

---

# KUPLOT

## Users Guide

### Version 6.20

---

written by

**Thomas Proffen**  
Email: tproffen@ornl.gov  
**Reinhard Neder**  
Email: reinhard.neder@krist.uni-erlangen.de

---

<http://tproffen.github.io/DiffuseCode>

---

*Document created: December 2, 2025*

# Preface

## Disclaimer

The KUPLOT software described in this guide is provided without warranty of any kind. No liability is taken for any loss or damages, direct or indirect, that may result through the use of KUPLOT. No warranty is made with respect to this manual, or the program and functions therein. There are no warranties that the programs are free of error, or that they are consistent with any standard, or that they will meet the requirement for a particular application. The programs and the manual have been thoroughly checked. Nevertheless, it can not be guaranteed that the manual is correct and up to date in every detail. This manual and the KUPLOT program may be changed without notice. KUPLOT is intended as a public domain program. It may be used free of charge. Any commercial use is, however, not allowed without permission of the authors.

## Acknowledgments

The routines for the line editing were taken from the program GNUPLOT. KUPLOT uses now the *PGPLOT* library written by T.J. Pearson, California Institute of Technology.

# Contents

<b>1</b>	<b>Introduction</b>	<b>6</b>
1.1	What is KUPLOT ? . . . . .	6
1.2	Getting started . . . . .	7
<b>2</b>	<b>Plotting 1D data</b>	<b>9</b>
2.1	File formats . . . . .	10
2.2	Making a nicer plot . . . . .	13
2.3	Fonts and special characters . . . . .	15
2.4	Drawing bonds . . . . .	16
2.5	Filling of areas . . . . .	18
2.6	Saving data sets . . . . .	18
2.7	Printing and exporting the plot . . . . .	19
<b>3</b>	<b>Plotting 2D data</b>	<b>20</b>
3.1	File formats . . . . .	20
3.2	Customizing the plot . . . . .	23
3.3	Using bitmaps . . . . .	24
3.4	Saving data . . . . .	25
<b>4</b>	<b>Plotting 3D data</b>	<b>28</b>
4.1	File formats . . . . .	28
4.2	Transformation of the view direction . . . . .	29
4.3	Saving data . . . . .	31
<b>5</b>	<b>Using frames</b>	<b>34</b>
5.1	Introduction . . . . .	34
5.2	Example 1: Using two different y-axes . . . . .	36
5.3	Example 2: Connected view graphs . . . . .	38
5.4	Example 3: Advanced frame usage . . . . .	40
<b>6</b>	<b>Using the mouse</b>	<b>43</b>
6.1	Mouse interface . . . . .	43
6.2	Mouse in macros . . . . .	44

<b>7 Manipulating and analyzing data</b>	<b>46</b>
7.1 Simple data calculations . . . . .	46
7.2 Calculating functions . . . . .	47
7.3 Data manipulation . . . . .	47
7.3.1 Data smoothing . . . . .	47
7.3.2 Sorting data . . . . .	49
7.3.3 Rebinning data . . . . .	49
7.3.4 Matching and merging data sets . . . . .	49
7.4 Data manipulation using variables . . . . .	49
7.5 Data analysis . . . . .	51
7.6 Fourier transform . . . . .	52
7.7 Other functions . . . . .	53
<b>8 Least square fitting</b>	<b>54</b>
8.1 Fitting 1d data sets . . . . .	54
8.2 Fitting 2d data sets . . . . .	57
8.3 User defined fit functions . . . . .	61
<b>A KUPLOT commands</b>	<b>62</b>
A.1 Summary . . . . .	62
A.2 News . . . . .	63
A.3 manual . . . . .	67
A.4 2dm . . . . .	67
A.5 achx/y/z . . . . .	69
A.6 adt . . . . .	70
A.7 alloc . . . . .	73
A.8 angle . . . . .	73
A.9 aver . . . . .	73
A.10 bond . . . . .	73
A.11 branch . . . . .	74
A.12 buffer . . . . .	74
A.13 branch . . . . .	74
A.14 ccalc . . . . .	74
A.15 close . . . . .	75
A.16 color . . . . .	75
A.17 convolute . . . . .	76
A.18 cost . . . . .	76
A.19 cmap . . . . .	76
A.20 derivative . . . . .	77
A.21 dsave . . . . .	77
A.22 ecolor . . . . .	77
A.23 etype . . . . .	77
A.24 excl . . . . .	78
A.25 fill . . . . .	78
A.26 fit . . . . .	78

A.27 fname . . . . .	87
A.28 font . . . . .	88
A.29 fset . . . . .	89
A.30 fft . . . . .	90
A.31 fourier . . . . .	90
A.32 frames . . . . .	90
A.33 function . . . . .	92
A.34 glat . . . . .	92
A.35 grid . . . . .	92
A.36 hart . . . . .	93
A.37 hcolor . . . . .	93
A.38 hlab . . . . .	93
A.39 hline . . . . .	93
A.40 hpackage . . . . .	94
A.41 hpak . . . . .	94
A.42 htype . . . . .	94
A.43 ident . . . . .	94
A.44 integral . . . . .	95
A.45 ksave . . . . .	95
A.46 kcalc . . . . .	97
A.47 kpara . . . . .	97
A.48 kpar_par . . . . .	98
A.49 lart . . . . .	98
A.50 lcolor . . . . .	98
A.51 load . . . . .	99
A.52 sm . . . . .	105
A.53 ltype . . . . .	106
A.54 lwidth . . . . .	106
A.55 mark . . . . .	106
A.56 match . . . . .	106
A.57 mcolor . . . . .	107
A.58 mean . . . . .	107
A.59 merge . . . . .	107
A.60 mouse . . . . .	108
A.61 mtype . . . . .	108
A.62 msizer . . . . .	109
A.63 nexus . . . . .	109
A.64 orientation . . . . .	110
A.65 plot . . . . .	110
A.66 print . . . . .	110
A.67 ptype . . . . .	111
A.68 rdef . . . . .	111
A.69 rebin . . . . .	111
A.70 reset . . . . .	111
A.71 rvalue . . . . .	112

A.72 sann . . . . .	112
A.73 save . . . . .	112
A.74 sdef . . . . .	113
A.75 show . . . . .	113
A.76 sleg . . . . .	115
A.77 smax . . . . .	115
A.78 smooth . . . . .	115
A.79 sort . . . . .	116
A.80 spline . . . . .	116
A.81 scale . . . . .	116
A.82 skal . . . . .	116
A.83 tit1/2 . . . . .	116
A.84 torefine . . . . .	117
A.85 window . . . . .	117
A.86 variables . . . . .	117
A.87 errors . . . . .	118

# Chapter 1

## Introduction

### 1.1 What is KUPLOT ?

KUPLOT is the interactive plotting section of the Discus\_Suite program. It is controlled by the same command language as all other parts of the program.

KUPLOT is now using the *PGPLOT* library for the actual plotting and supports all graphic devices supported by *PGPLOT* such as X-window terminal or POSTSCRIPT output. As mentioned before, the program is controlled by a command language which includes a FORTRAN style interpreter which allows the use of variables, loops and conditional i statements. This results in a high degree of flexibility and allows the creation of quite complex graphics. It also allows KUPLOT to be used to process large numbers of data files and produce the desired plots automatically.

KUPLOT can process simple 1D, 2D and 3D data files. The program supports normal line graphs, marker, error bars as well as spline interpolation between the data points. Line color, marker color, line type, line width and various other parameters can be adjusted. KUPLOT allows one to plot 3D data sets using contour lines, colored bitmaps or both. The color map for the bitmap can be freely changed using the FORTRAN interpreter (see section Variables in the package part of the manual DISCUS package ). The program also allows one to define different contour line sets for one data set, e.g. finer spaced lines for diffuse scattering and larger space lines in a different color for the Bragg peaks. A page can be divided into different plot areas (see chapter 5). Each frame can contain graphs or the contents of a text file. Frames can have different background colors. Each frame has its own parameter set like e.g. title, axis labels, fonts.

Loaded data sets can be manipulated using the FORTRAN style interpreter or a variety of build-in functions. An integrated FIT sub level, which allows to fit the following functions to a given 1D data set: polynomial, n Gaussians and n Lorentzians. Furthermore 2D data sets can be fitted using a set of 2D Gaussians. Additionally KUPLOT allows fitting a user defined function to a loaded data set.

Symbol	Description
"text"	Text given in double quotes is to be understood as typed.
<text>	Text given in angled brackets, is to be replaced by an appropriate value, if the corresponding line is used in KUPLOT . It could, for example be the actual name of a file, or a numerical value.
text	Text in type writer font exclusively refers to KUPLOT commands.
[text]	Text in square brackets describes an optional parameter or command. If omitted, a default value is used, else the complete text given in the square brackets is to be typed.
{text   text}	Text given in curly brackets is a list of alternative parameters. A vertical line separates two alternative, mutually exclusive parameters.

Table 1.1: Used symbols

## 1.2 Getting started

After the program KUPLOT is installed properly and the environment variables are set, the program can be started by typing: 'kuplot' at the operating systems prompt. Information about how to compile and install the program is described in detail in the DISCUS package reference guide. All array sizes that might need to be adjusted to your needs are defined in the file '*kuplot.inc*'. Check the comments within this file for an explanation of the various variables.

The program uses a command language to interact with the user. The command 'exit' terminates the program and returns control to the shell. All commands of KUPLOT consist of a command verb, optionally followed by one or more parameters. All parameters must be separated from one another by a comma ','. There is no predefined need for any specific sequence of commands. KUPLOT is case sensitive, all commands and alphabetic parameters MUST be typed in lower case letters. If KUPLOT has been compiled using the '-DREADLINE' option (see installation files) basic line editing and recall of commands is possible. For more information about the command language refer to the DISCUS reference manual or check the online help. A list of variables specific to KUPLOT is given in Tab. 1.2.

Names of input or output files are to be typed, as they will be expected by the shell. If necessary include a path to the file. Only the first four letters of any command verb are significant, all commands may be abbreviated to the shortest unique possibility. At least a single space is needed between the command verb and the first parameter. No comma is to precede the first parameter. A line can be marked as comment by inserting a '#' as first character in the line. The symbols used throughout this manual to describe commands, command parameters, or explicit text used by the program KUPLOT are listed in Table 1.1.

There are several sources of information, first KUPLOT has a build in online help, which can be accessed by entering the command `help` or if help for a particular command `<cmd>` is wanted by `help <cmd>`. This manual describes background and principle functions of KUPLOT and should give some insight in the ways to use this program.

Variable	Description
n[1] n[2]* n[3]* n[4]* n[5]*	Number of loaded data sets ( <b>now writable</b> ) Maximum allowed number of data sets Maximum allowed number of frames Maximum allowed number of annotations Maximum allowed number of bonds
nx[<ik>]* ny[<ik>]* ni[<ik>]* np[<ik>]*	Number of points in x-direction for data set <ik> Number of points in y-direction for data set <ik> Type of data set <ik>. (0 for 1D and 1 for 2D data) Number of points of 1D data set <ik>
xmin[<ik>]* xmax[<ik>]* ymin[<ik>]* ymax[<ik>]* zmin[<ik>]* zmax[<ik>]* pwin[i]*	Minimum x-value of data set <ik> Maximum x-value of data set <ik> Minimum y-value of data set <ik> Maximum y-value of data set <ik> Minimum z-value of data set <ik> Maximum z-value of data set <ik> Current plotting dimensions (1:xmin,2:xmax,3:ymin,4:ymax)
x[<ik>,<ip>] y[<ik>,<ip>] dx[<ik>,<ip>] dy[<ik>,<ip>] z[<ik>,<ix>,<iy>]	X-value of data point <ip> of data set <ik> Y-value of data point <ip> of data set <ik> Value of $\sigma_x$ for data point <ip> of data set <ik> Value of $\sigma_y$ for data point <ip> of data set <ik> Z-value of point <ix>,<iy> of data set <ik>
axis[1,i] axis[2,i] axis[3,i] axis[4,i] axis[5,i] axis[6,i]	Angle of labels for axis i (1=x,2=y) Length of major ticks for axis i (1=x,2=y) Length of minor ticks for axis i (1=x,2=y) Subdivisions between ticks, axis i (1=x,2=y) Distance numbers - axis i (1=x,2=y) Distance label - axis i (1=x,2=y)
cmap[<ic>,3] cmax[1]*	Color map entry in R,G,B (0..1) for color <ic> Number of bitmap colors available
p[<i>] s[<i>]	Fit variable number <i> (see section 8) Error of p<i>

**Table 1.2:** KUPLOT specific variables

## Chapter 2

# Plotting 1D data

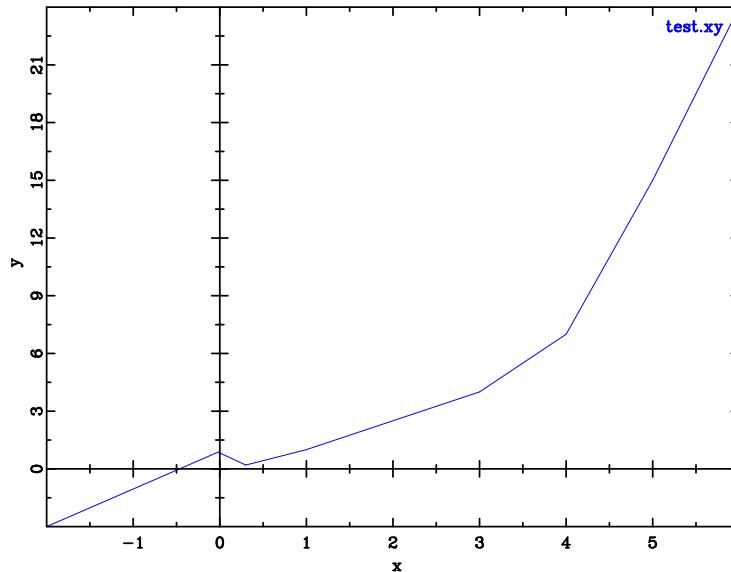
In this chapter we will learn how to read, plot and print simple 1D data sets, e.g. xy-data. Chapter 3 explains the plotting of 2D data sets. However, many of the commands to modify a plot discussed in this chapter also apply to the plotting of 2D and 3D data.

The simplest data file is a text file with x- and y-coordinates ( $x_i, y_i$ ) for each point in a separate line, as in the example data file *test.xy* listed below:

```
-2.00 -3.00
-0.02 0.87
:
:
```

In order to plot the file *test.xy*, we only need to enter the following two commands at the *KUPLOT* input prompt:

```
load xy,test.xy
plot
```



**Figure 2.1:** The first *KUPLOT* plot

Format	Description
cr	Special crystal structure format exported by <i>DISCUS</i> . Each line contains $x,y,z$ , marker type, color and marker size for the atom at the given coordinates. Note, the first two columns are used for plotting, the $z$ value is ignored (see <i>DISCUS</i> plot sublevel).
ma	Only the $x$ value of each data line is read, $y$ is set to zero. This allows the plotting of tick marks (e.g. at Bragg peak positions) at the x-axis.
sc/st	This command allows one to read scans from a SPEC file.
gs	This command allows one to read GSAS files.
xx	Each line contains only one value that is taken as $y$ value and the point number is stored as $x$ value.
xy	Each line of the data file contains $x,y$ and optionally standard deviations $\sigma_x$ and $\sigma_y$ from a multi-column text file. The default is $x, y, \sigma_x, \sigma_y$ , but the columns can also be manually assigned (see example).
csv	Allows to read quite flexible data files. The optional parameter for this format describes how you to define which columns to read, how many header lines to skip, what separator to assume between columns and whether a decimal point "." or a decimal comma "," is used in floating point numbers.

**Table 2.1:** Supported file formats for 1D data

The `load` command reads the specified data file, which is in our case of the type 'xy'. The command `plot` finally displays the plot on the screen. The resulting plot for this example can be seen in Figure 2.1. The following sections of this chapter describe the supported file formats and how to change, print and save the resulting plots.

## 2.1 File formats

*KUPLOT* reads data sets using the 'load' command. The simplest file format is a text file containing values for  $x$  and  $y$  on a separate line for each data point as in the previous example. To read more than one data set just repeat the `load` command. The maximum number of data sets that can be read and the maximum total number of data points are defined in the file *kuplot.inc* and might be adjusted before *KUPLOT* is compiled. The current limits can be displayed using the command `show config`. The `rese` command clears the currently loaded data sets and the next file is read as data set one again.

The various file formats for 1D data supported by *KUPLOT* are summarized in Table 2.1. The file format is specified as first parameter for the `load` command followed by the name of the file to be read. Note that the 'xy' and 'yx' formats may contain standard deviations  $\sigma_x, \sigma_y$  for each data point which may be used to plot error bars using the `etyp` command in *KUPLOT*. Optional parameters for the command `load xy` allow one to assign columns in the data file to  $x, y$  and  $\sigma_x, \sigma_y$ . The following command for example will assign column 2 to  $x$ , column 4 to  $y$  and column 9 to  $\sigma_x$  and 12 to  $\sigma_y$ :

```
load xy,datafile.dat,2,4,9,12
```

After a data set is read, *KUPLOT* displays the number of points read and the x- and y-limits. In our previous example, the screen output after entering the `load xy,test.xy` looks like that:

```
Reading 2 columns ...
Data set no.: 1

Filename : test.xy                               (    8 points )
Range x : -2.000      to     6.000
Range y : -3.000      to     24.00
```

Since our example file contains no  $\sigma$  values, *KUPLOT* can only read two columns. The file is associated with data set 1. This data set number is subsequently used to alter the appearance of the plot for a given data set (see section 2.2).

## Reading SPEC files

As mentioned earlier, *KUPLOT* can read so called SPEC files. Basically a SPEC file can contain several scans, each consisting of multiple columns. One can use *KUPLOT* to get basic information about the file contents: `load st,spec.dat` will simply list all scans in the file, but load no actual data. The command `load st,spec.dat,1` will list the columns available in scan number one. Again no actual data are loaded. An example output is shown here:

```
kuplot > load st,npdf_02305.sqa
Reading file npdf_02305.sqa ..
List of scans in file npdf_02305.sqa :
 6536 pts --> # 1 Bank at 46.00 degrees
 6536 pts --> # 2 Bank at 90.00 degrees
 6536 pts --> # 3 Bank at 119.00 degrees
 6536 pts --> # 4 Bank at 148.00 degrees

kuplot > load st,npdf_02305.sqa,2
Reading file npdf_02305.sqa ..
-----
Scan    2 in file npdf_02305.sqa :
-----
Data field  1 : Q
Data field  2 : S
Data field  3 : sigmaS
Data field  4 : F
Data field  5 : sigmaF
```

If we now want to read  $Q$ ,  $F(Q)$  and  $\sigma(F(Q))$  from bank or scan number two, we simply need to specify the desired columns after the scan number as shown here:

```
kuplot > load st,npdf_02305.sqa,2,1,4,0,0,5
Reading file npdf_02305.sqa ..
-----
Scan    2 in file npdf_02305.sqa :
-----
Axis x is      :          Q (# 1)
Axis y is      :          F (# 4)
Error column dy : sigmaF (# 5)
...
...
```

The scan number is 2 and columns 1 and 4 are assigned to  $x$  and  $y$ . The next parameter specifies a column used to normalize the  $y$  value, e.g. a measuring time. A value of zero tells *KUPLOT* to ignore this parameter. The last two numbers are the columns for  $\sigma_x$  and  $\sigma_y$  and again the zero

for  $\sigma_x$  indicates that no errors for  $x$  should be read and are in fact not present in our example file. The scan number can also be specified as `all` in which case all scans preset in the file are loaded as separate datasets into *KUPLOT*.

### Reading GSAS files

GSAS is a structure refinement program and we refer to powder diffraction data files in a format compatible with that program as GSAS files. This format is particularly common for time-of-flight neutron powder diffractometers. Similar to a SPEC file, it contains data for several scans or more commonly called banks. Similar to before, the command `load gs,file.gsa` will list the banks present in the file. Once a bank number or `all` is added to the command, the corresponding data are read as in this example reading data from bank 1:

```
load gs,file.gsa,1
```

Additional optional parameters allow one to select the units for the  $x$ -axis. Valid units are time-of-flight (`t`), d-spacing (`d`), momentum transfer Q (`q`) and wavelength (`l`). In addition data can be normalized by the incident neutron spectrum in case of spallation neutron data. Most of these conversions require a so-called instrument parameter file which is also needed to run GSAS itself. Let us look at an example

```
load gs,file.gsa,all,d,iparm.dat,norm
```

Here we read all banks from the file `file.gsa` and convert the  $x$ -axis to d-spacing. The instrument parameter file is `iparm.dat` and the last parameter `norm` causes the data to be normalized by the incident spectrum. As a side note, the incident spectrum is calculated from parameters given in the instrument parameter file. Refer to the GSAS documentation for more details.

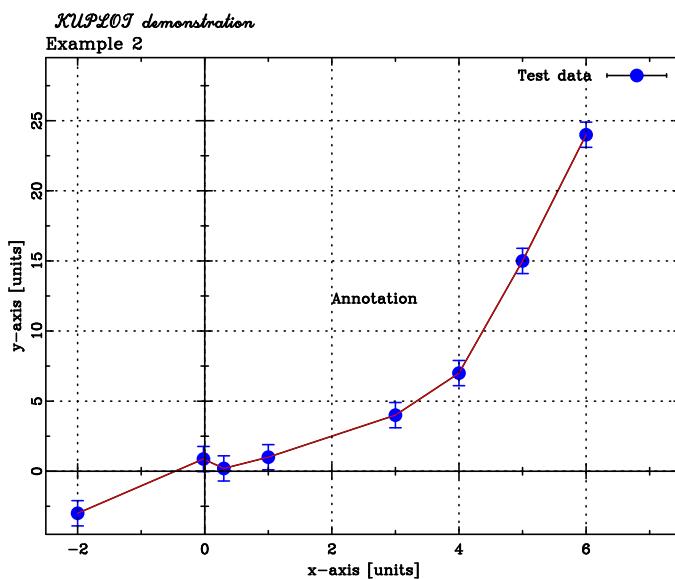


Figure 2.2: Customized *KUPLOT* plot

## 2.2 Making a nicer plot

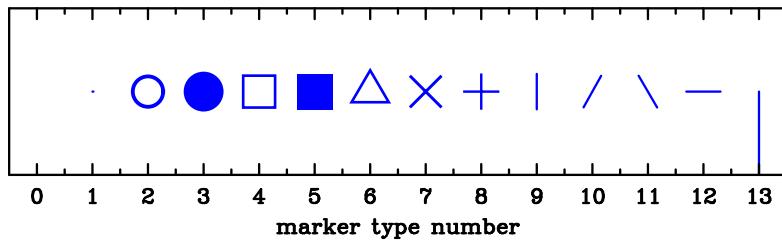
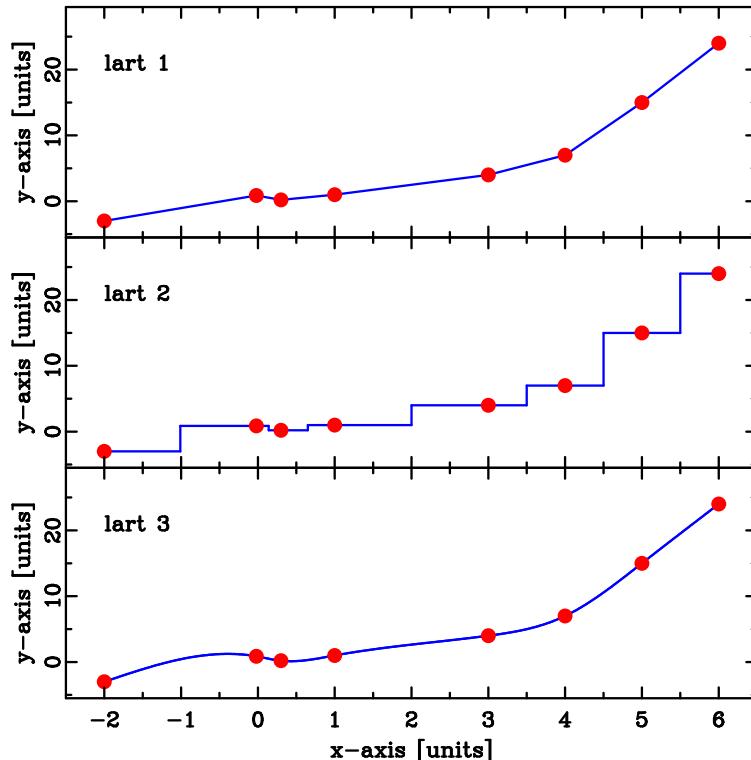
In the example shown in Figure 2.1 we have simply used the defaults of *KUPLOT* and plotted the data set directly. The program *KUPLOT* offers a variety of commands to alter the appearance of the plot itself and the representation of each loaded data set. We are using the same data set *test.xy* as in the previous example (with added  $\sigma$  values) and create the plot shown in Figure 2.2. The macro file used to create this plot is listed below. Note, that the line numbers are added for easy reference within this manual and are not part of the actual macro file.

```

1  load xy,test.xy
2  #
3  achx x-axis [units]
4  achy y-axis [units]
5  tit1 \fs KUPLOT demonstration
6  tit2 Example 2
7  #
8  grid on
9  fnam off
10 #
11 skal -2.5,7.5,-4.5,29.5
12 mark 2,5
13 #
14 lcol 1,6
15 lwid 1,0.5
16 mtyp 1,3
17 mcol 1,3
18 msiz 1,0.4
19 etyp 1,2
20 #
21 sleg 1,"Test data"
22 sann 1,"Annotation",2,12
23 plot

```

In line 1 of this macro we read the data set from file *test.xy* as in the previous example. The axes labels and the text for the first and second title line are set in lines 3–6. The grid of dashed lines at positions of the major tick marks is enabled in line 8 and the plotting of the filenames corresponding to the data sets in the upper left corner of the plot is switched off (line 9). Next we define the extend of the plot (line 11) to be from -2.5 to 7.5 in the x-direction and from -4.5 to 29.5 in the y-direction. The tick mark interval is set to 2.0 and 5.0 (line 12). All these settings affected the complete plot, whereas the following commands act on data set 1, which is given as the first parameter to all commands in lines 14–19. First the line color is set to black (line 14). The first six colors are coded as the 6 default pen colors on a HP plotter, i.e. red, green, blue, purple, yellow and black. Those default colors can be redefined using the command *color*. In the following lines, the line width, marker type, marker color and marker size are defined (lines 16–18). The different marker types supported by *KUPLOT* are shown in Figure 2.3. In addition to the markers shown, *KUPLOT* can access all markers defined in the *PGPLOT* library. The marker number is simply 100 plus the *PGPLOT* marker code. Refer to the *PGPLOT* documentation for a list of those markers. Error bars in y-direction based on the  $\sigma_y$  values read from the input file *test.xy* are enabled in line 19. Marker types -1, -2 and -3 will plot the  $x, y$  or  $x, y$  coordinates instead of a marker symbol. The caption "Test data" is defined in line 21 and the annotation "Annotation" at the coordinates (2.0,12.0) is specified by the *sann* command in line 22. Note, the value 1 in line 30 stands for the first annotation and not for data set one. Finally the plot is displayed on the screen (line 23). In this example, the data points  $(x_i, y_i)$  were

Figure 2.3: *KUPLOT* marker typesFigure 2.4: *KUPLOT* plotting styles

simply connected by a straight line. *KUPLOT* offers two alternative plotting styles: a histogram style and cubic spline interpolation. The style is selected by the `lart` command followed by the data set number and the desired style. Style 1 is the default and connects the data points by a straight line as in the previous example (`lart 1, 1`). This style is shown in Figure 2.4 in the bottom view graph. The histogram style 2 is shown in the middle view graph of the same Figure. Here each data point  $(x_i, y_i)$  sits in the center of a histogram step. The command to change the plotting style of data set 1 to histogram would be `lart 1, 2`. The third mode (`lart 1, 3` for data set 1) is cubic spline interpolation. The interpolated spline goes through all data points  $(x_i, y_i)$  and has a continuous first derivative. The spline interpolation of the data set `test.xy` can be seen as top view graph in Figure 2.4. The default number of interpolated points for a plot is 500. However, the value, determined by the variable `MAXSP` might be altered in

Command	Function
\u	start superscript, or end subscript
\d	start sub-, or end superscript
\b	backspace (draw next character on top of current one)
\fn	switch to normal font (1)
\fr	switch to roman font (2)
\fi	switch to italic font (3)
\fs	switch to script font (4)
\\\	backslash
\x	multiplication sign ×
\.	centered dot .
\A	Å

Table 2.2: Text control characters

the file *kuplot.inc* before *KUPLOT* is compiled.

## 2.3 Fonts and special characters

The program *KUPLOT* offers the user various ways to alter the appearance of the text parts of the plot. All these functions are accessed by the command `font`. If no parameters are given, the current settings will be displayed on the screen, as shown below:

```
Font settings for frame no.: 1
Overall font scaling factor : 2.50

Font where ?      size   fontname      font-id   color
-----
1 Main title line    16. Roman        2          6
2 Subtitle line     14. Roman        2          6
3 Axis labels       12. Roman        2          6
4 Numbers at axis   12. Roman        2          6
5 Text in text frame 12. Standard    1          6
6 Filename & caption 12. Roman        2          6
```

As can be seen from the output, there are six different types of fonts, the main and second title line, the axis labels, the axis numbering, text in a text frame (see section 5) and finally the font used for annotations and captions. Each of those types is associated with a font size given in points, a font style given as name (e.g. Roman) and number (e.g. 2) and finally a color in our example pen number 6 or black. The four font styles available in *KUPLOT* are shown in Figure 2.5. *KUPLOT* allows the user to apply an overall scale factor to the font size or change font sizes individually. As an example let us change the color of the first title line to red. This would be done using the command:

```
font col,1,1
```

The first parameter tells *KUPLOT* what we want to change, here the color. Next we have the number of the font type, here 1 for the first title line. Finally we specify the desired color, here pen 1 for red. One benefit of using the *PGPLOT* library is that the user can access special

alpha	\ga	$\alpha$	\gA	$A$	beta	\gb	$\beta$	\gB	$B$
gamma	\gg	$\gamma$	\gG	$\Gamma$	delta	\gd	$\delta$	\gD	$\Delta$
epsilon	\ge	$\epsilon$	\gE	$E$	zeta	\gz	$\zeta$	\gZ	$Z$
theta	\gh	$\theta$	\gH	$\Theta$	iota	\gi	$\iota$	\gI	$I$
kappa	\gk	$\kappa$	\gK	$K$	lambda	\gl	$\lambda$	\gL	$\Lambda$
mu	\gm	$\mu$	\gM	$M$	nu	\gn	$\nu$	\gN	$N$
xi	\gc	$\xi$	\gC	$\Xi$	omicron	\go	$\o$	\gO	$O$
pi	\gp	$\pi$	\gP	$\Pi$	rho	\gr	$\rho$	\gR	$P$
sigma	\gs	$\sigma$	\gS	$\Sigma$	tau	\gt	$\tau$	\gT	$T$
upsilon	\gu	$\upsilon$	\gU	$\Upsilon$	phi	\gf	$\phi$	\gF	$\Phi$
chi	\gx	$\chi$	\gX	$X$	psi	\gq	$\psi$	\gQ	$\Psi$
omega	\gw	$\omega$	\gO	$\Omega$					

**Table 2.3:** Access to Greek characters

\fn	KUPLOT normal font
\fr	KUPLOT roman font
\fi	KUPLOT italic font
\fs	KUPLOT script font

**Figure 2.5:** KUPLOT fonts

characters and commands within all text lines, e.g. titles, axis labels and text in text frames. The supported control commands are listed in Table 2.2 and a list of Greek characters is given in Table 2.3. For example to create an axis label  $\text{Å}^{-2}$  the text input would read  $\backslash A\text{\textit{u-2}}\text{\textit{d}}$ . Note that sub- and superscript always have to be given in pairs. These examples can only give a brief introduction in the different commands of *KUPLOT* allowing the user to alter the appearance of the plot. For details on these commands and their parameters refer to the online help, the command reference or the interactive tutorial.

