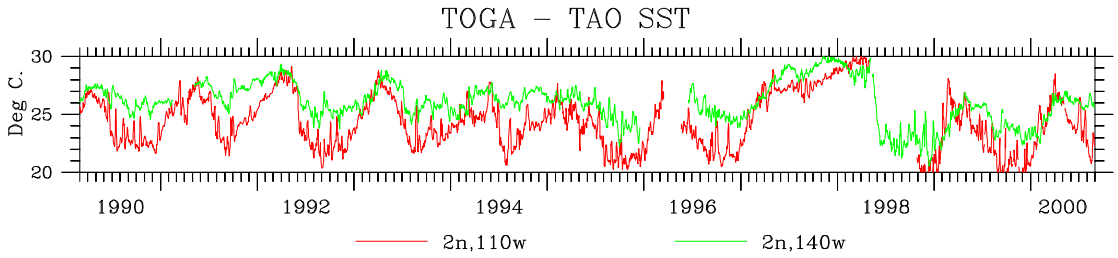
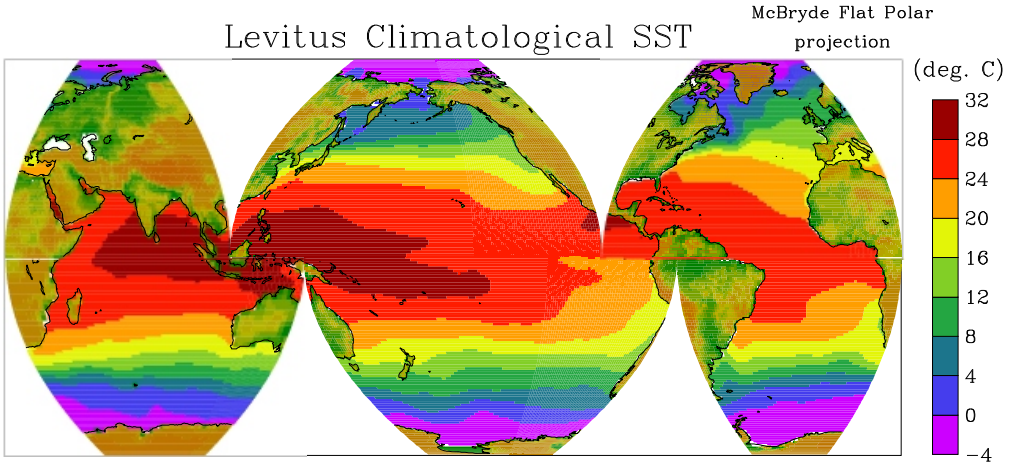


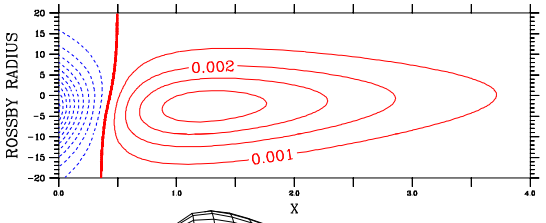
NOAA / PMEL

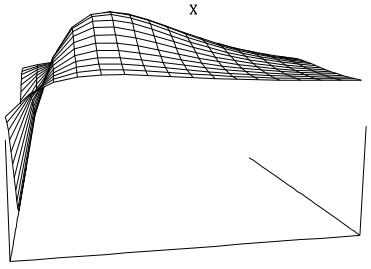
F E R R E T

An Analysis Tool for Gridded Data



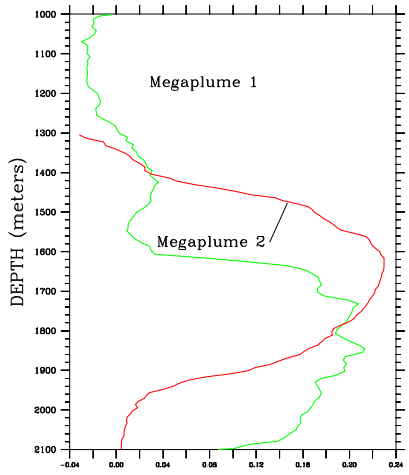
Perturbation Solution





$-(B+1/2)*D_y*e^B/D^2$

VENTS Megaplume Thermal Structure



FERRET

USER'S GUIDE

Version 6.02

NOAA/PMEL/TMAP

Steve Hankin
Jon Callahan, Ansley Manke
Kevin O'Brien, Jing Li
April 26, 2007

About the Cover

The cover of this User's Guide was produced by Ferret. From the top down the plots are: "TOGA-TAO SST," time series from the Tropical Pacific TAO array; "Levitus Climatological SST," an equal area projection of level one of the annual Climatological Atlas of the World Oceans by Sydney Levitus of NOAA/NODC; "Perturbation Solution," a visualization of abstract functions by Dr. Ping Chang; "Vents Megaplume Thermal Structure," vertical temperature profiles of undersea thermal vents from the NOAA Vents program.

Contents

CHAPTER 1: INTRODUCTION

CHAPTER 2: DATA SET BASICS

CHAPTER 3: VARIABLES AND EXPRESSIONS

CHAPTER 4: GRIDS AND REGIONS

CHAPTER 5: ANIMATIONS AND GIF IMAGES

CHAPTER 6: CUSTOMIZING PLOTS

CHAPTER 7: HANDLING STRING DATA: STRING VARIABLES AND "SYMBOLS"

CHAPTER 8: WORKING WITH SPECIAL DATA SETS

CHAPTER 9: COMPUTING ENVIRONMENT

CHAPTER 10: CONVERTING TO NETCDF

CHAPTER 11: WRITING EXTERNAL FUNCTIONS

PART II: COMMANDS REFERENCE

APPENDIX B: PPLUS USERS GUIDE

APPENDIX C: PLOTPLUS PLUS: FERRET ENHANCEMENTS TO PLOTPLUS

Chapter 1: INTRODUCTION

Ch1 Sec1. OVERVIEW

Ferret is an interactive computer visualization and analysis environment designed to meet the needs of oceanographers and meteorologists analyzing large and complex gridded data sets. "Gridded data sets" in the Ferret environment may be multi-dimensional model outputs, gridded data products (e.g., climatologies), singly dimensioned arrays such as time series and profiles, and for certain classes of analysis, scattered n-tuples (optionally, grid-able using Ferret's objective analysis procedures). Ferret accepts data from ASCII and binary files, and from two standardized, self-describing formats. Ferret's gridded variables can be one to four dimensions—usually (but not necessarily) longitude, latitude, depth, and time. The coordinates along each axis may be regularly or irregularly spaced

Ferret offers the ability to define new variables interactively as mathematical expressions involving data set variables and abstract coordinates. Calculations may be applied over arbitrarily shaped regions. Ferret's "external functions" framework allows external code written in FORTRAN, C, or C++ to merge seamlessly into Ferret at runtime. Using external functions, users may easily add specialized model diagnostics, advanced mathematical capabilities, and custom output formats to Ferret. A collection of general utility external functions is included with Ferret.

Ferret provides fully documented graphics, data listings, or extractions of data to files with a single command. Without leaving the Ferret environment, graphical output may be customized to produce publication-ready graphics. Graphic representations include line plots, scatter plots, line contours, filled contours, rasters, vector arrows, polygonal regions and 3D wire frames. Graphics may be presented on a wide variety of map projections. Interfaces to integrate with 3D and animation applications, such as Vis5D and XDataSlices are also provided.

Ferret has an optional point-and-click graphical user interface (GUI). The GUI is fully integrated with Ferret's command line interface. The user may freely mix text-based commands with mouse actions (push buttons, etc.). Ferret's journal file will log all of the actions performed during a session such that the entire session, including GUI inputs, can be replayed and edited at a later time. The GUI version is not currently supported, and is not available on all operating systems.

This User's Guide describes only the command line interface to Ferret. Other documents describe the point and click interface.

Ferret was developed by the Thermal Modeling and Analysis Project (TMAP) at NOAA/PMEL in Seattle to analyze the outputs of its numerical ocean models and compare them with gridded, observational data. Model data sets are often multi-gigabyte in size with mixed 3- and 4-dimensional variables defined on staggered grids.

Ferret graphics calls are made using the Plot Plus (PPLUS) graphics package, which is contained within Ferret. Plot Plus was written by Don Denbo. The Ferret version of PPLUS has diverged somewhat from the original, and the Ferret developers are responsible for these changes and for all of Ferret's graphics. Additions to PPLUS, for Ferret only, are documented in Appendix C of this manual (p. 555), which also has a brief history of the PPLUS graphics package.

Ferret is supported on a variety of Unix workstations with a version also available for Windows NT/9x/XP. Ferret is available at no charge from anonymous FTP [node ftp.ferret.noaa.gov] or from the World Wide Web [URL http://www.ferret.noaa.gov/Ferret].

Ch1 Sec1.1. Ferret User's Group

The Ferret User's Group provides a venue to ask experienced Ferret users for advice solving problems and to keep abreast of the latest Ferret updates. To (un)join simply send an e-mail message to

```
Majordomo@ferret.pmel.noaa.gov
```

and include a message which says simply

```
(un)subscribe ferret_users
```

(Note this must be in the e-mail message BODY—not in the subject line.) To learn about the user's list without joining send this message instead to the same address:

```
info ferret_users
```

Ch1 Sec1.2. Ferret Home Page

The Ferret Home Page contains source code distributions, on line documentation, Users' Group archives, Frequently Asked Questions and more. It is available at

http://ferret.pmel.noaa.gov/Ferret/FAQ/ferret_FAQ.html

Ch1 Sec2. GETTING STARTED

A quick way to get to know Ferret is to run the tutorial provided with the distribution.

```
% ferret
yes? GO tutorial
```

If Ferret is not yet installed consult the chapter "Computing Environment" (p. 257). (The tutorial is also available through the World Wide Web through Ferret's on-line demonstrations page.) The tutorial demonstrates many of Ferret's features, showing the user both the commands given and Ferret's textual and graphical output. You may find the explanations, terms and examples in this manual easier to understand after running the tutorial.

Ch1 Sec2.1. Concepts

Words in bold below are defined in the glossary of this manual.

In Ferret all variables are regarded as defined on grids. The grids tell Ferret how to locate the data in space and time (or whatever the underlying units of the grid axes are). A collection of variables stored together on disk is a data set.

To access a variable Ferret must know its name, data set and the region of its grid that is desired. Regions may be specified as subscripts (indices) or in world coordinates. Data sets, after they have been pointed to with the SET DATA command (alias "USE"), may be referred to by data set number or name.

Using the LET command new variables may be created "from thin air" as abstract expressions or created from combinations of known variables as arbitrary expressions. If component variables in an expression are on different grids, then regridding may be applied simply by naming the desired grid.

The user need never explicitly tell Ferret to read data. From start to finish the sequence of operations needed to obtain results from Ferret is simply:

1) specify the data set
2) specify the region
3) define the desired variable or expression (optional)
4) request the output

For example (Figure 1_1),

```
yes? USE coads              !global sea surface data
```

```
yes? SET REGION/Z=0/T="16-JAN-1982"/X=160E:160W/Y=20S:20N
yes? VECTOR uwnd,vwnd     !wind velocity vector plot
```

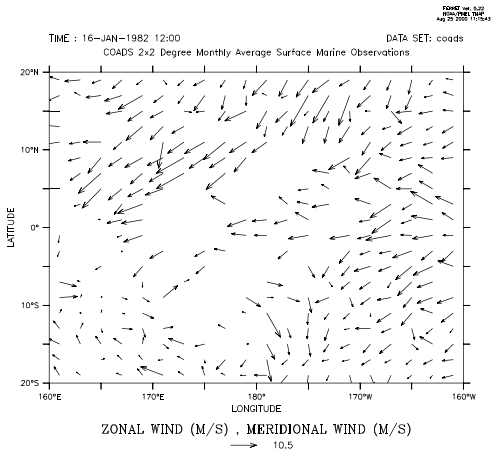


Figure 1_1

Ch1 Sec2.1.1. Thinking like a Ferret:

(A discussion on the Ferret outlook on the concepts of data, variables, grids and other basics of Ferret.)

Plottable variables

For this discussion we will coin the term "plottable variables." There are no non-plottable variables that will come up in this discussion but "variables" is a bit too generic. Plottable variables are of 3 types:

- file variables – read from disk files
- user-defined variables – defined by the LET command
- pseudo-variables – regions (I,J,K,L,X,Y,...) used as variables

As much as possible Ferret tries to make all types of variables indistinguishable. All plottable variables are defined on grids. No plottable variables exists in a vacuum for Ferret. The grid on which a plottable variable exists tells how to locate the variable in space and time. In cases where the variables are abstract in nature—disconnected from space and time—Ferret will associate those variables with grids that are abstract, too. Where a geographical grid will associate the Nth position along an axis with a location (like 20 degrees north latitude) an abstract grid will simply associate the Nth position with the number N. Plottable variables may be regridded to other grids than the one on which they are defined. (Done with "G=".)

All references to plottable variables must have a complete context. A complete context will be described in detail later—briefly it means a region in space, an interval in time and the data set(s) in which the variables will be found.

Grids

All Ferret grids are 4-dimensional. In most cases the axes have the obvious interpretation of 3 space coordinates and time but sometimes the axes are abstract.

A grid is composed of 4 axes, each describing the coordinates along one dimension. 3d, 2d, 1d and 0d grids are regarded as special cases of the full 4 dimensions in which 1 or more axes are set to "NORMAL".

Ferret tries to look at all axes equally—the same syntax of regions and transformations applies to each. Calendar dates, east-west longitudes and north-south latitudes are merely convenient ways to format positions along axes that have special interpretations to people—not to Ferret. (The only exception to this is that if the Y axis has units of Latitude Ferret will insert cosine(Latitude) factors into some calculations.)

Axes and grids may be defined by "grid files" (which normally have .GRD filename extensions). Axes may also be defined by the DEFINE AXIS command; grids by the DEFINE GRID command.

Contexts

A context is a region or point in space and time and a data set(s). This is the information needed by Ferret to make sense of a reference to a plottable variable. Suppose that "U" is a variable in a data set (file) called U_DATA. A command like "PLOT U" is meaningful only when Ferret knows that it is supposed to be looking for U in data set U_DATA and knows where in 4-dimensional space it is supposed to plot.

The context space-time region may be described by a mix of subscript and world coordinate positions. Subscripts are specified by I=,J=,K=,L= for axes 1 through 4, respectively. World coordinates are specified by X=,Y=,Z=,T=. On the right of the equal sign a single point may be given or a range specified by low:high may be given. Special formats are allowed for X= (longitude, e.g. 160W), Y=(latitude, e.g. 23.5S) and time (calendar dates like "7-NOV-1989:12:35:00" in quotation marks).

The data set may be given by name or number. The commands SET DATA and CANCEL DATA and the D= context descriptor all accept the name of the data set or its number. The data sets are numbered by the order in which they are pointed to with SET DATA. This order may be seen with SHOW DATA.

You can tell Ferret the context in 3 places:

1. The program context: Using the commands SET REGION and SET DATA you can describe a context in which all commands and expressions will be interpreted. You can look at the program context with SHOW REGION and SHOW DATA. (The command SET DATA is used both to initialize new data sets and to make previously initialized sets the current program context. When SET DATA initializes a new data set that set automatically becomes the data set for the program context.) Example: SET REGION/Z=50

2. The command context: Using the command qualifiers I,J,K,L,X,Y,Z,T and D commands like PLOT,CONTOUR,SHADE,LIST and VECTOR can specify additional context information. Command context information on any axis or on the data set will replace any program context information on the same axis or the data set.
3. The variable context: Using the same qualifiers as the command context any plottable variable name can be modified with additional context information in square brackets (e.g. LET U200 = U[Z=200,D=U_DATA], or LIST U[I=1:100:5]). Variable context information on any axis or the data set will replace any program or command context information on the same axis or the data set.

Transformations

Ferret can transform plottable variables along their axes. Transformations may be specified only in the variable context. Ferret understands a number of transformations that may be specified with the space-time region qualifiers. Some examples: PLOT U[Z=0:100@AVE] — the variable U averaged between Z=0 and Z=100 LIST/L=1:200 U[L=@SBX:5] — U with a boxcar smoother of width 5 points along L.

Also,

- @FAV (fill data holes with averages)
- @DIN (definite integral) @IIN (indefinite integral)
- @DDC (centered derivative)
- @SHF (shift data a number of points along an axis)
- @MIN (minimum value along an axis)

... and others (see HELP TRANSFORMATIONS inside Ferret)

Ch1 Sec2.2. Unix command line switches

```
ferret [-batch <file>.ps][-memsize Mwords] [-unmapped] [-gui] [-help]
[-gif] [-server] [-script [arg1] [arg2]...]
```

-memsize Mwords
specify the memory (data cache) size in Megawords (default is 6.4)

If memory is severely limited on a system Ferret's default memory cache size may be too large to permit execution. In this case use the "-memsize" qualifier on the command line to specify a smaller cache.

-unmapped
use invisible output windows (useful for creating animations and GIF files, or to create metafiles when window resizing is needed. In this case you can create metafiles, using any SET WINDOW/SIZE or /ASPECT commands required. Then use gksm2ps to convert to postscript, using gksm2ps options to control orientation and sizing.)

-gui
start Ferret in point-and-click mode (may not be available on all platforms). This option is not currently supported. Starting Ferret with ferret -gui will run the current version of Ferret, but some features may not work. If you need such features, you will need to use the command-line version of Ferret.

-help
obtain help on the Unix command line options

-nojnl
start Ferret without a journal file. Within the Ferret session, you can use SET MODE JOURNAL:<filename> to turn on journaling and set the journal file name if desired.

--gif
Ferret can run in batch mode—without an X server (see also -server below). Graphical output is buffered, and is stored in a GIF file by executing the FRAME command. For example:

```
> ferret -gif
yes? (commands that generate a plot...)
yes? FRAME/FILE=picture.gif
```

sends the stored graphical output from Ferret to the GIF file picture.gif.

Please note the following when using batch mode:

- Window resizing only works if the window is cleared before resizing the window. For instance:
  yes? set window/clear/size=0.25
  will resize the window while
  yes? set window/size=0.25/clear
  will cause an error.
- Avoid metafile commands when running in batch mode. In particular,
  yes? set mode meta
  may cause problems.
- Don't create new Ferret windows when running without an X server. The following command:
  yes? set window/new
  will cause Ferret to crash.

-batch
Ferret can generate PostScript files or metafiles without an X server. If you wish to use this mode, start Ferret with the -batch option:

```
ferret -batch <file>.ps
```

```
ferret -batch <file>.plt
```

where <file> is the name of the output file. <file>.ps will write postscript files and <file>.plt will write metafiles.

Please note the following when using PostScript mode:

- The PostScript output will not be fully written to the output file until you exit from Ferret.
- Window sizing commands do not have any effect on PostScript output. (If window sizing is needed, can start Ferret with the -unmapped option, and create metafiles, using any SET WINDOW/SIZE or /ASPECT commands required. Then use gksm2ps to convert to postscript, using gksm2ps options to control orientation and sizing if desired.)
- Avoid metafile commands when running in PostScript mode.
- Don't create new Ferret windows when running without an X server. The following command:

  yes? set window/new

  will cause Ferret to crash.

When in batch metafile mode:

A new myfile.plt.~*~ file is started with each new plot "page" that we'd see when running interactively.

The following commands are ignored in metafile batch mode:

- SET MODE METAFILE
- CANCEL MODE METAFILE
- SET WINDOW ! except for
  - SET WINDOW/ASPECT applies the new aspect ratio
  - SET WINDOW/CLEAR starts a new metafile

New plots may be started with these commands

- New plot command (if we are not in a viewport and not overlaying) PLOT, CONTOUR, SHADE, FILL, VECTOR, CONTOUR,WIRE
- CANCEL VIEWPORT
- SET WINDOW/CLEAR
- PPLUS/RESET

-server

Run in server mode -- don't stop on message commands. This mode uses primitive (but faster) command line reading, so it is generally preferred when setting up Ferret from a pipe or batch process. See the notes above under -gif regarding window sizing commands.

-script

Run a script, with optional arguments, and exit. Ferret starts, the script runs, and Ferret exits. If Ferret encounters an error, it will issue any error messages and exit to the command line. The switch also sets the -nojnl, -server, and -noverifyswitches (MODE VERIFY and MODE JOURNAL may be turned back on within the script). It supresses the banner lines. So that the command line reader can read and process any arguments to the script, this option must be specified last, after any other command-line switches (e.g. -gif or -memsize).

```
ferret -script file.jnl [arg1] [arg2] [arg3]
```

Ch1 Sec2.3. Sample sessions

This section presents a number of short Ferret sessions that demonstrate common uses. Data sets used in these sessions and throughout this manual are included with the distribution. If Ferret is installed on your system, you can duplicate the examples shown.

Ch1 Sec2.3.1. Accessing a netCDFdata set

In this sample session, the data set "monthly_navy_winds" is specified and certain aspects of it are examined. The command SHOW DATA/VARIABLES displays the variables in "monthly_navy_winds" and where on each axis they are defined. SET REGION specifies where in the grid the user wishes to examine the data. VECTOR produces a vector plot of the indicated variables over the specified region.

```
yes? USE monthly_navy_winds            ! specify the data set
yes? SHOW DATA/VARIABLES              ! what's in it?
     currently SET data sets:
    1> /opt/local/ferret/fer_dsets/descr/monthly_navy_winds.des
(default)
     FNOC 2.5 Degree 1 Month Average World-wide Wind Field
 name     title                          I         J         K         L
 UWND     ZONAL WIND                   1:144     1:73      ...
1:132
            M/S on grid FNOC251 with -99.9 for missing data
            X=18.8E:18.8E(378.8)  Y=91.2S:91.2N
 VWND     MERIDIONAL WIND              1:144     1:73      ...
1:132
            M/S on grid FNOC251 with -99.9 for missing data
            X=18.8E:18.8E(378.8)  Y=91.2S:91.2N
  time range: 16-JAN-1982 20:00 to 17-DEC-1992 03:30
```

Ch1 Sec2.3.2. Reading an ASCII data file

Many examples of accessing ASCII data are available later in this manual. See the chapter, "Data Sets" (p. 33) The simplest access, one variable with one value per record, looks like this:

```
% ferret
yes? FILE/VARIABLE=v1 snoopy.dat
yes? PLOT v1
yes? QUIT
```

Ch1 Sec2.3.3. Using viewports

The command SET VIEWPORT allows the user to divide the output graphics "page" into smaller display viewports.

In this sample session, we create two plots in two halves of a window (Figure 1_2):

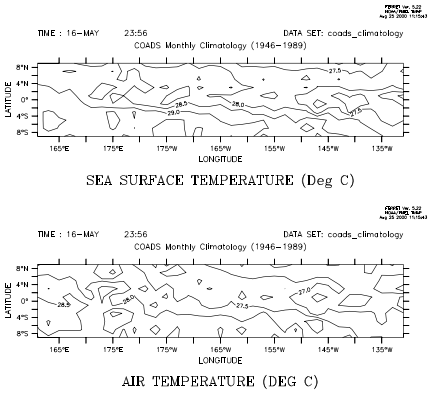


Figure 1_2

```
% ferret
yes? USE coads_climatology
yes? SET REGION/X=160E:130W
yes? SET REGION/Y=-10:10/L=5
yes? SET VIEWPORT upper
yes? CONTOUR sst
yes? SET VIEWPORT lower
yes? CONTOUR airt
yes? QUIT
```

Ch1 Sec2.3.4. Using abstract variables

Abstract variables (expressions that contain no dependencies on disk-resident data) can be easily displayed with Ferret. See the chapter "Variables and Expressions", section "Abstract variables" (p. 63), for several examples and detailed information.

For example, a user wishing to examine the function SIN(X) on the interval [0,3.14] might use (Figure 1_3):

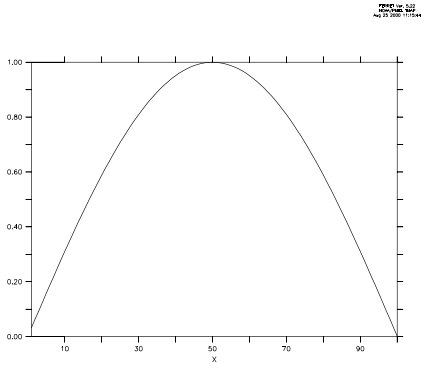


Figure 1_3

```
% ferret
yes? PLOT/I=1:100 sin(3.14*I/100)
yes? QUIT
```

Ch1 Sec2.3.5. Using transformations

A transformation is an operation performed on a variable along a particular axis and is specified with the syntax "@trn" where "trn" is the name of a transformation. See the chapter "Variables and Expressions", section "Transformations" (p. 112), for detailed information.

A user may wish to look at ocean temperatures averaged over a range of depths. In this sample session, we look at temperatures averaged from 0 to 100 meters of depth using a data set which has detailed resolution in depth (Figure 1_4). We plot the data along longitude 160 west from latitude 30 south to 30 north.

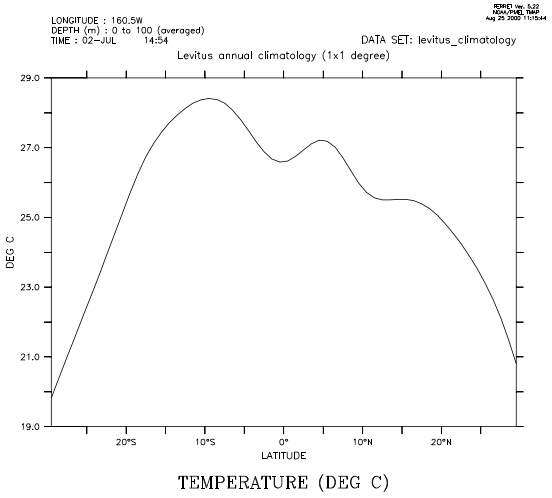


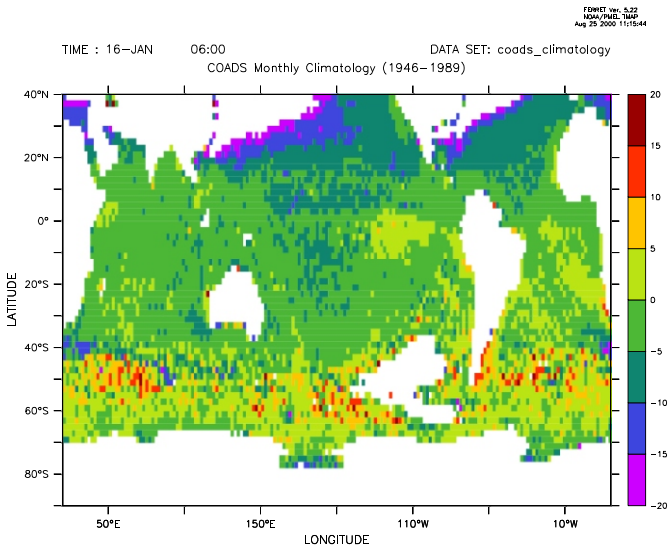
Figure 1_4

```
% ferret
yes? USE levitus_climatology
yes? SET REGION/Y=30s:30n/X=160W
yes? PLOT temp[Z=0:100@AVE]
yes? QUIT
```

Ch1 Sec2.3.6. Using algebraic expressions

See the chapter "Variables and Expressions", section "Expressions" (p. 73) for a description of valid expressions.

In this example, the data set contains raw sea surface temperatures, air temperatures, and wind speed measurements. We wish to look at a shaded plot of sensible heat at its first timestep (L=1) (Figure 1_5). We specify a latitude range and contour levels.



SENSIBLE HEAT

Figure 1_5

```
% ferret
yes? USE coads_climatology            !monthly COADS climatology
yes? LET kappa = 1                    !arbitrary
yes? LET/TITLE="SENSIBLE HEAT"  sens_heat = kappa * (airt-sst) * wspd
yes? SHADE/L=1/LEV=(-20,20,5)/Y=-90:40 sens_heat
yes? QUIT
```

Ch1 Sec2.3.7. Finding the 20-degree isotherm

Isotherms can be located with the "@LOC" transform, which returns the axis location where the value of the argument of @LOC first occurs. Thus, "TEMP[Z=0:200@LOC:20]" locates the first occurrence of the temperature value 20 along the Z axis, scanning all the data between 0 and 200 meters.

A session examining the 20-degree isotherm in mid-Pacific ocean data (Figure 1_6):

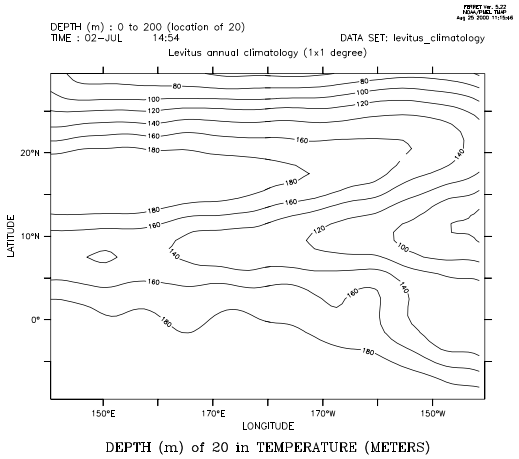


Figure 1_6

```
% ferret
yes? USE levitus_climatology
yes? SET REG/Y=10s:30n/X=140E:140W
yes? PPL CONSET .12  !label size
yes? CONTOUR temp[Z=0:200@LOC:20]
yes? QUIT
```

Note that the transformation @WEQ could have been used to display ANY variable on the surface defined by the 20 degree isotherm.

Ch1 Sec3. COMMON COMMANDS

A quick reference to the most commonly used Ferret commands (typing "SHOW COMMANDS" at the Ferret prompt lists all commands):

Command	Description
USE	names the data set to be analyzed (alias for "SET DATA")
SHOW DATA	produces a summary of a variable
SHOW GRID	examines the coordinates of a grid
SET REGION	sets the region to be analyzed
LIST	produces a listing of data
PLOT	produces a plot
CONTOUR	produces a line contour plot
FILL	produces a color filled contour plot
SHADE	produces a shaded-area plot
VECTOR	produces a vector arrow plot
POLYGON	plots polygonal regions

Command	Description
DEFINE	define new axes, grids, and symbols
STATISTICS	produces summary statistics about variables and expressions
LET	defines a new variable
SAVE	saves data in netCDFformat
GO	executes Ferret commands contained in a file

Information on all Ferret commands is available in Part II, Commands Reference, of this manual.

Ch1 Sec4. COMMAND SYNTAX

Commands in program Ferret conform to the following template:

```
COMM [/Q1/Q2...] [SUBCOM[/S1/S2...]] [ARG1 ARG2 ...] [!comment]
```

where

COMM	is a command name ***yes? LIST***
Q1...	are qualifiers of the command ***yes? CONTOUR/SET_UP***
SUBCOM	is a subcommand name ***yes? SHOW MODE***
S1...	are qualifiers of the subcommand ***yes? SET LIST/APPEND***
ARG1...	are arguments of commands ***yes? CANCEL MODE INTERPOLATE***

notes...

- The length of the command line is limited to a maximum of 2048 characters.
- Command lines ending with back slash are regarded as incomplete -- a special prompt is given to indicate that the next line is a continuation .
- Items in square brackets are optional.
- One or more spaces or tabs must separate the command from the subcommand and from each of the arguments. Spaces and tabs are optional preceding qualifiers.
- Multiple commands, separated by semi-colons, can be given on the same line.
- Command names, subcommand names, and qualifiers require at most 4 characters. (e.g., ***yes? CANCEL LIST/PRECISION*** is equivalent to ***yes? CANC LIST/PREC***)
- Some qualifiers take an argument following "=" (e.g., ***yes? LIST/Y=10S:10N***).
- An exclamation mark normally signifies the end of a command and the start of (optional) comment text.
- The backslash character (\), when placed directly before an exclamation point (!), apostrophe ('), semicolon (;), or forward slash (/), will hide it ("escape it") from Ferret.
- See the Expressions section (p. 73) for information on algebraic expressions as arguments to commands
- See the Symbols sections (p. 229) for information on symbol substitution in commands

Examples:

- A simple command and argument
  yes? LIST sst

- A comment on the command line
  yes***? SET REGION/L=1/X=130:290/Y=-23:23 ! January in the Tropical Pacific***

- Commands with qualifiers and arguments
  yes? VECTOR/L=30/COLOR=RED u,v
  yes? LET/UNITS=M ht = z[GZ=temp] - z0

- Subcommands
  yes? SET MODE METAFILE
  yes? SET REGION/X=130E:120W/J=20:40/Z=0/T=1-jan-1982:31-jan-1992

- Symbols used in a command(see p. 229) Note multiple commands on a line
  yes? DEFINE SYMBOL lower = -2; DEFINE SYMBOL upper = 6
  yes? SHADE/I=($lower):($upper) temp

- Use Square brackets to specify a variable's dataset or grid, range and optionally a delta- for the variable, or a transformation (see p.59)
  yes? PLOT temp[X=180,L=1:50]
  or
  yes? LIST temp[X=130:200@AVE,L=1:50:5]
  or
  yes? LET/UNITS=M ht = z[GX=temp] - z0

- Immediate mode expression: enclosed in grave accents. (see p. 135) (The expression must evaluate to a scalar, and is evaluated before the command is parsed or executed.)
  yes? CONTOUR/Z=`temp[X=180,Y=0,Z=@LOC:15]` salt

- A list of values (constant array) may be formed by enclosing values in curly brackets. For example in a function call:
  yes? LET aday = DAYS1900(1989,{3,6,9},1)

- Text for labels is enclosed in double quotes
  yes? VECTOR/TITLE="title_string" x_expr, y_expr
  If the string is to contain a quote, the backslash preserves it:
  yes? GO my_go_script "\"(-10,10,2)"\"
  sends the string ***"(-10,10,2)"*** to the script (see p. 18) for more on go scripts

Ch1 Sec5. GO FILES

GO files are files containing Ferret commands. They can be executed with the command "GO filename". Throughout this manual, these files are referred to as GO scripts or journal files (the file names end in *.jnl). There are two kinds of GO files provided with the distribution (differing in function, not form)—demos and tools. A list of the demonstrations and scripts can be found in Ferret's on-line documentation in "on-line demonstrations".

Ch1 Sec5.1. Demonstration files

Demonstration GO files provide examples of various Ferret capabilities (the tutorial is such a script) . The demonstration GO files may be executed simply by typing the Ferret command

```
yes? GO demo_name
```
example: `yes? GO vector_demo`

Below is a list of the demo files provided as of 4/99 (located in directory $FER_DIR/examples). The Unix command "Fgo demo" will list all GO scripts containing the string "demo". Use Fgo '*' to see all the scripts that are currently available on your system.

Name	Description
tutorial	brief tour through Ferret capabilities
bar_chart_demo	plotting bar charts
binary_read_demo	binary file reading (version 5.0 and after)
coads_demo	view of global climate using the Comprehensive Ocean-Atmosphere Data Set
constant_array_demo	shows {3,5,6} constant-array syntax
custom_contour_demo	customized contour plots
depth_to_density_demo	contour with a user-defined variable as an axis
dods_demo	using DODS to access remote datasets
edit_data_file_demo	"hand-editing" variables using netCDFdatasets and SAVE
ef_eof_demo	EOF functions
ef_fft_demo	FFT functions
ef_sort_demo	using the SORT and SAMPLE functions
ef_wv5d_demo	writing Vis5D-formatted files
error_bars_demo	making error bars on plots
file_reading_demo	reading an ASCII file

Name	Description
fnoc_demo	Naval Fleet Numerical Oceanography Center data
levitus_demo	T-S relationships using Sydney Levitus' climatological Atlas of the World Oceans
log_plot_demo	log plots using PPLUS in Ferret
mathematics_demo	abstract function calculation
mercator_demo	mercator map projection
minmax_label_demo	use FINDLO and FINDHI to label extrema on a plot
mp_demo	map projections
mp_stereo_demo	fancy map projection techniques
multi_line_labels_demo	many-line titles and other labels
multi_variable_demo	multiple variables with multiple dependent axes
objective_analysis_demo	interpolating scattered data to grids
overlay_on_time_axis_demo	PLOT/VS and POLYGON over a time axis
palette_demo	shows uses of various palettes
pattern_demo	patterns on shade and fill plots
plot_swath_demo	fill between line plots for "swaths" of color
plot_vectors	draw vectors from u,v,lat,lon
poly_vec_demo	use filled polygons to plot vector fields
polymark_demo	show use of polymark script
polytube_demo	"lagrangian" plots along a path using color fill
regridding_demo	tutorial on regridding data
sigma_coordinate_demo	how to work with sigma coordinates
spirograph_demo	for-fun plots from abstract functions
splash_demo	for-fun mathematical color shaded plots
statistics_demo	probability distributions
symbol_demo	how to use symbols for plot layouts
taylor_example1	using scripts to make Taylor diagrams
topographic_relief_demo	global topography
trackplot_demo	use of trackplot.jnl script
vector_demo	vector plots
viewports_demo	output to viewports
wire_frame_demo	3D wire frame representation

Ch1 Sec5.2. GO tools

GO tools are scripts which contain Ferret commands and perform dataset-independent tasks. For example, "GO land" overlays the outline of the continents on your plot. (Note: In order for Ferret to locate the GO scripts, the environment variable FER_GO must be properly defined. See the chapter "Computing Environment," p. 257, for guidance.)

To run any GO tool, from the Ferret command line, type,

```
Yes? GO scriptname
```

Or if the script has arguments, they follow the script name with optional comma separators.

```
yes? GO script2 arg1, arg2
```

To find out about the script, use the /HELP qualifier, which opens the script with the more command to type the first 20 lines of the script and allow you to see the documentation at the start of the script.

```
yes? GO/HELP scriptname
```

To omit arguments from a GO script,

```
yes? GO script arg1, , arg3
```

Or double quotes with a space to indicate the missing item.

```
yes? GO script arg1 "  " arg3
```

The Unix command Fgo has been provided to assist with locating tools within the Unix directory hierarchy. For example,

```
% Fgo grid   displays all tools with the substring "grid" in their names
% Fgo '*'    displays all GO tools and demonstrations
```

When passing arguments to GO commands sometimes it is necessary to pass enclosing quotation marks. An common example is the passing of the argument to the CONTOUR/LEVELS qualifier in cases such as

```
CONTOUR/LEVELS="(-100) (-10,10,2) (100)" my_var
```

where there may be blanks embeddd inside of the string. There are 3 methods to embed quotations inside of strings

1. use "\" to protect the quotation marks in the GO command line

```
yes? go my_go_script "\"(-100) (-10,10,2) (100)"\"
```

with the script containing the line

```
CONTOUR/LEVELS=$1 my_var
```

2. use "\" to define a symbol which contains the quotation marks

```
yes? DEFINE my_quoted_string \"$1\"
yes? CONTOUR/LEVELS=($my_quoted_string) my_var
```

3. use the symbol substitution syntax to add quotes to theGO argument

```
Yes? CONTOUR/LEVELS=$1&|*>"*"&
```

Of course, in the above examples one could also simply use

```
yes? CONTOUR/LEVELS="$1" my_var
```

Below is a table of the tools provided with your Ferret installation. Some tools accept optional arguments to control details. Use ***Fgo -more script_name*** for details on a script.

Tool name	Description
OVERLAYS	
basemap	a geographical basemap of continents to overlay on
land	overlays continental boundaries (color controls)
land_detailed	overlays detailed continents, national and state boundaries, rivers
bold_land	overlays darker continental boundaries
fland	overlays filled continents (color and resolution controls)
focean	overlays ocean mask (for terrestrial plots)

Tool name	Description
multi_xaxis_overlay	Overlay a line plot over an existing one, with a new horizontal axis
multi_yaxis_overlay	Overlay a line plot over an existing one, with a new vertical axis
graticule	sets the plot axis style to use a graticule (rather than tics) (See also the /GRATICULE qualifier on all plot commands, and MODE GRATICULE)
tics	resets the plot style to use axis tics (rather than a graticule)
gridxy	overlays a "graticule" at the I,J grid locations (see also the /GRAT qualifier for plot commands, and MODE GRATICULE)
gridxz	overlays a "graticule" at the I,K grid locations
gridxt	overlays a "graticule" at the I,L grid locations
gridyz	overlays a "graticule" at the J,K grid locations
gridyt	overlays a "graticule" at the J,L grid locations
gridzt	overlays a "graticule" at the K,L grid locations
box	draws a box at the specified location on the plot
ellipse	draws an ellipse at the specified location on the plot
MATHEMATICAL	
frequency_histogram	makes a frequency distribution plot (histogram) of data
ts_frequency	creates a 2-variable histogram (typically an oceanographer's TS density diagram)
polar	defines R and THETA from X and Y to perform (limited) polar plots
regressx	defines variables for linear regression along X axis
regressy	defines variables for linear regression along Y axis
regressz	defines variables for linear regression along Z axis
regresst	defines variables for linear regression along T axis
unit_square	sets unit square as default for abstract variables
variance	defines variables to compute variances and covariances
var_n	refines TVARIANCE with corrected n/n+1 factors
dynamic_height	defines Ferret variables for dynamic height calculations
SAMPLE DISPLAYS	
line_samples	draws specimens of the available line styles

Tool name	Description
line_thickness	draws examples of pen color/thickness styles in PPLUS
fill_samples	draws specimens of the available fill styles
show_symbols	draws specimens of the default symbols
show_88_syms	draws specimens of all 88 PPLUS symbols
GRAPHICS	
bar_chart	makes a color-filled bar chart from a line of data
bar_chart2	makes a bar chart using hollow rectangles
centered_vectors	makes a vector plot with coords at vector midpoints
scattered_vectors	makes a vector plot from an ASCII file: x,y,u,v
stick_vectors	makes a stick vector plot of a line of U,V values
extremum	annotate contour extrema on a plot
split_z	oceanographic-style plot with 2 z-axis scalings
taylor_example1	demonstrates tools for making Taylor diagrams
PLOT APPEARANCE	
margins	tweak the sizing of the plot on the page
magnify [factor]	increases the data plotting area (area inside the axes)
unmagnify	restores the plot origin and axis lengths to default values
black	sets video background to black, foreground to white
white	sets video background to white, foreground to black
bold	sets up PLOT+ and Ferret to produce bolder-looking plots
unbold	resets plot environment to normal after "GO bold"
unlabel [label #]	removes a specified (numbered) PPLUS movable label
remove_logo	removes labels 1–3 that form the Ferret logo
box_plot	produces a plot with "bare" axes (no tics, no labels)
portrait	set window for 8.5 x 11 portrait page
portrait1x2	set window for 8.5 x 11 portrait page and two viewports
portrait1x3	set window for 8.5 x 11 portrait page and three viewports
portrait1x4	set window for 8.5 x 11 portrait page and four viewports
portraitNxN	set window for 8.5 x 11 portrait page and NxN viewports
reminder	place small annotations in upper left corner of plot

Tool name	Description
COLOR	
try_palette [pal]	displays palette appearance for various numbers of color levels
try_centered_palette	displays centered palette appearance for various numbers of levels
exact_colors	sets up Ferret and PPLUS to modify individual colors in a color palette
squeeze_colors	modifies a color palette by squeezing and stretching the color scale
MULTIPLE X AND Y AXES (run demo: ***yes? GO multi_variable_plots***)	
left_axis_plot	plots a single variable preparing for a 2nd axis on the right
right_axis_plot	overlays a plot of a single variable using an axis on the right
multi_xaxis_plot1	draws a plot formatted for later overlays using multiple X axes
multi_xaxis_overlay	overlays a variable with a distinct X axis
multi_yaxis_plot1	draws a plot formatted for later overlays using multiple Y axes
multi_yaxis_overlay	overlays a variable with a distinct Y axis
MAP PROJECTIONS (run demo: ***yes? GO mp_demo***)	
mp_~name~	individual projections include bonne, craster_parabolic, eckert_greifendorff, eckert_iii, eckert_v, hammer, lambert_cyl, mcbryde_fpp, mercator, orthographic, plate_caree, polyconic, sinusoidal, stereographic_eq, stereographic_north, stereographic_south, vertical_perspective, wagner_vii, winkel_i
mp_aspect	set the appropriate window aspect ratio for this map projection
mp_fland	overlays "map projected" filled continents (color controls)
mp_graticule	overlays "map projected" graticule (color controls)
mp_grid.jnl	Associates a data grid with a predefined map projection.
mp_label	plots a label using world coordinates
mp_land	overlays "map projected" continental boundaries (color controls)
mp_land_stripmap	creates a land-centric, interrupted "stripmap" using the current map projection
mp_line	overlays "map projected" plotted data
mp_ocean_stripmap	creates an ocean-centric, interrupted "stripmap" using the current map projection

Tool name	Description
mp_polymark	overlays "map projected" polygons
mp_polymark	Plot polygons using a predefined map projection.
mp_polytube	Plot a colored tube using a predefined map projection.
mp_trackplot	Plot a trackplot using a predefined map projection
mp_viewport_aspect	Define a viewport for plotting map projections

SAMPLING A GRIDDED FIELD

bullseye	locate a bullseye in a 2-D field
digitize	obtain data values from a plot using the cursor
vertical_section	create 2-D vertical section from a 3-D field
samplexy_demo	create 2-D vertical section along any path

UTILITY SCRIPTS

datestring.jnl	create date string from year, month, day, etc

TESTS

test	tests proper functioning of FER_GO
ptest	produces a quick test plot
squares	creates a filled-area test plot

Ch1 Sec5.3. Writing GO tools

A GO tool ("GO script," "journal file," ...) is simply a sequence of Ferret commands stored in a file and executed with the GO command. Writing a simple GO tool requires nothing more than typing normal commands into a file.

To write a robust GO tool that may be shared, however, certain guidelines should be followed:

1) the GO tool should be well documented
2) the GO tool should leave the Ferret context unmodified

3) the GO tool may need to run "silently"
4) the GO tool may need to accept arguments (a maximum of 99 parameters)

Ch1 Sec5.3.1. Documenting GO tools

Documentation consists primarily of well-chosen comment lines (lines beginning with an exclamation mark). In addition, a line of this form should be included:

```
! Description:  [one-line summary of your GO tool]
```

This line is displayed by the Fgo tool.

Ch1 Sec5.3.2. Preserving the Ferret state in GO tools

Often a complex GO tool requires setting data sets, modifying the current region, etc. But to a user executing this tool its behavior may seem erratic if the user's previous context is modified by running the tool. A tool can restore the previous state of Ferret by these means:

region: Save the current default region with the command DEFINE REGION/DEFAULT save. Restore it at the end of your GO tool with SET REGION save.

data set: Save the current default data set with SET DATA/SAVE. Restore it at the end of your GO tool with SET DATA/RESTORE.

grid: Save the current default grid set with SET GRID/SAVE. Restore it at the end of your GO tool with SET GRID/RESTORE.

modes: If you modify a mode inside your GO tool by issuing a SET MODE or a CANCEL MODE command the original state of that mode can be restored using SET MODE/LAST.

Ch1 Sec5.3.3. Silent GO tools

If a user has set mode "verify" then by default every line of your GO tool, including comment lines, will be displayed at the screen as Ferret processes it. To make your GO tool run silently include the command CANCEL MODE VERIFY at the beginning of the GO tool and SET MODE/LAST VERIFY at the end. If the backslash character "\" is found at the beginning of any line that single line will not be displayed regardless of the state of MODE VERIFY. Thus the command "\CANCEL MODE VERIFY" is often the first line of a GO tool. Note also that the command LET/SILENT is useful in GO tools which need to define variables.

Ch1 Sec5.3.4. Arguments to GO tools

Arguments (parameters) may be passed to GO tools on the command line. There is an upper limit of 99 arguments allowed. For example,

```
yes? GO land red
```

passes the string "red" into the GO file named land.jnl. Inside the GO tool the argument string "red" is substituted for the string "$1" wherever it occurs. The "1" signifies that this is the first argument—similar logic can be applied to $1,... $99 or $0 where $0 is replaced by the name of the GO tool itself. "$*" is replaced by all the arguments as a single string, separated by spaces.

If there are more than 9 arguments, the syntax $nn (nn may be 1 through 99) is equivalent to to ($nn), however the parentheses enclosed form is generally preferred as it avoids ambiguities. Specifying $12.dat is equivalent to ($12).dat but is less clear.

As Ferret performs the substitution of $1 (or other) arguments it offers a number of string processing and error processing options. For example, without these options, if a user failed to supply an argument to "GO land" then Ferret would not know what to substitute for $1 and it would have to issue an error message. A default value can be supplied by the GO tool writer using the syntax

```
$1%string%
```

for example,

```
$1%black%
```

inside land.jnl would default to "black" if no color were specified. Note that in the example percent signs were used to delimit the default string but any of the characters ! # $ % or & also work as delimiters.

If the argument is a 2-digit number, and we are making a substitution, the replacement text goes inside the parentheses. For example, plot the variable passed as argument 1 with the color given by argument 12, or green if no argument 12 is given:

```
PLOT/COLOR=($12#green#) $1
```

In another case it might not be appropriate to supply a default string but instead it would be desirable to issue an instructional error message. The "<" character indicates an error message text:

```
$1"<you must supply an argument to this GO tool"
```

In still other cases there are a range of acceptable arguments but all other arguments are illegal. The allowable arguments can be specified following "|" (vertical bar) characters as in this example:

```
$1"|black|red|<You must specify black or red"
```

or a default of "black" could be specified together with the options as

```
$1"black|black|red|"
```

In the interest of "friendliness" a GO file may want to allow the user to specify a string other than the string actually needed by the GO tool. For example, in older Ferret versions red plot line was actually obtained by the PLOT command qualifier /LINE=2—the string "red" never appeared in this command. To allow a user to specify "red" and yet have the string "2" substituted, Ferret has provided the replacement arrow ">". Thus

```
$1"1|red>2|"
```

specifies a default string of "1" if no argument is given but substitutes "2" if "red" is supplied. In a typical GO tool line, defaults, options, substitutions, and an error message are combined like this:

```
PLOT/LINE=$1"1|red>2|green>3|blue>4|<must be red, green, or blue"
```

Note that the error message will be issued only if some color other than "red," "green," or "blue" is specified; if no argument is specified then "1" is substituted.

An asterisk (*) can be used to designate that any text whatsoever is acceptable as an option.

```
PLOT/LINE=$1"1|red>2|green>3|blue>4|*>7"
```

would never generate an error and would use line style 7 (thick black) if an unrecognized argument string such as "orange" were given.

An asterisk (*) can also be used on the right-hand side of a substitution, in which case it stands for the entire original argument string. For example

```
SET VARIABLE/TITLE=$1%*>"*"%
```

will place double quotation marks around the string in argument 1.

Ch1 Sec5.3.5. Documentation and checking arguments to GO tools

A final style note to keep in mind when writing GO tools that use arguments: providing error message feedback and appropriate documentation for the user is essential. In complex GO tools, all arguments should be checked at the beginning of the GO tool using the no-op command (has no effect) "QUERY/IGNORE". Thus the GO tool land.jnl might contain these lines at the beginning:

```
! check the argument
QUERY/IGNORE $1"1|red|green|blue|<must be red, green, or blue"
```

Once argument errors have been trapped and reported, the lengthy error text would not be needed again in the GO tool.

GO tools that use arguments should also be carefully documented. There are numerous examples provided with Ferret; try, for example, the Unix commands

```
% Fgo -more fland.jnl
% Fgo -more stick_vectors
```

or

```
% Fgo -more squeeze_colors
```

Ch1 Sec5.3.6. Flow Control in GO tools

There are several Ferret commands and techniques to assist with flow control in your GO scripts.

GO (subroutines)

The GO command may be used inside of a GO script (tool) to execute another (nested) GO script. If an error occurs inside of a nested GO script and SET MODE IGNORE_ERROR has not been issued then the GO script will be interrupted and control returns to the command line.

REPEAT (looping)

The REPEAT command may be used to execute loops within Ferret. The loop "counter" may be an index (I,J,K, or L) or a world coordinate (longitude, latitude, depth, or time). The increment between loop iterations need not correspond to the spacing of points on a grid. When used in conjunction with the "d" options of SET REGION, such as SET REGION/DI="-5:-5" the loops may be used to zoom in or out of a region or to pan a limited-width window of view across a larger region. See the Advanced Movie-Making section (p. 177) of this manual for further details.

IF-THEN-ELSE (conditional execution)

An IF-THEN-ELSE syntax can be used to conditionally execute Ferret commands. It may be used in two styles—single line and multi-line. See the IF command (p. 360) in the Commands Reference section of this manual for further details.

Ch1 Sec5.3.7. Debugging GO tools

As the complexity of Ferret GO scripts increases it becomes more challenging to locate and correct errors in GO scripts. This is especially true if, as so many GO scripts do, the scripts are

made silent by containing the command CANCEL MODE VERIFY. In a silent script it can be unclear from where within the script an error message is originating.

A special VERIFY mode has been provided to assist with locating the source of these error messages

```
SET MODE VERIFY:ALWAYS
```

The ALWAYS argument to this command instructs Ferret to ignore CANCEL MODE VERIFY commands inside of command files. All of the script commands that Ferret executes will be echoed when this mode is set. Error messages will appear with the commands that generated them. To restore normal non-debugging operations issue CANCEL MODE VERIFY or SET MODE VERIFY (no argument) interactively from the ***yes?*** prompt.

Complex webs of variable definitions (defined with LET or DEFINE VARIABLE) may also create challenges for debugging scripts. See Debugging Complex Hierarchies of Expressions (p. 144) for further discussion of this topic.

Ch1 Sec6. SAMPLE DATA SETS

A number of demonstration data sets are included with this distribution. Several of these data sets are used by the demonstration "GO" files, above. The data sets should be accessible simply by typing the Ferret command

```
yes? USE data_set_name
yes? USE coads_climatology
```
for example,

Data set	Description
etopo120	relief of the earth's surface at 120-minute resolution
etopo60	relief of the earth's surface at 60-minute resolution
levitus_climatology	subset of the Climatological Atlas of the World Oceans by Sydney Levitus (Note: the updated World Ocean Atlas, 1994, is also available with Ferret)
coads_climatology	12-month climatology derived from 1946–1989 of the Comprehensive Ocean/Atmosphere Data Set
monthly_navy_winds	monthly-averaged Naval Fleet Numerical Oceanography Center global marine winds (1982–1990)
esku_heat_budget	Esbensen-Kushnir 4×5 degree monthly climatology of the global ocean heat budget (25 variables)

Ch1 Sec7. UNIX TOOLS

A number of tools are provided with Ferret to assist with Unix-level activities: on-line help, converting data to Ferret's formats, locating files, etc. They are located in the Ferret installation area—typically $FER_DIR/bin. See the chapter "Copmuting Environment", section "Setting up to run Ferret" (p. 257), if the tools are not available on-line. They are described below.

Faddpath Usage: Faddpath new_path

Faddpath will add a new path name to the default lists of directories that Ferret searches a) in response to the SET DATA command; b) when looking for grid definition files; c) when looking for data files.

Fapropos Usage: Fapropos string (i.e. **% *Fapropos regridding***)

Fapropos searches the Ferret User's Guide for all occurrences of the given word or string. The string is not case sensitive. If the string contains multiple words it must be enclosed in quotation marks. Fapropos will list all lines of the User's Guide that contain the word or string and report their line numbers. The line numbers may be used with Fhelp to enter the User's Guide at the desired location.

Fdata Usage: Fdata data_file_substring

Searches the list of directories contained in the environment variable FER_DATA to find the data files whose names contain the indicated substring. For example,

```
% Fdata coads
```

locates the data files containing "coads" in their names. (Use this command to locate netCDFdata sets by giving the string "cdf".)

Fdescr Usage: Fdescr des_name_substring

Searches the list of directories contained in the environment variable FER_DESCR to find the descriptor files whose names contain the indicated substring. For example,

```
% Fdescr coads
```

locates the descriptor files containing "coads" in their names. ("Fdescr .des" will list all accessible descriptors.)

Fenv Usage: Fenv

Prints the values of environment variables used by Ferret

Fgo Usage: Fgo name_substring

Searches the list of directories contained in the environment variable FER_GO to find the GO command files whose names contain the indicated substring. For example,

```
% Fgo grid
```

locates the Ferret tools that contain "grid".

Fgrids Usage: Fgrids gridfile_substring

Searches the list of directories contained in the environment variable FER_GRIDS to find the grid definition files whose names contain the indicated substring. For example,

```
% Fgrids fnoc
```

locates the grid definition files containing "fnoc" in their names. ("***Fgrids .grd"*** will list all accessible grid files.)

Fpalette Usage: Fpalette name_substring

Searches the list of directories contained in the environment variable FER_PALETTE to find the palette files whose names contain the indicated substring. For example,

```
% Fpalette blue
```

locates the palette files containing "blue" in their names.

Fpurge Usage: Fpurge filename_template

Fpurge is a support routine to manage multiple versions of files created by Ferret—particularly journal files and graphic metafiles. Fpurge will remove all versions of a file except the current version. For example, "Fpurge ferret.jnl" will eliminate all past versions of ferret.jnl in the current directory.

Fsort Usage: Fsort filename_template

Fsort is a support routine for sorting file versions. Fsort reorders the incorrect ordering of emacs-style version numbers assigned by the Unix "ls" utility. For example, when sorting, ls will place filename.~19~ before filename.~2~. "Fsort filename*" will take care of this problem. Fsort may be used in Unix pipes.

Ch1 Sec8. HELP

Ch1 Sec8.1. Examples and demonstrations

As discussed earlier in this chapter (Getting Started, GO files), the demonstrations that come with the Ferret distribution are a source of help. See the introductory chapter, section "Demonstration files," (p. 16) for a list of demonstrations, or look in $FER_DIR/examples; you may find something that addresses your problem.

Ch1 Sec8.2. Help from within Ferret

Typing "help" while running Ferret will give you information on using the Unix tool Fhelp to access the User's Guide.

The Ferret command SHOW COMMANDS will list all Ferret commands; SHOW COMMAND "command" will display all qualifiers for the specified command.

The Ferret command SHOW FUNCTIONS lists all Ferret functions and their arguemnts. SHOW FUNCTION *string* will show all functions containing the string "string". SHOW FUNCTIONS EXTERNAL shows the names and arguments of external functions (see External Functions Chapter, page 293)

The Ferret command SHOW TRANSFORMS lists all Ferret transforms, including variable transforms and regridding transforms.

If you want to get details on a script, type 'GO/HELP scriptname" to see the documentation at the start of the script. For example:

```
GO/HELP land
```

When writing scripts, include documentation listing the purpose of the script and its arguments in the first few lines of the script. Then this feature will let you and others who may use the script get instant information about it.

Ch1 Sec8.3. Web-based information

From the Ferret web page, at http://www.ferret.noaa.gov/Ferret, see these sections:

1. Ferret support policy outlines the support available to users and sources of information

2. FAQ section discusses many topics where questions often arise.

3. Email archives, which are searchable and contain questions and solutions from the Ferret users group.

4. Documentation section, including release notes, this manual which is updated regularly on the web, and on-line information on demonstration scripts, data formats, and the Plot Plus graphics used by Ferret.

Chapter 2: DATA SET BASICS

Ch2 Sec1. OVERVIEW

Ferret accepts input data from both ASCII and binary files and recognizes two standardized, self-describing data formats—NetCDF, and TMAP. Network Common Data Format (NetCDF) is the suggested method of data storage.

SET DATA_SET or just SET DATA specifies a data set for access. ASCII and binary files can be read using SET DATA/EZ (also known as "FILE"). To unambiguously specify the format of a data set, include the extension .cdf or .des in its name, or use the qualifier /FORMAT=CDF.

To examine what each data set consists of (variables, grids, etc.) after specifying them with SET DATA, use SHOW DATA. This command displays the variables in the data set and over what geographical and time ranges they are defined.

Here is an example of Ferret's output:

```
       yes? SET DATA coads_climatology
       yes? SHOW DATA
      currently SET data sets:
     1> /home/e1/tmap/fer_dsets/descr/coads_climatology.des  (default)
name      title                          I         J         K         L
SST      SEA SURFACE TEMPERATURE      1:180     1:90      1:1       1:12
AIRT     AIR TEMPERATURE              1:180     1:90      1:1       1:12
SPEH     SPECIFIC HUMIDITY            1:180     1:90      1:1       1:12
WSPD     WIND SPEED                   1:180     1:90      1:1       1:12
UWND     ZONAL WIND                   1:180     1:90      1:1       1:12
VWND     MERIDIONAL WIND              1:180     1:90      1:1       1:12
SLP      SEA LEVEL PRESSURE           1:180     1:90      1:1       1:12
```

If multiple data sets have been requested in a single Ferret session, the last requested will be the default data set. To specify other data sets, use the name of the data set or the number of the set as given by the SHOW DATA statement. For example:

```
yes? LIST/D=2  temp
```

will list the data for the variable "temp" in data set number 2 as displayed by SHOW DATA/BRIEF, while

```
yes? LIST temp[D=levitus_climatology] - temp[D=coads_climatology]
```

will list the differences between the variable "temp" in data set "levitus_climatology" and data set "coads_climatology."

Once a data set has been opened, you can find the data set name via the RETURN keyword (see p. 138):

```
yes? say `var,RETURN=dset`
yes? say `var,RETURN=dsetnum`
```

If a filename begins with a number, Ferret does not recoginze it, but the file may be specified using its unix pathname, e.g.

```
yes? use "./123"
```

or

```
yes? file/var=a "./45N_180W.dat"
```

Ch2 Sec2. NETCDF DATA

The Network Common Data Format (NetCDF) is an interface to a library of data access routines for storing and retrieving scientific data. netCDFallows the creation of data sets which are self-describing and platform-independent. netCDFwas created under contract with the Division of Atmospheric Sciences of the National Scientific Foundation and is available from the Unidata Program Center in Boulder, Colorado (www.unidata.ucar.edu).

See the chapter "Converting Data to NetCDF" (p. 267), for a complete description of how to create netCDFdata sets or how to convert existing data sets into NetCDF.

To output a variable in NetCDF, simply use:

```
yes?  LIST/FORMAT=CDF variable_name
```

LIST/FORMAT=CDF (alias SAVE) can also be used with abstract variables:

```
yes? SAVE/FILE=example.cdf/I=1:100 sin(I/100)
```

This will create a file named example.cdf.

The current region and data sets determine the variable names in the saved file and the range over which they are saved. Saved data can then be accessed as follows:

```
yes? USE example
```

(USE is an alias for SET DATA/FORMAT=CDF, see)

To read a netCDFdataset that is on a DODS server, simply specify the DODS address in quotes:

```
yes? use
"http://www.ferret.noaa.gov/cgi-bin/nph-nc/data/coads_climatology.nc"
```

If a filename is not specified, Ferret will generate one. (See command SET LIST/FILE in the Commands Reference section, p. 407). An example of converting TMAP-formatted data to netCDFgoes as follows:

```
yes? SET DATA coads_climatology
yes? SAVE/L=1 sst,airt,uwnd,vwnd
```

These commands will save sst, airt, uwnd, and vwnd at the first time step over their entire regions to a netCDF file named by Ferret.

One advantage to using netCDF is that users on a different system (i.e., VMS instead of Unix) with different software (i.e., with an analysis tool other than Ferret) can share data easily without substantial conversion work. netCDF files are self-describing; with a simple command the size, shape and description of all variables, grids and axes can be seen.

Ch2 Sec2.1. NetCDF data and strides

Beginning with Ferret version 5.1 , the internal functioning of netCD Freads has been changed when "strides" are involved. Suppose that CDFVAR represent a variable from netCDFfile. In version 5.0 and earlier the command PLOT CDFVAR[L=1:1000:10] would have read the entire array of 1000 points from the file; Ferret's internal logic would have subsampled every 10th point from the resulting array in a manner that was consistent for netCDF variables, ASCII variables, user defined variables, etc. In V5.1 strides applied to netCDF variables are given special treatment -- subsampling is done by the netCDF library. The primary benefit of this is to make network access to remote data sets via DODS more efficient. Beginning with Ferret v5.4, strides can be applied across the "branch point" of a modulo variable without loss of efficiency for netCDF data set, as long as the stride is an integer fraction of the modulo length times the number of points on the axis. A remote satellite image of size, say, 1000x1000 points x 8 bit depth (8 megabytes) can efficiently be previewed using

```
SHADE DODS_VAR[i=1:1000:10,j=1:1000:10]
```

If a grid or axis from a netCDF file is used in the definition of a LET-defined variable (e.g. LET my_X = X[g=sst[D=coads_climatology]]) that variable definition will be invalidated when the data set is canceled (CANCEL DATA coads_climtology, in the preceding example). There is a single exception to this behavior: netCDF files such as climtological_axes.cdf, which define grids or axes that are not actually used by any variables. These grids and axes will remain defined even after the data set, itself, has been canceled. They may be deleted with explicit use of CANCEL GRID or CANCEL AXIS.

In Ferret version 6.02 we introduce a method whereby a grid may be redefined with strided axes. This "native stride" syntax means that the stride information needs to be specified only once, and variable names do not need to be changed.

Old syntax:

```
yes? SET DATA mydat.nc
yes? LET strided_var = var[i=1:1000:10,j=1:1000:10]
yes? FILL strided_var   ! Use the new name strided_var everywhere.
```

New syntax:

```
yes? SET DATA mydat.nc
yes? SET AXIS/STRIDE=10 `var,RETURN=xaxis`
yes? SET AXIS/STRIDE=10 `var,RETURN=yaxis`
yes? FILL var  ! The original variable name can be used
```

An offset may be specified on the SET AXIS/STRIDE command with SET AXIS/STRIDE=/OFFSET=. The offset value must be less than the stride value, and it refers to the first index to use:

Old syntax

```
yes? SET DATA mydat.nc
yes? LET strided_var = var[i=4:1000:10]
```

New syntax:

```
yes? SET DATA mydat.nc
yes? SET AXIS/STRIDE=10/OFFSET=4 `var,RETURN=xaxis`
```

This syntax associates a new strided axis with the original axis. Everywhere that original axis is used, the new strided behavior will be applied. This means that all variables from all datasets that share the same exact axis will appear on the new strided axis. The original axis definition still exists and we can cancel the stride behavior with

```
yes? CANCEL AXIS/STRIDE axisname
```

Ch2 Sec2.2. NetCDF data attributes

Beginning with Ferret V6.0, Ferret has access to attributes of netCDF variables, including coordinate variables. In fact, attributes can be defined and used for user variables and variables from any kind of dataset. See the section in the next chapter about dataset and variable attributes (p. 65)

Ch2 Sec2.3. NetCDF Data with the bounds attribute

The CF standard for netCDF files defines a bounds attribute for coordinate axes, where the upper and lower bounds of the grid cells along an axis are specified by a bounds variable which is of size n*2 for an axis of length N. See Section 7.1 of the CF document

http://www.cgd.ucar.edu/cms/eaton/cf-metadata/CF-1.0.html

The coordinates on the axis may be anywhere within the cells defined by the upper and lower cell bounds. Ferret uses these as the upper and lower bounds of of axis cells (also known as boxes). They may be listed or otherwise accessed using the pseudo-variables XBOXLO, XBHOXH, YBOXLO, etc.

For example, a simple netCDF file with bounds would have the following ncdump output:

```
netcdf irrx {
dimensions:
      XAX = 4 ;
      bnds = 2 ;
variables:
      double XAX(XAX) ;
            XAX:point_spacing = "uneven" ;
            XAX:axis = "X" ;
            XAX:bounds = "XAX_bnds" ;
      double XAX_bnds(XAX, bnds) ;
      float V(XAX) ;
            V:missing_value = -1.e+34f ;
            V:_FillValue = -1.e+34f ;
            V:long_name = "SEA SURFACE TEMPERATURE" ;

// global attributes:
            :history = "FERRET V5.60    4-Jun-04" ;
data:

 XAX = 1, 2, 5, 6 ;

 XAX_bnds =
  0., 1.5,
  1.5, 2.5,
  2.5, 5.5,
  5.5, 7. ;

 V =
  28.20222,
  28.36456,  28.35381,0
  28.2165,
  28.48889,
  28.31556 ;
}
```

The CF standard allows for axes in a file that may have discontiguous bounds (the upper bound of one cell is not the same as the lower bound of the next cell). Ferret does not allow such an axis. When discontiguous bounds are encountered in a file, we arbitrarily choose to use the lower bounds throughout, with the upper bound of the topmost cell to close the definition. This way all axes have contiguous upper and lower bounds. A warning message is issued.

DEFINE AXIS/BOUNDS may be used to create an axis with cell bounds. All irregular axes are saved with a bounds attribute (beginning with Ferret v5.70) and the user may request that all axes be written with the bounds attribute with the SAVE/BOUNDS command

Note that if you have a dataset that has an irregular time axis and a bounds attribute on that axis and you force Ferret to apply a regular time axis with

yes? USE/REGULART my_data.nc

then the bounds are ignored: the regular time axis is formed from the first and last coordinate and the number of points.

Ch2 Sec2.4. Multi-file NetCDF data sets

Ferret supports collections of netCDF files that are regarded as a single netCDF data set. Such data sets are referred to as "MC" (multi CDF) data sets. They are particularly useful to manage the outputs of numerical models. MC data sets use a descriptor file, in the style of TMAP-formatted data sets. The data set is referred to inside Ferret by the name of this descriptor file.

A collection of netCDF files is suitable to form a multi-file data set if

1) The files are connected through their time axis—each file represents one or more time snapshots of the variables it contains.

2) All non-time-dependent variables in the data set must be contained in the first file of the data set (or those variables will not appear in the merged, MC, data set).

Note that previous to version 5.2, each file is self-documenting with respect to the time axis of the variables—even if the time axis represents only a single point. (All of the time axes must be identically encoded with respect to units and date of the time origin.) In version 5.3 and higher these checks are not performed. This means that the MC descriptor mechanism can be used to associate into time series groups of files that are not internally self-documenting with respect to time. See Chapter 10, section 4 (p. 289)

Beginning with version 5.8 of Ferret the stepfiles may contain different scale and offset values for the variables they contain. (p. 275). Ferret reads and applies the scale and offset values as data from each stepfile is read. Note that the commands

```
yes? SAY `var, RETURN=nc_offset`
yes? SAY `var, RETURN=nc_scale`
```

return the latest scale and offset value that were applied.

A typical MC descriptor file may be found in the chapter "Converting to netCDF", in the section "Creating a multi-NetCDF data set." (p. 289)

Ch2 Sec2.5. Non-standard NetCDF data sets

As discussed in the Chapter, "Converting Data to NetCDF," (p. 267) Ferret expects netCDF files to adhere to the COARDS conventions (http://www.ferret.noaa.gov/noaa_coop/coop_cdf_profile.html). If the files do not adhere to the COARDS conventions, Ferret will still attempt to access them. Often, the user can use Ferret controls for regridding, reshaping, and otherwise transforming data to recover the intended file contents.

Here are a few common ways in which netCDF files may deviate from the COARDS standard and how one may cope with those situations in Ferret.

- Files with disordered coordinates

  In the COARDS conventions an axis (a.k.a. "coordinate variable") must have monotonically-increasing coordinate values. If the coordinates are disordered or repeating in a netCDF file, then Ferret will present the coordinates to the user (in SHOW DATA) as a dependent variable, whose name is the axis name, and it will substitute an axis of the index values 1, 2, 3, ... Note that Ferret will apply this same behavior when files have long irregular axis definitions that exceed Ferret's axis memory capacity.

- Files with reverse-ordered axes

  If the coordinates of an axis are monotonically decreasing, instead of increasing, Ferret will transparently reverse both the axis coordinates and the dependent variables that are defined upon that axis. Note that if Ferret writes a reverse-ordered variable to a new netCDF file (with the SAVE command), the coordinates and data in the output file will be in monotonically increasing coordinate order—reversed from the input file.

  If the values of a dependent variable are reversed, but there is no associated coordinate axis then use attach a minus sign to the corresponding axis orientation in the USE/ORDER= qualifier to designate that the variable(s) should be reversed along the corresponding axis.

- Files with "invalid" variable names

  The COARDS standard specifies that variable names should begin with a letter and be composed of letters, digits, and underscores. In files where the variable names contain other letters, references to those variable names in Ferret must be enclosed in single quotes.

- Files with permuted axis ordering

  The COARDS standard specifies that if any or all of the dimensions of a variable have the interpretations of "date or time" (a.k.a. "T"), "height or depth" (a.k.a. "Z"), "latitude" (a.k.a. "Y"), or "longitude" (a.k.a. "X") then those dimensions should appear in the relative order T, then Z, then Y, then X in the CDL definition corresponding to the file. In files where the axis ordering has been permuted the command qualifiers USE/ORDER= (Command Reference, p. 399) allow the user to inform Ferret of the correct permutation of coordinates. Note

that if Ferret writes a permuted variable to a new netCDF file (with the SAVE command), the coordinates and data in the output file will be in standard X-Y-Z-T ordering (as indicated in the user's /ORDER specification)—permuted from the original file ordering. See the Command Reference (p. 323) for a complete description of the ORDER qualifier.

- Files with more than four dimensions

  The COARDS standard specifies that a netCDF file may be created with more than four dimensions. However the Ferret framework allows just four dimensions at this time.

Ch2 Sec2.6. NetCDF and non-standard calendars

The netCDF conventions document discusses and defines usage for different calendar axes. hese conventions for calendars are implemented in Ferret version 5.3 See:

http://www.cgd.ucar.edu/cms/eaton/cf-metadata/CF-current.html#cal The calendars allowed are:

GREGORIAN or STANDARD (default)	Ferret uses the proleptic Gregorian calendar, which is the Gregorian calendar extended to dates before 1582-10-15.
NOLEAP or 365_DAY	All years are 365 days long.
NOLEAP or 365_DAY	All years are 365 days long.
ALL_LEAP or 366_DAYNL	All years are 366 days long.
360_DAY	All years are 360 days divided into 30 day months.
JULIAN	Julian calendar; leap years with no adjustment at the turn of the century.

These calendars are compatible with the Udunits standard which has slightly different naming conventions, except that the gregorian or standard calendar is a proleptic Gregorian calendar in Ferret. If the mixed Julian/Gregorian calendar is desired, use a time origin of 1-jan-0001:00:00 and Ferret will apply the 2-day shift that was made historically when the Gregorian calendar was introduced. The Udunits standard can be found at:

http://my.unidata.ucar.edu/content/software/udunits/udunits.txt

udunits_1998.dat (A local copy of the above link).

The netCDF conventions recommend that the calendar be specified by the attribute time:calendar which is assigned to the time coordinate variable when there is a non-Gregorian calendar associated with a data set, i.e.

time:calendar=noleap

Ferret reads this attribute when it is present in a netCDF file and assigns the appropriate calendar identifer to the variable. When a variable has a non-Gregorian calendar, the attribute is written to a netCDF file when the variable is output to a netCDF file.

Ch2 Sec3. TMAP-FORMATTED DATA

As of Ferret version 2.30, netCDF is the suggested format for data storage (see the chapter, "Converting to netCDF," p. 267). This section describing TMAP information is included only for users who already work with data in TMAP format.

To access TMAP-formatted data sets use

```
SET DATA_SET TMAP_set1, TMAP_set2, ...
```

TMAP_setn must be the name of a descriptor file for a data set that is in TMAP "GT" (grids-at-timesteps) or "TS" (time series) format. ("Ferret" format and "TMAP" format are synonyms.)

If the directory portion of the filename is omitted the environment variable FER_DESCR will be used to provide a list of directories to search. The order of directories in FER_DESCR determines the order of directory searches. If the extension is omitted a default of ".des" will be assumed (if the filename has more than one period, the extension must be given explicitly).

Descriptors

For every TMAP-formatted data set there is a descriptor file containing summary information about the contents of the data set. This includes variable names, units, grids, and coordinates. When the command SET DATA_SET is given to Ferret pointing to a GT-formatted or TS-formatted data set, it is the name of the descriptor file that must be specified.

Ch2 Sec4. BINARY DATA

Ferret can read binary data files that are formatted with and without FORTRAN record length headers (binary files without FORTRAN record length formatting are also known as "stream" files).

Ch2 Sec4.1. FORTRAN-structured binary files

Files containing record length information are created by FORTRAN programs using the ACCESS="SEQUENTIAL" (the FORTRAN default) mode of file creation and also by Ferret using LIST/FORMAT=unf. Files that contain FORTRAN record length headers must have all data aligned on a 4-byte boundary. Suppose "rrrr" represents 4 bytes of record length information and "dddd" represents a 4-byte data value. Then FORTRAN-structured files are organized in one of the following two ways:

Ch2 Sec4.1.1. Records of uniform length

A FORTRAN-structured file with records of uniform length (3 single-precision floating point data values per record in this figure) looks like this:

```
rrrr dddd dddd dddd rrrr ...
```

FORTRAN code that creates a data file of this type might look something like this (sequential access is the default and need not be specified in the OPEN statement):

```
REAL VARI(10), VAR2(10), VAR3(10)
...
OPEN(UNIT=20,FORMAT='UNFORMATTED',ACCESS='SEQUENTIAL',FILE='MYFILE.DAT')
...
DO 10 I=1,10
WRITE (20) VAR1(I), VAR2(I), VAR3(I)
10 CONTINUE
....
```

To access data from this file, use

```
yes? SET DATA/EZ/FORMAT=UNF/VAR=var1,var2,var3/COL=3  myfile.dat     or,
yes? FILE/FORMAT=UNF/VAR=var1,var2,var3/COLUMNS=3  myfile.dat
```

This is very similar to accessing ASCII data with the addition of the /FORMAT=unf qualifier. The /COLUMNS= qualifier tells Ferret the number of data values per record. Although optional in the above example, this qualifier is required if the number of data values per record is greater than the number of variables being read (examples follow in section "ASCII Data").

Ch2 Sec4.1.2. Records of non-uniform length

A FORTRAN-structured file with variable-length records might look like this:

```
rrrr dddd dddd rrrr
rrrr dddd rrrr
rrrr dddd dddd dddd dddd rrrr
etc.
```

With care, it is possible to read a data file containing variable-length records which was created using the simplest unformatted FORTRAN OPEN statement and a single WRITE statement for each variable. Use /FORMAT=stream to read such files. Note that sequential access is the FORTRAN default and does not need to be specified in the OPEN statement:

```
REAL VAR1(1000), VAR2(500)
...
OPEN (UNIT=20, FORMAT="UNFORMATTED", FILE="MYFILE.DAT")
...
WRITE (20) VAR1
WRITE (20) VAR2
....
```

Use the qualifier /SKIP to skip past the record length information (/SKIP arguments are in units of words), and define a grid which will not read past the data values. The /COLUMNS= qualifier can be used when reading multiple variables to specify the number of words separating the start of each variable:

```
yes? DEFINE AXIS/X=1:500:1  xaxis
yes? DEFINE GRID/X=XAXIS  mygrid
yes? FILE/FORMAT=stream/SKIP=1003/GRID=mygrid/VAR=var2  myfile.dat
```

The argument 1003 is the sum of the 1000 data words in record 1, plus 2 words of record length information surrounding the data values in record 1 (variable var1), plus 1 word of record information preceding the data in record 2.

Ch2 Sec4.1.3. Fortran binary files, variables on different grids.

Some FORTRAN-structured files have multiple variables per record which do not share a common grid. An example would be one year of a global monthly field stored as twelve records like this:

```
rrrr year month field(360x180) rrrr
```

The data file size is (1+1+1+360*180+1)*12*4 = 3110592 bytes. Such a file cannot be read with the /FORMAT=unf qualifier but can be read with the /FORMAT=stream qualifier described in the next section. By including the /SWAP qualifier, this technique can be used to read files created on a machine with a different byte ordering.

The following commands will read this file and assign the data to the appropriate grid:

```
yes? ! Create an X axis for an entire record.
yes? DEFINE AXIS/X=1:`3+360*180+1`:1 binary_x
yes? DEFINE AXIS/T=1:12:1 binary_t
yes? DEFINE GRID/X=binary_x/T=binary_t binary_g

yes? ! Read in everything.
yes? FILE/FORMAT=stream/G=binary_g/VAR=val binary_file
```

```
! Create the grid for the data field.
yes? DEFINE AXIS/MODULO/X=0.5:359.5:1 1deg_x
yes? DEFINE AXIS/Y=-89.5:89.5:1 1deg_y
yes? DEFINE AXIS/T=15-jan-1999:15-dec-1999:1/UNITS=month month_1999_t
yes? DEFINE GRID/X=1deg_x/Y=1deg_y/T=month_1999_t 1deg_1999_g

yes? ! Create a variable that uses this grid.
yes? LET dummy = x[GX=R_1deg_1999_g] + y[GY=R_1deg_1999_g] +
t[GT=R_1deg_1999_g]

yes? ! Reshape the data portion of val onto the data grid.
yes? LET field = RESHAPE(val[i=4:`3+360*180`],dummy)
```

Ch2 Sec4.2. Stream binary files

Files without embedded record length information are created by FORTRAN programs using ACCESS="DIRECT" in OPEN statements and by C programs using the C studio library. These files can contain a mix of integer and real numbers. The following types can be read from an unstructured file:

FORTRAN	C	Size in bytes
INTEGER*1	char	1
INTEGER*2	short	2
INTEGER*4	int	4
REAL*4	float	4
REAL*8	double	8

Ch2 Sec4.2.1. Simple stream files

Suppose "dddd" represents a 4-byte data value. Then a stream (or "direct access") binary file of FORTRAN REAL*4 or C floats is:

```
dddd dddd dddd dddd dddd dddd ...
```

The structure of the records is implied by the program accessing the data. FORTRAN code which generates a direct access binary file might look like this:

```
REAL*4 MYVAR(10,5)
...
C Use RECL=40 for machines that specify in bytes

OPEN(UNIT=20, FILE="myfile.dat", ACCESS="DIRECT", RECL=10)
...
DO 100 j = 1, 5
100   WRITE (20,REC=j) (MYVAR(i,j),i=1,10)
....
```

Use the following Ferret commands to read variable "myvar" from this file:

```
yes? DEFINE AXIS/X=1:10:1 x10
yes? DEFINE AXIS/Y=1:5:1 y5
yes? DEFINE GRID/X=x10/Y=y5 g10x5
yes? FILE/VAR=MYVAR/GRID=g10x5/FORMAT=stream  myfile.dat
```

If the file consisted of a set of FORTRAN REAL*8 or C doubles, then the data would look like:

```
dddddddd dddddddd dddddddd ...
```

and the following Ferret commands would read the data into "myvar":

```
yes? DEFINE AXIS/X=1:10:1 x10
yes? DEFINE AXIS/Y=1:5:1 y5
yes? DEFINE GRID/X=x10/Y=y5 g10x5
yes? FILE/VAR=MYVAR/GRID=g10x5/FORMAT=stream/type=r8  myfile.dat
```

Note the addition of the "type" qualifier. See the FILE command (p. 357) for more details.

Since Ferret represents all variables as REAL*4, some precision is lost when reading in REAL*8 or INTEGER*4 values. Also, some REAL*8 numbers cannot be represented as REAL*4 numbers; the internal Ferret value of such a number is system dependent.

Ch2 Sec4.2.2. Mixed stream files

Ferret can read binary files that contain a mix of numbers of different type. However, a given Ferret variable can only be one type. Say you have a file containing a mix of REAL*8 and REAL*4 numbers:

```
dddddddd dddd dddddddd dddd dddddddd ...
```

The following would successfully read the file:

```
yes? FILE/VAR=MYDOUBLE,MYFLOAT/GRID=somegrid/FORMAT=stream/type=r8,r4
myfile.dat
```

while:

```
yes? FILE/VAR=MYDOUBLE/GRID=someothergrid/FORMAT=stream/type=r8,r4
myfile.dat
```

would fail.

Ch2 Sec4.2.3. Byte-swapped stream files

Stream files with byte-swapped numbers can be read with the /SWAP qualifier. Note that the /ORDER and /SKIP qualifiers are also available (see chapter "Data Set Basics", section "Reading ASCII files," p. 46, for more details on /ORDER and /SKIP).

Ch2 Sec5. ASCII DATA

To access ASCII data file sets use

```
yes? SET DATA/EZ   ASCII_file_name    or equivalently
yes? FILE   ASCII_file_name
```

The following are qualifiers to SET DATA/EZ or FILE:

Qualifier	Description
/VARIABLES	names the variables in the file
/TITLE	associates a title with the data set
/GRID	indicates multi-dimensional data and units
/COLUMNS	tells how many data values are in each record
/FORMAT	specifies the format of the records
/SKIP	skips initial records of the file
/ORDER	specifies order of axes (which varies fastest)

Use command SET VARIABLE to individually customize the variables.

Ch2 Sec5.1. Reading ASCII files

Below are several examples of reading ASCII data properly. (Uniform record length, FORTRAN-structured binary data are read similarly with the addition of the qualifier /FORMAT= "unf". Seethe chapter on "Data Set Basics", section "Binary Data," p. 41, for other binary types). First, we look briefly at the relationship between Ferret and standard matrix notation.

Linear algebra uses established conventions in matrix notation. In a matrix A(i,j), the first index denotes a (horizontal) row and the second denotes a (vertical) column.

A11	A12	A13	...	A1n	
A21	A22	A23	...	A2n	Matrix A(i,j)
...					
Am1	Am2	Am3	...	Amn	

X-Y graphs follow established conventions as well, which are that X is the horizontal axis (and in a geographical context, the longitude axis) and increases to the right, and Y is the vertical axis (latitude) and increases upward (Ferret provides the /DEPTH qualifier to explicitly designate axes where the vertical axis convention is reversed).

In Ferret, the first index of a matrix, i, is associated with the first index of an (x,y) pair, x. Likewise, j corresponds to y. Element Am2, for example, corresponds graphically to x=m and y=2.

By default, Ferret stores data in the same manner as FORTRAN—the first index varies fastest. Use the qualifier /ORDER to alter this behavior. The following examples demonstrate how Ferret handles matrices.

Example 1—1 variable, 1 dimension

1a) Consider a data set containing the height of a plant at regular time intervals, listed in a single column:

```
2.3
3.1
4.5
5.6
. . .
```

To access, name, and plot this variable properly, use the commands

```
yes? FILE/VAR=height plant.dat
yes? PLOT height
```

1b) Now consider the same data, except listed in four columns:

```
2.3   3.1   4.5   5.6
5.7   5.9   6.1   7.2
. . .
```

Because there are more values per record (4) than variables (1), use:

```
yes? FILE/VAR=height/COLUMNS=4 plant4.dat
yes? PLOT height
```

Example 2—1 variable, 1 dimension, with a large number of data points.

The simple FILE command:

```
yes? FILE/VAR=height plant.dat
```

uses an abstract axis of fixed length, 20480 points. If your data is larger than that, you can read the data by defining an axis of appropriate length. Set the length to a number equal to or larger than the dimension of your data. The plot command will plot the actual number of points in the file.

```
yes? DEFINE AXIS/X/X=1:50000:1 longax
yes? DEFINE GRID/X=longax  biggrid
yes? FILE/VAR=height/GRID=biggrid plant.dat
yes? PLOT height
```

Example 3—2 variables, 1 dimension

3a) Consider a data set containing the height of a plant and the amount of water given to the plant, measured at regular time intervals:

```
2.3 20.4
3.1 31.2
4.5 15.7
5.6 17.3
. . .
```

To read and plot this data use

```
yes? FILE/VAR="height,water" plant_wat.dat
yes? PLOT height,water
```

3b) The number of columns need be specified only if the number of columns exceeds the number of variables. If the data are in six columns

```
2.3 20.4   3.1 31.2   4.5 15.7
5.6 17.3 ...
```

use

```
yes? FILE/VAR="height,water"/COLUMNS=6 plant_wat6.dat
yes? PLOT height,water
```

Example 4—1 variable, 2 dimensions

4a) Consider a different situation: a greenhouse with three rows of four plants and a file with a single column of data representing the height of each plant at a single time (successive values represent plants in a row of the greenhouse):

```
3.1
2.6
5.4
4.6
3.5
6.1
. . .
```

If we want to produce a contour plot of height as a function of position in the greenhouse, axes will have to be defined:

```
yes? DEFINE AXIS/X=1:4:1 xplants
yes? DEFINE AXIS/Y=1:3:1 yplants
yes? DEFINE GRID/X=xplants/Y=yplants gplants
yes? FILE/VAR=height/GRID=gplants greenhouse_plants.dat
yes? CONTOUR height
```

When reading data the first index, x, varies fastest. Schematically, the data will be assigned as follows:

```
        x=1        x=2         x=3        x=4
y=1     3.1        2.6         5.4        4.6
y=2     3.5        6.1 . . .
y=3    . . .
```

4b) If the file in the above example has, instead, 4 values per record:

```
3.1   2.6   5.4   4.6
3.5   6.1  . . .
```

then add /COLUMNS=4 to the FILE command:

```
yes? FILE/VAR=height/COLUMNS=4/GRID=gplants greenhouse_plants.dat
```

Example 5—2 variables, 2 dimensions

Like Example 3, consider a greenhouse with three rows of four plants each and a data set with the height of each plant and the length of its longest leaf:

```
3.1     0.54
2.6     0.37
5.4     0.66
4.6     0.71
3.5     0.14
6.1     0.95
.        .
.        .
```

Again, axes and a grid must be defined:

```
yes? DEFINE AXIS/X=1:4:1 xht_leaf
yes? DEFINE AXIS/Y=1:3:1 Yht_leaf
yes? DEFINE GRID/X=xht_leaf/Y=yht_leaf ght_leaf
yes? FILE/VAR="height,leaf"/GRID=ght_leaf greenhouse_ht_lf.dat
yes? SHADE height
yes? CONTOUR/OVER leaf
```

The above commands create a color-shaded plot of height in the greenhouse, and overlay a contour plot of leaf length. Schematically, the data will be assigned as follows:

```
        x=1          x=2          x=3          x=4
      ht ,  lf     ht ,  lf
y=1   3.1, 0.54    2.6, 0.37    5.4, 0.66    4.6, 0.71
y=2   3.5, 0.14    6.1, 0.95 . . .
y=3   . . .
```

Example 6—2 variables, 3 dimensions (time series)

Consider the same greenhouse with height and leaf length data taken at twelve different times. The following commands will create a three-dimensional grid and a plot of the height and leaf length versus time for a specific plant.

```
yes? DEFINE AXIS/X=1:4:1 xplnt_tm
yes? DEFINE AXIS/Y=1:3:1 yplnt_tm
yes? DEFINE AXIS/T=1:12:1 tplnt_tm
yes? DEFINE GRID/X=xplnt_tm/Y=yplnt_tm/T=tplnt_tm gplant2
yes? FILE/VAR="height,leaf"/GRID=gplant2 green_time.dat
yes? PLOT/X=3/Y=2 height, leaf
```

Example 7—1 variable, 3 dimensions, permuted order (vertical profile)

Consider a collection of oceanographic measurements made to a depth of 1000 meters. Suppose that the data file contains only a single variable, salt. Each record contains a vertical profile (11 values) of a particular x,y (long,lat) position. Supposing that successive records are successive longitudes, the data file would look as follows (assume the equivalencies are not in the file):

```
                z=0   z=10  z=20  . . .
x=30W,y=5S  35.89 35.90 35.93 35.97 36.02 36.05 35.96 35.40 35.13 34.89
34.72
x=29W,y=5S  35.89 35.91 35.94 35.97 36.01 36.04 35.94 35.39 35.13 34.90
34.72
            . . .
```

Use the qualifier /DEPTH= when defining the Z axis to indicate positive downward, and /ORDER when setting the data set to properly read in the permuted data:

```
yes? DEFINE AXIS/X=30W:25W:1/UNIT=degrees salx
yes? DEFINE AXIS/Y=5S:5N:1/UNIT=degrees saly
yes? DEFINE AXIS/Z=0:1000:100/UNIT=meters/DEPTH salz
yes? DEFINE GRID/X=salx/Y=saly/Z=salz salgrid
yes? FILE/ORDER=zxy/GRID=salgrid/VAR=sal/COL=11 sal.dat
```

Ch2 Sec5.2. Reading "DELIMITED" data files

SET DATA/FORMAT=DELIMITED[/DELIMITERS=][/TYPE=][/VAR=] filename

For "delimited" files, such as output of spreadsheets, SET DATA/FORMAT=DELIMITED initializes files of mixed numerical, string, and date fields. If the data types are not specified the file is analyzed automatically to determine data types.

The alias COLUMNS stands for "SET DATA/FORMAT=DELIMITED". (See p.402 for the full syntax.)

Example 1: Strings, latitudes, longitudes, and numeric data.

This file is delimited by commas. Some entries are null; they are indicated by two commas with no space between. File delimited_read_1.dat contains:

```
col1,  col2   col3   col4  col5  col6  col7
one    ,,     1.1,   24S,  130E  ,,    1e1
two    ,,     2.2,   24N,  130W, 2S
three  ,,     3.3,    24,   130, 3N,   3e-2

five   ,,     4.4,   -24,  -130, 91,   -4e2
extra line
```

If there is no /TYPE qualifier, the data type is automatically determined. If all entries in the column match a data type they are assigned that type. First let's try the file as is, using automatic

analysis. Record 1 contains 5 column headings (text) so V1 through V5 are analyzed as text variables.

```
yes? FILE/FORMAT=delim delimited_read_1.dat
yes? LIST v1,v2,v3,v4,v5,v6,v7,v8
            DATA SET: ./delimited_read_1.dat
            X: 0.5 to 7.5
 Column  1: V1
 Column  2: V2
 Column  3: V3
 Column  4: V4
 Column  5: V5
 Column  6: V6
 Column  7: V7
             V1           V2     V3     V4     V5   V6     V7
1   / 1: "col1"       "col2" "col3" "col4" "col5" " "    ....
2   / 2: "one"        " "    "1.1"  "24S"  "130E" " "    10.0
3   / 3: "two"        " "    "2.2"  "24N"  "130W" "2S"   ....
4   / 4: "three"      " "    "3.3"  "24"   "130"  "3N"    0.0
5   / 5: " "          " "    " "    " "    " "    " "    ....
6   / 6: "five"       " "    "4.4"  "-24"  "-130" "91" -400.0
7   / 7: "extra line" " "    " "    " "    " "    " "    ....
```

Now skip the first record to do a better "analysis" of the file fields. Explicitly name the variables. Note that v3 is correctly analyzed as numeric, A4 is latitude and A5 longitude. A6 is analyzed as string data, because the value 91 in record 5 does not fall in the range for latitudes, and records 2 and 3 contain mixed numbers and letters.

```
yes? FILE/FORMAT=DELIM/SKIP=1/VAR="a1,a2,a3,a4,a5,a6,a7,a8,a9"
delimited_read_ 1.dat
yes? LIST a1,a2,a3,a4,a5,a6,a7
            DATA SET: ./delimited_read_1.dat
            X: 0.5 to 6.5
 Column  1: A1
 Column  2: A2 is A2 (all values missing)
 Column  3: A3
 Column  4: A4 is A4 (degrees_north)(Latitude)
 Column  5: A5 is A5 (degrees_east)(Longitude)
 Column  6: A6
 Column  7: A7
              A1    A2     A3     A4     A5   A6     A7
1   / 1: "one"       ...  1.100 -24.00  130.0 " "    10.0
2   / 2: "two"       ...  2.200  24.00 -130.0 "2S"   ....
3   / 3: "three"     ...  3.300  24.00  130.0 "3N"    0.0
4   / 4: " "         ...   ....   ....   .... " "    ....
5   / 5: "five"      ...  4.400 -24.00 -130.0 "91" -400.0
6   / 6: "extra line"...   ....   ....   .... " "    ....
```

Now use the /TYPE qualifier to specify that all columns be treated as numeric.

```
yes? FILE/FORMAT=delim/SKIP=1/TYPE=numeric delimited_read_1.dat
yes? LIST v1,v2,v3,v4,v5,v6,v7,v8
            DATA SET: ./delimited_read_1.dat
            X: 0.5 to 6.5
 Column  1: V1
 Column  2: V2
 Column  3: V3
 Column  4: V4
 Column  5: V5
```

```
 Column  6: V6
 Column  7: V7
        V1  V2    V3     V4     V5     V6     V7
1   / 1:.....  1.100   ....   ....   ....   10.0
2   / 2:.....  2.200   ....   ....   ....   ....
3   / 3:.....  3.300  24.00  130.0   ....    0.0
4   / 4:.....   ....   ....   ....   ....   ....
5   / 5:.....  4.400 -24.00 -130.0  91.00 -400.0
6   / 6:.....   ....   ....   ....   ....   ....
```

Here is how to read only the first line of the file. If the variables are not specified, 7 variables are generated because auto-analysis of file doesn't stop at the first record. Use the command COLUMNS, the alias for FILE/FORMAT=delimited

```
yes? DEFINE AXIS/X=1:1:1 x1yes? DEFINE GRID/X=x1 g1
yes? COLUMNS/GRID=g1 delimited_read_1.dat
 LIST v1,v2,v3,v4,v5,v6,v7
             DATA SET: ./delimited_read_1.dat
             X: 1
 Column  1: V1
 Column  2: V2
 Column  3: V3
 Column  4: V4
 Column  5: V5
 Column  6: V6
 Column  7: V7
             V1     V2     V3     V4     V5  V6  V7
I / *:    "col1" "col2" "col3" "col4" "col5" " " ... " "
```

Define the variables to read.

```
yes? COLUMNS/GRID=g1/VAR="c1,c2,c3,c4,c5" delimited_read_1.dat
yes? LIST c1,c2,c3,c4,c5
             DATA SET: ./delimited_read_1.dat
             X: 1
 Column  1: C1
 Column  2: C2
 Column  3: C3
 Column  4: C4
 Column  5: C5
             C1     C2     C3     C4     C5
I / *:    "col1" "col2" "col3" "col4" "col5"
```

Example 2: File using blank as a delimiter.

