made for 5.0

(10) output data access corrected from ‘update’ to ‘write’

(11) work space alignment corrected

(12) other bug fix

Figaro 5.0 can be started from ICL simply with the command ‘figaro’. On-line help is available via the ICL command ‘help’ (not ‘fighelp’, which is only for use from the Unix shell). The ICL Figaro test script (demo.icl) has been changed from VMS to Unix.

The complete documentation (Starlink User Note 86 plus the on-line help library) has been reviewed and is available in HyperText format. It can be viewed with World Wide Web browsers like Mosaic. One set of documentation is held centrally at RAL. Starlink sites in all probability have a local set as well. The Mosaic browser can be started with the Figaro command ‘figwww’ so as to enter the Figaro documentation immediately. figwww works from ICL as well as from the Unix shell. Mosaic is started as a background process and the terminal remains free for use as the Figaro command interface.

The improvements to the DSA library that were introduced in Standalone Portable Figaro 4.2, are included in this version (Portable Figaro 5.0). These improvements are

- Figaro data access now prefers to create ‘simple’ NDF format instead of ‘primitive’ NDF format. This results in more efficient data access and better compatibility with applications that use the NDF library itself to handle NDF format data.

- The data access routines now allow N-dimensional axis data in NDF format (namely 2-D wavelengths arrays in échelle arc calibration).

- It is now permissible to have quality information in two distinct forms. Before, such information had to be either as flagged values or as a quality array. Now both forms may exist at the same time in the same data set.

- It is now possible to replace the input file with the output file even when a new file is required. This is usually a problem when the shape of the output data is different from that of the input data. The processing can then not happen in-situ. On VMS you could create an output file of the same name as the input, but with higher version number. On Unix you can now replace the original file with the new one: the output is created as a file with name ‘Temp_File_1.tmp’ or something similar. The temporary file is renamed when the data access is closed down, hence overwriting the input file. Note that if an application aborts or crashes, such temporary files may become permanent and you may have to delete them explicitly.

- The library is more tolerant (than NDF) in recognising whether data are free of flagged values. This should improve backwards compatibility with NDF data created by Figaro 3.0.
Where the version of DSA in Portable Figaro 3.2 was already better than that in Standalone Portable Figaro 4.2, these improvements have been retained. This is the case for the handling of file names and file units, handling of memory work spaces, and avoidance of the—interim—RTL library.

Two bugs introduced in DSA 4.2 have been fixed for version 5.0. One in effect prevented creation of complex data structures, the other allowed a certain form of invalid NDF axis to remain in the case of an N-D axis array.

A minor improvement to the DTA library (from 4.2) is that is now recognises a larger variety of error codes from the underlying HDS library. A new routine dta_dfned had to be introduced (in 5.0) to fix the N-D axis handling in DSA.

Improved versions of several GEN routines were adopted from Standalone Portable Figaro 4.2. A portable implementation for gen_rename was written for Portable Figaro 5.0 (i.e. this release).

Figaro does no longer use the NAG library. Starlink are in the process of gathering Public Domain routines in a new PDA library to replace most or all NAG routines currently used by Starlink applications. Figaro is the first package to be free of NAG. This has positive effects as regards porting to new platforms, but mostly in respect of the software distribution policy.

Due to the use of FFTPACK routines instead of NAG routines, there is no restriction any more on the shape of complex data sets.

‘gauss’ now uses a different fit algorithm for line fits. The new algorithm is copied from the ‘fitgauss’ application in the Specdre package. The change was necessary, in order to avoid use of the NAG library. As a side effect, the fit algorithm is more stable. A limit of 10 line components for a single optimisation is now enforced (NEXT option in the line fit menu). Before, you could enter more lines, but would mess up the application’s internal storage.

The applications ‘dvdplot’, ‘hopt’, ‘icont’, ‘igrey’, ‘image’, ‘splot’, and ‘esplot’ are now passively AGI-compliant. That means that information about the plot is stored in the AGI data base. Other applications (in non-Figaro packages) can then use this information. As an example, you can use the KAPPA command ‘cursor’ to enquire positions in an image or spectrum plot produced with Figaro. There will be one AGI data base for each machine you run graphics applications on. These files will be created either in SAGI_USER if defined, or in $HOME. The file names are agi_<host>.sdf and should be removed occasionally. These files grow ever bigger, while most of the information pertains to plots that are no longer visible.

Similarly, the Figaro-internal storage of the image/spectrum file name is increased to 132 characters. For example, ‘splot’ followed by ‘ccur’ will now work, even when the data files are in a deep level of the directory tree.

‘findsp’ and ‘overpf’ have been re-coded to use PGPLOT rather than GKS (and SGS) directly. Portable Figaro therefore uses only PGPLOT for graphics. Also, ‘findsp’, ‘overpf’, and ‘polext’ no longer use the NAG library, of course.

The application ‘hcross’, introduced in the previous version, now allows the template parameters to be given as redshifts or as radial velocities. Also, if the template and object spectra do not match well, the results will still be reported.

A typo was corrected in the standard star tables for BD +17 4708, both the linear and the AB magnitude table.
The application interfaces as used by the graphical user interface Xadam now include information as to which data structure is input and output. This should facilitate the auto-display in Xadam.

The package is now built by a makefile based on the Mark V Starlink template makefile. As a consequence the object libraries now are all in /star/figaro rather than sub-directories thereof. The directory /star/figaro/lib no longer exists. Similarly, the only include file available is /star/figaro/dynamic_memory, the directory /star/figaro/inc no longer exists.

G.5.1 Figaro 5.0-1 news item

Version 5.0-1 is an update to the previous release of Figaro 5.0-0. Several problems in the new data access library (DSA 4.2) regarding the handling of quality information have been fixed.

The previously recommended work-around of using ‘q2bad‘ whenever a quality array was created, is now obsolete.

The segmentation violation in ‘coadd‘ does no longer occur. And the floating-point exceptions in ‘isubset‘ and ‘coadd‘ also no longer occur. ‘optextract‘—which suffered from the same problem as COADD—works now as well.

Modified versions of ‘flag2qual‘ and ‘qual2flag‘ are used, which no longer cause the warning that the applications might not work correctly. (They did work correctly anyway.)

G.6 Figaro 5.1-3 release

In this version of Figaro the principal change is the use of a new data access library (FDA), but this made minor modifications to several applications necessary. The old libraries DSA and DTA are kept as part of the release, but are unused by Figaro itself.

With the new data access library, the principal data format is NDF. On the other hand, a number of foreign file formats can be used in addition to NDF itself. The old Figaro DST format is one such foreign format, others include FITS, IRAF, and GIF.

In order to use any foreign format, you must start up CONVERT in addition to Figaro. From the Unix shell:

```
% figaro
% convert
```

and from ICL:

```
ICL> figaro
ICL> convert
```

The data format is selected by the user on a per-file basis by specifying a file name extension:

```
ICL> istat myfile
```

will try all known formats in order of priority, to choose a format explicitly:
Currently the extensions recognised, the formats addressed, and the priority list are:

- .sdf NDF format
- .fit FITS format (disk FITS, IBM byte order)
- .dst DST format (also called Figaro format)
- .imh IRAF format
- .hdr GASP format
- .unf unformatted binary files
- .dat unformatted binary files
- .asc ASCII files
- .txt text files
- .gif GIF images
- .tif TIFF images
- .sdf.Z compressed NDF files

In addition to foreign file formats, users can now use NDF sections of data sets. You can use a number of ways to specify a section as shown in this 7-D example:

```
file(3:5,21.3:44.5,17,17.0,3:,:99)
```

(3:5, 1st axis: pixels 3 through 5
21.3:44.5, 2nd axis: pixel centre values 21.3 through 44.5
17, 3rd axis: pixel 17 only
17.0, 4th axis: only the pixel with centre = 17.0
3:, 5th axis: pixels 3 through end of NDF
, 6th axis: all pixels
:99) 7th axis: pixels from start of NDF through to pixel 99

Apart from the availability of foreign file formats and NDF sections, there should be hardly any perceptible change for the user.