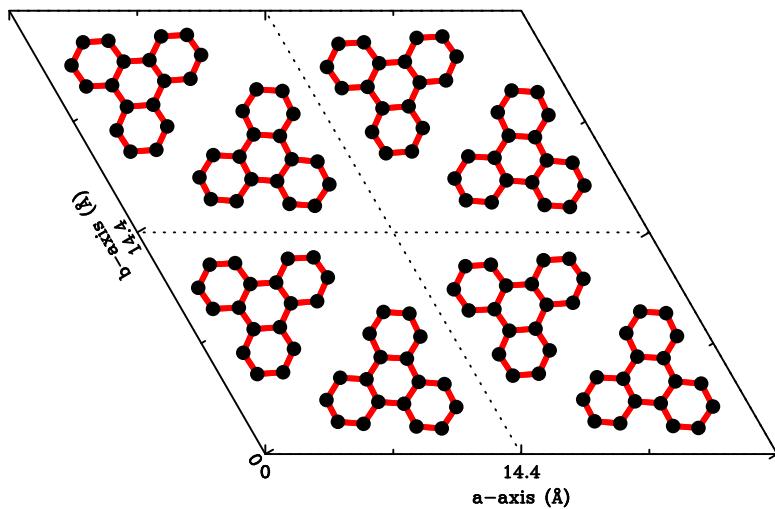
## 2.4 Drawing bonds

The program *KUPLOT* allows the user to connect points in the plot that are separated by a given distance. The distance is specified relative to the current x-axis. To calculate the distances, the aspect ratio and the angle between the axes is used. These values can be defined by the user via the commands `aver` and `angl` as we will discuss in somewhat more detail in section 3.2. Consequently you will get the desired connection only for the correct aspect ratio and angle between the axes. This feature can be used to include bonds when plotting a structure file exported by *DISCUS*. An example for the compound PHTP (Perhydrotriphenylen) is shown in Figure 2.6. The macro file used to create this picture is listed below:

```

1  rese
2  load cr,phtp.cr
3  #

```



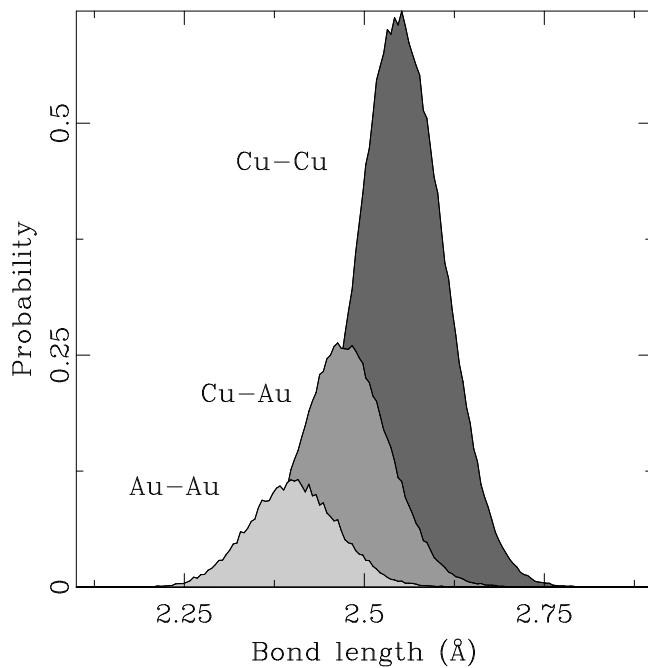
**Figure 2.6:** Example for plotting bonds

```

4  skal 0.0,28.8,0.0,28.8
5  mark 14.4,14.4
6  #
7  aver 1.0
8  angl 120.0
9  #
10 msiz 1,0.35
11 grid on
12 fnam off
13 #
14 achx a-axis (\AA)
15 achy b-axis (\AA)
16 #
17 bond 1,1.4,0.2,1,1,1.5
18 #
19 plot

```

First we reset the program and read the data (line 2). In line 4 the plot area is defined followed by the setting for the tick mark interval (line 5). In our case the lattice parameter is  $a = b = 14.4\text{\AA}$ . The command `aver` defines the ratio between units on the y- and x-axis. Since both are in  $\text{\AA}$ , the desired ratio is one. The angle between the axes is set to  $120^\circ$  (line 8). Next we set the marker size (line 10), turn the plotting of the grid on (line 11) and disable the plotting of the filenames (line 12). The axes labels are set in lines 14 and 15. So far nothing new, now comes the command `bond` that defines the bonds between carbon atoms we like to plot. The first parameter 1 is the number of the bond definition. *KUPLOT* allows one to define multiple distances. The second and third parameter set the distance to  $1.4\text{\AA} \pm 20\%$ . The last optional parameters in line 17 set the bond color to red (pen 1), select a solid line (type 1) and set the line width to 1.5. All that remains is to plot the result (line 19). One should be aware that *KUPLOT* calculates the distances between *all* data points and compares them to the selected distance interval which can be rather slow for large data file. To disable a bond definition simply reenter the command with distance zero, e.g. `bond 1, 0.0.`



**Figure 2.7:** Example for filling areas

## 2.5 Filling of areas

Sometimes to enhance the visibility of plot, it is desired to fill the area under a data set. A plot taken from our recent book *Diffuse scattering and defect structure simulations - A cook book using the program DISCUS* published by Oxford University Press is shown in Figure 2.7.

The part setting the colors and filling options of the macro file used to create the plot is shown here.

```

color 1,0.8,0.8,0.8
color 2,0.6,0.6,0.6
color 3,0.4,0.4,0.4
#
fill 1,3,5
fill 2,2,5
fill 3,1,5

```

The command `color` sets the color for the different 'pens' of *KUPLOT*. In this case colors one through three are assigned three different shades of grey. The color is specified as red, green, blue ranging from 0 to 1. The command `fill ik,c,t` then selects filling for data set `ik` using color or pen `c` and setting the fill style to `f`. In our case style five corresponds to a solid fill with a black border. Type `help fill` for a list of all fill styles.

## 2.6 Saving data sets

The program *KUPLOT* allows the user to save loaded data sets to a file. This might be necessary after a data set was altered or in cases where only part of the data shall be saved. To save a data

set, use the command `ksav` followed by the number of the data set to be saved. After entering the `ksav` sublevel, you need to specify the output filename and the file format for the output file. In this section we will only discuss the saving of 1D data sets and the only available file format is `xy` (see previous example). The following commands will save the loaded data set 1 (e.g. `test.xy` from the example above):

```

1  ksav 1
2  outfile export.xy
3  form xy,0.0,5.0
4  show
5  run

```

In line 1 we enter the `ksav` sublevel. The parameter specifies the data set we want to save, here data set 1. Next the output filename is set to `export.xy` (line 2). The file format is set to `xy` and the optional further parameters give x-limits (line 3), i.e. only data points with x-values between 0.0 and 5.0 will be saved. Note, that the default range are defined by the current plot window limits. Thus if the plot window set by the command `skal` is smaller than the extend of the actual data set, only point that are visible in the current plot are saved if no limits are given with the `form` command. The command `show` (line 4) lists the current limits and finally the data are written to the specified file via the `run` command (line 5). The command `run` also exits the `ksav` sublevel. To exit without actually writing a file use the `exit` command.

The `ksav` command has many additional options to save 2D data such as format conversion and the export of cross sections. These functions are discussed in chapter 3.4.

## 2.7 Printing and exporting the plot

Viewing the plot on the screen is one thing, but we certainly need to print plot at some stage or save it for e.g. import in a text processing program. *KUPLOT* supports different output formats that can be saved or used for printing. The defaults are POSTSCRIPT and PNG associated with the parameters `ps` and `png` respectively. However, they can be set to any device supported by *PGPLOT* by altering the variables `dev_name` in the file `blk_appl.f` of the *KUPLOT* distribution. The variable `dev_prn` in the same file sets the default print command. To print the current plot to the default printer, use `prin ps`. To reach other printers, the corresponding print command can be specified as a second parameter to the `prin` command as in the example below:

```
print ps,"lp -d myprinter -h "
```

Here the printer `myprinter` is used. Check local documentation or ask your system administrator how to access the desired printer. In order to save the graphics file rather than printing it, use the `save` command with the first parameter being `ps`, `png` or `pdf` for POSTSCRIPT, PNG or PDF output respectively. The second parameter is the name of the output file, e.g. `save ps,plot.ps`.

# Chapter 3

## Plotting 2D data

In the previous chapter we have learned about using *KUPLOT* to plot 1D data sets. Now we will extend the usage of *KUPLOT* to 2D data sets, i.e. xyz-data. The coordinates  $x$  and  $y$  are within the drawing plane and the  $z$  values are represented by contour lines at given  $z_c$  values, by the bitmap color or both. Changing the general appearance of the plot such as defining title lines, labels or tick mark intervals was described in section 2.2 of this manual and works exactly the same way when plotting 2D data sets.

### 3.1 File formats

As described in section 2.1 for 1D data sets, 2D data sets are read using the command `load`. Multiple data sets might be read by repeating the command `load` and 1D and 2D data sets might be imported in any combination. The command `rese` clears the currently loaded data sets and the next file is read as set number one again.

A summary of the supported 2D file formats is given in Table 3.1. All file formats are "normal" text files that can be viewed and edited with every text editor. Since these files are normally larger in size compared to a binary version, archived files might be compressed with standard UNIX tools like *compress* or *gzip*. The standard file format for 2D data is the so-called NIPL format which was developed at the Institut für Mineralogie und Kristallographie in München. The NIPL format (`ni`) is defined as follows:

```
1      nx ny
2      xmin xmax ymin ymax
3ff    z z z z ...
```

The first line contains the number of data points in x- and in y-direction. The next line gives the x- and y-limits of the plot area. All following lines contain the z-values row by row starting in the lower left corner. The z-values are real numbers and not restricted in any way. However, the value -9999.0 is reserved for excluded regions (see below).

The optional third parameter listed in Table 3.1 is the name of a file containing excluded regions, i.e. areas of the data file that should be excluded from the plot. An example is shown in Figure 3.1. The plot on the bottom was created by reading the NIPL file *test.nipl* using the command `load ni, test.nipl` and thus reading all data. The data shown here are actually neutron diffuse scattering from calcium stabilised zirconia collected by R.B. Neder at the neutron source

Format	add. Parameters	Description
de	$\Delta x, \Delta y$	Reads xy-file and creates a 2D data set with the number of points that are inside a specified grid box $\Delta x, \Delta y$ as z-values.
ni	[exclude]	Reads NIPL file format (see text). The optional parameter is a file containing excluded regions.
pg		Reads PGM file format (see text).
zz	$\Delta x, \Delta y$	Reads xyz-file and bins the data to the given grid size $\Delta x, \Delta y$

**Table 3.1:** Supported file formats for 2D data

in Garching, Germany. Sometimes one wants to exclude certain regions within the data from the actual plot, in our example the strong Bragg peaks. This can be done by creating a file with coordinates of rectangles within the plotting area that should be excluded. Such a file can be created using a text editor. In our example, the excluded region file *test.excl* looks like this:

```
10
0.87  1.13  0.81  1.17
2.85  3.15  0.81  1.26
1.85  2.34  3.70  4.28
2.82  3.18  2.79  3.28
0.87  1.18  2.81  3.23
0.00  0.24  3.68  4.28
0.00  0.12  1.73  2.24
0.00  0.58  0.00  0.79
1.78  2.22  0.00  0.35
1.90  2.17  1.81  2.13
```

The first line contains the number of rectangle coordinates in the file followed by *xmin*, *xmax*, *ymin*, *ymax* for each excluded region. Rereading the file *test.nipl* using the command `load ni, test.nipl, test.excl` results in the right view graph of Figure 3.1. Note that *KUPLOT* will replace all z-values within an excluded region by -9999.0 and those values will be ignored by most *KUPLOT* functions. If a file read with excluded regions is saved later on, the data within those excluded regions are lost.

The second more general 2D file format is the PGM (portable graymap) format which is defined as part of the PNM format Jef Poskanzer. Various programs are capable of reading PNM files and the *pnmplus-package* is a freely available collection of tools and conversion programs from PNM to virtually any other graphics format. *KUPLOT* only supports ASCII PGM files which are defined as follows:

```
1      P2
2      nx ny
3      255
4ff    z z z z z
```

The code *P2* in the first line identifies the file as ASCII PGM format. The dimensions *nx* and *ny* of the data set are given in the next line. The PGM format is an integer bitmap and the number in line 3 gives the maximum depth (here 255 or 8 bit). Although the PGM format is generally not restricted to 8 bit resolution, it was found that many programs are not capable of reading PNM files with a larger resolution. Comments can be included in the file starting with a # as first character. The actual data are integer values written row by row starting (in

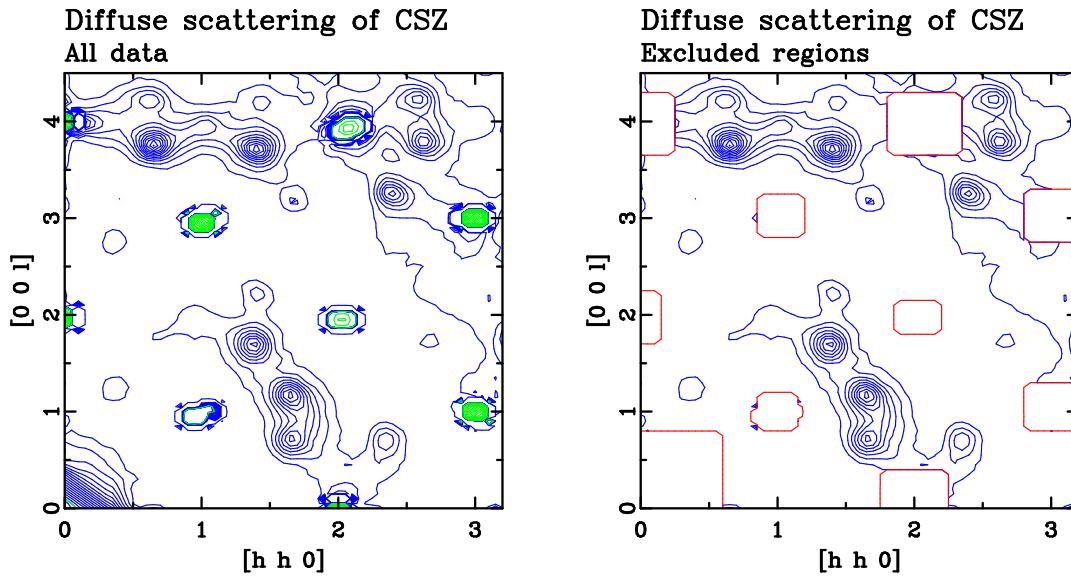


Figure 3.1: Usage of excluded regions for 2D files

contrast to NIPL files) from the top left corner. Besides being limited to integer values ranging from  $0 \rightarrow 255$ , the PGM file format does not contain any information about the  $x$  and  $y$  limits and *KUPLOT* sets  $x$ - and  $y$ -values to the pixel number. However, the  $x$ - and  $y$ -values might be manipulated (see chapter 7) and the file can be saved in NIPL format (see section 3.4), thus allowing the conversion of PGM files to NIPL files and *vice versa*.

The PGM and NIPL files are restricted to data points being on a equidistant rectangular grid. The file format 'zz' can be used to read any data file containing the data points  $(x_i, y_i, z_i)$  on a separate line. This is practically the extension of the 'xy' file format. However, *KUPLOT* can internally only work with  $z$ -values lying on a rectangular equidistant grid. Thus the read data points are binned to a grid size defined by  $\Delta x$  and  $\Delta y$  given as additional parameters to the *load* command. Each resulting grid point contains the average of all data points within the file that fall within the area of the specific grid point. If the specified grid size is too small, the resulting plot will contain empty grid points. The following command reads the XYZ file *test.xyz* and bins it to a grid size of  $\Delta x = \Delta y = 0.05$ :

```
load zz,test.xyz,0.05,0.05
```

This file format can also be used to rebin data sets to a broader grid by saving the NIPL (or PGM) file as XYZ file and rereading the file with the new desired grid size.

The file format de (for density) works similar to the previously discussed *zz* format but rather than averaging the  $z$ -values in each grid point, the resulting  $z$ -value is the number of points  $(x_i, y_i)$  falling within the corresponding grid volume which is defined as before by the additional parameters  $\Delta x$  and  $\Delta y$ . The program reads the first two real numbers in each line as  $x$ - and  $y$ -value. For example *DISCUS* can export the atom positions of all atoms without the translational part, i.e. all atoms are in one unit cell. The resulting file can be read using the *cr* format resulting in a marker for each atom. Alternatively the same file (since the first two

numbers are  $x$  and  $y$ ) can be read using the 'de' format which will result in a density plot which can be displayed either using contour lines or a bitmap.

## 3.2 Customizing the plot

The command `hart` selects the usage of contour lines, bitmaps or both and is discussed in the next section. Here we want to concentrate on commands used to change contour lines. The base level, the interval and the number of contour line levels are defined using the command `hlin`:

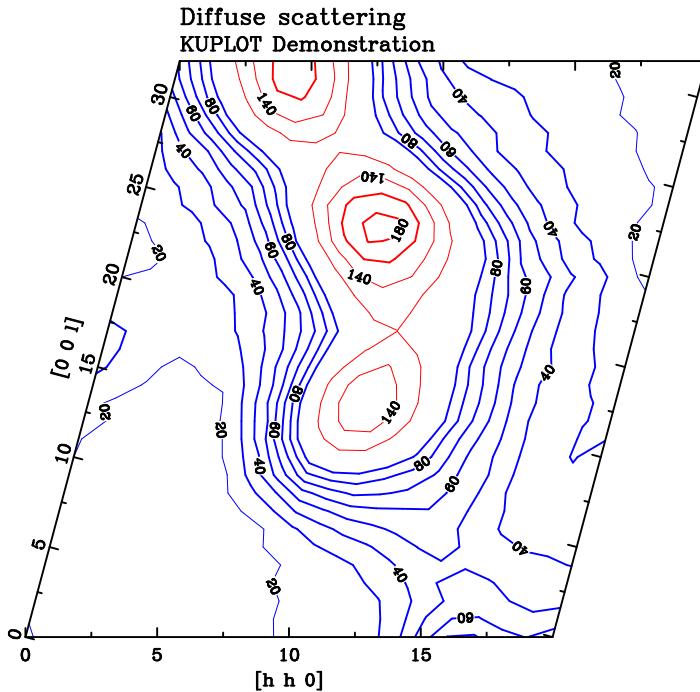
```
hlin 1,100,50,10
hlin 2,10,10,9,%
```

*KUPLOT* supports multiple sets of contour lines, e.g. one set at low levels to display diffuse scattering and one set at higher levels with a different spacing for the stronger Bragg peaks. Each set can be plotted in a different color. The two example commands above illustrate the `hlin` command. The first command sets the values for contour line set 1. The contours start at a value of 100 and increase in steps of 50. A total of 10 contour levels is drawn, e.g. the highest level corresponds to 600. The second `hlin` command sets values for the second contour line set, but the additional parameter '%' indicates, that the numbers are taken as percentage of  $\Delta z = z_{max} - z_{min}$  of all loaded data sets rather than absolute z-values. In our example the base level is 10% of  $\Delta z$ , the contour lines are stepped in 10% intervals and 9 levels are plotted. Thus the highest level corresponds to 100%. The command `hpak` determines how many contour line sets are actually plotted. The default is the usage of all sets defined by the `hlin` command. Color and line type for the individual contour line sets can be changed using the commands `hcol` and `htyp`.

Sometimes a specific aspect ratio of the x- and y-axis or a specific angle between the two axis is required to obtain a non distorted picture of the data. *KUPLOT* allows the user to specify an aspect ration using the `aver` command. As default *KUPLOT* determines the aspect ratio is such a way, that the resulting plot is as large as possible. This default can be restored by entering the command `aver` without further parameters. Alternatively, the desired ratio can be given as parameter to the `aver` command. In Figure 3.2 we show an example of a contour plot illustrating some of the features discussed above. The commands used to create the contour lines shown are listed below:

```
1 aver 0.707
2 angl 75.0
3 #
4 hlin 1, 10,10, 9
5 hlin 2,120,20,10
6 hcol 1,1,3
7 hcol 1,2,1
8 hlab 1,2
```

In line 1 we set the aspect ratio of the x- and y-axis to  $1/\sqrt{2} \approx 0.707$  and the angle between the axes to a value of  $75^\circ$  (line 2). In lines 4–5 we define two contour line packages, the first one giving contour lines at values of  $z_c = 10, 20, \dots, 100$  the second one at values  $z_c = 120, 140, \dots, 320$ . In line 6 the color for the first contour package for data set one is set to pen 3 (blue). Next the color for the second package for the same data set is set to pen 1 (red). Finally the labeling of



**Figure 3.2:** Customizing contour plots

contour lines for data set one is enabled. The second parameter in line 8 specifies that every second contour line starting from the base line is labeled. As always, check the online help for more detailed information on the commands used.

### 3.3 Using bitmaps

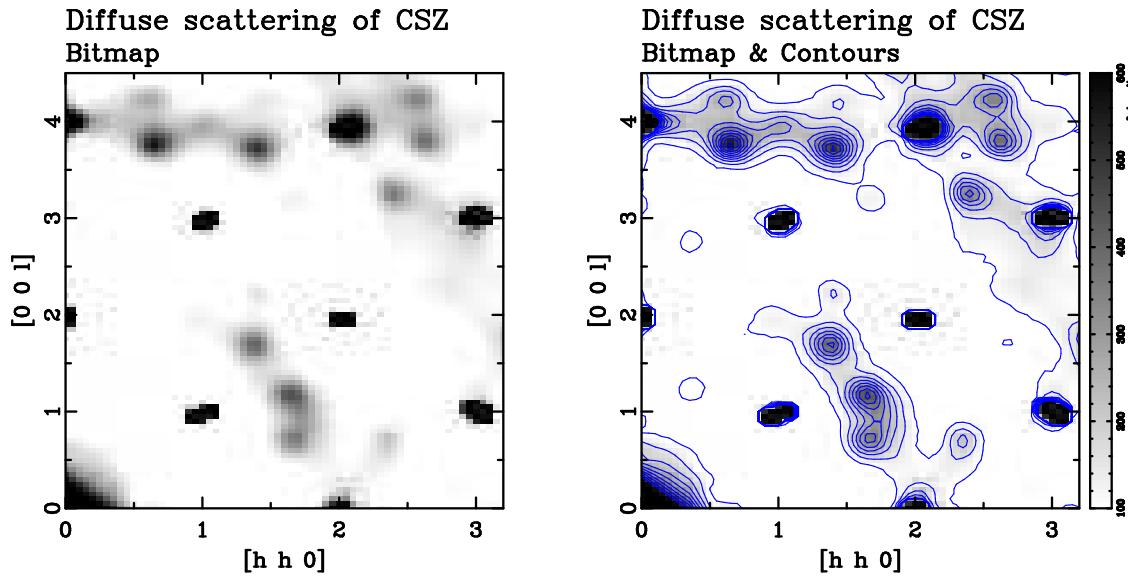
The command controlling the appearance of a 2D plot is `hart`. Its first parameter determines the data set, the second parameter the type of plot. To use only contour lines set the second parameter to 1, for bitmap display use 2 and to have both a bitmap and contour lines set the second parameter of the `hart` command to 3. In Figure 3.3 we can see the same plot using just a bitmap on the bottom and using additional contour lines on the top. The wedge showing the  $z$  range of the bitmap colors is activated by setting a label for the  $z$ -axis using the command `achz`.

The bitmaps used by *KUPLOT* have 240 color entries, the remaining 16 colors are reserved for other functions. The  $z$ -range to be converted to those 240 colors is determined by the minimal and maximal contour level for the first set defined by the command `hlin`. Subsequently the bitmap color range and the contour levels are the same for the first set of contour levels. In some cases one might want to create a plot where the bitmap and the contour lines have different ranges. This can be done like in the following example:

```

hlin 1,0,50,1,%
hlin 2,0,5,20,%
htyp 1,0
htyp 2,1

```



**Figure 3.3:** Example for bitmap plots

The levels set by the first `hlin` command determine the z-range used to create the bitmap, i.e. 0% to 50% of the z-range. We do this in a single step since only the maxima are used to determine the bitmap. The second `hlin` command sets the contour lines we actually want to plot on top of the bitmap. In order to suppress the first set of contour lines we set the line type for set 1 to 0, i.e. no line using the first `htyp` command. The line type for the second contour set is set to 1, i.e. solid line, as done in the last line of the example above.

*KUPLOT* has three default color maps used to display bitmaps. The map is selected by the command `cmap`. For a description of the different color maps refer to the online help. The selected map for the examples displayed in Figure 3.3 is gray. The command `cmap` is also used to save or read a color map from a file. Additionally the current color map can be altered using the variable `cmap`. For details about the usage of variables see the section `Varialse` in the general package manual `DISCUS` package.

### 3.4 Saving data

In contrast to 1D data where we could mainly save a given data set in the `xy` format, there are several options to extract and save data from a 2D data set. As described in section 2.6, the first step is to enter the save sub level using the command `ksav` followed by the number of the data set to be saved. As before the output filename is set by the command `outfile` and the saving process is started via `run`. However, there are now many more possible settings for the `form` command.

The first three of those formats listed in Table 3.2 save the data set or a subsection as 2D data set either in NIPL, PGM or XYZ format. As before the area that is saved is determined by the current size of the plot window determined by the command `ska1`. Alternatively the x-limits `x1` and `x2` and y-limits `y1` and `y2` can be specified as additional parameters to the 'form'

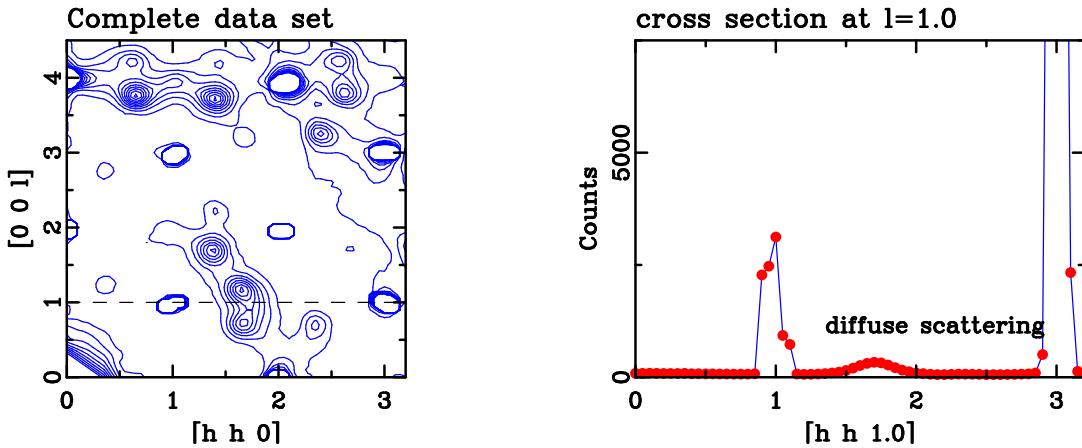


Figure 3.4: Extracting data from 2D data sets

command. Note that when using PGM as output format, the z-values are converted to integer and should range from  $0 \rightarrow 255$ . The command 'thresh' allows to control how the z-values are converted to this range. The other formats in Table 3.2 are used to extract a cross section from the 2D data set. The created output file is a normal xy file. The parameters `sx` and `sy` extract a cross section parallel to the x- or y-axis at the given y- or x-value (see example below). Rather than specifying these y- or x-values, the corresponding coordinates of maximum  $i$  determined by the command `smax` (see section 7) can be used (`mx` or `my`). The command above will extract the z-values at the corresponding grid points. The last two options can extract data from any value of  $x$  and  $y$  by interpolation. The format `sk` will use the coordinates  $x$  and  $y$  from data set (1D) number `ik` to determine the z-values to be extracted whereas `sl` will extract  $n$  points along a straight line defined by the points  $(x_1, y_1)$  and  $(x_2, y_2)$ .

A simple example how to extract a cross section is show in Figure 3.4. The original data set is shown on the left. The cross section parallel x (or  $[hh0]$ ) at y (or 1) equals 1.0 is marked by a dashed line. The resulting 1D plot of the cross section is shown on the right panel of the figure. The corresponding commands to create the data file shown at the top is listed below:

Format	Parameters	Description
ni	<code>[x1,x2,y1,y2]</code>	Nipl file, given area
pg	<code>[x1,x2,y1,y2]</code>	PGM file (ASCII), given area
gn	<code>[x1,x2,y1,y2]</code>	XYZ file (gnuplot), given area
<code>sx</code>	y-value	Cross section $\parallel$ x at y-value
<code>sy</code>	x-value	Cross section $\parallel$ y at x-value
<code>mx</code>	$i$	Cross section $\parallel$ x through maximum # $i$
<code>my</code>	$i$	Cross section $\parallel$ y through maximum # $i$
<code>sk</code>	$ik$	Cross section along xy of data set $ik$
<code>sl</code>	$x_1, y_1, x_2, y_2, n$	Cross section from $x_1, y_1$ to $x_2, y_2$ with $n$ points

Table 3.2: Save options for 2D data sets

```
1  ksav 1
2  outfile test.cut
3  form sx,1.0
4  run
```

In line 1 we enter the save sub level. In our example the data set to be used is data set number one. The output filename is set to *test.cut* (line 2) and the format is set to *sx*, i.e. cross section parallel to *x* at *y* = 1.0 (line 3). Finally the data file is written after the command *run* is entered (line 4).

## Chapter 4

# Plotting 3D data

In the previous chapter we have learned about using *KUPLOT* to plot 3D data sets. Now we will extend the usage of *KUPLOT* to 3D data sets, i.e. xyz-value-data. Most of the time *KUPLOT* will present a 2D rendering of the data values within an x-y-slice through the data. The interactive mode in chapter 6 "Using the mouse" allows to step through the adjacent slices. The coordinates *x* and *y* are within the drawing plane and the data *values* are represented by contour lines at given *val<sub>c</sub>* values, by the bitmap color or both. Changing the general appearance of the plot such as defining title lines, labels or tick mark intervals was described in section 2.2 of this manual and works exactly the same way when plotting 3D data sets. Changing the appearance of the contour lines or bitmaps works exactly as in chapter 3.2 "Customizing the plot" and is not repeated here. This chapter will focus on the file formats in section 4.1 "File formats" and the option to change the view direction of 3D-data in section 4.2 "Transformation of the view direction"

### 4.1 File formats

As described in section 2.1 for 1D data sets, 3D data sets are read using the command `load`. Multiple data sets might be read by repeating the command `load` and 1D, 2D and 3D data sets might be imported in any combination. The command `reset` clears the currently loaded data sets and the next file is read as set number one again.

A summary of the supported 3D file formats is given in Table 4.1. As the data volume of 3D data is usually quite large, data are stored in binary machine readable format.

The standard file format for 3D data is the Hierarchical Data Format HDF5 file format. The second file format is the MRC file format, often used for electron diffraction data, see the links at CCP4 format description and the Wikipedia page.

Format	Description
h5	Reads an HDF5 file. At the moment <i>KUPLOT</i> expects a file in the common format of Yell and DISCUS.
mrc	Reads an MRC file format.

Table 4.1: Supported file formats for 3D data

The short example shows how to read such an HDF5 file. As usual throughout DISCUS and KUPLOT the file extension bears no speacial meaning.

```
1 reset
2 load h5, example.h5
```

## 4.2 Transformation of the view direction

Often it will be desirable to view 3D-data along a different axis other than the default z-axis. Besides the fairly trivial view along x-and y, this can include a view along an arbitrary axis. This optional transformation is achieved in KUPLOT with an optional parameter `trans:yes` on the `load` command line. If this optional parameter is present, a sub menu is entered that allows to specify the desired transformation. The algorithm and the command sequence is identical to the optional transformation during the output of 3D diffraction data in DISCUS , see Chapter Fourier transform , section "Single crystal output with transformations" in the DISCUS manual. The description in this manual uses an identical example.

Three different combinations seem the likeliest to define a new viewing direction:

- Define a zone axis and an abscissa
- Define new abscissa, ordinate and top axes
- Define new corners in reciprocal space

These cases will be illustrated by examples. Each example assumes that reciprocal space has been calculated along  $a^*$ ,  $b^*$  and  $c^*$  as for example in the DISCUS Fourier macro:

```
1 fourier
2 11 -3.0, -3.0, -3.0
3 lr 3.0, -3.0, -3.0
4 ul -3.0, 3.0, -3.0
5 tl -3.0, -3.0, 3.0
6 na 241
7 no 241
8 nt 241
9 abs h
10 ord k
11 top l
12 temp use
13 disp off
14 run
15 exit
```

All examples will set a viewing direction along the [111] axis.

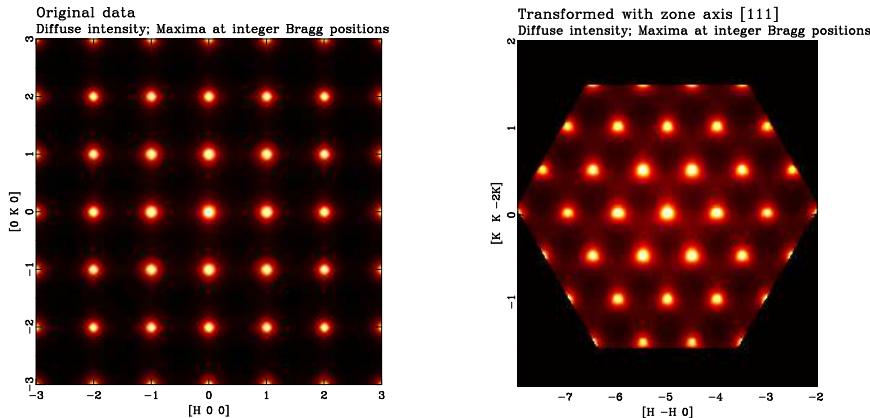
- Define a zone axis and an abscissa

```
1 kuplot
2 reset
3 load h5, example.h5, run trans:yes
4 reset
5 set abs:[0.04, -0.04, 0.00], zone:[1,1,1]
6 set npoints:[251, 251, 251]
7 run
8 exit ! Leave transformation menu; back to output menu
9 exit ! Leave Kuplot section
```

The run `trans:yes` command in line 3 instructs KUPLOT enter the transformation menu. After a `reset` to ensure clear conditions, line 5 defines a zone axis, here  $[uvw]=[111]$  and an increment vector along the abscissa. The abscissa in this example is parallel to the reciprocal vector  $[hkl]=[110]$ .