Ferret recognizes the file as containing date and time variables, further explored in Example 3 below. Here is the file delimited_read_2.dat. There is a record of many blanks in record 2.

```
1981/12/03     12:35:00

1895/2/6 13:45:05
```

Read the file using /DELIMITER=" "

```
yes? FILE/FORM=delimited/DELIMITER=" " delimited_read_2.dat
yes? LIST v1,v2
```

```
            DATA SET: ./delimited_read_2.dat
            X: 0.5 to 3.5
 Column  1: V1 is V1 (days)(Julian days since 1-Jan-1900)
 Column  2: V2 is V2 (hours)(Time of day)
              V1      V2
1   / 1:  37965.  12.58
2   / 2:    ....   ....
3   / 3:  39051.  13.75
```

Example 3: dates and times

Note that record 3 has syntax errors in the first 4 fields. Here is delimited_read_3.dat:

```
   12/1/99,  12:00,      12/1/99, 1999-03-01, 12:00,      13:45:36.5
   12/2/99,  01:00:13.5, 12/2/99, 1999-03-02, 01:00:13.5, 14:45:36.5
   12/3/99x, 2:00x,      12/3/99, 1999-03-03, 2:00,       15:45
   12/4/99,  03:00,      12/4/99, 1999-03-04, 03:00,      16:45:36.5
```

Read with auto-analysis. The records with syntax errors cause variables 1 and 2 to be read as string variables.

```
yes? COLUMNS delimited_read_3.dat
yes? LIST v1,v2,v3,v4,v5,v6
            DATA SET: ./delimited_read_3.dat
            X: 0.5 to 4.5
 Column  1: V1
 Column  2: V2
 Column  3: V3 is V3 (days)(Julian days since 1-Jan-1900)
 Column  4: V4 is V4 (days)(Julian days since 1-Jan-1900)
 Column  5: V5 is V5 (hours)(Time of day)
 Column  6: V6 is V6 (hours)(Time of day)
               V1          V2       V3      V4     V5     V6
1   / 1: "12/1/99"  "12:00"       36493.  36218.  12.00  13.76
2   / 2: "12/2/99"  "01:00:13.5"  36494.  36219.   1.00  14.76
3   / 3: "12/3/99x" "2:00x"       36495.  36220.   2.00  15.75
4   / 4: "12/4/99"  "03:00"       36496.  36221.   3.00  16.76
```

Use the date variables in v3 and v4 to define time axes. The date encodings are as expected.

```
yes? DEFINE AXIS/T/UNITS=days/T0=1-jan-1900  tax = v3
yes? SHOW AXIS tax
 name       axis              # pts   start                end
 TAX       TIME                 4 r   01-DEC-1999 00:00    04-DEC-1999
00:00

T0 = 1-JAN-1900

yes? DEFINE AXIS/T/UNITS=days/T0=1-jan-1900  tax = v4
yes? SHOW AXIS tax
 name       axis              # pts   start                end
 TAX       TIME                 4 r   01-MAR-1999 00:00    04-MAR-1999
00:00
T0 = 1-JAN-1900
```

Next we'll specify each column's type. Only the first two characters of the type are needed. Now we can read those columns which had errors, except for the record with the errors.

```
yes? COLUMNS/TYPE="da,ti,date, date, time, time"  delimited_read_3.dat
yes? LIST v1,v2,v3,v4,v5,v6
             DATA SET: ./delimited_read_3.dat
             X: 0.5 to 4.5
 Column  1: V1 is V1 (days)(Julian days since 1-Jan-1900)
 Column  2: V2 is V2 (hours)(Time of day)
 Column  3: V3 is V3 (days)(Julian days since 1-Jan-1900)
 Column  4: V4 is V4 (days)(Julian days since 1-Jan-1900)
 Column  5: V5 is V5 (hours)(Time of day)
 Column  6: V6 is V6 (hours)(Time of day)
              V1      V2      V3      V4      V5      V6
1   / 1:  36493.  12.00  36493.  36218.  12.00  13.76
2   / 2:  36494.   1.00  36494.  36219.   1.00  14.76
3   / 3:    ....   ....  36495.  36220.   2.00  15.75
4   / 4:  36496.   3.00  36496.  36221.   3.00  16.76
```

Delimiters can be used to break up individual fields. Use both the slash and a comma (indicated by backslash and comma \,)

```
FILE/FORM=delim/DELIM="/,\," delimited_read_3.dat
LIST V1,V2,V3,V4,v5,v6
             DATA SET: ./delimited_read_3.dat
             X: 0.5 to 4.5
 Column  1: V1
 Column  2: V2
 Column  3: V3
 Column  4: V4
 Column  5: V5
 Column  6: V6
              V1      V2    V3            V4      V5      V6
1   / 1:  12.00  1.000 "99"  "12:00"        12.00  1.000
2   / 2:  12.00  2.000 "99"  "01:00:13.5"   12.00  2.000
3   / 3:  12.00  3.000 "99x" "2:00x"        12.00  3.000
4   / 4:  12.00  4.000 "99"  "03:00"        12.00  4.000
```

Ch2 Sec6. TRICKS TO READING BINARY AND ASCII FILES

Since binary and ASCII files are found in a bewildering variety of non-standardized formats a few tricks may help with reading difficult cases.

- Sometimes variables are interleaved with data axes in unstructured (stream) binary files. A simple trick is to read them all as a single variable, say, "Vall," in which the sequence of variables in the file V1, V2, V3, ... is regarded as an axis of the grid. Then extract the variables by defining V1 = Vall[I=1] (if the I axis was used, else J=1, K=1, or L=1) as needed.

- In some ASCII files the variables are presented as blocks—a full grid of variable 1, then a full grid of variable 2, etc. These files may be read using Unix soft links so that the same file can be opened as several Ferret data sets. Then use the FILE command to point separately to each soft link using the /SKIP qualifier to locate the correct starting point in the file for each variable. For example,

  Unix commands:

```
ln -s my_data my_dat.v1
ln -s my_data my_dat.v2
ln -s my_data my_dat.v3
```

Ferret commands:

```
yes? FILE/SKIP=0/VAR=v1 my_dat.v1
yes? FILE/SKIP=100/VAR=v2 my_dat.v2
yes? FILE/SKIP=200/VAR=v3 my_dat.v3
```

- If an ASCII file contains a repeating sequence of records try describing the entire sequence using a single FORTRAN FORMAT statement. An example of such a statement would be (3F8.4,2(/5F6.2)). The slash character and the nested parentheses allow multi-record groups to appear as a single format. Note that the /COLUMNS qualifier should reflect the total number of columns in the repeating group of records.

- If an ASCII or binary file contains gridded data in which the order of axes is not X-Y-Z-T read the data in (which results in the wrong axis ordering) and use the LIST/ORDER= to permute the order on output. The resulting file will have the desired axis ordering.

- If the times and geographical coordinate locations of the grid are inter-mixed with the dependent variables in the file then 1) issue a FILE command to read the coordinates only; 2) use DEFINE AXIS/FROM_DATA to define axes and DEFINE GRID to define the grid; 3) use FILE/GRID=mygrid to read the file again.

Ch2 Sec7. ACCESS TO REMOTE DATA SETS WITH DODS

Ch2 Sec7.1. What is DODS?

DODS is now called OPeNDAP; we continue to refer to it as DODS in this manual for now. DODS, the Distributed Oceanographic Data System, allows users to access data anywhere from the Internet using a variety of client/server methods, including Ferret. Employing technology similar to that used by the World Wide Web, DODS and Ferret create a powerful tool for the retrieval, sampling, analyzing and displaying of datasets; regardless of size or data format (though there are data format limitations).

For more information, please see the OPeNDAPhome page at

http://www.opendap.org/

Similar to the WWW, DODS is an emerging technology and is under development. As a result, it is likely that the details with which things are accomplished will be changing.

Ch2 Sec7.2. Accessing Remote Data Sets

Datasets are accessed through Ferret using their raw Universal Resource Locator (URL) address. Make sure to enclose the address in quotes, as for any dataset name that includes a path. For example, to access the COADS climatology, hosted at PMEL:

```
yes? use
"http://www.ferret.noaa.gov/cgi-bin/nph-nc/data/coads_climatology.nc"
```

Once the dataset has been initialized, it is used just like any other local dataset.

```
yes? list/x=140w/y=2n/t="16-Feb" sst
            SEA SURFACE TEMPERATURE (Deg C)
            LONGITUDE: 141W
            LATITUDE: 1N
            TIME: 15-FEB 16:29
            DATA SET:
http://www.ferret.noaa.gov/cgi-bin/nph-nc/data/coads_climatology.nc
         26.39
```

To locate DODS data, you can search the NVODS /DODS List of DODS datasets at http://www.opendap.org/data/datasets.cgi?&exfunction=none&xmlfilename=datasets.xml or the Global Change Master Directory at http://gcmd.gsfc.nasa.gov/

For the time being, netCDF and HDF files can be read via DODS by Ferret. As DODS (OPeNDAP) netCDF libraries become available, other data types will be made available.

Note that HDF files can be read by Ferret only via DODS, that is the HDF file must first be put on a DODS server and then Ferret can access it by giving its DODS URL. Even by this means, Ferret will be successful in reading the file only if the HDF file is similar in its structure to a COARDS or CF netCDF file. Often, you will need to apply the USE/ORDER= qualifier to change the ordering of the coordinate axes.

If a file is on a DODS server, you can look at the DAS in your browser (the URL that ends in .das). When you look at the attribute data check to see if there are dimension variables with attributes that look like a Latitude or Longitude as in the COARDS conventions.

Ch2 Sec7.3. Debugging Access to Remote DODS Data Sets

To find out more information about a particular dataset, or to debug problems, there are three elements of the dataset which may be accessed via a web browser. To access this information, merely append a dds, das, or info to the dataset name. For example:

```
http://www.ferret.noaa.gov/cgi-bin/nph-nc/data/coads_climatology.nc.dds
```

DDS stands for Data Description Structure and this will return a text description of the data sets structure.

```
http://www.ferret.noaa.gov/cgi-bin/nph-nc/data/coads_climatology.nc.das
```

DAS stands for Dataset Attribute Structure and this will return a text description of attributes assigned to the variables in the data set.

```
http://www.ferret.noaa.gov/cgi-bin/nph-nc/data/coads_climatology.nc.info
```

This will return a text description of the variables in the dataset.

Ch2 Sec7.4. Security

Some DODS data providers will choose to control access to some or all of their data. When you request data from one of these servers, the DODS client will prompt you for a username and password. If you want to avoid the prompt, you can embed a username and password in it, like this:

```
http://user:password@www.dods.org/nph-dods/etc...
```

Ch2 Sec7.5. Sharing Data Sets via DODS

One of the most powerful aspect of DODS is the ease with which it allows for the sharing of data. With just a few simple steps, anyone running a web server can also be a DODS data server, thereby allowing data set access to anyone with an Internet connection.

Simply copying a few precompiled binaries into the cgi-bin directory of an already configure httpd server is all it takes to become a DODS server. Once the server is configured, adding or removing data sets is as simple as copying them to the server data directory or deleting them from that directory.

This ability has such immense potential that it bears extra emphasis. Imagine that within seconds of finishing a model run, a remote colleague is able to look at your dataset with whatever DODS client he/she desires, be it Ferret, or Matlab, etc. No need for you to package up the data or for your colleague to download and/or reformat it, it is ready to be analyzed right away.

Ch2 Sec7.6. DODS caching

This feature allows caching of frequently accessed DODS served datasets to produce a quicker response when requesting remote data. The first time you access a DODS data set, a file in the users home directory will be created called .dodsrc. This file is the DODS client initialization file. Please see the DODS Users Guide; http://www.opendap.org/user/guide-html/guide.html/guide_72.html for details of the paramaters that this file contains. Initially, DODS caching will be turned off. In order to turn

caching on, change the line in the newly created ~/.dodsrc file from

```
USE_CACHE=0
```

to

```
USE_CACHE=1
```

Note that if you edit the .dodsrc file, make sure that the lines within it all start in the first column.

The next time Ferret is run, and a DODS-served dataset is accessed, a file called .dods_cache will be created, typically in the users home directory. The location of the DODS cache directory can be controlled by the line

```
CACHE_ROOT=/home/twaits/.dods_cache/
```

in the user's .dodsrc file. This directory is where all the cached information is stored. To clear the DODS cache, simply delete the .dods_cache directory and all of it's contents (for example, rm -rf ~/.dods_cache). This directory will be recreated and repopulated with caching information the next time data is accessed via DODS, if caching is turned on. All of the paramater values in the .dodsrc file can be modified to better suit individual needs, and will be incorporated the next time Ferret is run and DODS served data is accessed. Again, see the DODS User guide at see the section "The OPeNDAP Client Initialization File (.dodsrc)" in the DODS Users Guide (http://www.opendap.org/user/guide-html/guide.html) for more detailed information

It is often a useful diagnostic exercise to turn caching off and/or clear out the cache directory when attempts to access datasets in Ferret appear inconsistent. For example, if Ferret attempted to access a DODS-served dataset that was unavailable because the DODS server was down, that information may get cached and adversely effect the next attempt at retrieving the data.

For more detailed information on using DODS, and on setting up a DODS server, see the DODS home page (http://unidata.ucar.edu/packages/dods).

Ch2 Sec7.7. Proxy servers

A DODS client can negotiate proxy servers, with help from directions in its configuration file. The parameters that control proxy behavior are fully documented in the DODS Users Guide, see the link above.

Chapter 3: VARIABLES AND EXPRESSIONS

Ch3 Sec1. VARIABLES

Variables are of 2 kinds:

1) file variables (read from disk files)
2) user-defined variables (defined by the user with LET command)

Both types may be accessed identically in all commands and expressions.

Variables, regardless of kind, possess the following associated information:

1) grid—the underlying coordinate structure
2) units
3) title
4) title modifier (additional explanation of variable)
5) flag value for missing data points

Use the commands SHOW DATA and SHOW VARIABLES to examine file variables and user-defined variables, respectively.

The pseudo-variables I, J, K, L, X, Y, Z, T and others may be used to refer to the underlying grid locations and characteristics and to create abstract variables.

For a description of string variables and arrays, see the chapter on "Handling String Data", p. 229.

Ch3 Sec1.1. Variable syntax

Variables in Ferret are referred to by names with optional qualifying information appended in square brackets. See DEFINE VARIABLE (p. 349) for a discussion of legal variable names.

The information that may be included in the square brackets includes

```
D=data_set_name_or_number        ! indicate the data set
G=grid_or_variable_name          ! request a regridding
X=,Y=,Z=,T=,I=,J=,K=,L=          ! specify region and transformation
                                   e.g. LIST V[x=1:50:5,l=1:30@ave]
```

See the chapter "Grids and Regions", section "Regions" (p. 162) for more discussion of the syntax of region qualifiers and transformations.

Some examples of valid variable syntax are

```
Myvar                 ! data set and region as per current context
myvar[D=2]            ! myvar from data set number 2 (see SHOW DATA)
myvar[D=a_dset]       ! myvar from data set a_dset.cdf or a_dset.des
myvar[D=myfile.txt]   ! myvar from file myfile.txt
myvar[G=gridname]     ! myvar regridded to grid gridname
myvar[G=var2]         ! myvar regridded to the grid of var2
                      ! which is in the same data set as myvar
myvar[G=var2[D=2]]    ! myvar regridded to the grid of var2
                      ! which is in data set number 2
myvar[GX=axisname]    ! myvar regridded to a dynamic grid which
                      ! has X axis axisname
myvar[GX=var2]        ! myvar regridded to a dynamic grid which
                      ! has the X axis of variable var2
myvar[I=1:31:5]       ! myvar subsampled at every 5th point
                      ! (regridded to a subsampled axis)
myvar[X=20E:50E:5]    ! myvar subsampled at every 5 degrees
                      ! (regridded to a 5-deg axis by linear
                      !  interpolation)
```

Ch3 Sec1.2. File variables

File variables are stored in disk files. Input data files can be ASCII, binary, netCDF, or TMAP-formatted (see the chapter "Data Set Basics", p. 33). File variables are made available with the SET DATA (alias USE) command.

In some netCDF files the variable names are not consistent with Ferret's rules for variable naming. They may be case-sensitive (for example, variables "v" and "V" defined in the same file), may be restricted names such as the Ferret pseudo-variable names I, J, K, L, X, Y, Z, T, XBOX, YBOX, ZBOX, or TBOX, or they may include "illegal" characters such as "+", "-", "%", blanks, etc. To access such variable names in Ferret, simply enclose the name in single quotes. For example,

```
yes? PLOT 'x'
yes? CONTOUR 'SST from MP/RF measurements'
```

By the same token when using Ferret to output into netCDF files that Ferret did not itself create, the results may not be entirely as expected. Case-sensitivity of names is one aspect of this. Since Ferret is (by default) case insensitive and netCDF files are case-sensitive writing into a "foreign" file may result in duplicated entities in the file which differ only in case. Starting with Ferret v6.0 you may specify

```
yes? CANCEL MODE upcase_output
```

and output to netCDF files will retain the original case of the variables and attribute names. Also see the documentation on SET ATTRIBUTE/OUTPUT for more about controlling which attributes are written to netCDF files.

Ch3 Sec1.3. Pseudo-variables

Pseudo-variables are variables whose values are coordinates or coordinate information from a grid. Valid pseudo-variables are

X – x axis coordinates	XBOX – size of x grid box	XBOXLO-grid cell lower bound
Y – y axis coordinates	YBOX – size of y grid box	XBOXHI-grid cell upper bound
Z – z axis coordinates	ZBOX – size of z grid box	YBOXLO-grid cell lower bound
T – t axis coordinates	TBOX – size of t grid box	YBOXHI-grid cell upper bound
I – x axis subscripts		ZBOXLO-grid cell lower bound
J – y axis subscripts		ZBOXHI-grid cell upper bound
K – z axis subscripts		TBOXLO-grid cell lower bound
L – t axis subscripts		TBOXHI-grid cell upper bound

A grid box is a concept needed for some transformations along an axis; it is the length along an axis that belongs to a single grid point and functions as a weighting factor during integrations and averaging transformations.

The pseudo-variables I, J, K, and L are subscripts; that is, they are a coordinate system for referring to grid locations in which the points along an axis are regarded as integers from 1 to the number of points on the axis. This is clear if you look at one of the sample data sets:

```
yes? USE levitus_climatology
yes? SHOW DATA
    1> /home/e1/tmap/fer_dsets/descr/levitus_climatology.des  (default)
       Levitus annual climatology (1x1 degree)
             diagnostic variables: NOT available
      name   title                    I         J         K       L
      TEMP   TEMPERATURE            1:360     1:180     1:20      ...
        ... on grid GLEVITR1   X=20E:20E(380) Y=90S:90N  Z=0m:5000m
      SALT   SALINITY               1:360     1:180     1:20      ...
        ... on grid GLEVITR1   X=20E:20E(380) Y=90S:90N  Z=0m:5000m
```

We see that there are 20 points along the z-axis (1:20 under K), for example, and that the z-axis coordinate values range from 0 meters to 5000 meters. Pseudo-variables depend only on the underlying grid, and not on the variables (in this case, temperature and salt).

Examples: Pseudo-variables

1) ***yes? LIST/I=1:10 I***
2) ***yes? LET xflux = u * xbox[G=u]***

Ch3 Sec1.3.1. Grids and axes of pseudo-variables

The name of a pseudo-variable (p. 61), alone, ("I", "T", "ZBOX", etc.) is not sufficient to determine the underlying axis of the pseudo-variable. The underlying axis may be specified explicitly, may be inherited from other variables used in the same expression, may be generated dynamically, or may be inherited from the current default grid. The following examples illustrate the possibilities:

```
TEMP + Y          ! pseudo-variable Y inherits the y axis of variable TEMP
Y[G=TEMP]         ! explicit: Y refers to the y axis of variable TEMP
Y[GY=axis_name]   ! explicit: Y refers to axis axis_name
Y[Y=0:90:2]       ! y axis is dynamically generated (See "dynamic axes">,
                  ! p. 148)
```

In the expression

```
LET A = X + Y
```

in which the definition provides no explicit coaching, nor are there other variables from which Y can inherit an axis, the axis of Y will be inherited from the current default grid. The current default grid is specified by the SET GRID command and may be queried at any time with the SHOW GRID command. SHOW GRID will respond with "Default grid for DEFINE VARIABLE is grid".

Note that when pseudo-variables are buried within a user variable definition they do not inherit from variables used in conjunction with the user variable. For example, contrast these expressions involving pseudo-variable Y

```
USE coads_climatology ! has variable SST
LET A = Y             ! Y buried inside variable A (axis indeterminate)
LIST SST + A          ! y axis inherited from current default grid
LIST SST + Y          ! y axis inherited from grid of SST
LIST SST + A[G=SST]   ! y axis inherited from grid of SST
```

Ch3 Sec1.4. User-defined variables

New variables can be defined from existing variables and from abstract mathematical quantities (such as COS(latitude)) with command DEFINE VARIABLE (alias LET). The section later in this chapter, Defining New Variable (p. 143) expands on this capability.

See command DEFINE VARIABLE (p. 349) and command LET (p. 363) in the Commands Reference. Example 3 shows the use of masking, a useful concept in constructing variables.

Examples: User-defined variables

```
1) yes? LET/TITLE="Surface Relief x1000 (meters)" r1000=rose/1000
2) yes? LET/TITLE="Temperature Deviation" tdev=temp - temp[Z=@ave]
3) yes? LET a = IF (sst GT 20. AND sst LT 30.) THEN sst ELSE 20.
```

Ch3 Sec1.5. Abstract variables

Ferret can be used to manipulate abstract mathematical quantities such as SIN(x) or EXP(k*t)—quantities that are independent of file variable values. Such quantities are referred to as abstract expressions.

Example: Abstract variables

Contour the function
```
COS(a*Y)/EXP(b*T)  where a=0.25 and b=-0.02
```

over the range
```
Y=0:45 (degrees) and  T=1:100 (hours)
```

with a resolution of
0.5 degree on the Y axis and 2 hours on the T axis.

Quick and dirty solution:
```
yes? CONTOUR COS(0.25*Y[Y=0:45:0.5])/EXP(-0.2*T[T=1:100:2])
```

Nicer (Figure 3_1); plot is documented with correct units and titles):

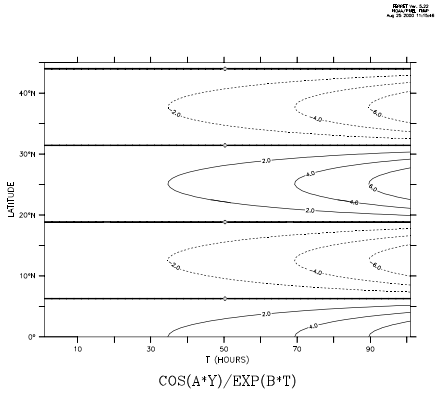


Figure 3_1

```
yes? DEFINE AXIS/Y=0:45:0.5/UNIT=DEGREES yax
yes? DEFINE AXIS/T=1:100:2/UNIT=HOURS tax
yes? DEFINE GRID/T=tax/Y=yax my_grid
yes? SET GRID my_grid
yes? LET a=0.25
yes? LET b=-0.02
yes? CONTOUR COS(a*Y)/EXP(b*T)
```

See the chapter "Grids and Regions", section "Grids" (p. 145), for more information on grids.

Ch3 Sec1.6. Missing value flags

Data values that are absent or undefined for mathematical reasons (e.g., 1/0) will be represented in Ferret with a missing value flag. In SHADE outputs a missing value flag embedded at some point in a variable will result in a transparent rectangular hole equal to the size of the grid cell of the missing value. In a CONTOUR or FILL plot it will result in a larger hole—extending past the grid box edge to the coordinate location of the next adjacent non-missing point—since contour lines cannot be interpolated between a missing value and its neighboring points. In the output of the LIST command for cases where the /FORMAT qualifier is not used the missing value will be represented by 4 dots ("...."). For cases where LIST/FORMAT=FORTRAN-format is used the numerical value of the missing value flag will be printed using the format provided.

Ch3 Sec1.6.1. Missing values in input files

Ferret does not impose a standard for missing value flags in input data sets; each variable in each data set may have its own distinct missing value flag(s). The flag(s) actually in use by a data set may be viewed with the SHOW DATA/VARIABLES command. If no missing value flag is specified for a data set Ferret will assume a default value of –1.E+34.

For EZ input data sets, either binary or ASCII, the missing data flag may be specified with the SET VARIABLE/BAD= command. A different value may be specified for each variable in the data set.

For netCDF input data sets the missing value flag(s) is indicated by the values of the attributes "missing_value" and "_FillValue." If both attributes are defined to have different values both will be recognized and used by Ferret as missing value indicators, however the occurrences of _FillValue will be replaced with the value of missing_value as the data are read into Ferret's memory cache so that only a single missing value flag is apparent inside of Ferret. The command SET VARIABLE/BAD= can also be applied to netCDF variables, thereby temporarily setting a user-imposed value for _FillValue. If there are values of NaN in the file, then NaN must be listed in either the as either the "missing_value" OR "_FillValue" attribute and then NaN is the missing value. Or, the user may specify SET VARIABLE/BAD=NAN (case insensitive) to designate the Fortran value NaN (not a number) as the bad value flag for a given variable in a netCDF dataset.

Ch3 Sec1.6.2. Missing values in user-defined variables

User-defined variables may in general be defined as expressions involving multiple variables. The component variables need not in general agree in their choice of missing value flags. The result variable will inherit the bad value flag of the first variable in the expression. If the first component in the expression is a constant or a pseudo-variable, then Ferret imposes its default missing value flag of –1.E+34.

The function MISSING(variable,replacement) provides a limited control over the choice of missing values in user-defined variables. Note, however, that while the MISSING function will replace the missing values with other values it will not change the missing value flag. In other words, the replacement values will no longer be regarded as missing.

Ch3 Sec1.6.3. Missingvalues in output NetCDF files

Values flagged as missing inside Ferret will be faithfully transferred to output files—no substitution will occur as the data are written. In the case of netCDF output files both of the attributes missing_value, and _FillValue will be set equal to the missing value flag.

Under some circumstances it is desirable to save a user-defined variable in a netCDF file and then to redefine that variable and to append further output. (An example of this is the process of consolidating several files of input, say, moored measurements, into a gridded output.) The process of appending will not change any of the netCDF attributes—neither long_name (title), units, nor missing_value or _FillValue. If the subsequent variable definitions do not agree in their choice of missing value flags the resulting output may contain multiple missing value flags that will not be properly documented.

An easy "trick" that avoids this situation is to begin all of the variable definitions with an addition of zero, "LET var = 0 + ..." The addition of zero will not affect the value of the output but it will guarantee that a missing value flag of –1.E+34 will be consistently used. Of course, you will want to use the SET VARIABLE/TITLE= command in conjunction with this approach.

Ch3 Sec1.6.4. Displaying the missing value flag

If the LIST command is used, missing values are, by default, displayed as "...." To examine the flag as a numerical value, use LIST/FORMAT=(E) (or some other suitable format).

Ch3 Sec1.7. Returning properties of variables

The keyword RETURN= can reveal the size and shape, title, bad flag, units, and other properties of a variable or expression. See p. 138for a description of this useful construct.

Ch3 Sec1.8. Variable and dataset attributes

Beginning with Ferret V6.0, Ferret has access to attributes of all variables, including netCDF variables, netCDF coordinate variables, user-defined variables and variables from non-netCDF datasets.. When a netCDF dataset is opened, its variables and attributes are stored. For other file variables and user variables, a basic set of attributes is created, including

at least the missing_value flag, _FillValue, and a long-name attribute with the variable's definition. If a variable is defined with /UNITS or /TITLE then those are also included among the attributes.

The power of this feature is in its access to attribute information as strings or numeric data. The text or values can be examined, edited, and used in computations. The attributes of a variable can be changed, removed, and new attributes defined, and when writing netCDF files, we can control which attributes that are written to the file.

The general syntax for this access is

```
varname.attname
```

For example,

```
yes? USE coads_climatology
yes? LIST sst.units
             VARIABLE : SST.UNITS
             FILENAME : coads_climatology.cdf
             FILEPATH : /home/porter/tmap/ferret/linux/fer_dsets/data/
        "Deg C"
```

To refer to a coordinate variable, put the name of the coordinate variable in parentheses. The RETURN= keyword is helpful for getting the names of coordinate axes.

```
yes? LIST (coadsx).point_spacing
             VARIABLE : (COADSX).POINT_SPACING
             FILENAME : coads_climatology.cdf
             FILEPATH : /home/porter/tmap/ferret/linux/fer_dsets/data/
        "even"
yes? LIST (`sst,return=taxis`).time_origin
 !-> list (TIME).time_origin
             VARIABLE : (TIME).TIME_ORIGIN
             FILENAME : coads_climatology.cdf
             FILEPATH : /home/porter/tmap/ferret/linux/fer_dsets/data/
        "1-JAN-0000 00:00:00"
```

The dataset itself can have attributes; the global attributes. To refer to the dataset, use a dot. The history attribute of the current dataset can be referred to by

```
..history
```

For example,

```
yes? LET a = 1; save/clobber/file=new.nc a
 LISTing to file new.nc
yes? USE new.nc
yes? LIST ..history
             VARIABLE : ..HISTORY
             FILENAME : new.nc
        "FERRET V6    5-Jul-06"
```

When there is more than one dataset open, the dataset specifier [d=1] or [d=datasetname] comes at the end of the varname.attname string:

```
yes? use dataset_1
yes? use dataset_2
yes? list var.units[d=1]
```

Ch3 Sec1.8.1. SHOW ATTRIBUTE commands

SHOW DATA/ATRIBUTE	Expands the SHOW DATA output, listing dataset attributes and variable attributes
SHOW ATTRIBUTE varname.attname	Lists the value(s) of an attribute. It gives the same information as LIST varname.attname
SHOW ATRIBUTE/ALL varname	Lists all of the attributes for the variable
`varname.attname,RETURN=size`	Returns the length of a string attribute, or the number of values in a numeric attribute

Example:

```
yes? USE etopo60
yes? SHOW DATA/ATT
     currently SET data sets:
    1> /home/porter/tmap/ferret/linux/fer_dsets/data/etopo60.cdf
(default)
 VARIABLE                ATTRIBUTE NAME   TYPE   SIZE OUTFLAG VALUE
------------------------------------------------------------------------

 .                       history          CHAR    28   T      FERRET
V4.45 (GUI) 22-May-97

(ETOPO60X)               units            CHAR    12   T
degrees_east
                         modulo           CHAR    1    T
                         point_spacing    CHAR    4    T      even

(ETOPO60Y)               units            CHAR    13   T
degrees_north
                         point_spacing    CHAR    4    T      even

 ROSE                    missing_value    FLOAT   1    T
-1.000000E+34
                         _FillValue       FLOAT   1    T
-1.000000E+34
                         long_name        CHAR    34   T      RELIEF OF
THE SURFACE OF THE EARTH
                         history          CHAR    12   T      From
etopo60
                         units            CHAR    6    T      METERS


yes? SHOW ATT rose.missing_value
     attributes for dataset:
/home/porter/tmap/ferret/linux/fer_dsets/data/etopo60.cdf
 ROSE.missing_value = -1.000000E+34
```

```
yes? SHOW ATT/ALL rose
     attributes for dataset:
/home/porter/tmap/ferret/linux/fer_dsets/data/etopo60.cdf
 ROSE.missing_value = -1.000000E+34
 ROSE._FillValue = -1.000000E+34
 ROSE.long_name = RELIEF OF THE SURFACE OF THE EARTH
 ROSE.history = From etopo60
 ROSE.units = METERS

yes? SAY `rose.missing_value,RETURN=SIZE`
 !-> MESSAGE/CONTINUE 1
1
yes? say `rose.long_name,return=size`
 !-> MESSAGE/CONTINUE 34
34
```

Ch3 Sec1.8.2. Attribute keywords

In addition to the attributes of variables we can also learn about the number and names of variables in a dataset, the dimensions and number of attributes for each variable, and the same information about the dimensions (coordinate variables). New keywords are introduced:

DATASET	
..nvars	number of variables (excluding coordinate variables
..varnames	variable names
..ndims	number of dimensions (coordinate variables)
..dimnames	dimension names
..nattrs	number of global attributes
..attnames	global attribute names
VARIABLE	
var.ndims	number of dimsnsions for variable
var.dimnames	names of variable's dimensions
var.nattrs	number of attributes for variable
var.attnames	attribute names for variable

Examples:

```
yes? use mydata.nc
yes? let names = ..varnames
yes? list names
yes? let anames = `names[i=2]`.attname

yes? list ..ndims
yes? list myvar.dimhnames
```

Ch3 Sec1.8.3. Programmatic access to attributes

To form a variable from attribute information simply use a LET command:

```
yes? let a = varname.missing_value
yes? let b = (axisname).units
yes? let h = ..history
```

Or, use attribute keywords

```
   ! The number of variables (not including coordinate variables)
yes? let nv = ..nvars

   ! The number of coordinate variables
yes? let nc = ..ndims

   ! A variable with a list of all coordinate variables
   ! from data set 1
yes? let lnames = ..dimnames[d=1]

   ! If two datasets have a variable TEMP, list the units
   ! of the variable in each dataset
yes? let uu = temp.units
yes? list/d=1 uu
yes? list/d=2 uu
```

Ch3 Sec1.8.4. Editing attributes

We can change an existing attribute for a variable, add new attributes, and control which are written to output netCDF files. We can define attributes to be of type STRING or FLOAT. (on ouput we will be able to request type conversions to other numeric types). Numeric attributes may be lists of values. If the type is note specified by the /TYPE qualifier, the type is inferred from the value of the attribute. The outupt flag is set with the /OUTPUT qualifier: SET ATTRIBUTE/OUTPUT varname.attname marks the attribute for output to netCDF files, and CANCEL ATTRIBUTE/OUTPUT causes that attribute to be hidden on output.

Example: add new attributes with DEFINE ATTRIBUTE

```
yes? USE etopo60

yes? DEFINE ATT/TYPE=float rose.floatval = 22
yes? DEFINE ATT rose.pp = {1.5, 1.9}
yes? DEFINE ATT/TYPE=string rose.strval = 2
yes? DEFINE ATT rose.some_text = "some text about ROSE "

yes? SHOW ATT/ALL rose
     attributes for dataset:
/home/porter/tmap/ferret/linux/fer_dsets/data/etopo60.cdf
 ROSE.missing_value = -1.000000E+34
 ROSE._FillValue = -1.000000E+34
 ROSE.long_name = RELIEF OF THE SURFACE OF THE EARTH
 ROSE.history = From etopo60
 ROSE.units = METERS
```

```
 ROSE.floatval = 22
 ROSE.pp = 1.5, 1.9
 ROSE.strval = 2
 ROSE.some_text = some text about the ROSE variable

   ! /D= specifies the dataset if needed
yes? DEFINE ATT/D=1 rose.another_attribute = 6

   ! /OUTPUT sets the output flag: this attribute will be
   ! written to output files with the variable.
yes? DEFINE ATT/OUTPUT rose.more_text = "Another notation"
```

We can also change an existing attribute. As in the commands DEFINE VARIABLE; SET VARIABLE; CANCEL VARIABLE we can use DEFINE ATTRIBUTE to redefine an existing attribute or SET ATTRIBUTE and CANCEL ATTRIBUTE to change settings or values of an attribute.

Examples: continuing the above example

```
   ! Change the text in long_name
yes? DEFINE ATT rose.long_name = "Relief of the Surface of the Earth"

   ! The rose.history attribute will NOT be written to output files
yes? CANCEL ATT/OUTPUT rose.history

   ! The new attribute rose.pp we defined above WILL be written
yes? SET ATT/OUTPUT rose.pp
```

For a coordinate variable, SET AXIS and SET ATTRIBUTE commands do the same thing:

```
yes? USE levitus_climatology

   ! These commands are equivalent
yes? SET AXIS/POSITIVE="up" `temp,return=zaxis`

yes? SET ATT (`temp,return=zaxis`).positive = "up"
```

Another way to edit attributes is to inherit them from another variable. Use the qualifer SET ATTRIBUTE/LIKE=

Example (note the long_name and units)

```
yes? use levitus_climatology
yes? SET ATT/LIKE=salt temp
yes? SHOW ATTRIBUTE/ALL temp
     attributes for dataset:
/home/porter/tmap/ferret/linux/fer_dsets/data/levitus_climatology.cdf
 TEMP.missing_value = -1.E+10
 TEMP._FillValue = -1.E+10
 TEMP.long_name = SALINITY
 TEMP.history = From levitus_climatology
 TEMP.units = PPT
```

As noted at the start of this section, the attribute commands can be applied to any variable, not just those in netCDF datsets. A user variable has a few default attributes when it is defined; a missing_value, _FillValue, and a long_name, and units if they are included in the definition. Here a user variable inherits all of the attributes from a file variable. Here salt2 initially has only a long-name, missing_value, and _FillValue. We can inherit the units and a more descriptive long_name from the dataset variable.

```
yes? USE levitus_climatology
yes? LET salt2 = 2* salt

yes? SHOW ATT/ALL salt2
     attributes for user-defined variables
 SALT2.long_name = 2* SALT
 SALT2.missing_value = -1.000000E+34

yes? SET ATT/LIKE=salt salt2
yes? SHOW ATT/ALL salt2
     attributes for user-defined variables
 SALT2.missing_value = -1.E+10
 SALT2._FillValue = -1.E+10
 SALT2.long_name = SALINITY
 SALT2.history = From levitus_climatology
 SALT2.units = PPT
```

Finally, fix the long_name of our new variable

```
yes? set att salt2.long_name = "2 * `salt.long_name`"
 !-> set att salt2.long_name = "2 * SALINITY"
 *** NOTE: Changing the value of attribute
```

Ch3 Sec1.8.5. Output attributes to NetCDF fies

As noted in the previous section, the value of the attribute output flag, set by the commands DEFINE ATTRIBUTE/OUTPUT, SET ATTRIBUTE/OUTPUT and CANCEL ATTRIBUTE/OUTPUT controls whether an attribute is written to a netCDF file when the variable is written. The SET ATTRIBUTE/OUTPUT= allows more precise control over the writing of attributes.

SET ATT/OUTPUT varname.attname	Sets an individual attribute to be written when the variable is written
SET ATT/OUTPUT=all varname	Output all attributes that have been defined for a variable
SET ATT/OUTPUT=default varname	Write only the outputs that Ferret typically writes by default (see p. 275)
SET ATT/OUTPUT=none varname	Output no attributes for the variable
CANCEL ATT/OUTPUT varname.attname	Suppresses output of the attribute when the variable is written.

Example:

```
yes? LET aa = 12.
yes? LET bb = {3,4.5,6,7,4}

yes? DEFINE ATT/OUTPUT att bb.my_title = "This is my new variable bb"
yes? DEFINE ATT bb.another_attr = 6

! Output just bb.mytitle, along with the default
! ones, missing_value, _FillValue, and long_name.

yes? SAVE/CLOBBER/FILE=ab.nc aa,bb

! Output all attributes

yes? SET ATT/OUTPUT=all bb
yes? SAVE/CLOBBER/FILE=ab.nc aa,bb

! Output default attributes

yes? SET ATT/OUTPUT=default bb
yes? SAVE/CLOBBER/FILE=ab.nc aa,bb
```

We can suppress output of an attribute that Ferret would otherwise add, the "axis" attribute for coordinate axes.

```
yes? USE levitus_climatology
yes? CANCEL ATT/OUT (`temp,return=xaxis`).axis
```

Ch3 Sec1.8.6. Output Variables to NetCDF files

The attribute-handling structure gives us flexibility in writing variables to netCDF files. We can specify that the upper- or lower-case spelling of variables and attributes from an input netCDF file be preserved on output, or that these should be upper-cased as has been done previously in Ferret. By default, Ferret still upcases variable names when it writes variables to netCDF files. If we want to keep the case of the names that they had on input, use

```
CANCEL MODE upcase_output
```

We can also control the data type of variables written to output netCDF files, with SET VAR/OUTTYPE=]. The netCDF library is used to convert the data type of a Ferret FLOAT value to the requested output type. The types allowed are FLOAT, INT, SHORT, and BYTE, or INPUT to preserve the type the data had on input. To write integers, for instance,

```
yes? SET DATA etopo60
yes? SET VAR/OUTTYPE=int4 rose
yes? SAVE/X=180/FILE=r_int.nc rose
 LISTing to file a.nc
 *** NOTE: Converting data type of missing_value NC_FLOAT to match
output type of variable NC_INT
 ** netCDF error:
 data in attribute missing_value not representable in output type NC_INT
```

Here, the missing_value of the variable cannot be represented as an integer, and the file was not written. Correct for this by assigning a new missing_value to the variable before writing:

```
yes? SET DATA etopo60
yes? SET VAR/OUTTYPE=int4/BAD=200000 rose
yes? SAVE/X=180/FILE=r_int.nc rose
 LISTing to file a.nc
 *** NOTE: Converting data type of missing_value NC_FLOAT to match
output type of variable NC_INT
```

Note that not all data can be represented in all types. When data is written to DOUBLE, it is always converted from the Ferret internal representation of single precision FLOAT data, even if the original data was double precision.

This mechanism lets us pack data when writing it to a netCDF file by using the add_offset and scale_factor attributes. Say a dataset has a variable called elev which is packed using these attributes. By default Ferret scales this data when it is read, and writes it in scaled (unpacked) form. To pack it, we turn on output of the scale_factor and add_offset attributes. Then Ferret will apply this scaling to data and its missing_value and _FillValue are rescaled on output.

```
yes? USE my_scaled_dset.nc
yes? SET ATT/OUTPUT elev.scale_factor
yes? SET ATT/OUTPUT elev.add_offset
yes? SET ATT/OUTTYPE=input elev
yes? SAVE/CLOBBER/FILE=scaled.nc/J=1 elev
```

We can use this technique to apply new scale factors to a variable or define scale and offset attributes when writing any variable.

Ch3 Sec2. EXPRESSIONS

Throughout this manual, Ferret commands that require and manipulate data are informally called "action" commands. These commands are:

PLOT
CONTOUR
FILL (alias for CONTOUR/FILL)
SHADE
VECTOR
POLYGON
WIRE
LIST
STAT
LOAD

Action commands may use any valid algebraic expression involving constants, operators (+,–,*,...), functions (SIN, MIN, INT,...), pseudo-variables (X, TBOX, ...) and other variables.

A variable name may optionally be followed by square brackets containing region, transformation, data set, and regridding qualifiers. For example, "temp", "salt[D=2]", "u[G=temp"], "u[Z=0:200@AVE]", "v[k=1:50:5]

The expressions may also contain a syntax of:

IF condition THEN expression_1 ELSE expression_2

Examples: Expressions

i) ***temp ^ 2***
temperature squared

ii) ***temp - temp[Z=@AVE]***
for the range of Z in the current context, the temperature deviations from the vertical average

iii) ***COS(Y)***
the cosine of the Y coordinate of the underlying grid (by default, the y-axis is implied by the other variables in the expression)

iv) ***IF (vwnd GT vwnd[D=monthly_navy_winds]) THEN vwnd ELSE 0***
use the meridional velocity from the current data set wherever it exceeds the value in data set monthly_navy_winds, zero elsewhere.

Ch3 Sec2.1. Operators

Valid operators are

+
–
*
/
^ (exponentiate)
AND
OR
GT
GE
LT
LE
EQ
NE

For instance the exponentiate operator can compute the square root of a variable as var^0.5

Ch3 Sec2.2. Multi-dimensional expressions

Operators and functions (discussed in the next section, Functions) may combine variables of like dimensions or differing dimensions.

If the variables are of like dimension then the result of the combination is of the same dimensionality as inputs. For example, suppose there are two time series that have data on the same time axis; the result of a combination will be a time series on the same time axis.

If the variables are of unlike dimensionality, then the following rules apply:

1) To combine variables together in an expression they must be "conformable" along each axis.
2) Two variables are conformable along an axis if the number of points along the axis is the same, or if one of the variables has only a single point along the axis (or, equivalently, is normal to the axis).
3) When a variable of size 1 (a single point) is combined with a variable of larger size, the variable of size 1 is "promoted" by replicating its value to the size of the other variable.
4) If variables are the same size but have different coordinates, they are conformable, but Ferret will issue a message that the coordinates on the axis are ambiguous. The result of the combination inherits the coordinates of the FIRST variable encountered that has more than a single point on the axis.

Examples:

Assume a region J=50/K=1/L=1 for examples 1 and 2. Further assume that variables v1 and v2 share the same x-axis.

1) ***yes? LET newv = v1[I=1:10] + v2[I=1:10] !same dimension (10)***

2) ***yes? LET newv = v1[I=1:10] + v2[I=5] !newv has length of v1 (10)***

3) We want to compare the salt values during the first half of the year with the values for the second half. Salt_diff will be placed on the time coordinates of the first variable—L=1:6. Ferret will issue a warning about ambiguous coordinates.

   yes? LET salt_diff = salt[L=1:6] - salt[L=7:12]

4) In this example the variable zero will be promoted along each axis.

```
yes? LET zero = 0 * (i+j)
yes? LIST/I=1:5/J=1:5  zero          !5X5 matrix of 0's
```

5) Here we calculate density; salt and temp are on the same grid. This expression is an XYZ volume of points (100×100×10) of density at 10 depths based on temperature and salinity values at the top layer (K=1).

```
yes? SET REGION/I=1:100/J=1:100
yes? LET dens = rho_un (salt[K=1], temp[K=1], Z[G=temp,K=1:10]
```

Ch3 Sec2.3. Functions

Functions are utilized with standard mathematical notation in Ferret. The arguments to functions are constants, constant arrays, pseudo-variables, and variables, possibly with associated qualifiers in square brackets, and expressions. Thus, all of these are valid function references:

- ***EXP(-1)***
- ***MAX(a,b)***
- ***TAN(a/b)***
- ***SIN(Y[g=my_sst])***
- ***DAYS1900(1989,{3,6,9},1)***

A few functions also take strings as arguments. String arguments must be enclosed in double quotes. For example, a function to write variable "u" into a file named "my_output.v5d", formatted for the Vis5D program might be implemented as

- ***LOAD WRITE_VIS5D("my_output.v5d", a)***

You can list function names and argument lists with:

```
yes? SHOW FUNCTIONS           ! List all functions
Yes? SHOW FUNCTIONS *TAN! List all functions containing string
```

Valid functions are described in the sections below. They are:

MAX	ATAN	XSEQUENCE	SAMPLEXY
MIN	ATAN2	YSEQUENCE	SCAT2GRIDGAUSS_XY
INT	MOD	ZSEQUENCE	SCAT2GRIDGAUSS_XZ
ABS	DAYS1900	TSEQUENCE	SCAT2GRIDGAUSS_YZ
EXP	MISSING	FFTA	SCAT2GRIDLAPLACE_XY
LN	IGNORE0	FFTP	SCAT2GRIDLAPLACE_XZ
LOG	RANDU	SAMPLEI	SCAT2GRIDLAPLACE_YZ
SIN	RANDN	SAMPLEJ	SORTI

COS	RHO_UN	SAMPLEK	SORTJ
TAN	THETA_FO	SAMPLEL	SORTK
ASIN	RESHAPE	SAMPLEIJ	SORTL
ACOS	ZAXREPLACE	SAMPLET_DATE	TAUTO_COR

See also the section on string functions (p. 230).

Grid-changing functions

It is generally advisable to include explicit limits when working with functions that replace axes. For example, consider the function SORTL(v). The expression

```
LIST/L=6:10 SORTL(v)
```

is not equivalent to

```
LIST SORTL(v[L=6:10])
```

The former will list the 6th through 10th sorted indices from the entire l range of variable v. The latter will list all of the indices that result from sorting v[l=6:10].

These functions in Ferret, including XSEQUENCE, SAMPLXY, and so on, are "grid-changing" functions. This means that the axes of the result may differ from the axes of the arguments. In the case of XSEQUENCE(sst), for example, the input grid for SST is

lon
lat
normal
time

whereas the output grid is

abstract
normal
normal
normal

so all axes of the input are replaced.

Grid-changing functions create a potential ambiguity about region specifications. Suppose that the result of XSEQUENCE(sst[L=1]) is a list of 50 points along the ABSTRACT X axis. Then it is natural that

```
LIST/I=10:20 XSEQUENCE(sst[L=1])
```

should give elements 10 through 20 taken from that list of 50 points (and it does.) However, one might think that "I=10:20" referred to a subset of the longitude axis of SST. Therein lies the ambiguity: one region was specified, but there are 2 axes to which the region might apply.

It gets a degree more complicated if the grid-changing function takes more than one argument. Since the input arguments need not be on identical grids, a result axis (X,Y,Z, or T) may be replaced with respect to one argument, but actually taken from another (consider ZAXREPLACE, for example.) Ferret resolves the ambiguities thusly:

If in the result of a grid-changing function, an axis (X, Y, Z, or T) has been replaced relative to some argument, then region information which applies to the result of the function on that axis will NOT be passed to that argument.

So, when you issue commands like

```
SET REGION/X=20E:30E/Y=0N:20N/L=1
LIST XSEQUENCE(sst)
```

the X axis region ("20E:30E") applies to the result ABSTRACT axis -- it is not passed along to the argument, SST. The Y axis region is, in fact, ignored altogether, since it is not relevant to the result of XSEQUENCE, and is not passed along to the argument.

Ch3 Sec2.3.1. MAX

MAX(A, B) Compares two fields and selects the point by point maximum.
MAX(temp[K=1], temp[K=2]) returns the maximum temperature comparing the first 2 z-axis levels.

Ch3 Sec2.3.2. MIN

MIN(A, B) Compares two fields and selects the point by point minimum.
MIN(airt[L=10], airt[L=9]) gives the minimum air temperature comparing two timesteps.

Ch3 Sec2.3.3. INT

INT (X) Truncates values to integers.
INT(salt) returns the integer portion of variable "salt" for all values in the current region.

Ch3 Sec2.3.4. ABS

ABS(X) absolute value.
ABS(U) takes the absolute value of U for all points within the current region

Ch3 Sec2.3.5. EXP

EXP(X) exponential e^x; argument is real.
EXP(X) raises e to the power X for all points within the current region

Ch3 Sec2.3.6. LN

LN(X) Natural logarithm $\log_e X$; argument is real.
LN(X) takes the natural logarithm of X for all points within the current region

Ch3 Sec2.3.7. LOG

LOG(X) Common logarithm $\log_{10} X$; argument is real.
LOG(X) takes the common logarithm of X for all points within the current region

Ch3 Sec2.3.8. SIN

SIN(THETA) Trigonometric sine; argument is in radians and is treated modulo 2*pi.
SIN(X) computes the sine of X for all points within the current region.

Ch3 Sec2.3.9. COS

COS(THETA) Trigonometric cosine; argument is in radians and is treated modulo 2*pi.
COS(Y) computes the cosine of Y for all points within the current region

Ch3 Sec2.3.10. TAN

TAN(THETA) Trigonometric tangent; argument is in radians and is treated modulo 2*pi.
TAN(theta) computes the tangent of theta for all points within the current region

Ch3 Sec2.3.11. ASIN

ASIN(X) Trigonometric arcsine (-pi/2,pi/2) of X in radians.The result will be flagged as missing if the absolute value of the argument is greater than 1; result is in radians.
ASIN(value) computes the arcsine of "value" for all points within the current region

Ch3 Sec2.3.12. ACOS

COS(X) Trigonometric arccosine (0,pi), in radians. The result will be flagged as missing of the absolute value of the argument greater than 1; result is in radians.
ACOS (value) computes the arccosine of "value" for all points within the current region

Ch3 Sec2.3.13. ATAN

ATAN(X) Trigonometric arctangent (-pi/2,pi/2); result is in radians.
ATAN(value) computes the arctangent of "value" for all points within the current region

Ch3 Sec2.3.14. ATAN2

ATAN2(X,Y) 2-argument trigonometric arctangent of X/Y (-pi,pi); discontinuous at Y=0.
ATAN2(X,Y) computes the 2-argument arctangent of X/Y for all points within the current region

Ch3 Sec2.3.15. MOD

MOD(A,B) Modulo operation (arg1 – arg2*[arg1/arg2]). Returns the remainder when the first argument is divided by the second.
MOD(X,2) computes the remainder of X/2 for all points within the current region

Ch3 Sec2.3.16. DAYS1900

DAYS1900(year,month,day) computes the number of days since 1 Jan 1900. This function is useful in converting dates to Julian days on the standard Gregorian calendar. If the year is prior to 1900 a negative number is returned. This means that it is possible to compute Julian days relative to, say, 1800 with the expression
LET jday1800 = DAYS1900 (year, month, day) - DAYS1900(1800,1,1)

Ch3 Sec2.3.17. MISSING

MISSING(A,B) Replaces missing values in the first argument (multi-dimensional variable) with the second argument; the second argument may be any conformable variable.
MISSING(temp, -999) replaces missing values in temp with –999
MISSING(sst, temp[D=coads_climatology]) replaces missing sst values with temperature from the COADS climatology

Ch3 Sec2.3.18. IGNORE0

IGNORE0(VAR) Replaces zeros in a variable with the missing value flag for that variable.
IGNORE0(salt) replaces zeros in salt with the missing value flag

Ch3 Sec2.3.19. RANDU

RANDU(A) Generates a grid of uniformly distributed [0,1] pseudo-random values. The first valid value in the field is used as the random number seed. Values that are flagged as bad remain flagged as bad in the random number field.
RANDU(temp[I=105:135,K=1:5]) generates a field of uniformly distributed random values of the same size and shape as the field "temp[I=105:135,K=1:5]" using temp[I=105,k=1] as the pseudo-random number seed.

Ch3 Sec2.3.20. RANDN

RANDN(A) Generates a grid of normally distributed pseudo-random values. As above, but normally distributed rather than uniformly distributed.

Ch3 Sec2.3.21. RHO_UN

RHO_UN(SALT, TEMP, P) Calculates the mass density rho (kg/m^3) of seawater from salinity SALT(salt, psu), temperature TEMP(deg C) and pressure P(dbar) using the 1980 UNESCO International Equation of State (IES80). Either in-situ or potential density may be computed depending upon whether the user supplies in-situ or potential temperature.

Note that to maintain accuracy, temperature must be converted to the IPTS-68 standard before applying these algorithms. For typical seawater values, the IPTS-68 and ITS-90 temperature scales are related by T_68 = 1.00024 T_90 (P. M. Saunders, 1990, WOCE Newsletter 10). The routine uses the high pressure equation of state from Millero et al. (1980) and the one-atmo-

sphere equation of state from Millero and Poisson (1981) as reported in Gill (1982). The notation follows Millero et al. (1980) and Millero and Poisson (1981).

RHO_UN(salt, temp, P)

Ch3 Sec2.3.22. THETA_FO

THETA_FO(SALT, TEMP, P, REF) Calculates the potential temperature of a seawater parcel at a given salinity SALT(psu), temperature TEMP(deg. C) and pressure P(dbar), moved adiabatically to a reference pressure REF(dbar).

This calculation uses Bryden (1973) polynomial for adiabatic lapse rate and Runge-Kutta 4th order integration algorithm. References: Bryden, H., 1973, Deep-Sea Res., 20, 401–408; Fofonoff, N.M, 1977, Deep-Sea Res., 24, 489–491.

THETA_FO(salt, temp, P, P_reference)

Ch3 Sec2.3.23. RESHAPE

RESHAPE(A, B) The result of the RESHAPE function will be argument A "wrapped" on the grid of argument B. The limits given on argument 2 are used to specify subregions within the grid into which values should be reshaped.

RESHAPE(Tseries,MonthYear)

Two common uses of this function are to view multi-year time series data as a 2-dimensional field of 12-months vs. year and to map ABSTRACT axes onto real world coordinates. An example of the former is

```
DEFINE AXIS/t=15-JAN-1982:15-DEC-1985/NPOINTS=48/UNITS=DAYS tcal
LET my_time_series = SIN(T[gt=tcal]/100)
! reshape 48 months into a 12 months by 4 year matrix
DEFINE AXIS/t=1982:1986:1 tyear
DEFINE AXIS/Z=1:12:1 zmonth
LET out_grid = Z[GZ=zmonth]+T[GT=tyear]
LET my_reshaped = RESHAPE(my_time_series, out_grid)
SHOW GRID my_reshaped
   GRID (G001)
name       axis              # pts   start                end
normal     X
normal     Y
ZMONTH     Z                  12 r   1                    12
TYEAR      T                   5 r   1982                 1986
```

For any axis X,Y,Z, or T if the axis differs between the input output grids, then limits placed upon the region of the axis in argument two (the output grid) can be used to restrict the geometry into which the RESHAPE is performed. Continuing with the preceding example:

! Now restrict the output region to obtain a 6 month by 8 year matrix

```
LIST RESHAPE(my_time_series,out_grid[k=1:6])
         RESHAPE(MY_TIME_SERIES,OUT_GRID[K=1:6])
                1       2       3       4       5       6
                1       2       3       4       5       6
1982   / 1:  0.5144  0.7477  0.9123  0.9931  0.9827  0.8820
1983   / 2:  0.7003  0.4542  0.1665 -0.1366 -0.4271 -0.6783
1984   / 3: -0.8673 -0.9766 -0.9962 -0.9243 -0.7674 -0.5401
1985   / 4: -0.2632  0.0380  0.3356  0.6024  0.8138  0.9505
1986   / 5:  0.9999  0.9575  0.8270  0.6207  0.3573  0.0610
```

For any axis X,Y,Z, or T if the axis is the same in the input and output grids then the region from argument 1 will be preserved in the output. This implies that when the above technique is used on multi-dimensional input, only the axes which differ between the input and output grids are affected by the RESHAPE operation. However RESHAPE can only be applied if the reshape operation preserves the ordering of data on the axes in four dimensions. The RESHAPE function only "wraps" the variable to the new grid, keeping the data ordered as it exists in memory, that is, ordered by X (varying fastest) then -Y-Z-T (slowest index). It is an operation like @ASN regridding. Subsetting is done if requested by region specifiers, but the function does not reorder the data as it is put on the new axes. For instance, if your data is in Z and T:

```
SHOW GRID G001
   GRID (G001)
name       axis              # pts   start                end
normal     X
normal     Y
ZMONTH     Z                  12 r   1                    12
T_ABSTR    T                   5 r   1                     5
```

and you wish to put it on a new grid, GRIDYZ

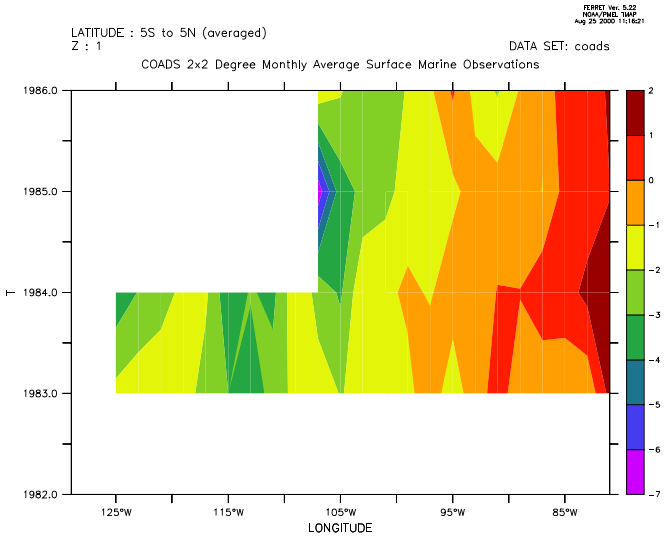
```
SHOW GRID gridyz
   GRID (GRIDYZ)
name       axis              # pts   start                end
normal     X
YAX        LATITUDE            5 r   15N                 19N
ZMONTH     Z                  12 r   1                    12
normal     T
```

then the RESHAPE function would NOT correctly wrap the data from G001 to GRIDYZ, because the data is ordered with its Z coordinates changing faster than its T coordinates, and on output the data would need to be reordered with the Y coordinates changing faster then the Z coordinates.

The following filled contour plot of longitude by year number illustrates the use of RESHAPE in multiple dimensions by expanding on the previous example: (Figure 3_2)

```
!  The year-by-year progression January winds for a longitudinal patch
!  averaged from 5s to 5n across the eastern Pacific Ocean. Note that
!  k=1 specifies January, since the Z axis is month
```

```
USE coads
LET out_grid = Z[GZ=zmonth]+T[GT=tyear]+X[GX=uwnd]+Y[GY=uwnd]
LET uwnd_mnth_ty = RESHAPE(uwnd, out_grid)
FILL uwnd_mnth_ty[X=130W:80W,Y=5S:5N@AVE,K=1]
```



RESHAPE(UWND, OUT_GRID)

Figure 3_2

In the second usage mentioned, to map ABSTRACT axes onto real world coordinates, suppose xpts and ypts contain time series of length NT points representing longitude and latitude points along an oceanographic ship track and the variable global_sst contains global sea surface temperature data. Then the result of

LET sampled_sst = SAMPLEXY(global_sst, xpts, ypts)

will be a 1-dimensional grid: NT points along the XABSTRACT axis. The RESHAPE function can be used to remap this data to the original time axis using RESHAPE(sampled_sst, xpts)

```
LET sampled_sst = SAMPLEXY(global_sst,\
  xpts[t=1-jan-1980:15-jan-1980],\
  ypts[t=1-jan-1980:15-jan-1980])

LIST RESHAPE(sampled_sst, xpts[t=1-jan-1980:15-jan-1980])
```

When the input and output grids share any of the same axes, then the specified sub-region along those axes will be preserved in the RESHAPE operation. In the example "RESHAPE(myTseries,myMonthYearGrid)" this means that if myTseries and myMonthYearGrid were each multidimensional variables with the same latitude and longitude grids then

```
RESHAPE(myTseries[X=130E:80W,Y=5S:5N],myMonthYearGrid)
```

would map onto the X=130E:80W,Y=5S:5N sub-region of the grid of myMonthYearGrid. When the input and output axes differ the sub-region of the output that is utilized may be controlled by inserting explicit limit qualifiers on the second argument

Ch3 Sec2.3.24. ZAXREPLACE

ZAXREPLACE(V,ZVALS,ZAX) Convert between alternative monotonic Zaxes, where the mapping between the source and destination Z axes is a function of X,Y, and or T. The function regrids between the Z axes using linear interpolation between values of V. See also the related functions ZAXREPLACE_BIN (p. 484) and ZAXREPLACE_AVG (p. 483) which use binning and averaging to interpolate the values.

Typical applications in the field of oceanography include converting from a Z axis of layer number to a Z axis in units of depth (e.g., for sigma coordinate fields) and converting from a Z axes of depth to one of density (for a stably stratified fluid).

Argument 1, V, is the field of data values, say temperature on the "source" Z-axis, say, layer number. The second argument, ZVALS, contains values in units of the desired destination Z axis (ZAX) on the same Z axis as V — for example, depth values associated with each vertical layer. The third argument, ZAX, is any variable defined on the destination Z axis, often "Z[gz=zaxis_name]" is used.

The ZAXREPLACE function takes three arguments. The first argument, V, is the field of data values, say temperature or salinity. This variable is available on what we will refer to as the "source" Z-axis -- say in terms of layer number. The second argument, ZVALS, contains the values of the desired destination Z axis defined on the source Z axis -- for example, it may contain the depth values associated with each vertical layer. It should always share the Z axis from the first argument. The third argument, ZAX, is defined on the destination Z axis. Only the Z axis of this variable is relevant -- the values of the variable, itself, and its structure in X, Y, and T are ignored. Often "Z[gz=zaxis_name]" is used for the third argument.

Note:

ZAXREPLACE is a "grid-changing" function; its output grid is different from the input arguments. Therefore it is best to use explicit limits on the arguments rather than a SET REGION command. (See p. 69)

An example of the use of ZAXREPLACE for sigma coordinates is outlined in the FAQ on Using Sigma Coordinates.

Another example:

Contour salt as a function of density:

```
yes? set dat levitus_climatology

! Define density sigma, then density axis axden
yes? let sigma=rho_un(salt,temp,0)-1000
yes? define axis/z=21:28:.05 axden

! Regrid to density
yes? let saltonsigma= ZAXREPLACE( salt, sigma, z[gz=axden])

! Make Pacific plot
yes? fill/y=0/x=120e:75w/vlimits=28:21:-1 saltonsigma
```

Note that one could regrid the variable in the third argument to the destination Z axis using whichever of the regridding transformations that is best for the analysis, e.g. z[gz=axdens@AVE]

Ch3 Sec2.3.25. XSEQUENCE, YSEQUENCE, ZSEQUENCE, TSEQUENCE

XSEQUENCE(A), YSEQUENCE(A), ZSEQUENCE(A), TSEQUENCE(A) Unravels the data from the argument into a 1-dimensional line of data on an ABSTRACT axis.

Note:

This family of functions are "grid-changing" functions; the output grid is different from the input arguments. Therefore it is best to use explicit limits on the argument rather than a SET REGION command. (See p. 69)

Ch3 Sec2.3.26. FFTA

FFTA(A) Computes Fast Fourier Transform amplitude spectra, normalized by 1/N

Arguments:	A	Variable with regular time axis.
Result Axes:	X	Inherited from A
	Y	Inherited from A
	Z	Inherited from A
	T	Generated by the function: frequency in cyc/(time units from A)

See the demonstration script ef_fft_demo.jnl for an example using this function. Also see the external functions fft_re, fft_im, and fft_inverse for more options using FFT's

FFTA returns a(j) in

f(t) = $_{(j=1 \text{ to } N/2)}$ (j) cos(j t + (j))]

where [ ] means "integer part of", =2 pi/T is the fundamental frequency, and T=N* t is the time span of the data input to FFTA. is the phase (returned by FFTP, see next section)

The units of the returned time axis are "cycles/ t" where t is the time unit of the input axis. The Nyquist frequency is yquist = 1./(2.*boxsize), and the frequency axis runs from freq1 = yquist/ float(nfreq) to freqn = yquist

Even and odd N's are allowed. N need not be a power of 2. FFTA and FFTP assume f(1)=f(N+1), and the user gives the routines the first N pts.

Specifying the context of the input variable explicitly e.g.

```
LIST FFTA(A[l=1:58])
```

will prevent any confusion about the region. See the note in chapter 3 (p. 77)on the context of variables passed to functions.

The code is based on the FFT routines in Swarztrauber's FFTPACK available at www.netlib.org. For further discussion of the FFTPACK code, please see the document, Notes on FFTPACK - A Package of Fast Fourier Transform Programs at http://ferret.pmel.noaa.gov/Ferret/Documentation/FFTpack_notes/FFTPACK_notes.html

Ch3 Sec2.3.27. FFTP

FFTP(A) Computes Fast Fourier Transform phase

Arguments:	A	Variable with regular time axis.
Result Axes:	X	Inherited from A
	Y	Inherited from A
	Z	Inherited from A
	T	Generated by the function: frequency in cyc/(time units from A)

See the demonstration script ef_fft_demo.jnl for an example using this function.

FFTP returns (j) in

f(t) = $_{(j=1 \text{ to } N/2)}$ (j) cos(j t + (j))]

where [ ] means "integer part of", =2 pi/T is the fundamental frequency, and T=N* t is the time span of the data input to FFTA.

The units of the returned time axis are "cycles/ t" where t is the time increment. The Nyquist frequency is yquist = 1./(2.*boxsize), and the frequency axis runs from freq1 = yquist/ float(nfreq) to freqn = yquist

Even and odd N's are allowed. Power of 2 not required. FFTA and FFTP assume f(1)=f(N+1), and the user gives the routines the first N pts.

Specifying the context of the input variable explicitly e.g.

```
LIST FFTA(A[l=1:58])
```

will prevent any confusion about the region. See the note in chapter 3 (p. 77)on the context of variables passed to functions.

The code is based on the FFT routines in Swarztrauber's FFTPACK available at www.netlib.org. See the section on FFTA for more discussion (p. 86). For further discussion of the FFTPACK code, please see the document, Notes on FFTPACK - A Package of Fast Fourier Transform Programs at http://ferret.pmel.noaa.gov/Ferret/Documentation/FFTpack_notes/FFTPACK_notes.html

Ch3 Sec2.3.28. SAMPLEI

SAMPLEI(TO_BE_SAMPLED,X_INDICES) samples a field at a list of X indices, which are a subset of its X axis

Arguments:	TO_BE_SAMPLED	Data to sample
	X_INDICES	list of indices of the variable TO_BE_SAMPLED
Result Axes:	X	ABSTRACT; length same as X_INDICES
	Y	Inherited from TO_BE_SAMPLED
	Z	Inherited from TO_BE_SAMPLED
	T	Inherited from TO_BE_SAMPLED

See the demonstration ef_sort_demo.jnl for a common useage of this function. As with other functions which change axes (see p. 69), specify any region information for the variable TO_BE_SAMPLED explicitly in the function call, e.g.

```
yes? LET sampled_data = samplei(airt[X=160E:180E], xindices)
```

Ch3 Sec2.3.29. SAMPLEJ

SAMPLEJ(TO_BE_SAMPLED,Y_INDICES) samples a field at a list of Y indices, which are a subset of its Y axis

Arguments:	TO_BE_SAMPLED	Data to be sample
	Y_INDICES	list of indices of the variable TO_BE_SAMPLED
Result Axes:	X	Inherited from TO_BE_SAMPLED
	Y	ABSTRACT; length same as Y_INDICES
	Z	Inherited from TO_BE_SAMPLED
	T	Inherited from TO_BE_SAMPLED

See the demonstration ef_sort_demo.jnl for a common useage of this function. As with other functions which change axes(see p. 69), specify any region information for the variable TO_BE_SAMPLED explicitly in the function call.

Ch3 Sec2.3.30. SAMPLEK

SAMPLEK(TO_BE_SAMPLED, Z_INDICES) samples a field at a list of Z indices, which are a subset of its Z axis

Arguments:	TO_BE_SAMPLED	Data to sample
	Z_INDICES	list of indices of the variable TO_BE_SAMPLED
Result Axes:	X	Inherited from TO_BE_SAMPLED
	Y	Inherited from TO_BE_SAMPLED
	Z	ABSTRACT; length same as Z_INDICES
	T	Inherited from TO_BE_SAMPLED

See the demonstration ef_sort_demo.jnl for a common useage of this function. As with other functions which change axes(see p. 69), specify any region information for the variable TO_BE_SAMPLED explicitly in the function call.

Ch3 Sec2.3.31. SAMPLEL

SAMPLEL(TO_BE_SAMPLED, T_INDICES) samples a field at a list of T indices, a subset of its T axis

Arguments:	TO_BE_SAMPLED	Data to sample
	T_INDICES	list of indices of the variable TO_BE_SAMPLED
Result Axes:	X	Inherited from TO_BE_SAMPLED
	Y	Inherited from TO_BE_SAMPLED
	Z	Inherited from TO_BE_SAMPLED
	T	ABSTRACT; length same as X_INDICES

See thedemonstration ef_sort_demo.jnl for a common useage of this function. As with other functions which change axes (see p. 69), specify any region information for the variable TO_BE_SAMPLED explicitly in the function call.

Ch3 Sec2.3.32. SAMPLEIJ

SAMPLEIJ(DAT_TO_SAMPLE,XPTS,YPTS) Returns data sampled at a subset of its grid points, defined by (XPTS, YPTS)

Arguments:	DAT_TO_SAMPLE	Data to sample, field of x, y, and perhaps z and t
	XPTS	X coordinates of grid points to sample
	YPTS	Y coordinates of grid points to sample
Result Axes:	X	ABSTRACT, length of list (xpts,ypts)
	Y	NORMAL (no axis)
	Z	Inherited from DAT_TO_SAMPLE
	T	Inherited from DAT_TO_SAMPLE

This is a discrete version of SAMPLEXY. The points defined in arguments 2 and 3 are coordinaets, but a result is returned only if those arguements are a match with coordinates of the grid of the variable.

As with other functions which change axes (see p. 69), specify any region information for the variable TO_BE_SAMPLED explicitly in the function call.

Ch3 Sec2.3.33. SAMPLET_DATE

SAMPLET_DATE (DAT_TO_SAMPLE, YR, MO, DAY, HR, MIN, SEC) Returns data sampled by interpolating to one or more times

Arguments:	DAT_TO_SAMPLE	Data to sample, field of x, y, z and t
	YR	Year(s), integer YYYY
	MO	Month(s), integer month number MM
	DAY	Day(s) of month, integer DD
	HR	Hour(s) integer HH
	MIN	Minute(s), integer MM
	SEC	Second(s), integer SS
Result Axes:	X	Inherited from DAT_TO_SAMPLE
	Y	Inherited from DAT_TO_SAMPLE
	Z	Inherited from DAT_TO_SAMPLE
	T	ABSTRACT; length is # times sampled. The length is determined from the length of argument 2.

As with other functions which change axes (see p. 69), specify any region information for the variable DAT_TO_SAMPLE explicitly in the function call.

Example:

List wind speed at a subset of points from the monthly navy winds data set. To choose times all in the single year 1985, we can define a variable year = constant + 0*month, the same length as month but with all 1985 data.

```
yes? use monthly_navy_winds

yes? set reg/x=131:139/y=29
yes? let month = {1,5,11}
yes? let day = {20,20,20}
yes? let year = 1985 + 0*month
yes? let zero = 0*month

yes? list samplet_date(uwnd, year, month, day, zero, zero, zero)
             VARIABLE : SAMPLET_DATE(UWND, YEAR, MONTH, DAY, ZERO, ZERO,
ZERO)
             FILENAME : monthly_navy_winds.cdf
             FILEPATH : /home/ja9/tmap/fer_dsets/data/
             SUBSET   : 5 by 3 points (LONGITUDE-T)
             LATITUDE : 30N
            130E   132.5E 135E   137.5E 140E
             45     46     47     48     49
 1   / 1: -0.771 -0.201  1.017  2.282  3.192
 2   / 2: -2.643 -2.670 -2.409 -2.052 -1.503
 3   / 3:  1.082  1.280  1.722  2.133  2.417
```

Ch3 Sec2.3.34. SAMPLEXY

SAMPLEXY(DAT_TO_SAMPLE,XPTS,YPTS) Returns data sampled at a set of (X,Y) points, using linear interpolation.

Arguments:	DAT_TO_SAMPLE	Data to sample
	XPTS	X values of sample points
	YPTS	Y values of sample points
Result Axes:	X	ABSTRACT; length same as XPTSand YPTS
	Y	NORMAL (no axis)
	Z	Inherited from DAT_TO_SAMPLE
	T	Inherited from DAT_TO_SAMPLE

Note:

SAMPLEXY is a "grid-changing" function; its output grid is different from the input arguments. Therefore it is best to use explicit limits on the first argument rather than a SET REGION command. (See p. 69)

Examples:

1) See the the script vertical_section.jnl to create a section along a line between two (lon,lat) locations; this script calls SAMPLEXY.

2) Use SAMPLEXY to extract a section of data taken along a slanted line in the Pacific.

First we generate the locations xlon, ylat (Figure3_3a). One could use a ship track, specifying its coordinates as xlon, ylat.

```
yes? USE levitus_climatology

     ! define the slant line through (234.5,24.5)
     ! and with slope -24./49

yes? LET xlon = 234.5 + (I[I=1:50]-1)
yes? LET slope = -1*24./49
yes? LET ylat = 24.5 + slope*(i[i=1:50] -1)

yes? PLOT/VS/LINE/SYM=27 xlon,ylat      ! line off Central America
yes? GO land
```

Now sample the field "salt" along this track and make a filled contour plot. The horizontal axis is abstract; it is a count of the number of points along the track. To speed the calculation, or if we otherwise want to restrict the region used on the variable salt, put that information in explicit limits on the first argument. (Figure3_3b)

```
yes? LET slantsalt = samplexy(salt[x=200:300,y=0:30],xlon,ylat)
yes? FILL/LEVELS=(33.2,35.2,0.1)/VLIMITS=0:4000 slantsalt
```

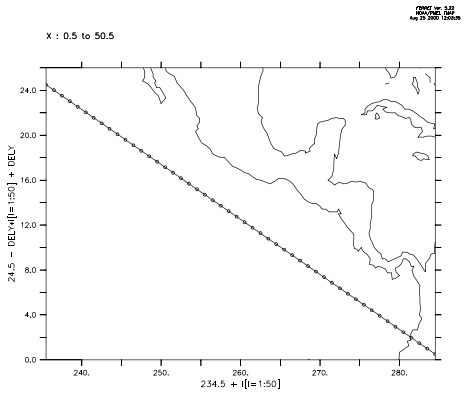


Figure3_3a

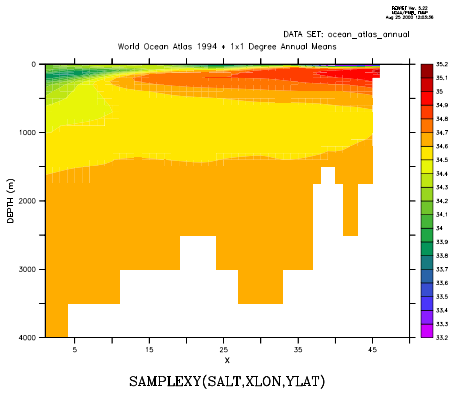


Figure3_3b

Ch3 Sec2.3.35. SAMPLEXY_CLOSEST

SAMPLEXY_CLOSEST(DAT_TO_SAMPLE,XPTS,YPTS) Returns data sampled at a set of (X,Y) points, using nearest grid intersection.

Arguments:	DAT_TO_SAMPLE	Data to sample
	XPTS	X values of sample points
	YPTS	Y values of sample points
Result Axes:	X	ABSTRACT; length same as XPTSand YPTS
	Y	NORMAL (no axis)
	Z	Inherited from DAT_TO_SAMPLE
	T	Inherited from DAT_TO_SAMPLE

Note:SAMPLEXY_CLOSEST is a "grid-changing" function; its output grid is different from the input arguments. Therefore it is best to use explicit limits on the first argument rather than a SET REGION command. (See p. 69)

This function is a quick-and-dirty substitute for the SAMPLEXY function. It runs much faster than SAMPLEXY, since it does no interpolation. It returns the function value at the grid point closest to each point in XPTS, YPTS. It is less accurate than SAMPLEXY, but may give adequate results in a much shorter time for large samples.

```
Example: compare with SAMPLEXY output

yes? USE levitus_climatology
yes? LET xlon = 234.5 + I[I=1:20]
yes? LET dely = 24./19
yes? LET ylat = 24.5 - dely*i[i=1:20] + dely

yes? LET a = samplexy(salt[X=200:300,Y=0:30,K=1], xlon, ylat)
yes? LET b = samplexy_closest(salt[X=200:300,Y=0:30,K=1], xlon, ylat)
```

```
yes? LIST a, b
             DATA SET:"./fer_dsets/descr/levitus_climatology.cdf
             Levitus annual climatology (1x1 degree)
             X: 0.5 to 20.5
             DEPTH (m): 0
             TIME: 02-JUL      14:54
 Column  1: A is SAMPLEXY(SALT[X=200:300,Y=0:30,K=1],XLON,YLAT)
 Column  2: B is SAMPLEXY_CLOSEST(SALT[X=200:300,Y=0:30,K=1],XLON,YLAT)
                A      B
1    /  1:  34.22  34.22
2    /  2:  34.28  34.26
3    /  3:  34.35  34.39
4    /  4:  34.41  34.43
5    /  5:  34.44  34.44
6    /  6:  34.38  34.40
7    /  7:  34.26  34.22
8    /  8:  34.09  34.07
9    /  9:  33.90  33.92
10   / 10:  33.74  33.78
11   / 11:  33.64  33.62
12   / 12:  33.63  33.62
13   / 13:  33.69  33.67
14   / 14:  33.81  33.75
15   / 15:  33.95  34.00
16   / 16:  34.11  34.11
17   / 17:  34.25  34.22
18   / 18:  34.39  34.33
19   / 19:  34.53  34.56
20   / 20:  34.65  34.65
```

Ch3 Sec2.3.36. SAMPLEXY_CURV

SAMPLEXY_CURV Returns data which is on a curvilinear grid, sampled at a set of (X,Y) points, using interpolation.

Arguments:	DAT_TO_SAMPLE	Data to sample
	DAT_LON	Longitude coordinates of the curvilinear grid
	DAT_LAT	Latitude coordinates of the curvilinear grid
	XPTS	X values of sample points
	YPTS	Y values of sample points
Result Axes:	X	ABSTRACT; length same as XPTSand YPTS
	Y	NORMAL (no axis)
	Z	Inherited from DAT_TO_SAMPLE
	T	Inherited from DAT_TO_SAMPLE

Note:SAMPLEXY_CURV is a "grid-changing" function; its output grid is different from the input arguments. Therefore it is best to use explicit limits on the first argument rather than a SET REGION command. (See p. 69)

Ch3 Sec2.3.37. SCAT2GRIDGAUSS_XY

SCAT2GRIDGAUSS_XY(XPTS, YPTS, F, XCOORD, YCOORD, XSCALE, YSCALE, CUTOFF, 0) Use Gaussian weighting to grid scattered data to an XY grid

Arguments:	XPTS	x-coordinates of scattered input triples, listed along an abstract X or Y axis. May be fcn of Z or time
	YPTS	y-coordinates of scattered input triples, listed along an abstract X or Y axis. May be fcn of Z or time
	F	F(X,Y) 3rd component of scattered input triples, listed along an abstract X or Y axis. May be fcn of Z or time
	XAXPTS	coordinates of X-axis of output grid. Must be regularly spaced.
	YAXPTS	coordinates of Y-axis of output grid. Must be regularly spaced.
	XSCALE	Mapping scale for Gaussian weights in Y direction, in data units (e.g. lon or m). See the discussion below.
	YSCALE	Mapping scale for Gaussian weights in Y direction, in data units (e.g. lat or m)
	CUTOFF	Cutoff for weight function. Only scattered points within CUTOFF*XSCALE and CUTOFF*YSCALE of the grid box center are included in the sum for the grid box.
	0	An unused argument: previously there had been a second "cutoff" argument but it was redundant. There is only one cutoff in the algorithm once the XSCALE and YSCALE mapping is applied. In future versions of Ferret this argument will be removed.
Result Axes:	X	Inherited from XAXPTS
	Y	Inherited from YAXPTS
	Z	Inherited from F
	T	Inherited from F

Note:

The SCAT2GRIDGAUSS functions are "grid-changing" functions; the output grid is different from the input arguments. Therefore it is best to use explicit limits on any of the arguments rather than a SET REGION command. (See p. 69)

Quick example:

```
yes? DEFINE AXIS/X=180:221:1 xax
yes? DEFINE AXIS/Y=-30:10:1 yax
yes? ! read some data
yes? SET DATA/EZ/VARIABLES="times,lats,lons,var" myfile.dat
yes? LET my_out = SCAT2GRIDGAUSS_XY(lons, lats, var, x[gx=xax],
y[gy=yax], 2, 2, 2, 0)
```

If the output X axis is a modulo longitude axis, then the scattered X values should lie within the range of the actual coordinates of the axis. That is, if the scattered X values are xpts={355, 358, 2, 1, 352, 12} and the coordinates of the X axis you wish to grid to are longitudes of x=20,23,25,...,379 then you should apply a shift to your scattered points:

```
yes? USE levitus_climatology  ! output will be on the grid of SALT
yes? LET xx = x[gx=salt]
yes? LET x1 = if lons lt `xx[x=@min]` then lons+360
yes? LET xnew = if x1 gt `xx[x=@max] then x1-360 else x1
yes? LET my_out = SCAT2GRIDGAUSS_XY(xnew, ypts, var, x[gx=xax],
y[gy=yax], 2, 2, 2, 0)
```

The SCAT2GRIDGAUSS* functions use 1a Gaussian interpolation method to map irregular locations (x_n, y_n) to a regular grid (x_0, y_0). The output grid must have equally-spaced gridpoints in both the x and y directions. For examples of the gridding functions, run the script ***objective_analysis_demo,*** or see the on-line demonstration

http://www.ferret.noaa.gov/Ferret/Demos/objective_analysis_demo/objective_analysis_demo.html

Parameters for a square grid and a fairly dense distribution of scattered points relative to the grid might be XSCALE=YSCALE = 0.5, and CUTOFF = 2. To get better coverage, use a coarser grid or increase XSCALE, YSCALE and/or CUTOFF.

The value of the gridded function F at each grid point (x_0, y_0) is computed by:

$$F(x_0,y_0) = \sum_{n=1 \text{ to } Np} F(x_n,y_n)W(x_n,y_n) \Big/ \sum_{n=1 \text{ to } Np} W(x_n,y_n)$$

Where Np is the total number of irregular points within the "influence region" of a particular grid point, (determined by the CUTOFF parameter, defined below). The Gaussian weight fucntion W_n is given by

$$W_n(x_n,y_n) = \exp\{-[(x_n-x_0)^2/(X)^2 + (y_n-y_0)^2/(Y)^2]\}$$

X and Y in the denominators on the right hand side are the mapping scales, arguments XSCALE and YSCALE.

The weight function has a nonzero value everywhere, so all of the scattered points in theory could be part of the sum for each grid point. To cut computation, the parameter CUTOFF is employed. If a cutoff of 2 is used (e.g. CUTOFF* XSCALE=2), then the weight function is set to

zerowhen $W_n < e^{-4}$. This occurs where distances from the grid point are less than 2 times the mapping scales X or Y.

(Reference for this method: Kessler and McCreary, 1993: The Annual Wind-driven Rossby Wave in the Subthermocline Equatorial Pacific, Journal of Physical Oceanography 23, 1192 -1207)

Ch3 Sec2.3.38. SCAT2GRIDGAUSS_XZ

SCAT2GRIDGAUSS_XZ(XPTS, ZPTS, F, XAXPTS, ZAXPTS, XSCALE, ZSCALE, CUTOFF, 0) Use Gaussian weighting to grid scattered data to an XZ grid

Arguments:	XPTS	x-coordinates of scattered input triples, listed along an abstract X or Z axis. May be fcn of Y or time
	ZPTS	z-coordinates of scattered input triples, listed along an abstract X or Z axis. May be fcn of Y or time
	F	F(X,Z) 3rd component of scattered input triples, listed along an abstract X or Z axis. May be fcn of Y or time
	XAXPTS	coordinates of X-axis of output grid. Must be regularly spaced.
	ZAXPTS	coordinates of Z-axis of output grid. Must be regularly spaced.
	XSCALE	Mapping scale for Gaussian weights in Y direction, in data units (e.g. lon or m). See the discussion under SCAT2GRIDGAUSS_XY.
	ZSCALE	Radius of influence in the Z direction, in data units (e.g. m or km)
	CUTOFF	Cutoff for weight function in the X direction. Only scattered points within CUTOFF*XSCALE and CUTOFF*ZSCALE of the grid box center are included in the sum for the grid box.
	0	An unused argument: previously there had been a second "cutoff" argument but it was redundant. There is only one cutoff in the algorithm once the XSCALE and ZSCALE mapping is applied. In future versions of Ferret this argument will be removed.
Result Axes:	X	Inherited from XAXPTS
	Y	Inherited from F
	Z	Inherited from ZAXPTS

	T	Inherited from F

See the description under SCAT2GRIDGAUSS_XY (p. 95). Note that The output grid must have equally-spaced gridpoints in both the x and z directions. For examples of the gridding functions, run the script ***objective_analysis_demo,*** or see the on-line demonstration

http://www.ferret.noaa.gov/Ferret/Demos/objective_analysis_demo/objective_analysis_demo.html

Ch3 Sec2.3.39. SCAT2GRIDGAUSS_YZ

SCAT2GRIDGAUSS_YZ(YPTS, zPTS, F, YAXPTS, ZAXPTS, YSCALE, ZSCALE, CUTOFF, 0) Use Gaussian weighting to grid scattered data to a YZ grid

Arguments:	YPTS	y-coordinates of scattered input triples, listed along an abstract Y or Z axis. May be function of X or time.
	ZPTS	z-coordinates of scattered input triples, listed along an abstract Y or Z axis. May be function of X or time.
	F	F(Y,Z) 3rd component of scattered input triples, listed along an abstract Y or Z axis. May be function of X or time.
	YAXPTS	coordinates of Y-axis of output grid. Must be regularly spaced.
	ZAXPTS	coordinates of Z-axis of output grid. Must be regularly spaced.
	YSCALE	Mapping scale for Gaussian weights in Y direction, in data units (e.g. lat or m). See the discussion under SCAT2GRIDGAUSS_XY.
	ZSCALE	Radius of influence in the Z direction, in data units (e.g. m or km)
	CUTOFF	Cutoff for weight function in the Y direction. Only scattered points within CUTOFF*YSCALE and CUTOFF*ZSCALE of the grid box center are included in the sum for the grid box.
	0	An unused argument: previously there had been a second "cutoff" argument but it was redundant. There is only one cutoff in the algorithm once the YSCALE and ZSCALE mapping is applied. In future versions of Ferret this argument will be removed.

Result Axes:	X	Inherited from F
	Y	Inherited from YAXPTS
	Z	Inherited from ZAXPTS
	T	Inherited from F

See the description under SCAT2GRIDGAUSS_XY (p. 95). Note that the output grid must have equally-spaced gridpoints in both the y and z directions. For examples of the gridding functions, run the script ***objective_analysis_demo,*** or see the on-line demonstration

http://www.ferret.noaa.gov/Ferret/Demos/objective_analysis_demo/objective_analysis_demo.html

Ch3 Sec2.3.40. SCAT2GRIDLAPLACE_XY

SCAT2GRIDLAPLACE_XY(XPTS, YPTS, F, XAXPTS, YAXPTS, CAY, NRNG) Use Laplace/ Spline interpolation to grid scattered data to an XY grid.

Arguments:	XPTS	x-coordinates of scattered input triples, listed along an abstract X or Y axis. May be fcn of Z or time.
	YPTS	y-coordinates of scattered input triples, listed along an abstract X or Y axis. May be fcn of Z or time.
	F	F(X,Y) 3rd component of scattered input triples, listed along an abstract X or Y axis. May be fcn of Z or time.
	XAXPTS	coordinates of X-axis of output grid. Must be regularly spaced.
	YAXPTS	coordinates of Y-axis of output grid. Must be regularly spaced.
	CAY	Amount of spline eqation (between 0 and inf.) vs Laplace interpolation
	NRNG	Grid points more than NRNG grid spaces from the nearest data point are set to undefined.
Result Axes:	X	Inherited from XAXPTS
	Y	Inherited from YAXPTS
	Z	Inherited from F
	T	Inherited from F

Note:

The SCAT2GRIDLAPLACE functions are "grid-changing" functions; the output grid is different from the input arguments. Therefore it is best to use explicit limits on any of the arguments rather than a SET REGION command. (See p. 69)

Quick example:

```
yes? DEFINE AXIS/X=180:221:1 xax
yes? DEFINE AXIS/Y=-30:10:1 yax
yes? ! read some data
yes? SET DATA/EZ/VARIABLES="times,lats,lons,var" myfile.dat
yes? LET my_out = SCAT2GRIDLAPLACE_XY(lons, lats, var, x[gx=xax],
y[gy=yax], 2., 5)
yes? SHADE my_out
```

If the output X axis is a modulo longitude axis, then the scattered X values should lie within the range of the actual coordinates of the axis. That is, if the scattered X values are xpts={355, 358, 2, 1, 352, 12} and the coordinates of the X axis you wish to grid to are longitudes of x=20,23,25,...,379 then you should apply a shift to your scattered points:

```
yes? USE levitus_climatology  ! output will be on the grid of SALT
yes? LET xx = x[gx=salt]
yes? LET x1 = if lons lt `xx[x=@min]` then lons+360
yes? LET xnew = if x1 gt `xx[x=@max] then x1-360 else x1
yes? LET my_out = SCAT2GRIDLAPLACE_XY(xnew, lats, var, x[gx=xax],
y[gy=yax], 2., 5)
```

For examples of the gridding functions, run the script ***objective_analysis_demo,*** or see the on-line demonstration

http://www.ferret.noaa.gov/Ferret/Demos/objective_analysis_demo/objective_analysis_demo.html

The SCAT2GRIDLAPLACE* functions employ the same interpolation method as is used by PPLUS, and appears elsewhere in Ferret, e.g. in contouring. The parameters are used as follows (quoted from the PPLUS Users Guide. A reference for this is "Plot Plus, a Scientific Graphics System", written by Donald W. Denbo, April 8, 1987.):

CAY
If CAY=0.0, Laplacian interpolation is used. The resulting surface tends to have rather sharp peaks and dips at the data points (like a tent with poles pushed up into it). There is no chance of spurious peaks appearing. As CAY is increased, Spline interpolation predominates over the Laplacian, and the surface passes through the data points more smoothly. The possibility of spurious peaks increases with CAY. CAY= infinity is pure Spline interpolation. An over relaxation process in used to perform the interpolation. A value of CAY=5 often gives a good surface.

NRNG
Any grid points farther than NRNG away from the nearest data point will be set to "undefined"
The default used by PPLUS is NRNG = 5

Ch3 Sec2.3.41. SCAT2GRIDLAPLACE_XZ

SCAT2GRIDLAPLACE_XZ(XPTS, ZPTS, F, XAXPTS, ZAXPTS, CAY, NRNG) Use Laplace/ Spline interpolation to grid scattered data to an XZ grid.