Users can abbreviate parameter names on the command line, e.g. instead of ‘istat image=myfile’, ‘istat im=myfile’ can be used. Name abbreviation can be turned on and off with ‘abbrev’ and ‘noabbrev’ respectively.

‘exam’ has been removed, use the Starlink utility ‘hdstrace’ instead.

‘figwww’ has been removed, use ‘showme sun86’ instead.

Bug fixes in applications:

- ‘cdist’ should now abort with an error message if the end of file is encountered before all information has been read.
- ‘echarc’ has separate parameter ‘output’ now for a copy of input with calibrated wavelengths.
- ‘echmask’ now maps the mask data for write instead of update.
- ‘echmerge’ when given two input files, will now actually map the first and second data, before it mapped the first data twice and ignored the second.
• ‘fet321’ now calls dsa_close at the end. Before it called dsa_open a second time.

• ‘ffcross’ now maps the cross correlation for write instead of update.

• ‘fft’ will now simply replace the axis information according to the Fourier transform that is being applied. It will no longer save the old axis information. ‘bfft’ will therefore simply delete all axis information and not restore the pre-fft information. Users can copy axis information from the pre-fft file to the post-bfft file, of course.

• ‘hcross’ will no longer crash when the power 0.0**0.0 is calculated.

• ‘isplot’ now constructs plot labels as ‘LABEL / UNITS’, before it was the other way round.

• ‘wdfits’ now has initial values for variables BLOCK and NOTERM.

• A possible infinite loop in ‘fitskeys’ has been removed.

The source code of ‘echfind’, ‘fet321’, ‘figs322’, ‘figs321’, ‘icur’, ‘gauss’ no longer uses the problematic splitting of string constants by simply breaking the source code line. Proper string concatenation is now used. Minor bugs in fig_hdsdig and fig_pgend were fixed.

The release has been tidied with respect to differences between Unix and VMS versions. All that remains of the VMS version is the link options file ‘linkmono.opt’. ‘rdispo’ and ‘wdispo’ have been formally added. ‘table’ has been formally added and ‘rspica’ and ‘wspica’ have been formally removed. Startup scripts exist only for C shell and Unix ICL, the test script exists only for ICL. The test script has been extended to cover a wider range of applications.

The move to the new data access library prompted:

• Changes to all interface files, where data sets now have type ‘NDF’ instead of ‘LITERAL’. (One parameter was left at ‘LITERAL’ and had its global association removed.) All NDF-type parameters have access information in the interface file, telling whether the data set is input or output data.


• Changes to call sequence with respect to mapping real part and deleting imaginary part in ‘cmplx2r’, ‘cmplx2i’, ‘cmplx2m’.

• Changes to call sequence with respect to mapping data and quality: ‘surfit’ (not available for use due to NAG call).

• Mapping quality with the correct data type in ‘adjoin’, ‘optextract’.

Changes to call sequence with respect to re-shaping data and axis in 'figs422'.

Use of new routine 'dsa_map_complex' in 'cmplx2r', 'cmplx2i', 'cmplx2m', 'cmplxadd', 'cmplxdiv', 'cmplxmult', 'cmplxadd', 'cmplxconj', 'cmplxfilt', 'fft', 'bfft', 'i2cmplx', 'r2cmplx'.

Disuse of DTA error messages in 'alasout', 'cspike', 'fet321', 'figs424', 'rcgs2'.

Avoidance of DTA mapping routines in 'irflat', 'rcgs2'.

Cosmetic changes to the DYNAMIC_MEMORY include file.

Addition of routines 'dsa_input_update', 'dsa_named_input_update', and 'dsa_map_complex' to the DSA library. These allow the modified applications to be linked against DSA/DTA.

Removal of routines 'fig_convchk' and 'fig_dtaerr', which are related to the now unused DTA library.

G.7 Figaro 5.1-4 release

This is a shortened copy of the News item describing Figaro 5.1-4. Some items described in previous release notes have been removed.

You may have noticed some recent changes to Starlink Figaro—perhaps unfavourably—and wondered what was happening and why. This news item describes the new features, their benefits, and the rationale behind the main changes. It also offers some tips on getting the most from Starlink Figaro, and alternative routes where some functionality has had to be withdrawn. Finally, it announces and gives details of the release of a new version: 5.1-4.

Here is a summary:

**Figaro 5.1-4 release—main changes:**

- New Starlink support team for Figaro.
- Addition of the ability to select between using pixel indices and pixel centres in addressing data arrays.
- Bug fixes.

See §G.7.1 for further details.

**Recent Developments**

- Rationale for the change.
- History Recording.
• Removal of the ‘exam’ command (and what replaces it).

§G.7.3 describes these.

**Practical Information**
§G.7.2 contains practical information on the following.

• How to process a list of files with a command.
• How to save disk space when using ‘isubset’.
• How to use the value of a parameter from one command when using a second command.
• How to explicitly access a non-NDF file via the new Figaro Data Access Library (FDA).

**G.7.1 Release notes**

**Switching between Pixel Indices and Axes Values**

With the introduction of FDA in Figaro 5.1-3, the handling of default axis-centre numbering changed from the Figaro-style indices (start from the lower bound and increment by one per pixel) to the Starlink standard default pixel co-ordinates; these are pixel indices less 0.5.

From Figaro 5.1-4, the old-style behaviour can be selected by defining the environment variable FIGARO_AXES. The value of the variable does not matter: its presence selects Figaro-style indices; its absence selects Starlink standard co-ordinates. If your dataset has existing axis centres, they will be used irrespective of FIGARO_AXES.

Since V5.1-3 appeared you may have created NDFs that have axis centres in Starlink pixel co-ordinates. To get the former behaviour you will either need to remove the entire axis structure (so that it stays a valid NDF), for instance

```
% delobj myndf.axis
```

removes the axis structure from the NDF named myndf. However, erasing the axis structure also loses any other axis information such as the label, and the axis centres along other dimensions. You can replace the axis centres with pixel indices along an individual axis using the KAPPA task SETAXIS. For instance,

```
% kappa    # only needed if you have not activated KAPPA already
% setaxis spectrum mode=expr exprs=index
% setaxis image 2 expr exprs=index
```

would assign pixel indices to the axis centres in the 1-dimensional NDF called spectrum, and to the second dimension’s axis centres in the NDF called image.

**Bug Fixes**

Fixes to applications:

• ‘adjoin’ — the input SPECTRUM can now be overwritten.
• ‘ирконв’ — a bug caused by the use of a locator to an existing Figaro extension in a file prior to obtaining the locator has been removed. This prevented the magnitude flag from being written.

• ‘ирфлат’ — the scan position array is now fully filled.

• ‘ирфлюкс’ — a parameter missing from the interface file has been added.

• ‘исубет’ — several bugs related to the propagation of 2-D axis data from the input to the output have been removed.

• ‘профиль’ — an error in the ordering of two ДСА_ calls has been fixed ensuring that the output dataset exists before being written to.

• ‘вдфитс’ — A bug which could generate a second END card in the FITS header has been removed.

• ‘имульт/идив’ — Now correctly propagate errors for negative values of FACTOR.

• Abort to Parameters - in many applications an abort response (!!) to a parameter prompt would not have the desired effect. This has been fixed in 28 tasks.

Other fixes:

• If UPDATE access to a dataset is requested via the FDA library, a check that the output target exists is made before attempting access. If the target does not exist it is created.

• Trailing blanks present in input NDF axes LABEL and UNITS strings are now propagated to the output. Previously, these were truncated, leading to an error in the case of a completely blank string.

• Accesses to undefined AXIS UNITS in FDA have been removed.

• Some applications request input or output datafile names other than for NDFs. The maximum length of the path and filename allowed for these has been standardised at 132 characters. Previously, this varied from 32 to 132 characters. 29 applications have been altered as a result.

• The maximum Y-dimension of images used by ‘findsp’ has been increased from 500 to 4096 pixels.