If line 6 in the macro is omitted, The overall size in reciprocal space is determined automatically and encompasses the original block in reciprocal space. Data points outside the original block will be set to zero.

To restrict the new reciprocal volume to a smaller or to expand it to a larger volume, the optional parameter `npoints:[251, 251, 251]` can be used. Within the transformation menu you can place this optional parameter either with other optional parameters onto a single set line or onto a set `npoints:[251, 251, 251]` line by itself. The numbers 251 are, of course just an example, feel free to vary as needed.



**Figure 4.1:** Example of an output transformation. Left original with  $[uvw]*[hkl]=[001]*[hkl]=0$ ; right  $[111]*[hkl]=0$ . To emphasize the surrounding zeros the plot has been drawn at slightly lower contour levels.

- Define new abscissa, ordinate and top axes

Essentially the identical transformation is achieved with:

```

1  kuplot
2  reset
3  load h5, example.h5, run trans:yes
4  reset
5  set abs:[0.04, -0.04, 0.00], ord:[0.04, 0.04,-0.08], top:[0.04, 0.04, 0.04]
6  set npoints:[251, 251, 251]
7  run
8  exit  ! Leave transformation menu; back to output menu
9  exit      ! Leave Kuplot section

```

Instead of a zone axis, explicit increment vectors along the abscissa, ordinate and top axis are defined.

- Define new corners in reciprocal space

Finally, if needed four corners in reciprocal space can be defined, in a similar fashion as in the `fourier` menu:

```

1 variable real, corners,dim:[3,4]
2 #           left      lower      bottom
3 corners[1,1] = -0.04*125 - 0.02*125 - 0.04*125
4 corners[2,1] = +0.04*125 - 0.02*125 - 0.04*125
5 corners[3,1] =  0.00*125 + 0.04*125 - 0.04*125
6 #
7 #           right     lower      bottom
8 corners[1,2] = +0.04*125 - 0.02*125 - 0.04*125
9 corners[2,2] = -0.04*125 - 0.02*125 - 0.04*125
10 corners[3,2] =  0.00*125 + 0.04*125 - 0.04*125
11 #
12 #           left      upper      bottom
13 corners[1,3] = -0.04*125 + 0.02*125 - 0.04*125
14 corners[2,3] = +0.04*125 + 0.02*125 - 0.04*125
15 corners[3,3] =  0.00*125 - 0.04*125 - 0.04*125
16 #
17 #           left      lower      top
18 corners[1,4] = -0.04*125 - 0.02*125 + 0.04*125
19 corners[2,4] = +0.04*125 - 0.02*125 + 0.04*125
20 corners[3,4] =  0.00*125 + 0.04*125 + 0.04*125
21 #
22 #   left   lower   bottom   corner :    -12.5000    -2.5000    0.0000
23 #   right  lower   bottom   corner :    -2.5000    12.5000    0.0000
24 #   left   upper   bottom   corner :    -7.5000    2.5000   -10.0000
25 #   left   lower   top     corner :    -2.5000    7.5000   10.0000
26 #
27 echo " ll %5.1f %5.1f %5.1f ", corners[1,1], corners[2,1], corners[3,1]
28 echo " lr %5.1f %5.1f %5.1f ", corners[1,2], corners[2,2], corners[3,2]
29 echo " ul %5.1f %5.1f %5.1f ", corners[1,3], corners[2,3], corners[3,3]
30 echo " tl %5.1f %5.1f %5.1f ", corners[1,4], corners[2,4], corners[3,4]
31 kplot
32 reset
33 load, h5, example.h5, trans:yes
34 reset
35 set ll:[corners[1,1], corners[2,1], corners[3,1]], lr:[corners[1,2], corners[2,2], corners[3,2]]
36 set ul:[corners[1,3], corners[2,3], corners[3,3]], tl:[corners[1,4], corners[2,4], corners[3,4]]
37 set npoints:[251,251, 251]
38 run
39 exit
40 exit ! leave Kplot section

```

As the macro exemplifies, in a general view direction it might be easier to define the corners as vector sum of the abscissa, ordinate and top vectors. Lines 22 to 25 summarize the effective corners that are set in this example.

### 4.3 Saving data

In a similar fashion to 2D data, there are several options to extract and save data from a 3D data set. As described in section 2.6, the first step is to enter the save sub level using the command `ksav` followed by the number of the data set to be saved. As before the output filename is set by the command `outfile` and the saving process is started via `run`. However, there are now many more possible settings for the `form` command.

The first line saves the file as HDF5 file. this is likely relevant after a transformation.

The next three of those formats listed in Table 4.2 save the data set or a subsection as 2D data set either in NIPL, PGM or XYZ format. As before the area that is saved is determined by the current size of the plot window determined by the command `skal`. Alternatively the x-limits `x1` and `x2` and y-limits `y1` and `y2` can be specified as additional parameters to the 'form'

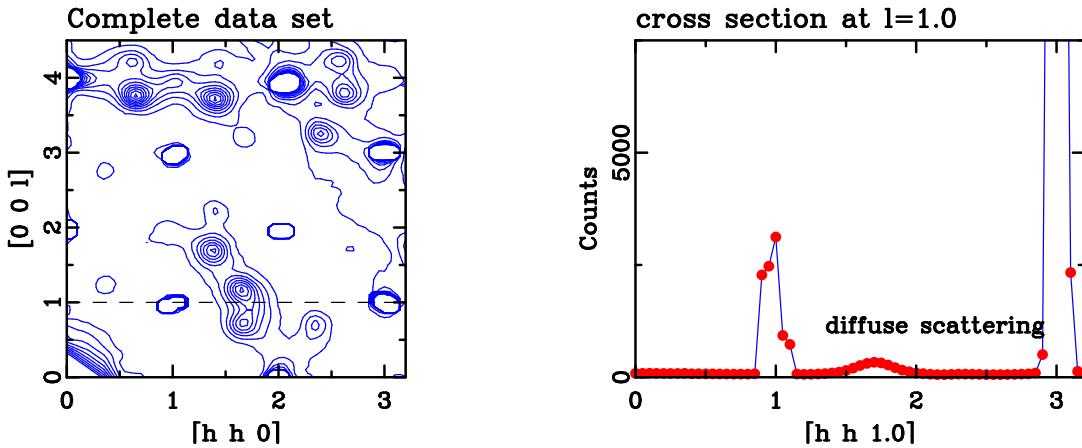


Figure 4.2: Extracting data from 3D data sets

command. Note that when using PGM as output format, the z-values are converted to integer and should range from  $0 \rightarrow 255$ . The command 'thresh' allows to control how the z-values are converted to this range. The other formats in Table 3.2 are used to extract a cross section from the 2D data set. The created output file is a normal xy file. The parameters sx and sy extract a cross section parallel to the x- or y-axis at the given y- or x-value (see example below). Rather than specifying these y- or x-values, the corresponding coordinates of maximum  $i$  determined by the command smax (see section 7) can be used (mx or my). The command above will extract the z-values at the corresponding grid points. The last two options can extract data from any value of  $x$  and  $y$  by interpolation. The format sk will use the coordinates  $x$  and  $y$  from data set (1D) number ik to determine the z-values to be extracted whereas sl will extract  $n$  points along a straight line defined by the points  $(x_1, y_1)$  and  $(x_2, y_2)$ .

A simple example how to extract a cross section is show in Figure 3.4. The original data set is shown on the left. The cross section parallel x (or [hh0]) at y (or l) equals 1.0 is marked by a dashed line. The resulting 1D plot of the cross section is shown on the right panel of the figure.

Format	Parameters	Description
h5		HDF5 file
ni	[x1,x2,y1,y2]	Nipl file, given area
pg	[x1,x2,y1,y2]	PGM file (ASCII), given area
gn	[x1,x2,y1,y2]	XYZ file (gnuplot), given area
sx	y-value, z-value	Cross section $\parallel$ x at pair y-z-value
sy	x-value, z-value	Cross section $\parallel$ y at pair x-z-value
sz	x-value, y-value	Cross section $\parallel$ z at pair x-y-value
mx	i	Cross section $\parallel$ x through maximum #i
my	i	Cross section $\parallel$ y through maximum #i
sk	ik	Cross section along xy of data set ik
sl	x1,y1,x2,y2,n	Cross section from x1,y1 to x2,y2 with n points

Table 4.2: Save options for 2D data sets

The corresponding commands to create the data file shown at the top is listed below:

```
1  ksav 1
2  outfile test.cut
3  form sx,1.0, 2.0
4  run
```

In line 1 we enter the save sub level. In our example the data set to be used is data set number one. The output filename is set to *test.cut* (line 2) and the format is set to *sx*, i.e. cross section parallel to *x* at *y* = 1.0 and *z* = 2.0 (line 3). Finally the data file is written after the command *run* is entered (line 4).

# Chapter 5

## Using frames

In this chapter we will introduce the usage of frames that enable *KUPLOT* to display more than one view graph in a single plot. A detailed description of all frame related commands is available as part of the online help which can be accessed by entering `help frames` at the input prompt.

### 5.1 Introduction

Frames enable *KUPLOT* to display multiple view graphs within a single plot. The layout is defined by the user. A simple example with two graphs being plotted on a single page is shown in Figure 5.1. Let us have a look at the macro file that was used to create the figure and learn step by step how to use this feature of *KUPLOT*. Note that the line numbers shown in the listing below are used for easy reference in this manual and are not part of the actual macro file.

```
1 load xy,s287.xy
2 load xy,s288.xy
3 #
4 tit1 Experiment A
5 achx \gw (\uo\d)
6 achy Counts
7 mark 0.1,100
8 buff 0.08
9 fnam off
10 fram on
11 #
12 nfra 2
13 #
14 afra 1
15 kfra 1,1
16 tit2 Scan 287
17 #
18 afra 2
19 kfra 2,2
20 tit2 Scan 288
21 #
22 plot
```

The macro starts with the reading of two data sets (lines 1–2). Next title, axes labels and marker intervals are set (lines 4–7) as in previous examples. The command 'buff' (line 8) alters the space

around the view graph reserved for axis numbering and titles. The value is the fraction of the total width or height of the plot. In our case we reserve 8% of the page on all four sides of the plot as buffer space. In line 9 the plotting of the filename is disabled and in the following line the plotting of a border around each frame is enabled. So far we have used no frame related commands and a plot at this stage would show both data sets in a single view graph. In line 12 we then specify that we want to use two frames, i.e. have two view graphs on our plot. At this stage, all settings from the current plot (fram 1 in case we have used frames before) are copied as defaults to all other frames. Entering command `plot` now would result in two equivalent view graphs side by side on the plot. Thus all setting that are common to all frames should be made *before* the command `nfra` is used. Now we need to customize the two frames. The command `afra` determines for which frame the following commands are used. First we alter settings for frame 1 by entering `afra 1` (line 14). Next we specify that this frame should only contain the data from data set one (line 15) and we enter an individual second title line (line 16). The same procedure is repeated for frame 2 (lines 18–22) which should include data set 2 and a different subtitle line. The command `plot` (line 22) will result in a picture similar to the one in Figure 5.1.

The usage of frames is rather simple and most *KUPLOT* settings are individual settings for the different frames. All frame related commands are summarized in table 5.1. The following two sections contain more complex examples using frames.

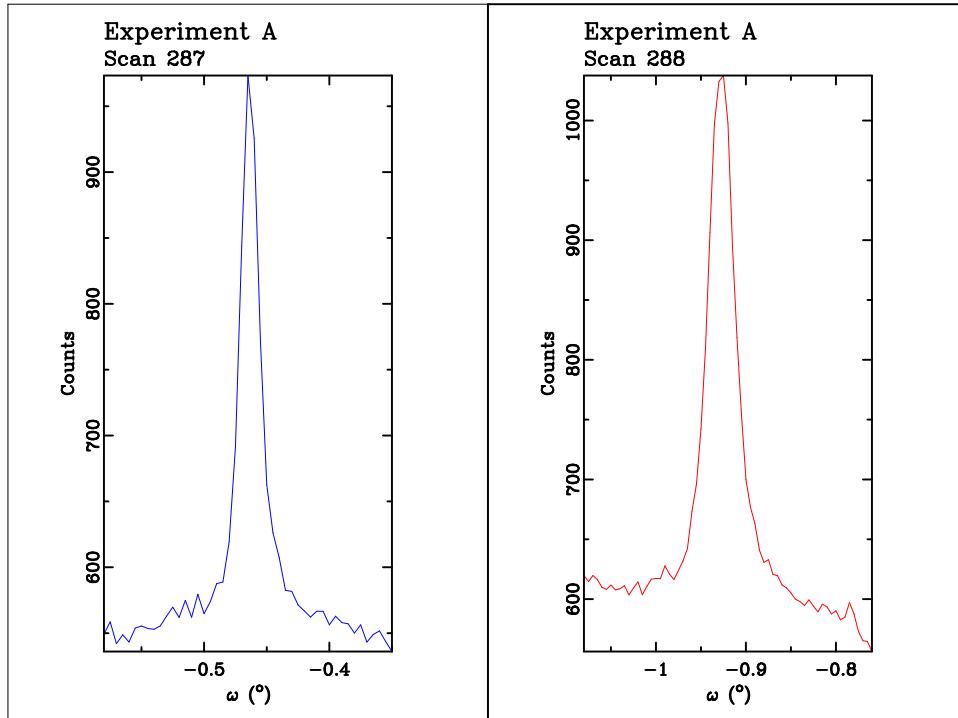


Figure 5.1: Simple example using frames

Command	Description
afra	Sets the active frame for user input
bfra	Sets background color for specified frame
cfra	Copies frame parameters
fram	Defines if a border is plotted around each frame
kfra	Defines contents of frames (data sets or text)
nfra	Sets number of frames (default = 1)
sfra	Defines position and size of a frame

Table 5.1: KUPLOT commands related to frames

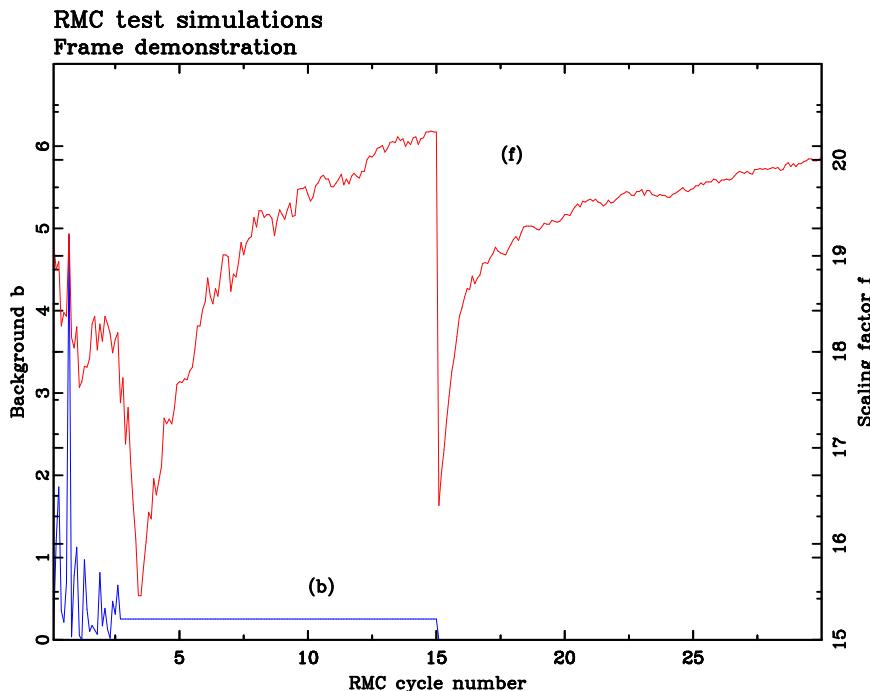


Figure 5.2: Using different y-axes with frames

## 5.2 Example 1: Using two different y-axes

In this example, we will use frames to produce a plot that uses two different y-axes to display two different data sets. One y-axis labeling will be on the left hand side of the view graph, the other one on the right hand side. The resulting plot is shown in figure 5.2. Here we display the scaling factor  $f$  (right axis) and the background parameter  $b$  (left axis) of a Reverse Monte Carlo refinement as function of the cycle number as an example plot.

We will discuss the corresponding macro file listed below step by step. Again the line numbers are not actually part of the macro. Lines 1–6 are nothing special, we read the two data files and set common features like title lines. Next we set the number of frames to two (line 8). Since we want to plot the two data sets on top of each other using just a different y-axes, we have to overwrite the default location of the frames using the `sfra` command (lines 10–11). The

coordinates of the plot area range from 0.0 to 1.0 in x- and y-direction. In our example both frames cover the complete plot area.

```

1  load xy,back.xy
2  load xy,scal.xy
3  #
4  fnam off
5  tit1 RMC test simulations
6  tit2 Frame demonstration
7  #
8  nfra 2
9  #
10 sfra 1,0.0,0.0,1.0,1.0
11 sfra 2,0.0,0.0,1.0,1.0
12 #

```

The next step is to input the settings for both frames. First we set the input focus to frame 1 (line 13) and specify data set 1 to be used in that frame (line 14). The fset command in line 15 determines the layout of the view graph and setting number 3 is the default plot layout, i.e. box, labels and tick marks in x- and y-direction and lines at  $x = 0, y = 0$ . Finally we set the size of the plotting window (line 16), the tick mark intervals (line 17) and the labels for the axes (lines 18–19).

```

13 afra 1
14 kfra 1,1
15 fset 3
16 skal 0.1,30.0,0.0,7.0
17 mark 5.0,1.0
18 achx RMC cycle number
19 achy Background b
20 #

```

The same as before has to be done for the second frame. Lines 21–22 set the focus to the second frame and select data set 2 to be displayed. The negative value of the parameter of the fset command (line 23) indicates that the y-axis label and numbers are to be plotted on the right hand side rather than on the left hand side which is the default. Again the extend of the plotting window and the tick marks are set (lines 24–25). Note that we need to specify the same plotting range ( $0.1 \rightarrow 30.0$ ) in x-direction and a different one in y-direction compared to the settings of frame 1. The x-axis label is switched off (line 26) because we have already the label from the other frame. The y-axis needs a new label (line 27). Finally we define two annotations (line 28–29). Note that the given coordinates are with respect to the scaling of frame 2.

```

21 afra 2
22 kfra 2,2
23 fset -3
24 skal 0.1,30.0,15.0,21.0
25 mark 5.0,1.0
26 achx
27 achy Scaling factor f
28 sann 1,"(b)",10.0,15.5
29 sann 2,"(f)",17.5,20.0
30 #
31 plot

```

Obviously the same plot could have been achieved by scaling one of the data sets to the y-range of the other, but the information about the original range of y-values of one of the data sets would have been lost in the plot in contrast to the example using frames given here.

### 5.3 Example 2: Connected view graphs

In this section we will learn how we can create plots where several graphs share one or more sides. This is done using the command `buff` we have used in the first part of this chapter. An example plot is shown in Figure 5.3.

The macro used to create the plot is displayed below. As usual we start resetting `KUPLOT` (line 1) and we select portrait orientation for this plot (line 2). Next the data sets are loaded (lines 4–7). This is followed by general settings (lines 8–12) since all our frames will have the save y-axis label, plot window and tick mark interval.

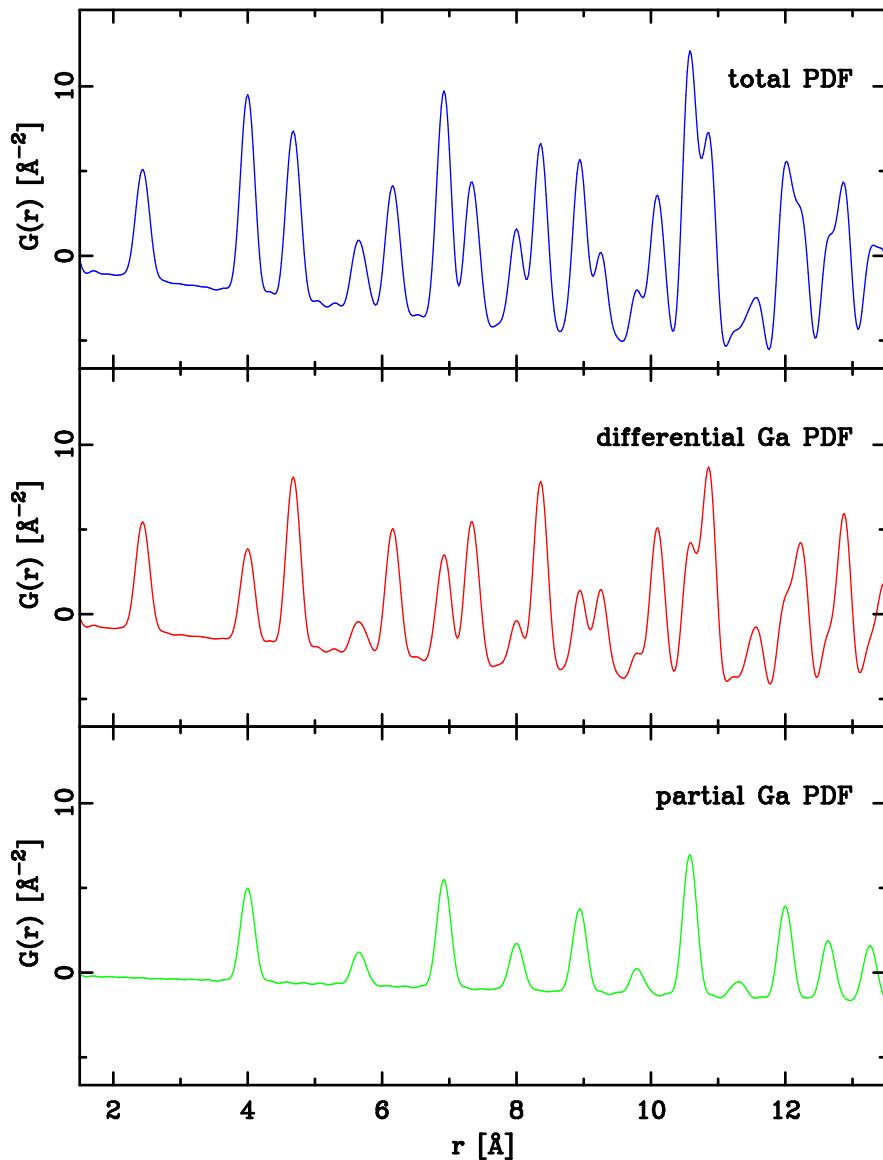


Figure 5.3: Example for connected view graphs

```

1  rese
2  orient port
3  #
4  load xy,tot.calc
5  load xy,dif_ga.calc
6  load xy,par_ga.calc
7  #
8  fnam off
9  fset 2
10 achy G(r) [\A\u-2\d]
11 skal xmin[1],xmax[1],1.2*ymin[1],1.2*ymax[1]
12 mark 2.0,10.0

```

Next we define the frames, three on top of each other with no gap in between them. Since the middle frame will have no buffer space on the bottom and top we need to make it smaller in order for the plot area to have the same size. Here we use variables for the task. The value of the buffer size for title and axes labels is stored in `r[1]` (line 14), here 0.1. The remaining plot space for each panel is calculated and stored in `r[2]` (line 15). It is simply the full size (1.0) minus two times the buffer size for the title of the top panel and the axis label and numbering of the bottom panel. Next we select three frames (line 17) and set the frame corners using the variables we just defined. The *x*-range for all panels is 0.0 to 1.0. The *y* ranges are determined by simply adding the heights of the panels and the buffering space together (lines 18–20). Simply calculate the numbers and it becomes clear.

```

13 #
14 r[1] = 0.1
15 r[2] = (1.0-2.0*r[1])/3.0
16 #
17 nfra 3
18 sfra 1,0.0,2.0*r[2]+r[1],1.0,3.0*r[2]+2.0*r[1]
19 sfra 2,0.0,      r[2]+r[1],1.0,2.0*r[2]+    r[1]
20 sfra 3,0.0,0.0      ,1.0,      r[2]+    r[1]

```

In the next section the setting for the individual frames are entered. Most of the commands were already discussed in the previous chapters. The command `buff` can either be used with a single parameter like in the example in section 5.1 in which case the buffer space around the view graph is the same in all directions. Alternatively one can supply four parameters for the required free space on the left and right side and the bottom and top of the view graph, respectively. In our example this first frame located on the top will have no buffer space at the bottom (line 23). Apparently we have to turn the numbering for the x-axis off (line 25). Now we repeat the settings for the other two frames. For the middle frame we have no buffer space on the top an bottom (line 29) and for the bottom frame we have only buffer space at the bottom (line 34). Also we need to set a x-axis label for the bottom panel (line 38).

```

21 #
22 afra 1
23 buff 0.1,0.1,0.0,r[1]
24 kfra 1,1
25 achx OFF
26 sann 1,"total PDF",13.0,10.0,right
27 #
28 afra 2
29 buff 0.1,0.1,0.0,0.0
30 kfra 2,2
31 achx OFF

```

```

32 sann 1,"differential Ga PDF",13.0,10.0,right
33 #
34 afra 3
35 buff 0.1,0.1,r[1],0.0
36 kfra 3,3
37 sann 1,"partial Ga PDF",13.0,10.0,right
38 achx r [\A]

```

Frames are a flexible tool and can be placed anywhere on the plot area and may also overlap. A even slightly more complex example using frames is discussed in the next section.

## 5.4 Example 3: Advanced frame usage

The final example for the usage of frames is more complex and the resulting plot is shown in Figure 5.4. *KUPLOT* is capable of creating quite complex plots and e.g. allows one to create complete transparencies for a talk.

The macro used to create the plot is displayed below. As usual the data sets are loaded first (lines 1–2) followed by general settings (line 4). Next we select 4 frames (line 6) and define the desired frame layout (lines 8–11) and individual background colors for each frame (lines 13–15). Note that the default background color is white. The colors are given as RGB (red, green, blue) values ranging from 0.0 to 1.0.

```

1 load xy,test.cut
2 load ni,test.nipl
3 #
4 fnam off
5 #
6 nfra 4
7 #
8 sfra 1,0.0,0.0,0.5,0.9
9 sfra 2,0.5,0.4,1.0,0.9
10 sfra 3,0.5,0.0,1.0,0.4
11 sfra 4,0.0,0.9,1.0,1.0
12 #
13 bfra 2,0.9,0.9,0.9
14 bfra 3,0.9,0.9,0.9
15 bfra 4,0.7,0.7,0.7

```

The next part of the macro file contains the settings for frame 1. After setting the focus to this frame (line 19), data set 1 is selected for this frame (line 20). The following commands specify various settings and were already explained in previous examples. The font command in line 32 increases the font size of all fonts used for this frame by 10%.

```

16 #
17 # Frame 1 with cross section
18 #
19 afra 1
20 kfra 1,1
21 buff 0.4
22 lcol 1,6
23 mtyp 1,3
24 mcol 1,1
25 msiz 1,0.2
26 skal 0.0,3.2,0.0,5000.0
27 mark 1,1000

```

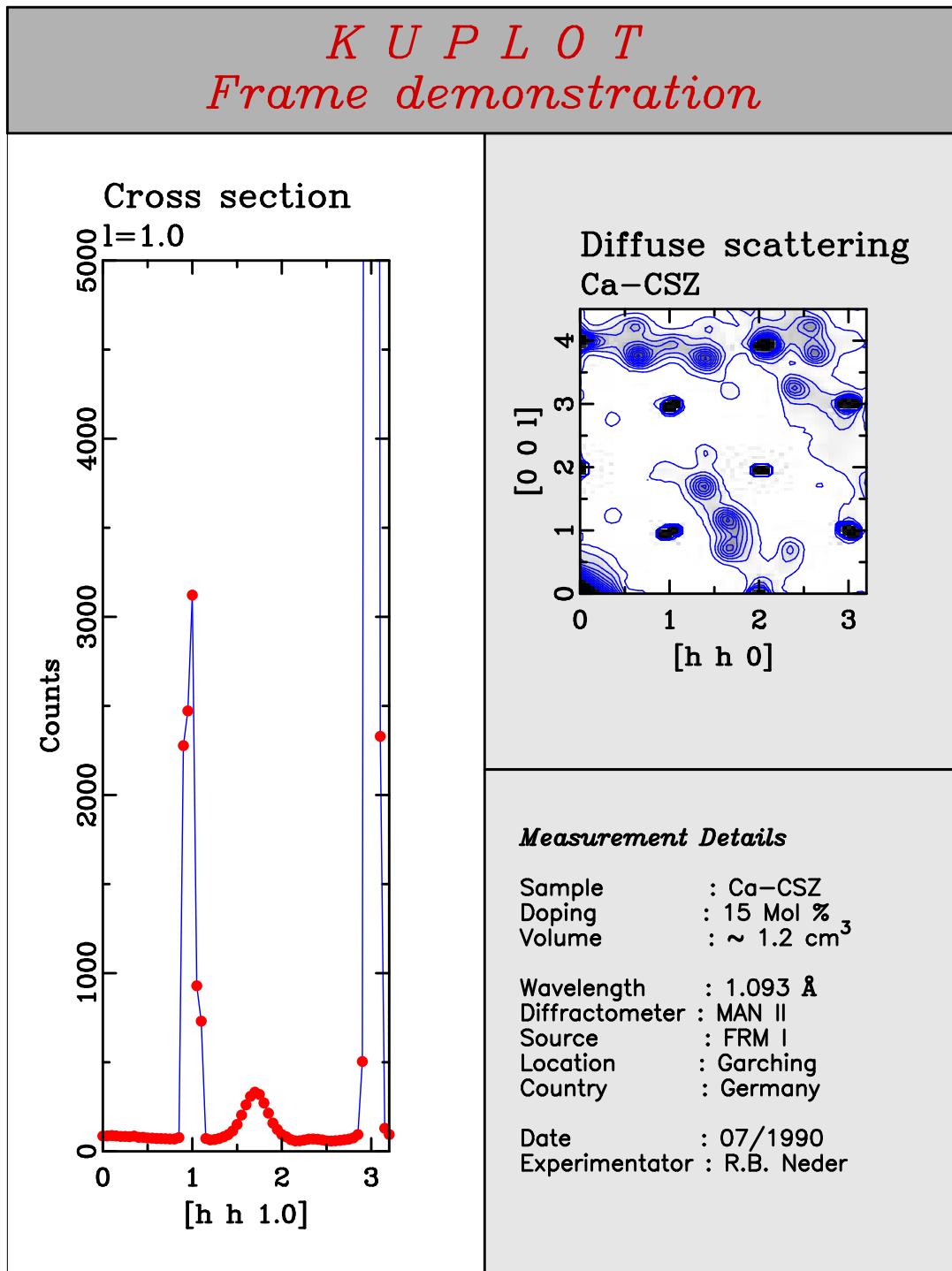


Figure 5.4: Advanced frame example plot

```

28 achx [h h 1.0]
29 achy Counts
30 tit1 Cross section
31 tit2 l=1.0
32 font size,1.1

```

Next we have the settings for frame 2 containing the 2D data set number 2, the diffuse neutron scattering of Ca-CSZ we used as example before. Again the various commands were explained in earlier examples.

```

33 #
34 # Frame 2 with data plot
35 #
36 afra 2
37 kfra 2,2
38 glat 2,3
39 hart 2,3
40 hcol 2,1,3
41 hlin 1,100,50,12
42 mark 1,1,0,0
43 aver 0.707
44 achx [h h 0]
45 achy [0 0 1]
46 tit1 Diffuse scattering
47 tit2 Ca-CSZ
48 font size,1.1

```

The next frame contains text rather than data. The text for this frame is read from the file *ref.txt* which contains exactly the text you can see at the bottom right corner of the plot in Figure 5.4. In lines 52–56 the justification of the text, the font type and font size are specified. The text file may contain the same special characters and control sequences discussed in section 2.3.

```

49 #
50 # Frame 3 with text
51 #
52 afra 3
53 kfra 3,ref.txt
54 font just,left
55 font typ,5,1
56 font siz,5,12

```

Finally we specify another text file containing the title line of the plot. We select the text to be centered (line 62) and set the font to *italics* (font 3), set the color to dark red (pen 7) and increase the size of the font to 24 points.

```

57 #
58 # Frame 4 with title text
59 #
60 afra 4
61 kfra 4,tit.txt
62 font just,center
63 font typ,5,3
64 font col,5,7
65 font siz,5,24

```

Currently one has not much control about the layout in a frame containing text but that might change in future version of the program *KUPLOT*.