Arguments:	XPTS	x-coordinates of scattered input triples, listed along an abstract X or Z axis. May be function of Y or time.
	ZPTS	z-coordinates of scattered input triples, listed along an abstract X or Z axis. May be function of Y or time.
	F	F(X,Z) 3rd component of scattered input triples, listed along an abstract X or Z axis. May be function of Y or time.
	XAXPTS	coordinates of X-axis of output grid. Must be regularly spaced.
	ZAXPTS	coordinates of Z-axis of output grid. Must be regularly spaced.
	CAY	Amount of spline eqation (between 0 and inf.) vs Laplace interpolation
	NRNG	Grid points more than NRNG grid spaces from the nearest data point are set to undefined.
Result Axes:	X	Inherited from XAXPTS
	Y	Inherited from F
	Z	Inherited from ZAXPTS
	T	Inherited from F

The gridding algorithm is discussed under SCAT2GRIDLAPLACE_XY (p. 101). For examples of the gridding functions, run the script ***objective_analysis_demo,*** or see the on-line demonstration

http://www.ferret.noaa.gov/Ferret/Demos/objective_analysis_demo/objective_analysis_demo.html

Ch3 Sec2.3.42. SCAT2GRIDLAPLACE_YZ

SCAT2GRIDLAPLACE_YZ(YPTS, ZPTS, F, YAXPTS, ZAXPTS, CAY, NRNG) Use Laplace/ Spline interpolation to grid scattered data to an YZ grid.

Arguments:	YPTS	y-coordinates of scattered input triples, listed along an abstract Y or Z axis. May be fcn of X or time.
	ZPTS	z-coordinates of scattered input triples, listed along an abstract Y or Z axis. May be fcn of X or time.
	F	F(Y,Z) 3rd component of scattered input triples, listed along an abstract Y or Z axis. May be fcn of X or time.
	YAXPTS	coordinates of Y-axis of output grid. Must be regularly spaced.
	ZAXPTS	coordinates of Z-axis of output grid. Must be regularly spaced.
	CAY	Amount of spline eqation (between 0 and inf.) vs Laplace interpolation
	NRNG	Grid points more than NRNG grid spaces from the nearest data point are set to undefined.
Result Axes:	X	Inherited from F
	Y	Inherited from YAXPTS
	Z	Inherited from ZAXPTS
	T	Inherited from F

The gridding algorithm is discussed under SCAT2GRIDLAPLACE_XY (p. 102). For examples of the gridding functions, run the script ***objective_analysis_demo,*** or see the on-line demonstration

http://www.ferret.noaa.gov/Ferret/Demos/objective_analysis_demo/objective_analysis_demo.html

Ch3 Sec2.3.43. SORTI

SORTI(DAT): Returns indices of data, sorted on the I axis in increasing order

Arguments:	DAT	DAT: variable to sort
Result Axes:	X	ABSTRACT, same length as DAT x-axis
	Y	Inherited from DAT
	Z	Inherited from DAT
	T	Inherited from DAT

SORTI, SORTJ, SORTK, and SORTL return the indices of the data after it has been sorted. These functions are used in conjunction with functions such as the SAMPLE functions to do sorting and sampling. See the demonstration ef_sort_demo.jnl for common useage of these functions.

As with other functions which change axes (see p. 69), specify any region information for the variable DAT explicitly in the function call.

Ch3 Sec2.3.44. SORTJ

SORTJ(DAT) Returns indices of data, sorted on the I axis in increasing order

Arguments:	DAT	DAT: variable to sort
Result Axes:	X	Inherited from DAT
	Y	ABSTRACT, same length as DAT y-axisInherited from DAT
	Z	Inherited from DAT
	T	Inherited from DAT

See discussion under SORTI

Ch3 Sec2.3.45. SORTK

SORTK(DAT) Returns indices of data, sorted on the I axis in increasing order

Arguments:	DAT	DAT: variable to sort
Result Axes:	X	Inherited from DAT
	Y	Inherited from DAT
	Z	ABSTRACT, same length as DAT x-axis
	T	Inherited from DAT

See the discussion under SORTI

Ch3 Sec2.3.46. SORTL

SORTL(DAT) Returns indices of data, sorted on the L axis in increasing order

Arguments:	DAT	DAT: variable to sort
Result Axes:	X	Inherited from DAT
	Y	Inherited from DAT
	Z	Inherited from DAT
	T	ABSTRACT, same length as DAT x-axis

See the discussion under SORTI

Ch3 Sec2.3.47. TAUTO_COR

TAUTO_COR(A): Compute autocorrelation function (ACF) of time series, lags of 0,...,N-1, where N is the length of the time axis.

Arguments:	A	A function of time, and perhaps x,y,z
Result Axes:	X	Inherited from A
	Y	Inherited from A
	Z	Inherited from A
	T	ABSTRACT, same length as A time axis (lags)

Note:

TAUTO_COR is a "grid-changing" function; its output grid is different from the input arguments. Therefore it is best to use explicit limits on the first argument rather than a SET REGION command. (See p. 69)

Ch3 Sec2.3.48. XAUTO_COR

XAUTO_COR(A): Compute autocorrelation function (ACF) of a series in X, lags of 0,...,N-1, where N is the length of the x axis.

Arguments:	A	A function of x, and perhaps y,z,t
Result Axes:	X	ABSTRACT, same length as X axis of A (lags)
	Y	Inherited from A
	Z	Inherited from A
	T	Inherited from A

Note:

XAUTO_COR is a "grid-changing" function; its output grid is different from the input arguments. Therefore it is best to use explicit limits on the first argument rather than a SET REGION command. (See p. 69)

Ch3 Sec2.3.49. TAX_DATESTRING

TAX_DATESTRING(A, B, C): Returns date string for time axis coordinate values

Arguments:	A	time steps to convert
	B	variable with reference time axis; use region settings to restrict this to a small region (see below)
	C	output precision (STRING), "second", "minute", "hour", "day", "month", or "year"
Result Axes:	X	normal
	Y	normal
	Z	normal
	T	Inherited from A

NOTE: The variable given for argument 2 is loaded into memory by this function, so restrict its range in X, Y, and/or Z so that it is not too large. Or, if you are getting the time steps from the variable, specify the time steps for both arguments.

Examples:

```
yes? use
"http://ferret.pmel.noaa.gov/cgi-bin/dods/nph-dods/data/PMEL/COADS/coads
_sst.nc"

yes? let tt = t[gt=sst]
yes? let tvar = sst[x=180,y=0]

yes? L=1400:1405 tax_datestring(tt,tvar,"hour")
```

```
             VARIABLE : TAX_DATESTRING(TT,TVAR,"hour")
             DATA SET : COADS Surface Marine Observations (1854-1993)
             FILENAME : coads_sst.nc
             FILEPATH :
http://ferret.pmel.noaa.gov/cgi-bin/dods/nph-dods/data/PMEL/COADS/
             SUBSET   : 6 points (TIME)
 15-AUG-1970 / 1400:"15-AUG-1970 00:00"
 15-SEP-1970 / 1401:"15-SEP-1970 00:00"
 15-OCT-1970 / 1402:"15-OCT-1970 00:00"
 15-NOV-1970 / 1403:"15-NOV-1970 00:00"
 15-DEC-1970 / 1404:"15-DEC-1970 00:00"
 15-JAN-1971 / 1405:"15-JAN-1971 00:00"
```

If the timesteps are specified based on the variable, we can specify the time steps instead of the "variable with reference time axis", as follows":

```
yes? list/t=15-aug-1970:15-jan-1971 tax_datestring(t[gt=sst], t[gt=sst],
"day")
...
 15-AUG-1970 / 1400:"15-AUG-1970"
 15-SEP-1970 / 1401:"15-SEP-1970"
 15-OCT-1970 / 1402:"15-OCT-1970"
 15-NOV-1970 / 1403:"15-NOV-1970"
 15-DEC-1970 / 1404:"15-DEC-1970"
 15-JAN-1971 / 1405:"15-JAN-1971"
```

Or, the times to list may not be steps chosen from the time axis. They can be any numbers within the range of the time steps on that axis.

```
yes? define axis/t=1-jan-2001:31-dec-2005:1/units=days/t0=31-dec-2000
dayaxis

yes? list tax_datestring({15.1,33.5,35.6}, t[gt=dayaxis], "minutes")

            VARIABLE : TAX_DATESTRING({15.1,33.5,35.6}, T[GT=DAYAXIS],
"minutes")
             SUBSET   : 3 points (X)
 1   / 1:"15-JAN-2001-2001 02:24"
 2   / 2:"02-FEB-2001 12:00"
 3   / 3:"04-FEB-2001 14:23"
```

Ch3 Sec2.3.50. TAX_DAY

TAX_DAY(A, B): Returns days of month of time axis coordinate values

Arguments:	A	time steps to convert
	B	variable with reference time axis; use region settings to restrict this to a small region (see below)
Result Axes:	X	normal
	Y	normal
	Z	normal

	T	Inherited from A

NOTE: The variable given for argument 2 is loaded into memory by this function, so restrict its range in X, Y, and/or Z so that it is not too large. Or, if you are getting the time steps from the variable, specify the time steps for both arguments. See the examples in the section on the TAX_DATESTRING function (p. 105)

Example:

```
yes? use
"http://ferret.pmel.noaa.gov/cgi-bin/dods/nph-dods/data/PMEL/reynolds_ss
t_wk.nc"

yes? list/t=1-feb-1982:15-mar-1982 tax_day(t[gt=wsst], wsst[i=1,j=1])

             VARIABLE : TAX_DAY(T[GT=WSST], WSST[I=1,J=1])
             DATA SET : Reynolds Optimum Interpolation Weekly SST
Analysis
             FILENAME : reynolds_sst_wk.nc
             FILEPATH :
http://ferret.pmel.noaa.gov/cgi-bin/dods/nph-dods/data/PMEL/
             SUBSET   : 7 points (TIME)
 02-FEB-1982 00 /  5:   2.00
 09-FEB-1982 00 /  6:   9.00
 16-FEB-1982 00 /  7:  16.00
 23-FEB-1982 00 /  8:  23.00
 02-MAR-1982 00 /  9:   2.00
 09-MAR-1982 00 / 10:   9.00
 16-MAR-1982 00 / 11:  16.00
```

Ch3 Sec2.3.51. TAX_DAYFRAC

TAX_DAYFRAC(A, B): Returns fraction of day of time axis coordinate values

Arguments:	A	time steps to convert
	B	variable with reference time axis; use region settings to restrict this to a small region (see below)
Result Axes:	X	normal
	Y	normal
	Z	normal
	T	Inherited from A

NOTE: The variable given for argument 2 is loaded into memory by this function, so restrict its range in X, Y, and/or Z so that it is not too large. Or, if you are getting the time steps from the variable, specify the time steps for both arguments. See the examples in the section on the TAX_DATESTRING function (p. 105)

Example: Show both the TAX_DATESTRING and TAX_DAYFRAC output. Because this is a climatological axis, there are no years in the dates.

```
yes? use coads_climatology
yes? let tt = t[gt=sst, L=1:5]
yes? let tvar = sst[x=180,y=0]

yes? list tax_datestring(tt, tvar, "minutes"), tax_dayfrac(tt,tvar)

             DATA SET:
/home/porter/tmap/ferret/linux/fer_dsets/data/coads_climatology.cdf
             TIME: 01-JAN 00:45 to 31-DEC 06:34
 Column  1: EX#1 is TAX_DATESTRING(TT,TVAR,"minutes")
 Column  2: EX#2 is TAX_DAYFRAC(TT,TVAR)
                                 EX#1    EX#2
16-JAN       /  1: "16-JAN      06:00"  0.2500
15-FEB       /  2: "15-FEB      16:29"  0.6869
17-MAR       /  3: "17-MAR      02:58"  0.1237
16-APR       /  4: "16-APR      13:27"  0.5606
16-MAY       /  5: "16-MAY      23:56"  0.9975
```

Ch3 Sec2.3.52. TAX_JDAY

TAX_JDAY(A, B): Returns day of year time axis coordinate values

Arguments:	A	time steps to convert
	B	variable with reference time axis; use region settings to restrict this to a small region (see below)
Result Axes:	X	normal
	Y	normal
	Z	normal
	T	Inherited from A

NOTE 1:The variable given for argument 2 is loaded into memory by this function, so restrict its range in X, Y, and/or Z so that it is not too large. Or, if you are getting the time steps from the variable, specify the time steps for both arguments. See the examples in the section on the TAX_DATESTRING function (p. 105)

NOTE 2: This function currently returns the date relative to a Standard (Gregorian) calendar. If the time axis is on a different calendar this is NOT taken into account. This will be fixed in a future Ferret release.

Example:

```
yes? list tax_jday(t[gt=sst], sst[i=1,j=1])
             VARIABLE : TAX_JDAY(T[GT=SST], SST[I=1,J=1])
             FILENAME : coads_climatology.cdf
             FILEPATH : /home/porter/tmap/ferret/linux/fer_dsets/data/
             SUBSET   : 12 points (TIME)
```

```
16-JAN      /  1:   16.0
15-FEB      /  2:   46.0
17-MAR      /  3:   77.0
16-APR      /  4:  107.0
16-MAY      /  5:  137.0
16-JUN      /  6:  168.0
16-JUL      /  7:  198.0
16-AUG      /  8:  229.0
15-SEP      /  9:  259.0
16-OCT      / 10:  290.0
15-NOV      / 11:  320.0
16-DEC      / 12:  351.0
```

Ch3 Sec2.3.53. TAX_MONTH

TAX_MONTH(A, B): Returns months of time axis coordinate values

Arguments:	A	time steps to convert
	B	variable with reference time axis; use region settings to restrict this to a small region (see below)
Result Axes:	X	normal
	Y	normal
	Z	normal
	T	Inherited from A

NOTE: The variable given for argument 2 is loaded into memory by this function, so restrict its range in X, Y, and/or Z so that it is not too large. Or, if you are getting the time steps from the variable, specify the time steps for both arguments. See the examples in the section on the TAX_DATESTRING function (p. 105)

Example:

```
yes? use
"http://ferret.pmel.noaa.gov/cgi-bin/dods/nph-dods/data/PMEL/reynolds_ss
t_wk.nc"
yes? list/t=18-jan-1982:15-mar-1982 tax_month(t[gt=wsst], wsst[i=1,j=1])

             VARIABLE : TAX_MONTH(T[GT=WSST], WSST[I=1,J=1])
             DATA SET : Reynolds Optimum Interpolation Weekly SST
Analysis
             FILENAME : reynolds_sst_wk.nc
             FILEPATH :
http://ferret.pmel.noaa.gov/cgi-bin/dods/nph-dods/data/PMEL/
             SUBSET   : 9 points (TIME)
 19-JAN-1982 00 /  3:  1.000
 26-JAN-1982 00 /  4:  1.000
 02-FEB-1982 00 /  5:  2.000
 09-FEB-1982 00 /  6:  2.000
 16-FEB-1982 00 /  7:  2.000
 23-FEB-1982 00 /  8:  2.000
 02-MAR-1982 00 /  9:  3.000
```

```
09-MAR-1982 00 / 10:  3.000
16-MAR-1982 00 / 11:  3.000
```

Ch3 Sec2.3.54. TAX_UNITS

TAX_UNITS(A): Returns units of time axis coordinate values, in seconds

Arguments:	A	variable with reference time axis
Result Axes:	X	normal
	Y	normal
	Z	normal
	T	Inherited from A

NOTE: to get the time units as a string use the RETURN=tunits keyword (p.)

Example:

```
yes? use
"http://ferret.pmel.noaa.gov/cgi-bin/dods/nph-dods/data/PMEL/reynolds_ss
t_wk.nc"
yes? say `wsst,return=tunits`
 !-> MESSAGE/CONTINUE DAYS
DAYS

yes? list tax_units(t[gt=wsst])

             VARIABLE : TAX_UNITS(T[GT=WSST])
             DATA SET : Reynolds Optimum Interpolation Weekly SST
Analysis
             FILENAME : reynolds_sst_wk.nc
             FILEPATH :
http://ferret.pmel.noaa.gov/cgi-bin/dods/nph-dods/data/PMEL/
          86400.
```

Ch3 Sec2.3.55. TAX_YEAR

TAX_YEAR(A, B): Returns years of time axis coordinate values

Arguments:	A	time steps to convert
	B	variable with reference time axis; use region settings to restrict this to a small region (see below)
Result Axes:	X	normal
	Y	normal
	Z	normal
	T	Inherited from A

NOTE :The variable given for argument 2 is loaded into memory by this function, so restrict its range in X, Y, and/or Z so that it is not too large. Or, if you are getting the time steps from the variable, specify the time steps for both arguments. See the examples in the section on the TAX_DATESTRING function (p. 105)

Example:

```
yes? use
"http://ferret.pmel.noaa.gov/cgi-bin/dods/nph-dods/data/PMEL/COADS/coads
_sst.nc"
yes? list/L=1403:1408 tax_year(t[gt=sst], sst[i=1,j=1])
             VARIABLE : TAX_YEAR(T[GT=SST], SST[I=1,J=1])
             DATA SET : COADS Surface Marine Observations (1854-1993)
             FILENAME : coads_sst.nc
             FILEPATH :
http://ferret.pmel.noaa.gov/cgi-bin/dods/nph-dods/data/PMEL/COADS/
             SUBSET   : 6 points (TIME)
 15-NOV-1970 00 / 1403:  1970.
 15-DEC-1970 00 / 1404:  1970.
 15-JAN-1971 00 / 1405:  1971.
 15-FEB-1971 00 / 1406:  1971.
 15-MAR-1971 00 / 1407:  1971.
 15-APR-1971 00 / 1408:  1971.
```

Ch3 Sec2.3.56. TAX_YEARFRAC

TAX_YEARFRAC(A, B): Returns fraction of year of time axis coordinate values

Arguments:	A	time steps to convert
	B	variable with reference time axis; use region settings to restrict this to a small region (see below)
Result Axes:	X	normal
	Y	normal
	Z	normal
	T	Inherited from A

NOTE: The variable given for argument 2 is loaded into memory by this function, so restrict its range in X, Y, and/or Z so that it is not too large. Or, if you are getting the time steps from the variable, specify the time steps for both arguments. See the examples in the section on the TAX_DATESTRING function (p. 105)

Example:

```
yes? use
"http://ferret.pmel.noaa.gov/cgi-bin/dods/nph-dods/data/PMEL/COADS/coads
_sst.nc"

yes? list/L=1403:1408 tax_yearfrac(t[gt=sst], sst[i=1,j=1])
```

```
            VARIABLE : TAX_YEARFRAC(T[GT=SST], SST[I=1,J=1])
            DATA SET : COADS Surface Marine Observations (1854-1993)
            FILENAME : coads_sst.nc
            FILEPATH :
http://ferret.pmel.noaa.gov/cgi-bin/dods/nph-dods/data/PMEL/COADS/
            SUBSET   : 6 points (TIME)
 15-NOV-1970 00 / 1403:  0.8740
 15-DEC-1970 00 / 1404:  0.9562
 15-JAN-1971 00 / 1405:  0.0411
 15-FEB-1971 00 / 1406:  0.1260
 15-MAR-1971 00 / 1407:  0.2027
 15-APR-1971 00 / 1408:  0.2877
```

Ch3 Sec2.4. Transformations

Transformations (e.g., averaging, integrating, etc.) may be specified along the axes of a variable. Some transformations (e.g., averaging) reduce a range of data to a point; others (e.g., differentiating) retain the range.

When transformations are specified along more than one axis of a single variable the order of execution is X axis first, then Y then Z then T.

The regridding transformations are described in the chapter "Grids and Regions" (p. 145).

Example syntax: ***`TEMP[Z=0:100@LOC:20]`*** (depth at which temp has value 20)

Valid transformations are

Transform	Default Argument	Description
@DIN		definite integral (weighted sum)
@IIN		indefinite integral (weighted running sum)
@AVE		average
@VAR		unweighted variance
@MIN		minimum
@MAX		maximum
@SHF	1 pt	shift
@SBX	3 pt	boxcar smoothed
@SBN	3 pt	binomial smoothed
@SHN	3 pt	Hanning smoothed
@SPZ	3 pt	Parzen smoothed
@SWL	3 pt	Welch smoothed
@DDC		centered derivative
@DDF		forward derivative
@DDB		backward derivative
@NGD		number of valid points
@NBD		number of bad (invalid) points flagged

Transform	Default Argument	Description
@SUM		unweighted sum
@RSUM		running unweighted sum
@FAV	3 pt	fill missing values with average
@FLN	1 pt	fill missing values by linear interpolation
@FNR	1 pt	fill missing values with nearest point
@LOC	0	coordinate of ... (e.g., depth of 20 degrees)
@WEQ		"weighted equal" (integrating kernel)
@CDA		closest distance above
@CDB		closest distance below
@CIA		closest index above
@CIB		closest index below

The command SHOW TRANSFORM will produce a list of currently available transformations.

Examples: Transformations

U[Z=0:100@AVE] – average of u between 0 and 100 in Z
sst[T=@SBX:10] – box-car smooths sst with a 10 time point filter
tau[L=1:25@DDC] – centered time derivative of tau
v[L=@IIN] – indefinite (accumulated) integral of v
qflux[X=@AVE,Y=@AVE] – XY area-averaged qflux

Ch3 Sec2.4.1. General information about transformations

Transformations are normally computed axis by axis; if multiple axes have transformations specified simultaneously (e.g., ***U[Z=@AVE,L=@SBX:10]***) the transformations will be applied sequentially in the order X then Y then Z then T. There are two exceptions to this: if @DIN is applied simultaneously to both the X and Y axes (in units of degrees of longitude and latitude, respectively) the calculation will be carried out on a per-unit-area basis (as a true double integral) instead of a per-unit-length basis, sequentially. This ensures that the COSINE(latitude) factors will be applied correctly. The same applies to @AVE simultaneously on X and Y.

Data that are flagged as invalid are excluded from calculations.

All integrations and averaging are accomplished by multiplying the width of each grid box or portion of the box by the value of the variable in that grid box—then summing and dividing as appropriate for the particular transformation.

If integration or averaging limits are given as world coordinates, the grid boxes at the edges of the region specified are weighted according to the fraction of grid box that actually lies within the specified region. If the transformation limits are given as subscripts, the full box size of each grid point along the axis is used—including the first and last subscript given. The region information that is listed with the output reflects this.

Some transformations (derivatives, shifts, smoothers) require data points from beyond the edges of the indicated region in order to perform the calculation. Ferret automatically accesses this data as needed. It flags edge points as missing values if the required beyond-edge points are unavailable (e.g., @DDC applied on the X axis at I=1).

When calculating integrals and derivatives (@IIN, @DIN, @DDC, @DDF, and @DDB) Ferret attempts to use standardized units for the grid coordinates. If the underlying axis is in a known unit of length Ferret converts grid box lengths to meters. If the underlying axis is in a known unit of time Ferret converts grid box lengths to seconds. If the underlying axis is degrees of longitude a factor of COSINE (latitude) is applied to the grid box lengths in meters.

If the underlying axis units are unknown Ferret uses those unknown units for the grid box lengths. (If Ferret does not recognize the units of an axis it displays a message to that effect when the DEFINE AXIS or SET DATA command defines the axis.) See command DEFINE AXIS/UNITS (p. 344) in the Commands Reference in this manual for a list of recognized units.

Ch3 Sec2.4.2. Transformations applied to irregular regions

Since transformations are applied along the orthogonal axes of a grid they lend themselves naturally to application over "rectangular" regions (possibly in 3 or 4 dimensions). Ferret has sufficient flexibility, however, to perform transformations over irregular regions.

Suppose, for example, that we wish to determine the average wind speed within an irregularly shaped region of the globe defined by a threshold sea surface temperature value. We can do this through the creation of a mask, as in this example:

```
yes? SET DATA coads_climatology
yes? SET REGION/l=1/@t        ! January in the Tropical Pacific
yes? LET sst28_mask = IF sst GT 28 THEN 1
yes? LET masked_wind_speed = wspd * sst28_mask
yes? LIST masked_wind_speed[X=@AVE,Y=@AVE]
```

The variable sst28_mask is a collection of 1's and missing values. Using it as a multiplier on the wind speed field produces a new result that is undefined except in the domain of interest.

When using masking be aware of these considerations:

- Use undefined values rather than zeros to avoid contaminating the calculation with zero values.

- The masked region is composed of rectangles at the level of resolution of the gridded variables; the mask does NOT follow smooth contour lines. To obtain a smoother mask it may be desirable to regrid the calculation to a finer grid.

- Variables from different data sets can be used to mask one another. For example, the ETOPO60 bathymetry data set can be used to mask regions of land and sea.

Ch3 Sec2.4.3. General information about smoothing transformations

Ferret provides several transformations for smoothing variables (removing high frequency variability). These transformations replace each value on the grid to which they are applied with a weighted average of the surrounding data values along the axis specified. For example, the expression u[T=@SPZ:3] replaces the value at each (I,J,K,L) grid point of the variable "u" with the weighted average

```
u at t = 0.25*(u at t-1) + 0.5*(u at t) + 0.25*(u at t+1)
```

The various choices of smoothing transformations (@SBX, @SBN, @SPZ, @SHN, @SWL) represent different shapes of weighting functions or "windows" with which the original variable is convolved. New window functions can be obtained by nesting the simple ones provided. For example, using the definitions

```
yes? LET ubox = u[L=@SBX:15]
yes? LET utaper = ubox[L=@SHN:7]
```

produces a 21-point window whose shape is a boxcar (constant weight) with COSINE (Hanning) tapers at each end.

Ferret may be used to directly examine the shape of any smoothing window: Mathematically, the shape of the smoothing window can be recovered as a variable by convolving it with a delta

function. In the example below we examine (PLOT) the shape of a 15-point Welch window (Figure 3_4).

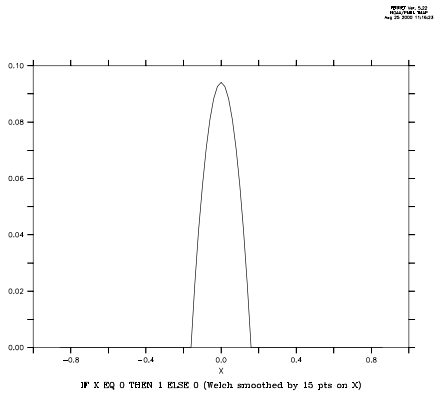


Figure 3_4

```
! define X axis as [-1,1] by 0.2
yes? GO unit_square
yes? SET REGION/X=-1:1
yes? LET delta =
      IF X EQ 0 THEN 1 ELSE 0
! convolve delta with Welch window
yes? PLOT delta[I=@SWL:15]
```

Ch3 Sec2.4.4. @DIN — definite integral

The transformation @DIN computes the definite integral—a single value that is the integral between two points along an axis (compare with @IIN). It is obtained as the sum of the grid_box*variable product at each grid point. Grid points at the ends of the indicated range are weighted by the fraction of the grid box that falls within the integration interval.

If @DIN is specified simultaneously on multiple axes the calculation will be performed as a multiple integration rather than as sequential single integrations. The output will document this fact by indicating a transformation of "@IN4" or "XY integ." See General Information (p 113) for important details about this transformation.

Example:

```
yes? CONTOUR/X=160E:160W/Y=5S:5N u[Z=0:50@DIN]
```

In a latitude/longitude coordinate system X=@DIN is sensitive to the COS(latitude) correction.

Integration over complex regions in space may be achieved by masking the multi-dimensional variable in question and using the multi-dimensional form of @DIN. For example

```
yes? LET salinity_where_temp_gt_15 = IF temp GT 15 THEN salt
yes?  LIST salinity_where_temp_gt_15[X=@DIN,Y=@DIN,Z=@DIN]
```

Ch3 Sec2.4.5. @IIN — indefinite integral

The transformation @IIN computes the indefinite integral—at each subscript of the result it is the value of the integral from the start value to the upper edge of that grid box. It is obtained as a running sum of the grid_box*variable product at each grid point. Grid points at the ends of the indicated range are weighted by the fraction of the grid box that falls within the integration interval. If See General Information (p 113) for important details about this transformation.

In particular, it's important to pay attention to how the limits are specified. If the limits of integration are given as world coordinates (var[X=10:22@IIN]), the grid boxes at the edges of the region specified are weighted according to the fraction of grid box that actually lies within the specified region. If the transformation limits are given as subscripts (var[I=1:13@IIN]), then the full box size of each grid point along the axis is used—including the first and last subscript given.

Example:

```
yes? CONTOUR/X=160E:160W/Z=0 u[Y=5S:5N@IIN]
```

Note 1: The indefinite integral is always computed in the increasing coordinate direction. To compute the indefinite integral in the reverse direction use

```
LET reverse_integral = my_var[X=lo:hi@DIN] - my_var[X=lo:hi@IIN]
```

Note 2: In a latitude/longitude coordinate system X=@IIN is sensitive to the COS(latitude) correction.

Note 3: The result of the indefinite integral is shifted by 1/2 of a grid cell from its "proper" location. This is because the result at each grid cell includes the integral computed to the upper end of that cell. (This was necessary in order that var[I=lo:hi@DIN] and var[I=lo:hi@IIN] produce consistent results.)

To illustrate, consider these commands

```
yes? LET one = x-x+1
yes? LIST/I=1:3 one[I=@din]
             X-X+1
             X: 0.5 to 3.5 (integrated)
           3.000
yes? LIST/I=1:3 one[I=@iin]
             X-X+1
             indef. integ. on X
 1   / 1:  1.000
 2   / 2:  2.000
 3   / 3:  3.000
```

The grid cell at I=1 extends from 0.5 to 1.5. The value of the integral at 1.5 is 1.000 as reported but the coordinate listed for this value is 1 rather than 1.5. Two methods are available to correct for this 1/2 grid cell shift.

Method 1: correct the result by subtracting the 1/2 grid cell error

```
yes? LIST/I=1:3 one[I=@iin] - one/2
            ONE[I=@IIN] - ONE/2
 1   / 1:  0.500
 2   / 2:  1.500
 3   / 3:  2.500
```

Method 2: correct the coordinates by shifting the axis 1/2 of a grid cell

```
yes? DEFINE AXIS/X=1.5:3.5:1 xshift
yes? LET SHIFTED_INTEGRAL =  one[I=@IIN]
yes? LET corrected_integral = shifted_integral[GX=xshift@ASN]
yes? LIST/I=1:3 corrected_integral
            SHIFTED_INTEGRAL[GX=XSHIFT@ASN]
 1.5 / 1:  1.000
 2.5 / 2:  2.000
 3.5 / 3:  3.000
```

Ch3 Sec2.4.6. @AVE—average

The transformation @AVE computes the average weighted by grid box size—a single number representing the average of the variable between two endpoints.

If @AVE is specified simultaneously on multiple axes the calculation will be performed as a multiple integration rather than as sequential single integrations. The output will document this fact by showing @AV4 or "XY ave" as the transformation.

See General Information (p. 113) for important details about this transformation. In particular, note the discussion about specifying the averaging interval in world coordinates, e.g. longitude, meters, time as in var[x=3.4:4.6@AVE] versus specifying the interval using indices, as in var[I=4:12@AVE]. When the interval is expressed in world coordinates, the weighting is done using partial grid boxes at the edges of the interval. If the interval is expressed using indices, the entire grid cells contribute to the weights.

Example:

```
yes? CONTOUR/X=160E:160W/Y=5S:5N u[Z=0:50@AVE]
```

Note that the unweighted mean can be calculated using the @SUM and @NGD transformations.

Averaging over complex regions in space may be achieved by masking the multi-dimensional variable in question and using the multi-dimensional form of @AVE. For example

```
yes? LET salinity_where_temp_gt_15 = IF temp GT 15 THEN salt
yes? LIST salinity_where_temp_gt_15[X=@AVE,Y=@AVE,Z=@AVE]
```

When we use var[x=@AVE] Ferret averages over the grid points of the variable along the X axis, using any region in X that is in place. IF a specified range is given X=x1:x2@ave, then Ferret uses portions of grid cells to average over that exact region.

```
yes? USE coads_climatology
yes? LIST/L=1/Y=45 sst[x=301:305@AVE]
             VARIABLE : SEA SURFACE TEMPERATURE (Deg C)
             LONGITUDE: 59W to 55W (averaged)
             LATITUDE : 45N
             TIME     : 16-JAN      06:00
          2.6557

yes? LET var = sst[x=301:305]
yes? LIST/L=1/Y=45 var
             VARIABLE : SST[X=301:305]
             SUBSET   : 3 points (LONGITUDE)
             LATITUDE : 45N
             TIME     : 16-JAN      06:00
                45N
 59W   / 141:  2.231
 57W   / 142:  2.604
 55W   / 143:  3.183

yes? LIST/L=1/Y=45 var[x=@AVE]
             VARIABLE : SST[X=301:305]
             LONGITUDE: 60W to 54W (averaged)
             LATITUDE : 45N
             TIME     : 16-JAN      06:00
          2.6730
```

The last average is taken not from a specific X to another specific X, but over all grid cells in the range where the variable var is defined. Note in each listing the LONGITUDE range of the average.

Ch3 Sec2.4.7. VAR—weighted variance

The transformation @VAR computes the weighted variance of the variable with respect to the indicated region (ref. Numerical Recipes, The Art of Scientific Computing, by William H. Press et al., 1986).

As with @AVE, if @VAR is applied simultaneously to multiple axes the calculation is performed as the variance of a block of data rather than as nested 1-dimensional variances. See General Information (p 113) for important details about this transformation.

Ch3 Sec2.4.8. MIN—minimum

The transformation @MIN finds the minimum value of the variable within the specified axis range. See General Information (p 113) for important details about this transformation.

Example:

For fixed Z and Y

```
yes? PLOT/T="1-JAN-1982":"1-JAN-1983"    temp[X=160E:160W@MIN]
```

plots a time series of the minimum temperature found between longitudes 160 east and 160 west.

Ch3 Sec2.4.9. @MAX—maximum

The transformation @MAX finds the maximum value of the variable within the specified axis range. See also @MIN. See General Information (p 113) for important details about this transformation.

Ch3 Sec2.4.10. @SHF:n—shift

The transformation @SHF shifts the data up or down in subscript by the number of points given as the argument. The default is to shift by 1 point. See General Information (p 113) for important details about this transformation.

Examples:

U[L=@SHF:2]
associates the value of U[L=3] with the subscript L=1.

U[L=@SHF:1]-U
gives the forward difference of the variable U along the L axis.

Ch3 Sec2.4.11. @SBX:n—boxcar smoother

The transformation @SBX applies a boxcar window (running mean) to smooth the variable along the indicated axis. The width of the boxcar is the number of points given as an argument to the transformation. The default width is 3 points. All points are weighted equally, regardless of the sizes of the grid boxes, making this transformation best suited to axes with equally spaced points. If the number of points specified is even, however, @SBX weights the end

points of the boxcar smoother as ½.. See General Information (p 113) for important details about this transformation.

Example:

```
yes? PLOT/X=160W/Y=0 u[L=1:120@SBX:5]
```

The transformation @SBX does not reduce the number of points along the axis; it replaces each of the original values with the average of its surrounding points. Regridding can be used to reduce the number of points.

Ch3 Sec2.4.12. @SBN:n — binomial smoother

The transformation @SBN applies a binomial window to smooth the variable along the indicated axis. The width of the smoother is the number of points given as an argument to the transformation. The default width is 3 points. The weights are applied without regard to the widths of the grid boxes, making this transformation best suited to axes with equally spaced points. See General Information (p 113) for important details about this transformation.

Example:

```
yes? PLOT/X=160W/Y=0/Z=0 u[L=1:120@SBN:15]
```

The transformation @SBN does not reduce the number of points along the axis; it replaces each of the original values with a weighted sum of its surrounding points. Regridding can be used to reduce the number of points. The argument specified with @SBN, the number of points in the smoothing window, must be an odd value; an even value would result in an effective shift of the data along its axis.

Ch3 Sec2.4.13. @SHN:n — Hanning smoother

Transformation @SHN applies a Hanning window to smooth the variable along the indicated axis (ref. Numerical Recipes, The Art of Scientific Computing, by William H. Press et al., 1986). In other respects it is identical in function to the @SBN transformation. Note that the Hanning window used by Ferret contains only non-zero weight values with the window width.The default width is 3 points. Some interpretations of this window function include zero weights at the end points. Use an argument of N-2 to achieve this effect (e.g., @SBX:5 is equivalent to a 7-point Hanning window which has zeros as its first and last weights). See General Information (p 113) for important details about this transformation.

Ch3 Sec2.4.14. @SPZ:n—Parzen smoother

Transformation @SPZ applies a Parzen window to smooth the variable along the indicated axis (ref. Numerical Recipes, The Art of Scientific Computing, by William H. Press et al., 1986). In other respects it is identical in function to the @SBN transformation. The default window width is 3 points. See General Information (p 113) for important details about this transformation.

Ch3 Sec2.4.15. @SWL:n—Welch smoother

Transformation @SWL applies a Welch window to smooth the variable along the indicated axis (ref. Numerical Recipes, The Art of Scientific Computing, by William H. Press et al., 1986). In other respects it is identical in function to the @SBN transformation. The default window width is 3 points. SSee General Information (p 113) for important details about this transformation.

Ch3 Sec2.4.16. @DDC—centered derivative

The transformation @DDC computes the derivative with respect to the indicated axis using a centered differencing scheme. The units of the underlying axis are treated as they are with integrations. If the points of the axis are unequally spaced, note that the calculation used is still $(F_{i+1} - F_{i-1}) / (X_{i+1} - X_{i-1})$. See General Information (p 113) for important details about this transformation.

Example:

```
yes? PLOT/X=160W/Y=0/Z=0 u[L=1:120@DDC]
```

Ch3 Sec2.4.17. @DDF—forward derivative

The transformation @DDF computes the derivative with respect to the indicated axis. A forward differencing scheme is used. The units of the underlying axis are treated as they are with integrations. See General Information (p 113) for important details about this transformation.

Example:

```
yes? PLOT/X=160W/Y=0/Z=0 u[L=1:120@DDF]
```

Ch3 Sec2.4.18. @DDB—backward derivative

The transformation @DDF computes the derivative with respect to the indicated axis. A backward differencing scheme is used. The units of the underlying axis are treated as they are with integrations. See General Information (p 113) for important details about this transformation.

Example:

```
yes? PLOT/X=160W/Y=0/Z=0 u[L=1:120@DDB]
```

Ch3 Sec2.4.19. @NGD—number of good points

The transformation @NGD computes the number of good (valid) points of the variable with respect to the indicated axis. Use @NGD in combination with @SUM to determine the number of good points in a multi-dimensional region.

Note that, as with @VAR, when @NGD is applied simultaneously to multiple axes the calculation is applied to the entire block of values rather than to the individual axes. See General Information (p 113) for important details about this transformation.

Ch3 Sec2.4.20. @NBD—number of bad points

The transformation @NBD computes the number of bad (invalid) points of the variable with respect to the indicated axis. Use @NBD in combination with @SUM to determine the number of bad points in a multi-dimensional region.

Note that, as with @VAR, when @NBD is applied simultaneously to multiple axes the calculation is applied to the entire block of values rather than to the individual axes. See General Information (p 113) for important details about this transformation.

Ch3 Sec2.4.21. @SUM—unweighted sum

The transformation @SUM computes the unweighted sum (arithmetic sum) of the variable with respect to the indicated axis. This transformation is most appropriate for regions specified by subscript. If the region is specified in world coordinates, the edge points are not weighted—they are wholly included in or excluded from the calculation, depending on the location of the grid points with respect to the specified limits. See General Information (p 113) for important details about this transformation.

Ch3 Sec2.4.22. @RSUM—running unweighted sum

The transformation @RSUM computes the running unweighted sum of the variable with respect to the indicated axis. @RSUM is to @IIN as @SUM is to @DIN. The treatment of edge points is identical to @SUM. See General Information (p 113) for important details about this transformation.

Ch3 Sec2.4.23. @FAV:n—averaging filler

The transformation @FAV fills holes (values flagged as invalid) in variables with the average value of the surrounding grid points along the indicated axis. The width of the averaging window is the number of points given as an argument to the transformation. The default is n=3. If an even value of n is specified, Ferret uses n+1 so that the average is centered. All of the surrounding points are weighted equally, regardless of the sizes of the grid boxes, making this transformation best suited to axes with equally spaced points. If any of the surrounding points are invalid they are omitted from the calculation. If all of the surrounding points are invalid the hole is not filled. See General Information (p 113) for important details about this transformation.

Example:

```
yes? CONTOUR/X=160W:160E/Y=5S:0 u[X=@FAV:5]
```

Ch3 Sec2.4.24. @FLN:n—linear interpolation filler

The transformation @FLN:n fills holes in variables with a linear interpolation from the nearest non-missing surrounding point. n specifies the number of points beyond the edge of the indicated axis limits to include in the search for interpolants (default n = 1). Unlike @FAV, @FLN is sensitive to unevenly spaced points and computes its linear interpolation based on the world coordinate locations of grid points.

Any gap of missing values that has a valid data point on each end will be filled, regardless of the length of the gap. However, when a sub-region from the full span of the data is requested sometimes a fillable gap crosses the border of the requested region. In this case the valid data point from which interpolation should be computed is not available. The parameter n tells Ferret how far beyond the border of the requested region to look for a valid data point. See General Information (p 113) for important details about this transformation.

Example: To allow data to be filled only when gaps in i are less than 15 points, use the @CIA and @CIB transformations which return the distance from the nearest valid point.

```
yes? USE my_data
yes? LET allowed_gap = 15
yes? LET gap_size = my_var[i=@cia] + my_var[i=@cib]
```

```
yes? LET gap_mask = IF gap_size LE gap_allowed THEN 1
yes? LET my_answer = my_var[i=@fln) * gap_mask
```

Ch3 Sec2.4.25. @FNR—nearest neighbor filler

The transformation @FNR is similar to @FLN, except that it replicates the nearest point to the missing value. In the case of points being equally spaced around the missing point, the mean value is used. See General Information (p 113) for important details about this transformation.

Ch3 Sec2.4.26. @LOC—location of

The transformation @LOC accepts an argument value—the default value is zero if no argument is specified. The transformation @LOC finds the single location at which the variable first assumes the value of the argument. The result is in units of the underlying axis. Linear interpolation is used to compute locations between grid points. If the variable does not assume the value of the argument within the specified region the @LOC transformation returns an invalid data flag. See also the discussion of @EVNT, the "event mask" transformation, (p. 132)

For example, temp[Z=0:200@LOC:18] finds the location along the Z axis (often depth in meters) at which the variable "temp" (often temperature) first assumes the value 18, starting at Z=0 and searching to Z=200. See General Information (p 113) for important details about this transformation.

```
yes? CONTOUR/X=160E:160W/Y=10S:10N    temp[Z=0:200@LOC:18]
```

produces a map of the depth of the 18-degree isotherm. See also the General Information about transformations section in this chapter (p. 113).

Note that the transformation @LOC can be used to locate non-constant values, too, as the following example illustrates:

Example: locating non-constant values

Determine the depth of maximum salinity.

```
yes? LET max_salt = salt[Z=@MAX]
yes? LET zero_at_max = salt - max_salt
yes? LET depth_of_max = zero_at_max[Z=@LOC:0]
```

Ch3 Sec2.4.27. @WEQ—weighted equal; integration kernel

The @WEQ ("weighted equal") transformation is the subtlest and arguably the most powerful transformation within Ferret. It is a generalized version of @LOC; @LOC always determines

the value of the axis coordinate (the variable X, Y, Z, or T) at the points where the gridded field has a particular value. More generally, @WEQ can be used to determine the value of any variable at those points. See also the discussion of @EVNT, the "event mask" transformation (p. 132). See General Information (p 113) for important details about this transformation.

Like @LOC, the transformation @WEQ finds the location along a given axis at which the variable is equal to the given (or default) argument. For example, V1[Z=@WEQ:5] finds the Z locations at which V1 equals "5". But whereas @LOC returns a single value (the linearly interpolated axis coordinate values at the locations of equality) @WEQ returns instead a field of the same size as the original variable. For those two grid points that immediately bracket the location of the argument, @WEQ returns interpolation coefficients. For all other points it returns missing value flags. If the value is found to lie identically on top of a grid point an interpolation coefficient of 1 is returned for that point alone. The default argument value is 0.0 if no argument is specified.

Example 1

```
yes? LET v1 = X/4
yes? LIST/X=1:6 v1, v1[X=@WEQ:1], v1[X=@WEQ:1.2]

 X      v1     @WEQ:1   @WEQ:1.2
___  ______   _______  _________

 1:  0.250     ....       ....
 2:  0.500     ....       ....
 3:  0.750     ....       ....
 4:  1.000    1.000     0.2000
 5:  1.250     ....     0.8000
 6:  1.500     ....       ....
```

The resulting field can be used as an "integrating kernel," a weighting function that when multiplied by another field and summed will give the value of that new field at the desired location.

Example 2

Using variable v1 from the previous example, suppose we wish to know the value of the function X^2 (X squared) at the location where variable v1 has the value 1.2. We can determine it as follows:

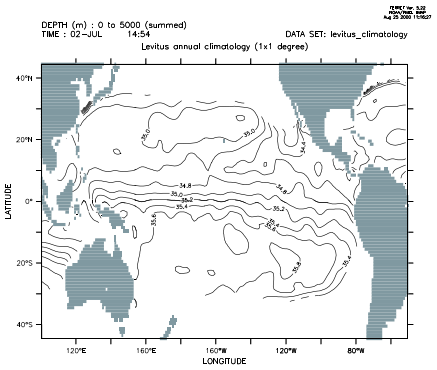
```
yes? LET x_squared = X^2
yes? LET integrand = x_squared * v1[X=@WEQ:1.2]
yes? LIST/X=1:6 integrand[X=@SUM]   !Ferret output below
            X_SQUARED * V1[X=@WEQ:1.2]
            X: 1 to 6 (summed)
         23.20
```

Notice that 23.20 = 0.8 * (5^2) + 0.2 * (4^2)

Below are two "real world" examples that produce fully labeled plots.

Example 3: salinity on an isotherm

Use the Levitus climatology to contour the salinity of the Pacific Ocean along the 20-degree isotherm (Figure 3_5).



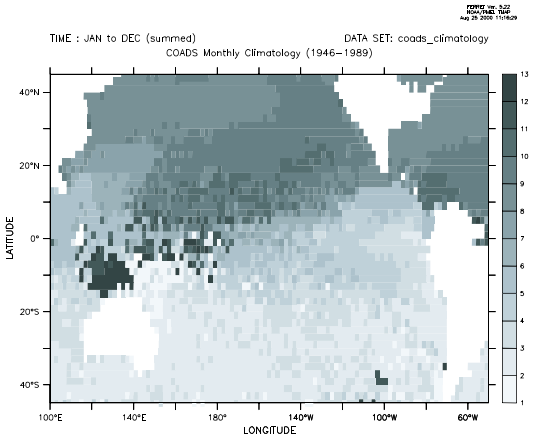
Salinity on the 20 degree isotherm

Figure 3_5

```
yes? SET DATA levitus_climatology       ! annual sub-surface climatology
yes? SET REGION/X=100E:50W/Y=45S:45N    ! Pacific Ocean
yes? LET isotherm_20 = temp[Z=@WEQ:20]  ! depth kernel for 20 degrees
yes? LET integrand_20 = salt * isotherm_20
yes? SET VARIABLE/TITLE="Salinity on the 20 degree isotherm"
integrand_20
yes? PPL CONSET .12                     !contour label size (def. .08)
yes? CONTOUR/LEV=(33,37,.2) integrand_20[Z=@SUM]
yes? GO fland                           !continental fill
```

Example 4: month with warmest sea surface temperatures

Use the COADS data set to determine the month in which the SST is warmest across the Pacific Ocean. In this example we use the same principles as above to create an integrating kernel on the time axis. Using this kernel we determine the value of the time step index (which is also the month number, 1–12) at the time of maximum SST (Figure 3_6).



Month of warmest SST

Figure 3_6

```
yes? SET DATA coads_climatology        ! monthly surface climatology
yes? SET REGION/X=100E:50W/Y=45S:45N   ! Pacific Ocean
yes? SET MODE CAL:MONTH
yes? LET zero_at_warmest = sst - sst[l=@max]
yes? LET integrand = L[G=sst] * zero_at_warmest[L=@WEQ]  ! "L" is 1 to
12
yes? SET VARIABLE/TITLE="Month of warmest SST" integrand
yes? SHADE/L=1:12/PAL=inverse_grayscale integrand[L=@SUM]
```

Example 5: values of variable at depths of a second variable:

Suppose I have V1(x,y,z) and MY_ZEES(x,y), and I want to find the values of V1 at depths MY_ZEES. The following will do that using @WEQ:

```
yes? LET zero_at_my_zees = Z[g=v1]-my_zees
yes? LET kernel = zero_at_my_zees[Z=@WEQ:0]
yes? LET integrand = kernel*v1
yes? LET v1_on_my_zees = integrand[Z=@SUM]
```

Ch3 Sec2.4.28. @ITP—interpolate

The @ITP transformation provides the same linear interpolation calculation that is turned on modally with SET MODE INTERPOLATE but with a higher level of control, as @ITP can be applied selectively to each axis. @ITP may be applied only to point locations along an axis. The result is the linear interpolation based on the adjoining values. Interpolation can be applied on an axis by axis and variable by variable basis like any other transformation. To apply interpolation use the transformation "@ITP" in the same way as, say, @SBX, specifying the desired location to which to interpolate. For example, on a Z axis with grid points at Z=10and Z=20 the syntax my_var[Z=14@ITP] would interpolate to Z=14 with the computation

0.6*my_var[Z=10]+0.4*my_var[Z=20].

The example which follows illustrates the interpolation control that is possible using @ITP:

```
SET DATA coads_climatology
! with modal interpolation
SET MODE INTERPOLATE
LIST/L=1/X=180/Y=0 sst     ! interpolates both lat and long
             SEA SURFACE TEMPERATURE (Deg C)
             LONGITUDE: 180E (interpolated)
             LATITUDE: 0 (interpolated)
             TIME: 16-JAN      06:00
             DATA SET:
/home/ja9/tmap/fer_dsets/descr/coads_climatology.des
          28.36

! with no interpolation
```

```
CANCEL MODE INTERPOLATE
LIST/L=1/X=180/Y=0 sst     ! gives value at 179E, 1S
             SEA SURFACE TEMPERATURE (Deg C)
             LONGITUDE: 179E
             LATITUDE: 1S
             TIME: 16-JAN      06:00
             DATA SET:
/home/ja9/tmap/fer_dsets/descr/coads_climatology.des
          28.20
! using @ITP to interpolate in longitude, only
LIST/L=1/Y=0 sst[X=180@ITP] ! latitude remains 1S
             SEA SURFACE TEMPERATURE (Deg C)
             LONGITUDE: 180E (interpolated)
             LATITUDE: 1S
             TIME: 16-JAN      06:00
             DATA SET:
/home/ja9/tmap/fer_dsets/descr/coads_climatology.des
          28.53
```

See General Information (p 113) for important details about this transformation.

Ch3 Sec2.4.29. @CDA—closest distance above

Syntax options:

@CDA	Distance to closest valid point above each point along the indicated axis
@CDA:n	Closest distance to a valid point above the indicated points, searching only within n points

The transformation @CDA will compute at each grid point how far it is to the closest valid point above this coordinate position on the indicated axis. The optional argument n gives a maximum distance to search for valid data. N is in integer units: how far to search forward in index space. The distance will be reported in the units of the axis. If a given grid point is valid (not missing) then the result of @CDA for that point will be 0.0. See the example for @CDB below. The result's units are now axis units, e.g., degrees of longitude to the next valid point above. See General Information (p 113) for important details about this transformation, and see the example under @CDB below (p 130).

Ch3 Sec2.4.30. @CDB—closest distance below

Syntax options:

@CDB	Distance to closest valid point below each point along the indicated axis
@CDB:n	Closest distance to a valid point below the indicated points, searching only within n points

The transformation @CDB will compute at each grid point how far it is to the closest valid point below this coordinate position on the indicated axis. The optional argument n gives a maximum distance to search for valid data. N is in integer units: how far to search backward in index space. The distance will be reported in the units of the axis. The distance will be reported in the units of the axis. If a given grid point is valid (not missing) then the result of @CDB for that point will be 0.0. The result's units are now axis units, e.g., degrees of longitude to the next valid point below. See General Information (p 113) for important details about this transformation.

Example:

```
yes? USE coads_climatology
yes? SET REGION/x=125w:109w/y=55s/l=1
yes? LIST sst, sst[x=@cda], sst[x=@cdb]   ! results below

  Column 1: SST is SEA SURFACE TEMPERATURE (Deg C)
  Column 2: SST[X=@CDA:1] is SEA SURFACE TEMPERATURE (Deg C) (closest
dist above on X ...)
  Column 3: SST[X=@CDB:1] is SEA SURFACE TEMPERATURE (Deg C) (closest
dist below on X ...)

                SST     SST     SST
125W   / 108:  6.700   0.000   0.000
123W   / 109:   ....   8.000   2.000
121W   / 110:   ....   6.000   4.000
119W   / 111:   ....   4.000   6.000
117W   / 112:   ....   2.000   8.000
115W   / 113:  7.800   0.000   0.000
113W   / 114:  7.800   0.000   0.000
111W   / 115:   ....   2.000   2.000
109W   / 116:  8.300   0.000   0.000

yes? list sst[x=121w], sst[x=121w@cda:2], sst[x=121w@cda:5], \
      sst[x=121w@cdb:5]
             DATA SET:
/home/ja9/tmap/fer_dsets/data/coads_climatology.cdf
             LONGITUDE: 121W
             LATITUDE: 55S
             TIME: 16-JAN      06:00
Column  1: SST is SEA SURFACE TEMPERATURE (Deg C)
Column  2: SST[X=@CDA:2] is SEA SURFACE TEMPERATURE (Deg C)
Column  3: SST[X=@CDA:5] is SEA SURFACE TEMPERATURE (Deg C)
Column  4: SST[X=@CDB:5] is SEA SURFACE TEMPERATURE (Deg C)
         SST  SST    SST    SST
I / *:   .... ....  6.000  4.000
```

Ch3 Sec2.4.31. @CIA—closest index above

Syntax options:

@CIA	Index of closest valid point above each point along the indicated axis

@CIA:n	Closest distance in index space to a valid point above the point at index or coordinate m, searching only within n points

The transformation @CIA will compute at each grid point how far it is to the closest valid point above this coordinate position on the indicated axis. The optional argument n gives a maximum distance to search from the index or coordinate location m for valid data. N is in integer units: how far to search forward in index space. The distance will be reported in terms of the number of points (distance in index space). If a given grid point is valid (not missing) then the result of @CIA for that point will be 0.0. See the example for @CIB below. The units of the result are grid indices; integer number of grid units to the next valid point above. See General Information (p 113) for important details about this transformation, and see the example under @CIB below (p 131).

Ch3 Sec2.4.32. @CIB—closest index below

Syntax options:

@CIB	Index of closest valid point below each point along the indicated axis
@CIB:n	Closest distance in index space to a valid point below the indicated points, searching only within n points

The transformation @CIB will compute at each grid point how far it is to the closest valid point below this coordinate position on the indicated axis. The optional argument n gives a maximum distance to search for valid data. N is in integer units: how far to search backward in index space. The distance will be reported in terms of the number of points (distance in index space). If a given grid point is valid (not missing) then the result of @CIB for that point will be 0.0. The units of the result are grid indices, integer number of grid units to the next valid point below. See General Information (p 113) for important details about this transformation.

Example:

```
yes? USE coads_climatology
yes? SET REGION/x=125w:109w/y=55s/l=1
yes? LIST sst, sst[x=@cia], sst[x=@cib]   ! results below

  Column 1: SST is SEA SURFACE TEMPERATURE (Deg C)
  Column 2: SST[X=@CIA:1] is SEA SURFACE TEMPERATURE (Deg C) (closest
dist above on X ...)
  Column 3: SST[X=@CIB:1] is SEA SURFACE TEMPERATURE (Deg C) (closest
dist below on X ...)

                SST     SST     SST
125W   / 108:  6.700   0.000   0.000
123W   / 109:   ....   4.000   1.000
```

```
121W   / 110:    ....   3.000   2.000
119W   / 111:    ....   2.000   3.000
117W   / 112:    ....   1.000   4.000
115W   / 113:  7.800   0.000   0.000
113W   / 114:  7.800   0.000   0.000
111W   / 115:    ....   1.000   1.000
109W   / 116:  8.300   0.000   0.000

yes? list sst[x=121w], sst[x=121w@cia:2], sst[x=121w@cia:5], \
    sst[x=121w@cib:5]
             DATA SET:
/home/ja9/tmap/fer_dsets/data/coads_climatology.cdf
             LONGITUDE: 121W
             LATITUDE: 55S
             TIME: 16-JAN      06:00
 Column  1: SST is SEA SURFACE TEMPERATURE (Deg C)
 Column  2: SST[X=@CIA:2] is SEA SURFACE TEMPERATURE (Deg C)
 Column  3: SST[X=@CIA:5] is SEA SURFACE TEMPERATURE (Deg C)
 Column  4: SST[X=@CIB:5] is SEA SURFACE TEMPERATURE (Deg C)
          SST  SST    SST    SST
I / *:   .... ....  3.000  2.00
```

@EVNT--event mask

This transformation locates "events" in data. An event is the occurrence of a particular value. The output steps up by a value of 1 for each event, starting from a value of zero. (If the variable takes on exactly the value of the event trigger the +1 step occurs on that point. If it crosses the value, either ascending or descending, the step occurs on the first point after the crossing.)

For example, if you wanted to know the maximum value of the second wave, where (say) rising above a magnitude of 0.1 in variable "ht" represented the onset of a wave, then

```
yes? LET wave2_mask = IF ht[T=@evnt:0.1] EQ 2 THEN 1
```

is a mask for the second wave, only. The maximum waveheight may be found with

```
yes? LET wave2_ht = wave2_mask * ht
yes? LET wave2_max_ht = wave2_ht[T=@max]
```

Note that @EVNT can be used together with @LOC and @WEQ to determine the location when an event occurs and the value of other variables as the event occurs, respectively. Since there may be missing values in the data, and since the instant at which the event occurs may lie immediately before the step in value for the event mask, the following expression is a general solution.

```
yes? LET event_mask = my_var[t=@evnt:<value>]
yes? LET event_n = IF ABS(MISSING(event_mask[L=@SBX],event_mask)-n) LE
0.67 THEN my_var
```

So that

```
event_n[t=@LOC:<value>]
```

is the time at which event "n" occurs, and

```
event_n[t=@WEQ:<value>]
```

is the integrating kernel (see @WEQ)

Ch3 Sec2.5. IF-THEN logic ("masking")

Ferret expressions can contain embedded IF-THEN-ELSE logic. The syntax of the IF-THEN logic is simply (by example)

```
LET a = IF a1 GT b THEN a1 ELSE a2
```

(read as "if a1 is greater than b then a1 else a2").

This syntax is especially useful in creating masks that can be used to perform calculations over regions of arbitrary shape. For example, we can compute the average air-sea temperature difference in regions of high wind speed using this logic:

```
SET DATA coads_climatology
SET REGION/X=100W:0/Y=0:80N/T=15-JAN
LET fast_wind = IF wspd GT 10 THEN 1
LET tdiff = airt - sst
LET fast_tdiff = tdiff * fast_wind
```

We can also make compound IF-THEN statements. The parentheses are included here for clarity, but are not necessary.

```
LET a = IF (b GT c AND b LT d) THEN e
LET a = IF (b GT c OR  b LT d) THEN e
LET a = IF (b GT c AND b LT d) THEN e ELSE q
```

The user may find it clearer to think of this logic as WHERE-THEN-ELSE to aviod confusion with the IF used to control conditional execution of commands. Compound and multi-line IF-THEN-ELSE constructs are not allowed in embedded logic.

Ch3 Sec2.6. Lists of constants ("constant arrays")

The syntax {val1, val2, val3} is a quick way to enter a list of constants. For example

```
yes? LIST {1,3,5}, {1,,5}
  X: 0.5 to 3.5
  Column  1: {1,3,5}
  Column  2: {1,,5}
         {1,3,5} {1,,5}
 1   / 1:   1.000  1.000
 2   / 2:   3.000   ....
 3   / 3:   5.000  5.000
```

Note that a constant variable is always an array oriented in the X direction To create a constant aray oriented in, say, the Y direction use YSEQUENCE

```
yes? STAT/BRIEF YSEQUENCE({1,3,5})

  Total # of data points: 3 (1*3*1*1)
  # flagged as bad  data: 0
  Minimum value: 1
  Maximum value: 5
  Mean    value: 3 (unweighted average)
```

Below are two examples illustrating uses of constant arrays. (See the constant_array_demo journal file)

Ex. 1) plot a triangle (Figure 3_7)

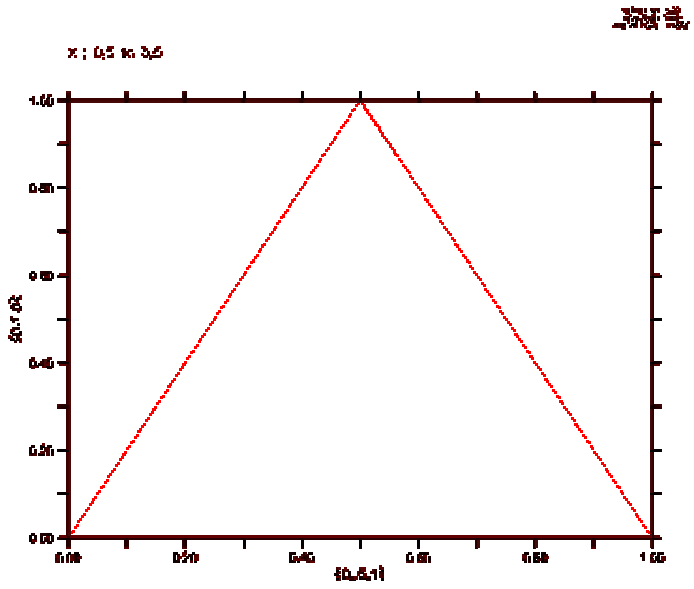


Figure 3_7

```
LET xtriangle = {0,.5,1}
LET ytriangle = {0,1,0}
POLYGON/LINE=8 xtriangle, ytriangle, 0
```

Or multiple triangles (Figure 3_8) See polymark.jnl regarding this figure

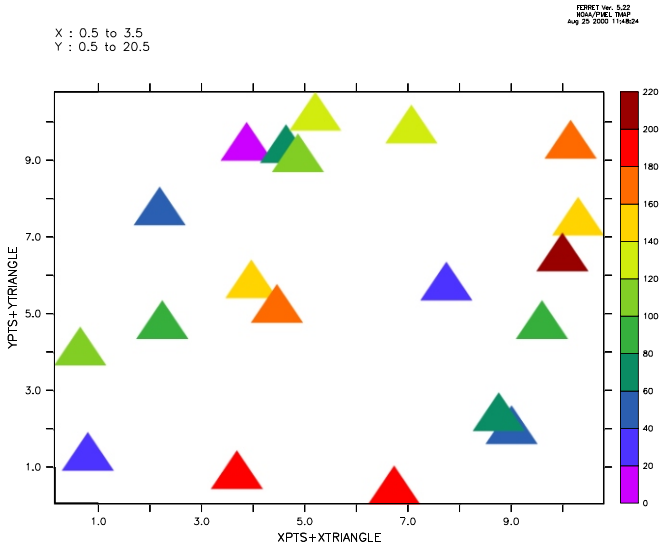


Figure 3_8

Ex. 2) Sample Jan, June, and December from sst in coads_climatology

```
yes? USE coads_climatology
yes? LET my_sst_months = SAMPLEL(sst, {1,6,12})
yes? STAT/BRIEF my_sst_months

 Total # of data points: 48600 (180*90*1*3)
 # flagged as bad data: 21831
 Minimum value: -2.6
 Maximum value: 31.637
 Mean value: 17.571 (unweighted average)
```

Ch3 Sec3. EMBEDDED EXPRESSIONS

Ferret supports "immediate mode" mathematical expressions—that is, numerical expressions that may be embedded anywhere within a command line. These expressions are evaluated immediately by Ferret—before the command itself is parsed and executed. Immediate mode expressions are enclosed in grave accents, the same syntax used by the Unix C shell. Prior to parsing and executing the command Ferret will replace the full grave accent expression, including the accent marks, with an ASCII string representing the numerical value. For example, if the given command is

```
CONTOUR/Z=`temp[X=180,Y=0,Z=@LOC:15]` salt
```

Ferret will evaluate the expression "temp[X=180,Y=0,Z=@LOC:15]" (the depth of the 15-degree isotherm at the equator/dateline—say, it is 234.5 meters). Ferret will generate and execute the command

```
CONTOUR/Z=234.5 salt
```

Embedded expressions:

Embedded expressions: the expression must evaluate to a single number, a scalar, or Ferret will respond that the command contains an error. If the result is invalid the numerical string will be "bad" (see BAD= in following section, p. 137). Region qualifiers that begin a command containing an embedded expression will be used in the evaluation of the expression. If multiple embedded expressions are used in a single command they will be evaluated from left to right within the command. This means that embedded expressions used to specify region information (e.g., the above example) may influence the evaluation of other embedded expressions to the right. When embedded expressions are used within commands that are arguments of a REPEAT command their evaluation is deferred until the commands are actually executed. Thus the embedded expressions are re-evaluated at each loop index of the REPEAT command. Grave accents have a higher priority than any other syntax character. Thus grave accent expressions will be evaluated even if they are enclosed within quotation marks, parentheses, square brackets, etc. Substitutions based on dollar-signs (command script arguments and symbols) will be made before embedded expressions are evaluated. A double grave accent will be translated to a single grave accent and not actually evaluated. Thus double grave accents provide a mechanism to defer evaluation so that grave accent expressions may be passed to the Unix command line with the SPAWN command or may be passed as arguments to GO scripts (to be evaluated INSIDE the script). The state of MODE VERIFY will determine if the evaluation of the embedded expression is echoed at the command line—similar to REPEAT loops.

The grave accent syntax may also be used to force immediate evaluation and substitution of a string variable in a command. Note that since region qualifiers that begin a command containing an embedded expression are used in the evaluation of the expression, the string variable may not contain a region qualifier.

Ch3 Sec3.1. Special calculations using embedded expressions

By default Ferret formats the results of embedded expressions using 5 significant digits. If the result of the expression is invalid (e.g., 1/0) the result by default is the string "bad". Controls allow you to specify the formatting of embedded expression results in both valid and invalid cases and to query the size and shape of the result.

The syntax to achieve this control is KEYWORD=VALUE pairs inside the grave accents, following the expression and set off by commas. The recognized keywords are "BAD=", "PRECISION=", and "RETURN=". Only the first character of the keyword is significant, so they may be abbreviated as "B=", "P=", and "R=".

PRECISION=, BAD=, and RETURN= may be specified simultaneously, in any order, separated by commas. If RETURN= is included, however, the other keywords will be ignored.

PRECISION=#digits

can be used to control the number of significant digits displayed, up to a maximum of 10 (actually at most 7 digits are significant since Ferret calculations are performed in single precision). Ferret will, however, truncate terminating zeros following the decimal place. Thus

```
SAY `3/10,PRECISION=7`
```

will result in

```
0.3
```

instead of 0.3000000.

If the value specified for #digits is negative Ferret will interpret this as the desired number of decimal places rather than the number of significant digits. Thus

```
SAY `35501/100,P=-2`
```

will result in

```
355.01
```

instead of 355.

In the case of a negative precision value, Ferret will again drop terminating zeros to the right of the decimal point.

Note that the precision of the embedded expression is used as the command is parsed, and any precision controls in the rest of the command are applied later. So

```
LIST/PRECISION=10 `100000000 + 12345`
```

will result in

W= ZW= set width and set zero-filled width.

Formatting immediate mode expressions may be done by specifying the width or zero-filled width:

```
yes? SAY Answer: `5.3,w=8`
Answer:      5.3
yes? SAY Answer: `5.3,zw=8`
Answer: 000005.3
```

BAD=string

can be used to control the text which is produced when the result of the immediate mode expression is invalid. Thus

```
SAY `1/0,BAD=missing`
```

will result in

```
missing
```

or

```
SAY `1/0,B=-999`
```

will result in

```
-999
```

RETURN=

The keyword RETURN= can reveal the size and shape of the result. RETURN= may take arguments

- SHAPE
- ISTART, JSTART, KSTART, or LSTART
- IEND, JEND, KEND, or LEND
- XSTART, YSTART, ZSTART, or TSTART
- XEND, YEND, ZEND, or TEND
- SIZE
- ISIZE, JSIZE, KSIZE, LSIZE
- BAD
- CALENDAR
- T0
- UNITS
- IUNITS, JUNITS, KUNITS, LUNITS
- XUNITS, YUNITS, ZUNITS, TUNITS
- TITLE
- GRID
- IAXIS, JAXIS, KAXIS, or LAXIS
- XAXIS, YAXIS, ZAXIS, or TAXIS
- DSET, DSETNUM, DSETPATH, DSETTITLE
- NC_SCALE, NC_OFF
- USER_SCALE, USER_OFF
- XMOD, TMOD

The RETURN= option in immediate mode expressions does not actually compute the result unless it must. For example, the expression

```
`sst, RETURN=TEND`
```

will return the formatted coordinate for the last point on the T axis of variable sst without actually reading or computing the values of sst. This allows Ferret scripts to be constructed so that they can anticipate the size of variables and act accordingly.

Note that this does not apply to variable definitions which involve grid-changing variables that return results on ABSTRACT axes. For those variables the size and shape of the result may depend on data values, so the entire result must be computed in order to determine many of the return= results

RETURN=SHAPE

Returns the 4-dimensional shape of the result—i.e., a list of those axes along which the result comprises more than a single point. For example, a global sea surface temperature field at a single point in time:

```
SAY `SST[T=1-JAN-1983],RETURN=SHAPE`
```

will result in

```
XY
```

See Symbol Substitutions in the chapter "Handling String Data" (p. 238) for examples showing the special utility of this feature.

RETURN=SIZE

Returns the total number of points in the variable -- Nx*Ny*Nz*Nt

RETURN=ISTART (and similarly JSTART, KSTART, and LSTART)

Returns the starting index of the result along the indicated axis: I, J, K, or L. For example, if CAST is a vertical profile with points every 10 meters of depth starting at 10 meters then Z=100 is the 10th vertical point, so

```
SAY `CAST[Z=100:200],RETURN=KSTART`
```

will result in

```
10
```

RETURN=IEND (and similarly JEND, KEND, and LEND)

Returns the ending index of the result along the indicated axis: I, J, K, or L. In the example above

```
SAY `CAST[Z=100:200],RETURN=KEND`
```

will result in

```
20
```

The size and shape information revealed by RESULT= is useful in creating sophisticated scripts. For example, these lines could be used to verify that the user has passed a 1-dimensional field as the first argument to a script

```
LET my_expr = $1
DEFINE SYMBOL SHAPE `my_expr,RESULT=SHAPE`
QUERY/IGNORE ($SHAPE%|X|Y|Z|T|<Expression must be 1-dimensional%)
```

RETURN=XSTART (and similarly YSTART, ZSTART, and TSTART)

Returns the first grid point in the current region, in world coordinates. Note that the format of the result can be controlled by setting or cancelling MODE LONG_LABEL for the X axis, MODE LAT_LABEL for the Y axis, or MODE CALENDAR for a time axis.

RETURN=XEND (and similarly YEND, ZEND, and TEND)

Returns the last grid point of specified world coordinate region, in world units.

RETURN=ISIZE (and similarly JSIZE, KSIZE, LSIZE

Returns the number of points along one axis, within the currently defined region.

RETURN=BAD

Returns the missing value flag from the expression

RETURN=T0

Returns the T0 string from the time axis of the variable

RETURN=CALENDAR

Returns the calendar name from the time axis of the variable

RETURN=UNITS

Returns the units string from the variable

RETURN=XUNITS (and similarly YUNIT, ZUNIT, and TUNIT)

Returns the units string from the axis

RETURN=IUNITS (and similarly JUNIT, KUNIT, and LUNIT)

Returns the units string from the axis

Example:

```
yes? say `sst, RETURN=UNIT`
  !-> MESSAGE/CONTINUE Deg C

yes? say `sst, RETURN=TUNIT`
  !-> MESSAGE/CONTINUE DAYS
```

RETURN=TITLE

Returns the title of a variable

RETURN=GRID

Returns the grid name of a variable

RETURN=IAXIS (and similarly JAXIS, KAXIS, and LAXIS)

Returns the name of an axis on which the variable is defined.

RETURN=XAXIS (and similarly YAXIS, ZAXIS, and TAXIS)

Returns the name of an axis on which the variable is defined.

RETURN=DSET

Returns data set name. This is the data set name without the file pathname.

Example:

```
yes? USE "/home/rmb_dat/testfile.nc"
yes? SAY `sst,RETURN=dset`
 !-> MESSAGE/CONTINUE testfile
testfile
```

RETURN=DSETNUM

Returns data set number from the expression.

```
yes? SAY `sst,RETURN=dsetnum`
 !-> MESSAGE/CONTINUE 1
1
```

RETURN=DSETPATH

Returns the path of the data set information from the expression. A leading slash on the pathname can cause trouble when the result is parsed by Ferret. Putting the result in a string variable is one way to deal with this.

```
yes? LET a =  "`sst,RETURN=dsetpath`"
 !-> DEFINE VARIABLE a =  "/home/rmb_dat/testfile.nc"
```

RETURN=DSETTITLE

Returns data set title from the expression, if it exists. This returns the title in a netCDF file which is specified as a global attribute :title= "Title text";

```
yes? LET a = "`sst,RETURN=dsettitle`"
 !-> DEFINE VARIABLE a =  "MERCATOR  SECTION ATL Gulf Cadiz"
```

RETURN=NC_SCALE, NC_OFF

Returns the scale and offset that were defined by a netCDF attribute for the variable. If the stepfiles of a multi-file netCDF file have different scale and offset values (see p. 289), these commands return the latest values that were applied.

RETURN=USER_SCALE, USER_OFF

Returns the scale and offset that were set using a SET VARIABLE command with the /SCALE= or /OFFSET qualifiers. (see p. 423)

RETURN=XMOD, TMOD

Returns the modulo length of the X or T axis of the variable, if it is modulo, in terms of the units of the axis.

```
yes? USE coads_climatology
yes? SAY `sst,RETURN=xmod`
 !-> MESSAGE/CONTINUE 360
360

yes? SAY `sst,RETURN=tmod`
 !-> MESSAGE/CONTINUE 8765.8
8765.8

yes? SAY `sst,RETURN=tunits`
 !-> MESSAGE/CONTINUE hour
hour
```

Ch3 Sec4. DEFINING NEW VARIABLES

The ability to define new variables lies at the heart of the computational power that Ferret provides. Complex analyses in Ferret generally proceed as hierarchies of simple variable definitions. As a simple example, suppose we wish to calculate the root mean squared value of variable, V, over 100 time steps. We could achieve this with the simple hierarchy of definitions:

```
LET v_rms     = v_mean_sq ^ 0.5
LET v_mean_sq = v_squared[L=@AVE]
LET v_squared = v * v
SET VARIABLE/TITLE="RMS V" v_rms

LIST/L=1:100 v_rms

(listed output not included)
```

As the example shows, the variables can be defined in any order and without knowledge in advance of the domain over which they will be evaluated. As variable definitions are given to Ferret with the LET (alias for DEFINE VARIABLE) command the expressions are parsed but not evaluated. Evaluation occurs only when an actual request for data is made. In the preceding example this is the point at which the LIST command is given. At that point Ferret uses the current context (SET REGION and SET DATA_SET) and the command qualifiers (e.g., "L=1:100") to determine the domain for evaluation. Ferret achieves great efficiency by evaluating only the minimum subset of data required to satisfy the request.

One consequence of this approach is that definitions such as

```
LET a = a + 1     ! nonsense
```

are nonsense within Ferret. The value(s) of variable "a" come into existence only as they are called for, thus it is nonsense for them to appear simultaneously on the left and right of an equal sign.

Variable names can be 1 to 24 characters in length and begin with a letter. See the command reference DEFINE VARIABLE (p. 349) for the available qualifiers.

Ch3 Sec4.1. Global, local, and default variable definitions

All of the above definitions are examples of "global variable definitions." A global variable definition applies to all data sets. In the above example the expression "v_rms[D=dset_1]" would be based on the values and domain of the variable V from data set dset_1 and "v_rms[D=dset_2]" would similarly be drawn from data set dset_2. The domain of v_rms, its size, shape, and resolution, will depend on the particular data set in which it is evaluated.

Although global variables are simple to use they can lead to ambiguities. Suppose, for example, that data sets dset_1 and dset_2 contain the following variables:

```
Dset_1        dset_2
______        ______
speed          u, v
```

If we would like to compare speeds from the two data sets we might be tempted to define a new variable, speed, as

```
LET speed = (u*u + v*v)^0.5
```

In doing so, however, we create an ambiguity of interpretation for the expression "speed[d=dset_1]".

To avoid this ambiguity we need to create a variable definition, "speed," that is local to data set dset_2. The qualifier /D= used as follows

```
LET/D=dset_2 speed = (u*u + v*v)^0.5          ! in dset_2, only
```

provides this capability. The use of /D=dset_2 indicates that this new definition of "speed" applies only to data set dset_2.

A convenient shortcut is often to define a "default variable." A default variable is defined using the /D qualifier with no argument

```
LET/D speed = (u*u + v*v)^0.5  ! where "speed" doesn't already exist
```

As a default variable "speed" is a definition that applies only to data sets that would otherwise not posses a variable named speed. In this sense it is a fallback default.

Ch3 Sec5. DEBUGGING COMPLEX HIERARCHIES OF EXPRESSIONS

A complex analysis generally proceeds within Ferret as a complex hierarchy of expressions: variables defined in terms of other variables defined in terms of other variables, etc., often containing many levels of transformation. When an error message such as "can only contour or vector a 2D region" occurs it may appear difficult to locate the reason for this message.

A simple strategy to locate the source of such problems is to use the command STAT which shows the size and shape of variables and expressions (simply edit the offending command line, replacing the PLOT, CONTOUR, VECTOR, etc. command with STAT and eliminating qualifiers if necessary) and use SHOW VARIABLE to see the variable definitions. By repeatedly using STAT to examine the component variables of definitions one can quickly locate the source of the problem.

Chapter 4: GRIDS AND REGIONS

Ch4 Sec1. OVERVIEW

Information describing a region in space/time, a data set, and a grid is collectively referred to as the "context." The current context may be examined with the commands SHOW DATA_SET, SHOW REGION, and SHOW GRID. The context may be set explicitly with the commands SET DATA_SET, SET REGION, and SET GRID.

The context may be modified for the duration of a single command with qualifiers to the command name (separated by slashes). The same qualifiers in square brackets may also modify single variables, changing the context only of that variable:

```
yes? PLOT/D=levitus_climatology  temp, salt

yes? CONTOUR  rose[D=etopo20]

yes? FILL/Z=0  temp[L=2] - temp[L=1]
```

Ch4 Sec2. GRIDS

Every variable has an underlying grid which defines a coordinate space. All grids are in a sense 4 dimensional (X, Y, Z, and T) but axes normal to the data are represented as "normal" (such as the Z axis of the surface wind stress).

Grids can be viewed, specified and created using SHOW GRID, SET GRID, DEFINE AXIS, and DEFINE GRID. These commands are all in the Commands Reference section of this manual. Data can be regridded by the G= modifier. (See this chapter, section "Regridding," p. 153)

Ch4 Sec2.1. Defining grids

Axes and grids can be explicitly created by DEFINE AXIS and DEFINE GRID. NetCDF and TMAP-formatted data set variables have all of the necessary grid and axis definitions embedded in the data set files, but if you are reading data from an ASCII or binary file, you must tell Ferret about the underlying grid of your data.

If you are creating abstract expressions entirely from pseudo-variables, you may want to define a grid in order to define the coordinate space of your calculation. This will also help produce a nicely labeled plot. (See the chapter "Variables and Expressions", "Grids and axes of pseudo-variables" (p. 61) and the example in the section on "Abstract Variables," p. 63.)

Example

This example is taken from the demonstration script "file_reading_demo.jnl". An ASCII file contains a grid of numbers, 50 rows by 6 columns. Suppose the data are on a 2D grid of 6 longitudes by 50 latitudes (Figure 4_1).

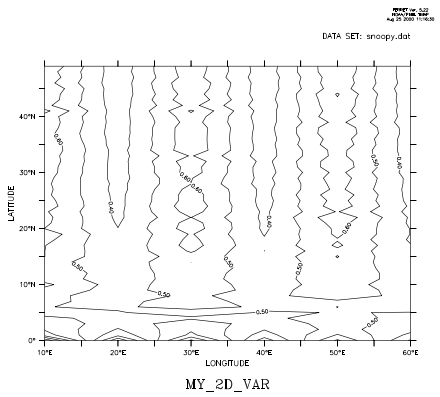


Figure 4_1

```
yes? DEFINE AXIS/X=10E:60E:10/UNIT=DEGREE xlong
yes? DEFINE AXIS/Y=0:49N:1/UNIT=DEGREE ylat
yes? DEFINE GRID/X=xlong/Y=ylat  gsnoopy2d
! By default only 1 column is read, /COLUMNS= specifies 6 columns
yes? FILE/VAR=my_2D_var/COL=6/GRID=gsnoopy2d snoopy.dat
yes? CONTOUR my_2D_var
```

Ch4 Sec2.2. Time axes and calendars

Data, particularly outputs from models, may be defined with time axes that are not on the standard Gregorian calendar. The netCDF conventions document discusses and defines usage for different calendars. These conventions for calendars are implemented in Ferret version 5.3 See:

http://www.cgd.ucar.edu/cms/eaton/cf-metadata/CF-current.html

NetCDF conforms to the conventions in the UDUNITS software package

http://www.unidata.ucar.edu/packages/udunits/

The concept of time units and formatted time needs some thought and explanation. The possibility of using different calendar definitions also compilcates the question. Time coordinates (seconds, days, years, etc) are used by the software for computation and comparison. Formatted time (30-DEC-2003 00:00:00) is for the convenience of the human user.

Time coordinates, given as so many units (seconds, days, years, etc) since a reference time is generally impossible to comprehend at a glance. There has to be internal code to convert to for-

matted time. Conversion between the time-since-reference form and formatted time requires that we know the calendar. The calendar says how many days there are in each month, and hence also implies the length of the year, which therefore depends on the calendar.

The units second, minute, hour and day (24 hours) are always the same in all calendars we use for Earth and so the utilities can assume this. Models would expect to use these units when they write out times in timesteps.

Conversion to units of time (year month day hour minute second) is also needed when processing data to calculate means over months or other calendar-related intervals and climatological statistics. For computation, comparisons, plotting and regridding, Ferret makes the choice to adopt a common length of year for all calendars.

TThe default calendar in Ferret is the Gregorian calendar. This is implemented as a "proleptic" calendar, where the definition of a year is consistent throughout time and does not have an offset in the 1500's as the historical calendars did. However, files written using the NOAA/CDC standard for the "blended" Julian/Gregorian calendar are read correctly by Ferret: If a time axis has a time origin of 1-1-1 00:00:00, and uses the default calendar, and if the coordinates of axis lie entirely after the year 1582, then the historical 2-day shift is applied.

Other calendars may be defined using DEFINE AXIS/CALENDAR= or by reading a variable with a calendar attribute from a netCDF file (see p. 40). You can set the calendar type in a descriptor file, with the D_CALTYPE attribute.

Example:

```
$BACKGROUND_RECORD
 D_TITLE          = 'Model Output, Daily Averages',
 D_T0TIME         = '30-DEC-0000 00:00:00',
 D_TIME_UNIT      = 3600.,
 D_CALTYPE        = 'NOLEAP',
$END
```

The calendars that are defined for use in Ferret are

calendar name	number of days/year	notes
GREGORIAN or STANDARD	365.2425	default calendar (proleptic gregorian)
JULIAN	365.25	with leap years
NOLEAPor COMMON_YEAR	365	no leap years
360_DAY	360	each month is 30 days

Calendar names are matched using the first three characters.

Example:

Define a calendar axis and regrid an existing variable to this axis:

```
yes? DEFINE
AXIS/CALENDAR=JULIAN/T="15-JAN-1982":"15-DEC-1985":30/UNITS=days tmodel
yes? LET twind = uwnd[GT=tmodel@NRST]
```

Regridding between different calendars is allowed using the transformations @LIN (the default), @ASN, or @NRST. When regridding with @LIN from one calendar axis to another the length of a year is assumed to be constant, therefore the regridding calculates a scale factor based on the length of a second in each calendar, computed from the number of seconds per year for the calendars.

The analysis of multi-year daily data is often awkward, because the length of the year changes for leap years. The analysis can often be made simpler by regridding the data to a NOLEAP calendar.

Ch4 Sec2.3. Dynamic grids and axes

The commands DEFINE AXIS and DEFINE GRID, described in the preceding section, should be used when the grid or axis will be referenced more than once and/or shared among several variables. In many cases it is more convenient to use dynamic (a.k.a. "implicit") grids and axes. Two quick examples:

```
PLOT SIN(X[X=0:3.14:.1])
```

– dynamically creates an axis from 0 to 3.14 by 0.1

```
SHADE SST[X=140E:160W:5, D=coads_climatology]
```

– dynamically creates a longitude axis extending from 140E to 160W by 5 degrees, dynamically creates a grid which is like the grid upon which the variable SST is defined but with the X axis replaced by the new dynamic axis, and automatically regrids to this new grid.

Ch4 Sec2.3.1. Dynamic grids

It is often possible to avoid explicitly defining grids. This is useful in two common situations:

- Situation 1

  Regridding to specified axes without the need for defining the destination grid.

  Syntax: ***G*=name@transform***, where

*	– The orientation of the axis to be regridded: "X," "Y," "Z," or "T"
name	– The name of an axis or of another variable defined on the desired axis
@transform	– The (optional) name of a regridding transform

Example:

```
sst[GX=x10deg]
```

Suppose the variable SST is defined on a 2×2 degree grid in latitude/longitude (e.g., SET DATA coads_climatology). If we wish to regrid to 10-degree spacing in longitude over a range from 175W to 75W we could use the commands

```
DEFINE AXIS/X=175w:75w:10/UNITS=degrees x10deg
LET sst10 = sst[GX=x10deg]
```

Several axes can be specified together when they are to be regridded similarly. For example, instead of ***sst[GX=x10deg, GY=y10deg]*** one can use the more concise ***sst[GXY=x10deg]***

Similarly, ***av_sst[GZ=@AVE, GT=@AVE]*** can be condensed to ***av_sst[GZT=@AVE]***

Ferret will dynamically create a grid equivalent to new_grid in

```
DEFINE GRID/LIKE=sst/X=x10deg new_grid.
```

Figure 4_2 shows the effects of regridding the 2×2 degree COADS data to a 10-degree spacing in longitude using (default) linear interpolation.

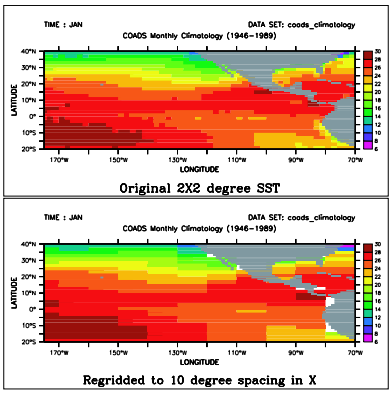


Figure 4_2

The command SHOW GRID SST10 will show the dynamically created grid. The names of dynamic grids and axes will always be displayed in parentheses.

Note that the transformation method to be used for regridding may also be specified, so LET SST10 = SST[GX=x10deg@ave] would create a 10-degree spaced result in which each grid

point was computed as the weighted sum of the source points that fell within its grid box. The default method for regridding is linear interpolation.

- Situation 2

  Automatic reconciliation of incompatible grid shapes

  Syntax: ***G=name@transform***

  where

name	– The name of a grid or of another variable defined on the desired grid
@transform	– The (optional) name of a regridding transform

  Example:

  ```
  VAR1[g=VAR2]
  ```

  If two variables are defined on grids that are mutually non-conformable because axes exist in one grid but do not exist (are NORMAL) in another, Ferret will now create a dynamic grid to resolve the non-conformabilities. This means that an expression of the form VAR1[G=VAR2] will be meaningful as long as the grid domains overlap.

  For example, TEMP[d=levitus_climatology] is defined on an XYZ (time-independent) grid whereas SST[d=coads_climatology] is defined on an XYT grid. So to evaluate the expression SST[d=coads_climatology,G=TEMP[d=levitus_climatology]] Ferret will create a dynamic intermediate grid equivalent to

  ```
  DEFINE GRID/LIKE=sst[D=coads_climatology]/X=temp/Y=temp
  ```

  so that regridding occurs on the X and Y axes but the original grid structure is maintained with respect to depth and time.

  The command SHOW GRID will reveal the resulting dynamically created grid structure.

Ch4 Sec2.3.2. Dynamic axes

The syntax "GX=lo:hi:delta" can be used in square brackets modifying a variable name to indicate the dynamic creation of an axis with the indicated range and spacing and the immediate regridding of the variable to a grid containing that axis. For example, SST[GX=175W:75W:10] will create a dynamic axis of 10-degree regular point spacing, will create a dynamic grid incorporating this axis (see previous section), and will regrid the variable SST to this grid.

Similarly, by referring to the grid indices rather than their world coordinates, the expression SST[GX=1:100:5] will create a dynamic axis that subsamples every 5th longitude point from

SST. In this case the points of the resulting axis may be irregularly spaced if the points of the original axis were also irregular.

As with the dynamic regridding described above, transformations can be specified to indicate the regridding technique. Thus SST[GX=1:100:5@AVE] will use averaging instead of the default linear interpolation to perform the regridding.

As a notational convenience the "G" may be dropped when referring to dynamic axes. Thus SST[X=175W:75W:10] is equivalent to SST[GX=175W:75W:10] and SST[I=1:100:5@AVE] is equivalent to SST[GX=1:100:5@AVE]. When using this notational convenience keep in mind that a regridding is taking place, so the transformation applied (if any) must be a regridding transformation (see SHOW TRANSFORMS in the command reference section, p. 441).

The lower plot of Figure 4_2 illustrates the effect of dynamic axes in the command

```
SHADE SST[GX=175W:75W:10]
```

Ch4 Sec2.3.3. Dynamic pseudo-variables

The same notation used for dynamic axes may also be applied to pseudo-variables providing a simple means for creating arrays of values. For example, X[GX=0.2:1:0.2] is a vector of 5 points from 0.2 to 1 at a regular spacing of 0.2 units. The vector is oriented in the X direction.

An example of using such a vector is (Figure 4_3)

```
PLOT SIN(X[GX=0:3.14:.1])
```

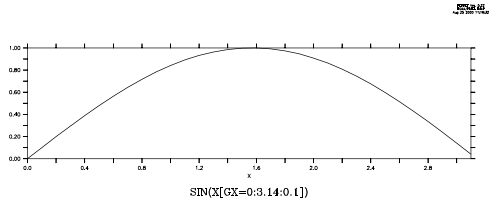


Figure 4_3

Note that when the lo:high:delta notation is applied to T or L expressed as calendar dates the units of the delta value will be hours. For example, L[GT=1-jan-1980:1-feb-1980:24] is the integers 1 to 32 defined on an axis of 32 days, 24 hours apart.

As a notational convenience the "G" may be dropped when referring to dynamic pseudo-variables. Thus X[X=0.2:1:0.2] is equivalent to X[GX=0.2:1:0.2].

See also the discussion of grids for pseudo-variables in section 3.1.3, p. 61.

Ch4 Sec2.4. Regridding

Syntax:

var[G=name] for (default) linear interpolation to new grid

or

var[G=name@trn] to regrid all axes using transform "trn" (see below)

or

var[G=name,GX=@TRN,GY=@TRN,...] to control regridding transformations along each axis separately

where

var	is the name of the variable to be regridded (e.g., temp, u, tau, ...)
name	is the name of a variable (e.g., temp[G=u]) or the name of a grid (e.g., temp[G=gu01])
trn	is the name of a special transformation (e.g., @AVE, @ASN, @LIN)

The syntax ***var[G=name,GX=@TRN,GY=@TRN,...]*** can be compressed when specifying that several axes are to be regridded similarly. For example, instead of

var[GX=sst, GY=SST]

one can now use the more concise

var[GXY=sst]

Similarly, if using a regridding transformation,

var[GZ=@AVE, GT=@AVE]

can be condensed to

var[GZT=@AVE]

Note that in Ferret Version 5 and after when the limits of a variable are unspecified v2[g=v1] will default to the full extent of the v1 grid. Previously, it would become the size of whatever region of the v2 native grid overlapped with the v1 grid.

The Ferret distribution provides a demonstration of many regridding techniques:

```
yes? GO regridding_demo
```

Regridding is essential for algebraic operations that combine variables on incompatible grids. Ferret provides the commands DEFINE AXIS and DEFINE GRID to assist with the creation of arbitrary grids.

The result grid of a regridding operation does not necessarily match exactly the destination grid requested. For example, suppose the native grid of variable TEMP3D (Ocean Temperature) is 1 degree resolution in X and Y and 50 meter spacing in Z. If the syntax "[G=sst]" is used to request regridding to the grid of variable SST (Sea Surface Temperature), which is 2 degree

resolution in X and Y, but normal to Z, then the resulting grid will be generated dynamically—inheriting X and Y axes from SST as requested, but retaining the Z axis of TEMP3D.

Examples

1) Suppose the variables u and temp are on staggered X, Y, and Z axes but share the same T axis. Then the two variables can be multiplied together on the axes (grid) of the u variable as follows:

```
yes? CONTOUR u * temp[G=u]
```

This will regrid temp onto the u grid by multi-axis linear interpolation before performing the multiplication.

2) Two variables, v1 and v2, are defined on distinct 4-dimensional grids (X, Y, Z, and T axes). The T axes of the two grids are identical but the X, Y, and Z axes all differ between the two variables. (This is often the case in numerical model outputs.)

To obtain the variable v1 on its original Z (depth) locations but regridded in the XY plane to the grid locations of the variable v2, define a new grid (say, named "new_grid") that has the X and Y axes of v2 but the Z axis of v1.

```
yes? DEFINE GRID/LIKE=v2/Z=v1 new_grid        !define new grid
yes? LIST/X=160E:140W/Y=5S:5N v1[G=new_grid]  !request regridding
```

3) In this example we look at temperature data from two data sets. levitus_climatology, an annual climatology, has the variable "temp" on an XYZ grid which is 1×1 degree in XY, and coads_climatology, a monthly climatology, has the variable "sst" on an XYT grid which is 2×2 degrees in XY. Suppose we wish to look at the sea surface temperatures in January at the higher XY resolution of the Levitus data.

```
yes? SET DATA levitus_climatology
yes? SET DATA coads_climatology
yes? SET REGION/L=1/Z=0
yes? !get the name of the grid on which temp is defined
yes? SHOW GRID temp[D=levitus_climatology]  ! -> "Glevitr1"
yes? DEFINE GRID/X=glevitr1/Y=glevitr1/Z=sst/L=sst glevitus_xy
yes? LIST/X=150E:155E/Y=0:5N sst[G=glevitus_xy]
```

Ch4 Sec2.4.1. Regridding transformations

Ferret supports several regridding transformations. Use the SHOW TRANSFORMATIONS command to obtain a list of the supported transformations from Ferret. The choice of regridding transformation determines the computation by which data from the source grid determine the values on the destination grid.