G.7.2 Figaro tips

1. How to process a list of files with a command

One way to process several files in a UNIX C-shell is using a foreach loop. For example, if you wanted to add a constant to all the files matching the wildcard run???.sdf you might:

```
% foreach f ( run???.sdf )
  ? icadd $f:r 5.6 $f:r
? end
```
If you wanted the resulting output files to have the same root name but with some string appended, for example image.sdf to become image.db.sdf, then the above would change to:

```
% foreach f ( run1??\sdf )
  ? icadd $f:r 5.6 $f:r_db
? end
```

Similarly, you might like to store the commands in a script (let’s call it icmadd) instead of having to type them each time:

```
% icmadd 5.6 run1??\sdf
```

The basic icmadd script would look like this:

```
#!/bin/csh
figaro # Loads Figaro commands.
set c = $1 # Gets constant to be added from first arg.
shift # Throws away constant from list of args.
foreach f ( $* ) # Processes list of files.
  icadd $f:r $f:r_db
end
```

(2) How to save disk space when using ISUBSET

If you use ‘isubset’ to remove unwanted parts of an image the output file produced remains approximately the same size as the input file. This is the case even if you select only a very small part of the input dataset.

Using NDF sections, this effect can be eliminated, resulting in output files of the expected size. For example, if you wanted to take a 100 × 100-pixel subset from an image, and would normally use:

```
% isubset im=inimage ys=200 ye=299 xs=250 xe=349 out=outimage
```

(note the use of abbreviated parameter names, see notes for 5.1-3), the same subset can be obtained by:

```
% isubset im=inimage\'(200:299,250:349)\' ys=min ye=max \n  xs=min xe=max out=outimage
```

The difference being that the file produced by this command will be smaller. If your dataset does not contain n-dimensional axis centres then the KAPPA command NDFCOPY is more succinct.

```
% ndfcopy inimage\'(200:299,250:349)\' outimage
```

(3) How to use the value of a parameter from one command when using a second command

Sometimes you may want, for example, to take the median obtained using ‘istat’ and use this value as the FACTOR in an ‘icsub’ command. This can be done as follows:
This assumes that the environment variable ADAM_USER points to the directory where your parameter files are stored; this is by default the directory adam in your home directory (~/adam).

(4) How to explicitly access a non-NDF file via the new Figaro Data-Access Library (FDA)

If you wish to access a data file which has the same root name as an NDF you must give the full file name to explicitly select that file. For example, if you have two files image.sdf (an NDF) and image.dst (a DST) in the same directory then

```bash
% figaro
% convert
% icsub image 4 output
```

will select the NDF, assuming you have not altered the default precedence used (see notes for 5.1-3), and

```bash
% figaro
% convert
% icsub image.dst 4 output
```

will select the DST file explicitly.

The environment variable FIGARO_FORMATS is no longer used.

G.7.3 New Figaro features

If it ain’t broke... 

The conversion to use the NDF library was requested by the Spectroscopy Software Strategy Group (cf. SGP/48, Appendix A2.1).

The change adds functionality to Figaro, namely automatic access to foreign data formats, NDF sections, and history recording. It gives more consistency with other Starlink packages; this is important when your data-processing path requires functionality not available in Figaro. Figaro will be able to take advantage of any future developments to the NDF library, such as astrometry and processing of wildcarded lists of files. The change to FDA also opens the way to have Figaro operating from the IRAF cl.

As automatic conversion requires the NDF library to be called, the move to FDA had the side effects of changing the default axis centres and the removal of the ‘exam’ command. The former is circumvented through the new FIGARO_AXES environment variable, and there is a ready replacement—HDSTRACE—for the latter.

History Recording

Since the FDA library calls the NDF library, you can record the data-processing history of an NDF. Since the information can be bulky compared to the data array, especially for a spectrum, history recording is disabled by default.

The KAPPA command HISSET lets you change the level of history recording from one of four levels: Disabled, Quiet, Normal, and Verbose. The following command sets the recording level to normal in NDF spectrum.
% hisset spectrum

Thereafter whenever you alter this NDF or create another NDF from it, the Figaro task will automatically record the name of the task run, the date and time, and some text comprising the command-line parameters and the full path of the application.

You can disable or erase the history using HISSET.

% hisset spectrum disabled
% hisset spectrum erase

KAPPA also provides commands for adding commentary (HISCOM), and listing the history records (HISLIST). If you enter

% findme "NDF History"

and select the link “NDF History” in SUN/95, you will find more information and examples of using these commands.

EXAM has been removed

The Figaro command ‘exam’ has been removed. This is for two reasons: first, ‘exam’ calls the DTA library, which is no longer used elsewhere in Starlink Figaro, and so would need major modification to call HDS directly; second, the functionality of ‘exam’ is available in the task HDSTRACE, which already calls HDS.

All the facilities of ‘exam’ are available in hdstrace. One common use for ‘exam’ was to list all the objects present in a file. This would have been done by:

% exam image..

The same thing can be done using hdstrace by:

% hdstrace image full

(and the full keyword is only required if you want to view all elements of any array of structures, such as the axis structure in a multi-dimensional NDF). Note that HDSTRACE displays the standard HDS data types so, for instance, you’ll see <_REAL> for Float, and a string of say six characters has type <_CHAR*6> rather than a six-element array of Char type.

You can access individual components like this:

% hdstrace image.more.figaro.airmass

might display:

STRUCT MORE FIGARO AIRMASS <_REAL>

AIRMASS 1.324

End of Trace.
This value could be used in a C-shell script like this:

```bash
set airmass = `hdstrace image.more.figaro.airmass \
    | grep " AIRMASS"`
shift airmass  # Throws away the object’s name
```

then $airmass will substitute the airmass value.

If you want to trace a DST file, the syntax is a little more complicated. Here are two examples.

```bash
% hdstrace @"image.dst"
% hdstrace @"image.dst".obs.airmass
```

Note the backslash is needed to escape the " special character from the UNIX shell.

### G.8 Figaro 5.1-6 release

This is a copy of the News item describing Figaro 5.1-6.

Version 5.1-5 of Figaro was only made available on the Starlink Linux CD, Version 1.3. Version 5.1-6 is identical to 5.1-5, except for the changes to the HELP text and the DSA_RESHAPE_DATA improvement (see below).

This news item accompanies the latest release of Figaro: version 5.1-6. This is a maintenance release with some small performance improvements and bug fixes. There has been an update of SUN/86 due to the recent withdrawal of XADAM. Some tips on tuning the performance of Figaro are included below.

#### G.8.1 Release notes

The main changes at this release are:

- The HELP text has been tidied as part of the preparation for an IRAF version of Figaro.

- Workspace for internal calculations is now allocated from memory, rather than in temporary NDFs. This leads to an improvement in speed for applications affected. The difference is generally quite small, except where the working directory resides on a ‘remote’ NFS-mounted disk. In this case the change is quite significant and is most noticeable with a command like ‘image’. Typically, the task completes in 15–25% of the time previously taken, depending on the level of network activity and disk usage. After the image is displayed, there is no longer a delay before the shell prompt returns. See ‘Figaro Tips’ below for more information on optimising your use of Figaro.

  The applications for which this change will be significant are:

  ```
  APERTURE CLEAN COADD ECHARC FIGS321 FIGS322 IGCONV IMAGE ISEDIT ISHIFT ISTAT ISTRETCH MEDFILT MEDFILTR MEDSKY OVERPF RETYPE
  ```

  ‘Istat’ is only effected if median=true. ‘Medfilt’, ‘medfiltr’, and ‘retype’ are only effected if the output is to the same file as used for input. About 40 other applications should see a slight improvement in performance.
The routine DSA_RESHAPE_DATA has been modified to ensure that the minimum space required for an output file is used. Previously, the size of an output file from a task using DSA_RESHAPE_DATA would be the same as the input file. This change affects several applications, notably ‘extract’ and ‘ystract’. These two commands, which produce spectra from input 2-D datasets, would previously produce output files much larger than needed.