# Chapter 6

## Using the mouse

### 6.1 Mouse interface

To enter the mouse mode, simply use the command `mouse`. The plot window will now contain a number off buttons in addition to the view graph as can be seen in Figure 6.1. The button *Coordinates* allows the user to display the coordinates of the mouse pointer by clicking the left mouse button. The values are displayed below the plot. In case of 2D files, the *z*-value of the closest data point is also shown. The right mouse button terminates the coordinate display function. The next group of buttons allows the user to use the mouse to modify the potting area. *Select region* does exactly that by clicking on the lower left corner of the desired area first. A moving frame will appear and the user can select the other corner of the new region. The buttons *Zoom* and it *Move* perform these functions using all three mouse buttons. A short help

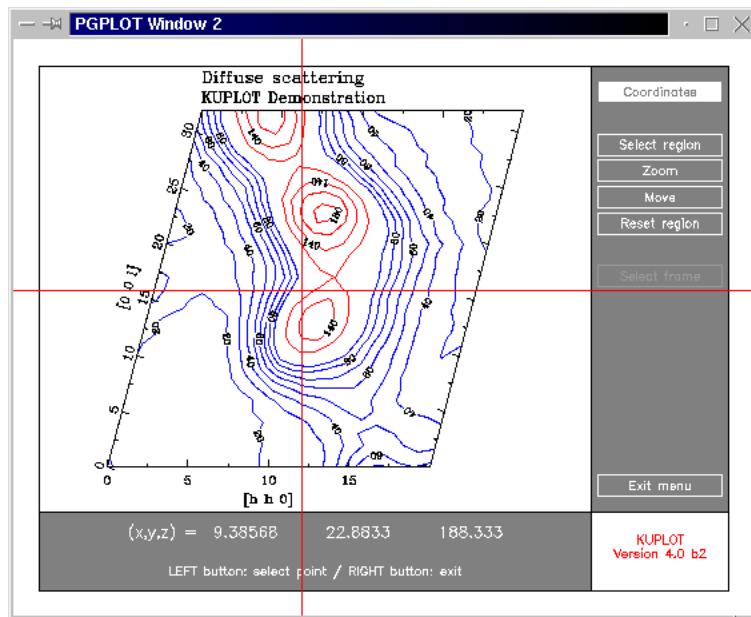


Figure 6.1: KUPLOT in mouse mode

text explaining which button is doing what appears below the plotting area once a function is selected. 'Reset region' allows the user to quickly display the complete range of data again. In cases where the current plot contains frames (see chapter 5), the active frame is marked by a light red background color. All functions will affect the active frame only. The focus can be moved to another frame via the *Select frame* button, which is only active when there is more than a single frame. Finally to get back to the command prompt of *KUPLOT* use the *Exit menu* button. Note that in cases where commands are written to a macro file via the command `learn`, all region functions will write the corresponding `skal` command in the macro file. One should also be careful when using the `mouse` command in macro files, since *KUPLOT* will not execute any further commands in until the *Exit menu* button is hit.

## 6.2 Mouse in macros

In the previous section, we learned about the interactive interface of *KUPLOT*. Alternatively, the `mouse` command can have additional parameters which specify what will be selected using the mouse pointer. In this case no buttons will show up and the coordinates will be returned in the `res[]` variables. Let us consider the following example. The goal is to use the mouse to select a number of points and store those in a data set. This could e.g. be used to define a background underneath some data manually. The macro file is show here:

```

1 variable integer,pt
2 variable integer,npt
3 variable integer,button
4 #
5 npt=0
6 button=0
7 #
8 echo Select points - right mouse button to finish
9 #
10 do while (button.ne.3)
11   mouse point
12   npt=npt+1
13   r[100+npt]=res[1]
14   r[200+npt]=res[2]
15   button=res[3]
16 enddo
17 #
18 npt=npt-1
19 alloc back,npt
20 do pt=1,npt
21   x[n[1],pt]=r[100+pt]
22   y[n[1],pt]=r[200+pt]
23 enddo
24 #
25 plot

```

We assume the data set of interest has been loaded and one has zoomed into the area where the background should be defined. In lines 1–3 we define the required variables. Note that this uses the new feature of named variables discussed in more detail in the chapter `Fortran style interpreter` of the general package manual `DISCUS` package. In lines 5 and 6 some variables are initially set to zero. In line 8 a message is printed on the screen to tell the user what to do. Basically every left click with the mouse will define a point and a right click will exit the macro and store the selected points. In line 10 we start a `while` loop which is executed until

Mode	Description
point	Selects a single point and returns $x, y$ or $x, y, z$ in variable <code>res[i]</code> .
line	Selects two points connected by a line.
rect	Selects two corners of a rectangle.
xrange	Selects two points defining range in $x$ .
yrange	Selects two points defining range in $y$ .

**Table 6.1:** Modes of command `mouse`

the variable `button` contains the number 3 which corresponds to the clicking the right mouse button. The command `mouse point` allows the user to select a point. The coordinates and the button pressed are returned in variables `res[i]` and are stored in variables (lines 13–15). This loop is now repeated until the right mouse button is pressed. Every time the variable `npt` is incremented, containing the number of points selected so far. The way the loop is constructed, `npt` actually contains the number of points plus one which is fixed in line 18. The command `alloc` (line 19) now creates a new and empty *KUPLOT data set* with a size of `npt` points. The loop in lines 20–23 is then used to store the coordinates selected in that data set and finally the data and background data set are plotted (line 25). In reality one would add commands to either interpolate and subtract the background directly or store it in a data file. The command `mouse` can also be used to select a region or range (see Table 6.1).

## Chapter 7

# Manipulating and analyzing data

In this chapter we will discuss *KUPLOT* functions that allow the analysis and manipulation of data. The first two sections deal with data manipulation, first using build-in functions and later using variables. The last section of this chapter summarizes data analysis functions of *KUPLOT*.

## 7.1 Simple data calculations

A common task is the numerical manipulation of a data set, e.g. to add some constant to a data set or inverse all y-values. The command `ccal` offers a variety of manipulation functions for a specific data set. The command `kcal` on the other hand allows simple arithmetic operations between two data sets. The commands and valid operations are listed in Table 7.1.

Note that  $x_i$  in Table 7.1 stands for x-, y-, z- and  $\sigma_x$ - or  $\sigma_y$ -values depending on the given parameters. The following simple command will multiply all y-values of data set one with the factor 1.75:

```
cocal mul,wy,1,1.75
```

Command	Operation	Description
ccal	abs	Performs $x_i =  x_i $
	add	Performs $x_i = x_i + a$
	exp	Performs $x_i = \exp(x_i)$
	inv	Performs $x_i = \frac{1}{x_i}$
	log	Performs $x_i = \ln(x_i)$
	mul	Performs $x_i = f \cdot x_i$
	sqr	Performs $x_i = \sqrt{x_i}$
	squ	Performs $x_i = x_i^2$
kcal	add	Performs $x''''_i = x''_i + x'_i$
	sub	Performs $x''''_i = x''_i - x'_i$
	mul	Performs $x''''_i = x''_i \cdot x'_i$
	div	Performs $x''''_i = x''_i / x'_i$

Table 7.1: Data manipulation functions

The parameter `mul` indicates that a multiplication is to be performed using the y-values which are selected by the next parameter (`wy`). Finally data set one and the desired factor of 1.75 are specified. For x- and z-values use `wx` and `wz`, the standard deviations  $\sigma_x$  and  $\sigma_y$  are selected using the parameters `dx` and `dy`.

## 7.2 Calculating functions

Another feature of *KUPLOT* is the capability to create a data set from an arithmetic expression rather than reading it from a file. This is done using the command `func`. The following two commands demonstrate usage of `func` to create a 1D data set ( $y = \sin(x)$ ) and a 2D data set ( $z = \sin(x) \cdot \cos(y)$ ).

```
func sin(r[0]),0.0,6.3,0.1
func sin(r[0])*cos(r[1]),0,6,0.1,0,6,0.1
```

Note that the variable `r[0]` is used for the x-argument and `r[1]` is used as y-argument. Thus values previously stored in these two variables are destroyed by the `func` command. The following commands are the range and the grid size in the two directions. In our first example, the desired x-range is  $0.0 \rightarrow 6.3$  with a grid size of  $\Delta x = 0.1$ . This results the creation of a data set with 64 points. The second `func` command shown above creates a 2D data set ranging from 0.0 to 6.1 in x- and y-direction with a grid size of  $\Delta x = \Delta y = 0.1$  given a size of 61x61 data points. Alternatively, space for a new data set can be initialize using the command `alloc` and the data values are then calculated using the FORTRAN style interpreter of *KUPLOT* (see section 7.4). However, the creation of large data sets this way from an arithmetic expression might be relatively slow.

## 7.3 Data manipulation

### 7.3.1 Data smoothing

*KUPLOT* has two different smoothing functions that can be used for 1D as well as 2D data. The first type of smoothing is a simple sliding average. The corresponding command is `glat`. The name is a reminder that the first *KUPLOT* version was in German (smoothing in German is *glätten*). The smoothing is performed by a sliding average of  $n$  neighboring points. The value of  $n$  is given as parameter of the command `glat`. An example for the smoothing operation is given in Figure 7.1. The top view graph shows the raw data showing quite noisy contour lines. The picture below shows the same data after the data set was smoothed with a value of  $n = 7$  using the command `glat 1, 7` assuming the values are stored as data set one. It is apparent from Figure 7.1 that this type of averaging broadens the peaks and does not preserve the peak heights. An alternative way to smooth data implemented in *KUPLOT* is the Savitzky-Golay algorithm which is in principle a weighted sliding average. The weight is given by a polynomial of user definable order (the default is 2). The command for the later type of smoothing is `smooth` with parameters identical to `glat`. However, the minimum number of  $n$  is five. The bottom view graph in Figure 7.1 was smoothed with  $n = 7$  using the command `smooth`. It can be clearly seen that the widths and heights of the peaks are much better preserved. For a

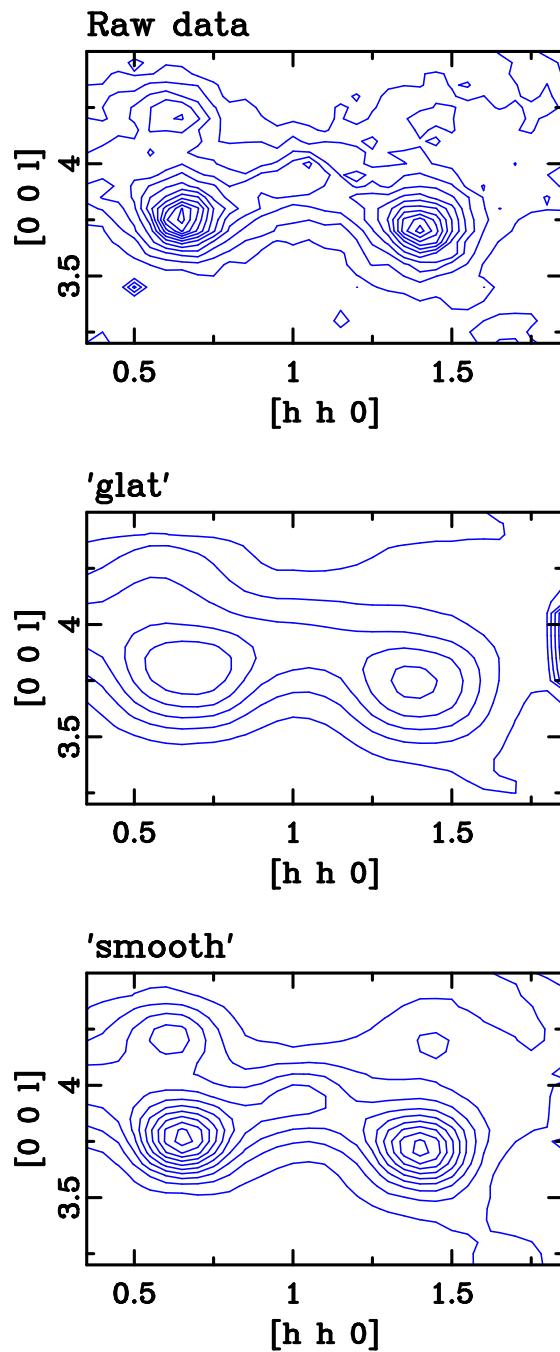


Figure 7.1: Demonstration of data smoothing

detailed discussion about the smoothing algorithm and its limits see e.g. Numerical Recipes by Press, Flannery, Teukolsky & Vetterling, Cambridge University Press, 1989.

As a default, 2D data sets are per smoothed in  $x$  and  $y$  direction. The user can restrict the smoothing to only one direction by adding the optional parameter  $x$  or  $y$  to the `glat` or `smooth` command. Again the reader might refer to the online help for more detailed information on the smoothing commands.

### 7.3.2 Sorting data

The command `sort ik` will sort the data points in data set `ik` by increasing  $x$  values. No other sorting is currently implemented in *KUPLOT*, however one might implement any sorting algorithm using the FORTRAN interpreter of the program.

### 7.3.3 Rebinning data

Sometimes a data set needs to be transformed onto a different grid of  $x$  or  $x, y$  values. This is usually referred to as rebinning. The most common application is to transform a data set from an irregular grid to an equidistant grid. *KUPLOT* offers a different mechanism for 1D and 2D data sets. The command `rebin ik, delta` will rebin the 1D data set `ik` to an equidistant grid with a bin width of `delta`. One needs to take care, that the new grid is at least slightly larger than the original to prevent 'holes' from appearing. In addition to an equidistant grid, a data set can be interpolated on a set of  $x$  values given by a different data set. The command `spline 1, 2` for example would interpolate data set one onto the  $x$  values given by data set 2 and store the result in a new data set.

To rebin a 2D data set, one needs to first save it in  $x, y, z$  or `gn` format. This data file is then loaded back into *KUPLOT* using the command `load zz, file, dx, dy` and rebinned on the desired grid `dx, dy`. Obviously one can load  $x, y, z$  files on an irregular grid the same way.

### 7.3.4 Matching and merging data sets

Sometimes two data sets differ by a scaling factor or an offset. The command `match` allows to find the scaling and/or offset that gives the best agreement between two data sets. The scaling and background is directly applied. Note that this function only works for data sets, that have identical points in  $x$ . The command `merge` allows one to combine data sets. Points with common  $x$ -values are averaged.

## 7.4 Data manipulation using variables

A very flexible way to manipulate data is the use of variables. Details about the FORTRAN style interpreter and the usage of variables were discussed in chapter *Fortran style interpreter* of the package manual *DISCUS* package. Two simple examples are given here for illustration. In the first example we assume having a data set containing measured intensities as function of some angle  $\omega$ . The following *KUPLOT* commands will create values for  $\sigma_y$  according to the relation  $\sigma_y(i) = \sqrt{y(i)}$ .

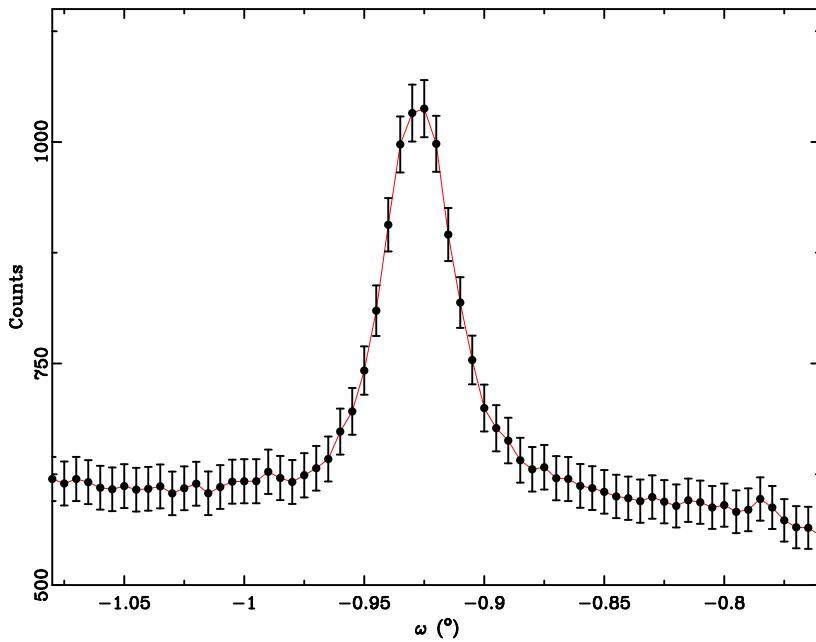


Figure 7.2: Plot of data manipulation example

```

1 do i[1]=1,np[1]
2   dy[1,i[1]]=sqrt(y[1,i[1]])
3 enddo

```

The first line starts a DO loop over all data points of data set one, assuming that is where our data are stored. The variable `i[1]` is the loop control variable and `np[1]` contains the number of data points of data set one. For a complete list of variables refer to section Variables in the package part of this users guide DISCUS package. Next the value of  $\sigma_y$  for data point `i[1]` is computed (line 2). The first parameter in `dy[1,i[1]]` specifies data set one. Finally (line 3) the loop is closed. The corresponding plot with the error bars of the newly created values of  $\sigma_y$  is shown in Figure 7.2.

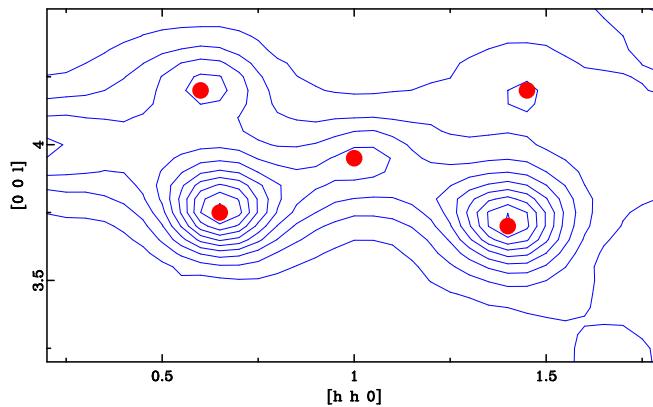
The second example shows how to create a new data set using variables. We assume that we have two data sets loaded, both having the same length and identical x-values. We want to create a new data set with the same x-values and y-values that are the average of the y-values of the two load data sets, thus  $y'''_i = \frac{1}{2}(y''_i + y'_i)$ . Here  $y'''$  stands for the new data set whereas  $y''$  and  $y'$  represent the values of the two loaded data sets. The KUPLOT commands for our task are listed below:

```

1 alloc aver.xy,np[1]
2 #
3 do i[1]=1,np[1]
4   x[3,i[1]]=x[1,i[1]]
5   y[3,i[1]]=0.5*(y[1,i[1]]+y[2,i[1]])
6 enddo

```

First we have to allocate space for the new data set. This is normally done by the command `load` or `func`, but the command `alloc` allows the user to create an empty data set, just what we need here. The name `aver.xy` given as parameter to the `alloc` command in line 1 is the



**Figure 7.3:** Marking of maxima within a plot

internal name for the data set, e.g. showing up in the top left corner of the plot or when using the `show` command. This is an arbitrary name and `no` file of that name needs to exist. The second parameter specifies the size of the new data set, in our case the same size as data sets one (variable `np[1]`) or two. Next we have a loop as in the previous example over all data points (line 3). In lines 4–5 the  $x$ - and  $y$ -values of the new data set number 3 are computed. Since we assumed that both original data sets have the same  $x$ -values, we just use one of them as  $x$ -values for the new set. Finally the loop is closed (line 6).

Although the usage of variables allows one to perform the same functions as the `ccal` and `kcal` commands, the usage of variables is much slower especially for large data sets and should only be used if no corresponding build-in function is available.

## 7.5 Data analysis

This section describes the data analysis functions of *KUPLOT*. Apparently variables can also be used to calculate averages and analyze data sets. The contents of a variable or result of an expression can be displayed with the command `eval`. Another wide area of data analysis is to fit a theory function to a data set. The least square fitting functions of *KUPLOT* are discussed in the next chapter. As example for this chapter we use a subsection of the diffuse scattering data displayed in previous examples. The data are shown in Figure 7.3. The circles mark positions of maxima found in the data set by the command `smax`.

A maximum determined by `smax` is defined as a point where all  $n$  neighboring points have smaller  $y$ - or  $z$ -values compared to the reference point. The value of  $n$  is the second parameter of the `smax` command, the first is the data set number. The maxima marked in Figure 7.3 were determined with `smax 1, 3` assuming we are dealing with data set one. The command `ptyp` allows one to select a symbol analog to `mtyp` to mark the positions of the determined maxima. Furthermore the positions of the found maxima are displayed on the screen. The output for our example is shown below:

```
Found maxima data set 1 (ifen = 3) :
No.      pos. x      pos. y      value
-----
```

1	.600	4.200	272.778
2	.650	3.750	567.111
3	1.000	3.950	273.889
4	1.400	3.700	509.667
5	1.450	4.200	156.778

You might verify these coordinates as the marked positions in figure 7.3. Other functions allow the user to determine the integral or mean values of a given area of the data set.

Next we will determine the integral of the left diffuse peak in figure 7.3. This is done using the command

```
inte 1, .423750, .903750, 3.4716, 3.99168
```

The first parameter specifies the data set number followed by the area to be integrated given as  $x_{min}, x_{max}, y_{min}$  and  $y_{max}$ . If those last 4 parameters are omitted, the complete current plotting window is used. The screen output of this command is:

```
Integration result for data set 1 :
x-range   : .4238      to   .9038
y-range   : 3.472       to   3.992
Integral   : 57.06      +-   .3650      (    100 pkt)
```

The command `mean` with similar parameters can be used to calculate mean values and standard deviations in the given region. The region above was determined using the `mouse` functions (see 6).

## 7.6 Fourier transform

*KUPLOT* can calculate a discrete Fourier transform of a 1D or 2D data set. Let us start with a little mathematics first. Details can again be found in the Numerical Recipes. The function we want to Fourier transform is  $f(x_i)$  with  $i = 1,..N$  with a constant grid size of  $\Delta x$ . This gives us the following range in Fourier space:

$$h_n = \frac{n}{N\Delta x} \text{ with } n = -\frac{N}{2}.. \frac{N}{2} \quad (7.1)$$

The Fourier transform is calculate as

$$F(h_n) = \Delta x \sum_{i=1}^N f(x_i) [\cos(2\pi h_n x_i) + i \sin(2\pi h_n x_i)]. \quad (7.2)$$

The Fourier transform is a complex quantity and *KUPLOT* allows one to select the real and imaginary part and/or amplitude and phase angle to be stored as new data sets.

Let us consider a simple example, a box function as displayed in the upper left panel of Figure 7.4. Our data set has  $N = 50$  points and a grid size of  $\Delta x = 1.0$ . Using equation 7.1 we find the grid size in Fourier space to be  $1/50 = 0.02$  and a calculated range of  $-1/2\Delta x = -0.5$  to  $0.5$ . In our example the command for the calculation of the Fourier transform is:

```
four ria
```

The parameter `ria` specifies that we want to keep the real and imaginary part as well as the amplitude of the Fourier transform. All three parts and the box function are shown in Figure 7.4.

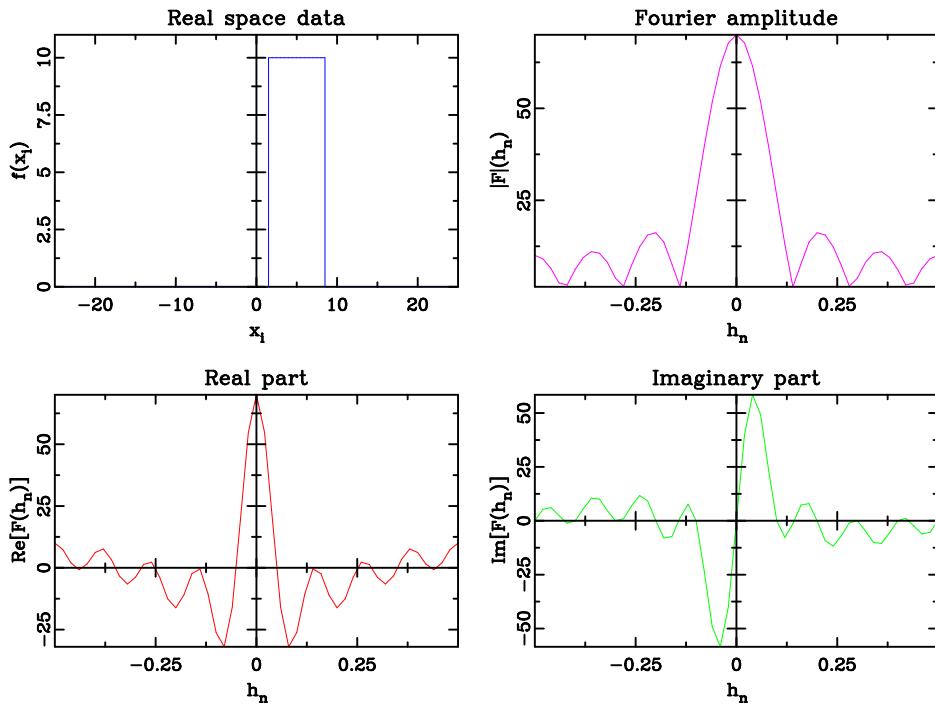


Figure 7.4: Fourier transform of box function

## 7.7 Other functions

The command `deriv ik,n` will calculate the numerical n-th derivative of data set `ik` and store the result a a new data set. The convolution of two data sets can be calculated using the command `conv`. Both data sets need to have a common equidistant set of  $x$  values.

# Chapter 8

## Least square fitting

This chapter describes the least-square (LS) fitting segment of the program *KUPLOT*. The user can choose between polynomial, Gaussian and Lorentzian for 1D data or a Gaussian for 2D data. Furthermore this version of *KUPLOT* supports user defined theory functions thus allowing for virtually any fit function desired. A description of the LS method itself is beyond the scope of this manual and the reader might refer to various text books on statistics or numerical mathematics. For a list of all fit related commands of *KUPLOT* check the online help or the command reference manual.

### 8.1 Fitting 1d data sets

Starting point for a LS refinement is a loaded data set  $(x_i, y_i)$ , the selection of a suitable theory function  $y(x_i, p)$  to describe the data and a set of starting values for the fit parameters  $p$ . The program *KUPLOT* offers the following three predefined theory functions and additionally allows the use of a user defined function (see section 8.3).

$$y(x_i, p) = \sum_{n=0}^N p_n x_i^n \quad (8.1)$$

$$y(x_i, p) = \sum_{n=0}^N p_n T_n(x_i) \quad (8.2)$$

with  $T_n(x) = \cos(n \arccos x)$

$$y(x_i, p) = p_1 + p_2 x_i + \sum_{n=1}^N p_{1,n} \cdot \exp \left\{ \frac{-(x_i - p_{2,n})^2}{\sigma_n^2} \right\} \quad (8.3)$$

$$y(x_i, p) = p_1 + p_2 x_i + \sum_{n=1}^N p_{1,n} \cdot \left\{ \frac{\sigma_n^2}{(\sigma_n^2 + 4(x_i - p_{1,n}))^2} \right\} \quad (8.4)$$

$$\text{with } \sigma_n = \begin{cases} p_{3,n} \cdot p_{4,n} & \text{for } x_i < p_{2,n} \\ \frac{p_{3,n}}{p_{4,n}} & \text{else} \end{cases}$$

Let us have a closer look at these theory functions. The first one shown in equation 8.1 is a simple polynomial of the order of  $N$  which can be defined by the user. The fit parameters

$p_n$  are the corresponding coefficients. Note that *KUPLOT* actually numbers the fit parameters starting with one, i.e. parameter one corresponds to the term  $x^0$ . Similarly equation 8.3 defines a Chebyshev polynomial.

The next function (equation 8.3) is a sum of  $N$  Gaussians and a linear background defined by the first two parameters  $p_1$  and  $p_2$ . The number of Gaussians to be used is defined by the user. Each Gaussian is represented by a set of four fit parameters:  $p_{1,n}$  is its peak height and  $p_{2,n}$  is its peak position. The half width of the Gaussian (as well as the Lorentzian) is defined by a half width parameter  $p_{3,n}$  and an asymmetry parameter  $p_{4,n}$ . Obviously the symmetric case is given by  $p_{4,n} = 1.0$ . The Lorentzian theory function (equation 8.4) is defined by a similar set of parameters. Currently both function types can not be used in combination except as user defined theory function as described in section 8.3.

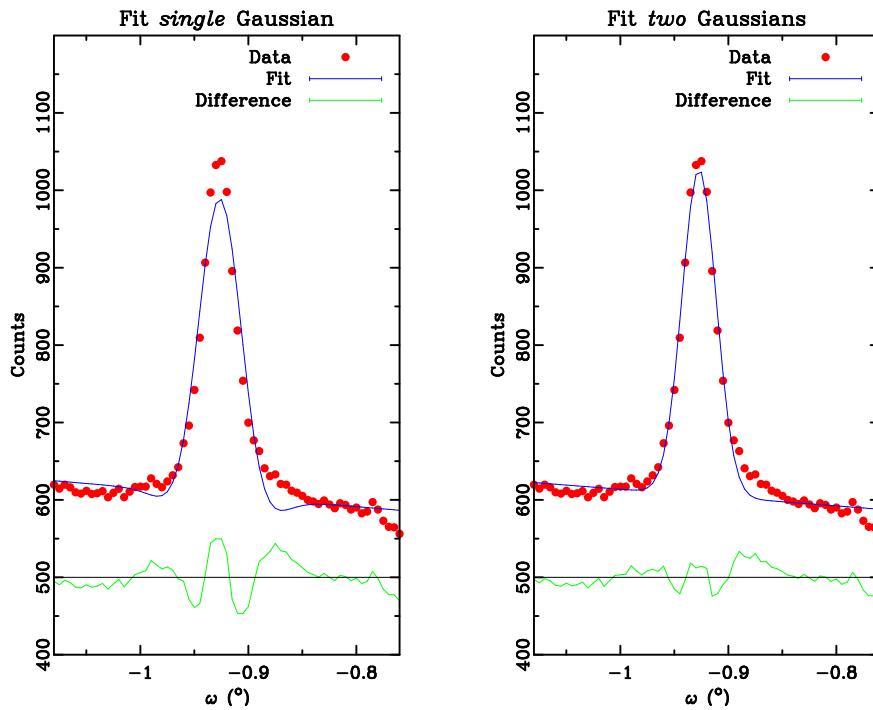


Figure 8.1: Fit results for 1D example

Now we will see the fitting segment of *KUPLOT* in action in the following example. The data set is the profile we have used in examples before in Figure 5.1. Assuming the corresponding data are loaded as set one, the next step is to enter the LS sub level of the program *KUPLOT*. This is done by the command `fit` followed by the data set number that should be used as observed data (line 1). The macro is shown below and as before the line number are not part of the macro file itself.

```

1  fit 1
2  #
3  mfen 7
4  func gaus,1
5  par
6  #

```

Parameter	One Gaussian	Two Gaussians	
		Gaussian 1	Gaussian 2
Background $p_1$	506.(16)	501.(10)	
Background $p_2$	-107.(18)	-106.(11)	
Peak height	421.(9)	315.(10)	134.(10)
Peak position	-0.9270(4)	-0.9277(3)	-0.9250*
Half width	0.0374(9)	0.0278(9)	0.0640*
R-value	2.1%	1.2%	

**Table 8.1:** Results of example 1D fit

7 run  
8 save

The command in line 3 sets the window size used to determine maxima within the plot which are used as defaults for the starting values. The procedure is equivalent to the command `smax` explained in section 7.5. For our example the setting to 7 points will determine the main maximum correctly. Next (line 4) the theory function is specified to be one Gaussian. Note that *KUPLOT* determines the default starting values for the fitting parameters  $p$  at this stage and the command `mfen` has to be entered before to have an effect. Furthermore all current values of the fit parameters, e.g. from a previous run, are replaced by the new default values. The command `par` (line 5) lists the current parameter settings and after checking them on the screen we are ready to go. The fit is started via the command `run` in line 7 of our example command file. The fit progress is displayed on the screen until the fits converges to a minimum or the maximum number of cycles is reached. This maximum cycle number can be altered using the `cyc` command in the fitting sub level. In our example it took 9 iterations to reach the minimum and the resulting parameters are listed in table 8.1 on the left. Finally the results are saved (line 8) as two data files containing the calculated and difference data set and a text file summarizing the results.

The resulting plot of the observed and calculated data as well as the difference between them is shown in Figure 8.1 again on the left. Although there a reasonable agreement is achieved, there are clearly visible differences especially at the tails of the Gaussian. Note that we have fitted a symmetric Gaussian which is the default, i.e. the parameter  $p_{4,n}$  is set to 1.0 and not refined. To refine this parameter use the command `par 6, 1`. The first number is the number of the fit parameter (since we have two background parameters it is  $4+2=6$ ) and the next number can be 1 for refining or 0 for keeping the corresponding parameter fixed. An optional third parameter allows one to alter the value of the fit parameter.