Regridding transformations provide a means to perform a given calculation over a limited span of coordinates and repeat that calculation for a series of contiguous spans. For example, if we wish to compute the variance of the variable SST over 10-degree longitude range from 180 to 170W we could use the syntax ***sst[X=180:170w@VAR]***. Now, if we wish to perform the same operation 10 times in 10-degree wide bands from 180 to 80W we could instead use ***G=@VAR*** regridding as in (see Dynamic Grids, p. 148, for an explanation of the "GX=" syntax):

```
DEFINE AXIS/X=175w:85w:10/UNITS=degrees x10deg
LET sst10 = sst[GX=x10deg@VAR]
```

The regridding transformations are:

@LIN—linear interpolation (the default if no transform is specified)
Performs regridding by multi-axis linear interpolation.

@AVE—averaging
Computes the length-weighted average of all points on the source grid that lie partly or completely within each grid cell of the destination grid. If any portion of a source grid cell containing data overlaps a given destination grid cell, then data from that source cell contributes to the destination cell, weighted by the fraction of the destination cell overlapped by the source cell. The source data are treated as continuous, extending to the edges of the grid cells.

Note: When @AVE is applied simultaneously to the X and Y axes, where X and Y are longitude and latitude, respectively, an area-weighted average (weighted by cos(latitude)) is used. The @AVE transformation is unique in this respect. In multiple axis applications other than X and Y @AVE will be applied sequentially to the axes, computing the "average of the average." This may not be the desired weighting scheme in some cases. See @VAR below for an example.

@ASN—(blind) association
Associates by subscript (blindly) the points from the source grid onto destination coordinates.

@VAR
Computes the variance of the points from the source grid that fall within each destination grid cell. This is a length-weighted computation like the @AVE transformation.

Note: This transformation is suitable for regridding only in a single axis. When applied simultaneously to two axes, for example, it will compute the variance of the variance. For example, V[gx=130E:80W:10@VAR, gy=205:20W:10@VAR] is equivalent to tmp[X=130E:80W:10@VAR] where tmp=V[y=20S:20N:10@VAR].

@NGD
Compute the number of points from the source grid that fall within each destination grid cell. Note that the results of this calculation need not be integers—this is a length-weighted

computation like the @AVE transformation. It is common for a grid cell on the source grid to span the boundary between grid cells on the destination grid, thereby contributing a fraction of its weight to multiple destination grid cells.

Note: This transformation is suitable only for regridding on a single axis. When applied simultaneously to two axes, for example, it will compute a constant. See @VAR for an example.

@NRST

Nearest coordinate regridding VAR[GX=newaxis@NRST] chooses the value from the source axis coordinate closest to the destination axis. If source coordinates above and below are equally close to a destination coordinate the value at the lower coordinate will be chosen. (This is most useful for regridding between axes whose coordinate values are very close, though not exactly matched -- e.g. between equally and unequally spaced monthly time axes.)

@SUM

Computes the length-weighted sum of the points from the source grid that fall within each destination grid cell. This is a length-weighted computation like the @AVE transformation.

@MIN

Finds the minimum value of those points from the source grid that lie within each destination grid cell. Note that this is NOT a weighted calculation; the destination grid cell that "owns" a source point is determined entirely from the coordinate location of the source point, not from the limits of the source grid cell.

(As of Ferret V5.1) If a point on the source axis lies exactly on the boundary between grid cells on the destination axis it will be included in the calculations for the higher indexed cell on the destination axis. If a point on the source axis lies exactly on the upper cell boundary of the highest point on the destination axis, then it will be included in the calculations for that highest destination grid cell.

If you have data on a single point axis and you wish to embed it in a larger axis range you can achieve this by using either the G=@MIN or G=@MAX. For example,

```
yes? define axis/x=163e/npoints=1 x1pt
yes? let var_1pt = randu(x[gx=x1pt])    ! a random value at a single
coordinate
yes? list var_1pt
            RANDU(X[GX=X1PT])
            LONGITUDE: 163E
       0.4914
yes? define axis/x=161e:165e:1 x5pt
yes? list var_1pt[gx=x5pt@max]     ! same value embedded within 5 point
axis
            RANDU(X[GX=X1PT])
             regrid: 1 deg on X@MAX
 161E   / 1:     ....
 162E   / 2:     ....
```

```
 163E   / 3:  0.4914
 164E   / 4:    ....
 165E   / 5:    ....
```

@MAX

Finds the maximum value of those points from the source grid that lie within each destination grid cell. Note that this is NOT a weighted calculation; the destination grid cell that "owns" a source point is determined entirely from the coordinate location of the source point, not from the limits of the source grid cell..

(As of Ferret V5.1) If a point on the source axis lies exactly on the boundary between grid cells on the destination axis it will be included in the calculations for the higher indexed cell on the destination axis. If a point on the source axis lies exactly on the upper cell boundary of the highest point on the destination axis, then it will be included in the calculations for that highest destination grid cell.

The @MAX transformation is useful as a mechanism to perform regridding between two axes that do not quite match. A common example would be to regrid between two monthly axes one of which has points located at the 15th of each month and the other having points exactly at mid-month. These Ferret commands illustrate the example using a 5-month axis in 1993:

```
! define axes for 15th of month and mid-month
yes? DEFINE AXIS/UNIT=DAYS/T0=1-JAN-1900 month_15 =
DAYS1900(1993,I[I1:5], 15)

yes? DEFINE AXIS/UNIT=DAYS/T0=1-JAN-1900/EDGES month_mid =
   DAYS1900(1993,I[I=1:6], 1)
yes? let my_var = L[gt=month_15
yes? list my_var
                L[GT=MONTH_15]

      15-JAN-1993 00 / 1:  1.000
     15-FEB-1993 00 / 2:  2.000
     15-MAR-1993 00 / 3:  3.000
     15-APR-1993 00 / 4:  4.000
     15-MAY-1993 00 / 5:  5.000
yes? list my_var[gt=month_mid]
               L[GT=MONTH_15]

                regrid: on T
      16-JAN-1993 12 / 1:  1.048
     15-FEB-1993 00 / 2:  2.000
     16-MAR-1993 12 / 3:  3.048
     16-APR-1993 00 / 4:  4.033
     16-MAY-1993 12 / 5:   ....     ! unable to interpolate
yes? list my_var[gt=month_mid@max]
                L[GT=MONTH_15]
                regrid: on T@MAX
     16-JAN-1993 12 / 1:  1.000
     15-FEB-1993 00 / 2:  2.000
     16-MAR-1993 12 / 3:  3.000
     16-APR-1993 00 / 4:  4.000
     16-MAY-1993 12 / 5:  5.000
```

@XACT

Regridding with G=@XACT (or GX=@XACT, etc.) is a request to transfer values from a source grid to a destination grid only at those positions where there is an exact coordinate match between the source and destination axis points on the axis in question. Other destination points will be set to "missing". This transformation is especially useful for taking multiple in-situ data profiles, such as oceanographic cast data, and regridding them onto a regular (sparse) grid. For example: grep

```
yes? LET xcoarse = sin(x[x=0:20:10])
yes? LIST xcoarse
                SIN(X[X=0:20:10])
          0  / 1:  0.0000
          10 / 2: -0.5440
          20 / 3:  0.9129
    yes? DEFINE AXIS/X=0:20:5 xfine
    yes? LIST xcoarse[gx=xfine@XACT]
                SIN(X[X=0:20:10])
                regrid: 5 delta on X@XACT
        0    / 1:    0.0000
        5    / 2:      ....
        10   / 3:  -0.5440
        15   / 4:     ....
        20  / 5:    0.9129
```

@MOD

Creates climatologies from time series by regridding to a time series axis with a modulo regridding transformation. See the section on Modulo Regridding (p. 159) for details.

Examples

1) Let variable temp be defined on a grid with points spaced regularly at 1-degree intervals in both longitude and latitude (X and Y). Let grid "g10" possess points spaced regularly at 10-degree intervals in both X and Y.

```
yes? PLOT temp[G=g10]           ! uses linear interpolation (@LIN)
yes? PLOT temp[G=g10@AVE]       ! area-averages 10x10 degrees of source\
                                  points into each destination point.
yes? PLOT temp[G=g10,GX=@AVE]   ! averages 10 degrees of longitude but\
                                  interpolates (@LIN) in Y.
```

2) @ASN has the effect of bypassing Ferret's protections against misrepresenting data (Figure 4_4).

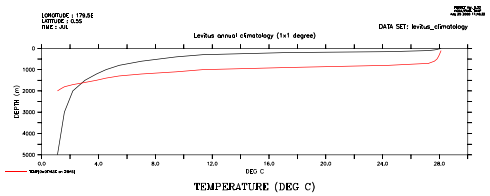


Figure 4_4

```
yes? SET DATA levitus_climatology
yes? SET REGION/X=180/Y=0     ! true profile
yes? PLOT/Z=0:5000 temp
yes? DEFINE AXIS/DEPTH /Z=100:2000:100  zfalse
yes? DEFINE GRID/LIKE=temp /Z=zfalse  gfalse   ! false profile
yes? PLOT/Z=0:5000/OVER temp[G=gfalse@ASN]
```

Ch4 Sec2.5. Modulo regridding

Ferret can create climatologies from time series simply by regridding to a climatological axis with a modulo regridding transformation. For example, if the axis named month_reg is a 12-point monthly climatological (modulo) axis then the expression

```
LET sst_climatology = sst[D=coads,GT=month_reg@MOD]
```

is a 12-month climatology computed by averaging the full time domain of the input variable (576 points for coads) modulo fashion into the 12 points of the climatological axis.

Ferret has three pre-defined climatological axes: seasonal_reg (Feb, May, Aug, Nov), month_reg (middle of every month), and month_irreg (15th of every month). In addition, there is an FAQ that describes how to create a daily climatological series, How do I compute a daily climatology for a time series? The analysis of multi-year daily data is often awkward, because the length of the year changes for leap years. The analysis can often be made simpler by regridding the data to a NOLEAP calendar.

```
yes? USE climatological_axes
*** NOTE: regarding ... climatological_axes.cdf
*** NOTE: Climatological axes SEASONAL_REG, MONTH_REG, and MONTH_IRREG
    defined
yes? CANCEL DATA climatological_axes ! the axes are still defined
```

To generate a climatology based on a restricted range of input data simply define an intermediate variable with the desired points. For example, a monthly climatological time series based on data from the 1960s could be computed using

```
LET sst_1960s = sst[D=coads,T=1-jan-1960:31-dec-1969]
PLOT sst_1960s[GT=month_reg@MOD]
```

In a similar fashion intermediate variables can be defined that mask out certain input points.

This example shows the entire sequence necessary to generate a plot of climatological SST at 40N, 40W based on the January 1982 to December 1992 Fleet Numerical wind data set. (Figure 4_5).

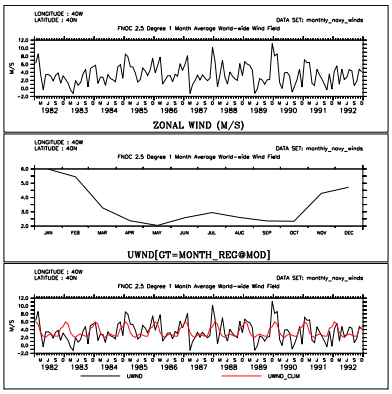


Figure 4_5

```
! use the predefined climatological axes
USE climatological_axes
CANCEL DATA climatological_axes

! use the Fleet Numerical winds
SET DATA monthly_navy_winds

! plot the raw data (top figure)
SET REGION/X=40w/Y=40n
plot uwnd

! plot the 12 month climatology (middle figure)
LET uwnd_clim = uwnd[GT=month_reg@MOD]
PLOT uwnd_clim

! since uwnd_clim is on a climatological axis
! Ferret can also plot it on the calendar axis with the raw data
PLOT/T=16-jan-1982:17-dec-1992 uwnd,uwnd_clim
```

In many cases the volume of input data needed to perform climatological calculations is very large. In the example above the command

```
CONTOUR/X=0:360/Y=90s:90n sst_climatology[L=1]
```

to plot January from the climatology would require Nx*Ny*Nt=72*72*576=3 Megawords of data. Such calculations may be too large to fit into memory. However, if the region is fully specified (as shown for the X and Y limits in the example) Ferret's internal memory manager

will break up the calculation as needed to produce the result. (See Memory Use in the chapter "Computing Environment", p. 259, for further details.)

Unlike other transformations and regridding, modulo regridding is performed as an unweighted average: each non-missing source point contributes 100% of its weight to the destination grid box within which it falls. If the source and destination axes are not properly aligned this can lead to apparent shifts in the data. For example, if a monthly time series has data points at the first of each month and a climatological axis is defined at midmonths, then unweighted modulo averaging will lead to an apparent 1/2-month shift. To avoid situations of this type simply regrid to the climatological axis using linear interpolation prior to the modulo regridding.

Here is an example that illustrates the situation and the use of linear interpolation to repair it. (Figure 4_6).

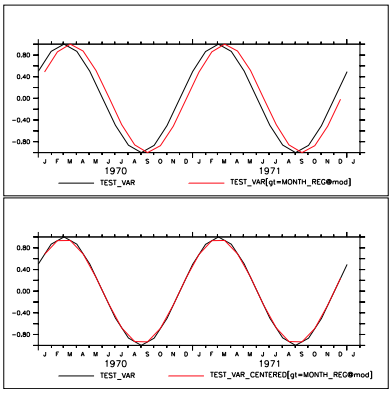


Figure 4_6

```
! define test_var, an illustrative variable with 1 year periodicity
! Note: test_var points are at the **beginnings** of months
DEFINE AXIS/T=1-jan-1970:1-jan-1974:`365.25/12`/UNITS=days t10years
DEFINE GRID/T=t10years gg
LET test_var = SIN(L[G=gg]*2*3.14/12)

! plot 4 years of the cycle
PLOT test_var

! define climatological axes at the midpoints of months
USE climatological_axes
CANC DATA climatological_axes

! notice that modulo regridding appears to shift the data
! (due to mis-aligned source and destination axes) (top figure)
PLOT/OVER/T=1-jan-1970:1-jan-1974 test_var[GT=month_reg@MOD]

! to avoid the shift we can first regrid test_var to mid-month
! points using linear interpolation (the default regridding method)
! notice that the function test_var is largely unchanged by this
LET test_var_centered = test_var[GT=month_reg]
PLOT/OVER/T=1-jan-1970:1-jan-1974 test_var_centered

! finally perform a modulo regridding on well-aligned data
! notice that the shift is gone (bottom figure)
PLOT/OVER/T=1-jan-1970:1-jan-1974 test_var_centered[GT=month_reg]
```

Ch4 Sec2.5.1. Modulo regridding statistics

In addition to the modulo averaging calculation performed by @MOD Ferret provides other statistics of the regridding. All modulo regridding calculations are unweighted as discussed under @MOD.

@MODVAR
the variance of source points within each destination grid box (SUM(var-varbar)^2)/(n-1))

@MODSUM
the sum of the source points within each destination grid box

@MODNGD
the number of source points contributing to each destination grid box

@MODMIN
the minimum value of the source points contributing to each destination grid box

@MODMAX
the maximum value of the source points contributing to each destination grid box

Ch4 Sec3. REGIONS

The region in space and time where expressions are evaluated may be specified in 3 different ways:

1) with the command SET REGION
2) with qualifiers to the command name (slash-delimited)
3) with qualifiers to variable names (in square brackets, comma-delimited)

If SET REGION is used, Ferret remembers the region as the default context for future commands, whereas a qualifier to a command name specifies the region for that command only, and a qualifier to a variable name specifies the region for that variable and command only.

Regions may be manipulated using DEFINE REGION, SET REGION, @ notation, and CANCEL REGION. The Commands Reference section of this manual covers all of these topics.

Region information is normally specified in the following form:

QUAL=val or
QUAL=lo_val:hi_val or
QUAL=val@transform (as a variable qualifier only) or
QUAL=lo_val:hi_val@transform (as a variable qualifier only)

When the region for an axis is specified as a single value (instead of a range) Ferret, by default, selects the grid point of the grid box containing this value. The Ferret mode "interpolate" can control this behavior. See command SET MODE INTERPOLATE in Commands Reference, p. 414.

Examples: Regions

Examples of valid region specifications.

1) Fully specify the region in an XY plane with the first vertical (Z) level and time 27739.

```
yes? SET REGION/X=140E:160W/Y=10S:20N/K=1/T=27739
```

2) Contour vertical heat advection within whatever region is the current default (previously set with SET REGION).

```
yes? CONTOUR qadz
```

3) Define, modify and set a named region and then modify with delta notation.

```
yes? DEFINE/REGION/Y=5S:5N YT          !define region YT to be 5S:5N
yes? DEFINE REGION/DY=-1:+1 YT         !modify region YT to be 6S:6N
yes? SET REGION/@YT                    !set current region to YT
yes? SET REGION/DY=-1:+1               !modify current region to 7S:7N
```

4) List meridional currents calculated by averaging values between the surface and a depth of 50 m.

```
yes? LIST v[Z=0:50@AVE]
```

5) Equivalent to ***v[Z=10] - v[Z=0:100@AVE]***, the anomaly at z=10 between v and the 0 to 100 meter depth average of v.

```
yes? LIST/Z=10 v - v[Z=0:100@AVE]
```

Ch4 Sec3.1. Latitude

Specify latitude or a latitude range with the qualifier Y or J. Specifications using J are integers between 1 and the number of points on the Y axis. Specifications using Y are in the units of the Y axis.

The units may be examined with SHOW GRID/Y. If the Y axis units are degrees of latitude then the Y positions may be specified as numbers followed by the letters "N" or "S".

Examples

```
yes? CONTOUR temp[Y=15S:10N]
yes? LIST/J=50 u
```

Ch4 Sec3.2. Longitude

Specify longitude or a longitude range with the qualifier X or I. Specifications using I are integers between 1 and the number of points on the X axis. Specifications using X are in the units of the X axis.

The units may be examined with SHOW GRID/X. If the units are degrees, then X values may be given as numbers followed by "W" or "E" (e.g., 160E, 110.5W) or as values between 0 and 360 with Greenwich at 0 increasing eastward. Note: If the X axis is "modulo" then it is sometimes desirable to use X greater than 360.

Examples

```
yes? CONTOUR  temp[Y=160E:140W]
yes? LIST/I=100  u
yes? SHADE/X=100:460  temp        !360 degrees centered at 100W
```

See the chapter "Grids and Regins", section "Modulo Axes" (p. 167), for help with globe-encircling axes.

Ch4 Sec3.3. Depth

Specify depth or a depth range with the qualifier Z or K. Specifications using K are integers between 1 and the number of points on the Z axis. Specifications using Z are in the units of the Z axis.

The units may be examined with SHOW GRID/Z.

Examples

```
yes? CONTOUR temp[Z=0:100]
yes? LIST/K=3 u
```

Ch4 Sec3.4. Time

Specify time or a time range with the qualifier T or L. Specifications using L are integers between 1 and the number of points on the T axis. Specifications using T may refer to calendar dates or to the time step units in which the time axis of the data set is defined.

Calendar date/time values are entered in the format dd-mmm-yyyy:hh:mm:ss, for example 14-FEB-1988:12:30:00. At a minimum the string must contain day, month, and year. If the string contains any colons it must be enclosed in quotation marks to differentiate from colons used to designate a range. If a time increment is specified with the repeat command given in calendar format (e.g., REPEAT/T="1-JAN-1982":"15-JAN-1982":6) it is interpreted as hours always. Calendar dates in the years 0000 and 0001 are regarded as year-independent dates (suitable for climatological data). Ferret cannot work with years larger than year 9999.

SHOW GRID/T can be used to display time step values. (Units may vary between data sets.) The commands SET MODE CALENDAR and CANCEL MODE CALENDAR can be used to view date strings or time steps, respectively.

Examples

```
yes? LIST/T="1-JAN-1982:13:50":"15-FEB-1982"   density
yes? CONTOUR temp[T=27740:30000]
yes? LIST/L=90 u
```

See the section in this chapter on "Modulo Axes" (p. 167) for help with climatological axes.

Ch4 Sec3.5. Delta

The notation q=lo:hi:delta (e.g., Y=20S:20N:5) specifies that the data in the requested range is regularly subsampled at interval "delta."

This notation is valid only for the REPEAT, SHOW GRID, and DEFINE AXIS commands, and the qualifiers /HLIMITS and /VLIMITS used in action commands with graphical output.

It can also be used in square brackets when specifying variable context:

```
yes? LIST temp[l=40:90:5]
```

(but this is NOT allowed: ***LIST/L=40:90:5 temp***)

Ch4 Sec3.6. @ notation

Regions may be named and referred to using the syntax "@name". Some commonly used regions are predefined. See commands SET REGION (p. 420) and DEFINE REGION (p. 347) in the Commands Reference section for further information.

If a region is specified using a combination of "@" notation and explicit axis limits the explicit axis limits will be evaluated after the "@" specification, possibly superseding the "@" limits.

Note: It is not advised to use the @notation inside of variable definitions, as redefinitions of the named region can cause code errors that lead to wrong results.

Using the @ notation only sets/alters the axis limits specified in the named region. For example, suppose that region "XY" is defined for the X and Y axes, but not for the Z and T axes. Then

```
yes? SET REGION/@XY
```

modifies only X and Y limits. BUT,

```
yes? SET REGION XY
```

modifies all axes—X and Y to the limits specified by XY, and Z and T to unspecified (even if they were previously specified).

Examples

1) Contour the 25th time step of temperature data at depth 10 within region T, the "Tropical Pacific."

```
yes? CONTOUR/@T/Z=10/L=25 temp
```

2) Produce a contour plot over region W, the "Whole Pacific Ocean," in the XY plane (the variable to be contoured as well as the depth and time will be inferred from the current context).

```
yes? CONTOUR/@W var1
```

3) Set the default region to "T", the Tropical Pacific Ocean (latitude 23.5S to 23.5N).

```
yes? SET REGION/@T
```

4) Define a region and then supersede with an axis limit specification.

```
yes? DEFINE REGION/X=180:140W/Y=2S:2N/Z=5   BOX1
yes? SET REGION/@BOX1/Z=15                  !replace Z
```

Pre-defined regions

As a convenience in the analysis of the Tropical Pacific Ocean the following regions are pre-defined:

Name	Region	Latitude	Longitude
T	Tropical Pacific	23.5S:23.5N	130E:70W
N	Narrow Pacific	10.0S:10.0N	130E:70W
W	Whole Pacific	30.0S:50.0N	130E:70W

These may be redefined by the user for the duration of a Ferret session or until the definitions are canceled.

Ch4 Sec3.7. Modulo axes

Some axes are inherently "modulo," indicating that the axis wraps around—the first point immediately following the last.

To determine if an axis is modulo use SHOW AXIS or SHOW GRID. A letter "m" following the number of points in the axis indicates a modulo axis. The command SHOW GRID qualified by the appropriate axis limits can be used to examine any part of the axis—including points beyond the nominal length of the axis. The commands SET AXIS/MODULO and CANCEL AXIS/MODULO can be used to control this feature on an axis-by-axis basis. Starting with Ferret version5.5, longitude axes which span 360 degrees or less, and climatological time axes are always detected as modulo, unless Ferret is specifically directed that the axis is NOT modulo, e.g. by a CANCEL AXIS/MODULO command.

Example

```
yes? SET DATA coads_climatology
yes? SHOW GRID/I=180:183 sst  !range request beyond last point
  GRID COADS1
  name       axis       # pts     start            end
  COADSX  LONGITUDE     180mr     21E              19E(379)
  [text omitted]
        I     X                  BOX_SIZ
      180>  19E(379)               2
      181>  21E(381)               2
      182>  23E(383)               2
      183>  25E(385)               2
```

The most common uses of modulo axes are:

1) As longitude axes for globe-encircling data sets. This allows any starting and any ending longitudes to be used, for example, X=140E:140E indicates the entire earth with data beginning and ending at 140E.

2) As time axes for climatological data. By this device the time axis appears to extend from 0 to infinity and the climatological data can be referred to at any point in time. This facilitates comparisons with data sets at fixed times.

Ch4 Sec3.7.1. Subspan Modulo Axes

Ferret V5.5 introduces the concept of a "sub-span modulo axis" -- an axis where the range is a sub-range of a fullmodulo cycle. As of V5.5, longitude axes and climatological time axes are always detected as modulo, or as sub-span modulo when appropriate, unless Ferret is specifically directed that the axis is NOT modulo, e.g. by a CANCEL AXIS/MODULO command. If the user does not specify the modulo length, it is set to 360 degrees for a longitude axis, or a year for a time axis. Time axes of lenght less than or equal to one year, and starting in year 0000 or 0001 are taken to be climatological axes.

The modulo length of an axis defined on the Ferret command line is set with an argument to the MODULO qualifier, or with an argument to the netCDF modulo attribute. Here is an example showing an axis defined explicitly as a modulo axis, and another which is modulo by default.

```
yes? DEFINE AXIS/MODULO=100/x=41:55:1 xax_subspan
yes? DEFINE AXIS/X=100:300:10/UNITS=degrees_longitude xax_lonspan
```

The output of SHOW AXIS includes the modulo length and span of the axis:

```
yes? show axis xax*
 name       axis               # pts   start                end
 XAX_SUBSPAN                     6mr   41                   46
   Axis span (to cell edges) = 6 (modulo length = 100)
 XAX_LONSPAN LONGITUDE          21mr   100E                 60W
   Axis span (to cell edges) = 210 (modulo length = 360)
```

In netCDF output files you will now see the modulo attribute taking a value. Continuing the example above, we write some variables using the axes to a file and use ncdump to show the modulo attribute in these files.

```
yes? LET v1 = X[GX=xax_subspan] +10
yes? LET v2 = SIN(X[GX=xax_lonspan])
yes? SAVE/FILE=test_subspan_modulo.nc v1, v2
yes? SPAWN ncdump -c test_subspan_modulo.nc

netcdf test_subspan_modulo {
dimensions:
      XAX_SUBSPAN = 15 ;
      XAX_LONSPAN = 21 ;
variables:
      double XAX_SUBSPAN(XAX_SUBSPAN) ;
            XAX_SUBSPAN:modulo = 100. ;
            XAX_SUBSPAN:point_spacing = "even" ;
            XAX_SUBSPAN:AXIS = "X" ;
      float V1(XAX_SUBSPAN) ;
            V1:missing_value = -1.e+34f ;
            V1:_FillValue = -1.e+34f ;
            V1:long_name = "X[GX=XAX_SUBSPAN] + 10" ;
```

```
      double XAX_LONSPAN(XAX_LONSPAN) ;
            XAX_LONSPAN:units = "degrees_east" ;
            XAX_LONSPAN:modulo = 360. ;
            XAX_LONSPAN:point_spacing = "even" ;
            XAX_LONSPAN:AXIS = "X" ;
      float V2(XAX_LONSPAN) ;
            V2:missing_value = -1.e+34f ;
            V2:_FillValue = -1.e+34f ;
            V2:long_name = "SIN(X[GX=XAX_LONSPAN])" ;

// global attributes:
            :history = "FERRET V5.50 15-Jan-03" ;
data:

 XAX_SUBSPAN = 41, 42, 43, 44, 45, 46, 47, 48, 49, 50, 51, 52, 53, 54,
55 ;

XAX_LONSPAN = 100, 110, 120, 130, 140, 150, 160, 170, 180, 190, 200,
210,
    220, 230, 240, 250, 260, 270, 280, 290, 300 ;
}
```

The importance of the sub-span modulo axes is to take the first of two steps that will make it possible for users largely to ignore differences in encodings of longitude and climatological time -- e.g. the blending of data in plots and analyses where the data come from data sets that are encoded variously as -180:180, 0:360, etc. Thus, for example, in V5.5 you can refer to my_subspan_var[g=another_var] and get a meaningful answer as long as the grids occupy the same region on the globe, regardless of longitude encoding. (The second step, for a future release, will address the longitude encoding of scattered data.)

Example:

Suppose we have data on an axis that was defined as follows

```
yes? DEFINE AXIS/X=520:550:1/UNITS=degrees  xax
```

and supposet we want to overlay it on a map showing the regional topography.

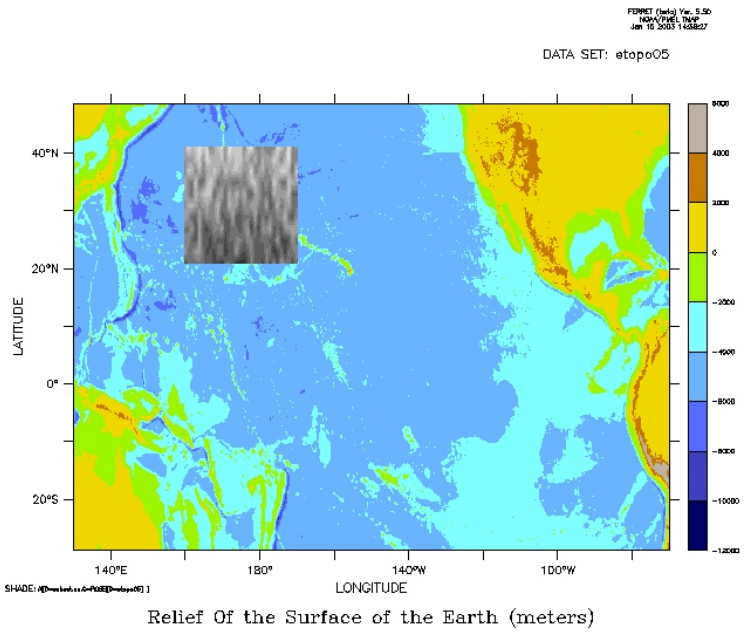


Figure 4_7

```
yes? USE etopo05
yes? SHOW GRID rose   ! We will want the names of the axes
    GRID GOZ1
 name       axis              # pts   start                end
 ETOPO05_X LONGITUDE         4320mr   0E                   0.079987W
 ETOPO05_Y LATITUDE          2161 r   90S                  90N
 normal    Z
 normal    T

yes? SET REGION W
yes? SHADE/PAL=land_sea rose[d=1]   ! draw the shade plot

yes? USE my_data.nc   ! The dataset containing the x=520:550 data
yes? SHOW AXIS xsub
 name       axis              # pts   start                end
 XSUB      LONGITUDE           31mr   160E(520)            170W(550)
   Axis span (to cell edges) = 31 (modulo length = 360)

yes? SHADE/OVER/PAL=greyscale a[GX=etopo05_x,GY=etopo05_y]
```

There is an implied void point in a sub-span modulo axis that fills the gap between the end of the axis and the start of the next modulo cycle. The data value at this point will always be the missing value flag (except for pseudo-variables such as "X[g=my_subspan_axis]"). Transformations such as smoothers do not operate across the void point.

In netCDF files, the modulo attribute is specified as follows:

1) Specify the modulo length of the axis with the attribute modulo = <value>, e.g. var:modulo=100;
2) The modulo attribute from previous netCDF files remains unchanged: modulo = " ". To set a modulo axis of standard length (360 degrees or one year). The modulo length is 360 degrees for a longitude axis, or one year for a climatological time axis.
3) The attribute value modulo = "FALSE", modulo = "NO", modulo="OFF" tells Ferret that the axis is not to be treated as modulo

Ch4 Sec3.8. Region Conflicts

Conflicting region information can be given to Ferret in obvious ways such as

```
LIST/I=1:3 I[I=1:10]
```

in which it is not clear if the request is for 10 points or for 3, or in subtler, disguised ways such as

```
LET A = I[I=1:10] LIST/I=1:3 A
```

In both examples Ferret would resolve the conflict by listing just the three values I=1:3.

Internally, Ferret uses the region closest to the variable to perform the calculation. Thus, in both of the examples above Ferret will perform the calculation on I=1:10, since the "[I=1:10]" directly modifies the variable name. If Ferret sees conflicting regions it attempts to use the regions further from the variable to clip the calculation. Thus 10 points are clipped to 3 in the above examples.

Unresolvable conflicts such as

```
LIST/I=11:13 I[I=1:10]
```

will result in a warning message that invalid limits have been ignored.

Ch4 Sec4. FERRET PROGRAM LIMITS

There are a number of hard limits in the Ferret code: the number of variables that may be defined, the number of datasets open at a time, the length of certain strings, etc. Some of these limits have been relaxed with successive Ferret versions as computing resources have expanded. Here are the limits as of Ferret version 5.41:

Parameter Name	Value	Description
memsize	6.4	Initial size of memory at startup, in Megawords. You can always change the memory at startup with the -memsize option (p. 6), or during a Ferret session with the SET MEMORY command. SHOW MEMOR gives the current size of the memory cache.
cmnd_buff_len	2048	Length of the command buffer. You can make long commands more readable using the continuation character backslash \ (p. 14)
Number of arguments to go scripts	99	Maximum number of arguments to a go script. Use the syntax ($nn) or $nn in the script. (p. 25)
Length of arguments to go scripts	511	Maximum length in characters of each argument to a go script.
maxvars	2000	Maximum number of all variables defined by SET DATA (including aliases USE and FILE)
max_uvars	2000	Maximum number of all user-defined variables (LET var =)
maxezvars	100	Maximum number of variables that can be read from a single delimited ASCII file, using SET DATA/FORMAT=DELIMITED (p. 402)
maxezfreefmtvars	20	Maximum number of variables that can be read in free format from a single ASCII file, e.g. in SET DATA/EZ/VARIABLES="var1,var2" (p.405)
maxdsets	100	Maximum number of data sets simultaneously open (as seen through SHOW DATA)
maxstepfiles	5000	Maximum number of files with time step data. These are read via descriptor files (p. 38). This is a limit on the cumulative sum of all files in all open multi-file data sets.
s_filename	128	Maximum length of the filenames listed within descriptor files (p. 289).
length of variable names	128	Maximum length of all variable names.
length of label text	2048	Maximum length of labels.
max_grids	500	Maximum number of static grids (grids defined by DEFINE GRID).
max_dyn_grids	1000	Total number of grids that can be defined at any time, static and dynamic. Dynamic grids are created by opening data sets and by implicit regridding operations such as strides (e.g. var[i=1:100:10]), regridding operations between grids of different dimensionality (e.g. temp4d[g=sst]), and external functions that create new grids (e.g. EOF_SPACE(A, F)) .

Parameter Name	Value	Description
max_lines	1000	Maximum number of static axes. Static axes are axes defined by DEFINE AXIS
max_dyn_lines	1500	Total number of axes, static and dynamic, thatcan be defined. Dynamic axes are defined by opening data sets and by implicit regridding operations such as strides (e.g. var[i=1:100:10]), regridding operations between grids or axes of different dimensionality (e.g. temp4d[gx=sst]), and external functions that create new grids (e.g. SAMPLEXY(sst, xpts, ypts))
maxlinestore	250000	Maximum number of coordinates in all irregular axes. This is the sum of all the coordinates of irregular axes currently defined via opening files and DEFINE AXIS, and includes storage for the edges of the grid cells defined by these axes. Coordinate storage may be recovered with the CANCEL AXIS command.
abstract_line_dim	20480	Dimension of the default abstract axis for reading ASCII data (p. 47). To read larger amounts of data, explicitly define an axis or grid.
ef_max_args	9	Maximum number of arguments that may be passed to an external function.
ef_max_work_arrays	9	Maximum number of work arrays defined by an external function for use by that function.
spec_size	250	Maximum number of levels in a spectrum, or palette file (.SPK) (p. 203)
pattern_num	50	Maximum number of patterns defined in a pattern file (.PAT) (p. 372)
year		Years in dates may take values from 0000 to 9999

Chapter 5: ANIMATIONS AND GIF IMAGES

Ch5 Sec1. OVERVIEW

There are two modes for animating in Ferret. One can animate "on the fly" in an interactive sesion, or a sequence of Ferret plots can be stored and then animated. For stored sequences of plots, each plot is stored as one frame in a movie file. Ferret stores movie frames in Hierarchical Data Format (HDF), a format designed by the National Center for Supercomputing Applications (NCSA). A movie file can then be displayed as an animated sequence of frames with NCSA's xds—X Data Slice (not distributed with Ferret; see the section in this chapter "Displaying an HDF movie" (p. 177), for details). A series of gif images can also be animated, see Ch5 Sec1.2 below.

Ch5 Sec1.1. Animating on the fly

In a Ferret session, display an animation with the command,

```
yes? REPEAT/ANIMATE[/LOOP=n]
```

to start an animation sequence. Given LOOP=n, the entire animation sequence will repeat n times.

Example:

```
yes? set data coads_climatology
yes? repeat/l=1:12/animate/loop=5 (shade sst; go fland)
```

NOTE: In order to properly display, it is necessary to have backing store enabled for the Xserver.

Ch5 Sec1.2. Note on using whirlgif to make a movie

The following sections detail making movies with HDF, but another method has been brought to our attention. An easy way to make movies from gif files generated by Ferret is a public domain utility called whirlgif. The documentation for whirlgif indicates that it is available for a variety of systems.

Whirlgif is extremely easy to use:

1. Make your gif files with a Ferret command like:

```
yes? REPEAT/J=1:36 (GO scriptfile `j`; FRAME/FILE=whirl-`j`.gif)
```

where the scriptfile uses the argument *j* to determine the plot characteristics. See sections later in this chapter for more on the REPEAT command (p. 177) and creating GIF files (p. 180).

2. Make a file (for example call it whirlgif-infile) that consists of a list of the gif files (including repeats if you want):

```
> more whirlgif-infile
   whirl-1.gif
   whirl-2.gif
…
```

This file can be as long as you want and may specify files more than once to repeat any of the images if you wish.

3. From the unix command line use whirlgif to make the movie:

```
> whirlgif -o movie_filename.gif -i whirlgif-infile
```

That's it. Whirlgif simply concatenates the gif files with some connecting information needed to do the animation. The resulting movie gif file is just about as large as the sum of the input frames.

These show nicely on the web, or you can use xanim (under unix) to view locally.

Download whirlgif from http://www.msg.net/utility/whirlgif/
or the mirror site: http://www.danbbs.dk/~dino/whirlgif/index.html

which has extensive documentation. But we have found that it is a simple program that works without much study.

Ch5 Sec2. CREATING AN HDF MOVIE

Creating a movie requires two steps:

1) designate an output file with SET MOVIE
2) generate a sequence of frames with REPEAT and FRAME

See commands SET MOVIE (p. 419), CANCEL MOVIE (p. 328), SHOW MOVIE (p. 440), FRAME (p. 358), and REPEAT (p. 388) in the Commands Reference section of this manual.

Example: basic movie

```
yes? SET DATA coads_climatology       !specify data set
yes? SET REGION/@W                    !specify Pacific Ocean
yes? LET/TITLE="SST Anomaly" SST_ANOM = SST - SST[L=1:12@AVE]
yes? REPEAT/L=1:12 (FILL sst_anom; FRAME/FILE=my_movie.mgm)
                                      !filled contour of sea surface\
```

```
                                    temp anomaly captured and\
                                    written to HDF file
```

Optionally, ".mgm" will be assigned to the movie file.

REPEAT executes its argument (in the above example, FILL) successively for each timestep specified. REPEAT can have multiple arguments separated by semi-colons and enclosed in parentheses.

FRAME is a stand-alone command, but also a qualifier for the graphical output commands PLOT, CONTOUR, FILL (alias for CONTOUR/FILL), SHADE, VECTOR and WIRE.

The saved animation frames are exactly the size and shape of the window from which they are created. Thus a large window results in a larger, slower animation that demands more disk space and memory to play back. The SET WINDOW/SIZE= command is generally used to specify minimally acceptable frame size.

See section "Advanced Movie-making" (p. 177), for more examples.

Note that when making an HDF movie you should not start Ferret with the -unmapped option.

Ch5 Sec3. DISPLAYING AN HDF MOVIE

Viewing a movie requires software which is not included with the Ferret distribution (although in some cases we have made the binary available in Ferret's anonymous ftp area). NCSA's X Data Slice reads HDF files and is available via anonymous ftp from NCSA. It requires about 1.7Mb of disk space. NCSA's ftp server is

ftp.ncsa.uiuc.edu login id is "anonymous", give your e-mail address as the password

Consult the README files you will find there for instructions on obtaining X Data Slice. Other utilities from NCSA can also be used for animations.

Ch5 Sec4. ADVANCED MOVIE-MAKING

Ch5 Sec4.1. REPEAT command

The REPEAT command is quite flexible. It allows you to repeat a sequence of commands, not just a single command as in the basic example above. You can give the GO command as an argument to REPEAT. The following examples demonstrate these techniques.

Note: MODE VERIFY must be SET (this is the default state) for loop counting to work.

Example 1

Note the method at the start of this chapter for making movies from a sequence of GIF files and the whirlgif utility. (p.175)

Example 2

Here we give multiple arguments to REPEAT; note the semi-colon separation and the parentheses. Note that FRAME, in this example, is used as a stand-alone command.

```
yes? REPEAT/L=1:12 (FILL SST; GO fland; FRAME/file=my_movie.mgm)
```

Example 3

In this example we use the REPEAT command to pan and zoom over a sea surface temperature field.

```
SET DATA coads_climatology
SET REGION/L=1
SET REGION/X=120E:60W/Y=45S:45N
SHADE sst; GO fland

! ZOOM
REPEAT/K=1:5 (SET REGION/DX=+8:-8/DY=+8:-8; SHADE sst; GO fland; FRAME)

! PAN
REPEAT/K=1:5 (SET REGION/DX=+5; SHADE/LEV=(20,30,.5) sst; FRAME)
```

Example 4

In this example the user calls setup_movie.jnl (text included below), title.jnl, which creates a title frame, then repeats main_movie.jnl (text included below) for each time step desired. Finally, the user adds a frame of credits at the end of the movie. Each of the scripts would end with the FRAME command (except setup_movie). Using GO scripts as arguments to REPEAT allows you to customize the plot with many commands before finally issuing FRAME, as the text of main_movie.jnl below demonstrates.

```
yes? ! make the movie
yes? GO setup_movie
yes? GO title
yes? REPEAT/L=1:12 GO main_movie
yes? GO credits

! Setup_movie.jnl
SET WINDOW/SIZE=.45/ASPECT=0.7
SET MOVIE/file=my_movie.mgm
SET DATA coads_climatology
SET REGION/X=130E:75W/Y=8S:8N
SET MODE CALENDAR:months
GO bold
PPL SHAKEY ,,.15,.2
PPL AXLEN 8.8,4.8

! Main_movie.jnl
```

```
FILL/SET_UP/LEVELS=(16,31,1) sst
PPL LABS; PPL TITLE
PPL FILL
LABEL 210,9.5,0,0,.22 @TRCOADS MONTHLY CLIMATOLOGY (1946-1989)
LABEL 210,-12,0,0,.22 @TRSEA SURFACE TEMPERATURE (DEG C)
LABEL 130,11,-1,0,.22 @TR'LAB4'
FRAME
```

Note: If you use the FILL command, we suggest that you use SHADE while customizing and fine-tuning your movie, then use FILL for the final run. SHADE is much faster.

Ch5 Sec4.1.1. Initializing the color table

If you create a movie with a title frame, or a first frame which otherwise uses different colors than the rest of the movie, you should be aware of an HDF peculiarity: all the colors that you plan to use in your movie must be in the first frame, or else color behavior will be unpredictable when you animate.

To "reserve" the colors you need, use overlapping full-window viewports. Make a representative plot in the title frame, then cover over it with either a black or white rectangle and finally write the title text. Here is a script which initializes the color table while creating a title frame.

```
! define 3 identical full-frame viewports
DEFINE VIEW full1;  DEFINE VIEW full2;  DEFINE VIEW full3

! draw frame one of the movie in full color
SET VIEW full1
SET DATA coads_climatology
SHADE/LEVELS=(16,31,1)/L=1 sst                  ! dummy frame

! white-out over the picture
SET VIEW full2
GO setup_text
SHADE/PALETTE=white/NOLAB/NOKEY/i=1:2/j=1:2  (i+j)*0

!put on title frame labels (using [0,1] coordinate space)
SET VIEW full3
GO setup_text
PPL PLOT
LABEL .5,.7,0,0,.3 @TRMy Title
PLOT/VS/LINE/OVER/NOAX/NOLAB/COLOR=black {2,8}, {.55,.55}
PLOT/VS/LINE/OVER/NOAX/NOLAB/COLOR=black {2,8}, {.53,.53}
LABEL .5,.4,0,0,.2 @CRBy me

!capture the title frame and clean up
FRAME
GO cleanup_text
```

Ch5 Sec4.1.2. Making movies in batch mode

Ferret, like other Unix applications, can be run in "batch" mode by redirecting standard input and output. Thus

```
ferret -unmapped <movie_commands.jnl >&movie.log&
```

will make a movie running in background mode based on the commands in file movie_commands.jnl logging standard output and standard error in file movie.log.

Note, however, that when used in this mode to make a movie Ferret will still require access to an X windows display (as in "setenv DISPLAY node:0"). To eliminate this requirement we recommend the use of the X11R6 "virtual frame buffer" (Xvfb). This application permits the movie frames to be generated in the absence of any physical display device. Consult your system manager for the availability of X11R6 for your system.

Ch5 Sec5. CREATING GIF IMAGES

GIF is a highly compressed format suitable for single images. (Ferret will not directly create GIF89 animations.) The procedure for creating a GIF image is nearly identical to the creation of a single frame of an HDF file. The modification is generally just to select a file name with the ".gif" extension; Ferret will automatically sense this as a request to create a GIF-formatted image file. Alternatively, any file name can be used if the GIF format is specified explicitly using

```
FRAME/FORMAT=GIF
```

If a number of GIF images are created using the same file name Ferret will automatically rename subsequent versions with a version number. Thus a repeat loop can be used to generate many GIF images.

Example:

```
REPEAT/L=1:12(FILL sst; GO fland; FRAME/file=myimage.gif)
```

Note: In this mode of grabbing an image, Ferret creates a GIF file by requesting the image back from your screen (your X server). In order for Ferret to correctly grab the image, the X server should be configured to be running either in 8-bit PseudoColor mode (i.e. direct color) or 24-bit TrueColor mode (i.e. indexed color) with X server backing store enabled. If the X server is configured in 16-bit TrueColor (also indexed color) mode, Ferret will be unable to grab the GIF image from the X server.

An alternative approach to creating GIF's (which does not share this restriction) is to invoke Ferret with the -gif command line switch "ferret -gif" (p. 6).

Ch5 Sec6. CREATING MPEG ANIMATIONS

MPEG animations can be created from the outputs of the FRAME command—either HDF animation files or a sequence of GIF images. Various public domain utilities are available to perform the conversion from Ferret's output formats into MPEG animations. The routine

hdf2mpeg (available in 2002 from ftp://ftp.ncsa.uiuc.edu/HDF/HDF/contrib/NCSA/HDF2MPEG/) can be used to convert HDF files into MPEG animations; mpeg_encode (available from mm-ftp.CS.Berkeley.EDU in /pub/multimedia/mpeg/encode) can be used to convert sequences of GIF files. New and improved routines may have become available since the time of this writing. See further documentation on this topic in the FAQ file from the Ferret home page.

Chapter 6: CUSTOMIZING PLOTS

Ch6 Sec1. OVERVIEW

Detailed control is possible over most aspects of Ferret graphical outputs. A custom modification will require the user to either add a qualifier to a Ferret command or communicate directly with the graphical package PPLUS, which is contained inside of Ferret. The most commonly used PPLUS commands are listed in the following sections of this chapter. Consult the PLOT PLUS for Ferret manual for complete command lists and the specifics of command syntax.

Ferret communicates with PPLUS by sending a sequence of commands to PPLUS (the command PPL ECHO ON causes the sequence of commands that Ferret sends to PPLUS to be logged in the file fort.41.). The user can give further commands to PPLUS directly using the Ferret command PPL (e.g., ***yes? PPL AXLEN 10,7***). Some results can be attained in two ways—with either Ferret or PPLUS commands. However, the interaction of the two is complex and the inexperienced user may get unexpected results, so when possible, use only Ferret commands.[1]

PPLUS uses a deferred mode of output—various commands are given to PPLUS which describe the plot state but produce no immediate output; the entire plot is then rendered by a single command. Some plot states (e.g., axis labels) are set by Ferret with every plotted output; to customize these states it is necessary to use the /SET_UP qualifier (which sets up the plot inside of PPLUS) and then modify the state with direct PPL commands. Other plot states are never set by Ferret and, if modified at any time, remain in their specified state for all subsequent plots. Still other states are modified by Ferret only under special circumstances. Here is a very simple customization (Figure 6_1):

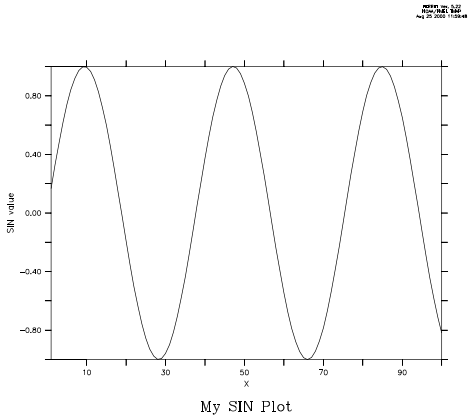


Figure 6_1

[1] Note that throughout this discussion a distinction has been made between Ferret commands and PPLUS commands. In reality, the user issues Ferret commands only. "PPLUS commands" in this context refers to PPLUS commands issued via the Ferret command PPL.

```
yes? PLOT/X=1:100/TITLE="My SIN Plot"/SET_UP sin(x/6)   !use /SET_UP
yes? PPL YLAB "SIN value"
yes? PPL PLOT
```

The examples throughout this chapter show how the /SET_UP qualifier on graphics commands can be used to delay rendering of a plot while the user modifies plot appearance with PPLUS commands.

Below is a list of PPLUS commands which are reset by Ferret. Please see the the PPLUS Users Guide for details of PPLUS syntax. (p. 485)

PPLUS command	**when reset by Ferret**
XFOR, YFOR	reset for every plot
XLAB, YLAB	reset for every plot
XAXIS, YAXIS	reset for every plot
LABS	reset for every plot
ALINE	reset for every plot
TAXIS OFF	reset for every plot
TITLE	reset for every plot
TICS	reset for every plot (small tic size, only)
WINDOW ON	reset for every plot
PEN 1,n	reset for every plot
LIMITS	reset for every plot
ORIGIN	reset by SET WINDOW/ASPECT and SET VIEWPORT; Y origin may be shifted to accommodate many line style keys
AXLEN	modified by SET WINDOW/ASPECT and SET VIEWPORT
VIEWPORT	modified by WIRE/VIEW
LEV	modified by CONTOUR and SHADE unless /LEVELS_SAME given
VECSET	modified by VECTOR unless /LENGTH_SAME given
WINDOW	modified for "fresh" plots but not for overlay plots

Ch6 Sec2. GRAPHICAL OUTPUT

Ch6 Sec2.1. Ferret graphical output controls

Ferret command	Function
CONTOUR	produces a contour plot of a single field
FILL	alias for CONTOUR/FILL; produces color-filled contour plot
PLOT	produces a line or symbol plot of one or more arrays
SHADE	produces a shaded representation (rectangular cells)

Ferret command	Function
VECTOR	produces a vector arrow plot
WIRE	produces a 3D wire frame plot
SET WINDOW	manipulates graphics windows
SET VIEWPORT	places graphics output into a sub-window (pane)

Ch6 Sec2.2. PPLUS graphical output commands

The plot commands, in the table below, can be customized using /SET_UP to delay display. The PLOT/SET_UP is followed by PPLUS commands which customize the settings for axes, labels, plot layout, and so on. Then the plot will ultimately be rendered using a PPLUS graphical output command (not the Ferret counterpart). A customized contour or filled-contour plot is rendered with PPL CONTOUR, a wire frame plot with PPL VIEW and so on. Please see the overview of this chapter (p. 183) and also the discussion in the Commands Reference section about PPLUS (p. 387).

In the following sections, there is a "PPLUS commands" subsection detailing which PPLUS commands are used for each type of customization. See the examples in those sections, and cross-references to the PPLUS command syntax in the PPLUS manual (Appendix B).

Command	Function
CONTOUR	makes a contour plot
PLOT	plots x-y pairs for all lines of data
PLOTUV	makes a stick plot of vector data
SHADE	makes a shaded representation
VIEW	makes a wire frame plot
VECTOR	makes a plot of a vector field

The graphical output command PLOTUV can be used to make stick plots easily, as the following time series example shows.

```
yes? SET DATA coads; SET REGION/X=180/Y=0/L=400:500
yes? PLOT/SET uwnd, vwnd
yes? PPL PLOTUV
```

Ch6 Sec3. AXES

By default, Ferret displays X- and Y-axes with tics and numeric labels at reasonable intervals and a label for each axis. Time axes are also automatically formatted and used as needed. These

axis features can be modified or suppressed using the following Ferret direct controls and PPLUS commands.

Ch6 Sec3.1. Ferret axis formatting

The following qualifiers are used with graphical output commands PLOT, VECTOR, SHADE, and CONTOUR to specify axis limits, tic spacing, and possible axis reversal:

Ferret qualifers
/HLIMITS, /VLIMITS, /NOAXIS

The /HLIMITS and /VLIMITS qualifiers use the syntax /HLIMITS=lo:hi:delta. Tic marks are placed every "delta" units, starting at "lo" and ending at "hi". Every other tic mark is labeled. "delta" may be negative, in which case the axis is reversed.

The /NOAXIS qualifier removes both X and Y axes from the plot. This is particularly useful for plots using curvilinear coordinates (map projections) where the final axis values represent transformed axis values rather than world coordinates.

The following arguments to SET MODE and CANCEL MODE determine axis style (e.g., SET MODE CALENDAR:days) :

Ferret arguments

CALENDAR
LATIT_LABEL
LONG_LABEL

See the next section for more about customizing axes, tic marks, labels and coordinate labels.

See the Commands Reference section of this manual (p. 323) for more information.

Ch6 Sec3.2. PPLUS axis commands

PPLUS commands can be used to customize axis settings. Note that Ferret makes settings for all of these automatically; you will only need to make PPLUS calls to change the axis properties. See the examples below, and the section on PPLUS graphical commands (p. 185) for more on the syntax to make PPLUS calls.

Command	Function
XAXIS*	controls numeric labeling and tics on the X axis (redundant with /HLIMITS) (p. 552)
YAXIS*	controls numeric labeling and tics on the Y axis (redundant with /VLIMITS) (p. 552)
AXATIC	sets number of large tics automatically for X and Y (p. 525)
AXLABP	locates or omits axis labels at top/bottom or left/right of plot (p. 525)
AXLEN**	sets axis lengths (p. 525)
AXLINT	sets numeric label interval for axes every nth large tic (p. 525)
AXLSZE	sets axis label heights (p. 525)
AXNMTC	sets number of small tics between large tics on axes (p. 525)
AXNSIG	sets number of significant digits in numeric axis labels (p. 525)
AXSET	allows omission of plotting of any axis (redundant with /AXES=) (p. 526)
AXTYPE	sets axis type (linear, log, inv. log) for x- and y-axis (p. 526) (See also /HLOG,/VLOG qualifiers on plot commands)
TICS	sets axis tic size and placement inside or outside axes (p. 546)
XFOR*	sets format of x-axis numeric labels (p. 189, p. 552)
YFOR*	sets format of y-axis numeric labels (p. 189, p. 553)
XLAB*	sets label of x-axis (p. 553)
YLAB*	sets label of y-axis (p. 553)
TXLABP	establishes time axis label position (or absence) (p. 547)
TXTYPE*	sets the style of the time axis (p. 548)
TXLINT*	specifies which time axis tics will be labeled (p. 547)
TXLSZE	sets height of time axis labels (p. 548)
TXNMTC	sets number of small tics between large tics on time axis (p. 548)

* issued by Ferret with every relevant plot

** issued by Ferret upon SET WINDOW/ASPECT or SET VIEWPORT

Examples

1) Plot with no axis labels (character or numeric) and no tics (Figure 6_2). (Equivalent to

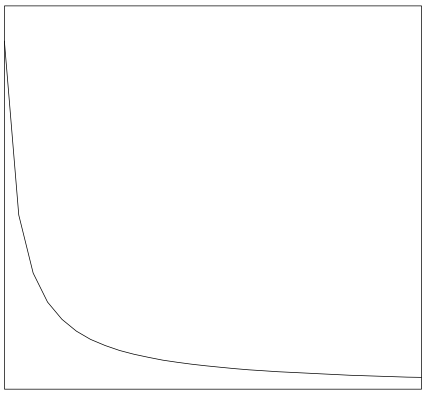


Figure 6_2

yes? GO box_plot PLOT/I=1:10/NOLABEL 1/i)

```
yes? PLOT/i=1:30/NOLABEL/SET 1/i
yes? PPL AXLABP 0,0                    !turn off numeric labels
yes? PPL TICS 0,0,0,0                  !suppress small and large tics
yes? PPL PLOT                          !render plot
yes? PPL TICS .125,.25,.125,.25        !reset tics to default
yes? PPL AXLABP -1,-1                  !reset numeric labels
```

2) customize x-axis label (Figure6_3); XLAB always reset by Ferret)

```
yes? PLOT/SET/i=1:100 sin(x/6)
yes? PPL XLAB My Custom Axis Label
yes? PPL PLOT
```

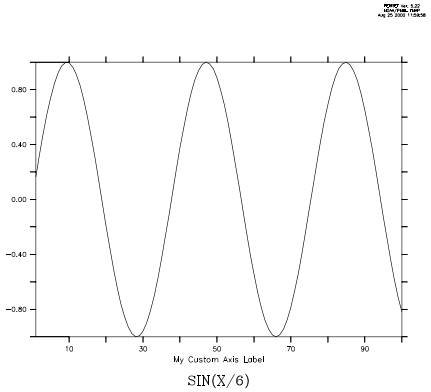


Figure 6_3

3) specify tic frequency for y axis

```
yes? PLOT/i=1:30/YLIM=0:1:.2 1/i
```

4) Specify the size and location of tic marks on the axes. The PPLUS tics command is

```
ppl tics,smx,lgx,smy,lgy,IX,IY
```

IX and IY are 1 for tics inside the plot box, 0 to straddle the axis line, and -1 for tics outside the axis with -1 as default. These commands put large tics inside the axes.

```
yes? SHADE/SET/i=1:100/j=1:15 sin(x/6)*10./j
yes? PPL TICS .0,.35,.0,.35,1,1
yes? PPL SHADE
```

See also the /GRATICULE qualifer available on all plotting commands (PLOT/GRATICULE, CONTOUR/GRATICULE, etc.)

The PPLUS commands XFOR and YFOR control the format of the labels for coordinates along the axes. These apply FORTRAN format syntax to override the automatic formatting.

Starting with Ferret v6.0, the PPL XFOR (p. 552) and PPL YFOR commands may be used with longitude and latitude axes to specify that the coordinates that are labeled along the axis should be shown as degrees minutes (and optionally seconds) instead of the default labels of degrees and decimal fractions of degrees.

Example:

```
yes? use coads_climatology
yes? SET VIEW upper
yes? SHADE  airt  ! Default axis formatting
yes? SET VIEW lower
yes? SHADE/SET airt   ! Showing XFOR (fortran format) and YFOR (dm)
yes? PPL XFOR (F7.2)
yes? PPL YFOR (dm)
yes? PPL SHADE
```

Ch6 Sec3.3. Overlaying symbols on a time axis

To overlay symbols or mark-up on a plot which has a formatted time axis (dates and times) it is necessary to specify positions using the internal time encoding of that axis. Typically, the easiest way to achieve this is to define a variable, say TT, which is the time encoding. This example illustrates.

Example:

demonstrate PLOT/VS and POLYGON over time axes (Figure 6_4)

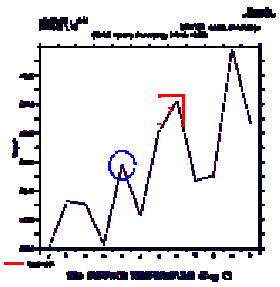


Figure 6_4

```
USE coads_climatology
LET xsqr = {-1,1,1,-1}                ! coordinates of a unit square
LET ysqr = {-1,-1,1,1}

LET xcircle = COS(6.3*i[i=1:42]/40)    ! coordinates of unit circle
LET ycircle = SIN(6.3*i[i=1:42]/40) ! Notice the units of the time axis
SHOW GRID/L=1:3 sst

PLOT/X=180/Y=0 sst     !  draw a  time series plot
LET tt = T[GT=sst]          ! tt is the coordinates along the T axis
! place an "X" at the value exactly at 7-aug
! "@ITP" causes interpolation to exact location
LET t0   = tt[T="7-aug-0000"@itp]
LET val0 = sst[X=180,Y=0,T="7-aug-0000"@itp]
PLOT/VS/OVER/NOLAB/SYM=2/LINE=8 t0,val0

! put a box around the "X"
POLYGON/OVER/LINE=8/TITLE="Special region" t0+500*xsqr, 0.05*ysqr+val0

! place an "X" on the data point nearest to 15-may
! Note that @ITP is absent, so behavior is set by MODE INTERPOLATE
LET t1   = tt[t="15-may-0000"]
LET val1 = sst[x=180,y=0,t="15-may-0000"]
PLOT/VS/OVER/NOLAB/SYM=2/LINE=10 t1,val1
! put a circle around the "X"
PLOT/VS/OVER/LINE=10/nolab t1+500*xcircle,0.05*ycircle+val1
```

Example (continued):

mark-up over a Hofmuller diagram (Figure 6_5)

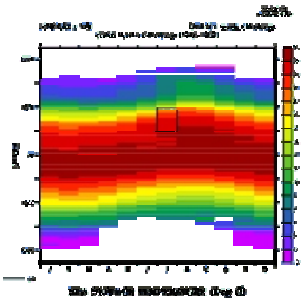


Figure 6_5

```
SHADE/X=180 sst                                  !  latitude vs time plot
LET tlo = tt[T="1-jul-0000"@itp]
LET thi = tt[T="1-aug-0000"@itp]
POLYGON/OVER/LINE=7/PAL=gray/PAT=lite_up_left_to_right {`tlo`, `thi`,`
thi`, `tlo`},  {20, 20, 40, 40}
```

Ch6 Sec4. LABELS

Ferret, by default, produces labeled axes, a plot title, documentation about the data set, the plot axes normal to the plot, nd a signature (current date and Ferret version number) when a plot is rendered. The /NOLABELS qualifier suppresses the plot title, the documentation and signature, and the axis labels of independent axes. Note that you can use the LABEL command to add any labels that you need.

Ch6 Sec4.1. Adding labels

The Ferret command LABEL adds a label to a plot and takes the following arguments:

```
yes? LABEL xpos,ypos,center,angle,size text
```

where xpos and ypos are in user (axis) units, size is in inches, angle is in degrees (0 at 3 o'clock) and center is -1, 0, or +1 for left, center, or right justification. There is an example in the section below on PPLUS label commands (p. 196). The label position will adjust itself automatically when the plot aspect ratio or the viewport is changed.

If you prefer to locate labels using inches rather than using data units issue the command

```
yes? LABEL/NOUSER xpos,ypos,...
```

Note, however, that the layout of a plot in inches—lengths of axes, label positions, etc.—shifts with changes in window aspect ratio (SET WINDOW/ASPECT) and with the use of viewports. Labels specified using LABEL/NOUSER will need to be adjusted if the aspect ratio or viewport is changed.

Beginning with Ferret v5.53, long labels may be specified, up to 2048 characters long. This applies to all kinds of labels: titles, axis labels, and moveable labels. To create multiple-line labels, use the separator <NL> to locate the line-breaks. If centered labels are requested, each line is centered separately. See examples on pages 197 and 197. Also try the demo script multi_line_demo.jnl for examples of this usage.

Notes:

1) If you use the command PPL LABS instead of LABEL, be aware that when defining a new movable label, all lower-numbered labels must already be defined.

2) The Ferret command LABEL is an alias for PPL %LABEL. PPLUS does NOT consider a label created with %LABEL to be a movable label. Consequently, no label number is assigned and the label cannot be manipulated as a movable label.

3) %LABEL is an unusual command in that the label appears on the plot immediately after the command is given, rather than being deferred. This has ramifications for the user who has multiple plot windows open and is in MODE METAFILE, since a metafile is not closed until a new plot is begun. If the user produces a plot in window B, and then returns to a previous window A and adds a label with LABEL, that label will appear on the screen correctly, but will be in the metafile corresponding to window B.

4) The LABWID function (Ferret version 5.81 and higher) returns the width in plot inches of a label in a given font and size. This can be used to position other plot elements relative to a label.

```
LABEL/NOUSER `xpos`, `ypos`, -1, 0, 0.15, "my label"
LET wid = LABWID ("my label", 0.15)
IF `next_label GT 0` THEN\
  LABEL/NOUSER `xpos+wid+0.15`, `ypos`, -1, 0, 0.15, "second label"
```

Example

```
yes? PLOT/I=1:100  sin(i/6)
yes? LABEL 50, 1.2, 0, 0, .2  @P2MY SIN PLOT
```

Ch6 Sec4.2. Listing labels

The PPLUS command PPL LIST LABELS can be used to list the currently defined labels. For example,

```
yes? PPL LIST LABELS
@ACSEA SURFACE TEMPERATURE (Deg C)
@ASLONGITUDE
@ASLATITUDE

          XPOS       YPOS     HGT   ROT   UNITS
LAB 1  8.000E+00  7.200E+00 0.060    0  SYSTEM  @ASFERRET Ver. 4.40
LINE PT:    0.000E+00 0.000E+00  NO LINE     CENTER JUSTIFY LABEL
LAB 2  8.000E+00  7.100E+00 0.060    0  SYSTEM  @ASNOAA/PMEL TMAP
LINE PT:    0.000E+00 0.000E+00  NO LINE     CENTER JUSTIFY LABEL
LAB 3  8.000E+00  7.000E+00 0.060    0  SYSTEM  @ASOct 22 1996 09:24
LINE PT:    0.000E+00 0.000E+00  NO LINE     CENTER JUSTIFY LABEL
LAB 4  0.000E+00  6.600E+00 0.120    0  SYSTEM  @ASTIME : 16-JAN
LINE PT:    0.000E+00 0.000E+00  NO LINE     LEFT   JUSTIFY LABEL
.
.
.
```

The first three lines of output show the plot title, the X axis label, and the Y axis label. These labels are controlled by the PPL TITLE, PPL XLAB, and PPL YLAB commands, respectively. The three characters "@AS" indicate the font of the label—in this case "ASCII Simplex" (see the section in this chapter, "Fonts," p. 207).

Next is a table of "movable labels"—labels that were defined using the PPL LABS command. Labels are generally simpler to control with the GO unlabel and LABEL commands described in the following sections, rather than with the PPL LABS command.

Each label is described with two lines. The column headers refer to the first of the two. The coordinates of each label, (XPOS,YPOS), may be in units of "inches" or may be in the units of the axes. This is reflected in the UNITS field of the output, which will contain "SYSTEM" if the coordinates are in inches or "USER" if the coordinates are axis units. (The /NOUSER qualifier on the PPL LABS command is used to indicate that coordinates are being given in inches.) Coordinates are calculated relative to the axis origins. The PPL HLABS and PPL RLABS commands control label height and rotations, respectively.

The second line of the label description contains information about an optional line on the plot which can be used to point to the label (refer to the PPLUS command LLABS or see the section in this chapter, "Positioning labels using the mouse pointer," p. 198). At the end of this line is the text of the movable label.

In addition to PPLUS LIST LABELS, you can also issue a SHOW SYMBOLS command; the labels that are automatically generated are available as symbols,

```
yes? SHOW SYMBOLS       ! lists all symbols that have been defined
yes? SHOW SYMBOLS LAB*  ! lists symbols starting with LAB
```

Ch6 Sec4.3. Removing movable labels

Removing a movable label is a two step process: identifying the label number and then deleting the label. PPLUS internally refers to all movable labels with label reference numbers. The PPLUS command LIST LABELS will list the PPLUS labels and the text strings they contain. Then the user can use "GO unlabel n", where n is the reference number, to delete a label.

Example
In this example we plot the same figure in two viewports, one plot with the default "signature," and one plot with the signature removed (Figure 6_6). The SHOW SYMBOLS command lists all the Ferret symbols that are defined for the plot; LAB1, LAB2, and LAB3 are always the three lines of the signature. (Alternatively, PPL LIST LABELS will list all currently defined labels, or PPL LISTSYM will list all symbols.)

```
!upper viewport has a "signature"
yes? PPL BOX on
yes? SET VIEW upper
yes? PLOT/I=1:100 sin(i/6)
yes? SHOW SYM
...
LAB1 = "FERRET Ver. 5.70"
LAB2 = "NOAA/PMEL TMAP"
LAB3 = "Jan  5 2005 08:54:22 "
...

!in the lower viewport
!the signature is removed as follows

yes? SET VIEW lower
yes? GO unlabel 1
yes? GO unlabel 2
yes? GO unlabel 3
yes? PPL PLOT
```

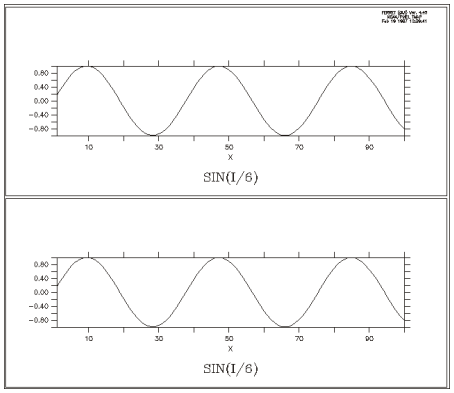


Figure 6_6

Ch6 Sec4.4. Axis labels and title

Special commands and special logic govern the labels of axes and titles. Use the PLOT+ commands XLAB, YLAB, and TITLE in conjunction with the Ferret plotting qualifier /SET_UP to

modify the labeling choices that Ferret makes. These are discussed in the section below, PPLUS label commands (see p. 195).

For two-dimensional plots (CONTOUR, FILL) Ferret will label the plot axes with the titles and units from the appropriate axes of the grid. The command SHOW GRID can be used to see the labels that will be used. The title will be the title of the variable (see SHOW VARIABLE, p. 442, and SHOW DATA/VARIABLE, p. 435) modified by the units and comments about transformations in parentheses.

For one-dimensional plots (PLOT) other than PLOT/VS the independent axis will be labeled using the title and units from the appropriate axis of the grid. The dependent axis will be labeled with the units of the variable being plotted. The title will be labeled as for two-dimensional plots.

For output of the PLOT/VS command the axes will be labeled with the titles of the variables (see SHOW VARIABLE, p. 442, and SHOW DATA/VARIABLE, p. 435) each modified by its units and comments about transformations in parentheses.

Ch6 Sec4.5. Ferret label controls

In addition to LABEL (discussed above, page 191), Ferret controls include the /NOLABELS qualifier, which suppresses default plot title, documentation and signature, axis labels, and /TITLE qualifier to graphical output commands PLOT, SHADE, CONTOUR, VECTOR, and WIRE:

Ferret qualifiers

/NOLABELS
/TITLE=

and arguments to SET MODE (p. 409) and CANCEL MODE (p. 327):

Ferret arguments

SET MODE ASCII_FONT
SET MODE CALENDAR
SET MODE LATIT_LABEL
SET MODE LONG_LABEL

Ch6 Sec4.6. PPLUS label commands

Ferret stores the text strings of the following labels in PPLUS symbols. The symbol names are:

symbol name	label
LABTIT	title label
LABX	X axis label
LABY	Y axis label
LABn	nth movable label

PPLUS commands can be used to customize labels. See the example below, and the section on PPLUS graphical commands (p. 185) for more on the syntax to make PPLUS calls. As stated above, PPLUS commands regarding movable labels are largely superceded by the Ferret command LABEL and "GO unlabel n". However the /SETUP qualifier on a plot command in conjuction with PPLUS commands LABSET, TITLE, XLAB, and YLAB are used to modify the labels that Ferret automatically puts on plots. See the section on PPLUS graphical commands for more on calling PPLUS plot commands (p. 185)

Command	Function
LIST LABELS	shows the currently defined labels (p. 537)
LABSET	sets character heights for labels (p. 534)
TITLE*	sets and clears main plot label (p. 546)
XLAB*	sets label of X axis (p. 553)
YLAB*	sets label of Y axis (p. 553)
LABS*	makes, removes, or alters a movable label (p. 533) (redundant with LABEL command)
HLABS	sets height of each movable labe (p. 532)l
RLABS	sets angle for each movable label (p. 543)
LLABS	sets start position for and draws a line to a movable label (p. 538)
XFOR	sets formatting of the coordinate labels along the x axis
YFOR	sets formatting of the coordinate labels along the x axis

* issued by Ferret with every relevant plot

Example

This example customizes a plot using PPLUS label controls. The LABSET command (See p. 534) is used here to control the size of the main label, x-label, and y-label. The Ferret LABEL command is used to add a label.

```
yes? PLOT/I=1:100/SET_UP   i * sin(i/6)
yes? PPL LABSET 0.3, 0.08, 0.3
yes? PPL TITLE
yes? PPL YLAB "Modiified SIN function"
yes? PPL PLOT
yes? LABEL 10.,20,-1,30,0.2 "Angled label"
```

Beginning with Ferret v5.53, long labels may be specified, up to 2048 characters long. To create multiple-line labels, use the separator <NL> to locate the line-breaks. If centered labels are requested, each line is centered separately. See the demo script multi_line_demo.jnl for examples of this usage.

Example

Use of long axis labels. Use the backslash continuation character for better readability.

```
yes? PLOT/I=1:100/TITLE="  "/SET  i*cos(i/8)
yes? PPL YLAB "A four-line y label.<nl>second line\
<nl>third line<nl>fourth line"
yes? PPL XLAB "a two-line X label. <nl>COSINE function"
yes? PPL PLOT
```

Ch6 Sec4.7. Positioning labels relative to other plot elements

Once a plot has been made, we can use the location and size of plot elements such as axis lengths to position any labels we would like to add. A number of global symbols are defined when a plot is drawn. See the section on "special symbols" for a compete list of these. (p. 219)

Use the LABEL command to position a label. To position a label using page inches, use LABEL/NOUSER which takes the units to be inches from the origin. When plotting in a viewport, plot inches are measured from the origin of the viewport.

Example:

```
yes? plot/i=1:100 (i/2)*sin(i/6)

  ! Put a label in the lower right, use user units
yes? label/user `($xaxis_max)`, `($yaxis_min)`, 1, 0, .2, "@P2lower
right"
 !-> PPL %LABEL/user 100, -50, 1, 0, .2, "@P2lower right"

  ! Use plot inches. Put a label just inside the plot area
yes? label/nouser `($ppl$xlen)/2`, `($ppl$ylen) - 0.4`, 0, 0, 0.2,
"@P2center top"
 !-> PPL %LABEL/nouser 4, 5.6, 0, 0, 0.2, "@P2center top"

  ! Put a label in the lower left, making sure it's not off the page.
yes? let xpl = -1*MIN(1,`($ppl$xorg)`)
yes? let ypl = -1*MIN(1,`($ppl$yorg)`)
yes? label/nouser `xpl`, `ypl`, -1, 0, 0.2, "@P2lower left"
 !-> PPL %LABEL/nouser -1, -1, -1, 0, 0.2, "@P2lower left"
```

Beginning with Ferret v5.53, long labels may be specified, up to 2048 characters long. To create multiple-line labels, use the separator <NL> to locate the line-breaks. If centered labels are

requested, each line is centered separately. See the demo script multi_line_demo.jnl for examples of this usage.

Example:

This is one LABEL command, used to put a block of text on the page. Use the backslash continuation character for better readability.

```
yes? LABEL 3,95,-1,0,0.14,\
"@CRFerret is an interactive computer visualization and analysis<NL>\
environment designed to meet the needs of oceanographers and<NL>\
meteorologists analyzing large and complex gridded data sets. It<NL>\
runs on most Unix systems, and on Windows NT/9x using X<NL>\
windows for display. It can be installed to run from a Web<NL>\
browser (WebFerret) for use while away from your desk or<NL>\
from a system lacking X windows software. It can transparently<NL>\
access extensive remote Internet data bases using OPeNDAP,<NL>\
formerly known as DODS."
```

Ch6 Sec4.8. Positioning labels using the mouse pointer

Often it is awkward precisely to position plot labels. Using the mouse pointer can simplify this as mouse clicks can be used to place labels and other annotations on plots. This command option works only in Window 1. It does not function in other windows that have been opened with SET WINDOW/NEW.

The full syntax of the LABEL command is

LABEL xpos, ypos, justify, rotate, height "text"

xpos,ypos are the (x,y) position of the label
justify = -1, 0, 1 for left, center, right justification — default = left
rotate is given in degrees counter-clockwise — default = 0
height is in "inches"
text to be plotted. This argument may include font and color specifications

Note that the LABEL /NOUSER qualifier is not relevant for mouse input.

If either of the first two arguments (label position) are omitted it is a signal that mouse input is desired. For example

```
yes? GO ptest
yes? LABEL "this is a test"
```

will wait for mouse input, using the indicated point as the lower left corner of the text string. Equivalent to this is

```
yes? LABEL ,,-1,0,.12 "this is a test"
```

Note that left justification will always be used in this mode, regardless of the value specified.

For mouse control over justification and/or to draw a line or arrow associating a label with a feature on the plot, explicitly omit the justification argument. Ferret will put up a menu requesting a selection of "Arrow", "Line", "Right", "Center", "Left". If Arrow or Line is selected two mouse inputs are required — the first indicating the feature to be marked, the second indicating the lower left corner of the text area. If "Right", "Center" or "Left" is specified the text will be justified accordingly.

Note that the mouse-driven LABEL command defines the symbols XMOUSE and YMOUSE and writes comments regarding their definitions into the current journal file (if any) as described under the WHERE alias.

The command (alias) WHERE requests mouse input from the user, using the indicated click position to define the symbols XMOUSE and YMOUSE in units of the plotted data. Comments which include the digitized position are also written to the current journal file (if open). The WHERE command can be embedded into scripts to allow interactive positioning of color keys, boxes, lines, and other annotations.

Ch6 Sec4.9. Labeling details with arrows and text

Using the technique described in section 4.7 it is also simple to create a label with a line or arrow indicating a detail of a plot. Follow the procedure outlined above but select "Line" or "Fancy line" (arrow) from the menu that appears in the plot window. Then click on the detail which is to be labeled. The menu will appear again—this time select the justification and click on the label position.

To see the precise numerical coordinates of the arrow and label use the PPL ECHO ON command prior to the PPLUS command which redraws the plot. The endpoint coordinates of the arrow will appear as a comment line which begins with "C LLABS" in the echo file, fort.41. The coordinates of the label will appear as a comment line which begins with "C LABS". (Easily viewed with "spawn tail -2 fort.41".)

Ch6 Sec5. COLOR

Ferret and PPLUS use colors stored by index. Storage indices 0 and 1 are used as window background and foreground colors. Indices 1–6 are reserved for lines. As the user makes SHADE and FILL requests, each color is assigned to the next available storage index beginning at 7, and that assignment is automatically "protected" when viewports or color overlays are added.

If your SHADE and FILL commands request more colors than there are storage indices (256), you will be informed with an error message and the color behavior may become unpredictable. For example, if you have multiple viewports defined within a window you may run out of color

storage indices. If you are using the same color palette(s) in each viewport, you can free up indices by canceling the color protections with PPL SHASET RESET. See the examples later in this section for details on removing color protection. Currently, there is no way to ask PPLUS how many colors it is using in a plot.

The following discussion is divided into a treatment of text and line colors, and a discussion of shade and fill color.

Ch6 Sec5.1. Text and line colors[2]

By default the background color is white and the text color is black. To reverse these, so the background is black, call the script "black.jnl". And to restorethe white background, call "white.jnl". Black and white are the only colors that can be used for the background.

```
yes? go black
yes? ! ...plot commands...
yes? go white
yes? ! ...more plot commands..
```

Line type and color for plot commands are most easliy controlled by the command qualifiers PLOT/COLOR=, PLOT/THICKNESS=, and PLOT/DASH in the Command Reference section (p. 375)

For text, and optionally for plot lines, line type text colors are regulated by use of storage indices 1–6, each index associated with a default color. These are listed in the table in the section "PPLUS text and line color commands" below (p. 200) It is possible to change the six available line colors with the PPLUS enhancements command COLOR. (See Plotplus Plus: Enhancements to Plotplus.) When you create a plot with multiple data lines, Ferret automatically draws each line in a different color. By default, axes, labels, and the first data line are all drawn in the same color. You can modify this behavior with the following Ferret and PPLUS commands.

Ch6 Sec5.1.1. Ferret color controls for lines

Plotted line colors can be set using the /COLOR= qualifier on PLOT, CONTOUR, VECTOR, or POLYGON commands. The available colors are black, red, green, blue, lightblue, purple, and white. In addition, starting with Ferret version 5.4, the user has direct control over dashed lines, and can combine them with choices of colors and thickness.

[2] In the following discussion, "line color/thickness" is used as equivalent to "line style" for the sake of simplicity. However, if you are using a black and white printer, then the metafile translator will substitute a dash pattern for each line color. See Plotplus Plus: Enhancements to Plotplus to see monochrome line styles.

Plotted colors and line type may also be set with the older syntax

```
yes? PLOT/LINE=n
yes? VECTOR/PEN=n
yes? CONTOUR/PEN=n
```

where "n" is an integer between 1 and 18.

More direct control over line color and thickness is available with the qualifiers /COLOR and /THICKNESS and the line type is controlled with /DASH, /SYMBOL=, and /SIZE=

Examples

1) Overlay three lines

```
yes? PLOT/i=1:10  1/i
yes? PLOT/OVER/COLOR=green/i=1:10 1/(i+3)
yes? PLOT/OVER/i=1:10/COLOR=purple/THICK=3 1/i+1/(10-i)
```

2) dashed lines with color and thickness settings

```
yes? PLOT/DASH/I=1:100 sin(i/5)
yes? PLOT/OVER/DASH=(0.3,0.1,0.3,0.1)/COLOR=RED/THICK/I=1:100 sin(i/7)
yes? PLOT/OVER/DASH=(0.6,0.2,0.1,0.2)/COLOR=RED/THICK/I=1:100 sin(i/9)
```

3) Symbols with color and thickness settings

```
yes? PLOT/THICK=2/I=1:100 sin(i/5)
yes? PLOT/OVER/COLOR=red/THICK=3/SYM=4/SIZ=0.10/i=1:100 sin(i/7)
yes? PLOT/OVER/COLOR=green/LINE/SYM=20/SIZ=0.15/i=1:100 sin(i/9)
```

Ch6 Sec5.1.2. PPLUS text and line color commands

Older syntax uses the PPLUS command PEN (p. 540) to assign a color and thickness index to a specified pen. The pen colors are also used to set pen colors for labels (see p. 207). The PPL PEN command takes the form:

```
yes? PLOT/SETUP var
yes? PPL PEN pen_#, color_thickness
yes? PPL PLOT
```

where pen_# is the PPLUS pen number and color_thickness is a color and thickness index. PPLUS uses different pens for different tasks. By default, color_thickness index 1 is assigned to pen 0. The following chart may be helpful.

pen number	default color_thickness index	drawing task
0	1 (black or white)	axes and labels

pen number	default color_thickness index	drawing task
1	1 (black or white)	first data line
2	2 (red)	second data line
3	3 (green)	third data line
4	4 (blue)	fourth data line
5	5 (cyan)	fifth data line
6	6 (magenta)	sixth data line

Note: Whether you plot several data lines simultaneously, or use the /OVERLAY qualifier on your Ferret commands, the color/thickness result will be the same. But the Ferret/PPLUS interaction is different. When Ferret plots multiple data lines simultaneously, PPLUS automatically cycles through pen numbers 1 through 6 combined with symbols. Type ***GO line_samples*** in Ferret to see the 36 different line styles. However, if you are using /OVERLAY for additional data lines, Ferret controls the color_thickness assigned to pen 1 and PPLUS draws each overlay line with pen 1.

Pen numbers range from 0 to 6, and color_thickness indices range from 0 to 18. The values 1 to 18 follow the formula:

color_thickness = 6 * (thickness - 1) + color

where thickness ranges from 1 to 3 and color from 1 to 6. Type "GO line_thickness" in Ferret to see actual colors and thicknesses. Further information is in the appendix, "Ferret Enhancements to PlotPlus, (p. 563)

The special color_thickness index 0 refers to the background color, which produces "invisible" lines that can be used as "white-out" for special purposes. Pen 19 is a thin, white line which can be used to draw in white over a colored area. Thicker white lines are not available.

The following PPLUS commands use the color_thickness index.

Command	Function
@Cnnn	uses color_thickness index "nnn" when embedded in a label (@c019 will draw in white)
PEN	sets color_thickness index for each data line (see chart above) (p. 540)
LEV	sets color_thickness index for contour plot lines (p. 534) (redundant with CONTOUR/LEVELS)

Examples

1) Ferret's default behavior—these two plots will look identical

```
yes? PLOT/i=1:10  1/i, 1/(i+3), 1/i + 1/(10-i)  !3 curves with 3 pens
yes? PLOT/i=1:10 1/i                            !first curve with pen 1
```

```
yes? PLOT/OVER/i=1:10 1/(i+3)          !overlay with pen 1 (next index)
yes? PLOT/OVER/i=1:10 1/i+1/(10-i)     !overlay with pen 1 (next index)
```

2) select different colors for pens 0 and 1

```
yes? PLOT/i=1:10/SET 1/i
yes? PPL PEN 1 4          !assign color_thickness 4 to pen 1 (plot curve)
yes? PPL PEN 0 3          !assign color_thickness 3 to pen 0 (axes &
labels)
yes? PPL PLOT             !render the plot
yes? PPL PEN 0 1          !reset pen 0 to default color_thickness (not\
                           reset by Ferret as is pen 1)
```

3) better way to do above plot:

```
yes? PLOT/i=1:10/LINE=4/SET 1/i  !include line style with qualifer /LINE
yes? PPL PEN 0 3 ; PPL PLOT
yes? PPL PEN 0 1
```

4) To make a white label

```
yes? SHADE/L=1 sst
yes? LABEL/NOUSER 4,4,0,0,0.14 "@C019White Text"1
```

Ch6 Sec5.2. Shade and fill colors

Colors specified with the PPLUS SHASET command or in pallette files (also called spectrum files) contain pre-defined color palettes. With Ferret 5.0 there are now three ways to specify how colors are set in SHADE, FILL, and POLYGON plots: the earlier Percent RGB mapping, and also By_value and By_level.

For examples of these palettes, try the demo script,

```
yes? go palette_demo
```

There is also an FAQ about choosing palettes,

How can I choose a good color palette for my plot? at http://www.ferret.noaa.gov/Ferret/FAQ/graphics/colorpalettes.html

The Percent method defines points along an abstract path in RGB color space that runs from 0 to 100 percent. The pallette file bluescale.spk, for example, contains these lines.

```
  0  0  0 95
100 95 95 95
```

The first number on each line is the percentage distance along the path in color space, and the following numbers are the percents of red, green, and blue, respectively. In this simple two-line file, the percentage runs from 0 to 100 % and the colors represent a range of blues from dark to

light. The percents in the first column must be in ascending order. The actual colors used by SHADE or FILL are determined by dividing this abstract color scale into N equal increments, where N is the number of colors, and linearly interpolating between the red, green, and blue values from the neighboring SHASET percentage points.

For compatibility with older palette files, the Percent RGB mapping method is the default, and pre-5.0 palette files will be interpreted correctly. Palette files using Percent RGB mapping written out with Ferret 5.0 will have a slightly different format. A starting line is optional, specifying "RGB_Mapping_Percent". Any line starting with a ! will be ignored as a comment line. Blank lines are ignored. for example the bluescale palette saved with Ferret 5.0 will look like this:

```
RGB_mapping Percent
! Level        Red        Green         Blue

0              0          0             95
100            95         95            95
```

The first line informs Ferret that the RGB mapping method is Percent. Lines beginning with an exclamation point are comments and ignored when read in—palette files created or modified using a text editor can contain comment lines as documentation. Note that palette files need to be unix-formatted files; values separated by tabs may not be read correctly.

The RGB mapping method By_value uses color interpolation similar to the Percent method, with the significant difference that colors are based on the values of the variable being plotted rather than an abstract zero to 100 percent axis. When you use the same By_value palette in several plots, identical values of one variable will be represented by the same color in each plot. Specify "RGB_Mapping_By_value" as the first line in the palette file. A line starting with a ! will be ignored as a comment line. Blank lines are ignored. The values in column 1 must be in ascending order. For example with the following palette, ocean_temp.spk:

```
RGB_Mapping By_value
!SetPt      Red          Green        Blue

-2.0        80.0         0.0          100.0
0.0         30.0         20.0         100.0
10.0        0.0          60.0         30.0
20.0        100.0        100.0        0.0
30.0        100.0        0.0          0.0
35.0        60.0         0.0          0.0
```

a particular temperature, say 25 degrees, will have the same color on a SHADE or FILL plot with levels ranging from 0 to 30, and on a plot with levels between 20 and 30 degrees.

The third RGB mapping method By_level allows the user to select the precise color to be used at each level in SHADE and FILL plots. Unlike the other methods, no interpolation of RGB values is done. Colors specified in the palette will be used exactly as defined. If there are more SHADE or FILL levels than colors specified, the color palette will repeat. Specify

"RGB_Mapping_By_level" as the first line in the palette file. A line starting with a ! will be ignored as a comment line. Blank lines are ignored. The levels listedin column 1 must be in ascending order.In the following palette, by_level_rainbow.spk,

```
RGB_Mapping By_level

!Level      Red         Green       Blue

1           80.0        0.0         100.0
2           30.0        20.0        100.0
3           0.0         60.0        30.0
4           100.0       100.0       0.0
5           100.0       0.0         0.0
6           60.0        0.0         0.0
```

for example, with 6 colors defined and used in a plot with 10 levels, the colors used at each plot level will be as follows:

Plot level	Color
1	1
2	2
3	3
4	4
5	5
6	6
7	1
8	2
9	3
10	4

Ch6 Sec5.2.1. Ferret shade and fill color controls

By default, Ferret will use the PPLUS spectrum file default.spk for shades and fills (normally default.spk is a Unix soft link to rnb.spk). Ferret comes with many color palettes. The UNIX command "Fenv" lists the environment variable $FER_PALETTE which is a list of paths to be searched for palette files (the palette file names all end in .spk). The UNIX command "Fpalette" allows you to find and examine these files (type "Fpalette -help" at the Unix prompt). You can easily create your own palette files with a text editor.

Use the Ferret qualifier /PALETTE= with Ferret graphical output commands CONTOUR/FILL and SHADE to specify a color palette. See the section in this chapter, "Contouring," p. 213, for details on the CONTOUR qualifier /LEV, which controls colors and dash patterns, as well as sets contour levels.

Ferret qualifiers

/PALETTE= (alias for PPL SHASET SPECTRUM=)
/LEV=

PALETTE is also a stand-alone command alias; it sets a new default color palette.

Be aware that when you use /PALETTE= in conjunction with /SET_UP, the color spectrum you specify becomes the new default palette; to restore the default palette use command PALETTE with no argument.

Ch6 Sec5.2.2. PPLUS shade color commands

Command	Function
SHASET	Sets colors used by SHADE (p. 562)
SHAKEY	Customizes the shade key (p. 561)

SHASET is an enhancement of PPLUS designed for Ferret. You can specify a color spectrum, save a spectrum, change an individual color in the spectrum, or remove the protection (PPL SHASET RESET) for colors already on the screen. See Plotplus Plus: Enhancements to Plotplus (p. 562) for more information.

If you need precise control over each individual RGB color on your plot, run "GO exact_colors", which contains instructions on modifying individual colors in a palette using SHASET.

The SHAKEY command (see p. 561) allows you to customize the location, size and labelling of the color key for SHADE and FILL plots.

Examples

1) look at the relief of the earth's surface

```
yes? SET DATA etopo120
yes? SHADE rose                          !Ferret's default plot
yes?   ! Emphasize land and sea with palette,customize the color key
yes? SHADE/PALETTE=land_sea/SET_UP rose palette
yes? PPL SHAKEY 1,0,0.1,2, , ,1.2,7.2,7.5,8.2
yes? PPL SHADE
```

2) Perhaps you would like to compare two topography resolutions. To illustrate what happens when you use more colors than are available, request an excessively large number of levels:

```
yes? SET DATA etopo120
yes? SET REGION/Y=-20:20
yes? SET VIEWPORT UPPER                    !upper half
yes? SHADE/LEV=(-8000,8000,100) rose       !160 colors, default palette
```

```
yes? SET VIEWPORT LOWER                    !lower half
yes? SET DATA etopo20                      !high resolution
yes? SHADE/LEV rose[d=etopo20]             !another 160 colors (320 >
256!)
yes? CANCEL VIEWPORT

PPL+ error: You're attempting to use more colors than are available.
            Using SHASET RESET to re-use protected colors may help.
```

If you reuse the same palette, as in this example, issue PPL SHASET RESET after the first plot. Now the second picture is made without error:

```
yes? SET DATA etopo120
yes? SET REGION/Y=-20:20
yes? SET VIEWPORT UPPER
yes? SHADE/LEV=(-8000,8000,100) rose
yes? SET VIEWPORT LOWER
yes? PPL SHASET RESET                      !reuse color storage indices
yes? SET DATA etopo20
yes? SHADE/LEV rose[d=etopo20]
yes? CANCEL VIEWPORT
```

Ch6 Sec6. FONTS

Ch6 Sec6.1. Ferret font and text color

By default, Ferret produces all plot labels using the fonts ASCII Simplex (code AS) and ASCII Complex (code AC). For upper and lower case letters these fonts are identical to the fonts Simplex Roman (SR) and Complex Roman (CR), respectively. In addition, however, fonts AS and AC include the complete set of ASCII punctuation characters and ignore the special PPLUS interpretations of the characters "^" (superscript), "_" (subscript), and "@" (change font or pen). Using a text editor, the ESCAPE character (decimal 27) may be inserted before the special characters to restore their special interpretation.

The Ferret command CANCEL MODE ASCII causes Ferret to generate PPLUS labels which have the font unspecified. When the font is unspecified the PPLUS command DFLTFNT determines the default font and PPLUS responds to the special characters "^", "_", and "@". SET MODE ASCII restores normal font behavior.

Ch6 Sec6.2. PPLUS font and text color commands

PPLUS commands can be used to customize the font settings. See the examples below, and the section on PPLUS graphical commands (p. 185) for more on the syntax to make PPLUS calls.

Command	Function
DFLTFNT	Sets default character font for all labeling.
@AB	In a label string, selects the font for which AB is a two-letter abbreviation (i.e., @CI for complex italic—see PPLUS manual for fonts, p. 553).
@Pn	Changes to pen color n (see p. 200for corresponding colors)
@Cnnn	Changes to pen color nnn when the pen number is greater than 9. You must use exactly 3 digits.

Note that many ASCII punctuation characters are printable only in ASCII simplex and complex fonts. In all other fonts these characters "@", "^", and "_" have special meanings: @ = font change; ^ = superscript; _ = subscript.