- Bug fixes.
- The Introduction and Beginners sections of SUN/86 have been updated; this reflects the withdrawal of XADAM.

G.8.2 Figaro tips

1) Speeding up processing of data stored on remote disks

When you access data stored on a disk connected to a different machine to the one on which your shell process is running, commands take more time to run. There are several reasons for this: the level of network activity, the load on the remote machine, and so on. Generally, you’re better off working with data on a disk connected to the machine you’re using; but it isn’t always possible, or helpful, to do this.

When accessing input data files, Figaro sometimes has to create temporary files. These are, by default, placed in your current working directory. What this means is that if you are working on ‘remote’ data, and your working directory is also ‘remote’, Figaro has to write and then read back data across the network. This can slow things down quite a bit! There is no once-and-for-all way around this, but you may be able to take advantage of a feature of HDS which lets you control where the temporary files are stored. For example, if your home directory is on the local machine you might

```bash
% setenv HDS_SCRATCH /home/username
```

Temporary files would then be placed in your home directory. Bare in mind that temporary files can be quite large for some applications, and the area available in the directory $HDS_SCRATCH needs to be of requisite size.

2) Speeding up ‘on-the-fly’ data format conversions

This is closely related to the first tip. Figaro can use on-the-fly conversions from, for example, FITS format. So, if you have some fits dataset starimage.fit, you can display it thus:

```bash
% figaro
% convert
% image starimage.fit
```

If starimage.fit is on an NFS-mounted disk, and in your working directory, it will take a while to display. This is because all the conversion data is being transferred over the network. The first action to tune this procedure is to setup $HDS_SCRATCH to point to a local disk (see above). The next thing to do is set $NDF_TEMP_FITS to be a local filename, the complete process is:
% figaro
% convert
% setenv HDS_SCRATCH /home/username
% setenv NDF_TEMP_FITS /home/username/tmp_ndf_~name
% image starimage.fit

The first four lines of the above example need be issued once per session. Remember the directory you use must be on a local disk, and there must be enough space for the temporary NDF created by CONVERT. The intermediate NDF will be stored in the file

/home/username/tmp_ndf_starimage.sdf

There are similar tuning parameters for other data formats, to get the full details see ‘Specifying Where the Native NDF is Stored’ in SSN/20.

G.8.3 Bug fixes

Fixes to applications:

• ‘alasin’ — Now works again, an error searching for End-of-file has been fixed.

• ‘icur’ — The output list of user-selected points is limited in size to the number of points, rather than the maximum number of points. This change will only effect those using Figaro under the IRAF cl.

• ‘medsky’ — The size of an internal workspace array has been enlarged by a factor of 16. For input data with quality information ‘medsky’ will run about 10 times faster than previously. There is a small improvement in speed for all data, regardless of the presence of a QUALITY component.

G.9 Figaro 5.2-0 release

This news item accompanied the release of Figaro version 5.2-0.

G.9.1 Release notes

The main changes at this release are:

• Enhanced and updated error propagation in many tasks. See below for more details.

• New task ‘ialog’ to take the antilog of pixels in a dataset. See SUN/86 or the on-line HELP for details.

• Bug fixes in Linux version.

There is a world-wide web page including Figaro documentation and related information:

• http://www.star.ucl.ac.uk/~mjc/figaro/
G.9.2 Error propagation

It is now possible to propagate the errors through the complete reduction of simple single-order spectra. A recommended reduction recipe is described in SUN/86.

Tasks which now include error propagation include:

- The majority of the mathematical operations
- B-star spectra
- Variance "fudging" for fitting data
- Removal of cosmic rays and bad columns

In the following tasks, error handling has been added or corrected at this release:

   BCLEAN   BSMULT   CSET   EXTRACT   FF   GROWX
   GROWY   IALOG   ICADD   ICDIV   ICMULT   ICSUB
   IPOWER   ISXADD   ISXDIV   ISXMULT   ISXSUB   ISYADD
   ISYDIV   ISYMULT   ISYSUB   POLYSKY   YTRACT

All the above now include propagation of the variance arrays into the output. The errors are propagated in the standard way (see e.g. Bevington and Robinson 9 for most routines).

There are a few circumstances in which the error propagation is not completely formal which are documented.

The following Figaro tasks also support error propagation:

   ABCONV   ADJOIN   COMBINE   ERRCON   FIGSFLUX   FSCRUNCH
   GAUSS   IADD   IDIV   ILOG   IMULT   ISUB
   IREVX   IREVY   IRLFUX   IROT90   IEDIT   ISUBSET
   OPTEXTRACT   POLYSKY   PROFILE   REMBAD

G.9.3 Bug fixes

Fixes to applications:

- 'istat' — A bug in the Linux version of 'istat’, resulting in a random output message for 1-D data, has been removed.

- 'medsky' — A bug in the Linux version of 'medsky', causing the task to fail completely, has been removed.

Fixes to Linux release:

- For some applications with variance propagation, a segmentation fault would occur if one or more of the input data files were incorrectly specified (if the file did not exist for example). This error has been removed.

G.10  Figaro 5.3-0 release

The following notes are a slightly modified version of the news item which accompanied the release of Figaro version 5.3-0 (spring 1998).

The major changes in version 5.3-0 are:

- incorporation of the SPECDRE spectroscopy package,
- incorporation of IRAFFIG (which allows Figaro to be run from IRAF).

In addition there are a couple of minor enhancements.

A.C. Davenhall (acd@roe.ac.uk), 19 December 1997.

G.10.1  Incorporation of SPECDRE

The spectroscopy package SPECDRE has been incorporated into Figaro. The SPECDRE commands are available automatically when Figaro is started. The SPECDRE commands available within Figaro differ from the previous stand-alone ones in the following respects:

- SPECDRE command RESAMPLE is now command RESAMP (to avoid a name clash with the Figaro command RESAMPLE),
- command GOODVAR has been withdrawn because the same functionality is already available within Figaro.

SUN/140, which described SPECDRE, has been withdrawn and its contents incorporated in SUN/86.

G.10.2  Incorporation of IRAFFIG

The IRAFFIG package is a set of files which allows Figaro to be run from the IRAF command language CL. In practice it makes Figaro available to IRAF users. For convenience it is now released as part of Figaro.

SUN/220, which described IRAFFIG, has been withdrawn and instructions for running Figaro from IRAF have been included in SUN/86.

G.10.3  Minor enhancements

Version 5.3-0 includes the following minor enhancements:

- Applications ARC and ECHARC now simply overwrite any pre-existing file called airlines.lis rather than terminating with a message,
- the maximum permitted number of elements in a re-binned spectrum created with SCRUNCH has been increased to 5,000,000.

In addition there have been a couple of bug fixes.
G.11 Figaro 5.4-0 release

The following notes are a slightly modified version of the news item which accompanied the release of Figaro version 5.4-0 (spring 1999).

The major change in version 5.4-0 is the addition of the application SCLEAN for removing bad lines and pixels from images obtained with the SCUBA instrument on the JCMT. SCLEAN reprises the functionality of the traditional Figaro application CLEAN, but it correctly handles the array of quality flags associated with the data array in SCUBA images. It also provides a new display mode suitable for use with SCUBA data.

In addition a number of bugs have been fixed.

G.12 Figaro 5.5-0 release

The following notes are a slightly modified version of the news item which accompanied the release of Figaro version 5.5-0 (autumn 1999).

The major change in version 5.5-0 is the incorporation of the Twodspec package, which is used for longslit spectroscopy data reduction and analysis. It provides tools for calibration/correction of data and fitting of 2-D longslit arrays, either automatically or as an interactive process. These tools also offer a number of options for generating hard copy output of the resulting fits.

The original stand-alone Twodspec accessed data files using the DSA and DTA subroutine libraries. As part of the process of incorporating it into Figaro it was modified to use the FDA library and hence it now accesses NDF data files. Consequently, the Twodspec applications are inter-operable with the basic Figaro ones. However, changing the data-access subroutine library introduced or revealed a number of bugs, some of which are still extant. These problems will be identified and corrected in future releases.