To improve the fit we will try to fit two Gaussians located at the same position to the data set. This will allow to model the sides of the peak in a better way. The corresponding macro is shown below. Lines 1–4 are the same as before except that we select two Gaussians rather than one. Again we assume the profile is loaded as data set one.

```
1 fit 1
2 #
3 mfen 7
4 func gaus,2
5 #
```

```

6  par 7,1,0.0
7  par 8,1,p[4]
8  par 9,1,p[5]
9  par10,0,p[6]
10 par
11 #
12 run
13 save

```

Since *KUPLOT* will only find one maximum a warning will be displayed and we have to set the starting values for our second Gaussian manually (lines 6–9). We set the peak height to 0.0 and take the other parameters from the settings for the first Gaussian using the variables `p[i]`. Just to be sure we list the current settings again (line 10) and the fit is started in line 12 by the `run` command. Finally the results are saved as before (line 13). The resulting parameters are listed in Table 8.1. A measure for the quality of a fit is the R-vale which is defined as:

$$R = \sqrt{\frac{\sum_{i=1}^N w_i (y_i - y(x_i, p))^2}{\sum_{i=1}^N w_i y_i^2}} \quad (8.5)$$

Here the sum goes over all  $N$  data points  $(x_i, y_i)$  and  $y(x_i, p)$  is the theory function. *KUPLOT* offers a variety of weights  $w_i$  to be selected to the fit. The default we have used in our example is  $w_i = \frac{1}{y_i}$ . Check the online help for the command `wic` to obtain more information about supported weighting schemes. Inspection of the results for this second fit shown in Table 8.1 and Figure 8.1 clearly show that the second fit describes the data much better. However the parameters marked with \* in Table 8.1 could not be refined and remained at their starting values. Possible steps to avoid that problem are to refine the parameters one by one or change the setting of URF which controlled the *speed* of the fit. A small value, e.g. 0.1, will converge quickly to the minimum but in case of a more complex problem, the fit might fail. A larger value of URF on the other hand will change the parameters in each iteration by a smaller amount and the fit might work. However care has to be taken to not end up in a local minimum. The value of URF (which stands for something very German - Unterer Relaxationsfaktor) is altered via the command `urf`. The authors recommend playing around with the various settings of the fit sub level to get familiar with the least square fitting process.

## 8.2 Fitting 2d data sets

Actually there is no principle difference between fitting a 2D or a 1D data set. The only theory function of 2D data currently available in *KUPLOT* is a set of  $N$  2D Gaussians as defined in equation 8.6. Each Gaussian is defined by eight parameters, peak height  $p_{1,n}$ , peak position  $p_{2,n}$  and  $p_{3,n}$  in x- and y-direction, half widths  $p_{4,n}, p_{5,n}$  and asymmetry parameters  $p_{7,n}, p_{8,n}$  for each direction and a new parameter  $p_{6,n}$  that defines the angle between the main axes of the Gaussian and the coordinate system. One additional parameter describes a flat overall background.

$$\begin{aligned}
 z(x_i, y_i, p) &= p_1 + \sum_{n=1}^N p_{1,n} \cdot \exp \left\{ \frac{r_x^2}{\sigma_n^2} \right\} \cdot \exp \left\{ \frac{r_y^2}{\tau_n^2} \right\} \\
 \text{with } r_x &= \cos(p_{6,n})(x_i - p_{2,n}) + \sin(y_i - p_{3,n}) \\
 r_y &= -\sin(p_{6,n})(x_i - p_{2,n}) + \cos(y_i - p_{3,n}) \\
 \sigma_n &= \begin{cases} p_{4,n} \cdot p_{7,n} & \text{for } x_i < r_x \\ \frac{p_{4,n}}{p_{7,n}} & \text{else} \end{cases} \\
 \tau_n &= \begin{cases} p_{5,n} \cdot p_{8,n} & \text{for } y_i < r_y \\ \frac{p_{5,n}}{p_{8,n}} & \text{else} \end{cases}
 \end{aligned} \tag{8.6}$$

This equation certainly looks more complex than the 1D equivalent and we will illustrate the fitting of three 2D Gaussians to a section of the diffuse scattering data used in the 2D examples before. The data and the fit result are shown in Figure 8.2. The observed data are shown in the top view graph. The maxima used to determine the starting values for the fit are marked by black circles. One of the major mistakes when using the fit routine of *KUPLOT* are wrong starting values. The calculated data are shown in the middle view graph using identical contour line settings as for the data. The bottom view graph finally shows the difference between observed and calculated data. The solid black line is the zero level, red lines mark negative values and blue lines stand for positive contour levels. Note that the contour line interval used for the difference plot is only  $\frac{1}{5}$  of the interval used for the observed and calculated data in order to make the differences more visible.

The macro file used for this example file is shown below. Again we assume the observed data are already loaded as data set one.

```

1  fit 1
2  #
3  mfen 3
4  func gau2,3
5  par
6  #
7  run
8  #
9  par 7,1
10 par 23,1
11 #
12 run
13 save
14 #
15 exit

```

First we enter the fit sub level (line 1) to fit the theory function to data set one. Before we select three 2D Gaussian theory functions in line 4 we set the window size for determination for maxima used as starting values to 3 data points. This will result in the three maxima marked in Figure 8.2. Line 5 displays the current parameter settings for checking. Next (line 7) the fit is started. As default the parameter  $p_{6,n}$  determining the angle between the principle axes of the Gaussian and the coordinate axes and the asymmetry parameters  $p_{7,n}$  and  $p_{8,n}$  will be kept fixed. After the first fit, the orientation of the Gaussian (parameter  $p_{6,n}$ ) for Gaussians one and three (the two strong peaks) are refined as well (lines 9–12). Finally the data sets containing the

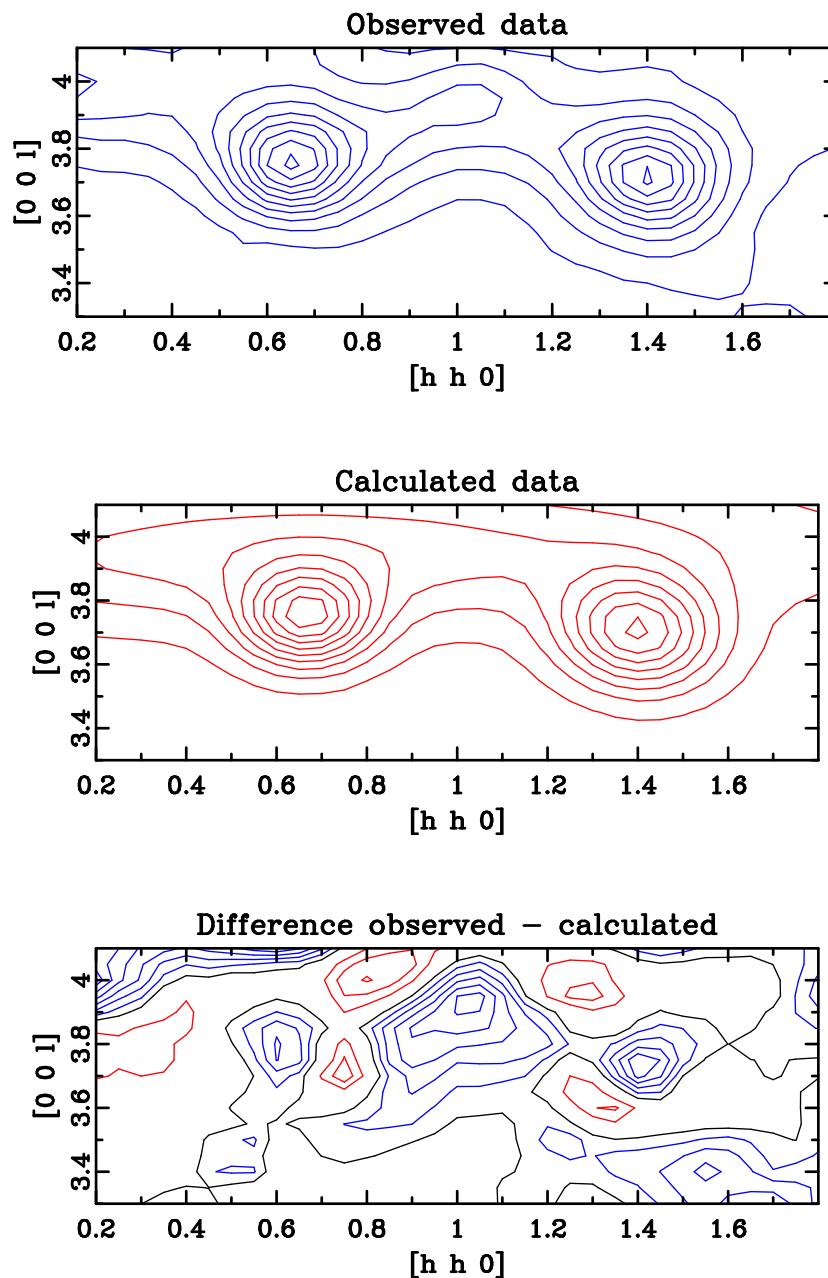


Figure 8.2: Fit results for 2D example

calculated data and the difference as well as the resulting parameters are saved. When fitting interactively the command `plot` within the fit sublevel will plot the current fit on the screen. Finally we want to have a look at the produced text file containing the fit results (file extension `.erg`) for this example fit. The line numbers are given for reference and as for the macro files not part of the actual output file. The first part contains general information about the fit such as a title (lines 2–3), the used theory function (line 4), the data file used (line 5), the number of data points and parameters (lines 6–7). Note that this is the total number of parameters including those not actually refined. Lines 8–10 show the current URF (see 8.1), the maximum number of iterations, the setting for maxima determinations and the used weighting scheme, in our case  $w_i = \frac{1}{z_i}$ . The next block of data in the output file show details of the fitting process itself. Line 14 lists the value for  $\sum(z_i - z(x_i, y_i, p))^2$  for the last and the previous iteration step. The difference between both is shown in the next line 15. A big difference normally indicates a *crashed* fit. The value of URF changes during the refinement and the final value is shown in line 16. If this value is not close to 1.0 at the end of the fit something has gone wrong. The next line shows the resulting R value and the expected R value. The ratio of these values is also known as the goodness-of-fit. Finally all correlations between fitted parameters that are greater than 0.8 are listed. We are lucky because our fit contains no highly correlated variables.

```

1 General fit parameter settings :
2   Title          : KUPLOT 2D fit demonstration
3                     January 1998
4   Fit function   : Gaussian (2D)
5   Data file name : test.fit
6   # of data pts. :      561
7   # of parameters :      25
8   Urf             :      .1000
9   Max. cycle     :      30
10  MFEN for maxima :      3
11  Weighting scheme : w(i) = 1.0/i
12
13 Information about the fit :
14  Sum n-1       : 1038.98           Sum n     : 1040.29
15  Difference    : 1.31653
16  Urf final    : 1.00047
17  R4 value     : .106378          R exp     : .077327
18
19 Correlations larger than 0.8 :
20  ** none **
21

```

All remaining information of the output file are results of the parameters used in the theory function. In line 22 the number and type of functions fitted are shown. Next we have the result for the overall background  $p_1$  and its standard deviation (line 24). The `pinc` in the last column indicates whether the parameter was refined (`pinc = 1`) or kept fixed to its starting value (`pinc = 0`). The values for each parameter and the corresponding `pinc` value can be altered via the command `par`.

```

22 Fitted 3 Gaussian(s) :
23
24  p( 1) : backgr. 1 : 69.2818    +- .998061    pinc : 1.
25

```

The rest of the output file contains three identical blocks with the resulting parameters of the three Gaussians fitted to the observed data. We show here just the results for Gaussian 1 (left maximum).

```

26 Gaussian : 1
27
28 p( 2) : peak      : 407.332    +- 11.3879   pinc : 1.
29 p( 3) : position x: .665616   +- .002372   pinc : 1.
30 p( 4) : position y: 3.74364   +- .002798   pinc : 1.
31 p( 5) : fwhm a     : .237793   +- .005545   pinc : 1.
32 p( 6) : fwhm b     : .235389   +- .005938   pinc : 1.
33 p( 7) : angle a,x : -98.8801  +- 85.1452   pinc : 1.
34 p( 8) : asym. a    : 1.00000   +- .000000   pinc : 0.
35 p( 9) : asym. b    : 1.00000   +- .000000   pinc : 0.
36 integral : 25.8344   +- 1.97653

```

Note that the asymmetry parameters were not refined, thus kept at the default setting for a symmetric function. The orientation for this Gaussian (line 7) has a very large error which can be understood from the resulting half width parameters (lines 31–32) which show a nearly isotropic result. Thus the definition of an orientation is quite arbitrary. This result is displayed here for demonstration purposes, normally the fit needs to be repeated with a fixed orientation parameter for this Gaussian. The third Gaussian however (right maxima) has a significant orientation parameter of  $p_{6,n} = 33(8)^\circ$ . Finally (line 36) the integral for each Gaussian with its standard deviation is given.

### 8.3 User defined fit functions

The final section of this chapter will demonstrate the use of an arbitrary fit function. The fit process is not different from the example described above, just the theory function is entered as *KUPLOT* type of expression. Assuming your theory function should be

$$y(x_i, p) = p_1 \cdot \exp(-p_2 x_i) \cdot \sin(x_i + p_3), \quad (8.7)$$

the corresponding command to define this theory function in the fit sublevel of *KUPLOT* would look like this

```
func fx,3,p[1]*exp(-p[2]*r[0])*sin(r[0]+p[3])
```

The parameter *fx* stands for a user defined function. The next parameter is the number of fit parameters to be used for that function. As last parameter, the function itself is given. The variable *r[0]* stands for  $x_i$  similar to the command *func* (see section 7.1). The fit parameters are represented by variables *p[i]*. Since the nature of the fit function is unknown, there is no calculation of default starting values. They have to be supplied by the user via the command *par*. Note that *KUPLOT* is calculating the expression once after the *func* command is given to perform a syntax check. If no proper starting values for *p[i]* have been defined this calculation might fail, e.g. by a division by zero. After the function is successfully defined, the fit process proceeds as described in the previous sections. The definition of a 2D theory function works exactly the same way, use *r[1]* for the values of  $y_i$ . The dimension of the user defined theory function (1D or 2D) is given by the data set used for the fit.

The derivatives needed for the least square refinement are calculated numerically in contrast to the usage of analytical derivatives used for the build in theory functions. In extreme cases the calculation of these derivatives and subsequently the fit itself might fail. In those cases it might be necessary to re-normalize the observed values or to use a specialized fit package after all.

## Appendix A

# KUPLOT commands

### A.1 Summary

Here is a list and brief description of valid KUPLOT commands. Further help can be obtained by typing the corresponding command name at the help prompt.

```

News      : Information on program updates
achx/y/z : Set labels for x, y and z-axis
alloc     : Allocate space for data set without loading one
angl      : Set angle between x- and y-axis
aver      : Set ratio between units on y and x-axis
branch   : Switch to discus, SUITE version only
buff      : Set reserved space outside of plotting frame
ccal      : Perform calculations with single data set
color     : Set pen colors
conv      : Calculate convolution of two data sets
costvalue : Set a value to be returned to DIFFEV
cmap      : Select color map
deriv     : Calculate the derivative
ecol      : Set color of error bars
etyp      : Set type of error bars
eval      : Evaluate expression
excl      : Exclude regions from 1D files
fill      : Set filling style
fit       : Enter least square fit sub level
fnam     : Toggle plot of file names in upper left corner
font      : Set font size, type and color
fset      : Set type of plot frame (axis, tick marks, ...)
frames   : More information about the use of frames
func      : Create data set from given function
glat      : Smooth data set (sliding average)
grid      : Toggle plotting of a grid at main marker points
hart      : Set type for 2D plots (bitmap, contours, ...)
hcol      : Set color of contour lines
hlab      : Labeling of contour lines
hlin      : Set contour line parameters (also for bitmap)
hpak      : Set number of contour line packages
htyp      : Set line type for contour lines
iden      : Toggle plotting of user, data and time information
inte      : Integrate data set
ksav      : Save data set in various formats
kpara    : Plots parameters fitted with DIFFEV
kcal      : Perform calculations with two data sets (add, ...)
lart      : Set type for 1D plot (line, steps, spline)
lcol      : Set line color

```

```

load      : Read data set
ltyp      : Set line type
lwid      : Set line width
mark      : Set tick mark interval
match     : Calculate scale and offset to get best match bw. data
mcol      : Set marker color
mean      : Calculate average and stand. deviation for data set
merge     : Merge data sets (2D only)
mouse     : Activate the mouse
mtyp      : Set marker type
msiz      : Set marker size
nexus     : NeXus file support routines
orient    : Set paper orientation (landscape, portrait)
plot      : Display plot
prin      : Print plot
ptyp      : Set marker type for maxima
rdef      : Read KUPLOT settings from file
rebin     : Allows rebinning data to user defined grid
reset     : Reset
sann      : Set text annotations for plot
save      : Save plot as hardcopy
sdef      : Store KUPLOT settings on disk
show      : Show various settings
sleg      : Sets legend of plot
smax      : Determines maxima of data set
smooth    : Smooth data set (Savitzky-Golay algorithm)
sort      : Sort data set
spline    : Interpolate data set on grid given by different set
skal      : Set plot window
tit1/2    : Set plot titles
window    : Select graphics window
variables : List of variables
errors    : List of error messages

```

## A.2 News

Here you can find updated info on the latest KUPLOT releases.

### 2023\_March

Added a Lanczos Filter to the 'smooth' command

### 2022\_February

Extensive internal revisions; added many options to work on 3D+value data 3D+value data are in a global storage

Added an optional data transformation to the load h5 command

Modified all FFT to use FFTW library, much faster

R-value works on 3-D data

### 2022\_March

The fit routine has been made more stable

A new parameter "sig" on the ccal command allows to create a data set with noise.

The FFT algorithm has been finished.

### **2021\_October**

All picture output is first written as postscript file and converted to the available formats via ImageMagick. This allows optional setting of the pixels per inch and the transparency. See ==> 'save' Furthermore a direct eps output was added

### **2021\_April**

Added the option to make line types of a data set negative, which will have the effect that this is drawn as a polygon. The fill command will fill just the area inside the polygon.

### **2021\_January**

Fixed the macro option to the ==> 'fit' command.

Added a double peak Pseudo-Voigt function for Kalpha1/Kalpha2 to ==> 'fit'

Added option to limit parameter range to ==> 'fit'

### **2020\_November**

Added a further optional parameter "decimal" to the 'load csv' command. This allows to read in data like "20.12" as well as "20,12" to give the same numerical value.

### **2020 July**

Added a new color scheme "pdf" that gives white colors at level zero, yellow to red colors for positive values and light to dark blue values for negative values. The ==> 'hlin' command has been modified to be better suited to PDF plots. New modes are: hlin 1, zmin, zdelta, n, %m !percentages with respect to ! zmax only hlin 1, zdelta, n, % ! Zmin is chosen automatically to ! give equal contour lines at ! positive and negative lines hlin 1, zdelta, n ! same on an absolute scale

### **2020\_June**

Added 'h5' for HDF5 capabilities to the load command.

The buttons in the interactive "mouse" mode have been revised and keyboard input has been added in case you do not have a three button mouse.

### **2019\_December**

Changed the fit algorithm to a more stable routine

### **2019\_November**

Added a ==> 'close' command that allows to deactivate plot windows

**2019\_April**

Added "latex" option to ==> 'save' command  
Extended 'merge' command to 2D data.  
Corrected fit weighting scheme "DAT" to  $1/\text{sig}^{**2}$   
Added a "thermal" color map  
Added a 2D option to build a ==> 'func' from a reference data set

**2019\_February**

Added a 'blank' separator to 'load csv, separator:blank'  
Added a new menu "2dm" that allows to concatenate several 2D data sets into a single 3D data set

**2018\_Oct**

Added the trend of the best value to the 'kpara' plot

**2018\_June**

Revised the reaction to a CTRL-C  
Added a ==> 'set error, ... , "save" option

**2018\_May**

Added the possibility to use a macro as fit 'function'

**2018\_Feb**

Added a command 'kpara' to create a plot for DIFFEV  
Added an optional parameter "partial" to the ==> 'rvalue' command  
Added optional parameters to the "load csv" command

**2018\_Jan**

The new command 'costvalue' allows to set a global value for the cost function that is returned to DIFFEV.  
The logical comparisons may now take the operators: <, <=, ==, /=, >=, >/ The classical fortran77 operators are still valid  
New logical functions "isvar" and "isexp" can be used within an "if" construction. See help entry ==>'function' in the general "Command\_lang" section.

**2017\_Oct**

The 'ni' format may now have a leading header that consists of lines that start with a '#' in the first column

**2017\_Sep**

Throughout the program the internal calculation of random numbers was changed to the FORTRAN 90 intrinsic function.

Added error messages for variable nx, ny, modified variable np to yield the total number of data points for 2D data sets as well.

**2017\_May**

The 'save' command can now save graphics as a PDF file

**2017\_Jan**

Added the option to read CSV files to 'load' command.

An unfortunate typing error in News/2016\_oct regarding the new refinement variable ref\_para[1...] (was misspelled as ref\_param[1...]) is corrected in the on-line help.

**2016\_Dec**

At a few select points colors are introduced into the output. Currently these are just the error messages.

**2016\_Oct**

Global variables have been introduced that use the same syntax as user defined variables. This include just "pi" and variables related to the refinement. DIFFEV sets the value to these variables: ref\_generation Current generation ref\_member Current population size ref\_children Current children size ref\_dimension Number of parameters ref\_kid Current child Updated for DISCUS and KUPLOT only ref\_indiv Current individuum Updated for DISCUS and KUPLOT only ref\_para[1..] Current trial parameters for current child

All colors can be referenced by their respective names as well as by their traditional numbers. See 'lcol', 'hcol', 'mcol' 'ecol'

All line types can be referenced by descriptive names as well as by their traditional numbers. See 'ltyp', 'htyp'

All marker types can be referenced by descriptive names as well as by their traditional numbers. See 'etyp', 'mtyp', 'ptyp'

**2016\_Feb**

Minor upgrade, added a threshold function to ==> 'ccal'

**2015\_Dec**

Minor upgrade, increases the size of bitmaps KUPLOT can plot

## 2015\_June

Starting with Version 5.1, we have migrated to a X-Window environment for WINDOWS as well. As a small side effect, the technique to jump to the desired folder has changed slightly. See the help entry on "cd" in the general "Command\_lang" section for further information. The process is described in the package manual as well.

### A.3 manual

```
manual ["section": {"suite" | "discus" | "diffev" |
    "kuplot" | "package" | "mixscat"}
    [, "viewer:<name>"]
```

Opens a PDF viewer for one of the Manuals

The section defaults to the current program section that you are working with. On Linux systems, the viewer defaults to "qpdfview", on Windows system it defaults to "firefox". If DISCUS does not find the default or the user provided viewer, DISCUS will search a list of common PDF viewers. If none is found an error message points to the folder that contains the manuals.

### A.4 2dm

**2dm**

Branches into the '2dm' menu. This menu allows you to concatenate several 2D i.e. xy data sets into a single 3D data set.

Valid commands at this level are:

**reset**

**reset**

Reset all numerical values back to system start.

**run**

**run**

Performs the actual calculation. The map will be stored in the next free data set number.

**back**

```
back "off"
back "csv", <filename>
    , skip:<nskip>,
    colx:<icolx>, coly:<icoly>, coldx:<icoldx>, coldy:<icoldy>,
    separator:<name>
    scale:<factor>
```

Defines the name and type of a background file. The y-values will be subtracted from the corresponding y-values of the files read with 'load'. If the command is used with the parameter "off", no background file will be read.

Optionally a scale factor for the background can be defined with scale:<factor>

The remaining optional parameters are identical to those of the == 'load csv' command at the main menu.

### **load**

```
load "csv", "filename_format", LOOP
    , skip:<nskip>,
    colx:<icolx>, coly:<icoly>, coldx:<icoldx>, coldy:<icoldy>,
    separator:<name>
```

Defines the name and type of the 2D input files. The file name must be specified within double quotation marks and contain a format descriptor that will be replaced by the formal "LOOP" loop index. The loop index must be called "LOOP".

A second, outer loop can be defined as well, which must use the counter variable "SLOW".

The remaining optional parameters are identical to those of the == 'load csv' command at the main menu.

### **loop**

```
loop "counter":["LOOP"|"SLOW"],
    "start:"<start_number>, "end:"<end_number>
    , "step:"<step_number>
    , "miss":{"error"|"ignore"|"blank"}
```

Defines the range of input file numbers that will be used for the "LOOP" or "SLOW" value on the ==> 'load' command.

The optional parameters "start:", "end:" and "step:" default to a value of 1. Allowed values for the numerical values are valid expressions.

The optional parameter "miss:" with default "error" sets the rules for missing or corrupt input files. With the default value "error", any missing or corrupt input file will interrupt the ==> 'run' command and no map will be produced. With "ignore" the erroneous file will simply be ignored and the map will be shorter by one (ore more) lines. With "blank" the corresponding line in the map will be set to a value of zero. The status of "miss" is a single status. If both loop counters "LOOP" and "SLOW" are used, use a common and identical status to describe the handling of missing files.

### **xrange**

```
xrange "xmin":{"xmin"|"x_min"}, "xmax":{"xmax"|"x_max"}
```

Defines the x-range of the input files that is to be written into the 3D map. Both parameters default to "xmin" respectively "xmax" the actual minimum and maximum values of the input files.

## yfunction

```
yfunction loop:LOOP
yfunction loop:inc
yfunction loop:<expression>
```

Defines the function that will be used to calculate the y-axis values of the map.

Default is "loop:LOOP". The y-axis will be the loop numbers in the range as defined on the ==> 'loop' command.

As these numbers may be unnecessarily large, the option "loop:inc" will increment the y-axis values from 1 in steps of 1.

Finally you might want to calculate the y-values from a user defined function, or use values in a previous data set: loop:1+LOOP-start\_number If start\_number is a variable that corresponds to the start value on the ==> 'loop' command, this function would be identical to the "inc" option. loop:y[1,LOOP] Would take the y-value of data set 1 point LOOP as y-value for the map. This might be appropriate if y[1,\*] contains temperature or pressure or time or ... values for the 2D data sets.

## example

Loop with a single counter

```
reset
back off
back csv, empty_measurement.xy, scale:1.0, skip:0
load csv, "DATA/my_data.%4D", LOOP, colx:1, coly:1, skip:0
loop counter:LOOP, start:1, end:201, step:2, miss:blank
yfunc loop:LOOP
xrange xmin:xmin, xmax:5.00
run
```

Loop with a two counters, SLOW and LOOP

```
reset
back off
back csv, empty_measurement.xy, scale:1.0, skip:0
load csv, "DATA/my_data_%5D.%4D", SLOW, LOOP, colx:1, coly:1, skip:0
loop counter:SLOW, start:100, end:101, step:1, miss:ignore
loop counter:LOOP, start:1, end:201, step:2, miss:ignore
yfunc loop:LOOP
xrange xmin:xmin, xmax:5.00
run
```

## A.5 achx/y/z

```
achx {<string> | "OFF"} [, {"lin" | "log"}]
achy {<string> | "OFF"} [, {"lin" | "log"}]
achz {<string> | "OFF"}
```

This command sets the labels for the x- or y-axis. The label for the z-axis is only active when bitmaps are plotted and is used to label a wedge on the left side of the plot. As for the other axes the parameter "OFF" will turn off the plotting of the wedge which is the default. The optional parameter "lin" or "log" allows the user to toggle between a linear and logarithmic axis.

Because KUPLOT treats the ',' as a parameter separator it can not be used within the label <string>. The command allows to build a label from a format text and variables/numbers. It is the same mechanism as described in -> filenames. If the given parameter is "OFF", no label AND no numbers will be plotted at the corresponding axis.

Examples:

```
achx 2 Theta      : will label x-axis with '2 Theta'
achy "PSD #%d",43 : will label y-axis with 'PSD #43'
```

## A.6 adt

**adt**

Menu to read and transform a volume of reciprocal space data from ADT treatment.

### Examples

**Examples**

To extract a volume into a normalized orientation use:

```
get cell:name.cell, log:screen get data:name.mrc set mode:volume set limits:[6.0,6.0, 6.0] set sigma:[0.00,0.00, 0.00] set steps:[0.05,0.05, 0.05] show extract
```

The volume will be extracted up to +- hkl as defined by set limit: The new grid will have steps in reciprocal space as defined by 'set steps:', here steps are 0.05 reciprocal lattice units along each of the axes a\*; b\*; c\*. Data are placed into the next data set with a\*; b\*; c\* parallel to the 'x', 'y', 'z' axes of the data set.

To extract a set of planes:

```
get cell:name.cell, log:screen get data:name.mrc set mode:plane set normal:[0.0,0.0,1.0, 1.00, 4.0] set sigma:[0.15] set abs:[0.05,0.0, 0.0, 6.0, 0.0, 0.0] set ord:[0.0, 0.05,0.0, 0.0, 6.0, 0.0] set zero:[152, 098, 50] show extract
```

In this example planes normal to [uvw] = [001] will be extracted. The individual planes are defined by [uvw]\*[hkl] in steps of 1, the fourth parameter of the "set normal:[...]" command. In this example, layers up to +- [uvw]\*[hkl] = 4 (the fifth parameter) will be extracted. Within each plane data are sampled at steps defined by "set abs:[]" and set ord:[]]. Here steps are [0.05, 0, 0] and [0, 0.05, 0] respectively. Within each plane the maximum hk0 is +- 6. Along the normal data are integrated in a range of [uvw]\*[hkl] as +- sigma around each plane. The new data set will consist of 'continous' intensity distribution with each plane along the "x" and "y" coordinates and discrete steps along "z", each "z" corresponding to an individual plane.

To extract a set of rods:

```
get cell:name.cell, log:screen get data:name.mrc set mode:rods set rods:[0.00, 0.00, 0.05, 85.] set sigma:[0.15, 0.15] set abs:[1.00, 0.00, 0.00, 6.0, 0.0, 0.0] set ord:[0.00, 1.00, 0.00, 0.0, 6.0, 0.0] set zero:[152, 098, 50] show extract
```

Rods will be extracted parallel [001] in steps of [0.0, 0.0, 0.05], up to +- [0.0, 0.0, 0.05]\*85 = +- [0, 0, 4.25]. Within the hk0 plane integer steps along h and k are performed as defined by the 'set abs:[]' and 'set:ord:[]' commands. Rods to +- h=6; k=6 are extracted. Around each rod data are integrated in the plane normal to the rod by delta h = +- 0.15 and delta k = +- 0.15. In the

new data set the "x" and "y" coordinates will be discrete steps from rod to rod while the "z" coordinate will be the 'continuous' intensity distribution along each rod.

Integrated Bragg reflections will be extracted with:

```
get cell:name.cell, log:screen get data:name.mrc set mode:bragg set lattice:P set sigma:[0.15, 0.15, 0.15] set steps:[1.00, 1.00, 1.00] set limits:[6,6,6] set output:bragg.data
```

All Bragg reflections to  $hkl = \pm 6$  will be extracted. Reciprocal space is integrated in the volume of  $\pm$  sigma around each integer hkl position. Only Bragg reflections that are allowed for a P-centered lattice are extracted.

### **extract**

```
extract
```

Extract the volume, planes, rods, Bragg reflections as defined by the current setting.

### **get**

```
get cell:<cell_file>
```

Read the adt cell file. DISCUS assumes this to be a plain ASCII text file that contains the orientation matrix for the data in the ==> 'data' file.

```
get data:<file_name>
```

Read the adt data file. DISCUS assumes this to be a MRC file format that has been processed to contain a reconstructed reciprocal space volume.

### **show**

```
show
```

Show the current settings.

### **set**

```
set {"mode:planes" | "mode:rods" | "mode:volume", | "mode:bragg"}
set {"sigma:"<value>"}
set {"abs:[<h>,<k>,<l>, <maxh>, <maxk>, <maxl>]"}
set {"ord:[<h>,<k>,<l>, <maxh>, <maxk>, <maxl>]"}
set {"normal:[<u>,<v>,<w>, <steps>, <maxsteps>]"}
set {"rods:[<h>,<k>,<l>, <nsteps>]"}
set {"lattice:"<Bravais>"}
set {"zero:[<ix>, <iy>, <iz>]"}
set ["output:"<file_name>}
"mode:volume"
```

Reciprocal volume will be extracted from the data and placed into the next data set. Within this data set the "x", "y" and "z" axes will be parallel to  $a^*$ ,  $b^*$  and  $c^*$ .

```
"mode:planes"
```

Reciprocal planes will be extracted from the data and placed into the next data set. Within this data set the abscissa and ordinates within the planes will be the "x" and "y" axes while the zone axis will be the "z" axis. Each plane will have been integrated along the zone axis.

```
"mode:rods"
```

Reciprocal rods will be extracted from the data and placed into the next data set. Within this data set the abscissa and ordinates from rod to rod will be the "x" and "y" axes while the 'continuous' intensity along each rod will be the "z" axis. Each rod will have been integrated within the plane normal to the rod at a width set by "sigma:"

```
"mode:bragg"
```

Integrated Bragg reflection intensities are extracted.

```
set {"abs:[<h>,<k>,<l>, <maxh>, <maxk>, <maxl>]"}
```

Set the abscissa increment vector with a plane and its maximum value.

```
set {"ord:[<h>,<k>,<l>, <maxh>, <maxk>, <maxl>]"}
```

Set the ordinate increment vector with a plane and its maximum value.

```
set {"normal:[<u>,<v>,<w>, <steps>, <maxsteps>]"}
```

Set the normal vector to a plane. Coordinates of this vector are in direct space coordinates. Reciprocal points that fulfill  $[uvw]^*[hkl] = N^*steps \pm \sigma$  are considered to be part of the plane. All planes to  $[uvw]^*[hkl] = \pm maxsteps$  are extracted

```
set {"rods:[<dh>,<dk>,<dl>, <nsteps>]"}
set {"rods:[<h>,<k>,<l>, <dh>,<dk>,<dl>, <nsteps>]"}
set {"rods:[<dh>,<dk>,<dl>, <mh>,<mk>,<ml>]"}
```

Set the rod direction in reciprocal space. The format "rods:[<dh>,<dk>,<dl>,<nsteps>]" is understood as steps along the rods. Data up to  $\pm [dh, dk, dl]^*nstepsd$  will be extracted. The format "rods:[<h>,<k>,<l>,<dh>,<dk>,<dl>,<nsteps>]" is understood as rods parallel to  $[hkl]$  at steps  $[dh, dk, dl]$ . Vectors  $[hkl]$  and  $[dh, dk, dl]$  must be parallel. Data up to  $\pm [hkl]^*nstepsd$  will be extracted.