Examples

1) axis labels in custom fonts (Figure 6_7)

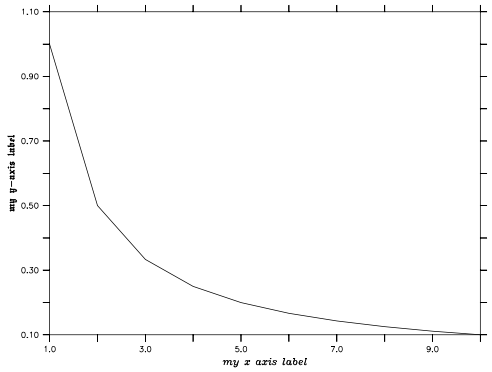


Figure 6_7

```
yes? PLOT/SET/i=1:10/NOLAB 1/i
yes? PPL XLAB @CImy x-axis label
yes? PPL YLAB @GEmy y-axis label
yes? PPL PLOT
```

2) set default font for all labeling (Figure 6_8)

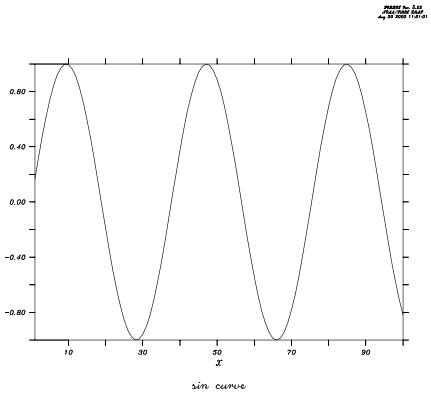


Figure 6_8

```
yes? CANCEL MODE ASCII
yes? PPL DFLTFNT CS      !complex script
yes? PLOT/I=1:100/TITLE="sin curve"  sin(i/6)
yes? SET MODE ASCII
yes? PPL DFLTFNT SR      !numeric axis labels unaffected by SET MODE
ASCII
```

Ch6 Sec7. PLOT LAYOUT

Ch6 Sec7.1. Ferret layout controls

Layout of plots can be controlled with commands which modify window size and aspect ratio, and viewports.

Ferret command

SET WINDOW/SIZE=/NEW/ASPECT=
DEFINE VIEWPORT/XLIMITS=/YLIMITS=/TEXT= view_name
SET VIEWPORT view_name
CANCEL VIEWPORT

Ch6 Sec7.1.1. Viewports

A viewport is a sub-rectangle of a full window. Viewports can be used to put multiple plots onto a single window. Issuing the command SET VIEWPORT is best thought of as entering "viewport mode." While in viewport mode all previously drawn viewports remain on the screen until explicitly cleared with either SET WINDOW/CLEAR or CANCEL VIEWPORT. If multiple plots are drawn in a single viewport without the use of /OVERLAY the current plot will erase and replace the previous one; the graphics in other viewports will be affected only if

the viewports overlap. If viewports overlap the most recently drawn graphics will always lie on top, possibly obscuring what is underneath. By default, the state of "viewport mode" is canceled. A number of the most commonly desired viewports are pre-defined.

Ch6 Sec7.1.2. Pre-defined viewports

Name	Description
FULL	full window
LL	lower left quadrant of window
LR	lower right quadrant of window
UR	upper right quadrant of window
UL	upper left quadrant of window
RIGHT	right half of window
LEFT	left half of window
UPPER	upper half of window
LOWER	lower half of window

Example: Graphics Viewports

Plot four variables from coads_climatology into the four quadrants of a single window (Figure 6_9).

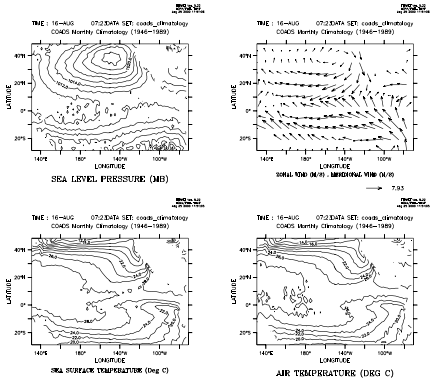


Figure 6_9

```
yes? SET DATA coads_climatology
yes? SET REGION/@W/L=8
yes? SET VIEWPORT LL
yes? CONTOUR sst                          !sea surface temperature
yes? SET VIEWPORT LR
yes? CONTOUR airt                         !air temperature
yes? SET VIEWPORT UL
yes? CONTOUR slp                          !sea level pressure
yes? SET VIEWPORT UR
yes? VECTOR/XSKIP=4/YSKIP=4 uwnd,vwnd     !zonal wind, meridional wind
yes? CANCEL VIEWPORT
```

Ch6 Sec7.1.3. Advanced usage of viewports

For the purposes of defining viewports, a graphics window is considered to have length 1 and height 1. All viewport commands refer to positions relative to the current aspect ratio of the window. Thus,

```
yes? DEFINE VIEWPORT/XLIM=.5,1/YLIM=.5,1  V5
```

will locate the origin of viewport V5 in the upper right of the output window regardless of the shape of the window.

```
yes? DEFINE VIEWPORT/XLIM=0.,1/YLIM= 0,.3 V1
yes? DEFINE VIEWPORT/XLIM=0.,1/YLIM=.3,.6 V2
yes? DEFINE VIEWPORT/XLIM=0.,1/YLIM=.6,.9 V3
```

defines three viewports; each takes a third of the height of the page, and the entire width.

The qualifiers /XLIMITS=x1,x2 and /YLIMITS=y1,y2 allow the user to specify a portion of the graphics window to be the defined viewport. The arguments must be values between [0,1] (NOT world coordinates). x1 and x2 indicate the lower and upper bounds for the length of the window to be defined as the viewport; y1 and y2 serve an analogous purpose for height.

The /TEXT=n qualifier allows the user control over the shrinkage or enlargement of text on the plot. A value of /TEXT=1 indicates that the text size should be the same as it is on the full screen output. If a value less than 1 is specified the text will shrink. If a value is not specified Ferret chooses a value appropriate to the viewport size. Acceptable values are $0 < n < \text{inf}$. but only values up to about 2 yield useful results.

Ch6 Sec7.1.4. Viewport Symbols

When we "set viewport viewport_name" a number of Ferret symbols are set, giving access to the viewport size, scaling, the values given to XLIM and YLIM when defining the viewport, and the margins. See the Ferret Special Symbols section (p. 221) for a list of these symbols.

Ch6 Sec7.2. PPLUS layout commands

The following PPLUS commands can be called to customize the plot layout. See the section on PPLUS graphical commands for how to call PPLUS plot commands (p. 185)

Command	Function
ORIGIN	sets distance of plot origin from lower left corner (p. 539)
BOX	controls drawing of a box around the plotting area (p. 526)
CROSS	controls drawing of lines through (0, 0) on graph (p. 528)

Command	Function
ROTATE	rotates plot by 90 degrees on screen and plotter (p. 543)
AXLEN	sets axis lengths (p. 525)
SHAKEY	locates the color key (p. 561)
VECKEY	locates the vector key (p. 549) (see also the VECTOR/NOKEY qualifier,p. 449)
AXSET	includes/excludes particular axes (p. 526) (see also PLOT/AXES=, CONTOUR/axes=, etc., p. 378)
SIZE	sets the overall size of the graphics window (p. 544)

Example:

A small plot, rotated 90 degrees, positioned with its origin at (4,4) on the plot page. Use the /AXES qualifier to plot just the left and bottom axes.

```
yes? PPL BOX ON
yes? PPL ORIGIN 4,4
yes? PPL CROSS ON
yes? PPL ROTATE ON
yes? PPL AXLEN 2,2
yes? PLOT/I=1:30/AXES=0,1,1,0 sin(i)
```

Ch6 Sec7.3. Controlling the white space around plots

The location and size of the axis rectangle within the viewport or window determines the amount of white space surrounding a plot. Complete control over this is possible using low level controls, DEFINE VIEWPORT/TEXT_PROMINENCE, PPL ORIGIN, and PPL AXLEN, but these commands are sometimes awkward to work with. A simpler strategy is to use the GO tool

```
yes? GO margins
```

When given without arguments this command will report the amount of white space surrounding a plot. With arguments it will adjust the axis origins and lengths according to the requested margins. Try the Unix command

```
> Fgo -more margins
```

for further documentation.

Ch6 Sec8. CONTOURING

Ch6 Sec8.1. Ferret contour controls

The following qualifiers to the Ferret command CONTOUR allow customization of a contour plot.

Qualfier	Function
/FILL	produces a color-filled contour plot (command FILL is an alias for CONTOUR/FILL)
/LEVELS	specifies contour levels, dash patterns, line thickness and color
/KEY	turns on display of color key for color-filled contour plots (default)
/NOKEY	turns off display of color key for color-filled plots
/NOAXIS	turns off display of X and Y axes (useful for map projections)
/LINE	adds contour lines to a color-filled plot (lines replace key)
/PALETTE=	specifies a color palette for color-filled contour plot
/PEN=	sets line style for contour lines (same arguments as PLOT/LINE=. See the section in this chapter, "Text and Line Colors," p. 200.)
/SIGDIG=	sets the number of significant digits for contour labels; default is 2
/SIZE=	sets the size of characters in the contour labels; default is 0.08
/SPACING=	sets spacing for contour lines, using PLOT+ definition of "inches"; default spacing is 5.0

Ch6 Sec8.1.1. /LEVELS qualifier

The /LEVELS qualifier for CONTOUR, SHADE and FILL commands is a powerful and multi-functional tool. It takes the form ***/LEVELS=levels_descriptor***

/LEVELS without an argument /LEVELS instructs Ferret to reuse CONTOUR or SHADE levels from the last CONTOUR or SHADE plot

/LEVELS=n specifying a simple numerical argument such as /LEVELS=25 instructs Ferret to select approximately 25 levels automatically, based upon the limits of the data to be plotted

/LEVELS=nC (centered levels) appending a "C" to the suggested number of levels instructs Ferret to select levels which are centered about the zero level. Such levels are suitable for zero-symmetric quantities such as anomalies and velocity components.

/LEVELS=x.xD (delta levels) Use of "D" as a suffix instructs Ferret to use the preceding value as the delta value between contour levels. Thus /LEVELS=0.25D will cause Ferret to select contour levels that span the range of the data to be contoured with a delta value of 0.25 be-

tween contour levels. The "D" and "C" notations can be combined. For example, /LEVELS=0.25DC instructs Ferret to create zero-centered levels with a delta of 0.25 spanning the range of the data.

/LEVELS=(lo, hi, delta)
or
/LEVELS=(value)
or
/LEVELS=(lo, hi, delta, ndigits)
or
/LEVELS=(-inf)(lo, hi, delta)(inf)

ndigits (applies to CONTOUR command only) is the number of decimal places to use when labeling the level on individual contour lines as:
-1 for integer format
or
-3 to omit numerical labels

(-inf) and (inf) are available starting with Ferret v5.81. When (-inf) or (inf) is specified for FILL or SHADE plots, the variable is filled in with a color on the end of the spectrum for all values below (-inf) or above (inf) the lo and hi values given in the levels specification. These specifiers have no effect on a CONTOUR command.

When a CONTOUR or SHADE plot is finished, the levels that were used are stored in a set of symbols so the settings can be used again or modified for subsequent plots. These symbols are

LEV_MIN	minimum level used; if (-inf) given, then the value of LEV_MIN is the minimum in the data field
LEV_MAX	maximum level used; if (inf) given, then the value of LEV_MIN is the maximum in the data field
LEV_NUM	number of levels used
LEV_DEL	Delta between values, if the levels were uniform, or "irregular"
LEV_TXT	The argument to the levels qualifier

Examples

Note that by default the contour lines of negative values will be dashed and the zero contour will be a heavy (DARK) line. See also (p.) for selecting color and thickness with the PEN option, below.

```
/LEVELS=(-20,10,2)     ! basic low,high,delta
/LEVELS=(5)            ! a single level at 5
/LEVELS=(-20,10,2,-3)  ! suppress numerical contour labels
```

```
/LEVELS=40              ! approximately 40 automatically selected
                        levels
/LEVELS=40C             ! approximately 40 automatic levels cen-
                        tered equally about zero
/LEVELS=0.2D            ! automatic levels with a delta value of 0.2
/LEVELS=0.2DC           ! automatic zero-centered levels with a
                        delta value of 0.2
```

Refinements to the basic levels may be applied using the syntaxes below. If blanks are included, surround the entire levels descriptor in double quotation marks.

To request additional levels, simply append additional (lo, hi, delta) and/or (value) specifiers.

```
 /LEVELS="(-100)(-10,10,2)(100)"        ! focus on -10 to10 range,
                                        but catch outliers
```

To specify the line type as dark (heavy line), append DARK(lo, hi, delta) or DARK(value). Similar syntax can be applied to LINE (solid, thin) or DASH. The range of levels must first be specified before customizing them with DARK or DASH.

```
/LEVELS="(-100,100,5)DARK(-100,100,  ! heavy line on  multiples of
25)"                                 25
/LEVELS="(0,10,2) DASH(2,10,2)"      ! use dashed lines for posi-
                                     tive values
```

To remove selected levels, append the specifier DEL(lo, hi, delta) or DEL(value).

```
 /LEVELS="(-10,10,2) DEL(0)"     ! -10 to 10 by 2's with the zero
                                 contour removed
```

To specify the color_thickness index of contour lines (see the section in this chapter, "Color, " p. 199, for a discussion of color_thickness indices), append PEN(lo, hi, delta, index).

```
/LEVELS=(0,1,.2) PEN(.6,1,.2,2)   ! use pen #2 (red) for the upper
                                  contour levels
/LEVELS="(-100,100,10)            !  Use Pen 2 (red) for negative
PEN(-100,-10,10,2)                levels and  pen 4 (blue) for pos-
PEN(10,100,10,4)"                 itive level
```
s.

To apply the previous levels to a new plot, use the /LEVELS qualifier alone. To do more, the levels symbols let you apply the settings in new ways:

```
yes? USE coads_climatology
yes? CONTOUR/L=1 sst
yes? SHOW SYM LEV*
LEV_MIN = "-5"
LEV_MAX = "35"
LEV_NUM = "9"
LEV_DEL = "5"
```

```
yes? SHADE/L=5/LEV=(($LEV_MIN), ($LEV_MAX), 2) airt
yes? SHOW SYM LEV*
LEV_TEXT = "(-5, 35, 5)"
LEV_MIN = "-5"
LEV_MAX = "35"
LEV_NUM = "21"
LEV_DEL = "2"
```

Ch6 Sec8.1.2. /PEN, /SIZE, /SIGDIG, /SPACING qualifiers

For contouring, the qualifiers /PEN, /SIZE, /SIGDIG, and /SPACING allow simple control over the contours: their color, the size and significant digits in the labels, and the spacing between contours.

CONTOUR/PEN=
Sets line style for contour lines (same arguments as PLOT/LINE=). Argument can be an integer between 1 and 18; run ***GO line_samples*** to see the styles for color devices.

Example:

```
yes? CONTOUR/PEN=2  sst
```

CONTOUR/SIZE, CONTOUR/SIGDIG, and CONTOUR/SPACING are alternative implementations of the PPL CONSET command. /SIZE is equivalent to hgt, /SIGDIG is nsig, and /SPACINGis dslab. They make the same settings, but in a more convenient way. For details see the CONSET command, (p. 558) The "inches" used in these commands is the PPLUS definition of inches. These are the same units as in, say, "ppl axlen 8,6", to specify plot axes of lengths 8 and 6 inches for horizontal and vertical axes, respectively. They are scaled within the plot window.

Note that to remove the labels from the contour lines, use the fourth argument on the /LEVELS qualifier: CONTOUR/LEVELS=(lo, hi, delta, ndigits), where ndigits is the number of decimal places to use when labeling the level on individual contour lines and we can set ndigit= -3 to omit numerical labels along the contour lines.

CONTOUR/SIZE=
Controls the size of characters in the contour labels; default is 0.08". See the example under CONTOUR/SPACING below.

CONTOUR/SIGDIG=
Sets the number of significant digits for contour labels; cefault is 2. See example under CONTOUR/SPACING below.

CONTOUR/SPACING=
Sets spacing for contour lines;default spacing is 5".

Example of CONTOUR/SIZE/SIGDIG/SPACING

```
yes? LET my_field = SIN(X[x=1:6:.1])*COS(Y[y=1:6:0.1])
yes? CONTOUR/SIGDIG=1/SIZE=0.15/SPACING=3 my_field
```

Specifies contour labels with a single significant digit using characters of height 0.15 "inches" at a nominal spacing of 3 "inches", consistent with the PLOT+ usage of "inches".

Ch6 Sec8.2. PPLUS contour commands

PPLUS commands can be used to customize contouring settings. Note that Ferret makes settings for all of these automatically; you will only need to make PPLUS calls to change the properties of the plot. See the examples below, and the section on PPLUS graphical commands (p. 185) for more on the syntax to make PPLUS calls.

Command	Function
CONPRE	sets prefix for contour labels (usually a font, e.g., "@TR") (p. 526)
CONPST	sets suffix for contour labels (usually units, e.g., "cm") (p. 527)
CONSET	controls various aspects of contour labels and curves (see below)

CONSET is a modified version of the PPLUS command. Two new parameters have been added—"spline_tension" and "draftsman". "spline_tension" controls a spline fitting routine for contour lines, and is primarily used in conjunction with the narc parameter. The new parameter "draftsman" enables the user to specify horizontally oriented contour labels (draftsman style) or the default, labels oriented along contour lines.

Examples:

```
! Contour a variable, enlarging the size of the contour label:
yes? CONTOUR/SET  var
yes? PPL CONSET 0.15
yes? PPL CONTOUR
```

Arguments for CONSET are as follows:

```
CONSET
hgt,nsig,narc,dashln,spacln,cay,nrng,dslab,spline_tension,draftsman
```

hgt = height of contour labels. default=.08 inches

nsig = no. of significant digits in contour labels. default=2

narc = number of line segments to use to connect contour points. default=1

dashln = dash length of dashes mode. default=.04 inches

spacln = space length of dashes mode. default=.04 inches

cay This argument has no effect on gridded data. It is documented in PLOT PLUS for Ferret User's Guide (p.527).

nrng This argument has no effect on gridded data. It is documented in PLOT PLUS for Ferret User's Guide (p. 527).

dslab= nominal distance between labels on a contour line. default=5.0 inches.

spline_tension = a real value that affects the fit of the contour line. default=0. This parameter is only applied if narc is greater than 1. Otherwise, straight lines are drawn between data points and no interpolated points are contoured. This value indicates the curviness desired.

abs(spline_tension) is nearly zero (e.g., .01). The resulting curve is approximately a cubic spline.
abs(spline_tension) is large (e.g., 10.). The resulting curve is nearly a polygonal line.
spline_tension = 0. The resulting curve is a cubic spline (the default algorithm in ppl).

A typical value for spline_tension is 1, and the typical useful range of values is .01 to 10.

draftsman = a real value that controls the label format. default = 0.
0. = original label style—labels oriented along contour arcs
> 0. = draftsman label style—labels oriented horizontally on the page
< 0. = reserved for future use

Examples

Run the demonstration on custom contouring for many examples of label styles, contour line styles (color, thickness dash pattern), and contour intervals— ***yes? GO custom_contour_demo***

1) Color-filled contour plot of sea surface temperature

```
yes? SET DATA coads_climatology
yes? SET REGION/@t/l=6          !specify tropical Pacific, month 6
yes? SET VIEWPORT upper
yes? FILL sst                   !filled contour plot
yes? SET VIEWPORT lower
yes? FILL/LINE sst              !make the plot with contour lines
```

2) Let's improve on the earlier example (5.2.2) of shaded bathymetry with blue palette

```
yes? SET DATA ETOPO60
yes? LET/TITLE="Surface relief x1000 (meters)" r1000 rose/1000
yes? FILL/PAL=ocean_blue/LINE/LEV=(-8,-1,1,-3)LINE(-8,-1,1,-3)/PEN=4
r1000
```

Here is a breakdown of the final command line:

FILL	color-filled contour plot (alias for CONTOUR/FILL)
PAL	specifies color palette for fill colors
LINE	specifies that contour lines be overlaid on the filled plot (in lieu of a key)
LEV	first arg specifies contour levels without numerical labels, next requests solid lines (dashed lines are the default for negative contour values)
PEN	assigns line style 4 (blue) to contour lines

Ch6 Sec9. SPECIAL SYMBOLS

When a plot command is executed, PPLUS automatically defines a number of global symbols which are available to the user with SHOW SYMBOL They are documented in the PPLUS Users Guide (p. 503), and listed here. These are not defined until associated plot commands have been issued. Also note that the user cannot redefine the value of these symbols.

Example: draw a plot and examine and use some of the symbols

```
yes? plot/i=1:10 1./i
yes? SHOW SYMBOL ppl$xlen
PPL$XLEN = "8.000"

    ! Try to show an undefined variable (no response)
yes? SHOW SYMBOL ppl$lf_var

yes? SHOW SYMBOL ppl$line_count
PPL$LINE_COUNT = " 1"

    ! Use the value of a symbol to position a label
yes? LET my_xlen = ($ppl$xlen) - 1.
DEFINE VARIABLE my_xlen = 8.00 - 1.

yes? LABEL/NOUSER `my_xlen`, 0.1, -1, 0, 0.1 "label at `my_xlen`"
```

SYMBOL	PPL COMMAND	DESCRIPTION
PPL$EOF	RD,RWD,SKP	"YES" if an EOF (end of file) was read.
PPL$FORMAT	FORMAT	The current format.
PPL$HEIGHT	SIZE	Height of the box.
PPL$INPUT_FILE	RD,SKP,RWD	The current input file.
PPL$LF_A	LINFIT	Constant from fit y= a + b*x
PPL$LF_A_STDEV	LINFIT	Standard error of A.

PPL$LF_B	LINFIT	Constant from fit.
PPL$LF_B_STDEV	LINFIT	Standard error of B.
PPL$LF_R2	LINFIT	Regression coefficient squared.
PPL$LF_RES_VAR	LINFIT	Residual variance.
PPL$LF_VAR	LINFIT	Total variance.
PPL$LINE_COUNT	-	The number of the last line read.
PPL$PLTNME	PLTNME	The name of the plot file.
PPL$RANGE_INC	%RANGE	See Advanced Commands Chapter
PPL$RANGE_HIGH	%RANGE	See Advanced Commands Chapter
PPL$RANGE_LOW	%RANGE	See Advanced Commands Chapter
PPL$TEKNME	TEKNME	The name of the tektronix file.
PPL$VIEW_X	VPOINT	X viewpoint (from a WIRE plot)
PPL$VIEW_Y	VPOINT	Y viewpoint (from a WIRE plot)t
PPL$VIEW_Z	VPOINT	Z viewpoint (from a WIRE plot)
PPL$WIDTH	SIZE	Width of the box.
PPL$XFACT(n)	TRANSXY	Xfact for line n.
PPL$XLEN	AXLEN	Length of X axis.
PPL$XOFF(n)	TRANSXY	Xoff for line n.
PPL$XORG	ORIGIN	Distance between origin and left edge.
PPL$XFIRST(n)	-	X value for first data point in line n.
PPL$XLAST(n)	-	X value for last data point in line n.
PPL$XMAX	RD	Xmax of contour grid
PPL$XMIN	RD	Xmin of contour grid
PPL$XMAX(n)	-	Xmax for valid data in line n.
PPL$XMIN(n)	-	Xmin for valid data in line n.
PPL$YFACT(n)	TRANSXY	Yfact for line n.
PPL$YLEN	AXLEN	Length of Y axis.
PPL$YOFF(n)	TRANSXY	Yoff for line n.
PPL$YORG	ORIGIN	Distance between origin and bottom edge.
PPL$YFIRST(n)	-	Y value for first data point in ine n.
PPL$YLAST(n)	-	Y value for last data point in line n.

PPL$YMAX	RD	Ymax of contour grid
PPL$YMIN	RD	Ymin of contour grid
PPL$YMAX(n)	-	Ymax for valid data in line n.
PPL$YMIN(n)	-	Ymin for valid data in line n.
PPL$ZMAX	-	Zmax for valid contour data.
PPL$ZMIN	-	Zmin for valid contour data.

In addition to the PPLUS symbols, Ferret sets other symbols on startup or when plotting commands are issued. They are summarized here:

SYMBOL	FERRET COMMAND	DESCRIPTION
FERRET_VERSION	Ferret startup	the Ferret version
FERRET_PLATFORM	Ferret startup	the platform Ferret is running o
BYTEORDER	Ferret startup	gives "BIG" or "LITTLE" according to the endianness of the CPU
SESSION_DATE	Ferret startup	date the current Ferret session started
SESSION_TIME	Ferret startup	time the current Ferret session started
LABTIT	plot commands	default title label; if title is set with /TITLE or PPL TITLE, this new title is stored in a moveable lable
LABX	plot commands	label for horizontal axis
LABY	plot commands	label for vertical axis
LABn	plot commands	nth moveable label
XAXIS_MIN	plot commands	data value corresponding to the start of the horizontal axis
XAXIS_MAX	plot commands	data value corresponding to the end of the horizontal axis
YAXIS_MIN	plot commands	data value corresponding to the start of the vertical axis
YAXIS_MAX	plot commands	data value corresponding to the end of the vertical axis
LABKEY	line PLOT, POLYGON	text for key of latest line or polygon put on the plot
VP_HEIGHT	SET VIEWPORT	height of current viewport
VP_WIDTH	SET VIEWPORT	width of current viewport

VP_XLO	SET VIEWPORT	lower x corner of viewport, as defined by DEFINE VIEWPORT/XLIM=xlo:xhi
VP_XHI	SET VIEWPORT	upper x corner of viewport, as defined by DEFINE VIEWPORT/XLIM=xlo:xhi
VP_YLO	SET VIEWPORT	lower y corner of viewport, as defined by DEFINE VIEWPORT/YLIM=ylo:yhi
VP_YHI	SET VIEWPORT	upper y corner of viewport, as defined by DEFINE VIEWPORT/YLIM=ylo:yhi
VP_RT_MARGIN	SET VIEWPORT	width of right margin(see ppl$xorg for left margin)
VP_TOP_MARGIN	SET VIEWPORT	width of top margin(see ppl$yorg for lower margin)
VP_SCALE	SET VIEWPORT	shrinking or expansion factor (see DEFINE VIEWPORT/TEXT, p. 352)
LEV_TEXT	CONTOUR, SHADE	command argument used to set levels.
LEV_MIN	CONTOUR, SHADE	minimum contour level used
LEV_MAX	CONTOUR, SHADE	maximum contour level used
LEV_NUM	CONTOUR, SHADE	number of levels used
LEV_DEL	CONTOUR, SHADE	delta-value between levels

Ch6 Sec10. MAP PROJECTIONS AND CURVILINEAR COORDINATES

Ch6 Sec10.1. Three-argument (curvilinear) version of SHADE, FILL, CONTOUR, and VECTOR

The SHADE, FILL, CONTOUR and VECTOR commands now have a 3-argument mode which allows them to create output in "curvilinear" coordinates; i.e. those on curvilienar grids (p. 252) or if a map projection has been defined (p. 224).This allows for easy generation of output plots using sigma coordinates as well as the application of various map projections. A typical command line entry will look like:

```
yes? SHADE sst, x_page, y_page
yes? VECTOR/OVER/PEN=1 uwnd, vwnd, x_page, y_page
```

where the last two arguments, ***x_page(i,j)*** and ***y_page(i,j)***, must be (at least) 2-dimensional grids which specify the X page (horizontal) position and Y page (vertical) position for

each ***(i,j)*** index pair. The page positions may be in any units; Ferret will scale the plot according to the ranges of values in the position fields.

Notes:

1. The default axis labeling for the 3-argument commands will be the ranges of the position fields: inappropriate when map projections are being used. The /NOAXIS qualifier is provided for this purpose.

The /NOAXIS qualifier causes the axes and axis labels to be omitted from the plot. The qualifier has been added to support the curvilinear coordinate and map projection capabilities of the 3-argument versions of SHADE, FILL, CONTOUR and VECTOR in which linear axes are inappropriate.

2. In the 3-argument SHADE syntax you can specify either the coordinates of the points or the coordinates of the cell boundaries. In the command

```
yes? SHADE values, xcoords, ycoords
```

say that nVx is the size of the "x" dimension of the values argument, and nCx is the size of the "x" dimension of the coordinate arguments.

If nCx = nVx then the xcoords argument is presumed to give the locations of the points in the values argument and (as you say), the boundaries between points are computed to be the midpoints.

However, if nCx = nVx + 1 then the xcoords and ycoords arguments are presumed to give the locations of the boundaries. For an example see the FAQ on Using the 3-argument SHADE command. In all cases the size of the xcoords argument must match the size of the ycoords argument.

3. Beginning with Ferret v5.81 the argument /MODULO for SHADE, CONTOUR, or FILL plots will draw modulo replications in longitude for curvilinear data in order to fill out the specified extent in the longitude direction. For instance, if the xcoords variable contains longitudes in the range -270:90 we can draw a plot with longitudes 0:360 with the command

```
yes? SHADE/HLIMITS=0:360/MODULO values, xcoords, ycoords
```

4. There is an alternative to the 4-argument VECTOR command. The script mp_poly_vectors sets up to plot vectors in curvilinear coordinates using filled polygons. See the script poly_vec_demo.jnl for a demonstration of this capability.

Ch6 Sec10.2. Gridded data sets on curvilinear coordinates

If a given gridded variable is defined on a curvilinear coordinate system (see p. 252), then one need only provide the X and Y coordinate fields in the 3-argument SHADE or FILL command to accurately depict the field. For example, if a data set contained a variable TEMP, which was Nx x Ny in the longitude-latitude plane, and the data set also contained variables LON_POSITION and LAT_POSITION of the same size, then the command:

```
yes? SHADE TEMP, LON_POSITION, LAT_POSITION
```

would render the curvilinear plot.

Ch6 Sec10.3. Layered (sigma) coordinates

The capability to render curvilinear coordinates allows Ferret to display sigma coordinate fields without interpolating or regridding the variable to be displayed.

In this example the variable flow is defined on the gg grid where the Z axis is in layers. To display the field we need only create multidimensional fields specifying the relative positions of (i,j) pairs and use the new curvilinear coordinate commands (Figure 6_10):

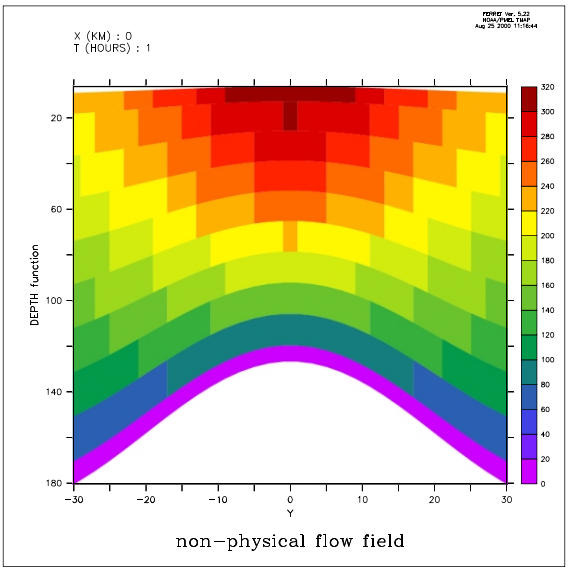


Figure 6_10

```
yes? LET depth = h[k=@rsum]-h/2
yes? SET VARIABLE/TITLE="DEPTH function"/UNIT=meters depth
yes? ! regrid 'Y' to the data grid
yes? LET ygg = y[g=gg]
yes? SET VARIABLE/TITLE="Y"/UNIT=kilometers ygg
yes? SHADE flow[x=0,l=1], ygg, depth[x=0,i=1]
```

For a detailed example illustrating the use of curvilinear coordinates to analyze sigma-coordinate fields see the Ferret FAQ Entry, How to handle sigma coordinate output in Ferret.

Ch6 Sec10.4. Map Projections

Along with general capabilities for curvilinear coordinates, version 4.9 of Ferret and later provide a series of scripts for many common map projections.

Each map projection script will create the following variables:

mp_central_meridian	central longitude calculated from the currently set region
mp_standard_parallel	central latitude calculated from the currently set region
x_page	two dimensional array mapping X world coordinates to page coordinates
y_page	two dimensional array mapping Y world coordinates to page coordinates
mp_mask	mask two hide "back side" data in orthographic or other 3-D projections

Ch6 Sec10.4.1. Using Map Projection scripts

To create output with a particular map projection you must do the following:

1. run the map projection script
2. associate the variable's grid with the projection: set grid var
3. adjust the window aspect ratio (if desired)
4. multiply the variable of interest by mp_mask (required for "3-D" projections)
5. give the three-argument plotting command

Example: (Figure 6_11)

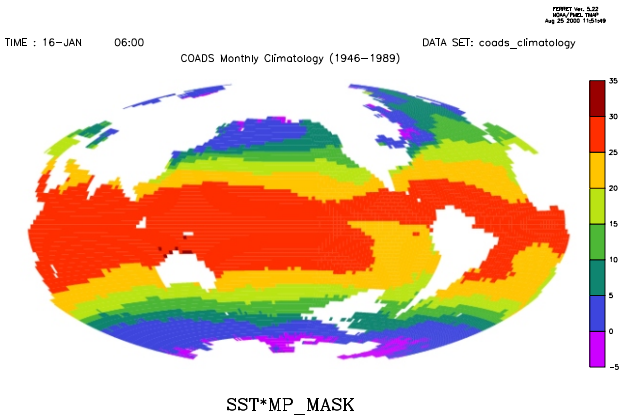


Figure 6_11

```
yes? USE coads_climatology
yes? SET REGION/L=1
yes? GO mp_hammer
yes? GO mp_grid sst
yes? GO mp_aspect
yes? SHADE/NOAXIS sst*mp_mask, x_page, y_page
```

Ch6 Sec10.4.2. Overlays with Map Projections

Overlays can be drawn once a map projection script has been run. To add a filled land mask, sea level pressure and wind vectors onto our SST map we would issue the following commands (Figure 6_12):

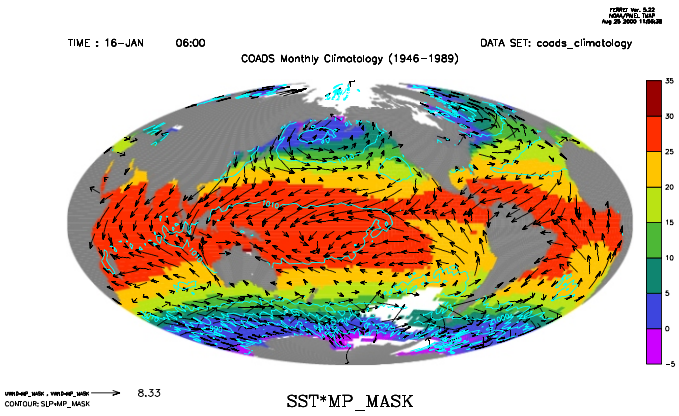
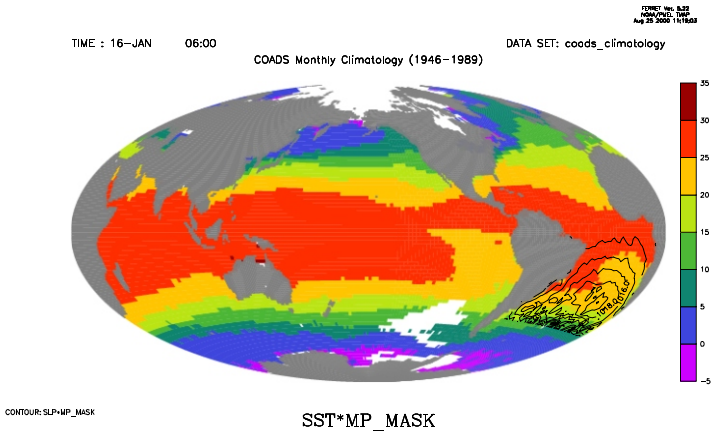


Figure 6_12

```
...
yes? GO mp_grid uwnd
yes? GO mp_fland
yes? VECTOR/OVER/PEN=1 uwnd*mp_mask, vwnd*mp_mask, x_page, y_page
yes? GO mp_grid slp
yes? CONTOUR/OVER/PEN=5 slp*mp_mask, x_page, y_page
```

If, instead, we wished to overlay sea level pressure for the South Atlantic only, we would need to take advantage of the ***mp_central_meridian*** and ***mp_standard_parallel*** variables. Normally, the map projection scripts calculate the central meridian and standard parallel from the currently set region and generate the ***x_page*** and ***y_page*** coordinate transformations accordingly. When we overlay a subregion, we need to rerun the map projection script and pass in values for ***mp_central_meridian*** and ***mp_standard_parallel*** so that they are match the previous values and are not calculated from the subregion associated with the overlay. (Figure 6_13)



SST*MP_MASK

Figure 6_13

```
yes? USE coads_climatology
yes? SET REGION/L=1
yes? GO mp_hammer
yes? GO mp_grid sst
yes? GO mp_aspect
yes? SHADE/NOAXIS sst*mp_mask, x_page, y_page
yes? GO mp_fland
yes? LIST mp_central_meridian, mp_standard_parallel
        LONGITUDE: 20E to 20E(380)
        LATITUDE: 90S to 90N
 Column  1: MP_CENTRAL_MERIDIAN is (MP_X[I=@MAX] + MP_X[I=@MIN])/2
 Column  2: MP_STANDARD_PARALLEL is (MP_Y[J=@MAX] + MP_Y[J=@MIN])/2
        MP_CENTRMP_STAND
I / *:     200.0  0.0000
yes? GO mp_hammer 200 0
yes? SET REGION/X=60w:20e/Y=45s:0n
yes? GO mp_grid slp
yes? CONTOUR/OVER slp, x_page, y_page
```

Note: Had we used ***go mp_hammer 200 0*** in the beginning we would not have had to rerun ***mp_hammer***.

Ch6 Sec10.4.3. Map Projection scripts

Here is the list of map projection scripts delivered with Ferret. (The techniques used are quite general and can be applied to most map projections.)

Ferret script	Projection name
mp_bonne.jnl	Bonne
mp_craster_parabolic.jnl	Craster Parabolic
mp_eckert_greifendorff.jnl	Eckert Grifendorff
mp_eckert_iii.jnl	Eckert III
mp_eckert_v.jnl	Eckert V
mp_hammer.jnl	Hammer
mp_lambert_cyl.jnl	Lambert Cylindrical Equal Area
mp_mcbryde_fpp.jnl	McBryde Flat Polar Parabolic
mp_mercator.jnl	Mercator
mp_orthographic.jnl	Orthographic
mp_plate_caree.jnl	Plate Caree
mp_polyconic.jnl	Polyconic
mp_sinusoidal.jnl	Sinusoidal
mp_stereographic_eq.jnl	Stereographic Equatorial
mp_stereographic_north.jnl	Stereographic North
mp_stereographic_south.jnl	Stereographic Soutth
mp_vertical_perspective.jnl	Vertical Perspective
mp_wagner_vii.jnl	Wagner VII
mp_winkel_i.jnl	Winkel I

Here is the list of utility scripts that support curvilinear coordinates

Ferret script	Function
mp_demo.jnl	demonstration of various map projections
mp_fland.jnl	curvilinear version of ***`fland.jnl`***
mp_graticule.jnl	creates a graticule (lines of longitude and latitude) over the whole globe or any portion
mp_grid.jnl	Associates a data grid with a predefined map projection.
mp_label.jnl	correctly places labels using lat-lon coordinates
mp_land.jnl	curvilinear version of ***`land.jnl`***
mp_land_stripmap.jnl	creates a land-centric interrupted map using the current projection
mp_line.jnl	correctly plots user lat-lon data on the map
mp_ocean_stripmap.jnl	creates an ocean-centric interrupted map using the current projection
mp_polymark	overlays "map projected" polygons

Chapter 7: HANDLING STRING DATA: STRING VARIABLES AND "SYMBOLS"

Ferret offers a variety of tools for manipulating strings through the use of "symbols" (variables defined to be strings). In addition, beginning with Version 5.4, string variables are supported, with much the same syntax as for numeric variables.

Ch7 Sec1. STRING VARIABLES

String variables are defined using DEFINE VARIABLE (or its alias LET). They can be read from and written to netCDF files. Arrays of strings may be defined and a limited number of algebraic operations are defined for string variables. They can be passed to go scripts and functions. Grave accents around a scalar string return the string.

```
yes? LET astring = "hello everyone"
yes? LIST/NOHEAD astring
        "hello everyone"
yes? message `astring`
 !-> message hello everyone
hello everyone
  Hit Carriage Return to continue
```

Ch7 Sec1.1. String arrays

Strings in arrays may be of variable length. The syntax {"a","b","c"} denotes an array of strings. Two commas in a row denotes a null (missing value) string. Single and double quoted strings are both allowed, but must match for an individual string, e.g. 'P" is not valid.

Examples: the following define and list valid string arrays:

```
yes? LET a =  {"s1","s2", ,"s3"}

yes? LIST a
             {"s1","s2", ,"s3"}
 1   / 1:"s1"
 2   / 2:"s2"
 3   / 3:""
 4   / 4:"s3"

yes? LET b = {, 'string1','s2',,'cccc'}
yes? LIST/i=3:5 b
             {, 'string1','s2',,'cccc'}
 3   / 3:"s2"
 4   / 4:""
 5   / 5:"cccc"

yes? LET c = {'p', 'q', 'a longer string'}
yes? LIST/NOHEAD/ORDER=x c
        "p"            "q"            "a longer string"
```

Ch7 Sec2. STRING FUNCTIONS

A number of functions are available for working with string variables. They are described in the following sections. See also the section later in this chapter, "PLOT+ string editing tools" (p. 240) for more ways to manipulate

Ch7 Sec2.1. STRCMP(string1, string2)

string1: string
string2: string
result: real

Compares two strings or string arrays. It computes the numerical difference in ASCII character value for the first character that differs between the two strings returns a number, > 0 if string1 > string2, 0 if they are equal and < 0 if string1 < string2.

Examples:

```
yes? list strcmp("b",{"a","b","c"})
             VARIABLE : STRCMP("b",{"a","b","c"})
             SUBSET   : 3 points (X)
 1   / 1:  1.000
 2   / 2:  0.000
 3   / 3: -1.000

yes? list strcmp({"a","b","c"},YSEQUENCE({"a","b","c"}))
             VARIABLE : STRCMP({"a","b","c"},YSEQUENCE({"a","b","c"}))
             SUBSET   : 3 by 3 points (X-Y)
             1      2      3
             1      2      3
 1   / 1:  0.000  1.000  2.000
 2   / 2: -1.000  0.000  1.000
 3   / 3: -2.000 -1.000  0.000

yes? let a = "a longer string"
yes? list strcmp (a,"a longer stringg")
             VARIABLE : STRCMP (a,"a longer stringg")
         -103.0
```

Ch7 Sec2.2. STRLEN(string1)

string1: string
result: real

Returns the length of the string passed in string string1

Ch7 Sec2.3. UPCASE(string1)

string1: string
result: string

Returns the string passed in string string1 in all upper case characters

Ch7 Sec2.4. DNCASE(string1)

string1: string
result: string

Returns the string passed in string string1 in all lower case characters

Ch7 Sec2.5. STRINDEX(string1, substring)

string1: string
substring: string
result: real

Locate first occurrence of substring in string1. Returns a 0 if substring doesn't exist in string1. If substring contains a zero-character string (i.e. ""), the function returns 1.

Ch7 Sec2.6. STRRINDEX(string1, substring)

string1: string
substring: string
result: real

Locates last occurence of string substring in string1. Returns a 0 if substring doesn't exist in string1. If substring contains a zero character string (i.e. ""), the function returns the length of string1

Ch7 Sec2.7. SUBSTRING(string1, offset, len)

string1: string
offset: integer
len: integer
result: string

Returns substring of length len from string string1 beginning from character offset in string1. If offset is 0, or if offset is greater than the length of string string1, a NULL value is returned. If length len exceeds the total length of string string1, the value of string string1 starting at offset is returned.

Ch7 Sec2.8. STRCAT(string1, str2)

string1: string
string2: string
result: string

Append a copy of string string2 onto string string1.

Ch7 Sec2.9. STRFLOAT(string1)

string1: string
result: real

Return float value of string string1 (e.g. STRFLOAT("3.14"))

Ch7 Sec2.10. LABWID(string, charsize)

string: string, possibly including a font specification, and may include <NL> newline character
charsize: real

result: real

Returns the size in plot-page inches of the string. If a font is specified, it takes the size of characters in that font into account. If the string is a multi-line label, the result is computed for the longest line as it will appear on the plot page.

Examples:

```
      ! LABWID with two different character sizes,
      ! lowercase vs capitals, and the italics font

yes?  LIST/NOHEADER  LABWID("abcdefg", .15)
        0.8857
 yes? LIST/NOHEADER  LABWID("abcdefg", .10)
        0.5905
 yes? LIST/NOHEADER  LABWID("ABCDEFG", .15)
        0.9929
yes?  LIST/NOHEADER  LABWID("@IIabcdefg", .15)
        1.0000
```

```
     ! multi-line label: result is the length of the longest line
yes? LIST/NOHEADER  LABWID("abcdefg<NL>hi", .15)
       0.8857

     ! Make a plot, drawing one label after another.
     ! Use the italics font for the first label.

yes? PLOT/VS/LINE/I=1:314 i*cos(i/20),i*sin(i/20)

yes? LET firstlab = "@iiMy label"
yes? LABEL/NOUSER 0, 3, -1, 0, .15, "`firstlab`"
yes? LET wid = labwid("`firstlab`", .15)
yes? LABEL/NOUSER `wid`, 3, -1, 0, .15,  "@P4Second Label"
```

Ch7 Sec2.11. SPAWN command

The SPAWN command executes a Unix system command and returns the result in a string array. The syntax SPAWN:"command" inside a string array definition allows the output of a Unix command to be mixed with other strings.

Examples:

```
LET a = {"first.nc", SPAWN:"ls *.nc","last.nc"}
```

Say we want to check whether a file is in the directory. We do not want a null result, so start with a dummy string. If "myfile.nc" exists, there will be 2 entries in array a.

```
yes? LET a = {"dummy", SPAWN:"ls myfile.nc"}
yes? LET nfiles = `a,RETURN=IEND`

yes? IF `nfiles EQ 2` THEN ...
```

Ch7 Sec2.12. Algebraic operations with string variables.

A number of algebraic operations are available for string variables. They are described in the following sections.

Ch7 Sec2.12.1. Logical operators with strings

The operators EQ, LT, LE, ... can be applied to string variables and arrays. These operators are case-insensitive (functions will be provided later that are case-sensitive and include UPCASE, DNCASE)

Examples:

```
yes? LIST/NOHEAD {"a","b","c"} EQ {"A","B","C"} ! case insensitive
 1   / 1:  1.000
 2   / 2:  1.000
 3   / 3:  1.000
yes? LIST/NOHEAD"b" GT {"a","b","c"}
 1   / 1:  1.000
 2   / 2:  0.000
 3   / 3:  0.000
```

Ch7 Sec2.12.2. Shift transformation of string arrays

The shift transformation can be applied to string arrays.

For example:

```
yes? LET a = {"a","b","c","d"}
yes? LIST a[i=@SHF]
             {"a","b","c","d"}
             shifted by 1 pts on X
 1   / 1:"b"
 2   / 2:"c"
 3   / 3:"d"
 4   / 4:""
yes? LIST a[i=@SHF:-1]
            {"a","b","c","d"}
             shifted by -1 pts on X
 1   / 1:""
 2   / 2:"a"
 3   / 3:"b"
 4   / 4:"c"
```

Ch7 Sec2.12.3. Strings in IF-THEN-ELSE

IF cond THEN string_array1 ELSE string_array2

Example:

```
yes? LIST/NOHEAD IF {0,1} THEN "hello" ELSE "goodbye"
 1   / 1:"goodbye"
 2   / 2:"hello"
```

Ch7 Sec2.12.4. String concatenation with "+":

Examples:

```
yes? let a = "good" + "bye"
```

```
yes? LIST/NOHEAD YSEQUENCE({"now","then"})+", " + (if {0,1} THEN
"hello"+", ") + "friend"
1   / 1:"now, friend"        "now, hello, friend"
2   / 2:"then, friend"       "then, hello, friend"
```

Ch7 Sec2.12.5. Strings as Function arguments

A few functions also take strings as arguments. String arguments must be enclosed in double quotes. For example, a function to write variable "u" into a file named "my_output.v5d", formatted for the Vis5D program might be implemented as

```
LOAD WRITE_VIS5D("my_output.v5d", a)
```

SAMPLE* functions may take string or numerical arrays as arguments

Example:

```
yes? LIST/NOHEAD  SAMPLEI({"a","b","c","d","e","f"},{3,2,,1})
 1   / 1:"c"
 2   / 2:"b"
 3   / 3:""
 4   / 4:"a"
yes? LIST/NOHEAD  SAMPLEJ(YSEQUENCE ({"a","b","c","d","e","f"}),
{3,2,,1})
 1   / 1:"c"
 2   / 2:"b"
 3   / 3:""
 4   / 4:"a"
```

Ch7 Sec2.12.6. Regridding string arrays

The regridding transformations @ASN, @XACT, @NRST can be used with character data.

Examples:

```
yes? LET a = {spawn:"ls *.nc"}
yes? LIST a
             {SPAWN:"ls *.nc"}
 1   / 1:"d1.nc"
 2   / 2:"d2.nc"
 3   / 3:"d3.nc"
 4   / 4:"d4.nc"
 5   / 5:"d5.nc"
 6   / 6:"d6.nc"
 7   / 7:"d7.nc"

yes? DEFINE AXIS/X=0.1:0.7:.1 xasn
yes? LIST a[gx=xasn@ASN]
             {SPAWN:"ls *.nc"}
             regrid: 0.1 delta on X@ASN
 0.1   / 1:"d1.nc"
 0.2   / 2:"d2.nc"
```

```
 0.3   / 3:"d3.nc"
 0.4   / 4:"d4.nc"
 0.5   / 5:"d5.nc"
 0.6   / 6:"d6.nc"
 0.7   / 7:"d7.nc"

yes? DEFINE AXIS/X=1:6:.5 xxact
yes: LIST a[gx=xxact@XACT]
             {SPAWN:"ls *.nc"}
             regrid: 0.5 delta on X@XACT
 1    /  1:"d1.nc"
 1.5  /  2:""
 2    /  3:"d2.nc"
 2.5  /  4:""
 3    /  5:"d3.nc"
 3.5  /  6:""
 4    /  7:"d4.nc"
 4.5  /  8:""
 5    /  9:"d5.nc"
 5.5  / 10:""
 6    / 11:"d6.nc"

yes? DEFINE AXIS/X=1:6:.4 xnrst
yes? LIST a[gx=xnrst@NRST]
             {SPAWN:"ls *.nc"}
             regrid: 0.4 delta on X@NRST
 1    /  1:"d1.nc"
 1.4  /  2:"d1.nc"
 1.8  /  3:"d2.nc"
 2.2  /  4:"d2.nc"
 2.6  /  5:"d3.nc"
 3    /  6:"d3.nc"
 3.4  /  7:"d3.nc"
 3.8  /  8:"d4.nc"
 4.2  /  9:"d4.nc"
 4.6  / 10:"d5.nc"
 5    / 11:"d5.nc"
 5.4  / 12:"d5.nc"
 5.8  / 13:"d6.nc"
 6.2  / 14:"d6.nc"
```

Ch7 Sec2.13. NetCDF input and output of string data

String variables can be input and output to netCDF files. In the file the string axis is the fastest moving dimension and all strings are the same length (equal to the maximum length of the strings being written). Extra character spaces are padded with nulls. If variable length strings are written out, then when read back they will again be variable length.

Example:

```
yes? SAVE/CLOBBER/FILE=test_string.cdf/HEADING=enhanced a[i=2:4]
```

Ch7 Sec3. SYMBOL COMMANDS

The following are the relevant commands:

DEFINE SYMBOL
usage:
DEFINE SYMBOL symbol_name = string

SHOW SYMBOL
usage:
SHOW SYMBOL/ALL
SHOW SYMBOL symbol_name
SHOW SYMBOL partial_name

CANCEL SYMBOL
usage:
CANCEL SYMBOL/ALL
CANCEL SYMBOL symbol_name

Legal symbol names must begin with a letter and contain only letters, digits, underscores, and dollar signs.

To invoke symbol substitution—the replacement of the symbol name with its (text) value—within a Ferret command include the name of the symbol preceded by a dollar sign in parentheses.

For example,

```
yes? DEFINE SYMBOL hi = hello everyone
yes? MESSAGE ($hi)              ! issues "hello everyone" msg
```

It is also possible to nest symbol definitions, as the following commands illustrate:

```
yes? DEFINE SYMBOL label_2 = My test label
yes? DEFINE SYMBOL number = 2
yes? SAY ($label_($number))
      My test label
```

Ch7 Sec4. AUTOMATICALLY GENERATED SYMBOLS

A number of useful symbols are automatically defined whenever Ferret sets up a plot. Following any plotting command issue the command SHOW SYMBOLS/ALL to see a list. Consult the PLOT PLUS for Ferret Users Guide (section "General Global Symbols") for detailed descriptions of the plot symbols. For example, if we wish to place a label “hello” at the upper right corner of a plot we might do the following

```
yes? PLOT/I=1:100 SIN(I/6)
yes? LABEL/NOUSER ($ppl$xlen) ($ppl$ylen) 1 0 .2 hello
```

This labeling procedure would work regardless of the aspect ratio of the plot. Use the command SHOW SYMBOL/ALL to see the symbols (and see "General Global Symbols" in the PLOT+ Users Guide).

Ch7 Sec5. USE WITH EMBEDDED EXPRESSIONS

When used together with Ferret embedded expressions symbols can be used to perform arithmetic on the plot geometry. For example, this command will locate the plot title in bold at the center of a plot regardless of the aspect ratio:

```
yes? LABEL/NOUSER `($ppl$xlen)/2` `($ppl$ylen)/2` 0 0 .2 @AC($labtit)
```

Ch7 Sec6. ORDER OF STRING SUBSTITUTIONS

The above example illustrates that the order in which Ferret performs string substitutions and evaluates immediate mode expressions in the command line is significant. The successful evaluation of the embedded expression `***($ppl$xlen)/2***` requires that ***(pplxlen)*** is evaluated before attempting the divide by 2 operation. The order of Ferret string substitutions is as follows:

1. substitute "GO" command arguments of the form "$1", "$2", ...
2. substitute symbols of the form ($symbol_name) (discussed here)
3. substitute command aliases
4. substitute immediate mode expressions. (But see example 3 below).

Example 1

If the script snoopy.jnl contains

```
DEFINE SYMBOL fcn = $1
DEFINE ALIAS ANSWER LIST/NOHEAD/FORMAT=("Result is ",$2)
ANSWER `($fcn)(($3^2)/2)`+5
```

then the command

```
yes? GO snoopy EXP F5.2 2.25
```

would evaluate to

```
DEFINE SYMBOL fcn = EXP
DEFINE ALIAS ANSWER LIST/NOHEAD/FORMAT=("Result is ",F5.2)
LIST/NOHEAD/FORMAT=("Result is ",F5.2) `EXP((2.25^2)/2)`+5
```

and would result in Ferret output of "Result is 17.57."

Example 2

We can use grave accent syntax and string variables to substitute the string into the command line.

```
yes? LET my_reg =  "X=0:180,Y=-40:40,L=1"
yes? SHADE sst[`my_reg`]
```

Example 3

Immediate mode substitution of a string variable may be used to set the values of qualifiers. However the region qualifiers (/X=/Y=etc.) on a command are used to set the context for the grave accent expression. So Ferret parses command qualifiers before it parses grave accent expressions. Thus we can use this syntax to set a region:

```
yes? let xreg = "40:180"
yes? let yreg = "60S:42S"
yes? set region/x=`xreg`/y=`yreg`
```

But including the qualifier name in the string variable is NOT valid (the qualifier is parsed BEFORE the grave accent expression is substituted, so Ferret would issue the error that `my_region` is an unknown qualifier).

```
yes? !THE FOLLOWING SYNTAX IS NOT VALID
yes? LET my_region = "x=40:180/y=-60:-42"; set region/`my_region`
```

Ch7 Sec7. CUSTOMIZING THE POSITION AND STYLE OF PLOT LABELS

All of the plot labels generated by Ferret are automatically defined as symbols. This includes the title ($labtit), X and Y axis labels ($labx),($laby), as well as the position labels (latitude, longitude, depth, time), which are normally placed at the upper left on a plot (see "Labels," p. 191). Sometimes it is desirable to change the location, size or fonts of these labels. The symbol facility makes it possible to do this in a way that is independent of the particular label strings or plot aspect ratio. See the demonstration script symbol_demo.jnl for an example.

Ch7 Sec8. USING SYMBOLS IN COMMAND FILES

Often in Ferret command files the identical argument substitutions must be repeated at several points in the file. Using symbols it is possible to write "cleaner" Ferret scripts in which the argument substitution occurs only once—to define a symbol which is used in place of the argument thereafter. See the demonstration script symbol_demo.jnl for an example.

Ch7 Sec9. PLOT+ STRING EDITING TOOLS

The PLOT+ program provides a variety of tools for editing symbol strings. See the PLOT+ Users Guide for further information (p. 544). The special functions manipulate and reformat character strings.

Note that many of these functions are handled directly by Ferret string functions such as STRINDEX, STRLEN, SUBSTRING, etc (p. 230).

The general format is SET sym $function(arg1, arg2,...). The functions are:

$EDIT(symbol,argument)	Edit a symbol: change to uppercase, remove extra blanks, or remove all blanks
$EXTRACT(start,length,symbol)	Extracts selected characters from the input string.
$INTEGER(symbol)	Converts a number to integer format
$LENGTH(symbol)	Returns the length of the input string
$LOCATE(substring,symbol)	Locates a substring in the input string (compares only the first 30 characters of the substring.)
$ELEMENT(position,delimiter,symbol)	Extracts an element from an input string in which the elements are separated by a specified delimiter.

Example:

```
yes? DEFINE SYMBOL test = my    string
yes? PPL SET upper_test $EDIT(test,COMPRESS)
yes? SHOW SYMBOL upper_test
UPPER_TEST = "my string"
```

Ch7 Sec10. SYMBOL EDITING

Symbols may be edited and checked using the same controls that apply to journal file arguments.

The section of this users guide entitled "Arguments to GO tools" (p 25) describes the syntax for checking and editing arguments. The identical syntax applies to symbols. As with the GO tool arguments (e.g., "$4"), all string manipulations are case insensitive.

In brief, the capabilities include:

default strings

If a symbol is undefined a default value may be provided using the pattern ($my_symbol%my default string%). For example,

```
($SHAPE%XY%)
```

check against list of acceptable values

A list of acceptable string values may be provided using the pattern ($my_symbol%|option 1|option 2|%). For example,

```
($SHAPE%|X|Y|Z|T|%)
```

will ensure that only 1-dimensional shapes (X, Y, Z, or T) are acceptable.

string substitution

Any of the optional string matches provided can invoke a substitution using the pattern ($my_symbol%|option 1>replacement|%). For example,

```
($SHAPE%|X>I|Y>J|Z>K|T>L|%)
```

will substitute I for X or J for Y, etc.

Asterisk ("*") provides default substitution

The asterisk character matches any string. For example,

```
($SHAPE%|X|Y|Z|T|*>other%)
```

will always result in "X," "Y," "Z," "T," or "other."

Asterisk ("*") provides limited string editing

The asterisk character, when used on the right hand side of a string substitution, inserts the original symbol contents

```
($SHAPE%|*>The shape is *|%)
```

error message control

An error message can be provided if the symbol is undefined or doesn't match any options. The pattern for this is
($my_symbol%|option 1|option 2|<error message text %). For example,

```
($SHAPE%|X|Y|Z|T|<Not a 1-dimensional shape%)
```

Ch7 Sec11. SPECIAL SYMBOLS

PPLUS defines a number of global symbols which are available to the user. They are documented in the PPLUS Users Guide, section 7.3, and listed in the chapter "Customizing Plots", section PPLUS special symbols (p.219).

There are a few symbols, generated automatically by plots, which are not documented in the PLOT PLUS for Ferret Users Guide. Those are shown like all symbols by SHOW SYMBOLS, but cannot be redefined by the user.

```
PPL$XPIXEL
PPL$YPIXEL
```

the number of pixels in the horizontal (X) and vertical (Y) size of the current Ferret output window. Note: these are "0" if there is no current window -- hence they can be used as a test of whether there is an open window.

```
BYTEORDER
```

gives "BIG" or "LITTLE" according to endianness of the CPU

```
FERRET_VERSION
FERRET_PLATFORM
```

give the Ferret version and the platform Ferret is running on.

```
SESSION_DATE
SESSION_TIME
```

gives the date and time when the current session began.

Chapter 8: WORKING WITH SPECIAL DATA SETS

Ch8 Sec1. WHAT IS NON-GRIDDED DATA?

Many data sets which are not normally regarded as "gridded" can nonetheless be managed, analyzed, and visualized effectively in a gridded data framework. Track lines, "point data", etc. are common examples of "non-gridded" data. Profiles and time series, although they are individually simple one-dimensional grids, have a non-gridded structure when considered as a collection, which is often essential.

This chapter addresses a number of classes of non-gridded data sets and offers approaches that make it straightforward to work with these data types in Ferret's gridded data framework. The approaches are all conceived to facilitate a fusion of these data types—so that multiple data types may be easily combined in calculations..

"Point data" refers to collections of values at scattered locations and times. An example would be the column burden of oceanic NO3 and the scattered locations and times at which the measurements were made.

- If at each point of the data scattered there is a vertical profile of values then see COLLECTIONS OF VERTICAL PROFILES (p. 247).
- If at each point of the data scattered there is a time series of values then see COLLECTIONS OF TIME SERIES (p. 250).
- If at each point of the data scattered there is a 2-dimensional grid in the ZT plane then see COLLECTIONS OF TIME SERIES (p. 250).
- If at each point of the data scattered there is a time series of values then see COLLECTIONS OF TIME SERIES (p. 250).

Ch8 Sec2. POINT DATA

In a gridded context point data is best viewed as a collection of 1-dimensional variables, where the axis of each variable is the index value, 1, 2, 3, ... of the individual point in the scatter. Thus, continuing our example of an oceanic NO3 data set, we would want to view this as four variables, longitude, latitude, date, and burden, where each variable was defined on a one-dimensional axis of earthquake number. Typically, this sort of data is organized in a table of the form

Index	longitude	latitude	year	month	day	N03
1	160	30	1968	11	-999	6.2
2	33.1	60.2	1992	5	13	5.5
...						

Ch8 Sec2.1. Getting point data into Ferret

Since point data sets are most commonly available in table form, where the columns of the table are the variables and each row of the table is a separate point. In the chapter "Data Set Basics", section "Reading ASCII Files" (p. 46), example 2 and subsequent examples show how such a file might be read into Ferret.

For example, let us suppose that the file above is introduced to Ferret with the command

```
yes? FILE/VAR="index,lon,lat,yr,mn,day,NO3"/SKIP=1 my_data_file.dat
yes? SHOW DATA my_data_file.dat

         currently SET data sets:
        1> ./my_data_file.dat  (default)
     name     title                          I          J          K
L
     LON      LON                          1:20480    ...        ...
…
     LAT      LAT                          1:20480    ...        ...
…
     YR       YR                           1:20480    ...        ...
…
     MN       MN                           1:20480    ...        ...
…
     DAY      DAY                          1:20480    ...        ...
…
     NO3      NO3                          1:20480    ...        ...
…
```

Note that the SET VARIABLE command would normally be used as well to assign titles, units, and missing value flags to the variables.

Also note that until the first data is actually requested from the file, Ferret does not know the size of the file. The /GRID= option may be used to tell Ferret what size to expect. Lacking a /GRID specification the "1:20480" is the size of the default grid "EZ." After the first data access SHOW GRID will reveal the true size of the file, instead. If the size still appears to be 20480 it may be that the default grid EZ was not large enough, and the /GRID qualifier must be used to pre-allocate sufficient space.

Ch8 Sec2.2. How point data is structured in Ferret

In table form (above) each column represents a dependent variable; the column for "burden" and the column for "latitude" have equal status. In many cases this is an adequate representation. For example, a plot of NO3 burden versus latitude could be produced with the command

```
yes? PLOT/VS lat, NO3
```

To combine point data organized in tables with gridded data sources, say a gridded field of oceanic temperature two approaches are available. Either the gridded data may be viewed in the structure of the table, or the scattered data may be viewed in a geo-referenced 1-dimensional

grid structure. The problem to be solved determines which approach is suitable. The next two sections describe these two approaches.

Ch8 Sec2.2.1. Working with dates

Ferret V5.0 does not understand formatted dates inside of generic data ASCII files. To use the dates intelligibly inside of Ferret you

1. Need to get the year, month, and day fields broken out separately or provide a Julian day. SET DATA/FORMAT=DELIMITED (p. 402) is helpful for inputting date information.
2. Can create a Julian date from year, month, day using function DAYS1900. If a time origin other that 1-jan-1900 is needed subtract DAYS1900(year0, mon0, day0). For help in creating the dates, see the FAQ, "How can I create a time axis from variables containing year, month, day, etc?" at http://ferret.pmel.noaa.gov/Ferret/FAQ/axes_and_data/time_axis_from_variables.html
3. Can create an axis of dates as done in the preceding latitude axis example.

See the chapter "Grids and Regions", section "Time" (p. 164) and the section in the chapter "Converting to NetCDF" on "Converting time word data to numerical data" (p. 282) for details of creating time axes.

Ch8 Sec2.3. Subsampling gridded fields onto point locations and times

Ferret can be used as a tool to extract variables from gridded data sets at time/space locations to match the scatter of the point data. In this form they may, effectively, be combined into the table of data read from the ASCII (or binary) file. For example, suppose we want to obtain values of sea surface temperature at the locations of our NO3 samples, from a climatological annual average SST field. This may be accomplished simply with

```
yes? use coads climatology
yes? let ssttav = sst[l=1:12@ave]
yes? let my_lon = lon[d=my_data_file.dat]
yes? let my_lat = lat[d=my_data_file.dat]
yes? LET sst_xy = SAMPLEXY(ssttav, my_lon, my_lat)
```

Suppose that instead we defined our XY sampling based upon the 12 month time series of SST grids as in

```
yes? LET sst_xy = SAMPLEXY(sst, my_lon, my_lat)
```

The variable sst_xy as defined above would then have a two-dimensional structure: sample index by 12 months. To sample this in time we use

```
yes? LET zero = 0 + 0*mn
yes? LET sst_t = SAMPLET_DATE(sst_xy,zero,mn,day,zero,zero,zero)
```

Note that the year is entered simply as 0, since SST is a climatological variable, and the variable "zero" is the same length as the variables "mn" and "day". To sample a field at hour 12 of each day, we could use "zero+12" for the hours argument.

In this example we sampled a field in X, Y, and T. The sst data was sampled at each time. If we were sampling a field which had a Z axis, that axis would be inherited from the first argument to SAMPLEXY in the same way; it would be sampled at the (x,y) points at each Z level.

Ch8 Sec2.4. Defining gridded variables from point data

There are functions to interpolate scattered data onto a grid. See the scat2gridgauss and scat2gridlaplace functions (p. 95ff). These functions map irregular locations to a regular grid.

For some calculations one may want to let Ferret know which of the variables are dependent (measurements) and which are independent (coordinates). For example, suppose we wish to compute the average column burden of NO3 as a function of latitude. Burden here is an integral of the concentration NO3 over depth. We will want to see our variable burden on an axis of latitude.

The steps to do this are

1. In general, the latitude variable will not be sorted into strictly increasing order — needed to create an axis. Determine the sorting order for latitude using
   ```
   yes? LET lat_index = SORTI(lat)
   ```
2. Create a latitude grid
   ```
   yes? DEFINE AXIS/FROM/NAME=lat_ax/Y/UNITS=degrees SAMPLEI(lat,
lat_index)
   yes? DEFINE GRID/Y=lat_ax glat
   yes? LET NEW = Y[g=glat] ! a dummy variable to use in RESHAPE below
   ```
3. Define your function for the burden based on the variable NO3, on the command line or using your script my_brdn.jnl.
   ```
   yes? GO my_brdn NO3 burden
   ```
4. Define a new variable burden_on_lat using this axis
   ```
   yes? LET sorted_burden = SAMPLEI(burden, lat_index)
   yes? LET burden_on_lat = RESHAPE( sorted_burden, new )
   ```
5. Now, to plot the NO3 burden averaged into 5 degree latitude bands we could use
   ```
   yes? PLOT burden_on_lat[Y=60s:30n:5@AVE]
   ```

Ch8 Sec2.5. Visualization techniques for point data

Scattered point data can be displayed in a number of ways.

A simple scatter plot showing the locations of points

```
yes? PLOT/VS lon,lat
yes? GO land
```

Use ***GO/help*** land for an explanation of resolving incompatible longitude encodings, should they arise.

A scatter plot in which the symbols are colored by value with control over the color palette and resolution can be made using the polymark.jnl script. For example, to plot using stars symbols in color levels by 10s use

```
yes? GO polymark POLYGON/LEV=(0,100,10) lon lat NO3 star.
```

Type ***GO/HELP*** polymark for more options.

See also the chapter "Customizing Plots", section "Map Projections" (p. 224) for guidance on plotting scattered data. The map projection scripts can be used in conjunction with the above.

Ch8 Sec3. VERTICAL PROFILES

A single profile, possibly consisting of multiple variables, can be regarded as a simple 1-dimensional data set. Ferret's plotting and analysis tools apply in a straightforward manner.

Collections of profiles resemble point data sets in their X,Y, and T structure, however at each point there is a 1-dimensional Z-axis structure. In general, the Z axes at each point may differ.

Ch8 Sec3.1. How collections of profiles are structured in Ferret

If the collection of profiles is sufficiently small (say 4 or fewer) then it is straightforward to handle them simply as 4 separate data sets. The D= qualifier may be used to designate which profile is being referred to. The IF ... THEN ... ELSE syntax may be used to combine the profiles into expressions.

As the number of profiles in the collection grows larger, however, it becomes necessary to merge them into a single structure. Typically, the sequence number of the profile, 1, 2, ...,N, becomes the X axis of the collection. The longitude, latitude, and time of each profile become dependent variables indexed by the sequence number. The Z structures of the profiles are blended into a single Z axis by a choice of techniques. The steps to creating a blended data set then become:

1. Determine the nature of the Z axis to be used and the collection of variables to be defined on the grid
2. Create an empty grid with the desired structure in a file
3. Populate the file with the profiles, each profile in turn.

The determination of the Z axis structure may be by any of these techniques:

1. Supply an arbitrary Z axis to which all of the individual profiles will be regridded by linear interpolation. This technique produces a data set which is very easy to work with and small in size, however, some of the data have been altered by linear interpolation. The default Ferret regridding (GZ=@LIN) is used for this technique.
2. Create a Z axis which is a superset of the Z axis points from all of the grids. In the final data set this axis will be sparsely populated, containing only those Z points that were actually present in each profile.
   This technique produces a data set which is 100% faithful to the original data and reasonably easy to work with, but may become very large if the number of profiles is large and the Z axes vary greatly. Ferret "exact match" regridding (GZ=@XACT) is used for this technique.
3. Do not create a Z axis at all — instead store the Z coordinates as a dependent variable. The Z axis becomes simply an index counter of length equal to the longest profile. This technique produces a data set which is 100% faithful to the original data and of modest size, however it is the most laborious to work with.

The choice of technique depends on the nature of the profile collection and the types of analysis or visualization to be done. Often it is desirable to combine technique 1, which is fast and simple with 2 or 3, which can be used for spot checking if there is a question of data fidelity. If method 3 is chosen (Z coordinates in a dependent variable) the techniques for handling the variables are very similar to sigma coordinate data, described in a separate section of this chapter (p. 251).

Ch8 Sec3.2. Getting profile data into Ferret

As of 4/99 the approaches to merging collections of profiles into a single structure are still "manual." (Data which are stored as global attributes in the input files, as is done in EPIC files, are lost in this process.) This text describes an example of the manual process used, where the target Z axis is created arbitrarily and data are interpolated to it. In this example the profiles are read from ASCII files, so the Z axis of each profile has to be created. This example does not save the longitude, latitude, and time positions of the casts.

```
! for this example we begin by manufacturing some data
! ... pretend this is one of your casts - unequal vertical spacing
LIST/FILE=test_cast.dat/NOHEAD/FORM=(2F)/I=1:10 10*i+randu(i), sin(i/6)

! create a grid suitable for ALL casts together
! make the points regular in X and Z ... they need not be, however
DEFINE AXIS/DEPTH/Z=0:1000:20/UNIT=meters zall ! Arbitrarty z axis
DEFINE AXIS/X=0:9:1/UNIT="sequence" xall
DEFINE GRID/X=xall/Z=zall gall

! create an empty output file
! if we were reading netCDF files we would create variables to hold
! longitude, latitude, and time (year, month, day).
! A latitude output variable, for example, is created below

LET outvar = 1/0 * x[g=gall] * z[g=gall]
SET VARIABLE/TITLE="My merged var"/UNITS="my units" outvar
```

```
SAVE/FILE=all_casts.cdf/ILIMITS=1:10/ZLIMITS=0:1000 outvar
LET LAT = 1/0*X[gx=gall]
SET VARIABLE/TITLE="Latitude"/UNITS="degrees" lat
SAVE/APPEND/FILE=all_casts.cdf/ILIMITS=1:10 lat

! read in a single cast (the fake data we created)
! if we were reading a netCDF file this block would be unnecessary
FILE/VAR=depth,invar test_cast.dat

! make Z axis for 1 profile
DEFINE AXIS/Z/DEPTH/UNIT=meters z1cast=depth
DEFINE AXIS/X=0:0:1/UNIT="sequence" x1cast   ! sequence no. of 1st cast
DEFINE GRID/X=x1cast/Z=z1cast g1cast
CANC DATA 1

! save first cast interpolated to many-point Z axis
FILE/VAR="-,invar"/GRID=g1cast test_cast.dat
LET outvar = invar[g=gall]
SAVE/APPEND/FILE=all_casts.cdf outvar[I=1]
CANC DATA 1

! if available, output latitude thusly
!   LET lat = 0*X[g=gall] + RESHAPE(Y[G=invar],X[gx=gall])
!   SAVE/append/file=all_casts.cdf lat[I=1]

! save next cast

DEFINE AXIS/X=1:1:1/UNIT="sequence" x1cast    ! X position of 2nd cast
FILE/VAR="-,invar"/grid=g1cast test_cast2.dat
SAVE/APPEND/FILE=all_casts.cdf outvar[I=2]
CANC DATA 1

! etc for next 8 casts …
! This may be automated with:   REPEAT/I=1:10 GO output_one_profile
! where the script output_one_profile.jnl reads profile file names
! from a list
```

The output data set which we create will be structured as follows:

```
yes? CANCEL VAR/ALL
yes? USE all_casts
yes? SHOW DATA
     currently SET data sets:
    1> ./all_casts.cdf  (default)
 name     title                                I         J         K
L
 OUTVAR   My merged var                      1:10      ...       1:51
…
 LAT      Latitude                           1:10      ...       ...
…
```

Ch8 Sec3.3. Defining vertical sections from profiles

In the data set created above the profiles may or may not be ordered as needed to create a valid section. There are many possible ways to order the data. Often more than one technique is applicable to a single data set. The data may be ordered along a ship track, ordered by increasing latitude, ordered by path distance along a regression line, etc.

Continuing with the example above, we can order the profiles into increasing latitude with:

```
yes? let order = SORTI(lat)
yes? let section = SAMPLEI(outvar, order)
```

Other definitions of the variable order may be created by straightforward means to apply other ordering principles.

As defined above, "section" has an X axis which is the values 1, 2, 3,...N from the Ferret ABSTRACT axis. To cast this on a proper latitude axis, use these two steps:

```
yes? DEFINE AXIS/Y/UNITS=degrees yax_sect=SAMPLEI(lat, order)
yes? LET ysection = RESHAPE(section,Y[gy=yax_sect]+Z[gz=all])
```

Ch8 Sec3.4. Visualization and analysis techniques for profile sections

The variables "section" and "ysection" defined above may be plotted and analyzed with the normal gridded plot commands. For examples,

```
yes? CONTOUR section  ! contour plot ordered on X=1,2,3,...
yes? FILL ysection    ! color contour plot on formatted latitude axis
yes? PLOT/Y=20S/Z=100:500 ysection ! profile at 20 south
yes? PLOT ysection[Z=@loc:20]      ! depth of 20 degree isotherm
```

Ch8 Sec3.5. Subsampling gridded fields onto profile coordinates

The technique described for sampling grids at scattered point values will work unmodified for collections of vertical profiles. The Z coordinate of the gridded variable will be retained unmodified throughout the sampling operations. Regrid the final result variable to other Z axes as desired.

Ch8 Sec4. COLLECTIONS OF TIME SERIES

Handling of collections of time series is analogous to handling collections of vertical profiles, described above. The choices of

1. a single interpolated time axis (using the default, GT=@LIN, regridding)
2. a super-set of all times axis (using "exact match," GT=@XACT, regridding)

should be considered. Choice 3, in which time would be handled as an independent variable, is possible, but awkward, due to the multiplicity of time encodings.

Ch8 Sec5. COLLECTIONS OF 2-DIMENSIONAL GRIDS

Handling collections of 2-dimensional grids (e.g. ZT grids from acoustic current profilers) is a straightforward extension of the techniques described under collections of profiles. If the time axes of the input grids are all identical, no additional work is needed beyond the techniques described there. If the time axes differ then follow the guidance given under Collections of Time Series, using intermediate variable definitions that reconcile the time axes into a single uniform axis before saving the input variables into a merged output file.

Ch8 Sec6. LAGRANGIAN DATA

Lagrangian data (ship tracks, drifters, etc.) is a special case of scattered point data described in a preceding section. In the terminology of "Defining gridded variables from point data" Lagrangian data is simply point data organized onto a 1-dimensional time axis grid.

Ch8 Sec6.1. Visualization techniques for Lagrangian data

Ferret has several visualization tools that specifically address the needs of Lagrangian data. There are three scripts:

polymark (polymark_demo)	marks value-colored symbol at each location
polytube (polytube_demo)	creates a line following the Lagrangian track with color varying according to a Lagrangian variable
trackplot (trackplot_demo)	creates a line plot of a Lagrangian variable where the zero line of the plot follows the Lagrangian track

Overlays of the trackplot script are useful to visualize more than one variable. Run the demonstration scripts noted above for each tool for an example of its use with Lagrangian data.

Ch8 Sec7. SIGMA COORDINATE DATA

With sigma coordinate data the vertical coordinate (or layer thickness) is available as a dependent variable and the Z axis of the sigma-encoded variables is layer number (the Z index). This is precisely analogous to method 3 of handling collections of profiles, above. (p. 248). The family of ZAXREPLACE functions may be used to regrid this kind of data to a Z axis with physical units (p. 85)

See also the FAQ on Using Sigma Coordinates.

Ch8 Sec7.1. Visualization techniques for sigma coordinate data

Visualizations of sigma coordinate data in vertical section planes are best handled with the 3-argument versions of the SHADE, FILL, CONTOUR and VECTOR commands. See further information in Customizing Plots (, p. 183).

For visualization of sigma coordinate data in other planes or orientations use the techniques described in the next section.

Ch8 Sec7.2. Analysis techniques for sigma coordinate data

Analysis of sigma coordinate data, which requires shifting to depth or pressure coordinates, is facilitated by the function ZAXREPLACE, which converts from layer number to other vertical coordinate axes. See sigma_coordinate_demo.jnl for an example. If the data set provides layer thickness rather than depth a depth variable may be created using integration with @iin.

Ch8 Sec8. CURVILINEAR COORDINATE DATA

By "curvilinear coordinate data" we refer to data which is curvilinear in the XY plane there. We presume that the X,Y coordinates (typically longitude, latitude) are available through other dependent variables.

Here is an example showing a curvilinear grid, taken from http://www.wldelft.nl/rnd/intro/topic/2003-swe/.

Figure 8_1

Curvilinear data may be defined by a map projection (see p.224), or by data in a file that has a curvilinear grid. A curvilinear grid has longitudes and latitudes defined by coordinates (lon[i, j],lat[i, j]) in 2D, and the data fields are also defined on the [i,j] index grid. In the CF standard for netCDF files at http://www.cgd.ucar.edu/cms/eaton/cf-metadata/CF-1.0.html, thses grids are discussed in the section titled " Two-dimensional latitude, longitude coordinate variables". The netCDF header for a file containing data on a curvilnear grid looks like this (when viewed

with the Unix command ncdump -h). Note how the coordinate variables lon and lat are 2-D fields. The coordinate variables and the data field tmp share a grid in index space.

```
dimensions:
  xc = 180 ;
  yc = 173 ;
variables:
  float xc(xc) ;
    xc:long_name = "x-index in Cartesian system" ;
    xc:units = "m" ;
  float yc(yc) ;
    yc:long_name = "y-index in Cartesian system" ;
    yc:units = "m" ;
  float lon(yc,xc) ;
    lon:long_name = "longitude" ;
    lon:units = "degrees_east" ;
  float lat(yc,xc) ;
    lat:long_name = "latitude" ;
    lat:units = "degrees_north" ;
  tmp:long_name = "temperature" ;
    tmp:units = "K" ;
    tmp:coordinates = "lon lat" ;
```

When such a dataset is opened in Ferret, the output of a SHOW DATA command will look like:

```
yes? show data
     currently SET data sets:
    1> ./tmp.nc  (default)
 name     title   I         J         K         L
 TMP temperature 1:180     1:173     ...       ...
 LON longitude   1:180     1:173     ...       ...
 LAT longitude   1:180     1:173     ...       ...
```

This data can be plotted with the 3-argument SHADE, FILL or CONTOUR commands

```
yes? SHADE tmp, lon, lat
```

You can see what the grid looks like by doing a shade plot of the coordinate variables:

```
yes? set view ul; shade lon
yes? set view ll; shade lat
```

Ch8 Sec8.1. Visualization techniques for curvilinear coordinate data

Visualizations of curvilinear coordinate data in the XY plane section planes are best handled with the 3-argument versions of the SHADE, FILL, and Contour commands. See further information in the chapter "Customizing Plots" (p. 183).

For visualization of curvilinear coordinate data in other planes or orientations use the techniques described under "Analysis techniques for curvilinear coordinate data."

Ch8 Sec8.2. Analysis techniques for curvilinear coordinate data

Analysis of curvilinear coordinate data may be done in the curvilinear coordinate system or in a rectilinear (including lat-long) coordinate system. If the analysis is done in the curvilinear coordinate system, it is the responsibility of the user to ensure that the proper geometric factors are applied when integrals and derivatives are computed. Converting other fields to the curvilinear coordinate system is most easily accomplished with the function RECT_TO_CURV. Curvilnear grids may be converted to rectilinear grids using the functions CURV_TO_RECT_MAP and CURV_TO_RECT.

Ch8 Sec9. POLYGONAL DATA

By "polygonal data" we refer to a class of point data set where each point represents a polygonal region rather than a single coordinate. An example of polygonal data would be a value associated with each state in the United States.

Ch8 Sec9.1. Visualization techniques for polygonal data

Visualizations of polygonal data is best handled with the POLYGON command. If the coordinates of the polygon vertices are available in 2-dimensional arrays, XPOLY and YPOLY, in which the axes of the arrays are the polygon vertices and the sequence of polygons the use of the POLYGON command is straightforward. The POLYGON command can also handle sequences of polygons encoded in 1-dimensional arrays with missing values separating each polygon.

Ch8 Sec9.2. Analysis techniques for polygonal data

Ferret version 5.0 does not have any tools specifically addressing the analysis of polygonal data sets. The analysis of these data sets in Ferret requires the creation of a gridded mask field corresponding to the polygonal regions (an external function could be written that would create a gridded mask of arbitrary resolution from polygonal coordinates.)

Once the mask is created, the standard gridded operators for averaging, integrating, etc. can be used. For example, if variable cal_mask contains a gridded mask of the state of California on latitude and longitude axes of 10 minute resolution then this definition would compute the average of a gridded variable, var, over California:

```
yes? let cal_var = mask * var[g=mask]
yes? let cal_average = cal_var[x=@ave, y=@ave]
```

Chapter 9: COMPUTING ENVIRONMENT

Ch9 Sec1. SETTING UP TO RUN FERRET

This discussion assumes that Ferret is already istalled on your system. Installation documentation is available separately from the Ferret Downloads web page

STEP 1

Execute interactively or add to your .login file the Unix C-shell command

```
% source /usr/local/ferret_paths
```

(Note: If this command doesn't work consult your system manager, who may have placed ferret_paths in a different directory.)

The Ferret program requires access to several files and directories. These Unix paths are stored in environment variables defined by the file "ferret_paths". Your Unix account must be "made aware" of where the Ferret utilities are located. This is done by adding to the definition of your environment variable PATH the directory "$FER_DIR/bin". Unless your system manager has modified the typical setup, this will occur automatically when you execute the above command.

STEP 2 (personal customization—optional)

Execute the "cp" command below:

```
% cp $FER_DIR/bin/my_ferret_paths_template \
     $HOME/my_ferret_paths
```

Then use a text editor to customize my_ferret_paths. Instructions are inside the file.

Some of the Ferret environment variables identify files and directories that are integral to the Ferret program, but others identify files that you may maintain—your data files, GO scripts, and palette files, for example. (The environment variables that you may want to customize are discussed at the end of this section.) To assist in customizing the Ferret environment variables the template file in the "cp" command, above, has been provided. The file is self-explanatory.

STEP 3

Execute the command below interactively or add it to your .login file.

```
% setenv DISPLAY node:0.0    e.g.,  % setenv DISPLAY anorak:0.0
```

This command sets the environment variable "DISPLAY" to point to the workstation console or X-terminal where you want Ferret graphical output displayed. In the example above, graphical output is directed to the screen of workstation "anorak." The X display must be set for indexed color (a.k.a. pseudo-color); a maximum of 65K colors.

Ch9 Sec2. FILES AND ENVIRONMENT VARIABLES USED BY FERRET

.ferret—the Ferret initialization file. This optional file holds a list of Ferret commands that will be executed immediately each time Ferret is started, permitting Ferret to be tailored to individual needs and styles. The file must be located in your $HOME (login) directory. A simple way to set up such a file is to enter Ferret, enter the commands that you want executed each time you enter Ferret, exit Ferret and rename the file "ferret.jnl" to ".ferret". Thereafter, all commands in ".ferret" will be executed automatically whenever you enter Ferret.

The following environment variables are defined in the file ferret_paths:

FER_DATA—a list of directories to be searched to locate data files. Usually this list includes ".", the current directory, and $FER_DSETS/data, a directory of sample data sets provided with Ferret. Your system manager may have set this variable to include other data areas as well. This is the list of directories searched to locate netCDF files.

FER_DESCR—a list of directories to be searched to locate descriptor files. Descriptors are required by Ferret to access data sets that are in Ferret's "GT" (grids at timesteps) or "TS" (time series) formats. Usually this list includes ".", the current directory, and $FER_DSETS/descr, a directory of sample descriptors provided with Ferret.

FER_GRIDS—a list of directories to be searched to locate grid definition files. Data sets will usually have a grid definition file associated with them so that the grids on which the data are defined may be known.

FER_DIR—top directory of the Ferret distribution on your system.

FER_DSETS—directory of sample data sets provided with the Ferret distribution.

FER_PALETTE—a list of directories to be searched to locate palette files. Usually this list includes "." and $FER_DIR/ppl. Note that to assist you in choosing a good palette for your plot, there is an FAQ, How can I find a good color palette for my plot? at http://ferret.pmel.noaa.gov/Ferret/FAQ/graphics/colorpalettes.html

FER_GO—a list of directories to be searched to locate GO scripts. This list usually includes ".", $FER_DIR/go, $FER_DIR/examples (demonstrations and tutorial), and $FER_DIR/contrib (user contributions demonstrating various applications; accuracy not guaranteed).

FER_EXTERNAL_FUNCTIONS—a list of directories to be searched to locate the shared object files (.so files) for external functions. By default this list includes the location of the example functions and the functions included with the Ferret distribution.

Ch9 Sec3. MEMORY USE

Ferret indicates memory problems by issuing the error message "insufficient memory." If memory is a problem running Ferret the following suggestions may help:

1) Use the command SET MEMORY/SIZE=nnn to increase the memory cache region available to Ferret.

2) Use the command SET MODE DESPERATE to determine the threshold size of memory objects at which Ferret will break a large calculation into fragments. A smaller argument value will induce stricter memory management but at a penalty in performance.

3) Use CANCEL MEMORY whenever you are sure that the data referenced thus far by Ferret will not be referenced again. This is particularly appropriate to batch procedures that use Ferret. This eliminates any memory fragmentation that may be left by previous commands.

4) Use CANCEL MODE SEGMENTS to minimize the memory usage by graphics (on a few X-window systems this may prevent windows from being restored after they are obscured).

5) When using DEFINE VARIABLE (alias LET) avoid embedding upper and lower axis bounds within the variable definition. Ferret cannot split up large calculations along axes when the limits are fixed in the definition. For example,

```
yes? LET V2=TEMP/10
yes? PLOT/K=1:10 V2
```

is preferable to

```
yes? LET V2=TEMP[K=1:10]/10
yes? PLOT V2
```

6) Try to group together calculations that are on smaller dimensioned objects. For example, the expression VAR[i=1:100, j=1:100]*2*PI will make less efficient use of cpu and memory than the expression VAR[i=1:100, j=1:100]*(2*PI). The former multiplies each of the 10000 points of VAR by 2 and then performs a second multiplication of the 10000 result points by PI. The latter computes the scalar 2*PI and uses it only once in multiplying the 10000 points of VAR.

7) After complex plots using viewports, use CANCEL VIEWPORTS to clear graphics memory.

8) If one has SET MODE STUPID:weak_cache, then make sure that the region is fully defined (i.e., check SHOW REGION and check the region qualifiers of your command). When the region along some axis is not specified Ferret defaults to the full span of the data along that axis and is unable to optimize memory usage.

Ch9 Sec4. HARD COPY AND METAFILE TRANSLATION

Ch9 Sec4.1. Hard copy: postscript output

To obtain hard copy of plots produced by Ferret, follow these steps:

1) Within Ferret, enter the command

```
yes? SET MODE METAFILE
```

This tells Ferret to generate a graphic metafile (ANSI/ISO GKSM format) for each plot created thereafter. To stop making the metafiles type

```
yes? CANCEL MODE METAFILE
```

2) Produce each plot as you would normally. Each new plot on your screen generates an additional file named "metafile.plt.~n~" where "n" will be incremented for each metafile. Overlay commands do not produce additional metafiles. (The metafile name may be set by the SET MODE METAFILE command.)

3) After exiting from Ferret use the command Fprint.

> Note: If it is necessary to use Fprint without exiting Ferret, then issue the command ***yes? PPL CLSPLT***. This will close the current metafile. Note that neither overlays nor additional viewports can be added to the plot after the metafile has been closed.

Fprint is a script which translates metafiles generated by Ferret. It uses the program "gksm2ps" and is intended to simplify sending plots to printers, to an output file only, or to a workstation screen.

On Windows systems, the Fprint command is not available. Run the gksm2ps command directly to translate your metafile to postscript. See the next section, Metafile Translation (p. 262), for a description of gksm2ps.

The Fprint script translates metafiles to Encapsulated PostScript or X-window output. Your system manager should customize the script at your site to permit your specification of the actual printers you have as output devices. Fprint uses standard Unix command line syntax.

Fprint [-h] [-P printer || -o file_name || -X]

[-p orient] [-# n] [-l line] [-R] metafile(s)

Options

-h	displays help on your terminal.
-P printer	Routes output to named printer. Files will not be renamed by previewing. You will be prompted, however, with an option to delete each metafile after previewing. The output window size will be equivalent to the default size in Ferret (SET WINDOW/SIZE=0.7).
-o file_name	Routes output to named disk postscript file.
-X	Routes output to your workstation screen. Files will not be renamed by previewing. You will be prompted, however, with an option to delete each metafile after previewing. The output window size will be equivalent to the default size in Ferret (SET WINDOW/SIZE=0.7).
-p orient	The page orientation option determines whether the plot will be placed on the page in landscape format, with the horizontal side longer than the vertical, or portrait, with the vertical side longer. Valid option values are "landscape" and "portrait". The default behavior is to orient the plot to best fit the page.
-# n	Specifies number of copies (n).
-l line	This option lets you specify line styles. Valid options are "ps" and "cps". "ps" uses dot-dashed line types; "cps" uses colored lines. The default is "ps" for monochrome printers and "cps" for color printers.
-R	Turns off the default behavior of the metafile translator to append a date stamp to metafile names when they are sent to a printer or a disk file. The default action is intended to distinguish metafiles that have been printed out; this option keeps the metafile names unmodified.
-C	Output a CMYK postscript file; default is RGB. See the FAQ on CMYK color, How can I get CMYK format for postscript files?

Examples

% Fprint metafile.plt

renders "metafile.plt" on the default printer identified by the environment variable PRINTER.

% Fprint -P myprinter -R metafile.plt*

renders all versions of "metafile.plt" on printer myprinter. Does not date stamp them.

% Fprint -o my_plot.ps metafile.plt.~1~

writes plot "metafile.plt.~1~" to a postscript file named "my_plot.ps".

Ch9 Sec4.2. Metafile translation

The command "gksm2ps" allows you to control the translation of the device-independent metafiles made by Ferret into device-specific output files. "gksm2ps" was written by Larry Oolman at the University of Wyoming and modified at NOAA/PMEL for use with Ferret. The "gksm2ps" command uses standard Unix command line syntax. See usage hints provided by the -h option.

```
gksm2ps [-h] [-p landscape||portrait] [-l ps||cps] [-d cps||phaser] \
    [-X || -o <ps_output_file>] [-R] [-a] [-g WxH+X+Y] file(s)
```

Options

-h	prints help message.
-p orient	The page orientation option determines whether the plot will be placed on the page in landscape format, with the horizontal side longer than the vertical, or portrait, with the vertical side longer. The default is to orient the plot to best fit the page.
-l line	This option permits specification of line styles in the hardcopy plot. Valid options are "ps" (the default) and "cps". "ps" renders lines as solid and dot-dashed and is suited for monochrome printers. "cps" renders lines in color.
-d devtype	The target device type of the translator. If the -d option is omitted and output is to a file gksm2ps will use devtype "ps". Valid devtype values: Cps – color PostScript phaser – Tektronix Phaser PX. The phaser is a PostScript printer, but it uses transfer sheets that reduce the usable page size.
-X	Sends the output to your X-window for preview.
-o ofile	The output will be directed to the file "ofile." Omit both this and the device type option when directing output to your workstation screen with -X. If neither -o nor -X is specified, gksm2ps creates a postscript file in the current directory called "gksm2ps_output.ps".
-a	Makes the plot the size of the original plot as specified in PPLUS inches (absolute size), rather than fitting the plot to the page (the default behavior).
-g WxH+X+Y	The -g option (-g WxH+X+Y) provides detailed control over the size, position, and aspect ratio of the plot on the printed page. The arguments W, H, X, and Y are given in units of points (1/72 of an inch).

Options

Normally when using this option you will want to specify an identical value for both W and H—the size (in points) you want the longer dimension of the plot to be. Unequal values of W and H will alter the aspect ratio of the plot relative to its appearance on your workstation screen.