In addition there are a number of bug fixes not related to Twodspec.

G.13 Figaro 5.5-1 release

The following notes are a slightly modified version of the news item which accompanied the release of Figaro version 5.5-1 (spring 2000).

Version 5.5-1 is a bug fix release which updates the interface file for the EMLT task to include some parameter defaults that were missing.

G.14 Figaro 5.5-2 release

The following notes are a slightly modified version of the news item which accompanied the release of Figaro version 5.5-2 (spring 2001).

Version 5.5-2 includes a number of bug fixes to Figaro and some minor corrections to SUN/86.

G.15 Figaro 5.6-0 release

The following notes are a slightly modified version of the news item which accompanied the release of Figaro version 5.6-0 (autumn 2001).
Version 5.6-0 contains relatively few changes: the automatic propagation of errors has been added to a few more applications, there have been a number of bug fixes and minor enhancements and there is one new minor application. In addition the user manual has been slightly revised. Brief details of the changes are given below.

G.15.1 Error Propagation

Error Propagation has been added to the following applications: BBODY, FWCONV, IRFLAT, ISHIFT, IXSMOOTH and SFIT.

G.15.2 Bug Fixes and Minor Enhancements

Either bugs have been fixed in, or minor enhancements have been made to, the following applications: CSPIKE, ECHSELECT, EMLT, FITGAUSS, SCROSS and SPFLUX. In addition, corrections have been made to the documentation for CENTERS and SPECCONT.

G.15.3 New Application

The new application IMPOS has been added. It simply reads a text file containing a list of $x, y$ positions and writes them to the environment variables which input values to application CENTERS. Thus, it provides a mechanism for using CENTERS from a script rather than interactively.

G.16 Figaro 5.6-1 release

The following notes are a slightly modified version of the news item which accompanied the release of Figaro version 5.6-1 (winter 2002).

Version 5.6-1 contains relatively few changes: there have been several bug fixes and an enhancement to application IARC. There have also been some revisions to the user manual. Brief details of the changes are given below.

Some spectrophotometric flux tables of standard stars suitable for use in flux calibration with Figaro have been recovered and made available again. These data were previously available on the now-defunct RAL central database machine STADAT.

G.16.1 Bug Fixes and Minor Enhancements

Bugs have been fixed in: EMLT, ICUR, IMAGE, POLSKY and XTRACT.

An option has been added to IARC to allow it to perform weighted as well as un-weighted fits.

G.16.2 Flux calibration standard stars

Two additional sets of spectrophotometric flux tables suitable for use in flux calibration with Figaro are available. One comprises 25 standard stars observed by Oke, the other 27 HST standard stars. Copies can be retrieved by anonymous ftp.
G.17  Figaro 5.6-2 release

The following notes are a slightly modified version of the news item which accompanied the release of Figaro version 5.6-2 (summer 2003).

The only changes in Version 5.6-2 are a couple of bug fixes.

G.18  Figaro 5.6-3 release

The following notes are a slightly modified version of the news item which accompanied the release of Figaro version 5.6-3 (spring 2004).

The changes in Version 5.6-3 comprise only a few minor bug fixes and enhancements, and a missing flux calibration file has been added. Brief details of the changes are given below.

G.18.1  Bug Fixes and Minor Enhancements

GAUSS and IARC have each received both a bug fix and a minor enhancement. Consequently, there has been an equally minor revision to the description of GAUSS in the user manual.

EMLT has a new NOISE keyword. If it is specified, allowance for noise is made when determining the line limits. This is intended for broad emission features or beam location rather than narrow arc lines. It defaults to the former behaviour.

SDIST allows for broader spectral profiles when tracing a spectrum. There was an arbitrary cut-off of the size of the extracted cross-section to be fitted.

G.18.2  Flux Calibration Files

The flux calibration file for Hiltner 102, hil1102.tab, was missing. It has been added.
Keyword index

The index does not include Figaro applications as keywords. The index points either to a section or sub-section of SUN/86, or to a different Starlink document (SUN, SSN, SG, SGP). Where the index points to a section it may be intended to point to the sub-sections of that section as well, without making this explicit.

A

@ 3.1 Parameters
3.2 Internals of a data file
G.1.6 Changes: Parameters and variables

ADAM
SG/4: ADAM — Starlink software environment
SG/5: ICL — Interactive command language for ADAM
SG/6: ADAM — Programmer’s facilities and documentation guide
SUN/101: ADAM — Introduction to ADAM programming
SUN/114: PAR — Interface to the ADAM parameter system
SUN/115: ADAM — Interface module reference manual
SUN/144: ADAM — Unix version
SSN/45: ADAM — Release 2.1
SSN/64: ADAM — Organization of Application Packages

ADAM parameter system
3.1 Parameters
G.4 Figaro 3.2-3 news item
SUN/114: PAR — Interface to the ADAM parameter system
SUN/115: ADAM — Interface module reference manual

ADAM_USER
2.1 Unix setup
G.5 Figaro 5.0 release

(A)PAR, see ADAM parameter system
abbreviated parameter names
3.1.2 Parameters: On the command line
abort flag
3.1.3 Parameters: Prompted
G.1.6 Changes: Parameters and variables
G.3 Figaro 3.2 news item
absorption lines
4.11 Abline command
4.12 Gauss command
accept
2.5 Looking at an image
3.1.3 Parameters: Prompted
alink
F.1 Programming
G.1.2 Changes: General
application: synonymous with command
arc
4.8 Wavelengths
4.8.3 Wavelengths: The arc command
4.8.4 Wavelengths: Applying an arc fit to other spectra
4.10 Arc command
4.13 Échelle reduction
arc calibration: see also arc
B.7 in Specdre
C.3 in Twodspec
arc line list: see arc
asymmetry
4.11 Abline command
axis
2.4 Looking at a spectrum
3.2.2 Internals of a data file: NDF format
3.2.5 Internals of a data file: DST format
4.6 FFT
4.8 Wavelengths
4.11 Abline command
4.13.10 Échelle reduction: Wavelength calibration
A.6 Commands: Arithmetic
A.18 Commands: Complex and FFT
G.1.8 Changes: Old data access routines
G.3 Figaro 3.2 news item
G.5 Figaro 5.0 release

AGI
2.1 Unix setup
G.1.2 Changes: General
G.1.3 Changes: Ported applications
G.1.9 Changes: High-level subroutines
G.5 Figaro 5.0 release
SUN/48: AGI — Applications graphics interface

AGI_USER
2.1 Unix setup
G.5 Figaro 5.0 release

accept
2.5 Looking at an image
3.1.3 Parameters: Prompted
alink
F.1 Programming
G.1.2 Changes: General
application: synonymous with command
arc
4.8 Wavelengths
4.8.3 Wavelengths: The arc command
4.8.4 Wavelengths: Applying an arc fit to other spectra
4.10 Arc command
4.13 Échelle reduction
arc calibration: see also arc
B.7 in Specdre
C.3 in Twodspec
arc line list: see arc
asymmetry
4.11 Abline command
axis
2.4 Looking at a spectrum
3.2.2 Internals of a data file: NDF format
3.2.5 Internals of a data file: DST format
4.6 FFT
4.8 Wavelengths
4.11 Abline command
4.13.10 Échelle reduction: Wavelength calibration
A.6 Commands: Arithmetic
A.18 Commands: Complex and FFT
G.1.8 Changes: Old data access routines
G.3 Figaro 3.2 news item
G.5 Figaro 5.0 release

B.7 in Specdre
C.3 in Twodspec
arc line list: see arc
asymmetry
4.11 Abline command
axis
2.4 Looking at a spectrum
3.2.2 Internals of a data file: NDF format
3.2.5 Internals of a data file: DST format
4.6 FFT
4.8 Wavelengths
4.11 Abline command
4.13.10 Échelle reduction: Wavelength calibration
A.6 Commands: Arithmetic
A.18 Commands: Complex and FFT
G.1.8 Changes: Old data access routines
G.3 Figaro 3.2 news item
G.5 Figaro 5.0 release