The format "rods:[<dh>,<dk>,<dl>, <mh>,<mk>,<ml>]" is understood as

steps along the rods. Data up to  $\pm [mh, mk, ml]$  will be extracted. The vectors  $[dh, dk, dl]$  and  $[mh, mk, ml]$  must be parallel.

```
set {"zero:[<i_x>, <i_y>, <i_z>]"}
```

By default reciprocal origin is assumed to be at the central pixel of the input data file. This command allows to adapt the zero point.

```
set ["output:<file_name>"]
```

The integrated Bragg reflections can be written into a file with 16 columns: No., 1.00, 0.00, Weight, observed pixels(x,y,z), Calculated pixels(x, y,z) Calculated real valued (h, k, l) , Nominal integer (h,k,l) This file predominantly serves to refine the orientation matrix. The weight is 1/integrated intensity.

## A.7 alloc

```
alloc <name>,<len>
alloc <name>,<x>,<y>
```

This command allocates space for a new data set. This is e.g. required if a data set should be created using the FORTRAN interpreter of KUPLOT. The command 'alloc <name,<len>' allocated space for <len> points for a 2d data set. To allocate a 3d data sets, the size in x- and y-direction is given after the name of the data set. The parameter <name> just specifies a name for the data set (also used as default file name if the data set is saved).

## A.8 angle

```
angle [<angle>]
angle data:<idata>
```

This command allows to set the angle between the x- and y-axis for the currently active frame. Suitable angles are between 30 and 120 degrees. If the command is entered without parameters, the value is set to 90.0 degrees.

If the data in data set <idata> were read as HDF5 files, KUPLOT can calculate the angle from the internal values for this data set. this allows an automatic angle.

## A.9 aver

```
aver [<yx>]
aver data:<idata>
```

This command sets the aspect ratio between the y- and the x-axis. A value of <yx>=1.0 will force one unit on the x-axis to be as long as one unit on the y-axis. The value <yx> is the ratio y/x of the length of a unit of both axis. Is the parameter <yx> omitted, the plot will fill the complete drawing area.

If the data in data set <idata> were read as HDF5 files, KUPLOT can calculate the aspect ratio from the internal values for this data set. this allows an automatic aver.

## A.10 bond

```
bond
bond <ib>,<dist> [,<sig> [,<it>,<ic>,<iw>]]
```

This command enables the plotting of lines between points that have a given distance, e.g. to include bonds in a crystal file plot. The first parameter <ib> is the number of the definition you are going to alter. The second parameter <dist> is the distance of points that shall be connected by a line. The distance is given relative to the current x-axis. To calculate the distances, the aspect ratio and the angle between the axes is used. Thus you will get the desired connection only for the correct aspect ratio (-> aver) and angle between the axes (-> angl). The other parameters are optional. The value <sig> sets the allowed relative deviation from the

given distance, e.g. a value of 0.1 will allow distances of +- 10% of the given value. The parameters `<ic>`, `<it>` and `<iw>` set the line color, line type and line width of the connecting line, respectively (-> `lcol`, `ltyp`, `lwid`).

In order to disable the plotting of a defined bond, just specify a zero distance, i.e. 'bond `<ic>,0.0`'.

## A.11 branch

```
branch discus [, "-macro" <macro_name> [ <par1> [ , <par2> ...]]]
branch diffev [, "-macro" <macro_name> [ <par1> [ , <par2> ...]]]
```

Active within the discus suite only!

Branches to the "discus" or "diffev" section.

Within this section any standard DISCUS command can be given. The behavior of "discus" is essentially the same as in the stand alone version. Likewise for DIFFEV.

The main use will branch to DISCUS while the discus section is run via `run_mpi` from a DIFFEV slave.

Optionally the "-macro" qualifier instructs the suite to run the macro `<macro_name>` (with its optional parameters) before the interactive session is started.

## A.12 buffer

```
buffer <all>
buffer <left>,<right>,<bottom>,<top>
```

This command allows to alter the amount of drawing space KUPLOT reserves outside the plotting area for labels, titles etc.. for the current frame. The parameters given are relative to the size of the plot, e.g. a value of 0.3 will save 30% of the width/height of the plot itself for axis numbers, labels etc. - note that if the reserved space is too small text might be wrapped around or cut off.

## A.13 branch

```
branch discus
```

Active within the discus suite only!

Branches to the "discus" section.

Within this section any standard DISCUS command can be given. The behavior of "discus" is essentially the same as in the stand alone version.

## A.14 ccalc

```
ccalc <oper>,<field>,<ik> [,<value>]
```

This command allows data manipulation. All functions could also be performed using the FORTRAN type interpreter, but especially for large data sets this can be very slow.

The general syntax is the operation `<oper>` (see list below) that should be performed as first parameter followed by the field identifier `<field>` (see below), followed by the number of the data set `<ik>`. Some operations require an additional parameter `<value>`.

Here is a list of allowed operation parameters `<oper>`:

```
"inv"   : x -> 1/x      "squ"   : x -> ***2
"log"   : x -> ln(x)    "add"   : x -> x + <value>
"exp"   : x -> exp(x)   "mul"   : x -> x * <value>
"sqr"   : x -> sqrt(x)  "abs"   : x -> |a|
"thr"   : x < value -> -9999; value is thus a threshold,
           data below are considered invalid
"sig"   : x _> x + gran(dx) ! Add Gaussian distributed noise
```

The value "x" in the table above represents the coordinate selected by the `<field>` parameter. Allowed values are:

```
"wx"   : x value of data set
"dx"   : standard deviation of x
"wy"   : y value of data set
"dy"   : standard deviation of y
"wz"   : z value of data set (for 3d data sets only)
```

## A.15 close

```
close [<iwin> [, <iwin> ...]] ]
```

KUPLOT creates persistent Xwindow frames for the plots. During the run of the program these can usually not be closed. This command deactivates the listed windows. You may then close them.

## A.16 color

```
color <ic>,<r>,<g>,<b>
color map:<name> [, range:[<min>,<max>]]
color map:<name> [, range:current]
```

This command allows the user to change the color settings for the different pens. The parameter `<ic>` gives the pen number (0 for background) and `<r>,<g>,<b>` are the red, green and blue values to be used ranging from 0.0 to 1.0. If no parameters are given, the current settings are displayed.

The second command form allows to use a color map for the lines. The first line will receive the lowest color entry in the color map, the last line will receive the highest color index. Thus the colors of multiple lines will change gradually.

Color maps are "fire", "ice", "pdf", "thermal", "kupl", "gray" color map:invert Reverses the color sequence.

The command "color:user" will use the current color map that has been build with ==> set color" or by explicitly defining the entries in the KUPLOT ==> variable "cmap"

The optional range allows to limit the colors to a section of the full color map. Specify the limits in the range 0. to 1.0 Defaults to "range:[0.0, 1.0]" Given this default, the option "range:current" will use the last user specified range.

## A.17 convolute

```
convolute <ik1>, <ik2>
```

This command calculates the convolution of data sets <ik1> and <ik2> and creates a new data set with the result. Both data sets must be on the same equidistant grid !

## A.18 cost

```
costvalue <rvalue> [, <partial_rvalue1> ...]
```

Set a value that is to be returned to DIFFEV as the overall R-value. It will overwrite the current weighted R-value that was calculated with the ==> 'rval' command. The value of <rvalue> can be any expression or real valued variable. This allows to calculate a combined, weighted R-value from several different calculations like single crystal, powder diffraction, PDF, etc. If the partial R-values are provided as well, they will overwrite the corresponding partial R-values.

## A.19 cmap

```
cmap { "gray" | "fire" | "ice" | "kupl" | "thermal" | "pdf" }
cmap { "read" | "write"}, <fname>
cmap "invert"
```

This command allows to choose the color map to be used for printing bitmaps (see 'hart' command). The three build in color map are "gray", "fire", "thermal" and "kupl" (the previous default color map). "gray" gives a white color at low data values and a black color at high data values. "fire" gives a black color at low data values and a red to yellow to white color with increasing data values. "ice" gives a black color at low data values and a blue to lighter blue color with increasing data values. "thermal" combines "fire" and "ice", with a "fire" map for positive data values and "ice" for negative data values. "pdf" is similar to "thermal". The zero level is plotted white, positive values are yellow to red to dark red, negative values are light blue to dark blue to black. This color map is best combined with the contour lines as: hlin 1, <inc>, <n> {, "%" } as this ensures that the contour lines are equally spaced around the zero level. Color maps can also be read from a file using the command 'cmap read,<fname>'. The format is of the form '#RRGGBB' for each color map entry on a new line and 'RR', 'GG' and 'BB' are the 'red', 'green' and 'blue' values given in hexadecimal ranging from 00 to FF. The current color map can be saved to a file using the command 'cmap write,<fname>'. Finally 'cmap invert' will simply invert the current color map.

The RGB values of the current color map can be shown using the command 'show color'. The color map entries can also be altered using the variable 'cmap [<ic>,3]'

## A.20 derivative

```
derivative <ik> [,<order>]
```

This command calculates the <order>th derivative of data set <ik>. The result is stored as new data set. If <order> is omitted, the first derivative is calculated.

## A.21 dsave

```
dsave "gsas",<file> [,<iname>]
dsave "xml",<file> [,<dtd-path>]
dsave "merge",<file>
```

The parameter "gsas" allows one to save all loaded data sets in GSAS file format. NOTE THAT THE CURRENT IMPLEMENTATION ASSUMES A LUJAN STYLE GSAS FILE.

The parameter "xml" will save the plot on PlotML (xml) format for viewing with 'plplot'. Currently only 1D data are supported. The optional second parameter allows one to specify the location of the DTD file needed to render the XML file.

The parameter "merge" will save a xyz-type file from all the loaded data sets with the following condition x=x, y=data set number and z=y. This can then be loaded using load zz, ..

## A.22 ecolor

```
ecolor <ik>,<c>
```

This command sets the error bar color for the data set <ik>. The colors are numbered like the pens on a HP7475 plotter. Allowed values for <c> are listed in the table, Alternatively you can use the color name (without blanks i.e. darkblue etc.)

The allowed default color values <icol> are listed below. Note that they can be changed by the user via the command 'color'.

1 : red	5 : yellow	9 : dark blue	13 : cyan
2 : green	6 : black	10 : dark magenta	14 : dark cyan
3 : blue	7 : dark red	11 : dark yellow	15 : white (!)
4 : magenta	8 : dark green	12 : gray	

## A.23 etype

```
etype <ik>,<p>
```

This command sets the type of error bars to be use for the data set number <ik>. Allowed values for <p> are:

0 : {"noerror" "none"} : no error bars are drawn
1 : "xbar" : only error bars in x-direction
2 : "ybar" : only error bars in y-direction
3 : "cross" : error bars in both directions

## A.24 excl

```
excl <ik>,<left>,<right>
```

This command allows the user to exclude the area between *<left>* and *<right>* from data set *<ik>*. The values within the range are replaced by the linear interpolation between *<left>* and *<right>*, i.e. the closest data points to the specified values. If *<left>* is outside the data range, all points up to *<right>* are removed. If *<right>* is outside the data range, all points from *<left>* onwards are removed.

## A.25 fill

```
fill <ik>,<fc>,<ft> [,<xmin>,<xmax>] [,<ymin>,<ymax>]
```

This command sets the filling style for data set *<ik>*. The data set is treated as a polygon and is filled between the data and the baseline. The fill color is defined by *<fc>* (see lcol for color definitions). The following fill types *<ft>* are currently supported:

```
1: "solid"      : solid fill      5: "bsolid"   : type 1 plus border
2: "slash"       : hashed (/)     6: "bslash"   : type 2 plus border
3: "backslash"  : hashed (\)     7: "bback"    : type 3 plus border
4: "cross"       : cross hashed  8: "bcross"   : type 4 plus border
```

The filled area is defined by the data points and a baseline, by default the connection between the end points of the current plotting area. However, the area to be filled can be limited using the optional parameters *<xmin>*, *<xmax>* and *<ymin>*, *<ymax>*.

## A.26 fit

```
fit
```

This command enters the fit sub level and data set *<ik>* is taken as target data set. For an easy start, select a fit function with 'func' and start the fit using the 'run' command. The result can be displayed with 'plot' and saved with 'save'. For more detailed information see the list of commands below:

### commands

```
Valid commands in this sub level are:
@      : Executes a macro (see general help)
=      : Algebra (see general help)
cycle  : Sets maximum number of fit cycles
echo   : Echoes a string on the screen (see general help)
eval   : Evaluates an expression (see general help)
exit   : Exits fit sub level
func   : Sets fit function
help   : Gives on-line help for 'fit' sub level (see general help)
macro  : Writes current fit parameters to a macro file
mark   : Sets interval for tick marks
mfen   : Sets parameter for maxima determination (fit start values)
output : Toggles fit screen output on/off
```

```

para   : Sets/modifies fit parameters
plot   : Plot results
range  : Sets data range used for refinement
run    : Start the fit
save   : Save fit results
show   : Show settings and fit results
skal   : Sets region for plotting window (see general help)
system : Executes operating system command (see general help)
urf    : Sets urf value
wait   : Waits for user input (see general help)
wic   : Sets weighting scheme

```

## convergence

```
convergence ["status:on" | "status:off"] [, "dchi:<delta>"]
            [, "pshift:<max>"] [, "conf:<level>"] ["chisq:<value>"]
```

Allows the user to define convergence criteria.

If the status is set to "on", the convergence criteria are used. Otherwise the refinement will run for the cycles defined by ==> 'set cycle'.

dchi:<delta> If the value of Chi\*\*2/(Ndata-Npara) decreases by less than <delta> convergence is reached. Defaults to 0.5 pshift:<max> If all refinement parameters change by less than |DeltaP/Sigma| convergence is reached. Defaults to 0.005 conf:<level> If the confidence level is greater than <level> convergence is reached. Defaults to 0.010 chi2:<level> If the value of Chi\*\*2/(Ndata-Npara) falls below <level> convergence is reached. Defaults to 0.500 Convergence is reached, if: Either one of the three conditions: (dchi AND pshift AND conf) or (dchi AND pshift>0) or chi2 are met.

## cycle

```
cycle <n>
```

This command sets the maximum number of cycles to <n>. If a minimum is found in less cycles, the fit will stop before <n> cycles are finished.

## func

This command allows to set the fit function. After this command is called, KUPLOT will calculate starting values and destroy possible previous fit results, which should be saved ('save') before this command is called. The starting values for some functions use the 'smax' function. You can set the window size using the 'mfen' command. See also -> 'smax'. The following table shows the currently allowed functions:

function	2d	3d	what	<par>
"poly"	yes	no	polynomial	order of the polynomial
"cheb"	yes	no	Chebyshew poly.	order of the polynomial
"gaus"	yes	yes	Gaussian	number of Gaussians
"lore"	yes	no	Lorentzian	number of Lorentzians
"pseudo"	yes	no	Pseudo-Voigt	number of Pseudo-Voigts
"dpseudo"	yes	no	Pseudo-Voigt	number of Double peak Pseudo-Voigts
"fx"	yes	yes	User defined	# parameters and function

```
"ma"      yes  yes  User macro      # parameters and macro name
"back"    yes  no   Background     # of background parameters
"gsas"    yes  no   TOF profile   see text
```

Select a fit function from the list below for more information:

### **poly**

```
func "poly" [,<par>]
```

A polynomial function  $p[1] + p[2]*r[0] + p[3]*r[0]**2 + \dots$  is used. The parameter is the order of the polynomial.

### **gaus**

```
func "gaus" [,<par>]
```

xY data: The fit uses <par> Gaussian functions. Two parameters (no. 1 and 2) are refined for the background and each Gaussian peak used four parameters:

```
p[3] Peak height
p[4] Position
p[5] FWHM
p[6] Asymmetry
```

A Gaussian function with linear background is used as:  $p[1] + p[2]*r[0] + p[3]*\exp(-4*\ln(2)*(r[0]-p[4])**2/p[5]**2)$  Note that the peak height rather than the integral intensity is refined.

xyZ data: The fit uses <par> Gaussian functions. One constant background parameter is used and each Gaussian peak used 8 parameters:

```
p[2] Peak height
p[3] Position x
p[4] Position y
p[5] FWHM x
p[6] FWHM y
p[7] Rotation of the Gaussian around the z-axis
p[8] Asymmetry x
p[9] Asymmetry y
```

### **lore**

```
func "lore" [,<par>]
```

The fit uses <par> Lorentzian functions. Two parameters (no. 1 and 2) are refined for the background and each Lorentzian peak used four parameters:

```
p[3] Peak height
p[4] Position
p[5] FWHM
p[6] Asymmetry
```

A Lorentzian function with linear background is used:  $p[1] + p[2]*r[0] + p[3]*p[5]**2/(p[5]**2 + 4*(r[0]-p[4])**2)$  Note that the peak height rather than the integral intensity is refined.

**pseudo**

```
func "pseudo" [, <par> [, <origin> [, <n_backgrd> ]]
```

The fit uses <par> Pseudo-Voigt functions. <n\_backgrd> parameters (no. 1,.. to <n\_backgrd>) are refined for the background and each Pseudo-Voigt peak used six parameters:

```
p[3] eta
p[4] integral intensity
p[5] Position
p[6] FWHM
p[7] Asymmetry parameter 1
p[8] Asymmetry parameter 2
```

The Pseudo-Voigt function is used in the form:

```
background + Intensity*(eta*L + (1-eta)*G)*Asymmetry
```

with Lorentzian part as:

```
L = 2/pi*FWHM / ( FWHM**2 + 4*(x-POSITION)**2 )
```

and Gaussian part as:

```
G = 2*sqrt(ln(2)/pi)/FWHM * exp ( -4ln(2) * (x-POSITION)**2/FWHM**2 )
```

The asymmetry function is defined as:

```
ASYMMETRY = 1 + (p[7]*FA(z) + p[8]*FB(z))/ tan(POSITION)
FA(z) = 2*z*exp(-z**2)
FB(z) = 2*(2*z**2-3)*FA(z)
z = (x-POSITION)/FWHM
```

The background is calculated as a polynomial of order (n\_backgrd-1):

```
p[1] + p[2]*(x-origin) + p[3]*(x-origin)**2 + ...
or more exactly:
SUM p[i]*(x-origin)**(i-1)
```

where origin is the third optional parameter from the command line. If this parameter is omitted, the origin is set to zero. To describe a locally bent background, one should set the origin of the background to a position within the refined x-range instead of zero. The value of the origin is not refined.

The last optional parameter defines the number of background parameters to be used. If it is omitted, the number of background parameters is set to 2.

**dpseudo**

```
func "dpseudo" [, "peaks:<par>"] [, "origin:<origin>"]
[, "nback:<n_backgrd>"]
[, "wave:<symb>, ["axis:tth" | "axis:Q"]]
```

All parameters use the form: "par\_name:par\_value".

Available parameters are:

```
"peaks:<par>"]
```

Number of double peaks within the fit range. Defaults to 1.

```
"origin: "<origin>"]
```

Origin of the background polynomial. Defaults to 0.0

```
"nback: "<n_backgrd> ]
```

Number of parameters for the background polynomial. The order of the polynomial will thus be <n\_backgrd>-1 Defaults to 2.

```
"wave: "<symbol>
```

Wave length symbol. Should be one of: Ti12, Cr12, Fe12, Co12, Cu12, Mo12, Ag12, Ta12, W12, Au12 Capitalization is not relevant. Defaults to "Cu12"

You can also give the wave length as symbol like "Cu", or as a numerical expression like "1.54". In any of these cases the intensity of the second peak is set to zero and the function essentially fits a single Pseudo-Voigt peak.

```
"axis:tth" | "axis:Q"
```

Define if the x-axis is in steps of 2Theta or steps of Q=2PIsin(Theta)/lambda. Capitalization is not relevant. Defaults to "tth"

The fit uses <par> idouble peak Pseudo-Voigt functions. <n\_backgrd> parameters (no. 1, .. to <n\_backgrd>) are refined for the background and each double peak Pseudo-Voigt peak used seven parameters:

```
p[3] eta
p[4] integral intensity
p[5] Position
p[6] FWHM
p[7] Asymmetry parameter 1
p[8] Asymmetry parameter 2
p[9] Itwo: Intensity ratio I(Ka2)/I(Ka1)
```

The Pseudo-Voigt function is used in the form:

```
background + Intensity*      (eta*L + (1-eta)*G)*Asymmetry
+ Intensity*Itwo*(eta*L + (1-eta)*G)*Asymmetry
```

with Lorentzian part as:

```
L = 2/pi*FWHM / ( FWHM**2 + 4*(x-POSITION)**2 )
```

and Gaussian part as:

```
G = 2*sqrt(ln(2)/pi)/FWHM * exp ( -4ln(2) * (x-POSITION)**2/FWHM**2 )
```

The position of the second peak is automatically calculated from the internally stored wave lengths for Kalpha1 and Kalpha2

The asymmetry function is defined as: For a 2Theta axis:

```
ASYMMETRY = 1 + (p[7]*FA(z) + p[8]*FB(z))/tanh(0.5*POSITION)
```

For a Q-axis

```
ASYMMETRY = 1 + (p[7]*FA(z) + p[8]*FB(z)) / tanh(asin(POSITION/4PI/lambda)
FA(z) = 2*z*exp(-z**2)
FB(z) = 2*(2*z**2-3)*FA(z)
z = (x-POSITION)/FWHM
```

The background is calculated as a polynomial of order (n\_backgrd-1):

```
p[1] + p[2]*(x-origin) + p[3]*(x-origin)**2 + ...
or more exactly:
SUM p[i]*(x-origin)**(i-1)
```

where origin is the optional "origin:" parameter from the command line. If this parameter is omitted, the origin is set to zero. To describe a locally bent background, one should set the origin of the background to a position within the refined x-range instead of zero. The value of the origin is not refined.

The optional parameter "nback:" defines the number of background parameters to be used. If it is omitted, the number of background parameters is set to 2.

#### fx

```
func "fx",<ip>,<function>
```

KUPLOT allows the user to fit a user defined function to a given 1D as well as 2D data set. The first parameter after "fx" specifies the number of parameters. Refer to the parameters in the expression for the function as p[1],p[2],.... The x-variable is r[0] and the y-variable (for 2D data sets) is r[1]. Note, that possible values stored in these variables are lost. It sounds complicated, but the following examples should help:

```
func fx,3,p[1]+p[2]*exp(p[3]*r[0]) : Fit a+b*exp(c*x)
func fx,2,p[1]*r[0]+p[2]*r[1]      : Fit a*x + b*y (2D data set)
```

For a syntax check, the expression is calculated once after entering this command. If the parameter values p[i] are not set to proper start values, a error might occur, i.e. division by zero.

#### ma

```
func "ma",<ip>,<macro_name>
```

KUPLOT allows the user to fit a user defined function inside a macro to a given 1D as well as 2D data set. The first parameter after "ma" specifies the number of parameters. Refer to the parameters in the expression for the function as p[1],p[2],.... The x-variable is r[0] and the y-variable (for 2D data sets) is r[1]. Note, that possible values stored in these variables are lost. The macro can use expressions, loops and if constructions, nested macros are allowed as well. For a full use of the discuss\_suite, use a combination of a 'top' and 'quit' command. With the 'top' command the macro steps to the top level at the discuss\_suite. From there on all commands can be used.

The final function value must be given on a 'value' command, the derivatives on a 'deriv' command. For these command you must have left the 'top' section via a 'quit'.

The macro must specify the function value into the special global variable 'F\_VALUE', and the derivatives into the entries in 'F\_DERIV[<i>]', where <i> takes the values 1 to the number of refined parameters.

As simple macro would be: F\_VALUE = p[1]\*r[0] ! Fit a\*x F\_DERIV[1]= r[0] ! Derivative with respect to p[1] finished

or absolutely equivalently: r[100] = p[1]\*r[0] F\_VALUE = r[100] ! Fit a\*x F\_DERIV[1] = r[0] ! Derivative with respect to p[1] finished

More involved, create a structure, calculate bond length as the function value:

r[101] = p[1] ! Copy KUPLOT variables to global r[102] = p[2] top ! Go to top level menu discuss read free insert C, r[101], 0,0, 0.1 insert C, r[102], 0,0, 0.1 r[200] = x[1]-x[2] ! The function value r[201] = 1.0 ! The derivative with respect to p[1] r[202] = -1.0 ! The derivative with respect to p[2] exit quit ! Quit the top level menu back to fit F\_VALUE = r[200] F\_DERIV[1] = 1, r[201] F\_DERIV[1] = 2, r[202] finished

It sounds complicated, but the following examples should help:

```
func ma, 3, one.mac : Fit a+b*exp(c*x)
```

Inside macro "one.mac" use the lines: F\_VALUE = p[1]+p[2]\*exp(p[3]\*r[0]) ! Fit a+b\*exp(c\*x) F\_DERIV[1] = 0.0 ! Derivative with respect to p[1] F\_DERIV[1] = exp(p[3]\*r[0]) ! Derivative with respect to p[2] F\_DERIV[1] = p[2]\*exp(p[3]\*r[0])\*r[0]! Derivative with respect to p[3] finished

The last command 'finished' is required inside these macros to return to the fit routine.

```
func ma, 2, two.mac : Fit a*x + b*y (2D data set)
```

Inside macro "two.mac" use the lines: F\_VALUE = p[1]\*r[0]+p[2]\*r[1] ! Fit a\*x + b\*y F\_DERIV[1] = r[0] ! Derivative with respect to p[1] F\_DERIV[1] = r[1] ! Derivative with respect to p[2] finished

```
func ma, 2
```

For a syntax check, the expression is calculated once after entering this command. If the parameter values p[i] are not set to proper start values, a error might occur, i.e. division by zero.

**gsas**

```
func "gsas", <ptype>, <bank>, <iparm-file>
```

KUPLOT allows the refinement of a GSAS profile function for TOF data. For details about the parameters and definitions of these functions, please refer to the GSAS manual. The parameters are <ptype> to select the profile function type, <bank> to select the bank number and <iparm-file> specifies the GSAS style instrument parameter file. The starting values are read from this file. Please also note, that the x-values are expected to be TOF.

**back**

```
func "back",<icalc> [,<par>]
```

The fitting of a background is calculated as:

```
y(dataset) = scale*y(dataset icalc) + SUM xxxxparameter
```

Thus KUPLOT takes the data set <icalc> as input values, which are scaled and to which a polynomial function is added. This sum is fitted to the current dataset which was selected by the 'fit' command.

The scale parameter is p[1], all higher parameters act as exponents to the background polynomial. Thus p[2] is a constant background, p[3] a linear background etc. The xx argument of the polynomial is calculated as (x - xmin), where x is the x-value of each given data point and xmin the minimum x-value. Thus, the values of the background polynomial change if you fit to a changed x-range!

```
<par> effect
1 scale factor only
2 scale factor + constant background
3 scale factor + constant + linear background
N scale factor + background polynomial of order (N-2)
```

**help**

```
help <command>
```

Displays this help text, for help for a certain command type help command. See help entry on main level for more details.

**macro**

```
macro <filename>
```

This command saves the current parameters in a KUPLOT macro file. This allows to set the parameter values in a later session using @<filename> from the FIT sub level.

**mfen**

```
mfen <if>
```

This command sets the 'window size' for maxima determination used to compute starting values for functions GAUS and LORE. For details about the <if> value see command -> smax.

**output**

```
output { "on" | "off" }
```

This command allows to specify if the progress of the fit should be displayed on the screen ("on") or not ("off").

**para**

```
para
para <ip>,<pinc> [,<value>] [,range:[lower,upper]]
para { "save" | "load" }
```

This command allows to modify the fit parameters. If the command is called alone, the current parameters are displayed. A single parameter *<ip>* can be set to a value *<value>*. The value *<pinc>* specifies if the parameter should be refined (*<pinc>* = 1) or should be kept fixed (*<pinc>* = 0). If *<value>* is omitted the current value will be used.

A set of parameters can be internally stored ('para save') and loaded later back in the parameter array ('para load'). Each save command will overwrite previously stored values.

The optional "range:[,]" parameter allows to restrict the range of a parameter to the interval specified by the *<lower>* and *<upper>* limits. Allowed formats are: range:[<lower>,<upper>] ! Expressions for lower and upper limit range:[,<upper>] ! Expressions for upper limit only, lower limit is set to -infinity range:[<lower>] ! Expressions for lower limit only, upper limit is set to +infinity

**plot**

```
plot
```

This command displays the current fit result with the current plot settings (like the plot command in the main level).

**range**

```
range {"all" | "plot"}
```

This command allows one to select which part of the data are used for the refinement. If set to "all", all data are used. If set to "plot" only the data shown on the current plot window are used. This is useful if one wants to refine e.g. just a single Gaussian in a data set without having to manipulate the data set itself.

**relax**

```
set relax ["start:<lamda_s>"] [, "fail:<lamda_f>"]
[, "success:<lamda_g>"]
```

Allows to fine tune the relaxation behavior. The initial Levenberg-Marquardt lamda parameter is defined with ["start:<lamda\_s>]. Default is 0.001

If a cycle was successful, the value of lamda is multiplied by <lamda\_g>, default is 0.5.

If a cycle was not successful, the value of lamda is multiplied by <lamda\_f>, default is 4.0.

A larger value of lambda implies smaller steps for the refinement parameters.

**run**

```
run
```

This command starts the fit.

**save****save**

This command saves the current fit results. The output file name is the file name of the input data set with the following extensions:

```
.erg : text file with fit results
.fit : calculated data set
.dif : difference between observed and calculated data set
```

**show****show** [ { "general" | "fit" | "para" } ]

This command shows fit settings and results. If the command is called without parameters, all information is printed on the screen. The command 'show general' will display only general fit settings, 'show fit' the fit results like R values and 'show para' will list the resulting parameters.

**urf****urf** <u>

This command has become obsolete, please use the ==> 'relax' command instead. The command urf will set the initial value of lamda for the Levenberg-Marquadt refinement a reasonable value is about 0.001

The value is automatically adjusted to allow a smooth transition into the minimum.

**wic****wic** <w> [, <K>]

This command defines the weighting scheme to be used for the fit. The valid values for <w> are:

"one" : w(y) = 1.0	"sqa" : w(y) = y**2
"sqrt" : w(y) = SQRT(y)	"inv" : w(y) = 1/y
"log" : w(y) = LOG(y)	"isq" : w(y) = 1/SQRT(y)
"lin" : w(y) = y	"dat" : w(y) = 1/DY**2 (error data set)
"bck" : w(y) = exp(-K*(y-ycalc))	

The second parameter is only needed for the weighting scheme "bck".

**A.27 fname****fname** [ { "on" | "off" } ]

This command determines weather the file name should be plotted for every data set in the upper right corner. Called without parameters, file name plotting is switched off.

## A.28 font

```
font <prop>,<value>,...
```

This command allows to set font attributes for the active frame. The settings can be displayed by entering the command 'font' without parameters or using the command 'show font'. For a list of escape sequences and special characters see entry PGPLOT below. The available font properties <prop> are listed below:

### PGPLOT

#### **Special characters**

The use of the PGPLOT library allows KUPLOT users to use a variety of escape sequences and special characters in all text related functions of KUPLOT.

```
\u : start superscript, or end subscript
\d : start sub, or end superscript (e.g. cm\u-2\d)
\b : backspace (draw next character on top of current one)
\fn : switch to normal font (1)
\fr : switch to roman font (2)
\fi : switch to italic font (3)
\fs : switch to script font (4)

\\ : backslash           \x : Multiplication sign
\. : centered dot        \A : Angstrom

\gx : Greek character 'x' (e.g. \ga gives alpha, \gb beta, ...)
\$(nnnn) : character (nnnn) from the Hershey character set
```

### color

```
font "color",<id>,<icol>
```

This command allows to alter the color of the font used to label the graph. The default is black for all fonts. The different types of text given by their <id> parameter are listed below:

1 : main title line	2: subtitle line
3 : axis labels	4: numbers at axis
5 : text in text frame	6: file name & caption

The allowed default color values <icol> are listed below. Note that they can be changed by the user via the command 'color'. Alternatively you can use the color name (without blanks i.e. darkblue etc.)

1 : red	5 : yellow	9 : dark blue	13 : cyan
2 : green	6 : black	10 : dark magenta	14 : dark cyan
3 : blue	7 : dark red	11 : dark yellow	15 : white (!)
4 : magenta	8 : dark green	12 : gray	

### just

```
font "just", { "left" | "center" }
```

This command sets the text justification for the two title lines and the text within a text frame.

**size**

```
font "size",<id>,<size>
font "size",<scale>
```

This command sets the font size for text type <id> to <size> points. If a specified font size is not available, a fixed font will be used instead. The allowed <id> parameters are:

<pre>1 : main title line 3 : axis labels 5 : text in text frame</pre>	<pre>2: subtitle line 4: numbers at axis 6: file name &amp; caption</pre>
---	---

Alternatively if only one parameter is given after "size", all fonts will be scaled by the given factor <scale>, i.e. a value of 0.5 will set all font sizes to half their specified original size.