The X and Y values are the offset of the lower left corner of the plot from the lower left corner of the page. If you want your plot's longer side to be 5 inches long, 3 inches right from the corner, and 2 inches up, for example, specify

> lpr my_plot.ps

-R Turns off the default behavior of the metafile translator to append a date stamp to metafile names when they are sent to a printer or a disk file. The default action is intended to distinguish metafiles that have been printed out; this option keeps the metafile names unmodified.

If the user does not specify an output option (-o or -X) gksm2ps translates the metafile and produces a PostScript file called gksm2ps_output.ps. After translation by gksm2ps, metafiles are renamed with a date stamp unless -R was specified. To get hard copy printed, the output PostScript file needs to be sent to the appropriate printer.

Ch9 Sec4.3. Hard Copy: gif files

To create gif graphics output, execute Ferret commands to produce a plot. Then use the command FRAME/FILE=filename.gif to write a gif-formatted file. See the section on the FRAME command (p. 358).

Or, start Ferret in gif mode,with the -gif command-line switch, to run Ferret without X server software. Execute the FRAME command to save the plot as a gif image.

```
< ferret -gif
yes? (commands that generate a plot...)
yes? FRAME/FILE=picture.gif
```

See p. 6for a complte description.

Ch9 Sec5. OUTPUT FILE NAMING

Ferret uses a file naming scheme to differentiate successive graphic metafiles and journal files. The scheme is styled after the gnu (Free Software Foundation) emacs editor. The scheme appends numbers to the end of the file name as in the following examples:

```
Metafile.plt.~2~
metafile.plt.~12~
metafile.plt
```

The third example, "metafile.plt" with no suffix appended, is the most recent file. When the next successive file is created, this file will have the suffix ".~nnn~" appended to its name. "nnn" is the current highest file suffix number plus one.

Two Unix tools are provided to assist with managing multiple file suffix numbers:

Fpurge	removes all but the current version of the named file (that is, all but the most recent). Example: ***% Fpurge ferret.jnl***
Fsort	sorts the versions of a file into increasing numerical order Example: ***% Fprint 'Fsort metafile.plt*'***

See the introductory chapter, section "Unix tools," p. 29, for further information.

Ch9 Sec6. INPUT FILE NAMING

There are several Ferret commands that use filenames. These include:

```
GO filename
SET DATA filename
LIST/FILE=filename  (do not use relative versions (below) with LIST)
USER/FILE=filename
SET MODE META filename
SET MODE JOURNAL filename
SET MODE PPLLIST filename
```

The filename specified can be just the filename itself, or it can include the path to the file. For example:

GO ferret.jnl or ***GO "/home/disk1/jnl_files/far_side.jnl"***

Note that if the path is specified as part of the filename, the entire name must be enclosed in quotation marks.

Ch9 Sec6.1. Relative version numbers

Under some circumstances (see the GO command, p. 359) a special syntax called "relative version numbers" will apply. If a filename has a version value of zero or less its value is interpreted relative to the current highest version number.

For example, if the current directory contains the files

```
ferret.jnl   ferret.jnl.~1~   ferret.jnl.~2~   ...  ferret.jnl.~99~
```

then the filename ***ferret.jnl.~0~*** refers to ***ferret.jnl*** and the filename ***ferret.jnl.~-1~*** refers to ***ferret.jnl.~99~***.

The syntax for relative version numbers is quite flexible. For example, if the desired file is ***ferret.jnl.~99~***, both of the following are valid:

```
yes? GO ferret.jnl.~-1~     or     yes? GO ferret.jnl~-1
```

Chapter 10: CONVERTING TO NetCDF

Ch10 Sec1. OVERVIEW

The Network Common Data Format (netCDF) is an interface to a library of data access routines for storing and retrieving scientific data. NetCDF allows the creation of data sets that are self-describing and network-transparent. NetCDF was created under contract with the Division of Atmospheric Sciences of the National Scientific Foundation and is available from the Unidata Program Center in Boulder, Colorado (on Internet: www.unidata.ucar.edu).

This chapter provides directions for creating netCDF data files. In addition to the documentation provided here, refer to the NetCDF User's Guide, published by Unidata Program Center, for further guidance. A user who uses and creates netCDF files within the Ferret environment needs no additional software.

NetCDF is a very flexible standard. In most cases there are multiple styles or profiles that could be used to encode data into netCDF. To resolve the ambiguities inherent in this multiplicity communities of users have banded together to develop profiles—documents that provide more detail on how data should be encoded into netCDF. Ferret adheres to the COARDS standard. The full standard is available through the Ferret home page on the World Wide Web,

http://www.ferret.noaa.gov/noaa_coop/coop_cdf_profile.html

The COARDS standards are a subset of the CF standard for netCDF files. Ferret does not implement all of the CF standard, but conforms to that standard for the portions that are implemented. The document document defining the CF standard may be found at

http://www.cgd.ucar.edu/cms/eaton/cf-metadata/CF-1.0.html

Note that Ferret variables may have only four dimensions.

Ch10 Sec2. SIMPLE CONVERSIONS USING FERRET

In straightforward conversion operations where ASCII or unformatted binary data files are already readable by Ferret, the conversion to direct access, self-describing netCDF formatted data can be accomplished by Ferret itself. The following set of examples illustrates these procedures:

Example 1

Consider an ASCII file uv.data, with two variables, u and v, defined on a grid 360 by 180. The following set of commands will properly read in u and v and convert them to a netCDF formatted data set:

```
yes? DEFINE AXIS/x=1:360:1/units=degrees xaxis
yes? DEFINE AXIS/y=1:180:1/units=degrees yaxis
yes? DEFINE GRID/x=xaxis/y=yaxis uv_grid
yes? FILE/GRID=uv_grid/BAD=-999/VAR="u,v"  uv.data
yes? SET VARIABLE/TITLE="zonal velocity" u
yes? SAVE/FILE=uv.cdf u,v
```

See command DEFINE AXIS in the Commands Reference (p. 338). See the chapter "Grids and Regions" (p. 145) for setting up formatted latitude, longitude and time axes.

Example 2

Consider now two separate ASCII files, u.data and v.data, defined on a grid 360 by 180. The following set of commands will properly read in u and v and convert them to a single netCDF formatted data set:

```
yes? DEF AXIS/x=1:360:1/units=degrees xaxis
yes? DEF AXIS/y=1:180:1/units=degrees yaxis
yes? DEF GRID/x=xaxis/y=yaxis uv_grid
yes? FILE/GRID=uv_grid/BAD=-999/VAR=u u.data
yes? FILE/GRID=uv_grid/BAD=-999/VAR=v v.data
yes? SAVE/FILE=uv2.cdf u[D=1]
yes? SAVE/APPEND/FILE=uv2.cdf v[D=2]
```

Example 3—multiple time steps

Consider 12 ASCII files, uv.data1 to uv.data12, each defined on the same grid as above but each representing a successive time step. The following set of commands illustrates how to save these data into a single netCDF data set (time series):

```
yes? DEF AXIS/x=1:360:1 xaxis
yes? DEF AXIS/y=1:180:1 yaxis
yes? DEF AXIS/t=1:1:1 taxis1
yes? DEF GRID/x=xaxis/y=yaxis/t=taxis1 uv_grid1
yes? FILE/GRID=uv_grid1/BAD=-999/VAR="u,v" uv.data1
yes? SAVE/FILE=uv1_12t.cdf u,v
yes? CANCEL DATA uv.data1
yes? DEF AXIS/t=2:2:1 taxis1
yes? FILE/GRID=uv_grid1/BAD=-999/VAR="u,v" uv.data2
yes? SAVE/APPEND/FILE=uv1_12t.cdf u,v
. . .
```

and so on, redefining the time axis to be 3:3:1, 4:4:1, ... each time a new file is set.

When the input data is in netCDF files, please see the following FAQ on using data from a set of input files. Note particularly the example showing how to add time information to the variable when the grid of the input data has no time axis.

FAQ: How can I use data as a time series when it exists in multiple files? http://www.ferret.noaa.gov/Ferret/FAQ/data_management/multi_dataset.html

Example 4—multiple slabs

The procedure used in example 3, above, is possible because netCDF files can be extended along the time axis. In order to append multiple levels (Z axis), the netCDF file must first be created including all of its vertical levels (the levels initially are filled with a missing data flag).

Consider 5 ASCII files, uv.data1 to uv.data5, each defined on the same grid as above but each representing a successive vertical level. Note that the output grid has an axis containing all the Z levels that the file will contain (and that the other Z axes, zaxis1, zaxis2, ... are defined only for the purpose of reading the data in). The following set of commands illustrates how to save these data into a single netCDF data set:

```
yes? DEF AXIS/x=1:360:1  xaxis
yes? DEF AXIS/y=1:180:1  yaxis
yes? DEF AXIS/Z=0:100:25/DEPTH  zaxis
yes? DEF GRID/X=xaxis/Y=yaxis/Z=zaxis uv_grid
yes? DEF AXIS/Z=0:0:1  zaxis1
yes? DEF GRID/LIKE=uv_grid/Z=zaxis1  uv_grid1

yes? FILE/GRID=uv_grid1/BAD=-999/VAR="u,v"  uv.data1
yes? LET/TITLE="My U data"  u1 = u[G=uv_grid]
yes? LET/TITLE="My V data"  v1 = v[G=uv_grid]
yes? SAVE/FILE=uv1_5z.cdf/KLIMITS=1:5/K=1  u1, v1

yes? CANCEL DATA uv.data1
yes? DEF AXIS/Z=25:25:1  zaxis1
yes? FILE/GRID=uv_grid1/BAD=-999/VAR="u,v"  uv.data2
yes? SAVE/FILE=uv1_5z.cdf/K=2/APPEND  u1,v1
. . .
yes? CANCEL DATA/ALL  ! Cancel definitions before using new file
yes? CANCEL VAR/ALL
yes? USE uv1_5z.cdf
```

The netCDF utilities "ncdump" and "ncgen" can also be combined with a text editor to make final refinements to the netCDF files created by SAVE. (These utilities can be obtained from http://www.unidata.ucar.edu/) Here is a simple example that removes all "history" attributes from a netCDF file using pipes and the Unix "grep" utility:

```
% ncdump old_file.cdf | grep -v history | ncgen -o new_file.cdf
```

In addition, the toolset "NetCDF Operator", or NCO, contains a number of operators for manipulating netCDF files. This software can be found at http://nco.sourceforge.net/

Ch10 Sec3. WRITING A CONVERSION PROGRAM

There are three steps required to convert data to netCDF if your data is not already readable by Ferret:

1. Create a CDL (the ASCII NetCDF Description Language) file that describes the axes, grids, and variables of the desired output data set. Note: Ferret itself often provides the simplest way to create the CDL file (see the following section).

2. Convert this CDL file into a netCDF file with the ncgen utility.

3. Create a program that will read your particular data and insert them into the netCDF file. The ncgen utility will create most of the FORTRAN or C code needed for this task.

The file converting_to_netcdf.f which is located in the Ferret documentation directory ($FER_DIR/doc) contains a complete description and example of these three steps. The remainder of this section provides further details.

Ch10 Sec3.1. Creating a CDL file with Ferret

Suppose that we wish to create a CDL file to describe a data set entitled "My Global Data" which contains variables u and v in cm/sec on a 5×5 degree global lat/long grid. The following commands would achieve the goal with Ferret doing the majority of the work:

- From Ferret issue the commands

```
DEFINE AXIS/X=2.5E:2.5W:5/UNITS=degrees xlong
DEFINE AXIS/Y=87.5S:87.5N:5/UNITS=degrees ylat
DEFINE GRID/X=xlong/Y=ylat my_grid
LET shape_2d =  x[G=my_grid]+y[G=my_grid]
LET U = 1/0*SHAPE_2D
LET V = 1/0*SHAPE_2D
SET VARIABLE/TITLE="Zonal Velocity"/UNITS="cm/sec" u
SET VARIABLE/TITLE="Meridional Velocity"/UNITS="cm/sec" v
SAVE/FILE=my_file.cdf/TITLE="My Global Data" u,v
QUIT
```

- From Unix issue the command

```
ncdump -c my_file.cdf > my_file.cdl
```

The resulting file my_file.cdl is ready to use or to make final modifications to with an editor.

Ch10 Sec3.2. The CDL file

A CDL file consists of three sections: Dimensions, Variables, and Data. All of the following text in `small Courier font` constitutes a real CDL file; it can be copied verbatim and used to generate a netCDF file. The full text of this file is included with the Ferret distribution as $FER_DIR/doc/converting_to_netcdf.basic.

```
netcdf converting_to_netcdf.basic {
```

Ch10 Sec3.2.1. Dimensions

In this section, the sizes of the grid dimensions are specified. One of these dimensions can be of "unlimited" size (i.e., it can grow).

```
Dimensions:

      lon   = 160 ;      //  longitude
      lat   = 100 ;      //  latitude
      depth = 27 ;       //  depth
      time  = unlimited ;
```

These are essentially parameter statements that assign certain numbers that will be used in the Variables section to define axes and variable dimensions. The "//" is the CDL comment syntax.

The dimension variables are available to you in Ferret commands as pseudo-variables. For example, to use the "depth" dimension variable from the above example, you can say:

```
yes? let valz = z[gz=depth]
yes? let offset = valz + a
```

See the next section (p. 273) about axes for more on dimension variables.

Ch10 Sec3.2.2. Variables

Variables, variable attributes, axes, axis attributes, and global attributes are specified.

```
variables:

     float temp(time, depth, lat, lon) ;
           temp: long_name = "TEMPERATURE" ;
           temp: units     = "deg. C" ;
           temp: _FillValue = 1E34 ;
     float salt(time, depth, lat, lon) ;
           salt: long_name = "(SALINITY(ppt) - 35) /1000" ;
           salt: units     = "frac. by wt. less .035" ;
           salt: _FillValue = -999. ;
```

The declaration "float" indicates that the variable is to be stored as single precision, floating point (32-bit IEEE representation). The declarations "long" (32-bit integer), "short" (16-bit integer), "byte" (8-bit integer) and "double" (64-bit IEEE floating point) are also supported by Ferret. Note that although these data types may result in smaller files, they will not affect Ferret's memory usage, as all variables are converted to "float" internally as they are read by Ferret.

Variable names in netCDF files should follow the same guidelines as Ferret variable names:

- case insensitive (avoid names that are identical apart from case)
- 1 to 24 characters (letters, digits, $ and _) beginning with a letter

- avoid reserved names (I, J, K, L, X, Y, Z, T, XBOX, ...)

See p. 60 for how to handle invalid variable names that are already in a netCDF file.

The _FillValue attribute informs Ferret that any data value matching this value is a missing (invalid) data point. For example, an ocean data set may label land locations with a value such as 1E34. By identifying 1E34 as a fill value, Ferret knows to ignore points matching this value. The attribute "missing_value" is similar to "_FillValue" when reading data; but "_FillValue" also specifies a value to be inserted into unspecified regions during file creation. You may specify two distinct flags for invalid data in the same variable by using "_FillValue" and "missing_value" together.

Scale and offset values may be specified by the scale_factor and add_offset attributes.

Ferret variables may have from 1 to 4 dimensions. If any of the axes have the special interpretations of: 1) latitude, 2) longitude, 3) depth, or 4) time (date), then the relative order of those axes in the CDL variable declaration must be T, then Z, then Y, and then X, as above. Any of these special axes can be omitted and other axes (for example, an axis called "distance") may be inserted between them.

```
axis definitions:

     double lon(lon) ;
            lon: units = "degrees";
     double lat(lat) ;
            lat: units = "degrees";
     double depth(depth) ;
            depth: units = "meters";
     double time(time) ;
            time: units = "seconds since 1972-01-01";
```

Axes, also known as coordinate variables, are distinguished from other 1-dimensional netCDF variables by their variable names being identical to their dimension names. Special axis interpretations are inferred by Ferret through a variety of "clues."

The direction of the axis may be specified by the attribute AXIS or CARTESIAN_AXIS. Files written by Ferret (as of version 5.5) include the AXIS attribute for coordinate variables.

```
lon: axis="X";
```

Date/time axes are inferred by units of "years," "days," "hours," "minutes," or "seconds," or by axis names "time," "date," or "t" (case-insensitive). Calendar date formatting requires the "units" attribute to be formatted with both a valid time unit and "since yyyy-mm-dd".

Vertical axes are identified by axis names containing the strings "depth", "elev", or "z", or by units of "millibars." Depth axes are positive downward by default. The attribute positive= "down" can also be used to create a downward-pointing Z axis. The positive= attribute may be used on any axis, with the values positive= "down" or positive="up", however positive="down" is applied only to Z axes and is ignored otherwise.

Latitude axes are inferred by units of "degrees" or "latitude" with axis names containing the strings "lat" or "y". Longitude axes are inferred by units of "degrees" or "longitude" with axis names containing the strings "lon" or "x".

Axis direction is determined by Ferret as in this order:

1) AXIS attribute
2) CARTESIAN_AXIS attribute
3) positive="down", indicating a z axis
4) Axis units
5) Axis name

Once the direction is determined, other conflicting information is ignored. Thus if an axis has the attribute AXIS="Y" it is a Y axis, even though its units might be "days" or its name might be "longitude".

Axes are either netCDF coordinate variables or are synthesized (as simple indices 1, 2, 3, ...) if coordinate definitions are missing for a variable. The axes of a variable are available as "pseudo-variables" using the syntax X[g=var], where "var" is the name of the netCDF variable, and similarly for the Y,Z, and T axes. When the data set is cancelled the associated axes are cancelled, too. The exception is that axes will be retained if they are in use in a DEFINE GRID definition -- and they will be erased from memory at the time all grids using them are cancelled.

Some files contain axis definitions (coordinate variables) without associated variables. Like all axes they are visible with the SHOW AXIS command. To obtain the values of those coordinate variables as Ferret pseudo-variables use the syntax X[gx=axname], where axname is the name of the coordinate variable (also the netCDF dimension name) and likewise for Y,Z, and T axes. Note that when the data set is cancelled, axis definitions of this variety are retained -- unlike axes that are used in variables.

Global attributes, or attributes that apply to the entire data set, can be specified as well.

```
global attributes:
     :title = "NetCDF Example";
     :message =   "This message will be displayed when the CDF file is
                   opened by Ferret";
     :history =   "Documentation on the origins and evolution of this
data
                   set or variable";
```

Ch10 Sec3.2.3. Data

In this section, values are assigned to grid coordinates and to the variables of the file. Below are 100 latitude coordinates entered (in degrees) into the variable axis "lat", 160 longitude coordinates in "lon", and 27 depth coordinates (in meters) in "depth". Longitude coordinates must be

specified with 0 at Greenwich, continuous across the dateline, with positive eastward (modulo 360).

```
Data:

lat=
-28.8360729218,-26.5299491882,-24.2880744934,-22.1501560211,-20.15135765
0,
-18.3207626343,-16.6801033020,-15.2428140640,-14.0134353638,-12.98742485
0,
-12.1513509750,-11.4834814072,-10.9547319412,-10.5299386978,-10.16939353
9,
-9.8333206177,-9.4999876022,-9.1666536331,-8.8333196640,-8.4999856949,
-8.1666526794,-7.8333187103,-7.4999847412,-7.1666512489,-6.8333182335,
-6.4999852180,-6.1666517258,-5.8333182335,-5.4999852180,-5.1666517258,
-4.8333187103,-4.4999852180,-4.1666517258,-3.8333187103,-3.4999852180,
-3.1666517258,-2.8333184719,-2.4999852180,-2.1666519642,-1.8333185911,
-1.4999852180,-1.1666518450,-0.8333183527,-0.4999849498,-0.1666515470,
0.1666818559,0.5000152588,0.8333486915,1.1666821241,1.5000154972,
1.8333489895,2.1666824818,2.5000159740,2.8333494663,3.1666829586,
3.5000162125,3.8333497047,4.1666831970,4.5000162125,4.8333497047,
5.1666831970,5.5000162125,5.8333497047,6.1666827202,6.5000162125,
6.8333497047,7.1666827202,7.5000166893,7.8333501816,8.1666841507,
8.5000181198,8.8333511353,9.1666851044,9.5000190735,9.8333530426,
10.1679363251,10.5137376785,10.8892869949,11.3138961792,11.8060989380,
12.3833675385,13.0618314743,13.8560228348,14.7786512375,15.8403968811,
17.0497493744,18.4128704071,19.9334945679,21.6128730774,23.4497566223,
25.4404067993,27.5786647797,29.8560409546,32.2618522644,34.7833900452,
37.4061241150,40.1139259338,42.8893203735,45.7137718201,48.5679702759;
lon=
130.5,131.5,132.5,133.5,134.5,135.5,136.5,137.5,138.5,139.5,140.5,
141.5,142.5,143.5,144.5,145.5,146.5,147.5,148.5,149.5,150.5,151.5,
152.5,153.5,154.5,155.5,156.5,157.5,158.5,159.5,160.5,161.5,162.5,
163.5,164.5,165.5,166.5,167.5,168.5,169.5,170.5,171.5,172.5,173.5,
174.5,175.5,176.5,177.5,178.5,179.5,180.5,181.5,182.5,183.5,184.5,
185.5,186.5,187.5,188.5,189.5,190.5,191.5,192.5,193.5,194.5,195.5,
196.5,197.5,198.5,199.5,200.5,201.5,202.5,203.5,204.5,205.5,206.5,
207.5,208.5,209.5,210.5,211.5,212.5,213.5,214.5,215.5,216.5,217.5,
218.5,219.5,220.5,221.5,222.5,223.5,224.5,225.5,226.5,227.5,228.5,
229.5,230.5,231.5,232.5,233.5,234.5,235.5,236.5,237.5,238.5,239.5,
240.5,241.5,242.5,243.5,244.5,245.5,246.5,247.5,248.5,249.5,250.5,
251.5,252.5,253.5,254.5,255.5,256.5,257.5,258.5,259.5,260.5,261.5,
262.5,263.5,264.5,265.5,266.5,267.5,268.5,269.5,270.5,271.5,272.5,
273.5,274.5,275.5,276.5,277.5,278.5,279.5,280.5,281.5,282.5,283.5,
284.5,285.5,286.5,287.5,288.5,289.5;
depth=
5.0,15.0,25.0,35.0,45.0,55.0,65.0,75.0,85.0,95.0,106.25,120.0,136.25,
155.0,177.5,205.0,240.0,288.5,362.5,483.5,680.0,979.5,1395.5,1916.0,
2524.0,3174.0,3824.0;  }
```

To use this CDL file type:

```
% ncgen -o my_data.cdf converting_to_netcdf.basic
```

This will create a file called "my_data.cdf" to which data can be directed (see next section).

Another netCDF command, "ncdump", can be used to generate a CDL file from an existing netCDF file. The command

```
% ncdump -h my_data.cdf
```

will list the CDL representation of a preexisting my_data.cdf without the Data section, while

```
% ncdump my_data.cdf
```

will list the CDL file with the Data section. The command

```
% ncdump -c my_data.cdf
```

will create a CDL file in which only coordinate variables are included in the Data section. The listed output can be redirected to a file as in the command

```
% ncdump -c my_data.cdf > my_data.cdl
```

Ch10 Sec3.3. Standardized NetCDF attributes

A document detailing the COARDS netCDF conventions to which the Ferret program adheres are available on line through the Ferret home page on the World Wide Web, at
http://www.ferret.noaa.gov/noaa_coop/ coop_cdf_profile.html and at

http://www.unidata.ucar.edu/packages/netcdf/conventions.html

The following are the attributes most commonly used with Ferret. They are described in greater detail in the reference document named above.

Global Attributes
:title = "my data set title"
:history = "general background information"

Data Variable Attributes
long_name = "title of my variable"
units = "units for this variable"
_FillValue = missing value flag
missing_value = alternative missing value flag
scale_factor = (optional) the data are to be multiplied by this factor
add_offset = (optional) this number is to be added to the data

Special Coordinate Variable Attributes
time_axis:units = "seconds since 1992-10-8 15:15:42.5 -6:00"; // example
y_axis:units = "degrees_north"
x_axis:units = "degrees_east"
z_axis:positive = "down"; // to indicate preferred plotting orientation
my_axis:point_spacing = "even"; // improved performance if present

axis: "X", "Y", "Z", or "T" indicating the direction of the axis.

See the SAVE command to write to a netCDF file. (p.392). Note that when using Ferret to output into netCDF files that Ferret did not itself create, the results may not be entirely as expected. Case-sensitivity of names is one aspect of this. Since Ferret is (by default) case insensitive and netCDF files are case-sensitive writing into a "foreign" file may result in duplicated entities in the file which differ only in case.

Ch10 Sec3.4. Directing data to a CDF file

The following is an example program which can be used on-line to convert existing data sets into netCDF files. It also should provide guidance on sending data generated by numerical models directly to netCDF files. This program assumes you have created the netCDF file as described in the previous section. It is included in the distribution as $FER_DIR/doc/converting_to_netcdf.f.

```
   program converting_to_netcdf

c written by Dan Trueman
c updated 4/94 *sh*

c This program provides a model for converting a data set to NetCDF.
c The basic strategy used in this program is to open an existing NetCDF
c file, query the file for the ID's of the variables it contains, and
c then write the data to those variables.

c The output NetCDF file must be created **before** this program is run.
c The simplest way to do this is to cd to your scratch directory and
c    % cp $FER_DIR/doc/converting_to_netcdf.basic converting_to_netcdf.cdl
c and then edit converting_to_netcdf.cdl (an ASCII file) to describe YOUR
c data set. If your data set requires unequally spaced axes,
climatological c time axes, staggered grids, etc. then
converting_to_netcdf.supplement may c be a better starting point then the
"basic" file used above.
c After you edit converting_to_netcdf.cdl then create the NetCDF file with
c the command
c    % ncgen -o converting_to_netcdf.cdf converting_to_netcdf.cdl

c Now we will read in **your** data (gridded oceanic temperature and
c salt in this example) and write it out into the NetCDF file
c converting_to_netcdf.cdf. Note that the axis coordinates can be written
c out exactly the same methodology - including time step values (as
below).
*********************************************************************
c An alternative to modifying this program is to use the command:

c     ncgen -f converting_to_netcdf.cdl

c This will create a large source code to which select lines can
c be added to write out your data.
*********************************************************************
c To compile and link converting_to_netcdf.f, use:

c     f77 -o converting_to_netcdf converting_to_netcdf.f -lnetcdf

*********************************************************************
```

```
c include file necessary for NetCDF

   include 'netcdf.inc'    ! may be found in $FER_DIR/fmt/cmn
**************************************************************************
c parameters relevant to the data being read in
c THESE NORMALLY MATCH THE DIMENSIONS IN THE CDL FILE
c (except nt which may be "unlimited")

   integer imt, jmt, km, nt, lnew, inlun
   parameter (imt=160, jmt=100, km=27, nt=5)

c imt is longitude, jmt latitude, km depth, and nt number of time steps
**************************************************************************
c variable declaration

   real temp(imt,jmt,km),salt(imt,jmt,km),time_step

   integer cdfid, rcode
c           ** cdfid = id number for the NetCDF file my_data.cdf
c     ** rcode = error id number

   integer tid, sid, timeaxid
c           ** tid = variable id number for temperature
c           ** sid = variable id number for salt
c           ** timeaxid = variable id for the time axis
   integer itime
c           ** itime = index for do loop
**************************************************************************
c dimension corner and step for defining size of gridded data

   integer corner(4)
   integer step(4)

c corner and step are used to define the size of the gridded data
c to be written out. Since temp and salt are four dimensional arrays,
c corner and step must be four dimensions as well. In each output
c to my_data.cdf within the do loop, the entire array of data (160 long.
c pts, 100 lat. pts., 27 depth pts.) will be written for one time step.
c Corner tells NetCDF where to start, and step indicates how many steps
c in each dimension to take.

         data corner/1, 1, 1, -1/      ! -1 is arbitrary; the time value

! of corner will be initialized
                                       ! within the do loop.

         data step/imt, jmt, km, 1/    ! NOT /1, km, jmt, imt/

c ***NOTE*** Since Fortran and C access data differently, the order of
c the variables in the Fortran code must be opposite that in the CDL
c file. In Fortran, the first index varies fastest while in C, the
c last index varies fastest.
**************************************************************************
c initialize cdfid by using ncopn

   cdfid = ncopn('converting_to_netcdf.cdf', ncwrite, rcode)
   if (rcode.ne.ncnoerr) stop 'error with ncopn'

**************************************************************************
c get variable id's by using ncvid
c THE VARIABLE NAMES MUST MATCH THE CDL FILE (case sensitive)

   tid = ncvid(cdfid, 'temp', rcode)
   if (rcode.ne.ncnoerr) stop 'error with tid'
```

```
   sid = ncvid(cdfid, 'salt', rcode)
   if (rcode.ne.ncnoerr) stop 'error with sid'
   timeaxid = ncvid(cdfid, 'time', rcode)
   if (rcode.ne.ncnoerr) stop 'error with timeaxid'
*******************************************************************
c this is a good place to open your input data file
   ! OPEN (FILE=my_data.dat,STATUS='OLD')
*******************************************************************
c begin do loop. Each step will read in one time step of data
c and then write it out to my_data.cdf.


   do 10 itime = 1, nt

   corner(4) = itime             ! initialize time step in corner
   time_step = float(itime)      ! or you may read this from your file

* insert your data reading routine here
!     CALL READ_MY_DATA(temp,salt)  ! you write this

   write (6,*) 'writing time step: ',itime, time_step  ! diagnostic output

   call ncvpt(cdfid,tid,corner,step,temp(1,1,1),rcode) ! write data to
   if (rcode.ne.ncnoerr) stop 'error with t-put'
   call ncvpt(cdfid,sid,corner,step,salt(1,1,1),rcode) ! my_data.cdf with
   if (rcode.ne.ncnoerr) stop 'error with s-put'
   call ncvpt1(cdfid,timeaxid,itime,time_step,rcode)   ! ncvpt
   if (rcode.ne.ncnoerr) stop 'error with timax-put'

c ncvpt1 writes a single data point to the specified location within
c timeaxid. The itime argument in ncvpt1 specifies the location within
c time to write.
c float(itime) is used (rather than simply itime) so the type matches the
c type of time declared in the CDL file.

10    continue
*******************************************************************
c close my_data.cdf using ncclos
   call ncclos(cdfid, rcode)
   if (rcode.ne.ncnoerr) stop 'error with ncclos'
*******************************************************************
   stop
   end
```

Ch10 Sec3.5. Advanced NetCDF procedures

This section describes:

1. Setting up a CDL file capable of handling data on staggered grids.
2. Defining coordinate systems such that the data in the netCDF file may be regarded as hyperslabs of larger coordinate spaces.
3. Defining boundaries between unevenly spaced axis coordinates (used in numerical integrations).
4. Setting up "modulo" axes such as climatological time and longitude.
5. Converting dates into numerical data appropriate for a netCDF time axis.

The final section of this chapter contains the text of the CDL file for the example we use throughout this section.

In this sample data set, we will consider wind, salt, and velocity calculated using a staggered-grid, finite-difference technique. The wind data is limited to the surface layer of the ocean (i.e., normal to the depth axis). We will also consider the salt data to be limited to a narrow slab from 139E to 90W (I=10 to 140), 32.5N to 34.9N (J=80 to 82), and for all depth and time values.

Ch10 Sec3.5.1. Staggered grid

Ferret permits each variable of a netCDF file to be defined on distinct axes and grids. Staggered grids are a straightforward application of this principle. Dimensions for each grid axis must be defined, the axes themselves must be defined (in Variables), and the coordinate values for each axis must be initialized (in Data). In the case of the example we use throughout this and the next section, there are two grids—a wind grid, and a velocity grid; slon, slat and sdepth are defined for the wind grid, and ulon, ulat, and wdepth for the velocity grid. The variables are then given dimensions to place them in their proper grids (i.e., wind(time, sdepth, slat, slon)).

Ch10 Sec3.5.2. Hyperslabs

There are a number of steps required to set up a netCDF data set that represents a hyperslab of data from a larger grid definition (a parent grid).

1. Define a dimension named "grid_definition." This dimension should be set equal to 1.

2. Define parent grids in Variables with the argument "grid_definition".

```
char wind_grid(grid_definition) ;
char salt_grid(grid_definition) ;
```

3. Define the 4 axes of the parent grids using the "axes" attribute.

```
wind_grid: axes = "slon slat normal time" ;
salt_grid: axes = "slon slat sdepth time" ;
```

The arguments are always a list of four axis names. Note that the order of arguments is opposite that in the variable declaration. The argument "normal" indicates that wind_grid is normal to the depth axis.

4. Define the variables that are hyperslabs of these grids with the proper dimensions.

```
float wind(time, slat, slon) ;
float salt(time, sdepth, slat80_82, slon10_140) ;
```

where the dimension slat80_82 = 3 and slon10_140 = 131. Optionally, these axes may be defined themselves with the attribute "child_axis".

```
float slat80_82(slat80_82) ;
slat80_82: child_axis = " " ;
```

These "child axes" need not be initialized in data, nor do edges need to be defined for them; Ferret will retrieve this information from the parent axis definitions. However, it is recommended that they be initialized to accommodate other software that may not recognize parent grids.

5. Use the "parent_grid" variable attribute to point to the parent grid.

```
wind: parent_grid = "wind_grid"
salt: parent_grid = "salt_grid"
```

6. Also, as a variable attribute, define the index range of interest within the parent grid.

```
wind: slab_min_index = 1s, 1s, 1s, 0s ;
wind: slab_max_index = 160s, 100s, 1s, 0s ;
salt: slab_min_index = 10s, 80s, 1s, 0s ;
salt: slab_max_index = 140s, 82s, 27s, 0s ;
```

The "s" after each integer indicates a "short" 16-bit integer rather than the default "long" 32-bit integer. If an axis dimension is designated as "unlimited" then the index bounds for this axis must be designated as "0s".

These attributes will effectively locate the wind and salt data within the parent grid.

Ch10 Sec3.5.3. Unevenly spaced coordinates

For coordinate axes with uneven spacing, the boundaries between each coordinate can be indicated by pointing to an additional axis that contains the locations of the boundaries. This can be done in one of two ways: The axis edges may be listed, N+1 edges for an axis of N coordinaes. Or cell bounds may be listed, with the lower and upper bound of each cell for a list of N*2 bounds. (The bounds attribute was implemented in Ferret version 5.70.)

If edges or bounds are not explicitly defined for an unevenly spaced axis, the midpoint between coordinates is assumed by default.

To define edges:

1. Define a dimension one larger than the coordinate axis. For the sdepth axis, with 27 coordinates, define:

```
sdepth_edges = 28 ;
```

2. Define an axis called sdepth_edges.
3. Initialize this axis with the desired boundaries (in Data).
4. As an attribute of the main axis, point to edges list:

```
sdepth: edges = "sdepth_edges" ;
```

To define bounds

1. Define a dimension twice as large as the coordinate axis. For the sdepth axis, with 27 coordinates, define:

```
sdepth_bnds = 54;
```

2. Define an axis called sdepth_bnds.
3. Initialize this axis with the desired boundaries (in Data). The ordering is box_lo_1, box_hi_1, box_lo_2, box_hi_2, ...
4. For a valid Ferret axis, the low bound of each cell must equal the high bound of the previous cell.
5. As an attribute of the main axis, point to boundslist:

```
sdepth: bounds = "sdepth_bnds" ;
```

Ch10 Sec3.5.4. Evenly spaced coordinates (long axes)

If the coordinate axes are evenly spaced, the attribute "point spacing" should be used:

```
slat: point_spacing = "even" ;
```

When used, this attribute will improve memory use efficiency in Ferret. This is especially important for very large axes—10,000 points or more.

Ch10 Sec3.5.5. "Modulo" axes

The "modulo" axis attribute indicates that the axis wraps around, the first point immediately following the last. The most common uses of modulo axes are:

1. longitude axes for globe-encircling data. If the modulo length is different from 360 degrees, specify the value.
2. time axes for climatological data

```
time: modulo = " " ;
xavr: modulo = "100" ;
```

If the climatological data occurs in the years 0000 or 0001 then the year will be omitted from Ferret's output format.

NetCDF time axes encoded as year 0000 or 0001 are automatically flagged as modulo axes.

As of Ferret version5.5, longitude axes and climatological time axes are always detected as modulo, or as sub-span modulo when appropriate, unless Ferret is specifically directed that the axis is NOT modulo. See the sections on modulo axes and subspan modulo axesfor more information (p. 167ff)

Ch10 Sec3.5.6. Reversed-coordinate axes

NetCDF axes may contain monotonically decreasing axis coordinates instead of monotonically increasing coordinates. Ferret will hide this aspect of the file data ordering.

Ch10 Sec3.5.7. Converting time word data to numerical data

To set up a time axis for data represented as dates (e.g., "1972 January 15 at 12:15") it is necessary to determine a numerical representation for each of the dates. Ferret can assist with this process, as the following example shows.

Suppose the data are 6-hourly observations from 1-JAN-1991 at 12:00 to 15-MAR-1991 at 18:00. These commands will assist in creating the necessary time axis for a netCDF file:

```
yes? DEFINE AXIS/T="1-JAN-1991:12:00":"15-MAR-1991:18:00":6/UNITS=hours\
       my_time
yes? DEFINE GRID/T=my_time tgrid
yes? SET REGION/T="1-JAN-1991:12:00":"15-MAR-1991:18:00"
yes? LIST T[g=tgrid]                                !to see the time
values
yes? SAVE/FILE=my_time.cdf T[g=tgrid]
```

The file my_time.cdf now contains a model of exactly the desired time axis. Use the Unix command

```
% ncdump my_time.cdf > my_time.cdl
```

to obtain the time axis definition as text that can be inserted into your CDL file.

Ch10 Sec3.6. Example CDL file

The following is an example CDL file utilizing many of the features described in the preceding section.

```
netcdf converting_to_netcdf_supplement {
//      CONVERTING DATA TO THE "NETWORK COMMON DATA FORM" (NetCDF):
//                          A SUPPLEMENT
//
// NOAA PMEL Thermal Modeling and Analysis Project (TMAP)
// Dan Trueman, Steve Hankin
// last revised: 1 Dec 1993:  slat80_82 and slon10_140 coordinates included
//
// I. INTRODUCTION
//
// This supplement to "Converting Data to the Network Common Data Form:
// an Introduction" describes:
//
//      1. How to set up a cdl file capable of handling data
//         on staggered grids.
//      2. How to define coordinate systems such that the data
//         in the NetCDF file may be regarded as hyperslabs of
//         larger coordinate spaces.
//      3. How to define variables of 1, 2, or 3 dimensions.
//      4. How to define boundaries between unevenly spaced axis
//         coordinates (used in numerical integrations).
//      5. How to set up climatological "modulo" time axes.
//      6. How to convert time word data into numerical data
//         appropriate for NetCDF.
//
// In this sample data set, we will consider wind, salt, and
// velocity calculated using a staggered-grid, finite-difference
// technique. The wind data is naturally limited to the surface
// layer of the ocean (i.e. normal to the  depth axis). We will
// also consider the salt data to be limited to a narrow slab from
// 139E to 90W (I=10 to 140), 32.5N to 34.9N (J=80 to 82), and for
// all depth and time values.
//
// II. STAGGERED GRIDS
//
// Dealing with staggered grids is fairly straightforward. Dimensions
// for each grid axis must be defined, the axes themselves must be
// defined (in Variables), and the coordinate values for each axis must
// be initialized (in Data). In this case, there are two grids, a
// wind grid, and a velocity grid, so tlon, tlat and tdepth are
// defined for the wind grid, and ulon, ulat, and udepth for the velocity
// grid. The variables are then given arguments to place them in their
// proper grids (i.e. wind(time, sdepth, slat, slon)).
//
// III. HYPERSLABS
//
// There are a number of steps required to set up a NetCDF data set that
// represents a hyperslab of data from a larger grid definition.
//
//      1. Define a dimension named "grid_definition". This dimension
//         should be set equal to 1.
//      2. Define parent grids in Variables with the argument
//         "grid_definition".
//
//              char wind_grid(grid_definition) ;
//              char salt_grid(grid_definition) ;
//
//      3. Define the 4 axes of the parent grids using the "axes" attribute.
//
//                  wind_grid: axes = "slon slat normal time" ;
//                  salt_grid: axes = "slon slat sdepth time" ;
//
//          Note that the order of arguments is opposite that in the
//          variable declaration. The argument "normal" indicates that
//          wind_grid is normal to the depth axis.
//
//       4. Define the variables which are hyperslabs of these grids with
//          the proper dimensions.
//
//              float wind(time, slat, slon) ;
//              float salt(time, sdepth, slat80_82, slon10_140) ;
```

```
//
//         where slat80_82 = 3 and slon10_140 = 131. The axis names are
///        arbitrary - chosen for readability. These axes (child axes)
//         must be defined with the attribute "child_axis" as follows:
//
//              float slat80_82(slat80_82) ;
//                    slat80_82: child_axis = " " ;
//
//         These "child axes" need not be initialized in Data, nor do their

//         edges need be defined; Ferret retrieves this information from
//         the parent axes.
//
//      5. Use the "parent_grid" variable attribute to point to the
//         parent grid.
//
//                   wind: parent_grid = "wind_grid"
//
//      6. Also as a variable attribute, define the index range of interest
//         within the parent grid.
//
//                   wind: slab_min_index = 1s, 1s, 1s, 0s ;
//                   wind: slab_max_index = 160s, 100s, 1s, 0s ;
//                   salt: slab_min_index = 10s, 80s, 1s, 0s ;
//                   salt: slab_max_index = 140s, 82s, 27s, 0s ;
//
//         The "s" after each integer indicates a "short" 16-bit integer
//         rather than the default "long" 32-bit integer. If an axis
//         dimension is designated as "unlimited" then the index bounds
//         for this axis must be designated as "0s".
//
// These commands will effectively locate the wind and salt data within
// the full grid.
//
// IV. VARIABLES OF 1, 2, or 3 DIMENSIONS
//
// One, two, or three dimensional variables may be set up in one of
// two ways - either using the parent_grid and child_axis attributes
// as illustrated in the 3-dimensional variable "wind" from the hyperslab
// example, above, or by selecting axis names and units that provide
// Ferret with adequate hints to map this variable onto 4-dimensional
// space and time. The following hints are recognized by Ferret:
//
//      Units of days, hours, minutes, etc. or an axis name of "TIME", "DATE"
//         implies a time axis.
//      Units of "degrees xxxx" where "xxxx" contains "lat" or "lon" implies
//         a latitude or longitude axis, respectively.
//      Units of "degrees" together with an axis name containing "LAT" or
//         "Y" implies a latitude axis else longitude is assumed.
//      Units of millibars, "layer" or "level" or an axis name containing
//         "Z" or "ELEV" implies a vertical axis.
//
// V. UNEVENLY SPACED COORDINATE BOUNDARIES
//
// For coordinate axes with uneven spacing, the boundaries between each
// coordinate can be indicated by pointing to an additional axis that
// contains the locations of the boundaries. The dimension of this "edge"
// axis will necessarily be one larger than the coordinate axis concerned.
// If edges are not defined for an unevenly spaced axis, the midpoint
// between coordinates will be assumed by default.
//
//      1. Define a dimension one larger than the coordinate axis. For
//         the sdepth axis, with 27 coordinates, define:
//
//              sdepth_edges = 28 ;
//
//      2. Define an axis called sdepth_edges.
//      3. Initialize this axis appropriately (in Data).
//      4. As a sdepth axis attribute, point to sdepth_edges:
//
//                   sdepth: edges = "sdepth_edges" ;
//
```

```
// If the coordinate axes are evenly spaced, the attribute "point spacing"
// should be used:
//
//                   slat: point_spacing = "even" ;
//
// When used, this attribute will improve memory use efficiency in Ferret.
//
// VI. CLIMATOLOGICAL "MODULO" AXES
//
// The "modulo" axis attribute indicates that the axis wraps around,
// the first point immediately following the last. The most common
// uses of modulo axes are:
//
//      1. As longitude axes for globe-encircling data.
//      2. As time axes for climatological data.
//
//                   time: modulo = " " ; // any arbitrary string is allowed
//
// If the climatological data occurs in the years 0000 or 0001 then Ferret
// will omit the year from the output formatting.
//
// VII. CONVERTING TIME WORD DATA TO NUMERICAL DATA
//
// If the time data being converted to NetCDF format exists in string format
// (i.e. 1972 - JANUARY 15 2:15:00), rather than numerical format (i.e. 55123
// seconds) a number of TMAP routines are available to aid in the conversion
// process. The steps required for conversion are as follows:
//
//      1. Break the time string into its 6 pieces. If the data is of the
//         form dd-mmm-yyyy:hh:mm:dd, the TMAP routine "tm_break_date.f" can
//         be used.
//      2. Choose a time_origin before the beginning of the time data to
//         assure that all time values are positive. i.e. if the data begins
//         at 15-JAN-1982:05:30:00, choose a time origin of
//         15-JAN-1981:00:00:00. This time_origin should then be an attribute
//         of the time axis variable in the CDL file.
//      3. Produce numerical time data by using "tm_sec_from_bc.f", which
//         calculates the number of seconds between 01-01-0000:00:00:00 and
//         the date specified. Continuing the example from (2), the time value
//         for the first time step with respect to the time_origin could be
//         calculated as follows:
//
//         time(1) = tm_sec_from_bc(1982, 1, 15, 5, 30, 0) -
//                   tm_sec_from_bc(1981, 1, 15, 0, 0, 0)
//
//         or more generally
//
//         time(n)=tm_sec_from_bc(nyear,nmonth,nday,nhour,nminute,nsecond) -
//                 tm-sec_from_bc(oyear,omonth,oday,ohour,ominute,osecond)
//
//         where nyear is the year for the nth time step and oyear is the year
//         of time_origin.
//
// VII. EXAMPLE CDL FILE dimensions:

// staggered grid dimension definitions:

             slon = 160 ;    // wind/salt longitude dimension
             ulon = 160 ;    // velocity longitude dimension
             slat = 100 ;    // wind/salt latitude dimension
             ulat = 100 ;    // velocity latitude dimension
             sdepth = 27 ;   // salt depth dimension
             wdepth = 27 ;   // velocity depth dimension

             slon10_140 = 131 ;      // for salt hyperslab
             slat80_82 = 3 ;         // for salt hyperslab
             time = unlimited ;

 // grid_definition is the dimension name to be used for all grid definitions

             grid_definition = 1 ;
```

```
// edge dimension definitions:

             sdepth_edges = 28 ;
             wdepth_edges = 28 ;

variables:

  // variable definitions:

             float wind(time, slat, slon) ;  // 3-dimensional variable
                          wind: parent_grid = "wind_grid" ;
                          wind: slab_min_index = 1s, 1s, 1s, 0s ;
                          wind: slab_max_index = 160s, 100s, 1s, 0s ;
                          wind: long_name = "WIND" ;
                          wind: units     = "deg. C" ;
                          wind: _FillValue = 1E34f ;
             float salt(time, sdepth, slat80_82, slon10_140) ; // 4-dim.
Variable
                          salt: parent_grid = "salt_grid" ;
                          salt: slab_min_index = 10s, 80s, 1s, 0s ;
                          salt: slab_max_index = 140s, 82s, 27s, 0s ;
                          salt: long_name = "(SALINITY(ppt) - 35) /1000" ;
                          salt: units     = "frac. by wt. less .035" ;
                          salt: _FillValue = -999.f ;

             float u(time, sdepth, ulat, ulon) ;
                          u: long_name    = "ZONAL VELOCITY" ;
                          u: units        = "cm/sec" ;
                          u: _FillValue = 1E34f ;
             float v(time, sdepth, ulat, ulon) ;
                          v: long_name    = "MERIDIONAL VELOCITY" ;
                          v: units        = "cm/sec" ;
                          v: _FillValue = 1E34f ;
             float w(time, wdepth, slat, slon) ;
                          w: long_name    = "VERTICAL VELOCITY" ;
                          w: units        = "cm/sec" ;
                          w: _FillValue = 1E34f ;

  // axis definitions:

             float slon(slon) ;
                          slon: units = "degrees" ;
                          slon: point_spacing = "even" ;
             float ulon(ulon) ;
                          ulon: units = "degrees" ;
                          ulon: point_spacing = "even" ;
             float slat(slat) ;
                          slat: units = "degrees" ;
                          slat: point_spacing = "even" ;
             float ulat(ulat) ;
                          ulat: units = "degrees" ;
                    ulat: point_spacing = "even" ;
             float sdepth(sdepth) ;
                          sdepth: units = "meters" ;
                          sdepth: positive = "down" ;
                          sdepth: edges = "sdepth_edges" ;
             float wdepth(wdepth) ;
                          wdepth: units = "meters" ;
                          wdepth: positive = "down" ;
                          wdepth: edges = "wdepth_edges" ;
             float time(time) ;
                          time: modulo = " " ;
                    time: time_origin = "15-JAN-1981:00:00:00" ;
                          time: units = "seconds" ;

  // child grid definitions:

             float slon10_140(slon10_140) ;
                          slon10_140: child_axis = " " ;

                          slon10_140: units = "degrees" ;
```

```
            float slat80_82(slat80_82) ;
                        slat80_82: child_axis = " " ;
                        slat80_82: units = "degrees" ;

  // edge axis definitions:

            float sdepth_edges(sdepth_edges) ;
            float wdepth_edges(wdepth_edges) ;

  // parent grid definition:

            char wind_grid(grid_definition) ;
                  wind_grid: axes = "slon slat normal time" ;
            char salt_grid(grid_definition) ;
                  salt_grid: axes = "slon slat sdepth time" ;

  // global attributes:
            :title = "NetCDF Title" ;

data:

//  //  ignore this block  //
//This next data entry, for time, should be ignored. Time is initialized here
// only so that Ferret can read test.cdf (the file created by this cdl file)
// with no additional data inserted into it.
Time=1000;
// // end of ignored block //

slat=
-28.8360729218,-26.5299491882,-24.2880744934,-22.1501560211,-20.1513576508,
-18.3207626343,-16.6801033020,-15.2428140640,-14.0134353638,-12.9874248505,
-12.1513509750,-11.4834814072,-10.9547319412,-10.5299386978,-10.1693935394,
-9.8333206177,-9.4999876022,-9.1666536331,-8.8333196640,-8.4999856949,
-8.1666526794,-7.8333187103,-7.4999847412,-7.1666512489,-6.8333182335,
-6.4999852180,-6.1666517258,-5.8333182335,-5.4999852180,-5.1666517258,
-4.8333187103,-4.4999852180,-4.1666517258,-3.8333187103,-3.4999852180,
-3.1666517258,-2.8333184719,-2.4999852180,-2.1666519642,-1.8333185911,
-1.4999852180,-1.1666518450,-0.8333183527,-0.4999849498,-0.1666515470,
0.1666818559,0.5000152588,0.8333486915,1.1666821241,1.5000154972,
1.8333489895,2.1666824818,2.5000159740,2.8333494663,3.1666829586,
3.5000162125,3.8333497047,4.1666831970,4.5000162125,4.8333497047,
5.1666831970,5.5000162125,5.8333497047,6.1666827202,6.5000162125,
6.8333497047,7.1666827202,7.5000166893,7.8333501816,8.1666841507,
8.5000181198,8.8333511353,9.1666851044,9.5000190735,9.8333530426,
10.1679363251,10.5137376785,10.8892869949,11.3138961792,11.8060989380,
12.3833675385,13.0618314743,13.8560228348,14.7786512375,15.8403968811,
17.0497493744,18.4128704071,19.9334945679,21.6128730774,23.4497566223,
25.4404067993,27.5786647797,29.8560409546,32.2618522644,34.7833900452,
37.4061241150,40.1139259338,42.8893203735,45.7137718201,48.5679702759;
ulat=
-27.6721439362,-25.3877544403,-23.1883945465,-21.1119174957,-19.1907978058,
-17.4507274628,-15.9094810486,-14.5761461258,-13.4507236481,-12.5241250992,
-11.7785758972,-11.1883859634,-10.7210769653,-10.3387994766,-9.9999876022,
-9.6666545868,-9.3333206177,-8.9999866486,-8.6666526794,-8.3333196640,
-7.9999856949,-7.6666517258,-7.3333182335,-6.9999847412,-6.6666512489,
-6.3333182335,-5.9999847412,-5.6666517258,-5.3333182335,-4.9999847412,
-4.6666517258,-4.3333182335,-3.9999849796,-3.6666517258,-3.3333184719,
-2.9999852180,-2.6666519642,-2.3333184719,-1.9999853373,-1.6666518450,
-1.3333184719,-0.9999850392,-0.6666516662,-0.3333182633,0.0000151545,
0.3333485723,0.6666819453,1.0000153780,1.3333487511,1.6666821241,
2.0000154972,2.3333489895,2.6666827202,3.0000162125,3.3333497047,
3.6666829586,4.0000162125,4.3333497047,4.6666827202,5.0000162125,
5.3333492279,5.6666827202,6.0000162125,6.3333492279,6.6666827202,
7.0000157356,7.3333497047,7.6666831970,8.0000171661,8.3333511353,
8.6666841507,9.0000181198,9.3333520889,9.6666860580,10.0000190735,
10.3358526230,10.6916217804,11.0869522095,11.5408391953,12.0713586807,
12.6953773499,13.4282865524,14.2837600708,15.2735414505,16.4072513580,
17.6922454834,19.1334934235,20.7334957123,22.4922523499,24.4072608948,
26.4735546112,28.6837768555,31.0283031464,33.4953994751,36.0713844299,
38.7408676147,41.4869842529,44.2916526794,47.1358833313,50.0000534058;
slon=
```

```
130.5,131.5,132.5,133.5,134.5,135.5,136.5,137.5,138.5,139.5,140.5,141.5,
142.5,143.5,144.5,145.5,146.5,147.5,148.5,149.5,150.5,151.5,152.5,153.5,
154.5,155.5,156.5,157.5,158.5,159.5,160.5,161.5,162.5,163.5,164.5,165.5,
166.5,167.5,168.5,169.5,170.5,171.5,172.5,173.5,174.5,175.5,176.5,177.5,
178.5,179.5,180.5,181.5,182.5,183.5,184.5,185.5,186.5,187.5,188.5,189.5,
190.5,191.5,192.5,193.5,194.5,195.5,196.5,197.5,198.5,199.5,200.5,201.5,
202.5,203.5,204.5,205.5,206.5,207.5,208.5,209.5,210.5,211.5,212.5,213.5,
214.5,215.5,216.5,217.5,218.5,219.5,220.5,221.5,222.5,223.5,224.5,225.5,
226.5,227.5,228.5,229.5,230.5,231.5,232.5,233.5,234.5,235.5,236.5,237.5,
238.5,239.5,240.5,241.5,242.5,243.5,244.5,245.5,246.5,247.5,248.5,249.5,
250.5,251.5,252.5,253.5,254.5,255.5,256.5,257.5,258.5,259.5,260.5,261.5,
262.5,263.5,264.5,265.5,266.5,267.5,268.5,269.5,270.5,271.5,272.5,273.5,
274.5,275.5,276.5,277.5,278.5,279.5,280.5,281.5,282.5,283.5,284.5,285.5,
286.5,287.5,288.5,289.5;
ulon=
131.0,132.0,133.0,134.0,135.0,136.0,137.0,138.0,139.0,140.0,141.0,142.0,
143.0,144.0,145.0,146.0,147.0,148.0,149.0,150.0,151.0,152.0,153.0,154.0,
155.0,156.0,157.0,158.0,159.0,160.0,161.0,162.0,163.0,164.0,165.0,166.0,
167.0,168.0,169.0,170.0,171.0,172.0,173.0,174.0,175.0,176.0,177.0,178.0,
179.0,180.0,181.0,182.0,183.0,184.0,185.0,186.0,187.0,188.0,189.0,190.0,
191.0,192.0,193.0,194.0,195.0,196.0,197.0,198.0,199.0,200.0,201.0,202.0,
203.0,204.0,205.0,206.0,207.0,208.0,209.0,210.0,211.0,212.0,213.0,214.0,
215.0,216.0,217.0,218.0,219.0,220.0,221.0,222.0,223.0,224.0,225.0,226.0,
227.0,228.0,229.0,230.0,231.0,232.0,233.0,234.0,235.0,236.0,237.0,238.0,
239.0,240.0,241.0,242.0,243.0,244.0,245.0,246.0,247.0,248.0,249.0,250.0,
251.0,252.0,253.0,254.0,255.0,256.0,257.0,258.0,259.0,260.0,261.0,262.0,
263.0,264.0,265.0,266.0,267.0,268.0,269.0,270.0,271.0,272.0,273.0,274.0,
275.0,276.0,277.0,278.0,279.0,280.0,281.0,282.0,283.0,284.0,285.0,286.0,
287.0,288.0,289.0,290.0;
sdepth=
5.0,15.0,25.0,35.0,45.0,55.0,65.0,75.0,85.0,95.0,106.25,120.0,136.25,155.0,
177.5,205.0,240.0,288.5,362.5,483.5,680.0,979.5,1395.5,1916.0,2524.0,3174.0,
3824.0;
sdepth_edges=
0.0,10.0,20.0,30.0,40.0,50.0,60.0,70.0,80.0,90.0,100.0,112.5,127.5,
145.0,165.0,190.0,220.0,260.0,317.0,408.0,559.0,801.0,1158.0,1633.0,2199.0,
2849.0,3499.0,4149.0;
wdepth=
10.0,20.0,30.0,40.0,50.0,60.0,70.0,80.0,90.0,100.0,112.5,127.5,145.0,165.0,
190.0,220.0,260.0,317.0,408.0,559.0,801.0,1158.0,1633.0,2199.0,2849.0,3499.0,
4149.0;
wdepth_edges=
5.0,15.0,25.0,35.0,45.0,55.0,65.0,75.0,85.0,94.375,105.625,119.375,135.625,
153.75,176.25,202.5,235.75,280.0,347.5,460.75,651.25,950.0,1372.75,1895.0,
2524.0,3174.0,3986.5,4311.0;
slon10_140=
      139.5, 140.5, 141.5, 142.5, 143.5, 144.5, 145.5, 146.5, 147.5,
      148.5, 149.5, 150.5, 151.5, 152.5, 153.5, 154.5, 155.5, 156.5, 157.5,
      158.5, 159.5, 160.5, 161.5, 162.5, 163.5, 164.5, 165.5, 166.5, 167.5,
      168.5, 169.5, 170.5, 171.5, 172.5, 173.5, 174.5, 175.5, 176.5, 177.5,
      178.5, 179.5, 180.5, 181.5, 182.5, 183.5, 184.5, 185.5, 186.5, 187.5,
      188.5, 189.5, 190.5, 191.5, 192.5, 193.5, 194.5, 195.5, 196.5, 197.5,
      198.5, 199.5, 200.5, 201.5, 202.5, 203.5, 204.5, 205.5, 206.5, 207.5,
      208.5, 209.5, 210.5, 211.5, 212.5, 213.5, 214.5, 215.5, 216.5, 217.5,
      218.5, 219.5, 220.5, 221.5, 222.5, 223.5, 224.5, 225.5, 226.5, 227.5,
      228.5, 229.5, 230.5, 231.5, 232.5, 233.5, 234.5, 235.5, 236.5, 237.5,
      238.5, 239.5, 240.5, 241.5, 242.5, 243.5, 244.5, 245.5, 246.5, 247.5,
      248.5, 249.5, 250.5, 251.5, 252.5, 253.5, 254.5, 255.5, 256.5, 257.5,
      258.5, 259.5, 260.5, 261.5, 262.5, 263.5, 264.5, 265.5, 266.5, 267.5,
      268.5, 269.5 ;
slat80_82=
      11.8060989379883, 12.3833675384522, 13.0618314743042 ;

}
```

Ch10 Sec4. CREATING A MULTI-FILE NETCDF DATA SET

Ferret supports collections of netCDF files that are regarded as a single netCDF data set. Such data sets are referred to as "MC" (multi CDF) data sets. They are defined via a descriptor file, in the style of TMAP-formatted data sets. These are FORTRAN NAMELIST-formatted files. Slight variations in syntax exist between systems. The requirements for an MC data set are described in the chapter "Data Set Basics", section "Multi-file NetCDF data sets".

Previous to version 5.2 Ferret performs sanity checking on the data set by comparing these time coordinates with those found in the data files as he data are read. In version 5.3 and higher no sanity checks are performed. This means that the MC descriptor mechanism can be used to associate into time series groups of files that are not internally self-documenting with respect to time, however, it also shifts an additional burden onto the user of carefully checking the validity of the STEPFILE records in the descriptor files.

Beginning with version 5.8 of Ferret the stepfiles may contain different scale and offset values for the variables they contain. (p. 275). Ferret reads and applies the scale and offset values as data from each stepfile is read. Note that the commands

```
yes? SAY `var, RETURN=nc_offset`
yes? SAY `var, RETURN=nc_scale`
```

return the latest scale and offset value that were applied.

The fields which are essential to consider are

$FORMAT_RECORD

```
D_TYPE          = '  MC' ,
D_FORMAT          = '  1A',
```

which must be exactly as shown.

$BACKGROUND_RECORD

```
D_TITLE         = 'Put your data set title here',
```

where you can insert a data sets title to appear on plots and listings;

```
D_T0TIME          = '14-JAN-1980 14:00:00',
```

which corresponds exactly to the /T0 qualifier on the DEFINE AXIS command

```
D_TIME_UNIT       = 3600.0,
```

which contains the same information as /UNITS= on the DEFINE AXIS command encoded as numbers of seconds . (/UNITS="minutes" corresponds to D_TIME_UNIT = 60., /UNITS="hours" corresponds to D_TIME_UNIT = 3600., etc.)

```
D_CALTYPE        = '360_DAY',
```

to specify the name of the calendar if your time axis is not on the standard Gregorian calendar. See the discussion of time axes and calendars (p. 146) for more on the calendars available.

```
$STEPFILE_RECORD

    S_FILENAME          = 'mtaa063-nc.001',
```

which points to the individual file names. Typically you will need one STEPFILE_RECORD for each file in the series, however if the files are named with extension .001, .002, ... you can use S_NUM_OF_FILES below.

```
    S_START             = 17592.0,
```

which contains the time step value of the first time step in the file. (For help determining the time step values use DEFINE AXIS to create the desired time axis (say, my_t_ax) and then use LIST T[gt=my_t_ax] to see the dates and time steps.)

```
    S_END               = 34309.0,
```

which contains the time step value of the last time step in the file (or group of files if S_NUM_OF_FILES is greater than 1). If there is only a single time step in the file set S_END identical to S_START.

```
    S_DELTA             = 73.0,
```

which contains the delta value separating time steps within in the file. If there is only a single time step in the file set S_DELTA = 1.

```
    S_NUM_OF_FILES      = 23,
```

Normally S_NUM_OF_FILES should be omitted or should have a value of 1. Use it for the special case that your files are named with ending .001, .002, ... in which case you can describe them all with a single STEPFILE_RECORD as in the example. (S_DELTA must then also describe the delta between the last time step in each file and the first time step in the next file of the series.)

Ch10 Sec4.1. Tools for making descriptor files

Two Ferret Users have written tools to create multi-file netCDF data sets. They are

1) A unix shell script, nc2mc which is available and documented at
http://www.mpch-mainz.mpg.de/~joeckel/nc2mc/

2) A perl script, make_des which is available and documented at
http://www.gfdl.noaa.gov/~atw/ferret/make_des

Ch10 Sec4.2. Example descriptor file

A typical MC descriptor file is given below. This file ties into a single data set the 23 files named mtaa063-nc.001 through mtaa063-nc.024. The time steps are encoded in the descriptor file through the S_START and S_END values.

Descriptor files have a space before the $ in the record headers, and comment lines begin with a *. In addition there are differences in the formatting of descriptor files depending on the operationg system. Here is an FAQ which addresses these differences:

http://www.ferret.noaa.gov/Ferret/FAQ/system/linux_mc_descriptors.html

```
*****************************************************************
***
*        NOAA/PMEL Tropical Modeling and Analysis Program, Seattle, WA.
*
*                       created by MAKE_DESCRIPT rev. 4.01
*
*****************************************************************
***
 $FORMAT_RECORD
      D_TYPE               = '  MC',
      D_FORMAT             = '  1A',
      D_SOURCE_CLASS       = 'MODEL OUTPUT',
 $END
 $BACKGROUND_RECORD
      D_EXPNUM             = '0063',
      D_MODNUM             = '  AA',
      D_TITLE              = 'MOM model output forced by Sadler winds',
      D_T0TIME             = '14-JAN-1980 14:00:00',
      D_TIME_UNIT          = 3600.0,
      D_TIME_MODULO        = .FALSE.,
      D_ADD_PARM           = 15*' ',
 $END
 $MESSAGE_RECORD
      D_MESSAGE            = ' ',
      D_ALERT_ON_OPEN      = F,
      D_ALERT_ON_OUTPUT    = F,
 $END
*************************************************
 $EXTRA_RECORD
 $END
```

```
$STEPFILE_RECORD
     s_filename           = 'mtaa063-nc.001',
     S_AUX_SET_NUM        = 0,
     S_START              = 17592.0,
     S_END                = 34309.0,
     S_DELTA              = 73.0,
     S_NUM_OF_FILES       = 23,
     S_REGVARFLAG         = ' ',
$END
************************************************
$STEPFILE_RECORD
     s_filename           = '**END OF STEPFILES**'

$END
************************************************
```

Chapter 11: WRITING EXTERNAL FUNCTIONS

Ch11 Sec1. OVERVIEW

External functions are user-written Fortran routines which are called from the Ferret command line just as internal Ferret functions (e.g. SIN, COS) are invoked. For example, you might create a routine to compute the amplitudes of the Fourier transform of a time series (the periodogram) and name your function "FFT_AMP". In Ferret you would use it like this:

```
LET my_fft = FFT_AMP(my_time_series)
```

Once the variable my_fft is defined, it can be used in other expressions, plotted, etc. External functions can be used in every way that Ferret internal functions are used and are distinguished only by their appearance after internal functions when the user issues a SHOW FUNC command.

A Ferret external function uses input arguments defined in a Ferret session and computes a result with user-supplied Fortran or C code. The external function specifies how the grid for the result will be generated. The axes can be inherited from the input arguments or specified in the external function code.

Utility functions, linked in when the external function is compiled, obtain information from Ferret about variables and grids. The utility functions are described in section 6 (p. 306).

Ferret external functions are compiled individually to create shared object (.so) files. These are dynamically linked with Ferret at run time. Ferret looks for shared object files in the directories specified in the FER_EXTERNAL_FUNCTIONS environment variable.

Ch11 Sec2. GETTING STARTED

Ferret Version 5.0 and later contains everything you need to run the external functions which are included with the distribution. The environment variable FER_EXTERNAL_FUNCTIONS is defined, listing the directory where the shared object files reside. To see a list of the included external functions and their arguments, type

```
> ferret
…
yes? SHOW FUNC/EXTERNAL
Externally defined functions available to Ferret:
ADD_9(A,B,C,D,E,F,G,H,I)
    (demonstration function) adds 9 arguments
AVET(A)
    (demonstration function) returns the time average
    A: data to be time averaged
...
```

Ch11 Sec2.1. Getting example/development code

To write your own external functions, you will need to get source code and set up a directory in which to work. All of the source code you need to get started (Makefiles, common files, simple examples) can be obtained from the Ferret Home Page. Go to the External Functions page and follow the instructions there.

You will need to download a tar file to get started. When you untar this file you will find that the ***ef_utility/*** directory contains the ***ferret_cmn*** subdirectory, containing common files that you need to compile external functions. The ***ef_utility/*** directory must be in place before you can compile any of the other external function code. The ***examples/*** directory contains source code and a ***Makefile***. You will create further directories with your external functions source code and Makefiles patterned on what is in the examples directory.

Also on the External Functions web page are documents about how to use code written in Fortran 90 and on compiling code with an F95 compiler, contributed by Ferret users.

Ch11 Sec3. QUICK START EXAMPLE

It's always easier to start coding from an example. Any of the external functions we provide should be documented well enough to serve as a starting point for writing a new function. In this section, we take the most trivial example function, pass_thru, and alter it to do something a little more interesting, if no more useful.

Ch11 Sec3.1. The times2bad20 function

We'll use the pass_thru(...) function as a template, modifying it into a times2bad20(...) function. This new function will multiply all values by 2.0 and will replace missing value flags with the value 20.0.

Inside any of the example functions, the areas that you need to (are allowed to) modify are set off with

```
c*********
c  USER CONFIGURABLE PORTION

        ->Insert your code here<-


c USER CONFIGURABLE PORTION
c*********
```

Here's what you need to do to create the new function:

1. move to the ***examples/*** directory

2. copy ***pass_thru.F*** to ***times2bad20.F***
3. use your favorite editor to change each "pass_thru" to "times2bad20"
4. go down into the "times2bad20_init" section and change the description of the function
5. go to the "times2bad20_compute" subroutine and change the code to look like this

```
c*            result(i,j,k,l) = bad_flag_result
             result(i,j,k,l) = 20

          ELSE

c*            result(i,j,k,l) = arg1(i,j,k,l)
             result(i,j,k,l) = 2 * arg1(i,j,k,l)
```

Assuming you have downloaded all of the ***ef_utility/*** directory development code and you are still in the ***examples/*** directory, you should be able to (Figure 11_1)

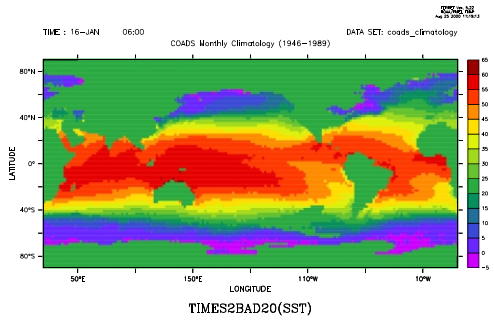


Figure 11_1

```
> make times2bad20.so
> setenv FER_EXTERNAL_FUNCTIONS .
> ferret
  ...
yes? use coads_climatology
yes? let a = times2bad20(sst)
yes? shade a[l=1]
```

Congratulations! You have just written your first external function.

Ch11 Sec4. ANATOMY OF AN EXTERNAL FUNCTION

Every Ferret external function contains an ***~_init*** subroutine which describes the external function's arguments and result grid and a ***~_compute*** subroutine which actually performs the calculation. If the code you wish to implement is written in C, then the ~_compute function will be a Fortran wrapper that calls the C routine. Three other subroutines are available for requesting memory allocation; creating axis limits for the result variable which are extended with respect to the defined region (useful for derivative calculations, etc.); and creating custom axes for the result.

For the following discussion we will assume that our external function is called efname (with source code in a file named efname.F). Examples are also taken from the external functions ***examples/*** directory which you installed when you downloaded the external functions code. This section will briefly describe the work done by the ***~_init*** and ***~_compute*** subroutines. The individual utility functions called by these subroutines are described in the section on Utility Functions below.

When you name your external functions, be aware that Ferret will search its internal function names before the external function names. So if you use a name that is already in use, your function will not be called. Use SHOW FUNCTION from Ferret to list the names that already are in use

Ch11 Sec4.1. The ~_init subroutine (required)

subroutine efname_init (id)

This subroutine specifies basic information about the external function. This information is used when Ferret parses the command line and checks the number of arguments; when it generates the output of SHOW FUNCTION/EXTERNAL; and in determining the result grid.

The following code from ***examples/subtract.F*** shows a typical example of an ***~_init*** subroutine. For an example with more arguments please look at ***examples/add_9.F***. For an example where a result axis is reduced with respect to the equivalent input axis take a look at ***examples/percent_good_t.F***.

```
      SUBROUTINE subtract_init(id)

      INCLUDE 'ferret_cmn/EF_Util.cmn'

      INTEGER id, arg

* ************************
* USER CONFIGURABLE PORTION
*
      CALL ef_set_desc(id,'(demonstration function) returns: A - B' )

      CALL ef_set_num_args(id, 2)   ! Maximum allowed is 9
      CALL ef_set_axis_inheritance(id, IMPLIED_BY_ARGS,
     .     IMPLIED_BY_ARGS, IMPLIED_BY_ARGS, IMPLIED_BY_ARGS)
      CALL ef_set_piecemeal_ok(id, NO, NO, NO, NO)

      arg = 1
      CALL ef_set_arg_name(id, arg, 'A')
      CALL ef_set_axis_influence(id, arg, YES, YES, YES, YES)

      arg = 2
      CALL ef_set_arg_name(id, arg, 'B')
      CALL ef_set_axis_influence(id, arg, YES, YES, YES, YES)
*
* USER CONFIGURABLE PORTION
* ************************
```

```
      RETURN
      END
```

Ch11 Sec4.2. The ~_compute subroutine (required)

subroutine efname_compute (id, arg_1, arg_2, ..., result, wkr_1, wrk_2, ...)

This subroutine does the actual calculation. Arguments to the external function and any requested working storage arrays are passed in. Dimension information for the subroutine arguments is obtained from Ferret common blocks in ***ferret_cmn/EF_mem_subsc.cmn***. The ***mem1lox:mem1hix***, etc. values are determined by Ferret and correspond to the region requested for the calculation.@Body Text = In the ***~_compute*** subroutine you may call other subroutines which are not part of the efname_compute.F source file.

```
      SUBROUTINE subtract_compute(id, arg_1, arg_2, result)

      INCLUDE 'ferret_cmn/EF_Util.cmn'
      INCLUDE 'ferret_cmn/EF_mem_subsc.cmn'

      INTEGER id

      REAL bad_flag(EF_MAX_ARGS), bad_flag_result
      REAL arg_1(mem1lox:mem1hix, mem1loy:mem1hiy,
     .     mem1loz:mem1hiz, mem1lot:mem1hit)
      REAL arg_2(mem2lox:mem2hix, mem2loy:mem2hiy,
     .     mem2loz:mem2hiz, mem2lot:mem2hit)
      REAL result(memreslox:memreshix, memresloy:memreshiy,
     .     memresloz:memreshiz, memreslot:memreshit)

* After initialization, the 'res_' arrays contain indexing information
* for the result axes.  The 'arg_' arrays will contain the indexing
* information for each variable's axes.

      INTEGER res_lo_ss(4), res_hi_ss(4), res_incr(4)
      INTEGER arg_lo_ss(4,EF_MAX_ARGS), arg_hi_ss(4,EF_MAX_ARGS),
     .     arg_incr(4,EF_MAX_ARGS)


* *************************
*  USER CONFIGURABLE PORTION
*

      INTEGER i,j,k,l
      INTEGER i1, j1, k1, l1
      INTEGER i2, j2, k2, l2

      CALL ef_get_res_subscripts(id, res_lo_ss, res_hi_ss, res_incr)
      CALL ef_get_arg_subscripts(id, arg_lo_ss, arg_hi_ss, arg_incr)
      CALL ef_get_bad_flags(id, bad_flag, bad_flag_result)

      …

*
* USER CONFIGURABLE PORTION
* *************************
```

```
      RETURN
      END
```

Please see the "Loop Indices" section for the example calculation.4.3

Ch11 Sec4.3. The ~_work_size subroutine (required when work arrays are defined)

This routine allows the external function author to request that Ferret allocate memory (working storage) for use by the external function. The memory allocated is passed to the external function when the ***~compute*** subroutine is called. The working storage arrays are assumed to be REAL*4 arrays; adjust the size of the arrays for other data types. See the sample code under ef_get_coordinates (p. 317) for an example of allocating a REAL*8 work array. The working storage is deallocated after the ***~compute*** subroutine returns.

When working storage is to be requested, a call to ***ef_set_num_work_arrays*** must be in the ***~init*** subroutine:

```
SUBROUTINE efname_init (id)
…
CALL ef_set_num_work_arrays (id,2)
```

A maximum of 9 work arrays may be declared. At the time the ***~work_size*** subroutine is called by Ferret, any of the utility functions that retrieve information from Ferret may be used in the determination of the appropriate working storage size.

Here is an example of a ***~work_size*** subroutine:

```
      SUBROUTINE efname_work_size(id)
      INCLUDE 'ferret_cmn/EF_Util.cmn'
      INCLUDE 'ferret_cmn/EF_mem_subsc.cmn'
      INTEGER id
*
* ef_set_work_array_lens(id, array #, X len, Y len, Z len, T len)
*
      INTEGER nx, ny, id
      INTEGER arg_lo_ss(4,1:EF_MAX_ARGS), arg_hi_ss(4,1:EF_MAX_ARGS),
     .     arg_incr(4,1:EF_MAX_ARGS)
      CALL ef_get_arg_subscripts(id, arg_lo_ss, arg_hi_ss, arg_incr)

      NX = 1 + (arg_hi_ss(X_AXIS,ARG1) - arg_lo_ss(X_AXIS,ARG1))
      NY = 1 + (arg_hi_ss(Y_AXIS,ARG1) - arg_lo_ss(Y_AXIS,ARG1))

      CALL ef_set_work_array_lens(id,1,NX,NY,1,1)
      CALL ef_set_work_array_lens(id,2,NX,NY,1,1)

      RETURN
```

In the argument list of the ***~compute*** subroutine, the work array(s) come after the result variable. Declare the workspace arrays using index bounds wrk1lox:wrk2hix, … which were set by the ef_set_work_array_lens call above.

```
      SUBOUTINE efname_compute (arg_1, result, workspace1, workspace2)
…

*   Dimension the work arrays
      REAL workspace1(wrk1lox:wrk1hix, wrk1loy:wrk1hiy,
     .                wrk1loz:wrk1hiz, wrk1lot:wrk1hit)
      REAL workspace2(wrk2lox:wrk2hix, wrk2loy:wrk2hiy,
     .                wrk2loz:wrk2hiz, wrk2lot:wrk2hit)
```

Ch11 Sec4.4. The ~_result_limits subroutine (required if result has a custom or abstract axis)

The result limits routine sets the limits on ***ABSTRACT*** and ***CUSTOM*** axes created by the external function.

An example ***~result_limits*** routine might look like this:

```
      SUBROUTINE my_result_limits(id)
      INCLUDE 'ferret_cmn/EF_Util.cmn'
      INTEGER id, arg, NF
*
      INTEGER arg_lo_ss(4,EF_MAX_ARGS), arg_hi_ss(4,EF_MAX_ARGS),
     .     arg_incr(4,EF_MAX_ARGS)
      INTEGER lo, hi

      CALL ef_get_arg_subscripts(id, arg_lo_ss, arg_hi_ss, arg_incr)

      arg = 1
      lo = 1
      hi = (arg_hi_ss(T_AXIS,arg) - arg_lo_ss(T_AXIS,arg) + 1)/ 2
      call ef_set_axis_limits(id, T_AXIS, lo, hi)

      RETURN
      END
```

Ch11 Sec4.5. The ~_custom_axes subroutine (required if result has a custom axis)

The ***~custom_axes*** subroutine allows the external function author to create new axes that will be attached the the result of the ***~compute*** subroutine. An example of such a function might take time series data with a time axis and create, as a result, a Fourier transform with a frequency axis.

The ***~custom_axes*** subroutine must be used with care because not all the Ferret internal information is available to the external function at the time Ferret calls this routine. Ferret must determine the grid on which a variable is defined before it actually evaluates the variable. This is fundamental to the delayed evaluation framework -- the aspect of Ferret that makes it possible to work with multi-gigabyte data sets while having minimal awareness of memory limitations. The ~custom_axes routines are called at the time that Ferret determines grid. Certain

types of information are not available to Ferret (or to you, as author of an external function) during this time. The information which is not available is

1. the values of arguments to the function (capability to get the value of a scalar argument is being implemented for a future version)

2. context information specified with SET REGION

3. context information set with command qualifiers such as

```
CONTOUR/X=130e:80w
```

Items two and three are because this information is mutable -- it may be changed when the function is actually invoked.

The context information which IS available is

1. information that is actually contained in the function call, such as the X limits of

```
LET myvar = MY_EFN(v[x=130e:80w])
```

2. information that is embedded in nested variable definitions, such as the X limits of

```
LET tmp_var = v[x=130e:80w]
LET myvar = MY_EFN(tmp_var)
```

If no context information is available through these means then the context information supplied by the call to ef_get_arg_subscripts will be the full span (low and high limits) of the relevant axes.

Examples:

You can set an axis explicitly in subroutine my_fcn_custom_axes:

```
SUBROUTINE custom_custom_axes(id)
INCLUDE 'ferret_cmn/EF_Util.cmn'
INTEGER id
CALL ef_set_custom_axis(id, T_AXIS, 0.0, 1000.0, 25.0, 'Hertz', NO)
RETURN
END
```

Also, you can define an axis using information about the argument, as in the FFT functions which set up a frequency axis based on the input time axis (somewhat simplified here):

```
SUBROUTINE ffta_sample_custom_axes(id)

INCLUDE 'ferret_cmn/EF_Util.cmn'
```

```
INTEGER id
INTEGER nfreq_lo_l, nfreq_hi_l

INTEGER arg_lo_ss(4,EF_MAX_ARGS), arg_hi_ss(4,EF_MAX_ARGS),
     .     arg_incr(4,EF_MAX_ARGS)
INTEGER arg
INTEGER nfreq, nd

REAL yquist, freq1, freqn
REAL boxsize(1)

arg = 1
CALL ef_get_arg_subscripts(id, arg_lo_ss, arg_hi_ss, arg_incr)

CALL ef_get_box_size(id, arg, T_AXIS, arg_lo_ss(T_AXIS,arg),
     .   arg_lo_ss(T_AXIS,arg), boxsize)

nfreq_lo_l = arg_lo_ss(T_AXIS,arg)
nfreq_hi_l = arg_hi_ss(T_AXIS,arg)

nd = abs(nfreq_hi_l - nfreq_lo_l) + 1

nfreq = nd/2
yquist = 1./(2.*boxsize(1))         ! Nyquist frequency

freq1 = 1.* yquist/ float(nfreq)
freqn = 1.001*yquist

C  Set label for the frequency axis CYC/units.

outunits = 'cyc/day'
CALL ef_set_custom_axis (id, T_AXIS, freq1, freqn, freq1, outunits, NO)

RETURN
END
```

Ch11 Sec5. NOTES AND SUGGESTIONS

Ch11 Sec5.1. Inheriting axes

When creating an external function, you can get Ferret to do a lot of conformability checking for you if you "inherit axes" properly. This means that Ferret can be responsible for making sure that the arguments you pass to the function are of the proper dimensionality to be combined together in basic operations such as addition, multiplication etc. For any given axis orientation, X, Y, Z, or, T, two arguments are said to be conformable on that axis if 1) they are either of the same length, or 2) at least one of the arguments has a size of 1 on the axis. (The terminology "size of 1" may equivalently be thought of as a size of 0. In other words, the data is normal to this axis.) When Ferret encounters a problem it will send an error message rather than passing the data to your external function which might result in a crash.

To get Ferret to do this kind of checking you should inherit axes from as many appropriate arguments as possible. For instance, in subtract.F we have the following sections of code:

```
 subtract_init(...)

 …
 CALL ef_set_axis_inheritance(id, IMPLIED_BY_ARGS,
.     IMPLIED_BY_ARGS, IMPLIED_BY_ARGS, IMPLIED_BY_ARGS)
...
```

This means that the axes of the result, and the index range of the result on those axes, will be determined by arguments.

```
…
arg = 1
CALL ef_set_arg_name(id, arg, 'A')
CALL ef_set_axis_influence(id, arg, YES, YES, YES, YES)

arg = 2
CALL ef_set_arg_name(id, arg, 'B')
CALL ef_set_axis_influence(id, arg, YES, YES, YES, YES)
...
```

Here we specify that each result axis is dependent upon the axes from both arguments. When Ferret sees this, it knows the arguments must be conformable before it passes them to the external function.

The advantages of this approach are best understood by thinking about this example function "MY_ADD_FUNCTION," which performs a simple addition:

```
LET arg1 = X[x=0:1:.1]
LET arg2 = Y[Y=101:102:.05]
LET my_result = MY_ADD_FUNCTION(arg1, arg2)
```

The desired outcome is that "my_result" is a 2-dimensional field which inherits its X axis from arg1 and its Y axis from arg2.

If arguments and result are on the same grid, you should inherit all axes from all arguments. In general, you should inherit axes from as many arguments as possible.

Ch11 Sec5.2. Loop indices

Note: Array indices need not start at 1.

Because the data passed to an external function is often a subset of the full data set, array indices need not start at 1.

Note: Indices on separate arguments are not necessarily the same.

This might occur, for instance, with variables from different data sets.

Because of this, we need to ask Ferret what the appropriate index values are for the result axes and for each axis of each argument. We also need to know whether the increment for each axis of each argument is 0 or 1. An increment of 0 would be returned, for example, as the Y axis increment of an argument which which was only defined on the X axis. The data for this argument would be replicated along the Y axis when needed in a calculation.

The following section of code from subtract.F retrieves the index and increment information:

```
...
CALL ef_get_res_subscripts(id, res_lo_ss, res_hi_ss, res_incr)
CALL ef_get_arg_subscripts(id, arg_lo_ss, arg_hi_ss, arg_incr)
...
```

Once we have this information we must make sure that we don't mix and match indices. It's possible that you can write code which will work in the very simplest cases but will fail when you try something like:

```
yes? let a = my_func(sst[d=1],airt[d=2])
yes? plot a[l=@ave]
```

The solution is straightforward if not very pretty: Assign a separate index to each axis of each argument and index them all separately. The code in ***subtract.F*** shows how to do it with two arguments:

```
...
i1 = arg_lo_ss(X_AXIS,ARG1)
i2 = arg_lo_ss(X_AXIS,ARG2)
DO 400 i=res_lo_ss(X_AXIS), res_hi_ss(X_AXIS)

   j1 = arg_lo_ss(Y_AXIS,ARG1)
   j2 = arg_lo_ss(Y_AXIS,ARG2)
   DO 300 j=res_lo_ss(Y_AXIS), res_hi_ss(Y_AXIS)

      k1 = arg_lo_ss(Z_AXIS,ARG1)
      k2 = arg_lo_ss(Z_AXIS,ARG2)
      DO 200 k=res_lo_ss(Z_AXIS), res_hi_ss(Z_AXIS)

      l1 = arg_lo_ss(T_AXIS,ARG1)
      l2 = arg_lo_ss(T_AXIS,ARG2)
      DO 100 l=res_lo_ss(T_AXIS), res_hi_ss(T_AXIS)


            IF ( arg_1(i1,j1,k1,l1) .EQ. bad_flag(1) .OR.
.                arg_2(i2,j2,k2,l2) .EQ. bad_flag(2) ) THEN

               result(i,j,k,l) = bad_flag_result

            ELSE

               result(i,j,k,l) = arg_1(i1,j1,k1,l1) -
.                    arg_2(i2,j2,k2,l2)

            END IF


            l1 = l1 + arg_incr(T_AXIS,ARG1)
            l2 = l2 + arg_incr(T_AXIS,ARG2)
```

```
100           CONTINUE

              k1 = k1 + arg_incr(Z_AXIS,ARG1)
              k2 = k2 + arg_incr(Z_AXIS,ARG2)
200         CONTINUE

            j1 = j1 + arg_incr(Y_AXIS,ARG1)
            j2 = j2 + arg_incr(Y_AXIS,ARG2)
300      CONTINUE

         i1 = i1 + arg_incr(X_AXIS,ARG1)
         i2 = i2 + arg_incr(X_AXIS,ARG2)
400   CONTINUE
      ...
```

Ch11 Sec5.3. Reduced axes

For external functions we introduce the concept of "axis reduction." The result of an external function will have axes which are either RETAINED or REDUCED with respect to the argument axes from which they are inherited. By default, all result axes have their axis reduction flag set to RETAINED. Every result axis which has it axis inheritance flag set to IMPLIED_BY_ARGS will have the same extent (context) as the argument axis from which it inherits. Setting the axis reduction flag to REDUCED means that the result axis is reduced to a point by the external function.

The axis reduction flag only needs to be applied when the result is reduced to a point but SET REGION information should still be applied to the external function arguments. (e.g. a function returning a status flag) In such a case the result axes should be IMPLIED_BY_ARGS and REDUCED. (as opposed to NORMAL and RETAINED)

The ***percent_good_t.F*** function is a good example of where the axis reduction flag needs to be set. This function takes a 4D region of data and returns a time series of values representing the percentage of good data at each time point. Inside the ***percent_good_t_init*** subroutine we see that the X, Y and Z axes are reduced with respect to the incoming argument:

```
* ************************
* USER CONFIGURABLE PORTION
*
*
      CALL ef_set_desc(id,
     . '(demonstration function) returns % good data at each time' )
      CALL ef_set_num_args(id, 1)
      CALL ef_set_axis_inheritance(id, IMPLIED_BY_ARGS,
     .     IMPLIED_BY_ARGS, IMPLIED_BY_ARGS, IMPLIED_BY_ARGS)
      CALL ef_set_axis_reduction(id, REDUCED, REDUCED, REDUCED,
     .     RETAINED)
      CALL ef_set_piecemeal_ok(id, NO, NO, NO, NO)
      arg = 1
      CALL ef_set_arg_name(id, arg, 'A')
      CALL ef_set_arg_desc(id, arg, 'data to be checked')
      CALL ef_set_axis_influence(id, arg, YES, YES, YES, YES)
*
*
```

```
* USER CONFIGURABLE PORTION
* ************************
```

This arrangement allows the user to specify an X/Y/Z region of interest and have this region information used when the argument is passed to the function. If we had specified X/Y/Z as NORMAL axes, Ferret would have understood this to mean that all region information for these three axes can be ignored when the percent_good_t function is called. This is not what we want.

Ch11 Sec5.4. String Arguments

Ferret can pass strings to external functions. This may be useful if you are writing external functions to write a new output format, for example, and wish to pass the output filename as an argument.

By default, all arguments are assumed to be of type ***FLOAT_ARG***. In the ***~init*** subroutine, the external function must tell Ferret which arguments are to be handled as strings:

```
arg = 1
CALL ef_set_arg_type(id, arg, STRING_ARG)
CALL ef_set_arg_name(id, arg, 'message')
CALL ef_set_arg_desc(id, arg, 'String to be written when executing.')
CALL ef_set_axis_influence(id, arg, YES, YES, YES, YES)
```

In the ***~compute*** subroutine, a pointer to the string argument is passed in and dimensioned as any other argument. A text variable must be declared and a utility function is used to get the actual text string. As an example:

```
      SUBROUTINE string_args_compute(id, arg_1, arg_2, result)

      INCLUDE 'ferret_cmn/EF_Util.cmn'
      INCLUDE 'ferret_cmn/EF_mem_subsc.cmn'

      INTEGER id

      REAL bad_flag(1:EF_MAX_ARGS), bad_flag_result
      REAL arg_1(mem1lox:mem1hix, mem1loy:mem1hiy,
     .           mem1loz:mem1hiz, mem1lot:mem1hit)

      REAL arg_2(mem2lox:mem2hix, mem2loy:mem2hiy,
     .           mem2loz:mem2hiz, mem2lot:mem2hit)
      REAL result(memreslox:memreshix, memresloy:memreshiy,
     .            memresloz:memreshiz, memreslot:memreshit)

      INTEGER res_lo_ss(4), res_hi_ss(4), res_incr(4)
      INTEGER arg_lo_ss(4,1:EF_MAX_ARGS), arg_hi_ss(4,1:EF_MAX_ARGS),
     .     arg_incr(4,1:EF_MAX_ARGS)

      CHARACTER arg1_text*160

* ************************
* USER CONFIGURABLE PORTION
*
      INTEGER i,j,k,l
```

```
      INTEGER i1, j1, k1, l1

      CALL ef_get_arg_string(id, 1, arg1_text)

      WRITE(6,49) arg1_text
49    FORMAT ('The text for arg1 is : ''',a,'''')

      …
```

Ch11 Sec6. UTILITY FUNCTIONS

The lists below describe the utility functions built into Ferret which are available to the external function writer. These are used to set parameters associated with the external function and to retrieve information provided by Ferret. (Input variables, sending information to Ferret, are in plain type and output variables, getting information from Ferret, are in italic.)

Ch11 Sec6.1. EF_Util.cmn

External functions need to include the ***EF_Util.cmn*** file in each subroutine in order to use various pre-defined parameters. These parameters are defined in the table below:

Parameters defined in ***EF_Util.cmn***

To make the code more readable:

X_AXIS (=1)	***ARG1 (=1)***	***ARG5 (=5)***	***ARG9 (=9)***
Y_AXIS (=2)	***ARG2 (=3)***	***ARG6 (=6)***	***YES (=2)***
Z_AXIS (=3)	***ARG3 (=3)***	***ARG7 (=7)***	***NO (=0)***
T_AXIS (=4)	***ARG4 (=4)***	***ARG8 (=8)***	

Internal parameters for Ferret:

CUSTOM	result axis is defined by the external function
IMPLIED_BY_ARGS	result axis is inherited from one (or more) of the arguments
NORMAL	this axis does not exist in the result
ABSTRACT	result axis is an indexed axis [1:N]
RETAINED	result axis has same extent as argument axis
REDUCED	result axis is reduced to a point

Ch11 Sec6.2. Available utility functions

Setting Parameter

- ef_set_desc(id, desc) (p. 308)
- ef_set_num_args(id, num) (p. 308)
- ef_set_piecemeal_ok(id, Xyn, Yyn, Zyn, Tyn) (p. 309)
- ef_set_axis_inheritance(id, Xsrc, Ysrc, Zsrc, Tsrc) (p. 308)
- ef_set_arg_name(id, arg, name) (p. 309)
- ef_set_arg_desc(id, arg, desc) (p. 309)
- ef_set_arg_unit(id, arg, unit) (p. 309)
- ef_set_arg_type(id, arg, type) (p. 310)
- ef_set_axis_influence(id, arg, Xyn, Yyn, Zyn, Tyn) (p. 310)
- ef_set_axis_reduction(id, Xred, Yred, Zred, Tred) (p. 311)
- ef_set_axis_extend(id, arg, axis, lo_amt, hi_amt) (p. 310)
- ef_set_axis_limits(id, axis, lo, hi) (p. 311)
- ef_set_custom_axis(id, axis, lo, hi, delta, unit, modulo) (p. 311)
- ef_set_num_work_arrays(id, num) (p. 312)
- ef_set_work_array_dims(id, array, Xlo, Ylo, Zlo, Tlo, Xhi, Yhi, Zhi, Thi) (p. 312)

Getting Information

For all calculations

- ef_get_res_subscripts(id, res_lo_ss, res_hi_ss, res_incr) (p. 312)
- ef_get_arg_subscripts(id, arg_lo_ss, arg_hi_ss, arg_incr) (p. 315)
- ef_get_bad_flags(id, bad_flag, bad_flag_result) (p. 316)

Text

- ef_get_arg_info(id, arg, name, title, units) (p. 313)
- ef_get_arg_string(id, arg, text) (p. 313)
- ef_get_axis_info(id, arg, name, units, bkwd, modulo, regular) (p. 314)
- ef_get_axis_dates(id, arg, tax, numtimes, datebuf) (p. 314)

Values

- ef_get_arg_ss_extremes(id, arg, ss_min, ss_max) (p. 316)
- ef_get_coordinates(id, arg, axis, lo, hi, coords) (p. 317)
- ef_get_box_size(id, arg, axis, lo, hi, size) (p. 318)
- ef_get_box_limits(id, arg, axis, lo, hi, lo_lims, hi_lims) (p. 319)
- ef_get_one_val(id, arg, value) (p. 319)

Other

- ef_version_test(version) (p. 321)
- ef_bail_out(id, text) (p. 321)

Ch11 Sec6.2.1. ef_set_desc(id, desc)

Assign a text string description to the external function.