C.3 in Twodspec
arc line list: see arc
asymmetry
4.11 Abline command
axis
2.4 Looking at a spectrum
3.2.2 Internals of a data file: NDF format
3.2.5 Internals of a data file: DST format
4.6 FFT
4.8 Wavelengths
4.11 Abline command
4.13.10 Échelle reduction: Wavelength calibration
A.6 Commands: Arithmetic
A.18 Commands: Complex and FFT
G.1.8 Changes: Old data access routines
G.3 Figaro 3.2 news item
G.5 Figaro 5.0 release

Figaro 5.0 release

Sun/86.21 —Keyword index

The index does not include Figaro applications as keywords. The index points either to a section or sub-section of SUN/86, or to a different Starlink document (SUN, SSN, SG, SGP). Where the index points to a section it may be intended to point to the sub-sections of that section as well, without making this explicit.
B

- BoundingBox
  2.7 Paper copies

- backslash
  3.1.3 Parameters: Prompted
  3.1.5 Parameters: Syntax conflicts
  G.1.8 Changes: Old data access routines

- bad value
  3.2.2 Files: NDF format
  3.2.5 Files: DST format
  A.12 Commands: Fudging
  G.1.3 Changes: Ported Applications
  G.1.8 Changes: Old data access routines
  G.1.11 Changes: Low-level subroutines
  G.5 Figaro 5.0 release

- batch mode
  4.11.2 Abline command: Running abline
  G.1 Changes

- bug report
  2.3.4 Getting help: Support

C

- C shell: see shell

- CHR
  SUN/40: CHR — Character handling routines

- CNF
  SGP/5: CNF — C and Fortran mixed programming

- CNV
  G.1.2 Changes: General

- CONVERT
  3.2.4 Internals of a data file: Foreign formats
  G.6 Figaro 5.1.3
  SGP/55: CONVERT — A format-conversion package

- Callable Figaro
  G.1.2 Changes: General
  G.1.5 Changes: Un-ported applications

- cabujy
  G.1.2 Changes: General

- centre of gravity
  4.10.5 Arc command: Moving about etc.
  4.11 Abline command
  A.9 Commands: Photometry

- centroid: see centre of gravity

- colour
  2.4 Looking at a spectrum
  2.5.1 Looking at an image: The image command
  2.5.3 Looking at an image: Adding colour
  2.6 Image display in monochrome
  A.3 Commands: Display
  G.1.2 Changes: General
  G.2 Figaro 3.1-1 news item

- colour lookup table
  2.5.1 Looking at an image: The image command
  2.5.3 Looking at an image: Adding colour
  2.6 Image display in monochrome
  G.1.2 Changes: General
  G.2 Figaro 3.1-1 news item

- command
  2.2 Starting Figaro
  2.3 Getting help
  A Classified list of commands
  F.1 Programming

- command line interface
  2.2 Starting Figaro
  2.3 Getting help
  3.1 Parameters

- compifl
  F.1 Programming
  G.1.2 Changes: General

- complex data: see FFT

- continuum
  4.2 B stars
  4.5 Fluxing
  4.5.1 Abline command
  4.12 Gauss command
  4.13 Échelle reduction
  A.5 Commands: B stars
  A.7 Commands: Flats
  A.16 Commands: Fluxing

- crepar
  F.1 Programming

- cshrc
  2.1 Unix setup

- cube manipulation
  B.5 in Specdre
  C.8 in Twodspec

D

- DCL Figaro
  G.1 Changes
• DSA
  G.1.6 Changes: Parameters and variables
  G.1.8 Changes: Old data access routines
  G.3 Figaro 3.2 news item
  G.5 Figaro 5.0 release
  G.5.1 Figaro 5.0-1 news item
  G.6 Figaro 5.1.3 release
• DSK
  G.1.2 Changes: General
  G.1.5 Changes: Un-ported applications
• DST
  1 Introduction
  3.2 Internals of a data file
  G.1.3 Changes: Ported applications
  G.1.8 Changes: Old data access routines
  G.3 Figaro 3.2 news item
  G.6 Figaro 5.1.3 release
• DTA
  G.1.8 Changes: Old data access routines
  G.3 Figaro 3.2 news item
  G.5 Figaro 5.0 release
  G.6 Figaro 5.1.3 release
• DUT
  G.1.2 Changes: General
• DYN
  G.1.8 Changes: Old data access routines
• data array
  3.2 Internals of a data file
  4.6 FFT
• data file
  3.2 Internals of a data file
• data format
  1 Introduction
  3.2 Internals of a data file
  4.3 Filters
  4.4 Spiketra
  4.5.2 Fluxing: Standard files
  4.10.1 Arc command: Arc line lists
  4.12.4 Gauss: Format of data file
  A.1 Commands: Input
  A.2 Commands: Output
  G.1 Changes
  G.3 Figaro 3.2 news item
  G.4 Figaro 3.2-3 news item
  G.6 Figaro 5.1.3 release
  SUN/55: CONVERT — A format conversion package
  SSN/20: Adding format conversion facilities to the NDF data access library
• data section: see NDF section
• data type
  3.2.1 Internals of a data file: HDS files
  G.1.3 Changes: Ported applications
• display
  2.4 Looking at a spectrum
  2.5 Looking at an image
  2.6 Image display in monochrome
  A.3 Commands: Display
  G.1.2 Changes: General
  G.2 Figaro 3.1-1 news item
• dta2hds
  G.1.2 Changes: General
• dvips
  2.7 Paper copies
  SUN/9: \LaTeX{} — A document preparation system
• EMS
  G.1.2 Changes: General
  SUN/104: MSG and ERR — Message and Error Reporting Systems
  SSN/4: EMS — Error message service
• ERR: see EMS
• Echomop
  4.13 \text{Échelle reduction}
  SUN/152: ECHOMOP — \text{Échelle} data reduction package
• Encapsulated PostScript: see PostScript
• échelle
  4.13 \text{Échelle reduction}
  A.20 Commands: \text{Échelle}
  G.5 Figaro 5.0 release
  SUN/152: ECHOMOP — \text{Échelle} data reduction package
  SUN/53: ECHWIND — UCL \text{échelle} spectrograph observation planning program
  SSN/3: ECH — \text{Échelle} spectrograph model package
• emission lines
  4.11 Line command
  4.12 Gauss command
  A.10 Commands: Line fits
• environment variables
  G.1 Changes
  G.1.2 Changes: General
• epsf, epsf_p, epsfcol, epsfcol_p
  2.7 Paper copies
equivalent width
4.11 Abline command
4.12 Gauss command

error (see also variance)
3.3 Error propagation
3.2.5 Internals of a data file: DST format
4.12 Gauss command
A.3 Commands: Display
A.8 Commands: Manipulations
A.23 Commands: Miscellany
G.1.8 Changes: Old data access routines

extinction
4.9 Extinction
A.17 Commands: Extinction

FFT
4.6 FFT
G.5 Figaro 5.0 release
G.6 Figaro 5.1-3 release

FFTPACK
4.6.1 FFT: Complex data structures
G.5 Figaro 5.0 release
SUN/194: PDA — Public Domain Algorithms' library

FIT
G.1.2 Changes: General

FITS (data format)
1. Introduction
3.2.4 Internals of a data file: Foreign formats
A.1 Commands: Input
A.2 Commands: Output
G.1 Changes
G.1.2 Changes: General
G.1.3 Changes: Ported applications
G.1.5 Changes: Un-ported applications
G.1.11 Changes: Low-level subroutines
G.3 Figaro 3.2 news item
G.4 Figaro 3.2-3 news item
G.6 Figaro 5.1-3 release

FITS (extension to DST and NDF formats)
3.2.6 Internals of a data file: Extensions
3.2.7 in Specdre
3.2.8 in Twodspec
A.1 Commands: Input

(F)PAR
3.1 Parameters
G.1.6 Changes: Parameters and variables

Figaro
SUN/86: FIGARO — A general data reduction system
SUN/19: NDPROGS — N-D image manipulation programs (FIGARO)