**typ**

```
font "typ",<id>,<typ>
```

This command sets the font of the text type <id> to <typ>. Allowed values for the text typ parameter <id> are:

<pre>1 : main title line 3 : axis labels 5 : text in text frame</pre>	<pre>2: subtitle line 4: numbers at axis 6: file name &amp; caption</pre>
---	---

Currently implemented fonts are:

```
1 : Normal single stroke font (default)
2 : Roman font
3 : Italics font
4 : Script font
```

**A.29 fset**

```
fset <typ>
```

This commands allows to alter the frame and axis plotted around the view graph for the active frame. The allowed values for <typ> are:

```
0 : no frame, axis and labels
1 : only a box around the view graph
2 : setting 1 & tick marks, numbers and labels
3 : setting 2 & axis at x,y=0.0
```

If you specify a negative number for <typ> rather than the positive number listed above, the y-axis label and numbers will be plotted on the right hand side of the view graph rather than the left hand side. This allows to plot data sets with two different y-axis in one plot by plotting two frames (-> frames) on top of each other.

Note, that the numbers and label at the axis can be switched off using the command 'achx/y' (-> achx, achy). A dotted line parallel to x and y at the major tick marks can be plotted using the command 'grid' (-> grid). The variable axis[i,j] allows one to change various details of the axes and frame (-> variables). Note that the values are given in character height units.

## A.30 fft

```
fft real:<ir>, imag:<ii>
```

This command calculates a Fast Fourier. The real part of the data reside in data set number <ik>, the imaginary in data set number <ii>. At least one part (real or imaginary) has to be given. The results are placed into the next two available data sets.

The data can be 1D, 2D or 3D. Both real and imaginary data have to be of the same dimension.

## A.31 fourier

```
fourier <ik> [,<Qmin>, <Qmax>, <deltaQ>] [,<code>]
```

This function allows the user to calculate the discrete Fourier Transform of data set <ik>. The range of the Fourier transform is given by <Qmin> and <Qmax> and the step size is <deltaQ>. If the parameters are omitted, the range in Fourier space is fixed from -1/2D .. 1/2D with D being the grid size in real space. Note, that the range can only be explicitly specified for 1D data sets. The optional parameter <code> determines how much output will be created. The parameter <code> is made up of any combination of the following four letters:

```
r : save real (cos) part of Fourier Transform
i : save imaginary (sin) part of Fourier Transform
a : save amplitude of Fourier Transform
p : save phase of Fourier Transform
```

So <code>'riap' would save all 4 parts in separate new data sets. The default is to store the real part. If the parameter <code> contains the letter 'H', the factor 2pi is used in the exponential.

## A.32 frames

**Using frames in KUPLOT**

The program KUPLOT allows to divide the plotting space into smaller areas called FRAMES. Each frame can be individually positioned on the plotting space. The user can define which loaded dataset should be displayed in which frame, alternatively, the contents of a text file can be printed in a frame. Each frame has its own set of parameters like title, axis labels, tick marks, ... . All commands related to frames are listed below:

### commands

```
List of commands:
aframe : sets the active frame for user input
bframe : sets background color for specified frame
cframe : copy frame parameters
frame : defines if a border is plotted around each frame
kframe : define contents of frames (data sets or text)
nframe : set number of frames (default = 1)
sframe : define position and size of frame
```

**aframe**

```
aframe <if>
```

The active frame is set to *<if>*. All parameters like title, axis labels, plotting window etc. are entered for the active frame. If 'afra' entered without parameters, the actual active frame is displayed. The setting of one frame can be copied to another frame with the 'cfra' command.

**bframe**

```
bframe <if>,<r>,<g>,<b>
```

This command sets the background color for frame *<if>*. The color is given as RGB (red,green,blue) value ranging from 0.0 to 1.0. The default is a white background (1.0 1.0 1.0).

If the background color is changed frequently, the X-window server might be unable to allocate additional colors if bitmaps are used. To free the used colors reenter the command 'cmap' which will create a new color map.

**cframe**

```
cframe <ifr>,<i1> [,<i2>,<i3>,...]
```

This command copies the settings of frame *<ifr>* (like title, contour line settings, plot window, ...) to frames *<i1>*, *<i2>*, ...

**frame**

```
frame [ {"on" | "off"} ]
```

This command determines whether a black border is plotted around the graph / each individual frame. Called without parameters, no border will be plotted.

**kframe**

```
kframe <if>,<ik1> [,<ik2>,...]
kframe <if>,<fname>
kframe <if>,{ "para" | "fit"}
```

This command sets the contents for frame *<if>*. A frame can either contain the graphs of dataset(s) *<ik1>*.. of the contents of a text file, specified by *<fname>*. Special text frames are "para" which prints data information and "fit" giving the fit results.

**nframe**

```
nframe <n>
```

This command sets the total number of frames to be used. For most values of *<n>* KUPLOT defines a default layout. If no default layout is found, a warning is given. The layout can be changes using the command 'sfra'.

**sframe**

```
sframe <if>, <xmin>, <ymin>, <xmax>, <ymax>
```

This command allows to define the layout of the frames on the plot space. The lower left corner of the plot space is 0,0, the upper right is 1,1. The lower left corner  $\langle \text{xmin} \rangle, \langle \text{ymin} \rangle$  and the upper right corner  $\langle \text{xmax} \rangle, \langle \text{ymax} \rangle$  for each frame  $\langle \text{if} \rangle$  can be specified with the 'sfra' command.

**A.33 function**

```
function <f(r[0])>,      <ioref>
function <f(r[0])>,      <xmin>, <xmax>, <dx>
function <f(r[0],r[1])>, <xmin>, <xmax>, <dx>, <ymin>, <ymax>, <dy>
function "fit", <ioref>
function "fit", <xmin>, <xmax>, <dx>
function "fit", <xmin>, <xmax>, <dx>, <ymin>, <ymax>, <dy>
```

This command allows to create a new data set from the function and range given as command parameters. Every call of 'func' will create a new data set similar to 'load'. Use 'rese' to discard previous created data sets.

The first parameter is the function to be calculated. See -> expressions for details on the function syntax. Use variable r[0] as x-variable and r[1] as y-variable in case of 3D data sets. Note that stored values in r[0] and r[1] will be destroyed ! The parameters  $\langle \text{xmin} \rangle$  and  $\langle \text{xmax} \rangle$  give the x-range for the function. The value of  $\langle \text{dx} \rangle$  is the step size in x-direction. For 3D data sets the y-range is given as additional parameters in a similar way. Alternatively the x-values can be taken from the dataset  $\langle \text{ioref} \rangle$  (implemented for 3D files as well as of Version 5.33).

If the first parameter is "fit", the value of the current fit function (see -> fit) is used.

**Examples**

```
func sin(r[0]),0.00,6.30,0.10      : sin(x) from 0->6.3, 64 pts
func sin(r[0])*cos(r[1]),0,6,0.1,0,6,0.1 : sin(x)*cos(y) 0->6 61x61 pts.
```

**A.34 glat**

```
glat <ik>, <ip> [, {"x" | "y"}]
```

This command smoothes the data set number  $\langle \text{ik} \rangle$  over  $\langle \text{ip} \rangle$  points. The number  $\langle \text{ip} \rangle$  should be a positive odd number. A normal sliding averaging without weighting is used (see also command -> smooth). Three dimensional files are per default smoothed in x and y direction. The user can restrict the smoothing to only one direction by adding the optional parameter "x" or "y".

**A.35 grid**

```
grid [ {"on" | "off"} ]
```

This command determines whether a dotted line parallel to the axis should be plotted at all major tick mark positions. Called without parameters, the grid plotting is switched off.

### A.36 hart

```
hart <ik>,<p>
```

This command controls the appearance of two dimensional plots. The data set is again specified by <ik>, the allowed values for <p> are:

```
1 : contour map
2 : bitmap
3 : contour map & bitmap
```

The z-range of the values to be converted to the color range of the bitmap is determined by the setting of the first contour line range (see command hlin). The color map is defined by the 'cmap' command.

### A.37 hcolor

```
hcolor <ik>,<ic>,<c>
```

This command sets the color for contour line set <ic> for the data set <ik>. The colors are numbered like the pens on a HP7475 plotter. Allowed values for <c> are listed in the table, Alternatively you can use the color name (without blanks i.e. darkblue etc.)

The allowed default color values <icol> are listed below. Note that they can be changed by the user via the command 'color'.

1 : red	5 : yellow	9 : dark blue	13 : cyan
2 : green	6 : black	10 : dark magenta	14 : dark cyan
3 : blue	7 : dark red	11 : dark yellow	15 : white (!)
4 : magenta	8 : dark green	12 : gray	

### A.38 hlab

```
hlab <ik>,<int>
```

This command controls optional labeling of the contour lines of data set <ik>. If the parameter <int> is set to zero (default), no labels are drawn. Otherwise every <int> contour level starting at the base level is labeled. The font characteristics of the axes labels are used, only the font size is reduced by 75%.

### A.39 hline

```
hline <ic>,<min>,<inc>,<n> [, { "abs" | "%" | "%m" }]
hline <ic>,      <inc>,<n> [, { "abs" | "%" }]
```

This command sets the contour line parameters. The first parameter <ic> specifies the contour line set, which is defined by the following numbers. The contour lines starting level is given by <min>, the increment by <inc> and the number of contours to be plotted by <n>. As default the given values are taken as absolute values. However, is a fifth parameter set to "%" the values are taken as percentages.

the values of <min> and <inc> are taken relative to the maximum/minimum value of loaded data set 1. If the last parameter is "%m" the percentage refers to zmax only.

If the second form is chosen, the contour levels are placed at equal intervals around the zero level within the data. If the data are either only positive or only negative, the contours start at the <inc> level and proceed to the <n>th level.

Is the command given without parameters, the current settings of the contour parameters are displayed on the screen.

## A.40 hpackage

**hpackage <ip>**

KUPLOT allows to define more than one set of contour lines see 'hlin' command). The command 'hpackage' allows to set the number of contour line sets <ip> to be used.

## A.41 hpak

**hpak <ip>**

KUPLOT allows to define more than one set of contour lines (see 'hlin' command). The command 'hpak' allows to set the number of contour line sets <ip> to be used.

## A.42 htype

**htype <ik>,<ic>,<p>**

This command sets the line typ for contour line set <ic> for the data set <ik>. The colors are numbered like the pens on a HP7475 plotter. Allowed values for <p> are (number or name):

0 : {"noline" "none"} :	no line
1 : "solid" :	solid line
2 : "broken" :	---- ----- ---- -----
3 : "dashed" :	- ----- - ----- - -----
4 : "shortbroken" :	- - - - - - - - - - - - - -
5 : "shortdashed" :	- - - - - - - - - - - - - -

## A.43 ident

**ident [ {"on" | "off"} ]**

This command determines whether user name, date and time of the plot are written on the bottom of the view graph.

## A.44 integral

```
integral <ik> [,<xmin>,<xmax>]
integral <ik> [,<xmin>,<xmax>,<ymin>,<ymax>]
```

This command calculates the integral of data set *<ik>*. If the additional parameters are omitted, the complete actual plotting window is integrated. Alternatively, the integration limits *<xmin>,<xmax>* (2d) or *<xmin>,<xmax>,<ymin>,<ymax>* (3d) can be given after the data set number *<ik>*.

The results of this command are stored in the *res[i]* variables in the following order:

```
res[0]   : Number of parameters given
res[1]   : Integral
res[2]   : Sigma of the integral
res[3]   : # of points integrated
res[4]   : Integration limit xmin
res[5]   : Integration limit xmax
res[6]   : Integration limit ymin (only for 3d data)
res[7]   : Integration limit ymax (only for 3d data)
```

## A.45 ksave

```
ksave <ik>
```

This command enters the 'ksav' level of KUPLOT which allows to save the given data set *<ik>* in different formats to a file. The allowed commands in the 'ksav' level are:

### commands

Here is a list of valid commands in the KSAV sub level:

```
exit      : Leaves KSAV sub level without saving the file
form     : Sets output file format
help     : Displays on-line help (see general help)
m999    : Treatment of excluded data points
outfile  : Specifies the output file name
run      : Saves file and leaves KSAV level
show     : Shows the current settings
thresh   : Sets threshold for PGM file output
wait     : wait for user input (see main help)
```

### form

```
form <ff> [,<p1>,<p2>,...]
```

This command determines the file format to be used when the file is saved. Some formats require additional parameters. If a parameter is not required, the default is the current plotting window. The following list shows the valid file formats:

<ff>	<p1>,....	Required	What
xy	xmin,xmax, [<style>]	no	Saves xy file from xmin,xmax
ni	xmin,xmax,ymin,ymax	no	Nipl file, given area
pg	xmin,xmax,ymin,ymax	no	PGM file (ASCII), given area
gn	xmin,xmax,ymin,ymax	no	XYZ file (gnuplot), given area
sx	y-value	yes	Cross section    x at y-value
sy	x-value	yes	Cross section    y at x-value
mx	i	yes	Cross section    x trough max #i
my	i	yes	Cross section    y trough max #i
sk	ik	yes	Cross section along xy of data set ik
sl	x1,y1,x2,y2,n	yes	Cross section from x1,y1 to x2,y2 with n points

For the xy format, the last optional parameter can be the string "four" to explicitly write the default four columns x, y, dx, dy. Alternatively the last parameter can be "two" to write a two column file. If the uncertainties for x and / or y are not zero, the values are lost! For either option, the parameters xmin, xmax are still optional, and the command might take the forms:

```
xy ,xmin,xmax, "four"
xy ,xmin,xmax, "two"
xy ,xmin,xmax
xy , "four"
xy , "two"
```

## m999

```
m999 "excl" | "data"
```

KUPLOT treats data points of value "-9999" as excluded data in a two dimensional data array. This command specifies the treatment of these data points while writing cross sections || x or y. With "excl" these data points are omitted from the output file, which may therefore have a length that varies from the corresponding dimension of the two-dimensional data. With "data" these data points are treated as regular data points and written to the output file. The output file is of constant length corresponding to the respective two dimensional dimension. The output file contains the data points with value -9999.

## outfile

```
outfile <fname>
```

This command specifies the name of the output file. See entry 'filenames' for more information about the building of file names.

## run

```
run
```

This command save the file with the current settings and leaves the KSAV level.

## show

```
show
```

This command shows the current settings.

## threshold

```
thresh { "high" | "low" | "sigma" | "zmin" | "zmax" } <value>
```

This command sets the threshold that is used when writing BITMAP output. All values less than the minimum threshold are set to zero, all values higher than the maximum threshold are set to the maximum threshold. The values in between are linearly scaled from zero to 255. Depending on the first parameter, the second parameter is interpreted in five different ways:

```
"high"   : Sets maximum threshold for BITMAP in percent of the maximum z
"low"    : Sets minimum threshold for BITMAP in percent of the maximum z
"sigma"  : Sets threshold to average z +- <value>*standard deviation z
"zmax"   : Sets maximum threshold for BITMAP
"zmin"   : Sets minimum threshold for BITMAP
```

## A.46 kcalc

```
kcalc {"add" | "sub" | "mul" | "div"}, <ik1>, <ik2>[, {"excl" | "neut" | "igno"}][, "over"]
```

This command allows to multiply ("mul"), add ("add"), divide ("div") or subtract ("sub") the y- or z- values of the data sets  $\langle ik1 \rangle$  and  $\langle ik2 \rangle$ . For a 2d data sets, the y- values are used, for a 3d data set the z-values. The optional last parameter triggers where the resulting data set is stored. The default is to create a new data set. If the parameter "over" is specified, the resulting data set will overwrite data set  $\langle ik1 \rangle$ .

The optional fourth parameter is effective only for 3d data sets. It tells KUPLOT how to treat the special z values of "-9999." The following parameters are valid:

```
"excl"  : These data points are treated as excluded regions and the
          result will be -9999 if one or both z-values of the data
          sets had been -9999. This is also the default used when no
          fourth parameter is given.
"neut"  : The data points are treated as neutral element for the
          operation. If both input values are -9999, the result is -9999,
          if only one input value is -9999, the result is the z-value
          of the other curve.
"igno"  : The special properties of these data points are ignored and
          the result is the corresponding operation between the two
          numbers.
```

## A.47 kpara

```
kpara {<par_name> | <par_number>}
kpara { {"last" | "-1"} | <generation_no> } ,
      {<par_name> | <par_number>} , {<par_name> | <par_number>}
```

This command serves to create a standardized plot of the parameters refined via DIFFEV. The first form with a single parameter name or number will plot the development of this parameter versus the refinement generations. The parameter name may include "Rvalue" or 0 to plot the development of the R-value. The plot displays the upper and lower most parameter value in red, the average parameter in blue with error bars and the best parameter in black. Examples kpara Rvalue ! Show the R-value versus generations kpara P\_one ! Same for a parameter named "P\_one"

## A.48 kpar\_par

```
kpar_par { {"last" | "-1"} | <generation_no> } ,
{<par_name> | <par_number>} , {<par_name> | <par_number>}
```

This command serves to create a standardized plot of the parameters refined via DIFFEV. The command serves to plot the distribution of two parameters in a given generation. If the first command parameter is either "last" or "-1" the last generation is used. This can be helpful to repeatedly plot the parameters during a refinement. The plot will display the distribution of the two parameters. The size of the markers indicates the R-value for each parameter pair, with the smallest marker signifying the smallest R-value. Either of the two parameter names/numbers may be "Rvalue" or "0" to show the correlation between parameter value and the R-value. Examples kpar\_par last, P\_one, Rvalue ! Show R-values versus Parameter value ! in the last generation kpar\_par last, P\_one, P\_two ! Display (possible) correlation ! between these two parameters

## A.49 lart

```
lart <ik>, <p>
```

This command sets the who the points of data set <ik> are to be connected. Allowed values for <p> are:

```
1 : points are connected by a line
2 : histogram style plot
3 : connection via cubic splines
```

## A.50 lcolor

```
lcolor <ik>, <c>
```

This command sets the line color for the data set <ik>. If <ik> is 0 the color set is taken for the grid lines (-> grid "on" | "off"). The colors are numbered like the pens on a HP7475 plotter. Allowed values for <c> are listed in the table, Alternatively you can use the color name (without blanks i.e. darkblue etc.)

The allowed default color values <icol> are listed below. Note that they can be changed by the user via the command 'color'.

1 : red	5 : yellow	9 : dark blue	13 : cyan
2 : green	6 : black	10 : dark magenta	14 : dark cyan
3 : blue	7 : dark red	11 : dark yellow	15 : white (!)
4 : magenta	8 : dark green	12 : gray	

## A.51 load

```
load <form>,<fname>[,<par>,<par>,...]
```

Reads a data set from file <fname>. The format of the input file is given by the value of <form>. Some formats require additional parameters <par>. If a history (PDFgetN) is prepended, KUPLOT extracts the title and stores some information in the res[n] array (see PDFFIT 'read data' for details). The parameter <fname> can also be a valid expression such as ".asc" just as a directory command.

## 4d

```
load 4d,<file>,<ix>,<iy>,<iz>,<ival>
```

This command allows to read a section from a NIPL type 4D files (x y z value). The first two parameters after the file name define which index is mapped to x and y in KUPLOT. The next parameter defined which index is used for cutting and <ival> specified the value at which the cut is performed.

## cr

```
load cr,<file>
```

Reads a "crystal" file format which contains coordinates for the atoms and typ, color and size for the marker for the specific atom. Those code numbers for typ, color and size are the same used with the 'mtyp', 'mcol' and 'msiz' commands. The points are plotted in the xy plane, i.e. projected along z.

```
0.5 0.5 0.5 5 3 1.00
1.0 1.0 1.0 5 6 0.75
```

The first atom at (0.5 0.5 0.5) is represented by a red (col=3) filled square (typ =5) of the size 1.00 whereas the second atom is a black (col=6) filled smaller square. The size is given relative to the size set with -> 'msiz'. The first atom is plotted with the full size set with 'msiz' whereas the second atom is only 75% of that size.

## csv

```
load csv,<file> load csv,<file>, skip:<nskip>, colx:<icolx>, coly:<icoly>, coldx:<icoldx>, coldy:<icoldy>, separator:<name>, decimal:<name>
```

Reads a csv type file that has been extracted from PANalytical powder data.

The second form is a generic attempt to read any CSV file. You can specify as optional parameters: skip: Number of header lines to skip Default : 25 colx: Column from which to take x-data Default : 1 coly: Column from which to take y-data Default : 2 coldx: Column from which to take uncertainties for x-data Default : 0 coldy: Column from which to take uncertainties for y-data Default : 0 separator: Character that separates the columns, possible values are: ";" or "semicolon" Default : "semicolon" ":" or "colon" "comma" "tab" "space" Will take as separator a string of blank(s) "[4,4,4,8,8]" An explicit list of columns widths The number of elements in [...]

must be at least as large as the actual columns in the file decimal: Character that separates the fractional part of numeric data Defaults to a "period", as used in most English speaking areas "comma" reads a numerical field "12,34" as 12.34 If the error columns are omitted or set to zero no error data are read

### de

```
load de,<file>,<gridx>,<gridy>
```

This command works similar to the 'load zz,..' command but rather than reading a xyz-file, a histogram of a xy-file is calculated, i.e. z represents the number of points x,y which fall in a grid of the size <gridx> and <gridy>.

### gs

```
load gs,<file>,<ibank> [,<unit>] [,<ifile>] [, "norm"]
```

This command allows to read GSAS files. AT THIS TIME THIS FEATURE IS ONLY TESTED FOR TIME-OF-FLIGHT POWDER DIFFRACTION DATA. The value <ibank> determines which bank to read. A range of banks can be specified like from>to and to read all banks use the keyword "all". The optional parameter <ifile> is the file name of the corresponding GSAS instrument parameter file, which is ONLY needed if certain unit conversion are desired. The desired unit is specified with the optional parameter <unit>. Valid units are:

"T"	: Time-of-Flight
"D"	: D spacing
"L"	: Wavelength (lambda)
"Q"	: Momentum transfer Q
"X"	: Used to indicate X-ray data in 2THETA

Finally if the last optional parameter is set to "norm", the intensities will be normalized by the incident spectrum. A valid instrument parameter file is required in this case.

### h5

```
load h5,<file> [, layer:<number>]
load h5,<file> [, layer:{ "bottom" | "middle" | "top" }]
load h5,<file> [, {"trans:yes" | "trans:no" }]
```

This command reads an HDF5 file format as written by the DISCUS section or the Yell program.

By default, the middle layer is presented, optionally you can specify a layer number or the "bottom" or "top" layer.

All layers in the HDF5 file are read, and you can use the ==> 'mouse' menu to step through the layers.

The optional parameter "trans" give you the choice to step into a transformation menu that allows to transform the 3D-data into a new orientation. The commands at this menu level are:

## Examples

### Examples

All examples assume a 3D volume parallel  $a^*$ ;  $b^*$ ;  $c^*$ . In all examples the new data set is transformed to an orientation with the z-axis parallel to [1,1,1] and the abscissa parallel to [1, -1, 0].

Sensible combinations of the optional parameters are: 1) Define an abscissa and a zone axis 2) Define the abscissa, ordinate and top axis 3) define the corners in space as: ll = lower\_left = left\_lower\_bottom lr = lower\_right = right\_lower\_bottom ul = upper\_right = left\_upper\_bottom tl = top\_left = left\_lower\_top

1) Define an abscissa and a zone axis set abs:[0.04, -0.04, 0.00], zone:[1,1,1]

The new zone axis is set to [111], the abscissa is in steps of [0.04, -0.04, 0.00]. The ordinate will be calculated as vector product zone \* abs, steps along the abscissa are of equal length as along the abscissa. The number of data points is determined by transformation of the hkl limits in the original data set.

set abs:[0.04, -0.04, 0.00], zone:[1,1,1] set npoints:[251,251, 125]

Same as above. The number of data points is given by the user. Along the abscissa the maximum value will be [0.04, -0.04, 0.00]\*(125), where 125 = (251-1)/2. Same for the other three axes. Datza points in the new data set outside the original data receive values of zero.

2) Define the abscissa, ordinate and top axis set abs:[0.04, -0.04, 0.00] set ord:[0.04, 0.04,-0.08] set top:[0.04, 0.04, 0.04]

The new axes are all explicitly given as increment vectors. hkl limits and number of data points are derived from the orginal data set.

set abs:[0.04, -0.04, 0.00] set ord:[0.04, 0.04,-0.08] set top:[0.04, 0.04, 0.04] set npoints:[251,251, 125]

Same as above. The number of data points is given by the user.

3) define the corners in space as: set ll:[-15.0, -3.0, 9.0] set lr:[ -3.0, -15.0, 9.0] set ul:[ -3.0, 9.0, -15.0] set tl:[ -9.0, 3.0, 15.0] set npoints:[251,251, 25]

Four new corners are defined and the number of data points along the abscissa, ordinate and top axis are provided.

### set

```
set "abs: [<h>, <k>, <l>]""
set "ord: [<h>, <k>, <l>]""
set "zone: [<h>, <k>, <l>]""
set "ll: [<h>, <k>, <l>]""
set "lr: [<h>, <k>, <l>]""
set "ul: [<h>, <k>, <l>]""
set "tl: [<h>, <k>, <l>]""
set "npoints: [<h>, <k>, <l>]"
```

Specified details of the intended transformation

```
set "abs: [<h>, <k>, <l>]"
```

Sets the new abscissa

```
set "ord: [<h>, <k>, <l>]"
```

Sets the new ordinate

```
set "zone:[ "<h>, <k>, <l>" ]"
```

Sets the new zone axis. The zone axis will be the "top" coordinate of the transformed data.

```
set "ll:[ "<h>, <k>, <l>" ]"
set "lr:[ "<h>, <k>, <l>" ]"
set "ul:[ "<h>, <k>, <l>" ]"
set "tl:[ "<h>, <k>, <l>" ]"
```

The combination of these four settings define the new 3D volume. The new axes are the vectors from the lower left "ll" corner to: abscissa :: lower right "lr" ordinate :: upper left "ul" top axis :: top left "tl"

```
set "npoints:[ "<nx>, <ny>, <nz>" ]"
```

Set the number of data points in the new transformed grid.

**reset**

```
reset
```

Set all values back to program start.

**run**

Perform the actual transformation.

**exit**

```
exit
```

Go back to the output menu.

**ma**

```
load ma,<file>
```

This commands reads a "marker" file which is a normal xy-file, but the program will only use the x coordinate , y is set to zero. So tick marks at the x-axis can be drawn.

**mc**

```
load mc,<file>,<snumber>
```

This commands reads spectrum number <snumber> from MCA files (e.g. those created at A2/CHESS).

**ni**

```
load ni,<file>[,<white>]
```

This file format is also an old "Garching format" used by an even older plot program. It contains gridded 3d data in the following format:

```
line 1    : nx ny          ! Number of points in x and y
line 2    : xmin xmax ymin ymax   ! Range in x and y
line 3-xx : z z z z z z z z      ! z-values increasing in x and y
```

**pg**

```
load pg,<file>
```

Here is finally a universal 3d data file format known by KUPLOT, PGM files (from netpbm library). The current KUPLOT version can only read ASCII PGM files which have the code P2 in the first line of the program.

**sc/st**

```
load sc,<file> [<iscan> [<cx>,<cy>] [<cN>] [<cdx>] [<cdy>]
load st,<file> [<iscan> [<cx>,<cy>] [<cN>] [<cdx>] [<cdy>]
load sc,<file>,<iscan>,"info"
load smca,<file>,[<iscan> ,<iscan>, <i point> [, "kev",par1,par2,par3]]]
```

This command allows one to read SPEC scan files. The parameter *<file>* is as usual the file name. If no further parameters are given, KUPLOT lists the scans present in the file. The next parameter *<iscan>* is the number of the scan. Rather than an individual scan, one can specify a range of scans to be read via 'start > end' in the *<iscan>* field. Alternatively one can use the string 'all' for *<iscan>* to extract all scans from the given file.

If the subcommand is 'sc' or 'st', KUPLOT will read a scan, if the subcommand is 'smca', KUPLOT will read a MCA data point embedded in the spec file.

With no further parameters again KUPLOT lists information about the given scan and descriptions of the columns in that scan. The next two parameters *<cx>* and *<cy>* specify the columns to be associated with x and y. These parameters can either be numbers (positive numbers are counted from the left, negative from the right) or the EXACT column title. The last optional parameters defines which column KUPLOT will use for normalization of the y-data. If the subcommand 'sc' is used, KUPLOT calculates the standard deviation 'dy' as  $\text{SQRT}(y)$ . If 'st' is entered, no errors are calculated. Columns *<cdx>* and *<cdy>* correspond to the errors of *<cx>* and *<cy>* allowing the user to read sigma values from any column of the file. The fields have to be specified in the given order, for used field enter "0". The scan info that KUPLOT derives from the file is in part stored into the result variable ==> 'res[]'. The entries are: 0 number of values stored in res 1 number of scans 2 number of MCA points 3...2+NSCANS number of data points in each scan 2+1\*NSCANS+1 ... 2+2\*NSCANS xmin for each scan 2+2\*NSCANS+1 ... 2+3\*NSCANS xmax for each scan

KUPLOT can also extract scan information for a given scan *<iscan>* from the SPEC file when the parameter "info" is specified. Note that no data are read.

---

```
load smca,<file>,[<iscan> [,<iscan>, <i point> [,,"kev",par1,par2,par3]]]
```

KUPLOT will extract an MCA from a SPEC file. <iscan> is the scan number. If the SPEC files does not contain any scans but just individual MCA data points, set <iscan> to zero. <i point> is the number of the data point within scan <iscan>. If there are no scans in the file, KUPLOT will read the file until it finds the individual MCA data number <i point>. Without the "kev" key word, KUPLOT will use the channel number for the x-axis. With the keyword, the channel number is converted into keV according to  $\text{kev} = \text{par1} * \text{channel}^{**2} + \text{par2} * \text{channel} + \text{par3}$

Examples:

```
load sc,scan.dat,12           ! Shows info for scan 12
load sc,scan.dat,pmQ,Elastic,14 ! Reads Q vs. elastic counts norm.
                                by data in column 14 (monitor)
load sc,scan.dat,1>3,1,2      ! Read scans 1 to 3 column 1 vs. 2
load sc,scan.dat,1,info        ! Get information for scan 1
```

#### xx

```
load xx,<file>
```

This command reads just the first column of a data file and takes the point number as x-value.

#### xy

```
load xy,<file> [,<cx>,<cy>] [,<cdx>,<cdy>] [,<header>]
```

Reads data from a multi column ASCII file. Blank lines and lines starting with # are ignored. If no parameters after <file> are given, column 1 is assigned to x and column 2 to y. If three columns are present, column 3 is assigned to dy. In case 4 or more columns are present, column 3 is dx and column 4 is dy. Alternatively, the desired columns can be specified using the optional parameters <cx> and <cy> for the data columns x and y as well as parameters <cdx> and <cdy> for assigning the errors to columns x and y. The final optional parameter <header> allows one to specify the number of header lines to be skipped.

#### zz

```
load zz,<file>,<gridx>,<gridy> [,,"mod"]
load zz,<file>,<gridx>,<gridy> [,<xmin>,<xmax>,<ymin>,<ymax>]
```

This command reads a xyz-file, each xyz triplet on a single line. The data points are binned to a regular grid of the size <gridx> and <gridy>. The number of points in each direction is calculated by the program. This command can also be used to rebin data on a regular grid to a broader one. Without the optional parameter "mod" the grid starts at the lowest x,y values found and continues in steps of <gridx> and <gridy>. If the optional parameter "mod" is given, the grid is shifted to the next integer multiple of the interval <gridx>, <gridy>. Alternatively, the desired boundaries can be given as additional parameters. All points outside these limits will be ignored.

**mp**

```
load mp,<file>,<gridx>,<gridy> [, "mod"]
load mp,<file>,<gridx>,<gridy> [, <xmin>,<xmax>,<ymin>,<ymax>]
```

This command reads a binary xyz-file, each xyz triplet is given on a single 16 byte record. The first 4 bytes of the first record are an integer number that gives the total number of records in the file including the first line. Each other record contains the xyz data as 4 byte real numbers. In case of data that form a plane in three dimensions, the data are: abscissa, ordinate, 'density', third axis.

The data points are binned to a regular grid of the size *<gridx>* and *<gridy>*. The number of points in each direction is calculated by the program. This command can also be used to rebin data on a regular grid to a broader one. Without the optional parameter "mod" the grid starts at the lowest x,y values found and continues in steps of *<gridx>* and *<gridy>*. If the optional parameter "mod" is given, the grid is shifted to the next integer multiple of the interval *<gridx>*, *<gridy>*. Alternatively, the desired boundaries can be given as additional parameters. All points outside these limits will be ignored.

**A.52 sm**

```
load sm,<file>
```

Loads the SIMREF \*.plt file.