Input arguments:
1. **INTEGER id:** external function's ID number
2. **CHARACTER*(*) desc:** description of this function

Ch11 Sec6.2.2. ef_set_num_args(id, num)

Specify the number of arguments this function will accept. The maximum number of arguments allowed is 9

Input arguments:
1. **INTEGER id:** external function's ID number
2. **INTEGER num:** number of arguments for this function

Ch11 Sec6.2.3. ef_set_axis_inheritance(id, Xsrc, Ysrc, Zsrc, Tsrc)

Specify where the result axes will come from. The acceptable values for each axis will be one of:

CUSTOM	result axis is defined by the external function
IMPLIED_BY_ARGS	result axis is inherited from one (or more) of the arguments
NORMAL	this axis does not exist in the result
ABSTRACT	result axis is an indexed axis [1:N]

Input arguments:
1. **INTEGER id:** external function's ID number
2. **INTEGER Xsrc:** inheritance flag for the X axis
3. **INTEGER Ysrc:** inheritance flag for the Y axis
4. **INTEGER Zsrc:** inheritance flag for the Z axis
5. **INTEGER Tsrc:** inheritance flag for the T axis

Ch11 Sec6.2.4. ef_set_piecemeal_ok(id, Xyn, Yyn, Zyn, Tyn)

Tell Ferret whether it is ok to break up calculations along a particular axis. (Not implemented as of Ferret v5.22, EF version 1.3)

Input arguments:
1. **INTEGER id:** external function's ID number
2. **INTEGER Xyn:** yes/no flag for the X axis
3. **INTEGER Yyn:** yes/no flag for the Y axis
4. **INTEGER Zyn:** yes/no flag for the Z axis
5. **INTEGER Tyn:** yes/no flag for the T axis

Ch11 Sec6.2.5. ef_set_arg_name(id, arg, name)

Assign a text string name to an argument.

Input arguments:
1. **INTEGER id:** external function's ID number
2. **INTEGER arg:** argument number
3. **CHARACTER*(*) name:** argument name

Ch11 Sec6.2.6. ef_set_arg_desc(id, arg, desc)

Assign a text string description to an argument.

Input arguments:
1. **INTEGER id:** external function's ID number
2. **INTEGER arg:** argument number
3. **CHARACTER*(*) desc:** argument description

Ch11 Sec6.2.7. ef_set_arg_unit(id, arg, unit)

Assign a text string to an argument's units.

Input arguments:
1. **INTEGER id:** external function's ID number
2. **INTEGER arg:** argument number
3. **CHARACTER*(*) unit:** unit description

Ch11 Sec6.2.8. ef_set_arg_type(id, arg, type)

Specify the type of an argument as either ***FLOAT_ARG*** or ***STRING_ARG***. In the ***~_compute*** subroutine, the ***ef_get_arg_string()*** function is used to obtain the desired text string.

Input arguments:
1. **INTEGER id:** external function's ID number
2. **INTEGER arg:** argument number
3. **INTEGER type:** either FLOAT_ARG or STRING_ARG

Ch11 Sec6.2.9. ef_set_axis_extend(id, arg, axis, lo_amt, hi_amt)

Tell Ferret to extend the range of data passed for an argument. This is useful for cases like smoothers where the result at a particular point depends upon a range of input values around that point.

Input arguments:
1. **INTEGER id:** external function's ID number
2. **INTEGER arg:** argument number
3. **INTEGER axis:** axis number
4. **INTEGER lo_amt:** extension to the lo range (–1 means get one more point than in the result)
5. **INTEGER hi_amt:** extension to the hi range (+1 means get one more point than in the result)

Ch11 Sec6.2.10. ef_set_axis_influence(id, arg, Xyn, Yyn, Zyn, Tyn)

Specify whether this argument's axes "influence" the result axes. A value of ***YES*** for a particular axis means that the result should have the same axis as this argument. If the result should have the same axis as several input arguments, then each argument should specify ***YES*** for the axis in question. Note that ***ef_set_axis_inheritance*** must have specified ***IMPLIED_BY_ARGS*** for this axis.

Input arguments:
1. **INTEGER id:** external function's ID number
2. **INTEGER arg:** argument number
3. **INTEGER Xyn:** influence flag for the X axis
4. **INTEGER Yyn:** influence flag for the Y axis
5. **INTEGER Zyn:** influence flag for the Z axis
6. **INTEGER Tyn:** influence flag for the T axis

Ch11 Sec6.2.11. ef_set_axis_reduction(id, Xred, Yred, Zred, Tred)

Specify whether the result axes are RETAINED or REDUCED with respect to the argument axes from which they are inherited. Setting the axis reduction flag to REDUCED means that the result axis is reduced to a point by the external function. The axis reduction flag need only be set when the result is reduced to a point but SET REGION information should still be applied to the external function arguments.

Input arguments:
1. **`INTEGER id`**: external function's ID number
2. **`INTEGER Xred`**: reduction flag for the X axis
3. **`INTEGER Yred`**: reduction flag for the Y axis
4. **`INTEGER Zred`**: reduction flag for the Z axis
5. **`INTEGER Tred`**: reduction flag for the T axis

Ch11 Sec6.2.12. ef_set_axis_limits(id, axis, lo, hi)

Specify the lo and hi limits of an axis. (This is not needed for most functions and must appear in a separate subroutine named ***`~func_name~_result_limits(id)`***).

Input arguments:
1. **`INTEGER id:`** external function's ID number
2. **`INTEGER axis:`** axis number
3. **`INTEGER lo:`** index value of the lo range of this axis
4. **`INTEGER hi:`** index value of the hi range of this axis

Ch11 Sec6.2.13. ef_set_custom_axis(id, axis, lo, hi, delta, unit, modulo)

Create a custom axis. This is only used by functions which create a custom axis and must appear in a separate subroutine named ***`~func_name~_custom_axes(id)`***. See also the discussion of the custom_axis subroutine (p. 299)

Input arguments:
1. **`INTEGER id:`** external function's ID number
2. **`INTEGER axis:`** axis number
3. **`REAL lo:`** coordinate value of the lo range of this axis
4. **`REAL hi:`** coordinate value of the hi range of this axis
5. **`REAL delta:`** increment for this axis
6. **`CHARACTER*(*) unit:`** unit for this axis
7. **`INTEGER modulo:`** flag for modulo axes (1 = modulo)

Ch11 Sec6.2.14. ef_set_num_work_arrays(id, nwork)

Set the number of work arrays to be allocated. The maximum number of work arrays allowed is 9.

Input arguments:
1. **INTEGER id:** external function's ID number
2. **INTEGER nwork:** number of storage arrays

Ch11 Sec6.2.15. ef_set_work_array_dims(id, iarray, xlo, ylo, zlo, tlo, xhi, yhi, zhi, thi)

Set the working array axis lengths.

Input arguments:
1. **INTEGER id:** external function's ID number
2. **INTEGER iarray:** array number
3. **INTEGER xlo:** index value of the lo range of x axis
4. **INTEGER ylo:** index value of the lo range of y axis
5. **INTEGER zlo:** index value of the lo range of z axis
6. **INTEGER tlo:** index value of the lo range of t axis
7. **INTEGER xhi:** index value of the hi range of x axis
8. **INTEGER yhi:** index value of the hi range of y axis
9. **INTEGER zhi:** index value of the hi range of z axis
10. **INTEGER thi:** index value of the hi range of t axis

Ch11 Sec6.2.16. ef_get_res_subscripts(id, res_lo_ss, res_hi_ss, res_incr)

Return lo and hi indices and increments to be used in looping through the calculation of the result.

Input arguments:
1. **INTEGER id:** external function's ID number

Output arguments:
1. **INTEGER res_lo_ss(4):** the lo end indices for the X, Y, Z, T axes of the result
2. **INTEGER res_hi_ss(4):** the hi end indices for the X, Y, Z, T axes of the result
3. **INTEGER res_incr(4):** the increment to be applied to the X, Y, Z, T axes of the result

Sample code:

```
CALL ef_get_res_subscripts(id,
res_lo_ss, res_hi_ss, res_incr) ... DO 400 i=res_lo_ss(X_AXIS),
res_hi_ss(X_AXIS) DO 300 j=res_lo_ss(Y_AXIS), res_hi_ss(Y_AXIS)
... 300 CONTINUE 400 CONTINUE
```

Ch11 Sec6.2.17. ef_get_arg_info(id, iarg, arg_name, arg_title, arg_units)

Return strings describing argument: name, title, units.

Input arguments:
1. **INTEGER id:** external function's ID number
2. **INTEGER iarg:** argument number

Output arguments:
1. **CHARACTER*24 arg_name:** the name of the argument
2. **CHARACTER*128 arg_title:** title associated with the argument
3. **CHARACTER*32 arg_units:** the argument's units.

Ch11 Sec6.2.18. ef_get_arg_string(id, iarg, text)

This name is deprecated, and renamed to ef_get_one_arg_string.

Ch11 Sec6.2.19. ef_get_one_arg_string(id, iarg, text)

Return the string associated with an argument of type STRING_ARG. This routine like ef_get_one_val, may be called during the init phase of an external function.

Input arguments:
1. **INTEGER id:** external function's ID number
2. **INTEGER iarg:** argument number

Output arguments:
1. **CHARACTER*(*) text:** the actual text string for the argument

Sample code:

```
      ...
      CHARACTER arg_text*160

* ************************
* USER CONFIGURABLE PORTION
*
```

```
*
      INTEGER i,j,k,l
      INTEGER i1, j1, k1, l1

      CALL ef_get_arg_string(id, 1, arg_text)
      WRITE(6,49) arg_text
 49   FORMAT ('The text is : ''',a,'''')

      …
```

Ch11 Sec6.2.20. ef_get_axis_info(id, iarg, axname, ax_units, backward, modulo, regular)

Return strings describing argument: name, title, units.

Input arguments:
1. **`INTEGER id:`** external function's ID number
2. **`INTEGER iarg:`** argument number

Output arguments:
1. **`CHARACTER*16 ax_name(4):`** the name of the four axes
2. **`CHARACTER*16 ax_units(4):`** units of the four axes
3. **`LOGICAL backward(4):`** true if axis is backward axis
4. **`LOGICAL modulo(4):`** true if axis is modulo axis
5. **`LOGICAL regular(4):`** true if axis is regular axis

Ch11 Sec6.2.21. ef_get_axis_dates(id, iarg, taxis, numtimes, datebuf)

Returns the string date buffer associated with the time axis of an argument.

Input arguments:
1. **`INTEGER id:`** external function's ID number
2. **`INTEGER iarg:`** argument number
3. **`REAL*8 taxis(numtimes):`** time axis coordinate values
4. **`INTEGER numtimes:`** number of time

Output arguments:
1. **`CHARACTER*20 datebuf(numtimes):`** the string-date buffer for each time.

Ch11 Sec6.2.22. ef_get_axis_calendar(id, iarg, calname, yrdays, nmonths, days_in_month)

Returns the calendar name, days per year, number of months, and days per month associated with the time axis of an argument.

Input arguments:
1. **INTEGER id:** external function's ID number
2. **INTEGER iarg:** argument number

Output arguments:
1. **CHARACTER calname:** the string name of the axis
2. **REAL yrdays:** the number of days in the year, including fractional days as in 365.2425 for the standard calendar
3. **INTEGER nmonths:** the number of months in the year. Currently Ferret has only 12-month years defined
4. **INTEGER days_in_month(12):** The number of days in each month

Ch11 Sec6.2.23. ef_get_arg_subscripts(id, arg_lo_ss, arg_hi_ss, arg_incr)

Return lo and hi indices and increments to be used in looping through the calculation of the result.. See the discussion under custom_axis (p. 299) if you call ef_get_arg_subscripts to generate a custom axis.

Input arguments:
1. **INTEGER id:** external function's ID number

Output arguments:
1. **INTEGER arg_lo_ss(4,EF_MAX_ARGS):** the lo end indices for the X, Y, Z, T axes of each argument
2. **INTEGER arg_hi_ss(4,EF_MAX_ARGS):** the hi end indices for the X, Y, Z, T axes of each argument
3. **INTEGER arg_incr(4,EF_MAX_ARGS):** the increment to be applied to the X, Y, Z, T axes of each argument

Sample code:

```
INTEGER i,j,k,l
INTEGER i1, j1, k1, l1
INTEGER i2, j2, k2, l2

CALL ef_get_res_subscripts(id, res_lo_ss, res_hi_ss, res_incr)
CALL ef_get_arg_subscripts(id, arg_lo_ss, arg_hi_ss, arg_incr)
CALL ef_get_bad_flags(id, bad_flag, bad_flag_result)

i1 = arg_lo_ss(X_AXIS,ARG1)
i2 = arg_lo_ss(X_AXIS,ARG2)

DO 400 i=res_lo_ss(X_AXIS), res_hi_ss(X_AXIS)
  …
  i1 = i1 + arg_incr(X_AXIS,ARG1)
  i2 = i2 + arg_incr(X_AXIS,ARG2)
400 CONTINUE
```

Ch11 Sec6.2.24. ef_get_arg_ss_extremes(id, num_args, ss_min, ss_max)

Return the maximum and minim index values for all the arguments. These define the domain of the data.

Input arguments:
1. **INTEGER id:** external function's ID number
2. **INTEGER num_args:** number of arguments for which to return index extremes

Output arguments:
1. **INTEGER ss_min(4,EF_MAX_ARGS):** the minimum indices for the X, Y, Z, T axes of each argument
2. **INTEGER ss_max(4,EF_MAX_ARGS):** the maximum indices for the X, Y, Z, T axes of each argument

Example:

```
INTEGER id, num_args
INTEGER  ss_min(4,EF_MAX_ARGS), ss_max(4,EF_MAX_ARGS)
num_args = 3
CALL ef_get_arg_ss_extremes(id, num_args, ss_min, ss_max)
```

Ch11 Sec6.2.25. ef_get_bad_flags(id, bad_flag, bad_flag_result)

Return the missing value flags for each argument and for the result.

Input arguments:
1. **INTEGER id:** external function's ID number

Output arguments:
1. **REAL bad_flag(EF_MAX_ARGS):** missing value flags for each argument
2. **REAL bad_flag_result:** missing value flag for the result

Sample code:

```
CALL ef_get_res_subscripts(id, res_lo_ss, res_hi_ss, res_incr)
CALL ef_get_arg_subscripts(id, arg_lo_ss, arg_hi_ss, arg_incr)
CALL ef_get_bad_flags(id, bad_flag, bad_flag_result)

...

IF ( arg_1(i1,j1,k1,l1) .EQ. bad_flag(ARG1) ) THEN
  result(i,j,k,l) = bad_flag_result
ELSE
  ...
```

Ch11 Sec6.2.26. ef_get_coordinates(id, arg, axis, lo, hi, coords)

Return the "world coordinates" associated with a particular arg, axis and lo:hi range.

Input arguments:

1. **INTEGER id:** external function's ID number
2. **INTEGER arg:** argument number
3. **INTEGER axis:** axis number
4. **INTEGER lo:** lo index of desired range
5. **INTEGER hi:** hi index of desired range

Output arguments:

1. **REAL*8 coords(*):** array of "world coordinate" values (NB_ these values are associated with index values lo:hi but are returned as ***coords(1:hi-lo)***.)

Sample code: in the work_size subroutine, define twice as many elements as coordinates so as to have storage for REAL*8 numbers

```
*
      SUBROUTINE myfcn_work_size(id)
      INCLUDE 'ferret_cmn/EF_Util.cmn'
      INCLUDE 'ferret_cmn/EF_mem_subsc.cmn'
      INTEGER id

*  Set the work arrays,  X/Y/Z/T dimensions

      INTEGER nxout, nx2
      INTEGER arg_lo_ss(4,1:EF_MAX_ARGS), arg_hi_ss(4,1:EF_MAX_ARGS),
          arg_incr(4,1:EF_MAX_ARGS)

      CALL ef_get_arg_subscripts(id, arg_lo_ss, arg_hi_ss, arg_incr)

      nxout = 1 + arg_hi_ss(X_AXIS,ARG4) - arg_lo_ss(X_AXIS,ARG4)
      nx2 = nxout* 2

* Define work array XAX

      CALL ef_set_work_array_dims (id, 1, 1, 1, 1, 1, nx2, 1, 1, 1)
      RETURN
      END
```

In the compute subroutine, dimension the REAL*8 coordinate array with half the wkr1hix dimension (wrk1lox:wrk1hix, etc are defined by the work_size subroutine)

```
      SUBROUTINE myfcn_compute(id, arg_1, arg_2, result,  xax)
...
      REAL arg_1(mem1lox:mem1hix, mem1loy:mem1hiy, mem1loz:mem1hiz,
     .     Mem1lot:mem1hit)
      REAL result(memreslox:memreshix, memresloy:memreshiy,
     .     memresloz:memreshiz, memreslot:memreshit)

      INTEGER res_lo_ss(4), res_hi_ss(4), res_incr(4)
      INTEGER arg_lo_ss(4,EF_MAX_ARGS), arg_hi_ss(4,EF_MAX_ARGS),
```

```
     .     Arg_incr(4,EF_MAX_ARGS)

C  Dimension the work array: X dimension was  defined twice as large
C  as the # coordinates, for double precision work array.

      REAL*8 xax(wrk1lox:wrk1hix/2, wrk1loy:wrk1hiy,
     .     wrk1loz:wrk1hiz, wrk1lot:wrk1hit)
…

      CALL ef_get_res_subscripts(id, res_lo_ss, res_hi_ss, res_incr)
      CALL ef_get_arg_subscripts(id, arg_lo_ss, arg_hi_ss, arg_incr)
      CALL ef_get_bad_flags(id, bad_flag, bad_flag_result)

      CALL ef_get_coordinates(id, ARG1, X_AXIS, arg_lo_ss(X_AXIS,
     .  ARG1), arg_hi_ss(X_AXIS, ARG1), xax )

…

     dummy = 1
     DO 30 i = arg_lo_ss(Y_AXIS, ARG1), arg_hi_ss(Y_AXIS, ARG1)
       cstr(i) = 1.0 / cos( xax(dummy) * (1.0/radian) )
       dummy = dummy + 1
30   CONTINUE
```

Ch11 Sec6.2.27. ef_get_box_size(id, arg, axis, lo, hi, size)

Return the box sizes (in "world coordinates") associated with a particular arg, axis and lo:hi range.

Input arguments:

1. **`INTEGER id:`** external function's ID number
2. **`INTEGER arg:`** argument number
3. **`INTEGER axis:`** axis number
4. **`INTEGER lo:`** lo index of desired range
5. **`INTEGER hi:`** hi index of desired range

Output arguments:

1. **`REAL size(*):`** array of box size values (NB_ these values are associated with index values lo:hi but are returned as ***`coords(1:hi-lo)`***.)

Sample code:

```
REAL tk_y(wrk1lox:wrk1hix, wrk1loy:wrk1hiy/2,
     .     wrk1loz:wrk1hiz, wrk1lot:wrk1hit)
REAL tk_dx(wrk2lox:wrk2hix, wrk2loy:wrk2hiy,
     .     wrk2loz:wrk2hiz, wrk2lot:wrk2hit)

INTEGER dummy

…

CALL ef_get_res_subscripts(id, res_lo_ss, res_hi_ss, res_incr)
CALL ef_get_arg_subscripts(id, arg_lo_ss, arg_hi_ss, arg_incr)
CALL ef_get_bad_flags(id, bad_flag, bad_flag_result)
CALL ef_get_coordinates(id, ARG1, Y_AXIS, arg_lo_ss(Y_AXIS, ARG1),
```

```
.     arg_hi_ss(Y_AXIS, ARG1), tk_y )
CALL ef_get_box_size(id, ARG1, X_AXIS, arg_lo_ss(X_AXIS, ARG1),
.     arg_hi_ss(X_AXIS, ARG1), tk_dx )

…

dummy = 1
DO 20 i = arg_lo_ss(X_AXIS, ARG1), arg_hi_ss(X_AXIS, ARG1)
  dxt4r(i) = 1.0 / ( 4.0 * tk_dx(dummy) * radius/radian )
  dummy = dummy + 1
20 CONTINUE
```

Ch11 Sec6.2.28. ef_get_box_limits(id, arg, axis, lo, hi, lo_lims, hi_lims)

Return the box limits (in "world coordinates") associated with a particular arg, axis and lo:hi range.

Input arguments:
1. **`INTEGER id:`** external function's ID number
2. **`INTEGER arg:`** argument number
3. **`INTEGER axis:`** axis number
4. **`INTEGER lo:`** lo index of desired range
5. **`INTEGER hi:`** hi index of desired range

Output arguments:
1. **`REAL lo_lims(*):`** array of box lower limit values (NB_ these values are associated with index values lo:hi but are returned as ***`coords(1:hi-lo)`***.)
2. **`REAL hi_lims(*):`** array of box upper limit values (NB_ these values are associated with index values lo:hi but are returned as ***`coords(1:hi-lo)`***.)

Ch11 Sec6.2.29. ef_get_one_val(id, arg, value)

Return the value of 1×1×1×1 variable.

Input arguments:
1. **`INTEGER id`**: external function's ID number
2. **`INTEGER arg`**: argument number

Output arguments:
1. **`REAL`** ***`value`*** : The value of the variable

Ch11 Sec6.2.30. ef_get_string_arg_element(id, arg, i,j,k,l, str_arg, slen, text)

Return one single string element from a string variable.

Input arguments:
1. **`INTEGER id`**: external function's ID number
2. **`INTEGER arg`**: argument number
3. **`REAL str_arg`**: the argument, e.g. arg_1, pointer to the string argument.
4-7. **`INTEGER i,j,k,l:`** the indices of the string element to return

Output arguments:
1. **`INTEGER slen`** : The length of the string element
2. **`CHARACTER text`** : The string. If the string variable you have declared in the external function is too short for the string in the argument, the variable text will be truncated, but the value of slen will contain the length of the string in the argument.

Ch11 Sec6.2.31. ef_get_string_arg_element_len (id, arg, str_arg, i,j,k,l, slen)

Return the length of a single string element from a string variable.

Input arguments:
1. **`INTEGER id`**: external function's ID number
2. **`INTEGER arg`**: argument number
3. **`REAL str_arg`**: the argument, e.g. arg_1, pointer to the string argument.
4-7. **`INTEGER i,j,k,l:`** the indices of the string element to return

Output arguments:
1. **`INTEGER slen`** : The length of the string element

Ch11 Sec6.2.32. ef_get_string_arg_max_len (id, arg, str_arg, slen)

Return the length of a single string element from a string variable.

Input arguments:
1. **`INTEGER id`**: external function's ID number
2. **`INTEGER arg`**: argument number
3. **`REAL str_arg`**: the argument, e.g. arg_1, pointer to the string argument.

Output arguments:
1. **`INTEGER slen`** : The maximum length of the strings in the array

Ch11 Sec6.2.33. ef_version_test (version)

Return the version number of the external functions code that is in place.

Output argument:
1. **REAL** ***version*** : The version number

Ch11 Sec6.2.34. ef_bail_out(id, text)

Bail out of an external function, returning to Ferret and issuing a message to the user.

Input arguments:
1. **INTEGER id**: external function's ID number
2. **INTEGER text**: text string to output.

The bail-out message looks like this, where the text supplied in the call to ef_bail_out is on the second line:

Bailing out of external function "ffta":
Time axis must be a regular axis
**ERROR: : error in external function

Part II: COMMANDS REFERENCE

Ref Sec1. ALIAS

An alias for DEFINE ALIAS (p. 337).

Ref Sec2. CANCEL

Cancels a program state or definition—generally paired with a SET or DEFINE command. See commands SET (p. 395) and DEFINE (p. 337) for further information.

Arguments:

The arguments, which are names of variables, data sets, or other definitions can be specified using wildcards. The * wildcard matches any number of characters in the name; the ? wildcard matches exactly one character.

Ref Sec2.1. CANCEL ALIAS

Cancels a user-defined command alias.

```
yes? CANCEL ALIAS ALIAS_NAME
```

The command UNALIAS is an alias for CANCEL ALIAS.

Ref Sec2.2. CANCEL ATTRIBUTE

/OUTPUT /DATASET

Removes an attribute from a variable, or if /OUTPUT is specified, sets the output flag so that the attribute is not written to netCDF files when the variable is written. (See the section on access to dataset and variable attributes, p. 65)

```
yes? CANCEL ATTRIBUTE[/qualifiers] varname.attname
```

Examples:

This command removes the long_name attribute entirely from the variable sst.

```
yes? CANCEL ATTRIBUTE sst.long_name
```

These commands set a flag so that the AXIS attribute is not written for the coordinate variables COADSX, COADSY, and TIME are written to an netCDF file. This applies to the default dataset

```
yes? CANCEL ATTRIBUTE/OUTPUT (COADSX).AXIS
yes? CANCEL ATTRIBUTE/OUTPUT (COADSY).AXIS
yes? CANCEL ATTRIBUTE/OUTPUT (TIME).AXIS
```

Ref Sec2.3. CANCEL AXIS

/MODULO /DEPTH /ALL /STRIDE

CANCEL AXIS forms the complement to DEFINE AXIS, or SET AXIS It is also applicable to "persistent" axes which are defined by netCDF files such as climatological_axes.cdf -- axes which are not associated with any variables in the netCDF file, itself, and are not automatically deleted when the data set is canceled.

```
yes? CANCEL AXIS AXIS_NAME
```

Attempts to CANCEL AXIS on a axis which is used by a variable in a currently open data set will be rejected with a message indicating the reason.

Command qualifiers for CANCEL AXIS:

CANCEL AXIS/MODULO

Cancels the modulo nature of a user-defined axis.

```
yes? CANCEL AXIS/MODULO my_x_axis
```

or

```
yes? CANCEL AXIS/MODULO my_t*
```

CANCEL AXIS/DEPTH

Cancels the depth setting of a Z axis, which may have been set with a positive="down" attribute in a netCDF file, or for a user-defined axis with a DEFINE AXIS/DEPTH or SET AXIS/DEPTH command. If applied to an X, Y, or T axis, this qualifier is ignored.

```
yes? CANCEL AXIS/DEPTH my_z_axis
```

CANCEL AXIS/ALL

Cancels all axes that have been defined by the user, and restores any coordinate storage that was used to define irregular axes. It does not cancel axes defined when a data set is opened.

CANCEL AXIS/STRIDE

Cancels the strided behavior of an axis that was set to native striding by a SET AXIS/STRIDE command. See the discussion of netCDF strides (p. 35.)

CANCEL DATA_SET

/ALL /NOERROR

Removes the specified data set from the list of available sets.

```
yes? CANCEL DATA_SET dset1, dset2, ..., dsetn
       where each dset may be the name or number of a data set; or
yes? CANCEL DATA/ALL
```

(See also SET DATA_SET, p. 398, and SHOW DATA SET, p. 434.)

Command qualifiers for CANCEL DATA_SET:

CANCEL DATA/ALL

Eliminates all data sets from the list of accessible data sets.

CANCEL DATA/NOERROR

Suppresses the error message otherwise generated when a data set that was never set is canceled. Useful in GO scripts for closing data sets that may have been opened in previous usage of the script.

Note that if a grid or axis from a netCDF file is used in the definition of a LET-defined variable (e.g. LET my_X = X[g=sst[D=coads_climatology]]) that variable definition will be invalidated when the data set is canceled (CANCEL DATA coads_climtology, in the preceding example). There is a single exception to this behavior: netCDF files such as climtological_axes.cdf, which define grids or axes that are not actually used by any variables. These grids and axes will remain defined even after the data set, itself, has been canceled. They may be deleted with explicit use of CANCEL GRID or CANCEL AXIS.

CANCEL EXPRESSION

Un-specifies the current context expression. Ferret's "action" commands can be issued without an argument (e.g., ***yes? PLOT***), in which case Ferret uses the current context expression. This expression is either the argument of the most recent action command, or an expression set explicitly with SET EXPRESSION.

```
yes? CANCEL EXPRESSION
```

The qualifier /ALL can be used with this command, but it exists for compatibility purposes only and has no effect.

CANCEL GRID

CANCEL GRID forms the complement to DEFINE GRID It is also applicable to "persistent" grids which are defined by netCDF files such as climatological_axes.cdf -- grids which are not associated with any variables in the netCDF file, itself, and are not automatically deleted when the data set is canceled.

Attempts to CANCEL GRID on a grid or axis which is used by a variable in a currently open data set will be rejected with a message indicating the reason.

CANCEL LIST

/ALL /APPEND /FILE /FORMAT /HEADING /PRECISION

Toggles the effects of the SET LIST command. See command SET LIST (p. 407).

```
yes? CANCEL LIST[/qualifiers]
```

Command qualifiers for: CANCEL LIST

CANCEL LIST/ALL

Restores all aspects of the LIST command to their default behavior.

CANCEL LIST/APPEND

Resets the listed output to NOT append to existing file.

CANCEL LIST/FILE

Resets the listed output to automatic file naming.

CANCEL LIST/FORMAT

Resets the listed output to its default formatting.

CANCEL LIST/HEAD

Instructs listed output to omit the descriptive data header.

CANCEL LIST/PRECISION

Resets the precision of listed data to 4 significant digits.

Ref Sec2.4. CANCEL MEMORY

/ALL /PERMANENT /TEMPORARY

Clears data currently cached in memory.

```
yes? CANCEL MEMORY[/qualifier]
```

Use this command to save memory space—by clearing data as soon as it is no longer needed virtual memory requirements can be reduced. This is especially useful for efficient batch processing. Default is CANCEL MEMORY/TEMPORARY.

Example:

To produce an animation using minimal virtual memory try:

```
yes? REPEAT/T=lo:hi:delta GO min_mem_movie
```

Where the file min_mem_movie.jnl contains

```
CONTOUR/FRAME  temp[Z=0]        ! contour plot
CANCEL MEMORY/ALL               ! clear memory for next time step
```

Command qualifiers for CANCEL MEMORY:

CANCEL MEMORY/ALL

Clears all variables stored in memory.

CANCEL MEMORY/PERMANENT

Clears all "permanent" variables stored in memory (i.e., variables loaded into memory with LOAD/PERMANENT).

CANCEL MEMORY/TEMPORARY (default)

Clears all non-permanent variables stored in memory.

CANCEL MODE

Sets the state of a mode to "canceled".

```
yes? CANCEL MODE mode_name
```

(See command SET MODE, p. 409, for descriptions of modes.)

Ref Sec2.5. CANCEL MOVIE

This command is unnecessary in Ferret version 3.1 and later; it is provided for compatibility with older versions of Ferret. It restores the default movie file name (ferret.mgm) but is not needed to conclude capturing graphics to a movie file.

```
yes? CANCEL MOVIE
```

The qualifier /ALL can be used with this command, but it exists for compatibility purposes only and has no effect.

Ref Sec2.6. CANCEL SYMBOL

/ALL

Deletes a user-defined symbol (string variable) definition.

```
yes? CANCEL STRING[/qualifier]   [symbol_name]
```

Command qualifiers for CANCEL SYMBOL:

CANCEL SYMBOL/ALL

Deletes all user-defined symbol definitions.

Examples:

```
yes? CANCEL SYMBOL my_x_label   !eliminate my_x_label from the
definitions
yes? CANCEL SYMBOL *x_label     !remove all strings ending in x_label
yes? CANCEL SYMBOL/ALL          !remove all user-defined symbols.
```

Ref Sec2.7. CANCEL REGION

/I/J/K/L /X/Y/Z/T /ALL

Cancels part or all of the current or named region.

```
yes? CANCEL REGION[/qualifier] [region_name]
```

Examples:

```
yes? CANCEL REGION               !clear the current region
yes? CANCEL REGION/T             !eliminate T from the current context
yes? CANCEL REGION reg1          !clear the region named "reg1"
```

Command qualifiers for CANCEL REGION:

CANCEL REGION/I /J /K /L /X /Y /Z /T

Eliminates I, J, K, L, X, Y, Z, or T axis information from current context or named region.

CANCEL REGION/ALL

Eliminates ALL stored region information (rarely used).

CANCEL VARIABLE

/ALL /DATASET

Deletes a user-defined variable definition.

```
yes? CANCEL VARIABLE[/qualifier]  [var_name]
```

Command qualifiers for CANCEL VARIABLE:

CANCEL VARIABLE/ALL

Deletes all user-defined variable definitions.

Examples:

```
yes? CANCEL VARIABLE my_sst  !eliminate my_sst from the definitions
yes? CANCEL VARIABLE *wind   !delete all variables ending in wind
yes? CANCEL VARIABLE tau?    !delete variables named tau plus one
character
yes? CANCEL VARIABLE/ALL     !delete all user-defined defined variables
```

CANCEL VARIABLE/DATASET

Deletes user define variables associated with the named dataset, which were defined by a DEFINE VARIABLE/DATASET command.

Ref Sec2.8. CANCEL VIEWPORT

Cancels a defined viewport or cancels use of viewports.

```
yes? CANCEL VIEWPORT view_name        !un-define view_name
yes? CANCEL VIEWPORT                  !return to full window output
```

Ref Sec2.9. CANCEL WINDOW

/ALL

Removes graphics window(s) from the screen.

```
yes? CANCEL WINDOW  n   !or
yes? CANCEL WINDOW/ALL
```

Command qualifiers for CANCEL WINDOW:

CANCEL WINDOW/ALL
Removes all graphics windows.

Ref Sec3. CONTOUR

/I/J/K/L /X/Y/Z/T /D /FILL /FRAME /KEY /LEVELS /LINE /NOAXIS /NOKEY /NOLABEL /OVERLAY /PALETTE /PATTERN /SIZE /SPACING /SIGDIG /PEN /SET_UP /TITLE /COLOR /TRANSPOSE /HLIMITS /VLIMITS /HLOG /VLOG /AXES /HGRATICULE /VGRATICULE /GRATICULE /MODULO

Produces a contour plot.

```
yes? CONTOUR[/qualifiers]   [expression]
```

In a curvilinear coordinate system (map projections)

```
yes? CONTOUR[/qualifiers]   expression, xcoords, ycoords (see p. 222)
```

Example:

```
yes? CONTOUR var1           !produce a contour plot of variable var1
yes? CONTOUR var1, xcoords, ycoords
                            !produce a contour plot of variable var1
                            ! using curvilinear coordinates
```

Parameters

Expressions may be any valid expression. See the chapter "Variables and Expressions", section "Expressions" (p. 73), for a definition of valid expressions. The expression will be inferred from the current context if omitted from the command line.

Command qualifiers for CONTOUR:

CONTOUR/I/J/K/L /X/Y/Z/T /OVERLAY /SET_UP /FRAME /D /TRANPOSE /FILL /LINE /NOLABEL /LEVELS /KEY /NOKEY /PALETTE /HLIMITS /VLIMITS /TITLE /COLOR /NOAXES /PATTERN /SIZE /SPACING /SIGDIG /PEN /AXES

CONTOUR/I/J=/K=/L=/X=/Y=/Z=/T=

Specifies value or range of axis subscripts (I, J, K, or L), or axis coordinates (X, Y, Z, or T) to be used when evaluating the expression being plotted.

CONTOUR/D=

Specifies the default data set to use when evaluating the expression being contoured.

CONTOUR/FILL (alias FILL)

Creates a color filled contour image.

CONTOUR/FRAME

Causes the graphic image produced by the command to be captured as an animation frame in the file specified by SET MOVIE. In general the FRAME command (p. 358) is more flexible and we recommend its use rather than this qualifier.

CONTOUR/FILL/KEY

Displays a color key for the palette used in a color-filled contour plot. Only valid in conjunction with /FILL (The key is displayed by default with CONTOUR/FILL or alias FILL). To control the color key position and labeling, see the command SHAKEY in the appendix, "Ferret Enhancements to PPLUS" (p. 561).

CONTOUR/FILL/KEY=CONTINUOUS

Chooses a continous color key for the palette used in a color-filled contour plot, without lines separating the colors. This option is particularly good for fill plots having many levels. Only valid in conjunction with /FILL

CONTOUR/LEVELS

Specifies the contour levels or how the levels will be determined. If the /LEVELS qualifier is omitted Ferret automatically selects reasonable contour levels.

See the chapter "Customizing Plots", section "Contouring" (p. 213) for examples and more documentation on /LEVELS and color_thickness indices. See also the demonstration "custom_contour_demo.jnl".

CONTOUR/LINE

Overlays contour lines on a color-filled plot. Valid only with /FILL (or as a qualifier to alias FILL). When /LINE is specified the color key, by default, is omitted. Use FILL/LINE/KEY to obtain both contour lines and a color key. The /SET_UP qualifier disables the action of /LINE. When using /SET_UP, follow the PPL CONTOUR (or PPL FILL) command with a CONTOUR/OVERLAY command to draw contour lines on the plot. In fact, CONTOUR/OVERLAY offers all the functionality of FILL/LINE and gives better control over the plot appearance.

CONTOUR/NOKEY

Turns off display of a color key for the palette used in a color-filled contour plot. Only valid in conjunction with /FILL (or with alias FILL).

CONTOUR/NOAXIS

Suppresses all axis lines, tics and labeling so that no box appears surrounding the contour plot. This is especially useful for map projection plots.

CONTOUR/NOLABELS

Suppresses all plot labels.

CONTOUR/OVERLAY

Causes the indicated expression to be overlaid on the existing plot.

Note (CONTOUR/OVERLAY with time axes):

A restriction in PPLUS requires that if time is an axis of the contour plot, the overlaid variable must share the same time axis encoding as the base plot variable. If this condition is not met, you may find that the overlaid contour fails to be drawn. The solution is to use the Ferret regridding capability to regrid the base plot variable and the overlaid plot variable onto the same time axis. See the section on overlaying with a time axis (p. 189).

CONTOUR/PALETTE=

Specifies a color palette (otherwise, the current default palette is used). Valid only with CONTOUR/FILL (or as a qualifier to the alias FILL). The file suffix *.spk is not necessary when specifying a palette. Try the Unix command `% Fpalette '*'` to see available palettes. See command PALETTE (p. 371) for more information.

Example:

```
yes? CONTOUR/FILL/PALETTE=land_sea  world_relief
```

The /PALETTE qualifier changes the current palette for the duration of the plotting command and then restores the previous palette. This behavior is not immediately compatible with the /SET_UP qualifier. See the PALETTE (p. 371) command for further discussion.

CONTOUR/PATTERN=

Specifies a pattern file (otherwise, the current default pattern specification is used). Valid only with CONTOUR/FILL (or as a qualifier to the alias FILL). The file suffix *.pat is not necessary when specifying a pattern. Try the Unix command % Fpattern '*' to see available patterns. See command PATTERN (p. 372) for more information.

CONTOUR/COLOR=

Sets line color (replaces the /PEN qualifier). The available color names are Black, Red, Green, Blue, LightBlue, and , Purple, and White (not case sensitive), corresponding to the /PEN values 1-6, respectively. (/COLOR also accepts numerical values.).

Example:

```
yes? CONTOUR/COLOR=red  sst
```

CONTOUR/PEN=
Sets line style for contour lines (same arguments as PLOT/LINE=). Argument can be an integer between 1 and 18; run ***GO line_samples*** to see the styles for color devices.

Example:

```
yes? CONTOUR/PEN=2  sst
```

CONTOUR/SIZE=
Controls the size of characters in the contour labels, using PLOT+ definition of "inches" .Default is 0.8' See example under CONTOUR/SPACING below.

CONTOUR/SIGDIG=
Sets the number of significant digits for contour labels. Default is 2. See example under CONTOUR/SPACING below.

CONTOUR/SPACING=
Sets spacing for contour lines, using PLOT+ definition of "inches". The default spacing is 5.0. (See the CONSET command in the on-line PLOT+ Users Guide)

Example o f CONTOUR/SIZE/SIGDIG/SPACING

```
yes? LET my_field = SIN(X[x=1:6:.1])*COS(Y[y=1:6:0.1])
yes? CONTOUR/SIGDIG=1/SIZE=0.15/SPACING=3 my_field
```

Specifies contour labels with a single significant digit using characters of height 0.15 "inches" at a nominal spacing of 3 "inches", consistent with the PLOT+ usage of "inches". (These are the same units as in, say, "ppl axlen 8,6", to specify plot axes of lengths 8 and 6 inches for horizontal and vertical axes, respectively.) Note that the PLOT+ CONPRE and CONPST commands are also useful (see p. 527), giving control over the text font and color used in the labels and adding units to the labels. For example, the commands

```
yes? PPL CONPRE @C002@CR
yes? PPL CONPST cm/sec
```

will transform the labels in the above CONTOUR example to red (002), Complex Roman font with a units label of "cm/sec".

CONTOUR/SET_UP
Performs all the internal preparations required by program Ferret for contouring but does not actually render output. The command PPL can then be used to make changes to the plot prior to producing output with the PPL CONTOUR command. This permits plot customizations that are not possible with Ferret command qualifiers. See the chapter "Customizing Plots", section "Contouring" (p. 213). Please note that /SET_UP disables the /LINE qualifier. When using /SET_UP certain plotting states may have to be reset manually.

CONTOUR/TITLE=
Allows user to specify a plot title (enclosed in quotation marks). Without this qualifier Ferret selects a title based on information about the expression.

CONTOUR/TRANSPOSE
Causes the horizontal and vertical axes to be interchanged. By default the X and T axes of the data are drawn horizontally on the plot and the Y and Z axes of the data are drawn vertically. For Y-Z plots the Z data axis is vertical by default.

Note that plots in the YT and ZT planes have /TRANSFORM applied by default in order to achieve a horizontal T axis. See /HLIMITS (below) for further details. Use /TRANSPOSE manually to reverse this effect.

CONTOUR/HLIMITS=
Specifies axis range and tic interval for the horizontal axis. Without this qualifier, Ferret selects reasonable values.

```
yes? CONTOUR/HLIMITS=lo_val:hi_val[:increment]  [expression]
```

The optional "increment" parameter determines tic mark spacing on the axis. If the increment is negative, the axis will be reversed.

This qualifier does not have any impact on the context of the expression being plotted. If data is on a modulo x axis but the arguments of the /HLIMITS qualifier represent a region outside the actual coordinates of the data, only the range including the actual coordinates is shown. Use /X=lo:hi or SET REGION or a context on the variable itself, var[X=lo:hi], to set the context for the expression.

The /HLIMITS and /VLIMITS qualifiers will retain their "horizontal" and "vertical" interpretations in the presence of the /TRANSPOSE qualifier. Thus, the addition of /TRANSPOSE to a plotting command mandates the interchange of "H" and "V" on the limits qualifiers.

CONTOUR/VLIMITS=
Specifies the axis range and tic interval for the vertical axis. See /HLIMITS (above)

CONTOUR/XLIMITS=/YLIMITS=
Note: XLIMITS and YLIMITS have been deprecated. Please use HLIMITS and VLIMITS instead.

CONTOUR/AXES[=top,bottom,left,right]

Turns plotting of individual axes off and on. This replaces the use of the "PPL AXSET" command. The syntax is

```
yes? CONTOUR/AXES[=top,bottom,left,right]  var
```

where the arguments are 1 to turn the axis on and 0 to turn it off. For example:

```
yes? CONTOUR/AXES=0,1,1,0  sst     ! Draws the bottom and left axes only
```

Note that contour plots with log axes can be drawn as explained in the FAQ, How can I make a 2D log (or log-log) plot?

CONTOUR/MODULO

Beginning with Ferret v5.81 the argument /MODULO for CONTOUR plots will draw modulo replications in longitude for curvilinear data in order to fill out the specified extent in the longitude direction. For instance, if the xcoords variable contains longitudes in the range -270:90 we can draw a plot with longitudes 0:360 with the command

```
yes? CONTOUR/HLIMITS=0:360/MODULO values, xcoords, ycoords
```

CONTOUR/GRATICULE[=line specifiers]

(Introduced in Ferret version 5.6) Turns on graticule lines for the horizontal and vertical axes. These are lines across the plot at the tic marks. /GRATICULE sets both horizontal and vertical lines; to set each separately see /HGRATICULE and /VGRATICULE, below. The syntax is

```
yes? CONTOUR/GRATICULE[=line or dash,COLOR=,THICKNESS=]  var
```

where the default is a thin, solid black line. The line colors available are Black, Red, Green, Blue, LightBlue, Purple, and White. The thickness codes are 1, 2, or 3 and as for plot lines, thickness=1 is a thin line, thickness=3 is the thickest, and THICK specified with no value defaults to thickness=2. For clarity the arguments to GRAT may be placed in parentheses

```
yes? CONTOUR/GRAT  sst     ! default graticules

yes? CONTOUR/GRAT=(LINE,COLOR=red,THIICK=3) sst

yes? CONTOUR/GRAT=(DASH,COLOR=lightblue) sst

yes? CONTOUR/FILL/GRAT=(DASH,COLOR=white) sst
```

The above commands make settings for the large tic marks. If small tic marks are being plotted on the axes, we can make settings for them as well using keywords SMALL and LARGE. Place all of the arguments for the /GRAT qualifier in double quotes. Note that the PPL AXNMTC command sets the plotting of small tics, and that small tics are used by default for many time axes.

```
yes? ppl axnmtc 2,2
yes? CONTOUR/GRAT="LARGE(COLOR=blue,thick),SMALL(COLOR=lightblue)" sst
```

CONTOUR/HGRATICULE[=line specifiers]/VGRATICULE[=line specifiers]

Turns on graticule lines and sets the line characteristics of the graticule for the horizontal or vertical axis separately. You may specify only one of /HGRAT or /VGRAT if desired. These are lines across the plot at the tic marks. The syntax is

```
     yes? CONTOUR/HGRATICULE[=line or dash,COLOR=,THICKNESS=]
/VGRATICULE=line or dash,COLOR=,THICKNESS=]  var
```

where the default is a thin, solid black line. The line colors available are Black, Red, Green, Blue, LightBlue, Purple, and White. The thickness codes are 1, 2, or 3 and as for plot lines, thickness=1 is a thin line, thickness=3 is the thickest, and THICK specified with no value defaults to thickness=2. For clarity the arguments to HGRAT may be placed in parentheses

```
yes? CONTOUR/HGRAT/VGRAT  sst    !this is equivalent to PLOT/GRAT

yes? CONTOUR/HGRAT=(LINE,COLOR=red,THIICK=3)/VGRAT=(color=green) sst

yes? CONTOUR/HGRAT=(DASH,COLOR=lightblue) sst  ! horizontal only
```

The above commands make settings for the large tic marks. If small tic marks are being plotted on the axes, we can make settings for them as well using keywords SMALL and LARGE. Place all of the arguments for the /HGRAT qualifier in double quotes. Note that the PPL AXNMTC command sets the plotting of small tics, and that small tics are used by default for many time axes.

```
yes? ppl axnmtc 2,2
yes? CONTOUR/HGRAT="LARGE(COLOR=blue,thick),SMALL(COLOR=lightblue)"
/VGRAT="LARGE(COLOR=blue,thick) sst
```

Ref Sec4. DEFINE

gratDefines a new alias, region, grid, axis, variable, or viewport.

Ref Sec4.1. DEFINE ALIAS

Defines an alias for a command. "ALIAS" is an alias for DEFINE ALIAS.

```
yes? DEFINE ALIAS NAME COMMAND
```

Example:

```
yes? DEFINE ALIAS SDF SHOW DATA/FULL
```

Ref Sec4.2. DEFINE ATTRIBUTE

/DATASET /TYPE /OUTPUT

Defines a new attribute for a variable. The variable may be from a netCDF dataset, a user-defined variable, or a variable from another type of dataset. (See the section on access to dataset and variable attributes, p. 65)

```
yes? DEFINE ATTRIBUTE[/qualifiers] varname.attname = expression
```

Example: add new attributes with DEFINE ATTRIBUTE

```
yes? USE etopo60

yes? DEFINE ATT rose.floatval = 22
yes? DEFINE ATT rose.pp = {1.5, 1.9}
yes? DEFINE ATT/TYPE=string/OUTPUT rose.strval = "some text about ROSE"
```

/DATASET=n applies the attribute to the variable in the specified dataset

/TYPE is string or float. If it is not specified, it is inferred from the expression.

/OUTPUT sets the flag for output so that this attribute will be written when the variable is written to a netCDF file. By default, only those attributes which Ferret has historically written to output files are written.

Ref Sec4.3. DEFINE AXIS

/X/Y/Z/T /DEPTH /FILE /FROM_DATA /MODULO /NAME /NPOINTS /T0 /UNITS /EDGES /CALENDAR /BOUNDS

Defines an axis (axis name up to 16 characters).

yes? DEFINE AXIS[/qualifiers] axis_name
or
yes? DEFINE AXIS[/qualifiers] axis_name = expr

Example:

yes? DEFINE AXIS/X=140E:140W:.2 AX140
or
yes? DEFINE AXIS/X myaxis = {1, 5, 15, 35}

Note on DEFINE AXIS:
Axes which are "in use", because they are used by currently open data sets may now be redefined using DEFINE AXIS.

In early versions of Ferret, attempting to redefine an in-use axis generated an error. This feature is especially useful to correct the interpretation of erroneous files, or files which exhibit minor incompatibilities with Ferret. Use this feature with caution as it can be used to "fool" Ferret into an incorrect interpretation of a data file.

Command qualifiers for DEFINE AXIS:

DEFINE AXIS/X=/Y=/Z=/T=

Specifies the limits and point spacing of an axis.

```
yes? DEFINE AXIS/X=lo:hi:delta   axis_name
```

The limits may be in longitude, latitude, or date format (for X, Y, or T axis, respectively) or may be simple numbers. No units are assumed unless units are given explicitly with the /UNITS qualifier.

Use /UNITS=degrees to obtain latitude or longitude axes. The X or Y qualifier determines which orientation "degrees" refers to. If an X axis has units of degrees longitude it is automatically marked as a modulo axis.

For T axis, the limits may be dates (dd-mmm-yyyy:hh:mm:ss) or may be time steps. The delta increment is regarded as hours unless the /UNITS qualifier specifies otherwise. Note that time axes may not extend beyond year 9999.

If the time limits are given as dates then this axis produces date-formatted output (unless CANCEL MODE CALENDAR is issued). If the time limits are given as time steps then all instances of this axis are labeled with time step values in the units specified with the /UNITS qualifier.

Examples (evenly-spaced axes):

```
yes? DEFINE AXIS/X=140E:140W:.2 ax140
yes? DEFINE AXIS/Y=15S:25N:.5 axynew
yes? DEFINE AXIS/Z=0:5000:20/UNITS=CM/DEPTH axzcm
yes? DEFINE AXIS/T="7-NOV-1953":"23-AUG-1988:11:00":24  axtlife
yes? DEFINE AXIS/T=25:125:5/UNITS=minutes   axt5min
```

DEFINE AXIS/CALENDAR=

Allows for non-Gregorian calendar axes. The calendars allowed are

calendar name	number of days/year	notes
GREGORIAN or STANDARD	365.2425	default calendar: proleptic Gregorian
JULIAN	365.25	with leap years
NOLEAP	365	no leap years
ALL_LEAP	366	366 days every year
360_DAY	360	each month is 30 days

The calendar definitions conform to the netCDF conventions document for calendars. See http://www.cgd.ucar.edu/cms/eaton/cf-metadata/CF-current.html#cal These calendar definitions are compatible with the Udunits standard (see http://www.unidata.ucar.edu/packages/udunits/udunits.dat) which has slightly different naming conventions. Also note that the default calendar in Ferret is the proleptic Gregorian calendar, i.e. the definition of a year is consistent throughout time and does not have an offset in the 1500's as the historical calendars did. However, files written using the NOAA/CDC standard for the "blended" Julian/Gregorian calendar are read correctly by Ferret: If a time axis has a time origin of 1-1-1 00:00:00, and uses the default calendar, and if the coordinates of axis lie entirely after the year 1582, then the historical 2-day shift is applied.

The netCDF conventions recommend that the calendar be specified by the attribute time:calendar when there is a non-Gregorian calendar associated with a data set, i.e.

time:calendar=noleap

Ferret reads this attribute from a netCDF file, or gets it from a definition made with DEFINE ATTRIBUTE and assigns the designated calendar to the time axis.

Example:

Define a calendar axis and regrid an existing variable to this axis:

```
yes? DEFINE
AXIS/CALENDAR=JULIAN/T="15-JAN-1982":"15-DEC-1985":30/UNITS=days tmodel
yes? LET twind = uwnd[GT=tmodel@NRST]
```

When regridding from one calendar axis to another the length of a year is assumed to be constant, therefore the regridding calculates a scale factor based on the length of a second in each calendar, computed from the number of seconds per year for the calendars.

DEFINE AXIS/DEPTH

Specifies the Z axis to be a depth, positive downward, axis. A depth axis is indicated by a "(-)" following its title in a SHOW GRID or SHOW AXIS command. Depth axes are notated by "UD" (up-down) in the grid definition file, while normal vertical axes (such as an elevation axis in meteorology) are notated by "DU" (down-up).

Example:

```
yes? DEFINE AXIS/Z=0:5000:20/DEPTH/UNITS=CM AXZDCM
```

DEFINE AXIS/EDGES

The /EDGES qualifier indicates that the coordinates provided refer to the edges or boundaries between grid cells. When /EDGES is used, the coordinates of the grid points will be computed at the midpoints between the indicated edges. When /EDGES is used in conjunction with /FROM_DATA the number of grid points created will be equal to the number of coordinates minus one, since the list of edges includes both the upper and lower edge of the axis. An example of defining an axis by its edges is

```
yes? DEFINE AXIS/Z=0:5010:20/EDGES/DEPTH/UNITS=CM AXZDCM
```

A class of especially important uses for the /EDGES qualifier is to create custom calendar axes. This example creates a true monthly axis, with axis cells beginning on the first of each month:

```
yes? ! Define a 20 year monthly axis starting in Jan 1950
yes? LET start_year = 1950
yes? LET nyears = 20
yes? LET indices = L[L=1:`nyears*12`]
yes? LET month = MOD(indices-1,12)+1
yes? LET year = start_year + INT((indices-1)/12)
yes? DEFINE AXIS/UNITS=days/T0=1-jan-1900/EDGES truemonth  =
DAYS1900(year,month,1)
```

```
yes? ! Examine one year of it
yes? SHOW AXIS/T=1-jan-1958:1-jan-1959 truemonth
 name      axis       # pts   start                end
 TRUEMONTH TIME       239 i   16-JAN-1950 12:00    16-NOV-1969 00:00
T0 = 1-JAN-1900
   Axis span (to cell edges) = 7274
     L     T                   TBOX   TBOXLO               TSTEP (DAYS)
    96>  16-DEC-1957 12:00:00   31  01-DEC-1957 00:00:00   21168.5
    97>  16-JAN-1958 12:00:00   31  01-JAN-1958 00:00:00   21199.5
    98>  15-FEB-1958 00:00:00   28  01-FEB-1958 00:00:00   21229
    99>  16-MAR-1958 12:00:00   31  01-MAR-1958 00:00:00   21258.5
   100>  16-APR-1958 00:00:00   30  01-APR-1958 00:00:00   21289
   101>  16-MAY-1958 12:00:00   31  01-MAY-1958 00:00:00   21319.5
   102>  16-JUN-1958 00:00:00   30  01-JUN-1958 00:00:00   21350
   103>  16-JUL-1958 12:00:00   31  01-JUL-1958 00:00:00   21380.5
   104>  16-AUG-1958 12:00:00   31  01-AUG-1958 00:00:00   21411.5
   105>  16-SEP-1958 00:00:00   30  01-SEP-1958 00:00:00   21442
   106>  16-OCT-1958 12:00:00   31  01-OCT-1958 00:00:00   21472.5
   107>  16-NOV-1958 00:00:00   30  01-NOV-1958 00:00:00   21503
   108>  16-DEC-1958 12:00:00   31  01-DEC-1958 00:00:00   21533.5
   109>  16-JAN-1959 12:00:00   31  01-JAN-1959 00:00:00   21564.5
```

The following example shows the computation of a custom climatological average. Given, for example, a multi-year time series of a daily measured variable, the climatological average of the variable for two unequal time periods could be computed by creating an axis with two points, using the FROM_DATA qualifier. The grid cells for these two points would extend from 15-Mar to 27-May (about 73 days), and from 27-May to 15-Mar (about 292 days). The actual dates on which the 2 points are located would be the midpoints of these two intervals, on 20-Apr and 20-Oct.

```
yes? DEFINE AXIS/t=1-jan-0001:1-jan-0002:1/unit=days/t0=1-jan-0000
tencoding
yes? LET tstep = t[gt=tencoding]
yes? LET start_date = tstep[t=15-mar-0001]
yes? LET end_date = tstep[t=27-may-0001]
yes? DEFINE AXIS/T/UNITS=days/T0=1-jan-0000/EDGES/MODULO
tax={`start_date,p=7`,`end_date,p=7`,`start_date+365.2425,p=7`}
yes? DEFINE GRID/T=tax taxgrid
yes? SHOW/L=1:2 grid taxgrid
     GRID TAXGRID
  name       axis              # pts   start                end
  normal     X
  normal     Y
  normal     Z
  TAX        TIME                2mi   20-APR      12:00    20-OCT
02:54
             L     T                BOX_SIZE       TIME_STEP (DAYS)
             1>  20-APR      12:00:00  73             475.5
             2>  20-OCT      02:54:35  292.2425       658.1212
```

DEFINE AXIS/BOUNDS

Define an axis from lists of coordinates and bounds. (See the discussion of netCDF bounds, page). The specificaion may be either in terms of 2*N bounds or N+1 edges. For 2*N bounds the upper bound of each cell must be the same as the lower bound of the next cell.The coordi-

nates must be inside (or may coincide with) the cell bounds. (Spaces are used here to clarify the pairs of bounds, but are not necessary and do not affect the way the data is read.)

```
DEFINE AXIS/X/BOUNDS axisname = coordinates, bounds
```

Examples:

First, a 2*N bounds definition and then an N+1 one. Note that the second definition has the coordinate z=0 at the same location as the edge of the first grid cell.

```
   yes? DEF AXIS/X/BOUNDS xax =  {1,2,5,6}, {0.5,1.5, 1.5,2.5, 2.5,5.5,
5.5,6.5}

   yes? LET a = {0,20,50,75,120}
   yes? LET b = {0,10,30,60,90,150}
   yes? DEF AXIS/Z/DEPTH/BOUNDS zax = a, b
```

DEFINE AXIS/FILE=

Reads a gridfile for grid and axis definitions. The gridfile specified should be in the standard TMAP gridfile format. There are several documents in $FER_DIR/doc regarding gridfiles and TMAP format (e.g., "about_grid_files.txt").

```
yes? DEFINE AXIS/FILE=grid_file.grd
```

DEFINE AXIS/FROM_DATA

Used only in conjunction with /NAME to define an axis from any expression that Ferret can evaluate.

```
yes? DEFINE AXIS/FROM_DATA/NAME=axis_name   expr
```

(This is a mechanism to convert dependent variables into independent axis data.)

When defining an axis from a LET-defined variable or expression the condensed syntax (e.g.)

```
yes? DEFINE AXIS/X axname=expression
```

replaces the older (still supported) syntax

```
yes? DEFINE AXIS/X/NAME=axname/FROM_DATA expression
```

Note that the values from which the axis is to be created must be in strictly increasing order. If the coordinates are repeated, Ferret will "micro-adjust" the values by adding multiples of 1 millionth of the axis range to the repeated values. Ferret will issue an informative message if it is micro-adjusting an axis.

Example (unevenly-spaced axis):

```
yes? DEFINE AXIS/X my_xaxis=pos[D=2]^0.5
```

defines each coordinate of the axis "my_xaxis" as the square root of variable "pos" from data set 2.

Variables in Ferret are always represented as single precision floating-point numbers. Coordinates are represented as double precision values. This presents a quandary when we wish to define an axis, and have double precision data representing the coordinates available as a variable (but not a coordinate variable) in a netCDF file. Once that variable is read by Ferret, it becomes single precision and we lose the precision needed to define the axis. Starting with V6 of Ferret, a means is provided to work around this, in particular for making graphics. We read the data, specifying an offset which is applied when the data is read, and which gives the numbers fewer significant figures, so the variations are represented in single precision. This may be "undone" via a PPLUS command which applies the reverse offset to the coordinates at the time we make a plot.

For example, say a variable called XVALS is in a netCDF file, which has data values {144.002, 144.0034, 144.005}. We want to create an axis from these values, and put the variable data_var onto that axis. Once XVALS is converted to single precision we would lose the variation. We can read the locations with the SET VAR/OFFSET command. It is convenient to define a variable to keep the constant offset, as it will be used again.

Example:

```
yes? SET DATA filename.nc
yes? LET xlon = 144
yes? SET VAR/OFFSET=`-1*xlon` xvals

yes?  ! when the data is read, the offset is added
yes? LIST/NOHEADER xvals
 1   / 1:  0.002000
 2   / 2:  0.003400
 3   / 3:  0.005000
yes?  ! Define an axis, and put the variable onto this axis...
yes? DEFINE AXIS/X/UNITS=deg  xxaxis = xvals
yes? LET var = data_var[gx=xxaxis@ASN]
yes?   ! Add back the offset. This works with any plotting command.
yes? PLOT/SET var
yes? PPL XVALOFF `xlon`
yes? PPL PLOT
```

DEFINE AXIS/MODULO[=len]

Specifies that the axis being defined be treated as modulo; that is, the first point will wrap around and follow the last point (e.g., a longitude axis).The optional modulo length is the length in axis units of the modulo repeat. If a length is specified, it may be longer than the axis span, so that the axis is treated as a subspan modulo axis, and if no length is specified then the

default modulo length for the type of axis is used. See the sections on modulo axes and subspan modulo axes for more information (p.167 ff).

DEFINE AXIS/NAME=

Used only in conjunction with /FROM_DATA to specify the name of the axis to be defined.

```
yes? DEFINE AXIS/FROM_DATA/NAME=axis_name   expr
```

DEFINE AXIS/NPOINTS=

Specifies the number of coordinate points on the axis being defined.

```
yes? DEFINE AXIS/Z=lo:hi/NPOINTS=n   ax_name
```

This qualifier can be used instead of specifying ***Z=lo:hi:delta***.

DEFINE AXIS/T0=

Specifies the date and time associated with the time step value 0.0

Example:

```
DEFINE AXIS/T="1-NOV-1980":"15-AUG-1988":72/T0="1-JAN-1800"  TNEW
```

Note: The /T0 qualifier is optional; the underlying time step values are transparent to Ferret users for most purposes. The default value is 15-JAN-1901.

DEFINE AXIS/UNITS=

Specifies the units of the axis being defined.

A DEFINE AXIS command such as DEFINE AXIS/X=130E:80W:2 xax infers from the formatting of the longitude coordinates the implied qualifier "/UNITS=degrees". Similar for latitudes.

Example:

```
yes? DEFINE AXIS/Z=0:2000:100/UNITS=CM   ZCM
```

Any string (up to 10 characters) is acceptable as a units string, but only the following units are recognized and used when computing axis transformations:

cm (or centimeter)	mm (or millimeter)	day
km (or kilometer)	mb (or millibar)	mon
m (or meter, or metre)	level	yr (or year) (365 days)
deg (or lat or lon)	layer	gregorian_year (365.2425 days)
ft (or feet or foot)	sec	year360 (360 days)
in	min	year366 (366 days)
mile	hour	M2 cycles
dbar	mbar	

NOTES:

1) As of Ferret version 5.1 the definition of the unit "month" has been redefined to be exactly 1/12 of a climatological year. This change applies both to files that use "units=months" and to the DEFINE VARIABLE command. The climatological month is the length of an average month in the Gregorian calendar, including leap years -- 1/12 or 365.2485 days. Thus the command

```
yes? DEFINE AXIS/T0=1-JAN-0000/T=0:12:1/EDGES/units=months/MODULO
month_reg
```

defines a climatological monthly axis which does not "drift" over time due to leap years. This non-drift behavior can be observed using a commands like

```
yes? SHOW AXIS /l=1:12001:1200 month_reg
```

which will show every 100th January over 1000 years. To create a monthly Gregorian calendar axis - with axis cells beginning on the first of each month, see the example under DEFINE AXIS/EDGES (p. 340).

2) The units dbar and mbar are recognized by Ferret, however, no automatic conversion is attempted between these and any other units.

TIP:

Ferret will convert recognized units of length to meters and recognized units of time to seconds during transformations such as integration (@IIN and @DIN) and differentiation (@DDB, @DDC, @DDF) (see "General Information about transformations," p. 113). Using this characteristic it is always possible to query Ferret about the conversion factors from meters or seconds by integrating a grid cell of width one on an axis of the units in question. For example:

```
yes? ! query conversion factor to meters
yes? define axis/x=0:1:1/edges/units=feet xtest  ! 1 point, cell width=1
unit
yes? let vx = 0*X[gx=xtest]+1                    ! vx = 1
yes? list/prec=7 vx[x=@din]
             0*X[GX=XTEST]+1
             X (FEET): 0 to 1 (integrated)
          0.3048000
```

```
yes? ! query conversion factor to seconds
yes? define axis/t=0:1:1/edges/units=month ttest  ! 1 point, cell
width=1 unit
 *** NOTE: /UNIT=MONTHS is ambiguous ... using 1/12 of 365 days.
yes? let vt = 0*T[gt=ttest]+1                     ! vt = 1
yes? list/prec=7 vt[t=@din]
             0*T[GT=TTEST]+1
             T (MONTH): 0 to 1 (integrated)
          2628000.
```

Ref Sec4.4. DEFINE GRID

/X/Y/Z/T /FILE /LIKE

Defines a grid (name may be up to 16 characters).

```
yes? DEFINE GRID[/qualifiers]   grid_name
```

Example:

```
yes? DEFINE GRID/LIKE=temp/T=my_t_axis   my_grid
```

Command qualifiers for DEFINE GRID:

DEFINE GRID/X=/Y=/Z=/T=
Specifies what particular axis is to be the X, Y, Z, or T axis for this grid.

```
yes? DEFINE GRID/X=axname  grid_name
```

The name axname may be the name of an axis, the name of a grid that uses the axis desired, or the name of a variable for which the defining grid uses the axis desired.

For example,

```
yes? DEFINE GRID/X=U   gx
```

will create a grid named gx which is one-dimensional—normal to Y, Z, and T.

Note: Many axes possess an orientation implicit in their units, especially latitude, longitude, and time axes. The effects of using an axis in an inappropriate orientation, such as /X=time_axis, are unpredictable.

DEFINE GRID/FILE=

Reads a gridfile for GRID and AXIS definitions. The gridfile specified should be in the standard TMAP gridfile format. There are several documents in $FER_DIR/doc regarding gridfiles and TMAP format (e.g., about_grid_files.txt).

Example:

```
yes? DEFINE GRID/FILE=new_grids.grd
```

DEFINE GRID/LIKE=
Specifies a particular grid (by name or by reference to a variable defined on that grid) to use as a template to create a new grid.

```
yes? DEFINE GRID/LIKE=grid_or_variable_name   grid_name
```

All axes of the grid being created will be identical to the axes of the "LIKE=" grid except those explicitly changed with the /X, /Y, /Z, or /T qualifiers. The argument may be an expression.

Example:

```
yes? DEFINE GRID/LIKE=temp[D=2]/Z=ZAX  gnew     !temp from data set 2
```

Examples: DEFINE GRID

1) ***yes? DEFINE AXIS/T="1-JAN-1980":"31-DEC-1983":24 axday***
   yes? DEFINE GRID/LIKE=temp/T=axday gday
   Define grid gday to be like the defining grid for temp but with a 4-year, daily-interval time axis.

2) ***yes? DEFINE GRID/LIKE=temp[D=ba022]/T=sst[D=nmc] gnmc3d***
   Define grid gnmc3d like temp from data set ba022 but with the same time axis as sst from data set nmc.

3) ***yes? DEFINE AXIS/X=140E:140W:.2 xnew***
   yes? DEFINE AXIS/Y=5S:5N:.2 ynew
   yes? DEFINE AXIS/T="15-FEB-1982":"15-FEB-1984":48 tnew
   yes? DEFINE GRID/X=xnew/Y=ynew/T=tnew gnew
   Define grid gnew from new axes. The grid, gnew, will be normal (perpendicular) to Z.

Ref Sec4.5. DEFINE REGION

/I/J/K/L /X/Y/Z/T /DI/DJ/DK/DL /DX/DY/DZ/DT /DEFAULT

Defines or redefines a named region_name (first 4 characters are recognized).

```
yes? DEFINE REGION[/qualifiers] region_name
```

If the qualifier /DEFAULT is not given only those axes explicitly named will be stored. If the qualifier /DEFAULT is given all axes will be stored.

Command qualifiers for DEFINE REGION:

DEFINE REGION/I=/J=/K=/L=/X=/Y=/Z=/T=
Specifies region limits (=lo:hi or =val).

DEFINE REGION/DI=/DJ=/DK=/DL=/DX=/DY=/DZ=/DT=
Specifies a change in region relative to the current settings (=lo:hi or =val). See examples below.

DEFINE REGION/DEFAULT
Saves all axes and transformations, not just those explicitly given. Commonly, a GO script begins with "DEFINE REGION/DEFAULT save" and ends with "SET REGION save".

Examples: DEFINE REGION

1) ***yes? DEFINE REGION/DEFAULT save***
   Stores the current default region under the name "save". The region may be restored at a later time by the command ***yes? SET REGION save***.

2) ***yes? DEFINE REGION/X xonly***
   Stores the current default X axis limits, only, as region xonly.

3) ***yes? DEFINE REGION/DX=-5 xonly***
   Stores the current default X axis limits minus 5 as region xonly.

4) ***yes? DEFINE REGION/Y=20S:20N/Z yanz***
   Stores the given limits from the Y axis and the default Z axis limits.

5) ***yes? DEFINE REGION/DEFAULT/L=5 15***
   Stores the current default region with the modification that L, the time step, is stored as 5.

6) ***yes? DEFINE REGION/DL=-1:+1 lp2***
   Stores an L region beginning 1 time step earlier and ending 1 time step later than the current default region as region lp2.

Ref Sec4.6. DEFINE SYMBOL

Allows the user to define a string variable. Symbol names must begin with a letter and contain only letters, digits, underscores, and dollar signs.

```
yes? DEFINE symbol symbol_name=string
```

Example:

```
yes? DEFINE symbol my_x_label = sample number
```

Ref Sec4.7. DEFINE VARIABLE

/D /QUIET /TITLE /UNITS /BAD=

Allows the user to define a variable from a valid algebraic expression. Note: LET is an alias for DEFINE VARIABLE.

```
yes? DEFINE VARIABLE[/qualifiers]  name=expression
```

Example:

```
yes? LET SPEED = U^2 + V^2
```

Parameters

The expression may be any valid expression. See the chapter "Variables and Expressions", section "Expressions" (p. 73) for a definition of valid expressions.

Variable names

The name specified with DEFINE VARIABLE can be 1 to 128 characters in length—letters, digits, $ and _, beginning with a letter. Pseudo-variable names (I, J, K, L, X, Y, Z, T , XBOX, YBOX, ZBOX, TBOX) and operators (AND, OR, GT, GE, LT, LE, EQ, NE) are reserved and cannot be used. It is not recommended to use function names such as SIN, EXP, LN or Ferret operators and keywords like PLOT or AXIS as variable names. See the chapter "Variables and Expressions" (p. 59) for recognized operators and functions, or use the commands SHOW COMMAND and SHOW FUNCTION for a list of all commands and functions.

If the name defined is the same as a variable name in a data set, the user-defined variable is used instead of the file variable. (Look for LET/D=d_set to control this behavior in future Ferret versions.)

Examples:

1) ***yes? DEFINE VARIABLE sum = a+b***
   or equivalently
   yes? LET sum = a+b

2) ***yes? DEFINE VARIABLE/TITLE="velocity"/UNIT="m/sec" pos[T=@DDC]*0.01***
   Defines velocity in m/sec from position, pos, in cm.

Command qualifiers for DEFINE VARIABLE:

DEFINE VARIABLE/BAD=value

Allows user to control the missing value of user-defined variables The specified value will be used whenever the variable is LISTed (or SAVEd) to a file. Note that the missing value will revert to its default (-1E34) when this variable is combined in further calculations.

Example:

```
yes? let/bad=3 gap_3 = I[I=1:5]
yes? list gap_3
             I[I=1:5]
 1   / 1:  1.000
 2   / 2:  2.000
 3   / 3:   ....
 4   / 4:  4.000
 5   / 5:  5.000
yes? let new_var = gap_3 + 5
yes? list new_var
             GAP_3 + 5
 1   / 1:   6.00
 2   / 2:   7.00
 3   / 3:   ....
 4   / 4:   9.00
 5   / 5:  10.00
yes? list/form=(1PG15.3) new_var
             GAP_3 + 5
             X: 0.5 to 5.5
       6.00
       7.00
     -1.000E+34
       9.00
       10.0
```

DEFINE VARIABLE/D=dataset

Restricts the scope of the variable name to the named data set. See further discussion in the chapter "Variables and Expressions", section "Defining New Variables" (p. 143).

The qualifier "DATASET=" (LET/DATASET=...) allows you detailed control over the multiple use of the same name.

If the name or number of a data set is supplied then the /dataset qualifier indicates that this variable name is to be defined only in the specified data set. For example

```
yes? LET/dataset=coads_climatology V_geostrophic = SLP[X=@DDC]/(F*RHO)
```

Defines V_geostrophic only in data set coads_climatology. In other data sets the name V_geostrophic may refer to file variables or it may be given different definitions or it may be undefined. The data set may be specified either by name as in this example or by number as

shown by SHOW DATA. Note that variables defined using LET/dataset=[name_or_number] will be shown in the SHOW DATA output for that data set as well as in SHOW VARIABLES.

If the /dataset qualifier is applied without specifying a data set name then the interpretation is different. In this case the named variable becomes a default definition -- one which applies only if a data-set specific variable of the same name does not exist. For example, if the command

```
yes? LET/DATASET sst = temp[Z=0]
```

is issued then sst[D=levitus_climatology] will evaluate to temp[D=levitus_climatology,Z=0] because the variable sst does not exist in levitus_climatology, but sst[D=coads_climatology] will refer to the file variable name sst within the coads_climatology data set.

LET/D is especially useful for editing data sets because it gives a ready way to distinguish between the pre-edit and post-edit versions of the variable. In this example we edit the data set etopo60, replacing a small rectangle in the Pacific Ocean.

Example:

```
! Do not use memory-cached data when editing.
! Always reread the most recent version from the file.
yes? SET MODE STUPID
! Save an exact copy of the original data for editing.
! We will call our edited file "new_etopo.cdf"
yes? SET DATA etopo60
yes? LET/D=etopo60 depth = rose
yes? SET VARIABLE/TITLE="edited etopo depth"/UNITS=meters depth
yes? SAVE/FILE=new_etopo.cdf depth
yes? USE new_etopo.cdf

     ! "rose[d=etopo60]"    is the original.
     ! "depth[d=new_etopo]" is the edited version.
     ! Redefine "depth[d=etopo60]" as a tool for for selective editing.
yes? LET/D=etopo60 depth = rose[D=etopo60]-rose[D=etopo60] + correction

     ! An example edit: replace a small region with the value 500
yes? LET correction = 500
yes? SAVE/APPEND/FILE=new_etopo.cdf depth[D=etopo60,X=180:175w,Y=0:2n]
yes? PLOT/X=160e:160w/Y=1n rose[D=etopo60], depth[D=new_etopo]
```

DEFINE VARIABLE/QUIET
Suppresses message that, by default, tells you when you are redefining an existing variable. This qualifier is useful in command files. (This is the default behavior starting with Ferret version 5.2)

DEFINE VARIABLE/TITLE=
Specifies a title (in quotation marks) for the user-defined variable. This title will be used to label plots and listings. If no title is specified the text of the expression will be used as the title. (See also SET VARIABLE/TITLE, p. 423.)

DEFINE VARIABLE/UNITS=
Specifies the units (in quotation marks) of the variable being defined. (See command SET VARIABLE/UNITS, p. 423.)

Ref Sec4.8. DEFINE VIEWPORT

/CLIP /ORIGIN /SIZE /TEXT /XLIMITS /YLIMITS/AXES

Defines a new viewport (a sub-rectangle of the graphics window).

```
yes? DEFINE VIEWPORT[/qualifiers]   view_name
```

Issuing the command SET VIEWPORT is best thought of as entering "viewport mode." While in viewport mode all previously drawn viewports remain on the screen until explicitly cleared with either SET WINDOW/CLEAR or CANCEL VIEWPORT. If multiple plots are drawn in a single viewport without the use of /OVERLAY the current plot will erase and replace the previous one; the graphics in other viewports will be affected only if the viewports overlap. If viewports overlap the most recently drawn graphics will always lie on top, possibly obscuring what is underneath. By default, the state of "viewport mode" is canceled.

Example:

```
yes? DEFINE VIEWPORT/XLIMITS=0,.5/YLIMITS=0,.5   LL
```

Defines a viewport that will place graphical output into the lower left quarter of the screen, and names the viewport "LL".

Command qualifiers for DEFINE VIEWPORT.

DEFINE VIEWPORT/XLIMITS=/YLIMITS=
Specifies the portion of the full window to be used.

```
yes? DEFINE VIEWPORT/XLIMITS=x1,x2/YLIMITS=y1,y2   view_name
```

The values of the limits must be in the range [0,1]; they refer to the portion of the window (of height and length 1) which defines the viewport. Together, /XLIMITS and /YLIMITS replace the CLIP, ORIGIN, and SIZE qualifiers in older Ferret versions.

DEFINE VIEWPORT/TEXT=
Controls shrinkage (or expansion) of text.

```
yes? DEFINE VIEWPORT/TEXT=n  view_name
```

In some cases text appearance may become unacceptable due to viewport size and aspect specifications. A value of 1 produces text of the same size as in the full window; $0 < n < 1$ shrinks

the text; n > 1 enlarges text. Sensible values go up to about 2. When the qualifier /TEXT is omitted, Ferret computes a text size that is appropriate to the size of the viewport.

Note that /TEXT modifies the prominence of the text through manipulation of axis lengths rather than through direct manipulation of the many text size specifications. A low value of text prominence produces axes that are "long" (as seen with SHOW SYMBOLS, p. 229, or PPL LIST XAXIS, p. 193), making the (fixed size) text appear less prominent.

DEFINE VIEWPORT/AXES
Specifies that user's limits are interpreted as the normalized positions of the plot axes rather than of the entire viewport .

You can change PPL ORIGIN and PPL AXLEN only after SET VIEW is issued. Use the new qualifier PLOT/NOYADJUST to avoid resetting the Y origin -- relevant during PLOT commands that require extra room for a large key block under the axes or for viewports that lie close to the bottom of the window where there may not be room below the Y origin. If /NOYADJUST is specified, and the viewport is near the bottom of the window, the labelling at the bottom of the plot will be lost.

DEFINE VIEWPORT/XLIMITS=x1,x2/YLIMITS=y1,y2/AXES view_name

Example 1:

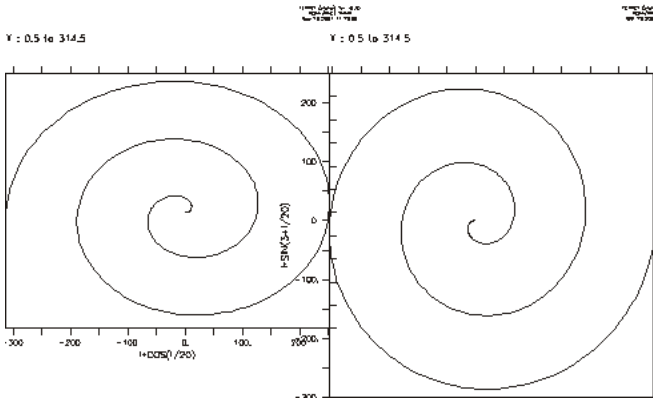


Figure Ref_1 a

This example shows the effect of the /YADJUST qualifier on the plot command. Define two viewports and plot; on the left the Y axis is adjusted automatically, on the right we specify /NOADJUST and the labelling below the plot is not plotted.

```
yes? DEFINE VIEW/AXES/XLIM=0:0.5/YLIM=0:0.5 llax
yes? DEFINE VIEW/AXES/XLIM=0.5:1/YLIM=0:0.5 lrax
yes? SET VIEW llax
yes? PLOT/VS/LINE/I=1:314 i*cos(i/20),i*sin(i/20)

yes? SET VIEW lrax
yes? PLOT/VS/LINE/I=1:314/NOYADJUST i*cos(3+i/20),i*sin(3+i/20)
```

Example 2:

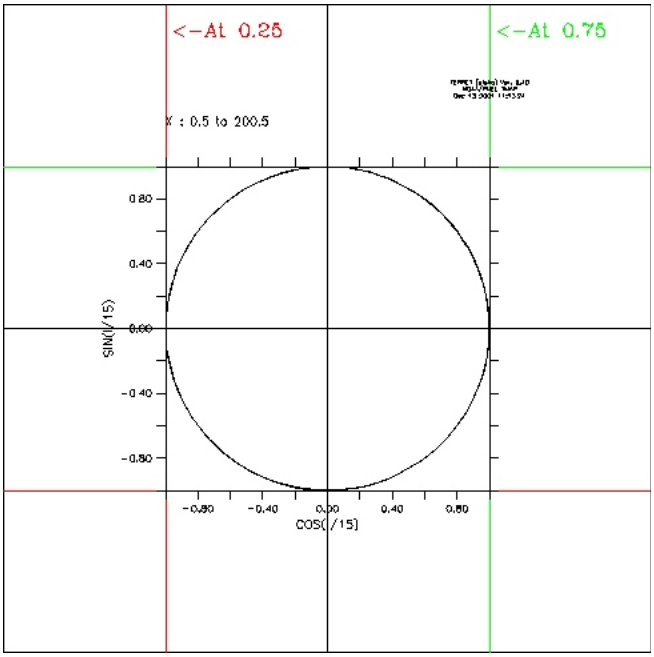


Figure Ref_1 b

DEFINE VIEWPORT/AXES can be used to set guide lines on a plot

```
yes? CANCEL VIEW
yes? DEFINE VIEW/AXES allax

yes? SET VIEW allax
yes? PLOT/VS/LINE/HLIM=0:1/VLIM=0:1/NOLAB {0.5,0.5,,0,1},{0,1,,0.5,0.5}
yes? PLOT/VS/LINE/OVER/NOLAB {0.25,0.25,,0,1},{0,1,,0.25,0.25}
yes? PLOT/VS/LINE/OVER/NOLAB {0.75,0.75,,0,1},{0,1,,0.75,0.75}

yes? LABEL 0.26,0.95,-1,0,.2 @P2@AC<-At 0.25
yes? LABEL 0.76,0.95,-1,0,.2 @P3@AC<-At 0.75

yes? DEFINE VIEW /XLIM=0.25:0.75/YLIM=0.25:0.75/TEXT=1/AXES mid
yes? SET VIEW mid
yes? PLOT/VS/HLIM=-1:1/VLIM=-1:1/LINE/I=1:200 cos(i/15),sin(i/15)
```

DEFINE VIEWPORT/CLIP=
This qualifier is obsolete; see XLIMITS= and /YLIMITS= (above). Specifies the location of the upper right corner of the viewport.

DEFINE VIEWPORT/ORIGIN=
This qualifier is obsolete; see /XLIMITS= and /YLIMITS= (above). Specifies the location of the lower left corner of the viewport.

DEFINE VIEWPORT/SIZE=
This qualifier is obsolete; see /XLIMITS and /YLIMITS (above). Specifies the scaling factor to use relative to the size of the full window.

Ref Sec5. ELIF

The ELIF command is a part of Ferret's conditional command execution capability: IF-THEN-ELIF-ELSE-ENDIF. It is valid only inside of an IF block. See further description under the IF command (p. 360) in this Commands Reference section.

Ref Sec6. ELSE

The ELSE command is a part of Ferret's conditional command execution capability: IF-THEN-ELIF-ELSE-ENDIF. It is valid only inside of an IF block. See further description under the IF command (p. 360) in this Commands Reference section.

Ref Sec7. ENDIF

The ENDIF command is a part of Ferret's conditional command execution capability: IF-THEN-ELIF-ELSE-ENDIF. It is valid only inside of an IF block. See further description under the IF command (p. 360) in this Commands Reference section.

Ref Sec8. EXIT

/LOOP /SCRIPT /PROMPT /PROGRAM /COMMAND /CYCLE

Without a qualifier, when issued interactively and without a qualifier EXIT terminates program Ferret. When executed within a command file, EXIT terminates the execution of the command file and returns control to the level in Ferret that executed the file (the user or another command file).

"QUIT" in a command file is an alias for EXIT without a qualifier. It will exit the current script, or the program if you are at the Ferret prompt.

Command qualifiers for EXIT:

EXIT/LOOP

When executed from within a REPEAT loop, Ferret will stop execution of that loop and return to the level in Ferret which executed the loop.

EXIT/CYCLE

When executed from within a REPEAT loop, Ferret will skip the remaining commands in the loop and continue at the start of the next repetition of the loop.

EXIT/SCRIPT

When executed from within a script, this command will terminate the execution of that script and return control to the level in Ferret which executed the script (either the user or another command file).

EXIT/PROMPT

When executed at any point, either in a script or loop, this command will immediately terminate execution and Ferret will return to the"yes?" prompt and return control to the user.

EXIT/PROGRAM

EXIT/COMMAND_FILE
When executed from within a command file EXIT/COMMAND_FILE or EXIT/PROGRAM forces an immediate exit from Ferret without returning control to the user or another command file.

Ref Sec9. FILE

The FILE command is an alias for SET DATA/EZ (p. 403). All qualifiers and restrictions are identical to SET DATA/EZ

Example:

```
yes? FILE/VARIABLES="u,v" velocities.dat
            is equivalent to
yes? SET DATA/EZ/VARIABLES="u,v" velocities.dat
```

Ref Sec10. FILL

Alias for CONTOUR/FILL (p. 331), color-filled contour plot. All qualifiers and restrictions are identical to CONTOUR/FILL.

Example:

```
yes? FILL/PAL=land_sea  etopo60
```

is equivalent to

```
yes? CONTOUR/FILL/PAL=land_sea  etopo60
```

In a curvilinear coordinate system (map projections)

```
yes? FILL[/qualifiers]   expression, xcoords, ycoords (see p. 222)
```

Ref Sec11. FRAME

/FORMAT /FILE

Saves the current graphics display image as a frame in the movie file initialized with the command SET MOVIE. FRAME is also a qualifier for the "action" commands PLOT, CONTOUR, POLYGON, SHADE, VECTOR and WIRE.

```
yes? CONTOUR my_var
yes? FRAME
```

Note that FRAME follows a command which creates an image.

FRAME/FORMAT=format controls the format of the file produced.

FRAME/**FORMAT=HDF** appends an HDF raster 8 drawn to the specified or implied input file. The default format is HDF.

FRAME/FORMAT=GIF creates a new GIF file, any existing GIF file with the specified or implied name using relative version number or less. Note that in this mode of grabbing an image Ferret creates a GIF by requesting the image back from your screen (your X server). This means that the X server normally has to be configured as pseudo-color. An alternative approach which does not share this restriction is to start Ferret with "ferret -gif" (see p. 6)

FRAME/FILE=filename specifies the name of the output file. If /FORMAT is not specified the output format is inferred from filename extensions of .hdf, .HDF, .gif, or .GIF.

The maximum filename length, including path, that is allowable is 255 characters.

Ref Sec12. GO

/HELP

Executes a list of commands stored in a file.

```
yes? GO file_name
```

If no filename extension is specified a default of .jnl will be assumed. If the full path is specified then the filename must be enclosed in double quotation marks.

The GO command can pass arguments to the script (tool) it executes. See the introductory chapter, section "Writing GO Tools" (p. 23) for more information. Arguments to the GO command may be separated by blanks or commas. To specify multiple words as a single argument, enclose them in quotation marks. To specify an argument that is deliberately omitted, use " " or two consecutive commas.

The response of Ferret to errors encountered during execution of the command file is determined by mode IGNORE_ERRORS. (See command SET MODE, p. 409.)

The echoing of command file lines is controlled by mode VERIFY.

The GO command understands a special syntax called "relative version numbers." If a filename is specified for the GO command which has a version value of zero or less its value is interpreted as relative to the current highest version number. See the chapter “Computing Environment”, section “Relative version numbers” (p. 264) for a discussion of relative version numbers of files.

Note: The command SET MODE IGNORE_ERRORS is useful when rerunning past sessions which may have errors.

/HELP
The command GO/HELP filename opens the named script with the Unix "more" command and displays the first 20 lines of the named file. Use this command to quickly see the documentation in a GO script.

Ref Sec13. HELP

On Unix systems interactive Ferret help is available from the command line with the commands Fapropos, Fhelp, and Ftoc. If multiple windows are not available on your system the ^Z key can be used to suspend the current Ferret session and access the help; the Unix command "fg" will then restore the suspended session.

See the introductory chapter, section "Unix on-line help" (p. 30) for more information.

Ref Sec14. IF

Ferret provides an IF-THEN-ELSE syntax to allow conditional execution of commands.

In addition Ferret uses an "masking" IF-THEN-ELSE syntax for masking. These share keywords but have different usage.

Ref Sec14.1. IF-THEN-ELSE conditional execution

This syntax may be used in two styles—single line and multi-line. In both the single and multi-line styles the true or false of the IF condition is determined by case-insensitive recognition of a condition based on evaluating a single string or a numeric value evaluated as a scalar. The conditional statement may contain one of these options:

TRUE condition:

- a valid, non-zero numerical scalar value
- TRUE
- T
- YES
- Y

FALSE condition:

- a single zero value
- an invalid embedded expression (see next paragraph)
- FALSE
- F
- NO
- N
- BAD
- MISSING

If the expression is a numeric expression, then it may be a digit (e.g. 1 or 0), a variable, or a symbol. If it is a variable, it must evaluate to a scalar, and be enclosed in grave accents to force its evaluation in the IF expression. If it is a symbol, it must be evaluated with ($symbolname).

If a variable is not a scalar, it cannot be used with this particular syntax for conditional execution, however the use of masking allows you to operate on a whole field based on a condition by defining a new variable (See p.133 and p.114).

Examples:

```
IF `i GT 5` THEN SAY "I is too big" ENDIF
```

writes message if the value of I is greater than 5

```
IF `a` THEN SAY "A is nonzero" ENDIF
```

writes message if the value of A is something other than zero

```
IF ($yes_or_no) THEN GO yes_script ELSE GO no_script
```

executes yes_script or no_script according to the value of the symbol yes_or_no

```
IF ($dset%|coads>TRUE|%) THEN GO my_plot
```

executes the script my_plot.jnl only if the symbol dset contains the exact string "coads"

```
IF `i LT 3` THEN
    GO option_1
ELIF `i LT 6` THEN
    GO option_2
ELSE
    GO option_3
ENDIF
```

uses the multi-line IF syntax to select among GO scripts.

Embedded (grave accent) expressions can be used in conjunction with the IF syntax. For example, `3 GT 2` (Is three greater than 2?) evaluates to "1" (TRUE) and `3 LT 2` (Is three less than 2?) evaluates to "0" (FALSE). If the result of a grave accent expression is invalid, for example division by zero as in `1/0`, the string "bad" is, by default, generated. Thus invalid expressions are regarded as FALSE.

Symbol substitution permits IF decisions to be based on text-based conditions. Suppose, for example, the symbol ($DSET) contains a string: either coads or levitus. Then an IF condition could test for coads using ($DSET%|coads>TRUE|*>FALSE%).

```
IF ($DSET%|coads>TRUE|*>FALSE%) THEN
      GO cscript
ELSE
      GO lscript
ENDIF
```

The single line style allows IF-THEN-ELSE logic to be applied on a single line. For example, to make a plot only when the surface (Z=0) temperature exceeds 22 degrees we might use

```
IF `TEMP[X=160W,Y=2N,Z=0] GT 22` THEN PLOT TEMP[X=160W,Y=2N]
```

The single line syntax may be any of the following:

IF condition THEN clause_1
IF condition THEN clause_1 ENDIF

IF condition THEN clause_1 ELSE clause_2
IF condition THEN clause_1 ELSE clause_2 ENDIF

Note that both ELSE and ENDIF are optional in the single line syntax. Groups of commands enclosed in parentheses and separated by semicolons can be used as clause_1 or as clause_2. There is no ELIF (pronounced "else if") statement in the single line syntax. However, IF conditions can be nested as in

```
IF `i1 GT 5` THEN (IF `j1 LT 4` THEN go option_1 ELSE go option_2)
```

The multi-line style expands the IF capabilities by adding the ELIF statement. Multi-line IF statement follows the pattern

```
IF condition_1 THEN
      clause_1_line_1
      clause_1_line_2
      ...
ELIF condition_2 THEN
      clause_2_line_1
      ...
ELIF condition_3 THEN
      ...
ELSE
      ...
ENDIF
```

Note that THEN is optional at the end of IF and ELIF statements but the ENDIF statement is required to close the entire IF block. Single line IF statements may be included inside of multi-line IF blocks.

Ref Sec14.2. IF-THEN-ELSE logic for masking

Ferret expressions can contain embedded IF-THEN-ELSE logic. The syntax of the IF-THEN logic is simply (by example)

```
LET a = IF a1 GT b THEN a1 ELSE a2
```

This syntax is especially useful in creating masks that can be used to perform calculations over regions of arbitrary shape. For example, we can compute the average air-sea temperature difference in regions of high wind speed using this logic:

```
SET DATA coads_climatology
SET REGION/X=100W:0/Y=0:80N/T=15-JAN
LET fast_wind = IF wspd GT 10 THEN 1
LET tdiff = airt - sst
LET fast_tdiff = tdiff * fast_wind
```

We can also make compound IF-THEN statements. The parentheses are included here for clarity, but are not necessary. Multi-line IF-THEN-ELSE constructs are not allowed in embedded logic

```
LET a = IF (b GT c AND b LT d) THEN e
LET a = IF (b GT c OR  b LT d) THEN e
LET a = IF (b GT c AND b LT d) THEN e ELSE q
```

The user may find it clearer to think of this logic as WHERE-THEN-ELSE to aviod confusion with the IF used to control conditional execution of commands.

Ref Sec15. LABEL

/NOUSER

Places a label on the current plot; alias for PPL %LABEL. %LABEL is one of PPLUS's primitive plot commands. It places a label on the plot immediately after being issued (rather than deferring placement). PPLUS does not assign numbers to labels created with LABEL, so they cannot be manipulated as movable labels. The label can also be placed on the plot using the mouse to point and click (see the chapter "Customizing Plots", section "Positioning labels using the mouse pointer," p. 198).

```
yes? LABEL xpos, ypos, center, angle, size, text
```

xpos, ypos	position in user units (world coordinates)
center	-1 left justification (the default)
	0 centered
	1 right justification
angle	angle in degrees, 0 degrees at 3 o'clock (default 0)
size	size of text in inches (default 0.12)

See the chapter "Customizing Plots", section "Labels" (p. 191) for examples.

Command qualifiers for LABEL:

LABEL/NOUSER
Locates labels in inches instead of user units (xpos and ypos are specified in inches rather than in world coordinates).

Ref Sec16. LET

The LET command is an alias for DEFINE VARIABLE (p.349). All qualifiers and restrictions are identical to DEFINE VARIABLE.