Figaro startup
2.1 Unix setup
2.2 Starting Figaro

Fourier
4.6 FFT

false colour: see colour lookup table

fig_prefix
2.2 Starting Figaro
G.1.9 Changes: High-level subroutines

figaro_link
F.1 Programming
G.5 Figaro 5.0 release

file name
3.1 Parameters
3.2.1 Internals of a data file: HDS files
4.3 Filters
4.4 Spiketra
4.5.2 Fluxing: Standard files
4.7 S-distortion
4.9 Extinction
4.10.1 Arc command: Arc line lists
4.10.2 Arc command: Other initial prompts

FIG
G.1.2 Changes: General
G.1.9 Changes: High-level subroutines

FIGARO_PROG_[LNSU]
2.5.3 [Looking at an image: Adding colour
4.10.1 Arc command: Arc line lists
4.5 Fluxing
4.9 Extinction
G.1.2 Changes: General
G.2 Figaro 3.1-1 news item

FIO
G.1.2 Changes: General
SUN/143: FIO — Fortran I/O Routines
4.13 Échelle reduction
G.6 Figaro 5.1-3 release

• file search path
G.1.2 Changes: General
G.1.3 Changes: Ported applications

• filter
4.3 Filters
4.6.3 FFT: Going smoothly to zero at the ends
4.6.7 FFT: Complex filters and data smoothing
A.8 Commands: Manipulations
A.18 Commands: Complex and FFT

• fit function to spectrum
B.6 in Specdre
C.2 in Twodspec

• flagged value: synonymous with bad value

• flat: see flat field

• flat field
4.1 Flats
A.7 Commands: Flats
A.19 Commands: Infra-red

• flux (definition)
4.5.9 Fluxing: The final step revisited
4.8.5 Wavelengths: Linear wavelength scales—scrunching

• flux calibration
4.4 Spiketra
4.5 Fluxing
A.16 Commands: Fluxing
A.19 Commands: Infra-red

• flux density (definition)
4.5.9 Fluxing: The final step revisited
4.8.5 Wavelengths: Linear wavelength scales—scrunching

• foreign data formats
1 Introduction
3.2.4 Internals of a data file: Foreign formats
3.2.5 Internals of a data file: DST format
3.2.6 Internals of a data file: Extensions
G.6 Figaro 5.1-3 release
SUN/55: CONVERT — A format conversion package
SSN/20: Adding format conversion facilities to the NDF data access library

G

• GEN
G.1.2 Changes: General

G.1.11 Changes: Low-level subroutines
G.4 Figaro 3.2-3 news item
G.5 Figaro 5.0 release

• GKD
G.1.12 Changes: Other libraries

• GKS
2.7 Paper copies
G.1.2 Changes: General
G.1.3 Changes: Ported applications
G.5 Figaro 5.0 release
SUN/83: GKS — Graphical Kernel System
SUN/113: ADAM graphics programmer’s guide

• GNS
G.1.2 Changes: General
G.1.3 Changes: Ported applications
SUN/57: GNS — Graphic workstation name service

• GWM
G.1.2 Changes: General
SUN/130: GWM — X graphics window manager

• GWM window
2.4 Looking at a spectrum
2.5 Looking at an image
2.6 Image display in monochrome
SUN/130: GWM — X graphics window manager

• Gauss
4.12 Gauss command
G.1.3 Changes: Ported applications
G.3 Figaro 3.2 news item
G.5 Figaro 5.0 release

• ghostscript (gs), ghostview
2.7 Paper copies

• global parameter
3.1 Parameters
4.1.2 Flats: Spectral data
4.7.4 S-distortion: Making do without an image display
F.1 Programming
G.1.6 Changes: Parameters and variables

• graphical user interface
G.8 Figaro 5.1-6 release

• graphics
2.1 Unix setup
2.4 Looking at a spectrum
2.5 Looking at an image
2.6 Image display in monochrome
B.3 Graphics in Specdre
• grey: see colour
• gs: see ghostscript

H
• HDS
  3.1 Parameters
  3.2 Internals of a data file
  A.12 Commands: Fudging
  G.1.2 Changes: General
  G.1.3 Changes: Ported applications
  G.1.8 Changes: Old data access routines
  G.2 Figaro 3.1-1 news item
  G.4 Figaro 3.2-3 news item
  G.5 Figaro 5.0 release
SUN/92: HDS — Hierarchical Data System programmer's manual
SUN/102: HDSTRACE — Listing HDS Data Files

• HELP
  SUN/124: HELP — Interactive help system
  SUN/175: MOSAIC — A hypertext help system

• HyperText-based help
  G.5 Figaro 5.0 release
  SUN/175: MOSAIC — A hypertext help system

• hard-copy (see also PostScript)
  2.7 Paper copies
  A.3 Commands: Display

• hds2dta
  G.1.2 Changes: General

• hdstrace
  3.1.5 Parameters: Syntax conflicts
  3.2 Internals of a data file
  4.8.2 Wavelength calibration: Wavelength arrays
  4.13.11 Échelle reduction: Scrunching
  G.6 Figaro 5.1-3 release
  SUN/102: HDSTRACE — HDS data file listing

• heliocentric
  4.11.2 Abline command: Running abline

• help
  2.3 Getting help
  3.1.3 Parameters: Prompted
  A.23 Commands: Miscellany
  G.1 Changes
  G.3 Figaro 3.2 news item
  G.5 Figaro 5.0 release
SUN/124: HELP — Interactive help system
SUN/175: MOSAIC — Hypertext help system

I
• ICL
  2.3 Getting help
  3.1.4 Parameters: Output
  G.4 Figaro 3.2-3 news item
  G.5 Figaro 5.0 release
SG/4: ADAM — Starlink software environment
SG/5: ICL — Interactive command language for ADAM

• IFL file
  G.1.6 Changes: Parameters and variables
  SUN/115: ADAM — Interface module reference manual

• IRAF (data format)
  1 Introduction
  3.2.4 Internals of a data file: Foreign formats
  G.6 Figaro 5.1-3 release

• in-situ
  G.4 Figaro 3.2-3 news item
  G.5 Figaro 5.0 release

• include file
  F.1 Programming

• interface file
  G.1.2 Changes: General
  G.1.6 Changes: Parameters and variables
  F.1 Programming
  SUN/115: ADAM — Interface module reference manual

J
• JT: see JTY
• JTY
  G.1.10 Changes: The JT-library: JTY

K
• KAPPA
  2.7 Paper copies
  G.1.2 Changes
  G.4 Figaro 3.2-3 news item
  G.5 Figaro 5.0 release
  SUN/95: KAPPA — Kernel application package
L

- **LSR**
  4.11.2 | Abline command: Running abline

- **\LaTeX**
  2.7 | Paper copies
  SUN/9: \LaTeX — A document preparation system
  SUN/12: \LaTeX — \LaTeX cookbook
  SUN/154: \TEX — Additional \TeX and \LaTeX facilities
  SUN/155: FORMLOAD — \LaTeX form generator

- **line analysis**
  4.11 | Abline command
  4.12 | Gauss command
  A.10 | Commands: Line fits
  B.6 | in Spectre
  C.2 | in Twodspec

- **line integral**
  4.11 | Abline command
  4.12 | Gauss command

- **line list**
  4.10.1 | Arc command: Arc line lists
  4.13.10 | \textit{Échelle reduction: Wavelength calibration}

- **line profile**
  4.11 | Abline command
  4.12 | Gauss command

- **line width**
  4.11 | Abline command
  4.12 | Gauss command

- **linear scrunching**: see scrunching

- **logarithmic scrunching**: see scrunching

- **look up table**: see colour lookup table

M

- **MEM**
  G.1.2 | Changes: General

- **MSG**: see EMS

- **MTPCKG**
  G.1.2 | Changes: General

- **Mosaic**: see HyperText-based help

- **magic value**: synonymous with bad value

- **max**: see parameter extrema

- **mean** (see also centre of gravity)
  4.1.1 | Flats: Image data
  A.8 | Commands: Manipulations