**special**

```
load <form>,<file>
```

KUPLOT can read certain special file formats produced by the program MAN2 and MAN1 used at the Institut fuer Kristallographie of the University Munich. Those formats are summed up in the following list:

T1/T2	MAN2-Pfausdatei	2-Theta gegen Zaehlrohr 1/2
C1/C2	MAN2-Pfausdatei	Chi gegen Zaehlrohr 1/2
P1/P2	MAN2-Pfausdatei	Phi gegen Zaehlrohr 1/2
O1/O2	MAN2-Pfausdatei	Omega gegen Zaehlrohr 1/2
H1/H2	MAN2-Pfausdatei	H gegen Zaehlrohr 1/2
K1/K2	MAN2-Pfausdatei	K gegen Zaehlrohr 1/2
K1/K2	MAN2-Pfausdatei	K gegen Zaehlrohr 1/2
L1/L2	MAN2-Pfausdatei	L gegen Zaehlrohr 1/2
TE	MAN2-Pfausdatei	Messpunktnumer gegen Temperatur (822)
DM	MAN2-Pfausdatei	Messpunktnumer gegen DMM-Wert
ZE	MAN2-Pfausdatei	Messpunktnumer gegen Messzeit
Z1/Z2	Messdatei von PULx	Zaehlrohr 1 bzw. 2

Two other special formats are the output files of the FAST data collection software of the area detector used in Munich. The two formats are:

DA	MP-Datei (binaer)	Flaechenzaehlerdatei
AS	MP-Datei (ascii)	Flaechenzaehlerdatei

## A.53 ltype

```
ltype <ik>,<p>
```

This command sets the line typ to be used for data set *<ik>*. The allowed values for *<p>* are (number or name):

```
0 : {"noline"} | {"none"} : no line
1 : {"solid"} : solid line
2 : {"broken"} : ---- ---- ---- ----
3 : {"dashed"} : - ---- - ---- - ---- -
4 : {"shortbroken"} : - - - - - - - - - -
5 : {"shortdashed"} : - - - - - - - - - -
```

If the line type is negative, the data set will be treated as a polygon. You can use the 'fill' command to fill the inside of this polygon.

## A.54 lwidth

```
lwidth <ik>,<width>
```

This command sets the line width *<width>* in cm for the data set *<ik>*. If *<ik>* is zero, the line width for the frame itself is set. The current settings can be displayed using the 'show style' command.

## A.55 mark

```
mark [<inx> [,<iny>]
```

This command sets the interval and starting point for the tick marks at the x- and y-axis. If all parameters are omitted, KUPLOT will do an automatic setting. The two parameters *<inx>* and *<iny>* give the interval of the major tick marks on the x- and y-axis. If only *<inx>* is given, *<iny>* remains unchanged.

## A.56 match

```
match {"scal" | "offset" | "all"},<i0>,<ic>
match "mix",<i0>,<i1>,<i2>
```

This command calculates the scale factor and/or offset which gives the best match between of data set *<ic>* to *<i0>*. The first command determines whether only a scale ("scal") or offset ("offset") or both ("all") will be calculated. Data set *<ic>* is then modified using these values. NOTE: This command assumes that a given data point i in both data sets *<i0>* and *<ic>* corresponds to the same x-value, i.e. the data sets must be equal with respect to their x-values ! Equivalently, xy-z data are assumed to be on identical grid points in (x,y).

If the first parameter is set to "mix" then data set *<i0>* is matched by  $f \cdot <i1> + (1-f) \cdot <i2>$ . The result is added as new data set. All data sets *<i0>*, *<i1>* and *<i2>* need to have identical x data points.

The scale factor and the offset are written into the result variable as entries 1 and 2. *res[0]* is set to 2

## A.57 mcolor

```
mcolor <ik>,<c>
```

This command sets the marker color for the data set  $<ik>$ . The colors are numbered like the pens on a HP7475 plotter. Allowed values for  $<c>$  are: listed in the table, Alternatively you can use the color name (without blanks i.e. darkblue etc.)

The allowed default color values  $<icol>$  are listed below. Note that they can be changed by the user via the command 'color'.

1 : red	5 : yellow	9 : dark blue	13 : cyan
2 : green	6 : black	10 : dark magenta	14 : dark cyan
3 : blue	7 : dark red	11 : dark yellow	15 : white (!)
4 : magenta	8 : dark green	12 : gray	

## A.58 mean

```
mean <ik> [,<xmin>,<xmax>]
mean <ik> [,<xmin>,<xmax>,<ymin>,<ymax>]
```

This command allows to calculate the average and standard deviation for data set  $<ik>$ . The area used for the calculation can be given by  $<xmin>,<xmax>$  for 2d data or  $<xmin>,<xmax>,<ymin>,<ymax>$  for 3d data. If these parameters are omitted, the complete actual plotting range is used for the calculation.

The results are stored in the  $res[i]$  variables in the following order:

3d	2d	
<hr/>		
$res[0]$	$res[0]$	= Number of stored values
$res[1]$	$res[1]$	= average of x-values
$res[2]$	$res[2]$	= average of y-values
$res[3]$		= average of z-values
$res[4]$	$res[3]$	= standard deviation of x-values
$res[5]$	$res[4]$	= standard deviation of y-values
$res[6]$		= standard deviation of z-values
$res[7]$	$res[5]$	= # points used for calculation
$res[8]$	$res[6]$	= used range : xmin
$res[9]$	$res[7]$	= used range : xmax
$res[10]$		= used range : ymin
$res[11]$		= used range : ymax

## A.59 merge

```
merge <i1>,<i2>,... [, "add"]
merge {"-1"|"all"} [, "add"]
```

This command allows one to merge different data sets given by the parameters  $<i1>$ ,  $<i2>$  and so on. If the first parameter is equal to "-1" or "all", all currently loaded data sets are merged. If as last parameter "add" is given, the values are simply added, otherwise they are normalized by the number of contributing data points. The new data set is assigned the next free data set number.

For 1D data sets, the extend of the new data set is defined by the maximum x-values of the data sets to be merged. The grid size is taken as the  $(\text{xmax}-\text{xmin})/(\text{np}-1)$ , i.e. the average step size of the first specified data set.

For 2D dat sets the grid is equivalently determined by the average grid size of the first data set.

## A.60 mouse

```
mouse
mouse {"point" | "line" | "rect" | "xrange" | "yrange" }
```

When the command is given without an additional parameter, KUPLOT redraws the plot with a button area and enters the interactive 'mouse mode'. Use the 'EXIT' button to return to the normal command mode. The command 'mouse point' allows the user to select a point within the active frame (-> afra) and returns the coordinates in the variable res[]:

3d	2d
<hr/>	
res[0]	res[0] = Number of values stored
res[1]	res[1] = x-coordinate of point selected
res[2]	res[2] = y-coordinate of point selected
res[3]	res[3] = z-coordinate of nearest grid pt.
res[4]	res[4] = Mouse button clicked

(1=left, 2=middle, 3=right)

The commands 'mouse line' allows one to select two points that are connected with a line. Select the first point using the left mouse button. The command 'mouse rect' selects two points as well, however they make up the corners of a rectangle. Commands 'mouse xrange' and 'mouse yrange' mark the two points by horizontal/vertical lines. The resulting coordinates are stored in res[] again:

```
res[0] = Number of values stored
res[1] = x-value of first point
res[2] = y-value of first point
res[3] = x-value of second point
res[4] = y-value of second point
res[5] = Mouse button clicked when selecting 2nd point
```

or -1 if action was aborted.

## A.61 mtype

```
mtype <ik>, <p>
```

This command sets the marker type to be used for data set <ik>. The allowed standard values for <p> are listed below. Additionally one can access all PGPLOT markers by adding 100 to the PGPLOT marker number (see users guide). A value of 118 for example will give the PGPLOT symbol 18, a filled star.

Allowed values for <p> are listed in the table. Alternatively you can use the name (without blanks)

```

0 : no marker|none   5 : filled square    10 : slash (/)
1 : dot              6 : empty triangle  11 : backslash (\)
2 : circle empty    7 : times (x)       12 : minus (-)
3 : filled circle   8 : plus (+)        13 : vertical line
4 : square empty    9 : bar (|)          from y-axis
-1 : x_coordinate   -2 : y_coordinate   -3 : "xy": both

```

## A.62 msize

```
msize <ik>, [<size> [, {"x" | "y" | "0" | <data_set>}]]
```

This command sets the marker size for data set *<ik>* to *<size>*. The marker size is given in cm (for our friends in the US: 1 inch = 2.54 cm).

If the third parameter is given and not equal to "0", the marker are scaled proportionally. x The marker size is proportional to the value along the x-axis y The marker size is proportional to the value along the y-axis *<data\_set>* The marker size is proportional to the y-value of data set *<data\_set>*.

## A.63 nexus

Here is a list of the NeXus file related commands. You have to enable NeXus support when compiling KUPLOT. The commands are

### **nxopen**

```
nxopen <file>
```

This command opens the NeXus file called *<file>*. The file stays open until the command 'nxclose' is issued.

### **nxclose**

```
nxclose
```

This command closes the current NeXus file.

### **nxdir**

```
nxdir
```

This command lists all NXdata entries in the currently open NeXus file. Note currently it will only list NXdata entries directly below the top NXentry level.

**nxload**

```
nxload <sdsname>
nxload <sdsname>,<i>,<j>
nxload <sdsname>,<cx>,<cy>
nxload <sdsname>,<xmin>,<xmax>,<ymin>,<ymax>
nxload <sdsname>,<i>,<j>,<k>,<cx>,<cy>
nxload <sdsname>,<i>,<j>,<k>,<xmin>,<xmax>,<ymin>,<ymax>
```

This command reads the content of an SDS (data set) of the NeXus file into a KUPLOT data set. The first parameter is always the name of the data block <sdsname>. To list all data blocks, use the command 'nxdir'. The other parameters allow one to extract a section or rebin the data. In general, parameters <i>, <j> and <k> allow the selection of a cut or slice from the data. If one of these variables is '\*', it represents the direction to be extracted. The parameters <cx> and <cy> allow one to compress the data, i.e. a value of 2 would combine any neighboring data points while reading. Finally, <xmin>,<xmax>,<ymin> and <ymax> allow one to extract only the specified subsection.

**A.64 orientation**

```
orientation [{ "landscape" | "portrait" }]
```

This command sets the orientation of the resulting plot. The default is landscape orientation, which is set, if the command is given without parameters.

**A.65 plot**

```
plot
```

This command is used to plot the graph on the screen.

**A.66 print**

```
print [<dev>], [<command>]
```

This command allows to print the current view graph. The output device type <dev> can be given as first parameter. The only supported printing device is currently 'ps' for POSTSCRIPT. The file is then automatically send to the default printer. As an alternative the print command to be used can be specified as second parameter. Note the command has to be in ". The print device and the default print command can be altered in the file 'blk\_appl.f' of the KUPLOT source code.

**Examples:**

```
print : Print to default device/printer
print ps,"lpr -drscgf" : Print Postscript using command lpr -drscgf ..
```

## A.67 ptype

```
ptype <ik>,<p>
```

This command sets the marker typ to be used for the found maxima of data set  $<ik>$ . The allowed values for  $<p>$  are listed below. Additionally one can access all PGPLOT markers by adding 100 to the PGPLOT marker number (see users guide). A value of 118 for example will give the PGPLOT symbol 18, a filled star.

Allowed values for  $<p>$  are listed in the table. Alternatively you can use the name (without blanks)

0 : no marker none	5 : filled square	10 : slash (/)
1 : dot	6 : empty triangle	11 : backslash (\)
2 : empty circle	7 : times (x)	12 : minus (-)
3 : filled circle	8 : pluss (+)	13 : vertical line
4 : empty square	9 : bar ( )	from y-axis
-1 : x-coordinate	-2 : y-coordinate	-3 : both

## A.68 rdef

```
rdef [<fname>]
```

This command reads the KUPLOT setting from a defaults file. If no file name  $<fname>$  is give, the setting are read from 'kuplot.def'.

## A.69 rebin

```
rebin <ik>,<delta>
```

This command allows one to rebin a 2D data set  $<ik>$  to a new equi-distant grid with a spacing of  $<\text{delta}>$ . The command creates a new data set with the rebinned data. Note that this command will only work for 2D data sets. 3D data can be rebinned by saving them in GN format and reading them in via 'load zz,..' with the desired grid size.

## A.70 reset

```
reset ["all"]
```

This command resets KUPLOT. The next data set will be loaded as set number 1. Values for aspect ratio, distance of tick marks etc. are also reset to automatic determination. The optional parameter "all" will reset KUPLOT back to the initial state including titles, colors, labels and so on.

## A.71 rvalue

```
rvalue <ik>,<ij>[,<weight>] [, partial:<no>]
```

Calculates the residual between curves  $<ik>$  and  $<ij>$ . The last parameter defines the weighting scheme to be used for the calculation. The valid values for  $<weight>$  are:

"one" : $w(y) = 1.0$	"sqa" : $w(y) = y^{**2}$
"sqrt" : $w(y) = \text{SQRT}(y)$	"inv" : $w(y) = 1/y$
"log" : $w(y) = \text{LOG}(y)$	"isq" : $w(y) = 1/\text{SQRT}(y)$
"lin" : $w(y) = y$	"dat" : $w(y) = 1/DY$ (error data set)

The "partial" parameter tells KUPLOT that this is one out of several R-values that will be send back to DIFFEV. If these partial R-values are to be averaged evenly, it is sufficient to provide all partial R-values via the 'rvalue' command. If you want to weigh these R-values differently, use the ==> 'costvalue' command.

The R-values are also copied into the result variable as entries one and two (unweighted and weighted).

## A.72 sann

```
sann <ia>, "OFF"
sann <ia>, <text>, <x>, <y> [, {"l"|"c"|"r"}] [, <angle>] [, <xp>, <yp>]
```

This command allows to add annotations to a plot. The first parameter  $<ia>$  is the annotation number. The text is given as second parameter  $<text>$ . It is possible to use formats (-> filenames) to alter the text using variables. The next two parameters  $<x>$  and  $<y>$  give the position where the text should appear in plot coordinates. You can use 'mouse' to determine the coordinates of a particular point. The default justification of the text is LEFT, i.e.  $<x>$  and  $<y>$  specify the lower left corner of the text. Other justifications can be given as optional parameter. Next the angle between the text and the x-axis  $<angle>$  can be given as optional last parameter. The last two optional parameters  $<xp>$  and  $<yp>$  give the coordinates to the head of an arrow drawn from the text position to this point. If omitted, no arrow is drawn.

Font type, size and color is similar to the settings for file names and caption (-> font). Call 'sann' without parameters or use 'show annotation' to get a list of the currently set annotations.

## A.73 save

```
save <dev>, [<filename>]
```

$<dev>$  : "ps", "eps", "png", "pic", "pdf", "la" This command allows to save the current view graph. The output device type  $<dev>$  can be given as first parameter. The choices for  $<dev>$  are: "ps" for POSTSCRIPT "eps" for Encapsulated POSTSCRIPT "pic" for GIF "png" for a PNG "pdf" for a PDF file format "latex" Latex picture environment  
for the bitmaps 'gif', 'png' and 'pdf' two optional parameters are available:

```
res:<value>
trans:<flag>
```

The resolution expects a numerical value in pixels per inch, default is 300.

The transparency can be either: "trans:trans" for a transparent image or "trans:none" or "trans:solid" for a non-transparent image.

## A.74 sdef

```
sdef [<fname>]
```

This command writes the current setting of KUPLOT in a file. If no file name <fname> is given, the setting are written to 'kuplot.def'.

## A.75 show

```
show <sub> [,<par>]
```

This command displays various parameters and settings of KUPLOT. The current subcommands are:

### annotation

```
show "annotation"
```

Shows the currently set annotation for the active frame.

### color

```
show "color"
```

This command shows the list of current color settings for the background and pens. These settings might be altered using the command 'col'.

### cmap

```
show "cmap"
```

This command shows the RGB values of the current color map used to print bitmaps (only POSTSCRIPT output and X11).

### config

```
show config
```

This command lists configuration information on the screen such as the maximum number of data sets that can be read or the maximum allowed number of frames.

**data**

```
show "data" [,<ik>]
```

Shows information about the loaded data set <ik>. If the parameter <ik> is omitted, information about all loaded data sets is printed.

**error**

```
show "error"
```

The error status of KUPLOT is displayed.

**font**

```
show "font"
```

This command shows the font settings for the active frame.

**frames**

```
show "frames"
```

This command shows the actual frame settings. For more information who to use frames in KUPLOT see help entry 'frames'.

**hlin**

```
show "hlin"
```

Shows the current contour line settings for the active frame.

**plot**

```
show "plot"
```

Shows plot information for the active frame. A value of -9999. means automatic setting of the corresponding parameter before the next plot.

**result**

```
show "result"
```

This command shows the values currently stored in the res[i] variable array.

**style**

```
show "style"
```

This command shows plotting style setting like line color, line style etc. are displayed on the screen.

## A.76 sleg

```
sleg <ik>, ["off", <text>] [,<ox>, <oy>]
```

This command sets a possible caption for data set  $<ik>$ . The text with a sample line, marker, .. will appear in the upper left corner of the plot. The option can be turned off via 'sleg  $<ik>$ ,off'. The caption text can also be a format string followed by the necessary parameters (see filenames entry !). The optional parameters  $<ox>$  and  $<oy>$  allow to specify the UPPER RIGHT corner of the box containing the captions in data coordinates. If the parameters are omitted, the upper right corner of the plot is used by default.

## A.77 smax

```
smax <ik>,<ifen> [,<fname>]
```

This command searches for maxima within data set  $<ik>$ . The parameter  $<ifen>$  determines how many points (in each direction) around a possible maximum must have a lower value as the maximum itself. For smooth data use larger values, for rough data smaller values. If the parameter  $<fname>$  is given, the results are saved to the file  $<fname>$  rather than printed in the screen.

The results of the search are stored in the variable array res[i]:

```
res[0]           : Number of stored values.
res[i]           : Number of x point for i-th maximum (2d data)
res[2i-1],res[2i] : Number of x,y point for i-th maximum (3d data)
```

The actual values can be determined by e.g.  $x[ik,res[5]]$ . For more information, see variables entry.

## A.78 smooth

```
smooth <ik>,<ip> [,<o>] [, {"x" | "y"}]
smooth <ik>, filter:lanczos, width:<points>, damp:<radius>, scale:<factor>
```

This command smoothes the data set number  $<ik>$  over  $<ip>$  points. The number  $<ip>$  should be a positive odd number. KUPLOT uses the Savitzky-Golay algorithm. The order of the smoothing polynomial defaults to two. Alternatively the user can give a higher order (4,6) via the optional parameter  $<o>$ . Within limits this algorithm allows smoothing without resolution (see section 14.8. in Numerical Recipes). Three dimensional files are per default smoothed in x and y direction. The user can restrict the smoothing to only one direction by adding the optional parameter "x" or "y". An normal gliding average smoothing is available via the command -> glat.

The "filter:lanczos" version starts a Lanczos filter. This will include the +-  $<points>$  data points. Default: 4 The damping determines how much further data points are weighted, a smaller  $<radius>$  will use less of the data points further away. Default:0.5 The number on scale:<factor> must be an integer number  $\geq 1$ . It allows to interpolate the data set with more additional points.

The lanczos filer relies on a FFT, thus the data must be on a regular grid.

## A.79 sort

```
sort <ik>
```

This command sorts the data set  $\langle ik \rangle$  increasing in its x-value.

## A.80 spline

```
spline <ik>,<ig>
```

This command will interpolate data set  $\langle ik \rangle$  for x-values given by data set  $\langle ig \rangle$ . The result is stored as a new data set. The interpolation is done using a spline.

## A.81 scale

```
scale [<xmin>, <xmax>] [,<ymin>, <ymax>]
```

This command sets the plotting window to  $\langle xmin \rangle$  to  $\langle xmax \rangle$  and  $\langle ymin \rangle$  to  $\langle ymax \rangle$ . If the data exceed the given window, they will be clipped. If no parameters are given, the plotting window will be set to the maximum / minimum values of the read data sets. If only the parameter pair  $\langle xmin \rangle$  and  $\langle xmax \rangle$  are given, the plotted y-range remains at the current setting.

## A.82 skal

```
skal [<xmin>, <xmax>] [,<ymin>, <ymax>]
```

This command sets the plotting window to  $\langle xmin \rangle$  to  $\langle xmax \rangle$  and  $\langle ymin \rangle$  to  $\langle ymax \rangle$ . If the data exceed the given window, they will be clipped. If no parameters are given, the plotting window will be set to the maximum / minimum values of the read data sets. If only the parameter pair  $\langle xmin \rangle$  and  $\langle xmax \rangle$  are given, the plotted y-range remains at the current setting.

## A.83 tit1/2

```
tit1 <string>
tit2 <string>
```

This command sets the text for the two title lines. Because KUPLOT treats the '/' as a parameter separator it can not be used within the label  $\langle string \rangle$ . The command allows to build a label from a format text and variables/numbers. It is the same mechanism as described in -> filenames.

```
Examples:
tit1 Powder data ZrO2           : 'Powder data ZrO2'
tit1 "Run %d / Sample %d",i[1],i[2] : 'Run #1 / Sample #5'
                                         (for i[1]=1 & i[2]=5)
```

## A.84 torefine

```
torefine ["kuplot:<value>] [, "refine:<value>"]
```

Transfers data sets from kuplot to the global storage that is accessible to refine. These data are persistent, even upon a 'reset' in kuplot.

Possible values are: **kuplot:last** Take the last data set **kuplot:1** Take data set "1", or whatever expression is used instead of "1".

**refine:sigma** The data represent the uncertainties of the experimental data  
**refine:obs** Both place the kuplot data set into the global storage as teh "experimental" or absolutely equivalently " observed" data.  
**refine:exp** The kuplot data set corresponds to the "cost" function value, that is the result of the current Refine parameter set.  
**refine:opti** The kuplot data set is the result of the best Refine parameter set. This entry should be read only, it is usually set by Refine.  
**refine:1** The kuplot data set represents the derivative of the cost function with respect to parameter 1. Same for parameters 2, 3, etc.  
**refine:<par>** The kuplot data set represents the derivative of the cost function with respect to the parameter named <par>, as defined on the 'newparam' command in REFINE.

Instead of the keywords numerical values can be used: **sigma == -3** **obs == -2** **exp == -2** **opti == -1** **cost == 0**

Defaults are: **kuplot:last** **refine:cost**

## A.85 window

```
window <id> [,<sf>]
```

This command allows the user to select a graphics output window. The first window has <id> of 1, the second window 2 and so on. The window command allows to create and toggle between graphics windows (provided you use the default \XSERV device). Note that print and save command will use the content of the currently active window. Optionally the windows can be scaled using the additional scale factor parameter <sf> (default = 0.7).

## A.86 variables

The program KUPLOT recognizes various variables. The contents of a variable can be displayed using the 'eval' command. Some variables are READ ONLY (RO) and can not be changed.

```
Free variables :
```

---

```
"i[<n>]"           : integer variables (currently 0 <= <n> <= 59)
"r[<n>]"           : real variables   (currently 0 <= <n> <= 59)
KUPLOT specific variables :
-----
"x[<ik>,<ip>]"    : x value of point <ip> in loaded data set <ik>.
"y[<ik>,<ip>]"    : y value of point <ip> in loaded data set <ik>.
"dx[<ik>,<ip>]"   : value of sigma x of <ip> in loaded data set <ik>.
```

```

"dy[<ik>,<ip>]"      : value of sigma y of <ip> in loaded data set <ik>.
"z[<ik>,<ix>,<iy>]" : z value of point <ix>,<iy> of loaded data set <ik>.
"nx[<ik>]"           : # points in x-direction of data set <ik>. (RO)
"ny[<ik>]"           : # points in y-direction of data set <ik>. (RO)
"ni[<ik>]"           : data set <ik> is 2D (=0) or 3D (=1). (RO)
"np[<ik>]"           : # points of 1D data set <ik>, (RO)

: # or nx[<ik>]*ny[<ik>] for 2D set <ik>. (RO)

"xmin[<ik>]"         : minimum x-value of data set <ik> (RO)
"xmax[<ik>]"         : maximum x-value of data set <ik> (RO)
"ymin[<ik>]"         : minimum y-value of data set <ik> (RO)
"ymax[<ik>]"         : maximum y-value of data set <ik> (RO)
"zmin[<ik>]"         : minimum z-value of data set <ik> (RO)
"zmax[<ik>]"         : maximum z-value of data set <ik> (RO)
"n[1]"                : number of loaded data sets (RO)
"n[2]"                : maximum number of data sets (RO)
"n[3]"                : maximum number of frames (RO)
"n[4]"                : maximum number of annotations (RO)
"n[5]"                : maximum number of bonds (RO)
"p[i]"                : Fit parameters
"s[i]"                : Fit parameter sigmas
"axis[1,i]"           : Angle of labels for axis i (1=x,2=y)
"axis[2,i]"           : Length of major ticks for axis i (1=x,2=y)
"axis[3,i]"           : Length of minor ticks for axis i (1=x,2=y)
"axis[4,i]"           : Subdivisions between ticks, axis i (1=x,2=y)
"axis[5,i]"           : Distance numbers - axis i (1=x,2=y)
"axis[6,i]"           : Distance label - axis i (1=x,2=y)
"pwin[i]"             : Current plotting dimensions
                        (1:xmin,2:xmax,3:ymin,4:ymax) (RO)
"cmap[<ic>,3]"       : color map entry RGB for color <ic>
"cmax[1]"              : maximum number of bitmap colors (RO)
"size[i]"              : scale factor for output (1=x11,2=ps,3=pic)
Variables containing command results :

-----
"res[<n>]"           : Result of several calculations that return more
                        than one value. res[0] contains the number of
                        values returned, res[1] to res[res[0]] contain the
                        actual values. The values of "res" change every time,
                        a command is applied that returns more than one
                        value. The old values are lost !

```

## A.87 errors

### Error messages

The program has been written such that it should handle almost any typing error when giving commands and hopefully all errors that result from calculation with erroneous data. When an error is found an error message is displayed that should get you back on track. Command language specific errors can be found in the 'command language' section of the on-line help. The error messages are grouped in the following categories:

#### KUPL Errors specific to KUPLOT usage

Each error message is displayed together with the corresponding category <cccc> and the error number <numb> in the form:

\*\*\*\*\*CCCC\*\*\*\* Message

\*\*\*\*\* numb \*\*\*\*\*

More information to a particular error message can be found by typing 'help <cccc> <numb>' where <cccc> is the category and <numb> the error number.

## appl

KUPLOT errors

These messages describe application specific errors (here KUPLOT) in contrast to errors concerning the command language handling.

### Error -1: Maximum number of data sets exceeded

There is no space left for a new data set. The maximum number of data sets is defined by MAXKURVTOT in 'kuplot.inc'. In order to change this setting you have to recompile the program.

### Error -2: Unknown file format

The given file format with the 'load' command is unknown. Check the on-line help for 'load' for a list of valid file formats.

### Error -3: Maximum number of data points exceeded

There is not enough space for the requested data set left. The maximum number of data points for ALL data sets is defined by MAXARRAY in 'kuplot.inc'.

### Error -4: Invalid data set selected

The given data set number is not valid. Try 'show data' for a list of loaded data sets.

### Error -5: Not an ASCII PGM file

The PGM file to be read is not ASCII (Code P2). Currently KUPLOT can only read ASCII PGM files. If you have the 'pbmplus' package installed, use the tool 'pnmmoraw' to convert from binary to ASCII format.

### Error -6: Number of data points exceeds limit

There is not enough space for the requested data set left. The maximum number of data points for ALL data sets is defined by MAXARRAY in 'kuplot.inc'.

### Error -7: Invalid parameter value entered

The given parameter value like color, line type or font id is invalid. Check the on-line help for the corresponding command for valid parameters.

**Error -8: Use only if NO data set is loaded**

The command can only be used, if NO data sets are loaded. Use the command 'rese' and reload the data sets after executing the command.

**Error -9: Too many excluded regions**

The number of excluded regions exceeds the upper limit. The number is defined in 'load.f' and the program must be recompiled after changing the value.

**Error -10: Too many major tick marks**

The number of major tick marks is too large to be displayed. Use the 'mark' command to adjust the number of tick marks.

**Error -11: Invalid plot device selected**

The output device selected is invalid. Check the on-line help for supported output device types.

**Error -12: No data present to plot**

There are no data present to be plotted. You are doing something wrong :-)

**Error -13: Too much points for spline / steps**

The number of points for the current data set is too large for 'step' or 'spline' mode. The value is set by MAXSP in the file 'config.inc'. The program needs recompiling after the value is changed.

**Error -14: Invalid contour line set selected**

An invalid contour line set was selected. The maximum number of contour line sets is specified by MAXHL in the file 'config.inc'. The program must be recompiled after the variable was altered.

**Error -15: Invalid frame selected**

An invalid frame was selected. Check 'show frames' for the current frame settings.

**Error -17: Maximum number of frames exceeded**

You want more frames than KUPLOT can handle. If you absolutely need more frames, change the setting of MAXFRAME in 'config.inc' and re- compile the program.

**Error -18: Invalid RGB color found**

RGB colors must have values between 0.0 and 1.0. The entered value is invalid.

**Error -19: Invalid peak number found**

There is no maximum with the specified number. Use the 'smax' command to see a list of found maxima.

**Error -20: No data points in selected area**

The area selected for the command contains no points.

**Error -21: Too many maxima found at search**

The search routine called by 'smax' has found too many maxima. Try limiting the number by changing the parameter <ifcn> (see help for smax). The maximum number is set by MAXMAX in the file 'plot.f' and 'math.f'.

**Error -22: No maxima found**

No maxima were found. Try a different setting of the <ifcn> parameter (see help for command 'smax') or are there no maxima after all ???

**Error -23: Incompatible data sets for KCAL**

The command 'kcal' works only for data sets of the same typ (2d or 3d) and identical size.

**Error -24: Size of res[] array exceeded**

The size of the result variable array res[i] is exceeded. The size is defined in 'param.inc'.

**Error -25: Invalid fit function selected**

An invalid fit function was selected. Note that not all functions are available for 3D data sets. Check the help entry for 'fit func' for more information.

**Error -26: Invalid fit parameter selected**

An invalid fit parameter was selected.

**Error -27: Invalid weighting scheme selected**

The selected weighting scheme is invalid. Check the help entry for 'fit wic' for more information.

**Error -28: Invalid annotation number selected**

The given annotation number (command 'sann') is not valid. If you need more annotations, you have to change MAXAN in the file 'kuplot.inc' and recompile KUPLLOT.

**Error -29: x or y-range of data set is zero**

The x- or y-range of the read 3D file is zero. Check your input file.

**Error -30: Not enough maxima found for start values**

The number of Gaussians or Lorenzians exceeds the number of maxima found in your data set. You can set the start parameters manually (-> 'par') or adjust the size of the search window (-> 'mfen') until all needed maxima are found.

**Error -31: To many fit parameters**

The number of fit parameters exceeds the maximum limit. Either use a theory function with less parameters or adjust the value of MAXPARA in 'config.inc' and recompile KUPLOT.

**Error -33: Invalid bond definition selected**

An invalid bond definition was specified. Check your input.

**Error -34: Data set to large for bitmap drawing**

In order to use PGPLOT contour and bitmap routines, the data set must fit in an array of the size MAXZ x MAXZ regardless of the current setting of MAXDAT. Either use a smaller data set or adjust the value of MAXZ in 'config.inc' and recompile KUPLOT.

**Error -35: No valid frame selected**

During the select frame mouse operation and error occurred. Try selecting the appropriate frame again. Note that there might be areas of the plotting surface that do not belong to any frame.

**Error -36: Configuration mismatch file & KUPLOT**

The loading of KUPLOT defaults works only for default files that were created with a KUPLOT version using the identical array size definitions.

**Error -37: Scan not found in SPEC file**

An invalid scan number was specified. Check your SPEC file for the correct number (#S xxx).

**Error -38: Invalid column value specified**

The column name or number was not found for the specified scan. Note that names must be given in full and are case sensitive.

**Error -39: Invalid smoothing size specified**

An invalid smoothing size was given. Valid numbers are odd integers. The minimum value for a smoothing polynomial of the order of 2 is 5, for 4th order 7.

Error -40: Maximum derivative exceeded  
Error -41: MCA scan not found  
Error -42: Invalid window ID selected  
Error -43: Invalid value/range for log. axis  
Error -44: Data sets must have same x-range  
Error -45: Step size of data sets is different  
Error -46: Invalid bank number found in iparm. file  
Error -47: Error reading instrument parameter file  
Error -48: Unsupported GSAS binning type found  
Error -49: No TIME\_MAP entry found in GSAS file  
Error -50: Requested bank not found in GSAS file  
Error -51: Unit for GSAS TOF conversion invalid  
Error -52: Unsupported incident spectr. function  
Error -53: This KUPLOT has NeXus support disabled  
Error -54: Close current NeXus file first  
Error -55: No NeXus file currently open  
Error -56: NeXus SDS has too many dimensions  
Error -57: Invalid SDS name specified (try nxdir)  
Error -58: SDS section value out of range  
Error -59: Invalid column number specified  
Error -60: Multiple point with same x, no spline  
Error -61: Singular matrix in Savitzky calculation  
Error -62: Data set is not 2D  
Error -63: Parameter no outside range  
Error -64: DIFFEV Summary file not found  
Error -65: DIFFEV LOG or Current file not found  
Error -66: DIFFEV Rvalue file not found