- **median**
  4.11.3 | Abline command: Line parameter estimation
  A.8 | Commands: Manipulations

- **meta-character**
  2.2 | Starting Figaro
  3.1.5 | Parameters: Syntax conflicts

- **min**: see parameter extrema

- **monolith**
  F.1 | Programming
  G.4 | Figaro 3.2-3 news item

N

- **NAG**
  4.6.1 | FFT: Complex data structures
  G.1.2 | Changes: General
  G.1.3 | Changes: Ported applications
  G.5 | Figaro 5.0 release
  SUN/28: NAG — Mathematical Subroutine Library

- **NDF**
  3.2 | Internals of a data file
  G.1.2 | Changes: General
  G.1.3 | Changes: Ported applications
  G.1.8 | Changes: Old data access routines
  G.3 | Figaro 3.2 news item
  G.5 | Figaro 5.0 release
  G.6 | Figaro 5.1-3 release
  SUN/33: NDF — Subroutine library for accessing NDF data structures
  SSN/20: Adding format conversion to the NDF data access library

- **NDF section**
  3.2.3 | Internals of a data file: NDF sections
  G.6 | Figaro 5.1-3 release
  SUN/33: NDF — Subroutine library for accessing NDF data structures

- **NDPROGS**
  SUN/19: NDPROGS — N-D image manipulation programs (FIGARO)

- **National Figaro**
  G.1 | Changes
• named parameter
  2.4 Looking at a spectrum
  3.1.2 Parameters: On the command line

• negated-name parameter
  3.1.2 Parameters: On the command line

O

• on-line help: see help

• origin
  3.2.2 Internals of a data file: NDF format
  SUN/33: NDF — Subroutine library for accessing NDF data structures
  SUN/95: KAPPA—Kernel application package

P

• PAR: see (A)PAR and (F)PAR

• PDA
  4.6.1 FFT: Complex data structures
  4.6.4 FFT: Taking the Fourier transform
  G.1.2 Changes: General
  G.1.3 Changes: Ported Applications
  G.5 Figaro 5.0 release
  SUN/194: PDA — Public Domain Algorithms’ library

• PGPLOT
  G.1.2 Changes: General
  G.1.3 Changes: Ported Applications
  G.1.6 Changes: Parameters and variables
  G.1.8 Changes: Old data access routines
  G.1.9 Changes: High-level subroutines
  G.5 Figaro 5.0 release
  SUN/15: PGPLOT — Graphics subroutine library

• PRIMDAT
  G.1.2 Changes: General
  G.1.8 Changes: Old data access routines
  SUN/39: PRIMDAT — Processing of primitive numerical data

• PSX
  G.1.2 Changes: General
  SUN/121: PSX — POSIX interface routines

• Portable Figaro: see Figaro

• PostScript
  2.7 Paper copies
  SUN/9: \LaTeX — A document preparation system

SUN/131: Printing on a PostScript laser printer
SUN/164: PSMERGE—Encapsulated PostScript file manipulation
SUN/184: A2PS — ASCII to PostScript formatter

• Programmer’s Guide
  F.1 Programming

• par2ifl
  F.1 Programming

• parameter
  2.3 Getting help
  2.4 Looking at a spectrum
  2.5 Image display in monochrome
  2.6 Looking at an image
  B.2 Parameters in Specdre 3.1 Parameters

• parameter extrema
  2.5.2 Looking at an image: Advanced use of image
  3.1.3 Parameters: Prompted
  G.4 Figaro 3.2-3 news item

• parameter system (see also PAR)
  2.3 Getting help
  3.1 Parameters

• polynomial
  4.1.2 Flats: Spectral data
  4.4 Spiketra
  4.5 Fluxing
  4.6.8 FFT: Cross correlation
  4.7 S-distortion
  4.10 Arc command
  4.11.2 Abline command: Running abline
  4.12.2 Gauss command: Running the program
  4.13 Échelle reduction
  A.8 Commands: Manipulations
  A.16 Commands: Fluxing

• positional parameter
  3.1.2 Parameters: On the command line

• printed documentation
  2.3 Getting help

• procedure
  2.2 Starting Figaro

• programming
  F.1 Programming

• prompt
  2.5 Looking at an image
  3.1.3 Parameters: Prompted

• ps_l, ps_p, pscol_l, pscol_p
  2.7 Paper copies
Q

- quality
  - Files
    - Figaro 5.0 release
    - Figaro 5.0-1 news item
  - quotes, double quotes, back-quotes
    - Parameters: Prompted
    - Parameters: Syntax conflicts
    - Internals of a data file: HDS files
  - Programming

R

- RTL
  - Changes: Other libraries
    - Figaro 5.0 release
- redshift
  - Figaro 5.0 release
- references
  - for Specdre
- reset
  - Changes: Parameters and variables
  - Parameters: On the command line
- reverse axis
  - Flats: Spectral data
  - Wavelengths: The arc command
  - Commands: Arithmetic
  - Commands: Manipulations
- run-time help
  - Getting help
  - Parameters: Prompted

S

- S distortion
  - S-distortion
  - Commands: Distortion
- SGS
  - Changes: General
  - Changes: Ported applications
  - Figaro 5.0 release
  - SGS — Simple graphics system
  - ADAM — graphics programmers guide
- Specdre
  - Commands: Spectroscopy data reduction
  - Demo
  - Extension
  - Introduction
  - Parameters
  - Graphics
  - Slicing data sets
  - Cube manipulation
  - Spectral fits
  - Arc spectrum axis calibration
  - References
  - Changes: Ported applications
  - Figaro 5.0 release
- Starlink
  - Unix setup
  - Unix and Starlink
  - The Starlink project
  - Starlink software collection
  - Starlink software on Unix
  - Starlink software organization on Unix
  - script: see procedure
  - scrunching
    - FFT: Creating a complex data structure
    - Wavelengths: Linear wavelength scales
    - Wavelengths: Two-dimensional scrunching
    - Échelle reduction
    - Commands: Wavelengths
    - Commands: Manipulations
    - in Twodspec
    - Figaro 3.2 news item
  - section: see NDF section
  - setorigin
    - Internals of a data file: NDF format
  - Unix — An introduction
  - Showme
    - Unix setup
    - Starting Figaro
    - Parameters: Prompted
    - Parameters: Syntax conflicts
    - Unix — An introduction
  - shell
    - Unix setup
    - Starting Figaro
    - Getting help via Mosaic
    - Figaro 5.1-3 release
    - HTX—Hypertext CrossReference Utilities
  - slicing data sets
    - Slicing through images and cubes
• wavelength calibration
  4.10 Arc command
  4.13 Echelle reduction
  4.8 Wavelengths
  B.7 in Speqred
  C.3 in Twodspec
  A.4 Commands: Wavelengths

• xwindows: see xw
• xwindows2: see x2w

Y
• ystart/yend
  2.5 Looking at an image

X
• X display
  2.4 Looking at a spectrum
  2.5 Looking at an image

• Xadam
  G.8 Figaro 5.1-6 release

• Xdefaults
  2.4 Looking at a spectrum
  G.2 Figaro 3.1-1 news item

• x2w (see also xw)
  2.5 Looking at an image
  G.2 Figaro 3.1-1 news item
  SUN/57: GNS — Graphic workstation name service
  SUN/130: GWM — X graphics window manager

• x2windows: see x2w
• xdestroy: see xmake
• xdisplay
  2.4 Looking at a spectrum
  2.5 Looking at an image
  SUN/129: TPAU — The Peter Allan Utilities

• xhost
  2.4 Looking at a spectrum

• xmake
  2.4 Looking at a spectrum
  SUN/130: GWM — X graphics window manager

• xstart/xend
  G.1.6 Changes: Parameters and variables
  2.5 Looking at an image

• xw
  2.4 Looking at a spectrum
  2.5 Looking at an image
  G.2 Figaro 3.1-1 release
  SUN/57: GNS — Graphic workstation name service
  SUN/130: GWM — X graphics window manager