Example:

```
yes? LET A = B
```
is equivalent to
```
yes? DEFINE VARIABLE A = B
```

Ref Sec17. LIST

/I/J/K/L /X/Y/Z/T /D /ILIMITS /JLIMITS /KLIMITS /LLIMITS /XLIMITS /YLIMITS /ZLIMITS /TLIMIT /APPEND /FILE /FORMAT /HEADING /NOHEAD /TITLE /ORDER /RIGID /PRECISION /CLOBBER /SINGLE /QUIET /WIDTH /EDGES /BOUNDS

Produces a listing of the indicated data.

```
LIST[/qualifiers]   [expression_1 , expression_2 , ...]
```

Example:

```
yes? LIST/Z=10 u , v , u^2 + v^2
```

Lists the 3 quantities specified using the current default data set and region (at depth 10).

Parameters

Expressions may be any valid expression. See the chapter "Variables and Expressions", section "Expressions" (p. 73) for a definition of valid expressions. If multiple variables or expressions are specified they may be listed together in columns or in sequence depending on the /SINGLY qualifier. The expression(s) will be inferred from the current context if omitted from the command line.

If multiple expressions are given on the command line and /SINGLY is not specified, then the expressions must be conformable. See the chapter "Variables and Expressions", section "Multi-dimensional expressions" (p. 75) for a definition of conformable expressions. Degenerate or single point axis limits will be promoted up (values repeated) as needed.

Example:

```
yes? LIST/I=1:3/J=1:2  i+j, i
```

Command qualifiers for LIST:

LIST/I= /J= /K= /L=/X= /Y= /Z= /T=
Specifies value or range of axis subscripts (I, J, K, or L), or axis coordinates (X, Y, Z, or T) to be used when evaluating the expression(s) being listed.

LIST/ILIMITS=/JLIMITS=/KLIMITS=/LLIMITS=
Specifies the size of the desired netCDF output file independently from the actual data being saved. By specifying axis limits in excess of the saved expression's limits it is possible to /APPEND data later. (See the chapter "Converting to NetCDF", section "Simple Conversions Using Ferret," p. 267, ex. 4).

LIST/XLIMITS=/YLIMITS=/ZLIMITS=/TLIMITS=
Specifies the size of the desired netCDF output file independently from the actual data being saved. By specifying axis limits in excess of the saved expression's limits it is possible to /APPEND data later. (See the chapter "Converting to NetCDF", section "Simple Conversions Using Ferret," p. 267, ex. 4).

LIST/D=
Specifies the default data set to be used when evaluating the expression(s) being listed.

LIST/APPEND
Use this qualifier together with the /FILE qualifier to indicate that the listed data should be appended to a pre-existing file. If no file exists by the name indicated a new file is created. This qualifier is not applicable to /FORMAT=GT. When used with /FORMAT=CDF it permits any data in the file to be overwritten, new variables to be added to the file, and appending of new indices along the T axis of the variables in the file. To append slabs of data in other dimensions, see the example in the netCDF chapter (p. 268). This qualifier overrides the command CANCEL LIST/APPEND.

LIST/FILE [=file_name]
Names a file to receive the listed data. If /FILE is specified with no name then the default name is used from the SET LIST/FILE command.

Example:

```
yes? LIST/FILE=my_file.dat sst[D=coads_climatology]
```

See command SET LIST (p. 407) for further information on automatic filename generation.

LIST/CLOBBER
Used with LIST/FILE. Indicates that any existing file with the name used is to be deleted, before writing. If CLOBBER is not specified and the file exists, and error message is given.

Example:

```
yes? LIST/FILE=my_file.dat/CLOBBER sst[D=coads_climatology]
```

LIST/FORMAT=
Specifies an output format (=format_choice) for the data to be listed.

yes? SET LIST/FORMAT=format_choice

or

yes? SET LIST/FORMAT (use format set by SET LIST/FORMAT)

Format choices:

FORTRAN format	produces ASCII output
"UNFORMATTED"	produces unformatted (binary) output using FORTRAN record structure
"CDF"	produces netCDF format output
"GT"	produces TMAP GT format
"STREAM"	produces unstructured binary floating point (C-style)
"tab"	produces tab-delimited output
"comma"	produces comma-delimited output

This command has the same function as SET LIST/FORMAT except that it does not affect future LIST commands. See command SET LIST/FORMAT (p. 408) for detailed documentation.

Notes for LIST/FORMAT:

1) All output values, regardless of the /FORMAT designation, will be of type single precision floating point. For FORTRAN output formats this means all numerical field specifiers must be "F", "E", or "G".

2) For FORTRAN-formatted and UNFORMATTED (binary) output, the contents of a single output "record" are determined by the /ORDER qualifier. For example, each record will be a line of Y values for LIST/ORDER=YX. If /ORDER is omitted, the records will be the first output axis of greater than unity length taken in the order X, Y, Z, then T. FORTRAN-formatted output records may be further split by the usual rules of FORTRAN output formatting.

3) FORTRAN formats must be enclosed in parentheses. If blanks are included in the format it must be enclosed in quotation marks. Output strings are permitted in the format.

   Example:

```
yes? LIST/FORMAT=("The temperature is:", F6.3)  sst[X=180, Y=0]
```

4) When FORTRAN formats are used, and more than one value per record is desired, the /ORDER qualifier (p368) must be used, even if the variable is defined along only one axis.

   Example:

```
yes? LIST/FORMAT=(8F6.3)/ORDER=T  sst[X=180, Y=0]
```

5) The default listing style includes labels for the rows and columns of the output. When a FORTRAN format is specified, these labels are omitted.

6) On Unix systems the /FORMAT=UNFORMATTED specifier produces FORTRAN-style variable-length records. On most implementations this means that a 4-byte field containing the record length begins and ends each record of data.

7) The command alias SAVE is provided for the commonly used LIST/FORMAT=CDF. netCDF outputs are self-documenting, including grid definitions. The output files can be used as input with the command USE—alias for SET DATA/FORMAT=CDF. See command SAVE (p. 392) for further notes about netCDF files.

8) For tab and comma foratted output, the coordinates are checked to see if the default format gives enough precision to distinguish all of the coordinates. For fine grids, the coordinates are output with enough digits to distinguish them. This may result in the coordinates not lining up with the columns of the variable values. If there is missing data, a tab or comma will be written as a placeholder.

LIST/BOUNDS

Only valid for netCDF output (LIST/FORMAT=cdf). Outputs the data, including the bounds attribute on all of its axes. The bounds attribute is used by default on all irregular axes, for Ferret ersion 5.70 and later. See the comments under SAVE (p. 393).

LIST/EDGES

Only valid for netCDF output (LIST/FORMAT=cdf). Outputs the data, having the edges attribute on all of its axes.

LIST/HEAD
For ASCII data listings this command determines whether to precede the listing with a heading describing data set, variable and region. This qualifier overrides the CANCEL LIST/HEAD command. Starting with Ferret version 5.4 the default heading output of the list command is expanded to include the filename, file path, and complete information on the subset of the data that's listed. See the example under LIST/WIDTH= (p. 369)

LIST/HEADING[=ENHANCED]
For ASCII data listings this qualifier determines whether to precede the listing with a heading that describes the data set, variable, and region. This qualifier overrides the CANCEL LIST/HEAD command. When the argument /HEADING=ENHANCED is used a self-documenting heading is provided that includes the axis coordinates.

For netCDF output files (alias SAVE) the /HEADING=ENHANCED option causes the netCDF file structure to include extra coordinate information that describes how the particular data subset being written fits within the broader coordinate system of the grid from which it is

extracted. When a netCDF file with an enhanced heading is accessed by Ferret (using SET DATA or USE) the index values will appear to be consistent with the parent data set.

LIST/NOHEAD

Does not precede listing with a heading describing data set, variable and region. This qualifier overrides the SET LIST/HEAD command.

LIST/ORDER=
Specifies the order (ORDER=permutation) in which axes are to be laid out in the listing.

Examples:

```
yes? LIST/ORDER=XY sst          !X varies fastest
yes? LIST/ORDER=YX sst          !Y varies fastest
```

The "permutation" string may be any permutation of the letters X, Y, Z, and T. /ORDER is applicable only to /FORMAT=unf and FORTRAN formats.

Note that a 1-dimensional list will, by default, place only one value per record. The /ORDER qualifier can cause the 1-dimensional list to occur in a single record. For example,

```
LIST/I=1:5 I
```

will list as 5 records whereas

```
LIST/I=1:5 /ORDER=X I
```

will list 5 values on a single record.

LIST/PRECISION=#
Controls the digit precision of LIST output

Using the qualifier /PRECISION=#digits the output precision of the LIST command may be easily controlled. This qualifier functions exactly as does the SET LIST/PRECISION= command but it applies only to the current command. Note that this controls only the output precision of the data, and that in fact at most 7 digits are significant since Ferret calculations are performed in single precision.

LIST/QUIET
Using the qualifier /QUIET will prevent the message "LISTing to file XXXX.XXXX" from being displayed.

LIST/RIGID
Valid only with /FORMAT=CDF. Indicates that Ferret should not create a netCDF "record" axis as the time axis for any of the variables listed with this command. Time axes are, instead, of fixed length and the /APPEND qualifier is not usable to extend the listing.

LIST/SINGLY
This qualifier is relevant only when multiple expressions are specified in the LIST command. When the /SINGLY qualifier is specified the entire listing of each expression including (optional) heading and all data is completed before proceeding to the next expression.

By default the expressions are not listed singly—each line contains one value of each expression. The qualifier has no effect if only a single expression is specified. If the /FILE qualifier is specified to use automatic filename generation and /APPEND is not specified, then each expression is listed to a separate file.

LIST/TITLE="title string"
Valid only with /FORMAT=CDF. Causes the global attribute "title" to be defined in a netCDF file, thereby setting its title.

LIST/WIDTH=columns
For multi-column output, controls the width of the listing on the page so the output line is no longer than "columns" characters.

Example:

```
yes? USE coads_climatology
yes? LIST/L=1/WIDTH=50/Y=0:4 sst
             VARIABLE : SEA SURFACE TEMPERATURE (Deg C)
             DATA SET : COADS Monthly Climatology (1946-1989)
             FILENAME : coads_climatology.des
             FILEPATH : /home/ja9/tmap/fer_dsets/descr/
             SUBSET   : 180 by 2 points (LONGITUDE-LATITUDE)
             TIME     : 16-JAN      06:00
      ... listing every  36th point
               21E    93E   165E   123W    51W
                1     37     73    109    145
 3N   / 47:   ....  28.30  29.04  25.36  27.49
 1N   / 46:   ....  28.29  29.12  24.82  27.49


yes? list/l=1/wid=70/y=0:4 sst
             VARIABLE : SEA SURFACE TEMPERATURE (Deg C)
             DATA SET : COADS Monthly Climatology (1946-1989)
             FILENAME : coads_climatology.des
             FILEPATH : /home/ja9/tmap/fer_dsets/descr/
             SUBSET   : 180 by 2 points (LONGITUDE-LATITUDE)
             TIME     : 16-JAN      06:00
      ... listing every  23th point
               21E    67E   113E   159E   155W   109W    63W    17W
                1     24     47     70     93    116    139    162
 3N   / 47:   ....  28.15  27.54  29.09  26.90  25.34   ....  27.58
 1N   / 46:   ....  28.18   ....  29.24  26.49  24.90   ....  27.05v
```

Ref Sec18. LOAD

/I/J/K/L /X/Y/Z/T /D /NAME /PERMANENT /TEMPORARY

Loads a variable or expression into memory.

```
yes? LOAD[/qualifiers]   [expression_1 , expression_2 , ...]
```

Loading may speed execution of later commands that will require the loaded data. Often it is helpful to LOAD a large region of data encompassing several small regions in which the analysis will be pursued.

Load interacts with the current context exactly as other "action" commands CONTOUR, PLOT, SHADE, VECTOR, LIST, etc. do.

Parameters

Expressions may be any valid expression. See the chapter "Variables and Expressions", section "Expressions" (p. 73) for a definition of valid expressions. If multiple variables or expressions are specified they are treated in sequence. The expression(s) will be inferred from the current context if omitted from the command line.

Command qualifiers for LOAD:

LOAD/I=/J=/K=/L=/X=/Y=/Z=/T=
Specifies value or range of axis subscripts (I, J, K, or L), or axis coordinates (X, Y, Z, or T) to be used when evaluating the expression(s) being loaded.

LOAD/D=
Specifies the default data set to be used when evaluating the expression(s) being loaded.

LOAD/NAME
Obsolete. Provided for compatibility with much older Ferret versions.

LOAD/PERMANENT
Data loaded with LOAD/PERMANENT are kept in memory until a LOAD/TEMPORARY command is given that refers to the same data. See command LOAD/TEMPORARY (p. 370). Note that this command may cause memory fragmentation. It should generally be given immediately following CANCEL MEMORY and preferably is used only to load file variables (as opposed to expressions).

LOAD/TEMPORARY (default)
Data loaded with LOAD or LOAD/TEMPORARY is brought into memory but may be unloaded based on a priority scheme of least recent use when memory space is required.

Ref Sec19. MESSAGE

/CONTINUE /QUIET /JOURNAL /ERROR

Displays a message at the terminal.

```
yes? MESSAGE text
```

By default a carriage return is required from the keyboard for program execution to continue (used to halt the execution of a command file). PAUSE is an alias for MESSAGE.

Command qualifiers for MESSAGE:

MESSAGE/CONTINUE
Continues program execution following the display of the message text without waiting for a carriage return from the operator. SAY is an alias for MESSAGE/CONTINUE.

MESSAGE/JOURNAL
Writes the message to the journal file.

MESSAGE/ERROR
Writes the message to standard error.

MESSAGE/QUIET
Waits for a carriage return from the operator but does not supply a prompt for it.

Ref Sec20. PALETTE

Alias for PPL SHASET SPECTRUM=. Specifies or restores the default color. PALETTE is also implemented as a qualifier for the 2-Dimension graphics commands FILL, CONTOUR/FILL, SHADE, and POLYGON commands.

```
yes? PALETTE pal_name
```

or

```
yes? FILL/PALETTE=pal_name var
yes? SHADE/PALETTE=pal_name var
yes? POLYGON/PALETTE=pal_name var
```

The argument is the name of a palette file. Many palettes are included in the Ferret distribution. Try the Unix command "Fpalette '*'" to see a list of available palette files.

Some of the palettes are designed for particular needs. "centered.spk", for example, emphasizes the contrast between positive and negative shade levels. "land_sea.spk" uses blue tones

for negative values and browns and greens for positive values, making it suitable for topography displays.

Palette files end in the file suffix .spk, but the suffix is not necessary when specifying a palette. Use ***GO try_palette pal_name*** to display a palette. The GO files "exact_color.jnl" and "squeeze_colors.jnl" can be used to modify palettes. You can also create new palette files with a text editor. See the chapter "Customizing Plots", section "Shade and fill colors" (p. 203) for the format of a palette file.

PALETTE with no argument restores the default palette. When you use the qualifier /PALETTE= in conjunction with /SET_UP, PPLUS makes the specified color spectrum the new default palette, and all subsequent shaded or color-filled plots will use that palette as the default. To restore the previous palette to the default, use PALETTE with no argument after your customization.

To assist you in choosing a good palette for your plot, there is an FAQ, How can I find a good color palette for my plot? at http://ferret.pmel.noaa.gov/Ferret/FAQ/graphics/colorpalettes.html

Ref Sec21. PATTERN

Alias for PPL PATSET PATTERN=. Specifies or restores the default pattern. PATTERN is also implemented as a qualifier for the 2-Dimension graphics commands FILL, CONTOUR/FILL, SHADE, and POLYGON commands.

```
yes? PATTERN patt_name
```

or

```
yes? FILL/PATTERN=patt_name var
yes? SHADE/PATTERN=patt_name var
yes? POLYGON/PATTERN=patt_name var
```

The argument is the name of a pattern file. Many patterns are included in the Ferret distribution. Try the Unix command "Fpattern '*'" to see a list of available pattern files, or run the script

```
yes? go show_all_patterns.jnl
```

to see examples of the different pattern fills available.

Ferret has the capability to make color fill plots using solid color only, and also with colors laid on in patterns.

The script

```
yes? go pattern_demo
```

illustrates the use of patterns and overlaying patterns on color fill plots.

The PATTERN command sets the patterns to be used in a plot generated with the SHADE, FILL and POLYGON commands. It is similar to the PALETTE command, which sets colors, but the PALETTE and PATTERN commands act independently.When Ferret is started up, only one pattern is set, SOLID. The SOLID pattern is equivalent to not using any pattern, and SHADE, FILL and POLYGON fill their cells with solid color.

Pattern files end in the file suffix .pat, but use of the suffix is not necessary when specifying a pattern. Use ***GO show_pattern patt_name*** to display the patterns specified in a pattern file. ***GO show_all_patterns*** draws a plot showing all the available pattern files and their names. Notice that patterns can be used with a single color, or multiple colors, depending entirely on the PALETTE specification.

A pattern file may specify one or more patterns. If there are fewer patterns specified in a pattern file than there are levels in a particular plot, the patterns will be repeated.

Ref Sec22. PAUSE

Alias for MESSAGE (p. 371)

Ref Sec23. PLOT

/I/J/K/L /X/Y/Z/T /OVERLAY/ SET_UP /FRAME /D/ TRANPOSE/ VS/ SYMBOL/ NOLABEL /LINE /COLOR /THICKNES /SIZE /HLIMITS /VLIMITS /TITLE /STEP /NOAXES /DASH /NOYADJUS /AXES /HLOG /VLOG /NOKEY

Produces a line plot.

```
yes? PLOT[/qualifiers]   [expression_1 , expression_2 , ...]
```

The indicated expression(s) must represent a line (not a plane) of data (PLOT/VS is an exception). Unless the /VS qualifier is used, the independent variable is the underlying coordinate axis for this line of data.

Example:

```
yes? PLOT/l=1:100 sst
```

produces a time series plot of the first 100 points of sst.

Parameters

The argument(s) for PLOT specify the variable or expression to be plotted.

When the /VS qualifier is used the indicated expressions may have any geometry in 4D space but they must match in the total number of points in each expression. The points are associated in the order of their underlying axes. When the /VS qualifier is not used the indicated expression(s) must describe a line (not a plane) of data.

The expression(s) are inferred from the current context if omitted from the command line—i.e., if no expression is given then the argument most recently given is used, or the default expression may be explicitly set with SET EXPRESSION.

When Ferret plots multiple data lines simultaneously, PPLUS automatically cycles through pen colors and symbols, creating up to 26 distinct line types. Try GO line_samples to see samples of these styles.

Command qualifiers for PLOT:

PLOT/I=/J=/K=/L=/X=/Y=/Z=/T=
Specifies value or range of axis subscripts (I, J, K, or L), or axis coordinates (X, Y, Z, or T) to be used when evaluating the expression(s) being plotted.

PLOT/D=
Specifies the default data set to be used when evaluating the expression(s) being plotted.

PLOT/FRAME
Causes the graphic image produced to be captured as an animation frame and written to the movie file specified by SET MOVIE. In general the FRAME command (p. 358) is more flexible and we recommend its use rather than this qualifier.

PLOT/COLOR=/THICKNESS=
Simple syntax for line plots. For line plots it is possible with these qualifiers to control line thickness and color with commands such as

```
yes?  PLOT/COLOR=blue/THICK=2 I[i=1:3]
```

This is equivalent to the (still supported) use of the /LINE qualifiers in

```
yes? PLOT/LINE=10 I[i=1:3] ! 4(blue) + 6*(2-1)
```

The available color names are Black, Red, Green, Blue, LightBlue, Purple, and White (not case sensitive), corresponding to the /LINE values 1-6, respectively. (/COLOR also accepts numerical values.) The line thickness may be 1, 2, or 3 corresponding to pixel thickness on the screen or corresponding to multiples of the default line thickness on hard copy. Note that White is only available for THICKNESS=1 (the default thickness).

Prior to Ferret v5.80, /THICK with no accompanying /COLOR plotted all lines in black. Starting with v5.80 PLOT/THICK for plot lines overlaid on the same plot cycles through the colors Black, Red, Green, Blue, LightBlue and Purple. PLOT//THICK alone produces six in thickness 2 and six in thickness 3.

When plotting a number of lines together with PLOT/OVER/COLOR=/THICKNESS= the default behavior is for symbols to be plotted starting with the seventh line plotted. To plot lines only, add the /LINE qualifier.

After you have created your plot, if you need the lines even thicker to show up properly when printed, see the FAQ, How can I control line thickness when I translate a metafile to a postscript file? at http://www.ferret.noaa.gov/Ferret/FAQ/custom_plots/changing_linewidths.html

PLOT/COLOR=/THICKNESS=/SYMBOL=/SIZE=
Simple syntax for plots using symbols. For symbol (scatter) plots (PLOT/VS or PLOT/SYMBOL), control the color, size, and line thickness of the symbols with commands such as:

```
yes?  PLOT/COLOR=red/THICKNESS=2/SYMBOL=4/SIZE=0.2 I[i=1:5]
```

The available color names are Black, Red, Green, Blue, LightBlue, Purple, and White (not case sensitive), corresponding to the /LINE values 1-6, respectively. (/COLOR also accepts numerical values.) The line thickness may be 1, 2, or 3 corresponding to pixel thickness on the screen or corresponding to multiples of the default line thickness on hard copy; note that White is only available in the default THICKNESS=1. The /SIZE is given in units of "inches", consistent with the PLOT+ usage of "inches". (These are the same units as in, say, "ppl axlen 8,6", to specify plot axes of lengths 8 and 6 inches for horizontal and vertical axes, respectively.)

PLOT/DASH[=]

(New qalifier with version 5.4) Simple syntax control over the dash characteristics using the same arguments as in the PPLUS "LINE" command: DOWN1, UP1, DOWN2, UP2, where these are in inches. For simple dashes let DOWN1=DOWN2 and UP1=UP2. For alternating long and short dashes, make DOWN2 longer or shorter. The parentheses are optional.

Example:

```
yes? PLOT/DASH/I=1:100 sin(i/5)
yes? PLOT/OVER/DASH=(0.3,0.1,0.3,0.1)/COLOR=RED/THICK/I=1:100 sin(i/7)
yes? PLOT/OVER/DASH=(0.6,0.2,0.1,0.2)/COLOR=RED/THICK/I=1:100 sin(i/9)
```

PLOT/LINE[=]
The /LINE qualifier without =n causes the PLOT command to connect the plotted points with a line regardless of the state of the /SYMBOLS qualifier.

For simpler specification of line characteristics see PLOT/COLOR=/THICKNESS=/DASH= above (p. 374) . /LINE=n specifies a pre-defined line style. "n" is an integer between 1 and 18. ***GO line_thickness*** draws samples of the available line styles. Line style "1" is always a solid line in the foreground color (black or white). Other line styles are device dependent (colors or dash patterns). For color devices, n=1–6 draws single-thickness lines each a different color. n=7–12 draws double-thick lines in the same color order, and n=13–18 draws tri-

ple-thick lines. See the chapter "Customizing Plots", section "Text and line colors" (p. 200) for a chart of the default colors.

If more than 18 different plot lines are drawn using PLOT var1, var2, ... or with PLOT/OVER, then Ferret starts making symbols on the plots

PLOT/NOLABELS
Suppresses all plot labels.

PLOT/OVERLAY
Causes the indicated field(s) to be overlaid on the existing plot. This qualifier can also be used to overlay lines or symbols on 2D plots (SHADE, CONTOUR, or VECTOR) provided the axis scalings are appropriate.

PLOT/SET_UP
Performs all the internal preparations required by program Ferret for plotting but does not actually render the plot. The command PPL can then be used to make changes to the plot prior to producing output with the PPL PLOT command. This makes possible certain customizations that are not possible with Ferret command qualifiers. See the chapter "Customizing Plots" (p. 183).

PLOT/SYMBOL[=]
The /SYMBOL qualifier causes the PLOT command to mark each plotted point with a symbol. If the /LINE qualifier is given too the symbols are also connected with a line; if /LINE is omitted no connecting line is drawn.

Optionally, the symbol number may be explicitly specified as an integer value between 1 and 88. The integer refers to the PPLUS plot marker numbers (e.g., 1 for x, 3 for +, etc.). Type "GO show_symbols" and "GO show_88_syms" at the Ferret prompt to see available symbols and their reference numbers. The symbols are also documented on page 1 of the document $FER_DIR/doc/pplus_fonts.ps. The PLOT MARK font can be accessed using the font code @PM.

PLOT/SYMBOL=DOT
This command uses the smallest dot that can be represented on the display device. Note that the dots may not show up well on all devices.

PLOT/TITLE=
Allows user to specify a plot title (enclosed in quotation marks). Without this qualifier Ferret selects a title based on information about the expression(s).

PLOT/TRANSPOSE
Causes the horizontal and vertical axes to be interchanged. By default the X axis is drawn horizontally on the plot and the Y and Z axes are drawn vertically. For Y-Z plots the Z data axis is vertical by default.

PLOT/VS
Specifies that the first expression given in the command line is to be used as the independent axis.

Example:

```
yes? PLOT/Y=20S:20N/X=180/T=27740:27741/Z=100/VS  temp , salt
```

Produces a plot of salinity (vertical axis) against temperature (horizontal axis) along the indicated range of latitudes and times. The plot will be labeled "salt"; the vertical (dependent) variable is the one that determines the key. The qualifier /TRANSPOSE can be used in conjunction with /VS to further manipulate the labeling and axis orientation.

PLOT/VS implies /SYMBOL by default to produce scatter plots. Use PLOT/VS/LINE to produce a line plot, and PLOT/VS/DASH[=] to plot a dash line and optionally set its characteristics.

PLOT/STEP[=CONNECTED]

PLOT/STEP give a plot style that is consistent with grid cell interpretation of data.This is especially useful for time series to show the span of each time bin. /STEP=CONNECTED draws a line segment between the steps.

PLOT/HLIMITS=
Specifies axis range and tic interval for the horizontal axis. Without this qualifier Ferret selects a reasonable range.

```
yes? PLOT/HLIMITS=lo:hi:[increment]  [expression(s)]
```

The optional "increment" parameter determines tic mark spacing on the axis. If the increment is negative, the axis is reversed.

This qualifier does not have any impact on the context of the expression being plotted. If data is on a modulo x axis but the arguments of the /HLIMITS qualifier represent a region outside the actual coordinates of the data, only the range including the actual coordinates is shown. Use /X=lo:hi or SET REGION or a context on the variable itself, var[X=lo:hi], to set the context for the expression.

The /HLIMITS and /VLIMITS qualifiers will retain their "horizontal" and "vertical" interpretations in the presence of the /TRANSPOSE qualifier. Thus, the addition of /TRANSPOSE to a plotting command mandates the interchange of "H" and "V" on the limits qualifiers.

PLOT/VLIMITS=
Specifies the axis range and tic interval for the vertical axis. See /HLIMITS (above).

```
yes? PLOT/VLIMITS=lo:hi:[increment]  [expression(s)]
```

The optional "increment" parameter determines tic mark spacing on the axis. If the increment is negative, the axis is reversed.

PLOT/AXES[=top,bottom,left,right]

Turns plotting of individual axes off and on. This replaces the use of the "PPL AXSET" command. The syntax is

```
yes? PLOT/AXES[=top,bottom,left,right]  var
```

where the arguments are 1 to turn the axis on and 0 to turn it off. For example:

```
yes? PLOT/AXES=0,1,1,0  sst     ! Draws the bottom and left axes only
```

PLOT/NOYADJUST

Avoid resetting the Y origin -- relevant during PLOT commands that require extra room for a large key block under the axes or for viewports that lie close to the bottom of the window where there may not be much room below the Y origin. See the examples for DEFINE VIEWPORT/AXES (p. 353)PLOT/HLOG /VLOG

/VLOG sets a vertical log axis, /HLOG sets a horizontal log axis. If /VLIMITS or /HLIMITS is specified, they should be in data units (not log10 axis units). If the axis is a depth axis, an inverse log axis is drawn.

Note: Setting axtype with a PPLUS call before the plot call will not result in a log plot, though setting axis type with PLOT/SET_UP still works as in previous versions.

new syntax:

```
yes? PLOT/VLOG/VLIMITS=10:1000 my_ fcn  ! plots my_fcn on a log axis
```

Replaces older syntax. The following no longer produces a log plot. As of Version 5.4, the PLOT command resets the status of the axes to linear.

```
yes? ppl axtype 1, 3
yes? PLOT/VLIMITS=1:3 my_fcn
```

These commands in the older PPL syntax duplicate the effect of the new /VLOG qualifier.

```
yes? PLOT/SET_UP/vlimits=1:3 my_fcn   ! These commands duplicate the
yes? ppl axtype, 1, 3                 ! effect of the  new syntax.
yes? ppl plot
```

PLOT/XLIMITS= /YLIMITS=
Note: XLIMITS and YLIMITS have been deprecated. Please use HLIMITS and VLIMITS instead.

PLOT/GRATICULE[=line specifiers]

(Introduced in Ferret version 5.6) Turns on graticule lines for the horizontal and vertical axes. These are lines across the plot at the tic marks. /GRATICULE sets both horizontal and vertical lines; to set each separately see /HGRATICULE and /VGRATICULE, below. The syntax is

```
yes? PLOT/GRATICULE[=line or dash,COLOR=,THICKNESS=] var
```

where the default is a thin, solid black line. The line colors available are Black, Red, Green, Blue, LightBlue, Purple, and White. The thickness codes are 1, 2, or 3 and as for plot lines, thickness=1 is a thin line, thickness=3 is the thickest, and THICK specified with no value defaults to thickness=2. For clarity the arguments to GRAT may be placed in parentheses

```
yes? PLOT/GRAT/i=1:12 1./i     ! default graticules

yes? PLOT/GRAT=(LINE,COLOR=red,THIICK=3) var

yes? PLOT/GRAT=(DASH,COLOR=lightblue) var

yes? PLOT/FILL/GRAT=(DASH,COLOR=white) var
```

The above commands make settings for the large tic marks. If small tic marks are being plotted on the axes, we can make settings for them as well using keywords SMALL and LARGE. Place all of the arguments for the /GRAT qualifier in double quotes. Note that the PPL AXNMTC command sets the plotting of small tics, and that small tics are used by default for many time axes and for logarithmic axes.

```
yes? DEFINE AXIS/Z zlog=EXP(k[k=1:10])
yes? LET fcn = k[gz=zlog]yes?
PLOT/VLOG/GRAT="LARGE(COLOR=red),SMALL(COLOR=lightblue)" fcn
```

PLOT/HGRATICULE[=line specifiers]/VGRATICULE[=line specifiers]

Turns on graticule lines and sets the line characteristics of the graticule for the horizontal or vertical axis separately. You may specify only one of /HGRAT or /VGRAT if desired. These are lines across the plot at the tic marks. The syntax is

```
     yes? PLOT/HGRATICULE[=line or dash,COLOR=,THICKNESS=]
/VGRATICULE=line or dash,COLOR=,THICKNESS=]  var
```

where the default is a thin, solid black line. The line colors available are Black, Red, Green, Blue, LightBlue, Purple, and White. The thickness codes are 1, 2, or 3 and as for plot lines, thickness=1 is a thin line, thickness=3 is the thickest, and THICK specified with no value defaults to thickness=2. For clarity the arguments to HGRAT may be placed in parentheses

```
yes? PLOT/HGRAT/VGRAT  var    !this is equivalent to PLOT/GRAT

yes? PLOT/HGRAT=(LINE,COLOR=red,THIICK=3)/VGRAT=(color=green) var
```

```
yes? PLOT/HGRAT=(DASH,COLOR=lightblue) var ! horizontal only
```

The above commands make settings for the large tic marks. If small tic marks are being plotted on the axes, we can make settings for them as well using keywords SMALL and LARGE. Place all of the arguments for the /HGRAT qualifier in double quotes. Note that the PPL AXNMTC command sets the plotting of small tics, and that small tics are used by default for many time axes and for logarithmic axes.

```
yes? DEFINE AXIS/Z zlog=EXP(k[k=1:10])
yes? LET fcn = k[gz=zlog]yes?
PLOT/VLOG/HGR="LARGE(COLOR=red),SMALL(COLOR=lightblue)"/VGRAT fcn
```

PLOT/NOKEY

Turns off the automatically-generated key showing line types on plots with multiple lines. There are several scripts which will let you position your own keys in the plot window

Ref Sec24. POLYGON

/I/J/K/L /X/Y/Z/T /OVERLAY /SET_UP /FRAME/D /TRANSPOSE /COORD_AX /SYMBOL /NOLABELS /LEVELS /LINE /COLOR /PALETTE /TITLE /THICKNESS /NOAXES /PATTERN /FILL /KEY /NOKEY /HLIMITS /VLIMITS

Produces a color-filled or line plot of polygons. By default a color key is drawn and lines are not drawn.

```
POLYGON[/qualifiers]  x-vertices, y-vertices [, values]
```

Example:

This is the first example in the script poly_vec_demo.jnl The scripts poly_vector.jnl and mp_poly_vector.jnl make definitions to set up a polygon command to draw a vector field using

arrow-shaped polygons, optionally filled with a color representing another quantity. Here is a wind field, colored according to Sea Level Pressure.

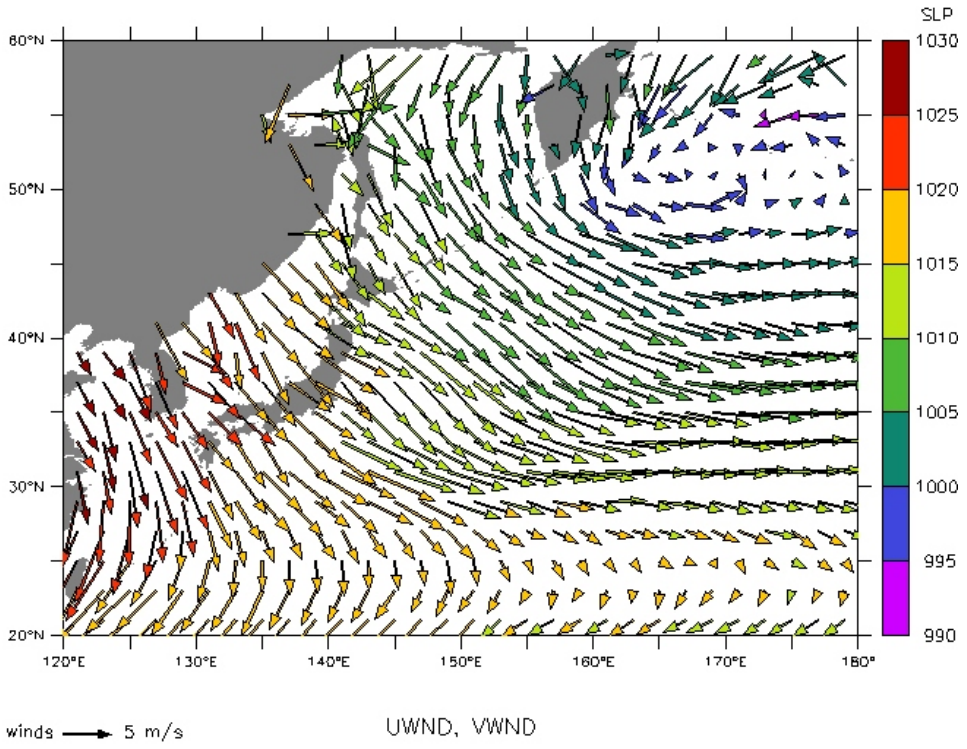


Figure Ref-2

Parameters

The two x- and y- vertices parameters separately specify the x and y coordinates of the vertices of the polygons to be plotted.

The values may be any valid expression. If a color-filled plot is specified, the numerical value of the expression associated with each polygon determines the color of that polygon, as in SHADE and FILL plots. See the chapter "Variables and Expressions", section "Expressions" (p. 73) for a definition of valid expressions. If values are omitted the /FILL option is not valid—only /LINE plots may be made.

The POLYGON command accepts single and multi-dimensional arguments.

- 1D FORM

```
yes? POLYGON xpoly1D, ypoly1D, values
```

where if ***xpoly1D*** or ***ypoly1D*** contain missing values, those represent the end of one polygon and the start of the next. The length of the values array must equal the number of polygons in which case the X coordinate might ve visualized as

x1,x1,x1,x1,BAD,x2,x2,x2,BAD,x3,x3,x3,x3,x3,x3,x3,x3,BAD,...

where the "1","2","3" refer to the successive polygons The script polymark.jnl makes a polygon plot by generating the correct 1-D arrays from a set of x and y coordinates and a polygon-shape specification. See polymark_demo for examples. polytube.jnl also makes use of the polymark command to draw "Lagrangian" plots along a track using color fill.

- 2D FORM

```
yes?  POLYGON xpoly2D, ypoly2D, values
```

where ***values*** must be 1-dimensional and its axis must match in size and orientation one of the axes from the 2D arrays. This axis represents the list of successive polygons. The other axis of the 2D coordinates is the coordinates within each polygon. In the default case the X coordinate is the axis of the coordinates within polygons, and might be visualized as

x1,x1,x1,x1,BAD,BAD,...
x2,x2,x2,BAD,BAD,...
x3,x3,x3,x3,x3,x3,x3,x3,BAD,BAD,...

(with each list of polygon coordinates along the first axis padded with BAD to become the same length)

If the "values" argument is not given the coordinate axis may be specified using the /COORD_AX qualifier.

Example:

```
yes? LET XTRIANGLE = YSEQUENCE({-1,0,1})
yes? LET YTRIANGLE = YSEQUENCE({-1,1,-1})
yes? LET XPTS = 180+30*RANDU(I[i=1:10])
yes? LET YPTS = 30*RANDU(1+I[i=1:10])
yes? POLYGON XTRIANGLE+XPTS, YTRIANGLE+YPTS, I[I=1:10]
```

Command qualifiers for POLYGON:

POLYGON /I=/J=/K=/L=/X=/Y=/Z=/T=
Specifies value or range of axis subscripts (I, J, K, or L), or axis coordinates (X, Y, Z, or T) to be used when evaluating the expression being plotted.

POLYGON/COORD_AX=
For the 2-dimensional version of POLYGON, if "values" is omitted or is a constant then there is no information on which to determine which is the axis of the vertices within each polygon

and which is the axes of successive polygons. The qualifier /COORD_AX can be used to specify which is the axis of successive polygons.
If COORD_AX is not specified, and values is unspecified or a constant, Ferret assumes that /COORD-AX is the second axis of the 2-dimensional coordinate arrays and issues a message to this effect.

POLYGON/D=
Specifies the default data set to be used when evaluating the expression being plotted.

POLY/FILL
Fills the polygons with colors according to the third argument, values. If the values argument is omitted the /FILL option is not valid—only /LINE plots may be made.

POLYGON/FRAME
Causes the graphic image produced by the command to be captured as an animation frame in the file specified by SET MOVIE. In general the FRAME command (p. 358) is more flexible and we recommend its use rather than this qualifier.

POLYGON/KEY
Displays a color key for the palette used in the color-filled plot. By default a key is drawn unless the /LINE or /NOKEY qualifier is specified. To control the color key position and labeling, see the command SHAKEY in the appendix, "Ferret Enhancements to PPLUS" (p. 561).

POLYGON/LEVELS
Specifies the POLYGON levels or how the levels will be determined. If the /LEVELS qualifier is omitted Ferret automatically selects reasonable POLYGON levels.

See the chapter "Customizing Plots", section "Contouring" (p. 213) for examples and more documentation on /LEVELS.

POLYGON/LINE
Outlines polygons specified by x and y vertices on a POLYGON plot. When /LINE is specified the color key is omitted unless specifically requested via /KEY. The line type is controlled by the /COLOR= and /THICK= qualifiers

POLYGON/LINE/COLOR=/THICK=
Simple specification of outline characteristics f or polygon plots which specify an outline line we control line thickness and color with commands such as

```
yes?  POLYGON/LINE/COLOR=blue/THICK=2  {1,2,1}, {3,2,1}
```

This is equivalent to the (still supported) use of the /LINE qualifiers in

```
yes?  POLYGON/LINE=10 {1,2,1}, {3,2,1}  ! 4(blue) + 6*(2-1)
```

The available color names are Black, Red, Green, Blue, LightBlue, Purple, and White (not case sensitive), corresponding to the /LINE values 1-6, respectively. (/COLOR also accepts numerical values.) The line thickness may be 1, 2, or 3 corresponding to pixel thickness on the screen or corresponding to multiples of the default line thickness on hard copy, however the color White is only available in the default THICKNESS=1. The /DASH qualifier is not available for the outlines of POLYGON but dashes can be drawn using

```
POLYGON x-vertices, y-vertices
PLOT/OVER/VS/DASH   x-vertices, y-vertices
```

POLYGON/NOKEY
Suppresses the drawing of a color key for the palette used in the plot.

POLYGON/NOLABELS
Suppresses all plot labels.

POLYGON/OVERLAY
Causes the indicated POLYGON plot to be overlaid on the existing plot.

POLYGON/PALETTE=
Specifies a color palette (otherwise, a default rainbow palette is used). Try the Unix command ***% Fpalette '*'*** to see available palettes. The file suffix *.spk is not necessary when specifying a palette. See command PALETTE (p. 371) for more information.

The /PALETTE qualifier changes the current palette for the duration of the plotting command and then restores the previous palette. This behavior is not immediately compatible with the /SET_UP qualifier. See the PALETTE command (p. 371) for further discussion.

POLYGON/PATTERN=
Specifies a pattern file (otherwise, a default SOLID pattern is used). Try the Unix command ***% Fpattern '*'*** to see available pattern files. The file suffix *.pat is not necessary when specifying a pattern file. See command PATTERN (p. 372) for more information.

POLYGON/SET_UP
Performs all the internal preparations required by program Ferret for a POLYGON plot but does not actually render output. Then PPL commands can then be used to make changes to the plot prior to producing output, to make customizations that are not possible with Ferret command qualifiers. For this command, the syntax for producing the plot is different from the other plot commands. Output is made with the PPL FILLPOL or PPL POLYGON command. See the chapter "Customizing Plots" (p. 183).

Example:

```
yes? ! use the SHAKEY command with the second argument = 0,
yes? ! to place the colorkey at the top of the plot

yes? POLYGON/SET x_vertices, y_vertices, values
yes? PPL SHAKEY 1,0
```

```
yes? PPL FILLPOL  !draws the plot  (or use PPL POLYGON)
```

POLYGON/TITLE=
Allows user to specify a plot title (enclosed in quotation marks). Without this qualifier Ferret selects a title based on information about the expression(s). To include font change and color_thickness specifications (e.g., @TI@C002) in the title string, it is necessary either to CANCEL MODE ASCII or to include a leading ESC (escape) character. See the chapter "Customizing Plots", section "Fonts" (p. 207).

```
yes? POLYGON/TITLE="title string" x-vertices, y-vertices, values
```

POLYGON/TRANSPOSE
Causes the horizontal and vertical axes to be interchanged. By default the X axis is drawn horizontally on the plot and the Y and Z axes are drawn vertically. For Y-Z plots the Z data axis is vertical.

Note that plots in the YT and ZT planes have /TRANSFORM applied by default in order to achieve a horizontal T axis. See /HLIMITS (below) for further details. Use /TRANSPOSE manually to reverse this effect.

POLYGON/HLIMITS=
Specifies the horizontal axis range and tic interval (otherwise, Ferret selects reasonable values).

```
yes? POLYGON/HLIMITS=lo:hi:increment
```

The optional "increment" parameter determines tic mark spacing on the axis. If the increment is negative, the axis will be reversed.

This qualifier does not have any impact on the context of the expression being plotted. If data is on a modulo x axis but the arguments of the /HLIMITS qualifier represent a region outside the actual coordinates of the data, only the range including the actual coordinates is shown. Use /X=lo:hi or SET REGION or a context on the variable itself, var[X=lo:hi], to set the context for the expression.

The /HLIMITS and /VLIMITS qualifiers will retain their "horizontal" and "vertical" interpretations in the presence of the /TRANSPOSE qualifier. Thus, the addition of /TRANSPOSE to a plotting command mandates the interchange of "H" and "V" on the limits qualifiers.

POLYGON/VLIMITS=

Specifies the vertical axis range and tic interval. See /HLIMITS (above)

POLYGON/XLIMITS= /YLIMITS=
Note: XLIMITS and YLIMITS have been deprecated. Please use HLIMITS and VLIMITS instead.

POLYGON/HLOG /VLOG

For 1-D plots only. /VLOG sets a vertical log axis, /HLOG sets a horizontal log axis. If /VLIMITS or /HLIMITS is specified, they should be in data units (not log10 axis units). If the axis is a depth axis, an inverse log axis is drawn.

See the notes under PLOT/VLOG/HLOG (p. 378)

Example:

```
yes? POLYGON/VLOG/VLIMITS=1:1000 xpts, ypts, my_ fcn
```

POLYGON/AXES[=top,bottom,left,right]

Turns plotting of individual axes off and on. This replaces the use of the "PPL AXSET" command. The syntax is

```
     yes? POLYGON/AXES[=top,bottom,left,right] x-vertices, y-vertices,
values
```

where the arguments are 1 to turn the axis on and 0 to turn it off. For example, this command draws the bottom and left axes only

```
yes? POLYGON/AXES=0,1,1,0    x-vertices, y-vertices, values
```

POLYGON/GRATICULE[=line specifiers]

(Introduced in Ferret version 5.6) Turns on graticule lines for the horizontal and vertical axes. These are lines across the plot at the tic marks. /GRATICULE sets both horizontal and vertical lines; to set each separately see /HGRATICULE and /VGRATICULE, below. The syntax is

```
     yes? POLY/GRATICULE[=line or dash,COLOR=,THICKNESS=]  x-vertices,
y-vertices, values
```

where the default is a thin, solid black line. The line colors available are Black, Red, Green, Blue, LightBlue, Purple, and White. The thickness codes are 1, 2, or 3 and as for plot lines, thickness=1 is a thin line, thickness=3 is the thickest, and THICK specified with no value defaults to thickness=2. For clarity the arguments to GRAT may be placed in parentheses

```
yes? POLY/GRAT x-vertices, y-vertices, values   ! default settings

yes? POLY/GRAT=(LINE,COLOR=red,THIICK=3)  x-vertices, y-vertices, values

yes? POLY/GRAT=(DASH,COLOR=lightblue)  x-vertices, y-vertices, values

yes? POLY/FILL/GRAT=(DASH,COLOR=white)  x-vertices, y-vertices, values
```

The above commands make settings for the large tic marks. If small tic marks are being plotted on the axes, we can make settings for them as well using keywords SMALL and LARGE. Place all of the arguments for the /GRAT qualifier in double quotes. Note that the PPL AXNMTC command sets the plotting of small tics.

```
POLYGON/grat="small(color=red,dash),large(thick,color=green)"  {1,2,1},
{2,1,0.5}
```

POLYGON/HGRATICULE[=line specifiers]/VGRATICULE[=line specifiers]

Turns on graticule lines and sets the line characteristics of the graticule for the horizontal or vertical axis separately. You may specify only one of /HGRAT or /VGRAT if desired. These are lines across the plot at the tic marks. The syntax is

```
     yes? POLYGON/HGRATICULE[=line or dash,COLOR=,THICKNESS=]
/VGRATICULE[=line or dash,COLOR=,THICKNESS=]   x-vertices, y-vertices,
values
```

where the default is a thin, solid black line. The line colors available are Black, Red, Green, Blue, LightBlue, Purple, and White. The thickness codes are 1, 2, or 3 and as for plot lines, thickness=1 is a thin line, thickness=3 is the thickest, and THICK specified with no value defaults to thickness=2. For clarity the arguments to HGRAT may be placed in parentheses

```
yes? POLYGON/HGRAT/VGRAT xpts, ypts  !equivalent to POLY/GRAT

yes? PLOT/HGRAT=(LINE,COLOR=red,THIICK=3)/VGRAT=(color=green) xpts,ypts

yes? PLOT/HGRAT=(DASH,COLOR=lightblue) xpts, ypts! horizontal only
```

The above commands make settings for the large tic marks. If small tic marks are being plotted on the axes, we can make settings for them as well using keywords SMALL and LARGE. Place all of the arguments for the /HGRAT qualifier in double quotes. Note that the PPL AXNMTC command sets the plotting of small tics, and that small tics are used by default for many time axes and for logarithmic axes.

```
yes? POLY/HGRAT="small(color=red,dash),large(thick,color=red)"
/VGRAT="large(thick,color=white),small(dash,color=white)" xpts,ypts
```

Ref Sec25. PPLUS

/RESET

Invokes PPLUS ("PLOT PLUS" written by Don Denbo), to execute a command or commands.

```
yes? PPLUS                  !(also PPL); invokes PPLUS interactively
```

or

```
yes? PPL pplus_command   !executes a single PPLUS command
```
or
```
yes? PPL/RESET           !restores PPLUS to start-up defaults
```

Example:

```
yes? PPL CROSS 1          !reference line through zero
```

Executes the PPLUS command "CROSS" and immediately returns control to Ferret.

When PPLUS is invoked interactively the prompt is "PPL>" instead of the usual "yes?". The EXIT command given at the "PPL>" prompt returns control to Ferret.

See the chapter "Customizing Plots" (p. 183) for more information on Ferret/PPLUS interactions. A complete list of PPLUS commands is in PLOT PLUS for Ferret User's Guide.

Command Qualifiers for PPLUS:

PPLUS/RESET
Restores PPLUS to start-up settings.

Ref Sec26. QUERY

Non-operating command (no result) checks the value of arguments to a GO script. See p. 26 for examples.

Ref Sec27. QUIT

Alias for EXIT; also just Q. See p. 356

Ref Sec28. REPEAT

/I/J/K/L /X/Y/Z/T /RANGE= /NAME= /ANIMATE/LOOP=

Repeats a command or group of commands over a range of values along an axis.

```
yes? REPEAT/q=lo:hi[:increment] COMMAND
```

```
yes? REPEAT/RANGE=lo:hi[:increment][/NAME=] COMMAND
```

The units of lo, hi, and increment are the units of the underlying grid axis if the qualifier is X, Y, Z, or T. The qualifiers I, J, K, or L advance the repeat loop by incrementing the indicated index (the default index increment is 1). Use SHOW GRID to examine the axis units (if the units are not displayed try CANCEL MODE LATITUDE, LONGITUDE, or CALENDAR as appropriate). To use an arbitrary loop counter use /RANGE=lo:hi:inc and optionally /NAME=string to name the loop counter. To run the loop from the highest value decreasing towards the lowest value, specify increment to be less than zero. Any command or group of commands that can be specified at the command line can also be given as an argument to REPEAT. If MODE VERIFY is SET, the current loop index is displayed at the console as REPEAT executes. The value of any symbols e.g. "($symbol)") that are used inside of REPEAT loops are re-translated at each repetition of the loop.

We can exit from a REPEAT loop with the command EXIT/LOOP, stop execution of that loop and return to the level in Ferret which executed the loop. We can skip the remaining commands in a loop and return to the next iteration with the command EXIT/CYCLE. See the documentation on EXIT (p. 357).

Examples:

1) ***yes? REPEAT/L=1:240 CONTOUR/Y=30S:50N/X=130E:70W/LEV/FRAME sst***
   Produces a 240-frame movie of sea surface temperature.

2) ***yes? REPEAT/Z=300:0:-30 GO compz***
   Executes the command file compz.jnl at Z=300, Z=270, ..., Z=0.

3) ***yes? REPEAT/L=1:250:5 (GO set_up; CONTOUR sst; FRAME)***
   Repeats three commands—execution of a GO script, CONTOUR, and FRAME—for each timestep specified.

4) ***yes? CANCEL MODE verify***
   ***yes? REPEAT/RANGE=1:100:5/NAME=m ***
   ***(USE wind_`m`.nc; ***
   ***LIST/NOHEAD wspd[X=@AVE,Y=@AVE,Z=@AVE,T=@AVE]; ***
   CANCEL DATA wind_`m`)
   Uses REPEAT/RANGE to open a set of data files, and do computations in all four dimensions.

5) See also the examples in the section on Animations (p. 177)

Command qualifiers for REPEAT:

REPEAT/I=/J=/K=/L=/X=/Y=/Z=/T=

Repeats the requested command(s) for the specified range of axis subscripts (I, J, K, or L) or axis coordinates (X, Y, Z, or T). Note that when T axis limits are specified as dates, the units of increment are hours.

REPEAT/ANIMATE[/LOOP=]

The /ANIMATE qualifier creates an animation on the fly. In a Ferret session, display an animation with the command,

```
yes? REPEAT/ANIMATE[/LOOP=n]
```

to start an animation sequence. Given LOOP=n, the animation sequence will repeat n times.

Example:

```
yes? set data coads_climatology
yes? repeat/l=1:12/animate/loop=5 (shade sst; go fland)
```

For a general discussion of animations, see the chapter Animations and GIF Images (p. 175)

NOTE: In order to properly display on SGI's, it is necessary to have backing store enabled for the Xserver.

REPEAT/RANGE=[/NAME=]
(Introduced in Ferret version 5.6) Repeats a command or group of commands over an arbitrary range of values. /RANGE= may be used with /NAME= to give a name to the repeat counter. The syntax is:

```
yes? REPEAT/RANGE=LO:HI[:INC]  ! for an unnamed loop counter
yes? REPEAT/RANGE=LO:HI[:INC]/NAME=string  ! for a named loop counter
```

If INC is not specified the increment is 1. If LO is larger than HI and INC is negative, then the loop is executed on decreasing values of the counter. If LO is larger than HI and INC is positive, the loop is not executed (or if LO is less than HI and INC is negative, the loop is not executed). The syntax can be used to create nested loops, including nesting with REPEAT /I/J/K/L or /X/Y/Z/T commands.

REPEAT/RANGE=LO:HI[:INC]/NAME=var assigns the name "var" to the repeat counter so it may be used within the commands to be executed. After the REPEAT terminates, this variable is undefined.

Examples:

Example 1) Factorial calculation with REPEAT/RANGE

```
yes? ! Factorial
yes? CANCEL MODE verify
```

```
yes? LET a = 0; LET f = 1

yes? REPEAT/RANGE=1:8 ( LET a = `a+1`;LET f = `f*a`;\
  LIST/NOHEAD/FORMAT=(F3.0," factorial", F7.0) a, f )

  1. factorial     1.
  2. factorial     2.
  3. factorial     6.
  4. factorial    24.
  5. factorial   120.
  6. factorial   720.
  7. factorial  5040.
  8. factorial 40320.
```

Example 2) REPEAT/RANGE=/NAME= with a nested repeat. The first two commands are not executed, their LO:HI:INC ranges indicate that no repeat is to be done.

```
yes? rep/range=3:7:-1/name=s (list s)

yes? rep/range=3:-3:3/name=s (list s)

yes? rep/range=3:-3:-3/name=s (rep/range=1:2/name=tt list s + sin(tt))

!-> REPEAT: S:3
!-> REPEAT: TT:1
             VARIABLE : S + SIN(TT)
          3.841
!-> REPEAT: TT:2
             VARIABLE : S + SIN(TT)
          3.909

!-> REPEAT: S:0
!-> REPEAT: TT:1
             VARIABLE : S + SIN(TT)
          0.8415
!-> REPEAT: TT:2
             VARIABLE : S + SIN(TT)
          0.9093

!-> REPEAT: S:-3
!-> REPEAT: TT:1
             VARIABLE : S + SIN(TT)
         -2.159
!-> REPEAT: TT:2
             VARIABLE : S + SIN(TT)
         -2.091
```

Example 3) The REPEAT/RANGE counter is unaffected by whether a dataset is open, or a region has been set. Here we use the @MIN and @MAX transformations and the INT function to set the minimum and maximum for the range counter, and plot the MAX function computed using the range counter variable.

```
yes? USE levitus_climatology
yes? SET REGION/x=100:300/Y=0/Z=10
yes? PLOT temp

yes? LET r1 = INT( temp[X=@MIN]) + 1
yes? LET r2 = INT( temp[X=@MAX]) - 1
```

```
yes? REPEAT/RANGE=`r1`:`r2`:3/NAME=tt  (PLOT/OVER MAX(temp,`tt`) )
 !-> rep/range=18:28:3/name=tt  (plot/over max(var,`tt`) )
!-> REPEAT: TT:18
 !-> plot/over max(var,18)
!-> REPEAT: TT:21
 !-> plot/over max(var,21)
!-> REPEAT: TT:24
 !-> plot/over max(var,24)
!-> REPEAT: TT:27
 !-> plot/over max(var,27)
```

A word of caution about REPEAT/RANGE. If you find yourself using it to do regridding, or compute integrals or averages, or if you are doing some operation to every element of a variable along an axis, you might want to rethink whether you are unnecessarily complicating your scripts by duplicating Ferret's capability to operate on entire grids or axes in a single command.

Ref Sec29. SAVE

The SAVE command is an alias for LIST/FORMAT=CDF (p.365). All qualifiers and restrictions are identical to LIST/FORMAT=CDF.

Example:

```
yes? SAVE temp, salt
```
is identical to
```
yes? LIST/FORMAT=CDF temp, salt
```

Notes:

1) Gaps in netCDF outputs are filled with the missing value flag of the variable being written. (See the chapter "Variables and Expressions", section "Missing value flags," p. 64.) In the example below, if "temp" and "salt" share the same time axis then the L=2:4 values of salt will be so filled.

```
yes? SAVE/FILE=test.cdf   temp[L=1:5], salt[L=1], salt[L=5]
```

2) Transformations that compress an axis to a point produce results that Ferret regards as time-independent. Thus, this 12-month average:

```
yes? SAVE/FILE=annual.cdf  sst[L=1:12@AVE]
```

creates a netCDF file with no time axis. It would not be possible to append the average of the next 12 months as the next time step of this file. (See p. 268 for examples on appending to netCDF files.) However, a time location can be inherited from another variable. In this example, we inherit the time axis of "timestamp" in order to create a time axis in the netCDF file.

```
yes? DEFINE AXIS/T="1-JUL-1980":"1-JUL-1985":1/UNIT=year  tannual
yes? DEFINE GRID/T=tannual  gannual
yes? LET  timestamp = T[G=gannual] * 0              !always 0
yes? LET  sst_ave = sst[L=1:12@AVE] + timestamp
yes? SAVE/FILE=annual.cdf  sst_ave[L=1]
yes? LET  sst_ave = sst[L=13:24@AVE] + timestamp
yes? SAVE/FILE=annual.cdf/APPEND  sst_ave[L=2]
.
. etc.
```

3) Background documentation about the definition and data set of origin for a variable are saved in the "history" attribute of a variable when it is first saved in the netCDF file. If the definition of the variable is then changed, and more values are inserted into the file using SAVE/APPEND, the modified definition will NOT be documented in the output file. If the new definition changes the defining grid for the variable the results will be unpredictable.

4) If you have created a data file with variables that you DEFINE, you will need to cancel those previous definitions of the variables before you USE the new data set. If you USE the data file while the DEFINE VARIABLE definition still exists in Ferret then the one that you defined is the one that you will see with LIST and other commands. Either CANCEL VARIABLE, or QUIT and start a new Ferret session where you access your new data set.

```
. . .
yes? SAVE/FILE=annual.cdf/APPEND  sst_ave[L=2]
. . .
yes? CANCEL VAR sst_ave
yes? USE annual.cdf
```

yes? SHOW DATA

5) To write packed data to a netCDF file, make sure the variable has add_offset and scale_factor attributes and that these attributes have the /OUTPUT flag set. Then the data and its missing value flag will have this scaling applied on output. (See p. 73)

6) Note that when writing netCDF files Ferret, by default, does NOT include the point_spacing attribute. This is because Ferret's default file characteristic is to be append-able, with no guarantees that the appended time steps will be regularly spaced. For output files of fixed length with regular time steps it is advisable to use the SAVE/RIGID qualifier. This allows Ferret to include the point_spacing="even" attribute. If the files will be very large (too large for the full time range to be in memory), then use the /RIGID/TLIMITS= qualifiers to specify the full, ultimate fixed size and use SAVE/APPEND to insert data into the file piecemeal.

If a coordinate is irregular it is output to the netCDF file with a bounds attribute (beginning with Ferret v5.70). To require that all axes be written with bounds attributes, use the /BOUNDS qualifier:

yes? SAVE/BOUNDS v1,v2

If the file has the bounds attribute for the record axis (files written with Ferret version 5.70 and after include the bounds attribute for all irregular axes), then we can append further timesteps

to the file and keep the correct point spacing. If there is a gap between the last time in the existing file and the first time being appended, Ferret adds a void point, consisting of missing data, centered on the interval between the upper cell bound of the time axis in the file and the lower cell bound of the new data being written.

Examples:

```
yes? USE climatological_axes
yes? LET v = L[GT=month_irreg]   ! variable with an irregular axis
yes? SAVE/FILE=mnth.nc/L=1:4 v   ! bounds attribute on time axis.
```

We will be able to append more time steps to the file.

```
yes? SAVE/APPEND/FILE=mnth.nc/L=5:6 v
```

If there is a gap between time steps in the file and the first time step being appended, Ferret inserts a void point whose bounds are the upper bound of the last time in the file, and the lower bound of the first time step being appended. Append steps 10:12 to the file, and then look at the variable and its coordinates:

```
yes? SAVE/APPEND/FILE=mnth.nc/L=10:12 v
yes? CANCEL VAR v
   yes? USE mnth.nc
   yes? LIST v, TBOXLO[gt=v], TBOXHI[gt=v], TBOX[gt=v]

            DATA SET: ./mnth.nc
            TIME: 01-JAN      00:00 to 31-DEC      05:49
 Column  1: V is L[GT=MONTH_IRREG]
 Column  2: TBOXLO is TBOXLO (axis MONTH_IRREG1)
 Column  3: TBOXHI is TBOXHI (axis MONTH_IRREG1)
 Column  4: TBOX is TBOX (axis MONTH_IRREG1)
                         V TBOXLO TBOXHI   TBOX
16-JAN      12 /  1:   1.00    0.0   31.0  31.00
15-FEB      02 /  2:   2.00   31.0   59.2  28.24
15-MAR      17 /  3:   3.00   59.2   90.2  31.00
15-APR      05 /  4:   4.00   90.2  120.2  30.00
15-MAY      17 /  5:   5.00  120.2  151.2  31.00
15-JUN      05 /  6:   6.00  151.2  181.2  30.00
15-AUG      05 /  7:   ....  181.2  273.2  92.00
15-OCT      17 /  8:  10.00  273.2  304.2  31.00
15-NOV      05 /  9:  11.00  304.2  334.2  30.00
15-DEC      17 / 10:  12.00  334.2  365.2  31.00
```

If appending data to a file which has a regular time axis and no bounds, if there is a gap between the time steps in the file and those being appended, older versions of Ferret had always written an axis which is irregular and which has large box sizes at the gap. Now, Ferret will issue a note suggesting that bounds would define the time axis more accurately.

```
   yes? use coads_climatology  ! Has a regular time axis
   yes? save/l=1:4/clobber/file=b.nc sst[x=180,y=0]
   yes? save/append/L=6:9/file=b.nc sst[x=180,y=0]
 *** NOTE: Appending to NetCDF record axis which has no bounds
attribute.
 *** NOTE: This will result in incorrect box sizes on record axis: TIME
 *** NOTE: Write the data initially with the /BOUNDS qualifier
```

In version 6.0 of Ferret, a number of features were added to allow control over attributes written to NetCDF files. In particular you can SET ATTRIBUTE/OUTPUT to force attributes to be written to files. See p. 65 for a complete description. Also see MODE UPCASE_OUTPUT for control over whether variable and attribute names are converted to uppercase.

Ref Sec30. SAY

Alias for MESSAGE/CONTINUE (p. 371)

Ref Sec31. SET

Sets features of the operating environment for program Ferret.

Generally, features may be toggled on and off with SET and CANCEL. Features affected by SET may be examined with SHOW (see also CANCEL, p. 323, and SHOW, p. 431).

Ref Sec31.1. SET ATTRIBUTE

/TYPE /OUTPUT /LIKE

Set or reset attribute characteristics for a variable and/or attribute. (See the section on access to dataset and variable attributes, p. 65)

SET ATTRIBUTE/ TYPE=[float or string] varname.attname

```
yes? SET ATTRIBUTE/TYPE=float varname.attname
```

SET ATTRIBUTE/OUTPUT= varname[.attname] sets a flag for each attribute which determines whether it is written to output netCDF files. It can turn on writing of all or none of the attributes, or /OUTPUT=default gives the standard set of Ferret output attributes. It is applied as follows.

SET ATT/OUTPUT varname.attname	Sets an individual attribute to be written when the variable is written
SET ATT/OUTPUT=all varname	Output all attributes that have been defined for a variable
SET ATT/OUTPUT=default varname	Write only the outputs that Ferret typically writes by default
SET ATT/OUTPUT=none varname	Output no attributes for the variable

Ref Sec31.2. SET AXIS

/MODULO /DEPTH /CALENDAR /T0 /UNITS /STRIDE /OFFSET

Set or resets characteristics of an existing axis.

/DEPTH

Indicates that an axis is to be treated as a depth axis (graphics made with positive down).

```
yes? SET AXIS/DEPTH z_ax
```

/CALENDAR=

Resets the calendar definition for a time axis.The allowed calendars are GREGORIAN, NOLEAP, JULIAN, 360_DAY, and ALL_LEAP. This qualifier is ignored if it is applied to a non-time axis. Applying this qualifier does not change the coordinates of the axis, but only their interpretation. For example, create an axis which is by default a Gregorian calendar axis, containing a leap day. Then reset it to a NOLEAP calendar. The coordinate values are the same but their interpretation as dates is changed.

```
yes? DEFINE AXIS/T=0:100:1/T0="1-JAN-2004 00:00:00"/UNITS=days tax
yes? LIST/L=58:62 T[GT=tax]
             VARIABLE : T
                        axis TAX
             SUBSET   : 5 points (TIME)
 27-FEB-2004 00 / 58:  57.00
 28-FEB-2004 00 / 59:  58.00
 29-FEB-2004 00 / 60:  59.00
 01-MAR-2004 00 / 61:  60.00
 02-MAR-2004 00 / 62:  61.00

yes? SET AXIS/CALENDAR=noleap tax
yes? LIST/L=58:62 T[GT=tax]
             VARIABLE : T
                        axis TAX
             SUBSET   : 5 points (TIME)
             CALENDAR : NOLEAP
 27-FEB-2004 00 / 58:  57.00
 28-FEB-2004 00 / 59:  58.00
 01-MAR-2004 00 / 60:  59.00
 02-MAR-2004 00 / 61:  60.00
 03-MAR-2004 00 / 62:  61.00
```

/MODULO[=LEN]

Indicates that an axis is to be treated as a modulo axis (the first point "wraps" and follows the last point, as in a longitude axis). The optional modulo length is the length in axis units of the modulo repeat. If a length is specified, it may be longer than the axis span, so that the axis is treated as a subspan modulo axis. If it is not specified, the axis length is used as the modulo length. See the sections on modulo axes and subspan modulo axes for more information (p. 167 ff).

```
yes? SET AXIS/MODULO  x_ax
yes? SET AXIS/MODULO/len=365.2425 days_axis
```

/STRIDE=

Indicates that the axis is to be treated everywhere it is used, with the stride specified. The stride value directs Ferret to use every nth point along the axis. If variables and grids from different data sets have this exact same axis, then the striding is applied for all those variables and grids. The striding may be cancelled with CANCEL AXIS/STRIDE. See the discussion of netCDF strides (p. 35.)

/OFFSET=

This qualifier is valid only if /STRIDE= is also specified. Otherwise it is ignored. The value of the offset is the first index used, so that

```
yes? SET AXIS/STRIDE=5/OFFSET=2 my_x_axis
```

is equivalent to this old stride syntax

```
yes? LET new_var = old_var[i=2:1000:5]
```

The previous stride syntax is also still valid.

/T0=

Sets, or resets, the time origin of a time axis.

```
yes? SET AXIS/T0="1-JAN-1990" t_ax
yes? SET AXIS/T0="15-MAR-1950 12:00:00" timeax
```

If the axis is not a time axis, this qualifier is ignored. Applying this qualifier does not change the coordinates of the axis, but only their interpretation. If /T0 is applied to an axis that did not previously have a time origin, the axis will become a calendar-formatted axis, with time coordinates automatically translated into date/time strings on listings, and formatted time axes on plots.

/UNITS=

Sets, or resets, the units of an axis.

```
yes? SET AXIS/UNITS=days t_ax
yes? SET AXIS/UNITS=meters xax
```

Applying this qualifier does not change the coordinates of the axis, but only their interpretation. UNITS are expressed as a string and are converted according to the usual rules for Ferret axes, see the description of units under DEFINE AXIS (p. 344)

Ref Sec31.3. SET DATA_SET

/EZ /VARIABLE /TITLE /FORMAT /GRID /SKIP /COLUMNS /SAVE /RESTORE /ORDER /TYPE /SWAP /REGULART/DELIMITED

SET DATA/EZ /COLUMNS /FORMAT /GRID /SKIP /TITLE /VARIABLE

Specifies ASCII, binary, netCDF, GT, or TS-formatted data set(s) to be analyzed.

1) ASCII or binary:

```
yes? SET DATA/EZ[/qualifiers] data_set1, data_set2, …
      or equivalently, with alias FILE:
yes? FILE[/qualifiers] data_set1, data_set2, ...
```

2) netCDF:

```
yes? SET DATA/FORMAT=cdf  netCDF_file
      or equivalently, with alias USE
yes? USE netCDF_file

      or for a netCDF dataset on a DODS server,
yes? use
"http://www.ferret.noaa.gov/cgi-bin/nph-nc/data/coads_climatology.nc"
```

3) GT or TS-formatted:

```
yes? SET DATA data_set1, data_set2, ...
```

In the case of GT or TS-formatted files, an extension of .des is assumed. A previously SET data set can be SET by its reference number, as shown by SHOW DATA, rather than by name.

If a Unix filename includes a path (with slashes) then the full path plus name must be enclosed in double quotation marks.

```
yes? use "/home/mydirectory/mydata/new_salinity.cdf"
```

If the filename begins with a numeric character, Ferret does not recognize the file, but it can be specified using the Unix pathname, e.g.

```
yes? use "./123"
```

or

```
yes? file/var=a "./45N_180W.dat"
```

Note: Maximum simultaneous data sets: 30 (as of Ferret ver. 5.41). Use CANCEL DATA if the limit is reached.

Command qualifiers for SET DATA_SET:

SET DATA/FORMAT=
Specifies the format of the data set(s) being SET. Allowable values for "file_format" are "cdf", "delimited", "free", "stream", "unformatted", or a FORTRAN format in quotation marks and parentheses.

```
yes? SET DATA/FORMAT=file_format [data_set_name_or_number]
```

Valid arguments for /FORMAT=

1) SET DATA/FORMAT=free (default for SET DATA/EZ)
   To use the format "free" a file must consist entirely of numerical data separated by commas, blanks or tabs.

2) SET DATA/FORMAT=cdf
   If SET DATA/FORMAT=cdf (alias USE) is used, the data file must be in CDF format. The default filename extension is ".cdf", or ".nc"

   Example:

```
yes? SET DATA/FORMAT=CDF  my_netcdf
       or equivalently,
yes? USE  my_netcdf
```

See the chapter "Data Set Basics", section "NetCDF data, p. 34."

Command qualifiers for SET DATA_SET/FORMAT=CDF:

SET DATA /FORMAT=CDF /ORDER=<permutation> /REGULART

The permutation argument contains information both about the order of the axes in the file and the direction.

The order indicated through the /ORDER qualifier should always be exactly the reverse of the order in which the dimensions of variables as revealed by the netCDF ncdump -h command are declared. (This ambiguity reflects the linguistic difference between "C ordering" and "FORTRAN ordering". The default X-Y-Z-T ordering used in the COARDS standard and in Ferret documentation would be referred to as T-Z-Y-X ordering if we used C terminology.)

Thus, to USE a file in Ferret in which the data on disk transposes the X and Y axes we would specify

```
USE/ORDER=YX my_file.nc
```

To use a file in which the data were laid down in XZ "slabs", such as might occur in model outpus we would specify

```
USE/ORDER=XZYT my_model.nc
```

To indicate that the coordinates along a particular axis are reversed from the "right hand rule" ordering, for example a Y axis which runs north to south (not uncommon in image data), we would precede that axis by a minus sign. For example

```
USE/ORDER=X-Y my_flipped_images.nc
```

The minus sign should be applied to the axis position **after** transposition. Thus if a file both transposed the XY axis ordering and used north-to-south ordering in latitude one would access the file with

```
USE/ORDER=Y-X my_transposed_flipped_images.nc
```

NetCDF files, while in principle self-documenting, may be contain axis ambiguities. For example, a file which is supposed to contain a time series, but lacks units on the coordinate variable in the file may appear to be a line of data on the X axis. The /ORDER qualifier can be used to resolve these ambiguities. For this example, one would initialize the file with the command

```
USE/ORDER=T my_ambiguous_time_series.nc
```

Notes for USE/ORDER:

a) Note that specifying USE/ORDER=XYZT is not always equivalent to specifying default ordering. For example, if a netCDF file contained variables on an XYT grid, the /ORDER=XYZT specifaction would tell Ferret to interpret it as an XYZ grid.

b) Also note, the /ORDER qualifier will be ignored if either the file is not netCDF, or the file is netCDF but has an "enhanced" header (see SAVE/HEADING=enhanced, p 367)

SET DATA/FORMAT=cdf/REGULART

Speeds initialization of large netCDF data sets with known regular time axes. When Ferret initializes a netCDF file (the SET DATA/FORMAT=cdf command or the USE command alias), it checks for the attribute point_spacing = "even" on the time axis. If found, Ferret knows that the coordinates are evenly spaced and reads only the first and last coordinate on the axis to obtain a complete description. If not found, Ferret must read the full list of coordinates -- a time-consuming procedure for very large files. After reading the coordinates, Ferret determines if they are regular. The /REGULART qualifier instructs Ferret to treat the time axis as regular regardless of the presence or absence of a point_spacing attribute in the file, speeding up the initialization time on files lacking point_spacing, but known to be regular. If the file happens to contain a bounds attribute on the time axis, and /REGULART is specified, then the bounds are ignored.

Note that when writing netCDF files Ferret, by default, does NOT include the point_spacing attribute. This is because Ferret's default file characteristic is to be append-able, with no guarantees that the appended time steps will be regularly spaced. For output files of fixed length with regular time steps it is advisable to use the SAVE/RIGID qualifier. This allows Ferret to include the point_spacing="even" attribute. If the files will be very large (too large for the full time range to be in memory), then use the /RIGID/TLIMITS= qualifiers to specify the full, ultimate fixed size and use SAVE/APPEND to insert data into the file piecemeal.

If the file has the bounds attribute for the record axis (files written with Ferret version 5.70 and after include the bounds attribute for all irregular axes), then we can append further timesteps to the file and keep the correct point spacing. If there is a gap between the last time in the existing file and the first time being appended, Ferret adds a void point, consisting of missing data, centered on the interval between the upper cell bound of the time axis in the file and the lower cell bound of the new data being written.

3) SET DATA/FORMAT=UNFORMATTED
To use the format "unformatted" the data must be floating point, binary, FORTRAN-style records with all of the desired data beginning on 4-byte boundaries. This option expects 4 bytes of record length information at the beginning and again at the end of each record. The "-" designator (/VARIABLES) can be used to skip over unwanted 4-byte quantities (variables) in each record. See the chapter "Data Set Basics", section "Binary data" (p. 41).

4) SET DATA/FORMAT=FORTRAN format string
FORTRAN format specifications should be surrounded by parentheses and enclosed in quotation marks. Integer format is not supported by Ferret; to read integer data use, for instance, FILE/FORMAT=(f5.0). See also /FORMAT=DELIMITED for reading string data or data with dates which have mixed string and numeric information (p. 402)

The format specified is for reading each record; it cannot be used to specify the total number of values read. That is determined by the grid for the variables. The grid is by default an abstract axis of length 20408, or it may be set explicitly using DEFINE AXIS, DEFINE GRID and SET DATA/GRID= commands. (See p. 404)

Example:

```
yes? SET DATA/EZ/FORMAT="(5X,F12.0)" my_data_set
        or equivalently,
yes? FILE/FORMAT="(5X,F12.0)" my_data_set
```

5) SET DATA/FORMAT=STREAM
/FORMAT=stream is used to indicate that a file contains either unstructured binary output (typical of C program output) or fixed-length records suitable for direct access (all records of equal length, no record length information embedded in the file). With caution it is also possible to read FORTRAN variable-length record output. This sort of file is typically created by "quick and dirty" FORTRAN code which uses the simplest FORTRAN OPEN statement and outputs entire variables with a single WRITE statement.

This format specifier allows you to access any contiguous stretch of 4-byte values from the file. The /SKIP=n qualifier specifies how many values should be skipped at the file start. The /GRID=name qualifier specifies the grid onto which the data should be read and therefore the number of values to be read from the file (the number of points in the grid). Note that an attempt to read more data than the file contains, or to read record length information, will result in a fatal FORTRAN error on UNIX systems and will crash the Ferret program.

For multiple variables, use the /COLUMNS=n specifier to specify how many 4-byte values separate each variable in the file. Each variable is assumed to represent a contiguous stream of values within the file and all variables are assumed to possess the same number of points. (A "poor man's method" is to create multiple Unix soft links pointing to the same file and multiple SET DATA/EZ commands to specify one variable from each link name.)

See the chapter "Data Set Basics", section "Binary data" (p. 41) for further discussion and examples of binary types. Also see the section below under SET DATA/TYPE for a list of the data types that may be read using SET DATA/FORMAT=STREAM (p. 404)

6) SET DATA/FORMAT=DELIMITED

SET DATA/FORMAT=DELIMITED[/DELIMITERS=][/TYPE=][/VAR=] filename

Initializes a file of mixed numerical, string, and date fields. If the data types are not specified the file will be analyzed automatically to determine data types. Using delimited files, the number of variables that can be read from a single file is increased from 20 to 100.

The alias COLUMNS stands for "SET DATA/FORMAT=DELIMITED"

/DELIMITER - list of field delimiters. Default is tab or comma e.g. /DELIM="X,\t,\,". Special characters include

- \b - blank
- \t - tab
- \n - newline
- \nnn - decimal value from ASCII table

/TYPE is the list of data types of the fields. Field types may be any of

- "-" - skipped
- NUMERIC
- TEXT
- LATITUDE - e.g. 87S or 21.5N (interpreted as negative or positive, respectively)
- LONGITUDE - e.g. 160W or 30E (interpreted as negative or positive, respectively)
- DATE - e.g. mm/dd/yy or mm/dd/yyyy or yyyy-mm-dd or yyyymmdd - value returned is days
- from 1-jan-1900 (consistent with the DAYS1900 function)

- EURODATE - e.g. dd/mm/yy or dd/mm/yyyy or yyyy-mm-dd TIME - e.g. hh:mm or hh:mm:ss.s/

See the section on delimited files in the Data Set Basics chapter (p. 50) for examples.

SET DATA/RESTORE
Restores the current default data set number that was saved with SET DATA/SAVE.

This is useful in creating GO files that perform their function and then restore Ferret to its previous state.

SET DATA/SAVE
Saves the current default data set number so it can be restored with SET DATA/RESTORE.

This is useful in creating GO files that perform their function and then restore Ferret to its previous state.

SET DATA/TITLE=
Associates a title with the data set.

```
yes? SET DATA/EZ/TITLE="title string"  file_name
yes? USE/TITLE="pmel data set"
"http://www.ferret.noaa.gov/cgi-bin/nph-nc/data/coads_climatology.nc"
```

For EZ, netCDF, or DODS datasets, set a title. This title appears on plotted outputs at the top of the plot.

SET DATA/EZ
Accesses data from an ASCII or unformatted file that is not in a standardized format (TMAP or netCDF). The command FILE is an alias for SET DATA/EZ.

```
yes? SET DATA/EZ[/qualifiers]   ASCII_or_binary_file
```

or, equivalently,

```
yes? FILE[/qualifiers]  ASCII_or_binary_file
```

Example:

```
yes? FILE/VARIABLE=my_var my_data.dat
```

See the chapter "Data Set Basics", section "ASCII data" (p. 46) for more information and examples. Used on its own, SET DATA/EZ/VAR= uses a default axis length which may be shorter than the size of your data. If this is the case, use DEFINE AXIS and DEFINE GRID commands, and FILE/GRID= to read your data, as discussed in the "Data Set Basics" chapter.

Command qualifiers for SET DATA_SET/EZ:

SET DATA/EZ/COLUMNS=n
Specifies the number of columns in the EZ data file.

By default the number of columns is assumed to be equal to the number of variables (including "-"'s) specified by the /VARIABLES qualifier.

SET DATA/GRID=
Specifies the defining grid for the data in the data set. The argument can be the name of a grid or the name of a variable that is already defined on the desired grid.

Example:

```
yes? SET DATA/EZ/GRID=sst[D=coads] snoopy
```

This is the mechanism by which the shape of the data (1D along T axis, 2D in the XY plane, etc.) is specified. See also the examples in the section Reading ASCII files (p. 46). By default Ferret uses grid EZ, a line of up to 20480 points oriented along the X axis.

SET DATA/SKIP=n

Specifies the number of records to skip at the start of a data set before beginning to read the data. By default, no records are skipped.

For ASCII files a "record" refers to a single line in the file (i.e., a newline character). If the FORMAT statement contains slash characters the "data record" may be multiple lines; the /SKIP qualifier is independent of this fact.

For FORTRAN-structured binary files the /SKIP argument refers to the number of binary records to be skipped.

For unstructured (stream) binary files (e.g., output of a C program) the /SKIP argument refers to the number of words (4-byte quantities) to skip before reading begins.

SET DATA/SWAP
Stream files only. Change the byte ordering of numbers read from the file; big-endian numbers are converted to little-endian numbers and vice versa.

SET DATA/TYPE=
For ASCII delimited files, /TYPE specifies the type of each variable to be read (see p. 402). For unstructured (stream) binary files (p. 401), /TYPE specifies the data type of a set of variables in the file. For stream files, the available values and their corresponding types are:

Value	FORTRAN	C	size in bytes
i1	INTEGER*1	char	1
i2	INTEGER*2	short	2
i4	INTEGER*4	int	4
r4	REAL*4	float	4
r8	REAL*8	double	8

```
yes? SET DATA/EZ/FORMAT=STREAM/TYPE=14,R4/VAR=V1,V2 foobar.dat
```

will read a file containing INTEGER*4 and REAL*4 numbers into the variables v1 and v2.

SET DATA/VARIABLES=
Names the variables of interest in the file. Default is v1.

```
yes? FILE/VARIABLES="var1,var2,..."  file_name
```

Except in the case of /FORMAT=stream, Ferret assumes that successive values in the data file represent successive variables. For example, if there are three variables in a file, the first value represents the first variable, the second represents the second variable, the third the third variable, and the fourth returns to representing the first variable. The maximum number of variables allowed in a single free-formatted data set is 20. See SET DATA/FORMAT=DELIMITED (p. 402) for reading from a delimited file.

Variable names may be 1 to 24 characters (letters, digits, $, and _) beginning with a letter. To indicate a column is not of interest use "-" for its name.

Example: (the third column of data will be ignored)

```
yes? SET DATA/VARIABLES="temp,salt,-,u,v" ocean_file.dat
```

SET DATA/ORDER= (Ferret version 3.11)
Specifies the order (ORDER=permutation) in which axes are to be read.

Examples:

```
yes? FILE/ORDER=XY sst          !X varies fastest
yes? LIST/ORDER=YX sst          !Y varies fastest
```

The "permutation" string may be any permutation of the letters X, Y, Z, and T. If the /format=stream qualifier is used, the string may also contain V (for variable). This allows variables to be "interleaved."

Ref Sec31.4. SET EXPRESSION

Specifies the default context expression. When Ferret's "action" commands (PLOT, CONTOUR, SHADE, VECTOR, WIRE, etc.) are issued with no argument, the default context expression is used. This is the expression last used as argument to an action command, or it may be set explicitly with SET EXPRESSION. See the chapter "Variables and Expressions", section "Expressions" (p. 73) for a full list of action commands.

```
yes? SET EXPRESSION expr1 , expr2 , ...
```

Examples:

1) ***yes? SET EXPRESSION temp***
   Sets the current expression to "temp".

2) ***yes? SET EXPRESSION u , v , u^2 + v^2***
   Set the current expressions to "u , v , u^2 + v^2"

Ref Sec31.5. SET GRID

/RESTORE /SAVE

Specifies the default grid for abstract expressions. Type ***GO wire_frame_demo*** at the Ferret prompt for an example of usage.

```
yes? SET GRID[/qualifier] [grid_or_variable_name]
```

Examples:

```
yes? SET GRID sst[D=coads]

yes? SET GRID           ! use grid from last data accessed
```

See the chapter "Grids and Regions" (p. 145).

Command qualifiers for SET GRID:

SET GRID/RESTORE
Restores the current default grid last saved by SET GRID/SAVE. Useful together with SET GRID/SAVE to create GO files that restore the state of Ferret when they conclude.

SET GRID/SAVE
Saves the current default grid to be restored later. Useful together with SET GRID/RESTORE to create GO files that restore the state of Ferret when they conclude.

When using curvilinear data, call the script mp_grid varnam.jnl" to associate the grid with the map region.

Example:

```
yes? use coads_climagoloty

yes? set region/x=40e:110e/y=60s:20s/L=6
yes? go mp_stereographic_north

yes? go mp_aspect
yes? go mp_grid
yes? shade/noaxes sst, x_page, y_page
```

Ref Sec31.6. SET LIST

/APPEND /FILE /FORMAT /HEADING /PRECISION

Uses SET LIST to specify the default characteristics of listed output.

```
yes? SET LIST/qualifiers
```

The state of the list command may be examined with SHOW LIST. See command CANCEL LIST (p. 326) and LIST (p. 364).

Command qualifiers for SET LIST:

SET LIST/APPEND
Specifies that by default the listed output is to be appended to a pre-existing file. Cancel this state with CANCEL LIST/APPEND.

SET LIST/FILE=
Specifies a default file for the output of the LIST command.

```
yes? SET LIST/FILE=filename
```

The filename specified in this way is a default only. It will be used by the command

```
yes? LIST/FILE variable
```
but will be ignored in
```
yes? LIST/FILE=snoopy.dat variable
```

Ferret generates a filename based on the data set, variable, and region if the filename specified is "AUTO". The resulting name is often quite long but may be shortened by following "AUTO" with a minus sign and the name(s) of the axes to exclude from the filename.

Note: the region information is not used in automatic netCDF output filenames.

Examples:

```
yes? SET LIST/FILE=AUTO
yes? LIST/L=500/X=140W:110W/Y=2S:2N/FILE sst[D=coads]
```

Sends data to file WcoadsSST.X140W110WY2S2NL500.

```
yes? SET LIST/FILE=AUTO-XY
yes? LIST/L=500/X=140W:110W/Y=2S:2N/FILE sst[D=coads]
```

Sends data to file WcoadsSST.L500.

SET LIST/FORMAT=
Specifies an output format for the LIST command. (When a FORTRAN format is specified the row and column headings are omitted from the output.)

```
yes? SET LIST/FORMAT=option
yes? SET LIST/FORMAT        !reactivate previous format
```

Options

FORTRAN format	produces ASCII output
"UNFORMATTED"	produces unformatted (binary) output
"CDF"	produces netCDF output
"GT"	produces TMAP GT format

Examples:

1) ***yes? SET LIST/FORMAT=(1X,12F6.1)***
Specifies a FORTRAN format (without row or column headings).

2) ***yes? SET LIST/FORMAT=UNFORMATTED***
Specifies binary output. (FORTRAN variable record length record structure.)

Notes:

- When using GT format all variables named in a single LIST command will be put into a single GT-formatted timestep.
- Very limited error checking will be done on FORTRAN formats.
- FORTRAN formats are reused as necessary to output full record.
- Latitude axes are listed south to north when /FORMAT is specified.

SET LIST/HEAD
Specifies that ASCII output is to be preceded by a heading that documents data set, variable, and region. Cancel the heading with CANCEL LIST/HEAD.

SET LIST/PRECISION
Specifies the data precision (number of significant digits) of the output listings. This qualifier has no effect when /FORMAT= is specified.

```
yes? SET LIST/PRECISION=#_of_digits
```

Note that this controls only the output precision of the data. and that in fact at most 7 digits are significant since Ferret calculations are performed in single precision.

Ref Sec31.7. SET MEMORY

/SIZE

```
yes? SET MEMORY/SIZE=megawords
```

The command SET MEMORY provides control over how much "physical" memory Ferret can use. (In reality the distinction between physical and virtual memory is invisible to Ferret. The SET MEMORY command merely dictates how much memory Ferret can attempt to allocate from the operating system.)

SET MEMORY controls only the size of Ferret's cache memory—memory used to hold intermediate results from computations that are in progress and used to hold the results of past file IO and computations for re-use. The default size of the memory cache is 6.4 megawords (equivalently, 6.4×4=25.6 megabytes). Cache memory size can be set larger or smaller than this figure.

Example:

```
yes? SET MEMORY/SIZE=8.2
```

Sets the size of Ferret's memory cache to 8.2 million (4-byte) words.

Notes:

- As a practical matter memory size should not normally be set larger than the physical memory available on the system.
- The effect of SET MEMORY/SIZE= is identical to the "-memsize" qualifier on the Ferret command line.
- See SET MODE DESPERATE (p. 412) and MEMORY USAGE (p. 259) in this users guide for further instructions on setting the memory cache size appropriately.
- Using the SET MEMORY command automatically resets the value of SET MODE DESPERATE to a default that is consistent with the memory size.
- The effects of SET MEMORY/SIZE last only for the current Ferret session. Exiting Ferret and restarting will reset the memory cache to its default size.
- If memory is severely limited on a system Ferret's default memory cache size may be too large to permit execution. In this case use the "-memsize" qualifier on the command line to specify a smaller cache.

Ref Sec31.8. SET MODE

/LAST

Specifies special operating modes or states for program Ferret.

```
yes? SET MODE[/LAST] mode_name[:argument]
```

Mode	Description	Default State
ASCII_FONT	imposes PPLUS ASCII font types on plot labels	set
CALENDAR	uses date strings for T axis (vs. time step values)	set
DEPTH_LABEL	uses "DEPTH" as Z axis label	set
DESPERATE	attempts calculations too large for memory	canceled
DIAGNOSTIC	turns on internal program diagnostic output	canceled
GRATICULE	sets drawing of graticule lines on all subsequent plots	cancelled
GUI	unsupported; used in GUI development	
IGNORE_ERROR	continues command file after errors	canceled
INTERPOLATE	automatically interpolates data between planes	canceled
JOURNAL	records keyboard commands in a journal file	set
LATIT_LABEL	uses "N" "S" notation for labeling latitudes	set
LONG_LABEL	uses "E" "W" notation for labeling longitudes	set
METAFILE	captures graphics in GKS metafiles	canceled
PPLLIST	listed output from PPLUS is directed to the named file	canceled
REFRESH	refreshes graphics on systems lacking "backing store"	canceled
SEGMENT	utilizes GKS segment storage	set
STUPID	controls cache hits in memory (diagnostic)	canceled
VERIFY	displays each command file line as it is executed	set
WAIT	waits for carriage return after each plot	canceled

Command qualifiers for SET MODE:

SET MODE/LAST
Resets mode to its last state.

```
yes? SET MODE/LAST mode_name
```

Example: (a command file that will not alter Ferret modes)

```
yes? SET MODE IGNORE_ERRORS      ! 1st line of command file
        .
        . ... code which may encounter errors
        .
yes? SET MODE/LAST IGNORE_ERRORS ! last line of command file
```

Ref Sec31.8.1. SET MODE ASCII_FONT

The SET MODE ASCII_FONT command causes program Ferret to precede plot labels with the PPLUS font descriptor "@AS" (ASCII SIMPLEX font). This assures that special charac-

ters (e.g., underscores) are faithfully reproduced. For special plots it may be desirable to use other fonts (e.g., to obtain subscripts). CANCEL MODE ASCII_FONT is for these cases.

default state: set

Ref Sec31.8.2. SET MODE CALENDAR

SET MODE CALENDAR causes program Ferret to output times in date/time format (instead of time axis time step values). This affects both plotted and listed output.

This mode accepts an optional argument specifying the degree of precision for the output date. If the argument is omitted the precision is unchanged from its last value.

default state: set (argument: minutes)

Arguments

SET MODE CALENDAR accepts the following arguments:

Argument	Equivalent precision
SECONDS	-6
MINUTES	-5 (default)
HOURS	-4
DAYS	-3
MONTHS	-2
YEARS	-1

The argument is uniquely identified by the first two characters.

Example:

```
yes? SET MODE CALENDAR:DAYS
```

Causes times to be displayed in the format dd-mmm-yyyy.

When CALENDAR mode is canceled the "equivalent" in the table above determines the precision of the time steps displayed exactly as in SET MODE LONGITUDE.

Ref Sec31.8.3. SET MODE DEPTH_LABEL

SET MODE DEPTH_LABEL causes Ferret to label Z coordinate information in the units of the Z axis. This affects both plotted and listed output. This mode accepts an optional argument

specifying the degree of precision for the output. If the argument is omitted the precision is unchanged from its last value.

```
yes? SET MODE DEPTH:argument
```

default state: set (argument: -4)

Arguments

See SET MODE LONG (p. 415) for a detailed description of precision control.

Ref Sec31.8.4. SET MODE DESPERATE

Ferret checks the size of the component data required for a calculation in advance of performing the calculation. If the size of the component data exceeds the value of the MODE DESPERATE argument Ferret attempts to perform the calculation in pieces.

For example, the calculation "LIST/I=1/J=1 U[K=1:100,L=1:1000@AVE]" requires 100*1000=100,000 points of component data although the result is only a line of 100 points on the K axis. If 100,000 exceeds the current value of the MODE DESPERATE argument Ferret splits this calculation into smaller sized chunks along the K axis, say, K=1:50 in the first chunk and K=51:100 in the second.

Ferret is also sensitive to the performance penalties associated with reading data from the disk. Splitting the calculation along axis of the stored data records can require the data to be read many times in order to complete the calculation. Ferret attempts to split calculations along efficient axes, and will split along the axis of stored data only in desperation, if MODE DESPERATE is SET.

Example:

```
yes? SET MODE DESPERATE:5000
```

default state: canceled (default argument: 80000)

Note: Use MODE DIAGNOSTIC to see when splitting is occurring.

Arguments

Use SHOW MEMORY/FREE to see the total memory available (as set with SET MEMORY/SIZE).

Whenever the size of memory is set using SET MEMORY the MODE DESPERATE argument is reset at one tenth of memory size. For most purposes this will be an appropriate value. The user may at his discretion raise or lower the MODE DESPERATE value based on the nature of

a calculation. A complex calculation, with many intermediate variables, may require a smaller value of MODE DESPERATE to avoid an "insufficient memory" error. A simple calculation, such as the averaging operation described above, will typically run faster with a larger MODE DESPERATE value. The upper bound for the argument is the size of memory. The lower bound is "memory block size."

Ref Sec31.8.5. SET MODE DIAGNOSTIC

SET MODE DIAGNOSTIC causes Ferret to display diagnostic information in real time about its internal functioning. It is intended to help Ferret developers diagnose performance problems by displaying what the Ferret memory management subsystem is doing. The message "strip gathering on xxx axis" indicates that Ferret has broken up a calculation into smaller pieces. Subsequent "strip" and "gathering" messages indicate that sub-regions of the calculations are being processed and brought together.

default state: canceled

See the FAQ, How do I interpret the output of "SET MODE DIAGNOSTIC? for help interpreting the output.

Ref Sec31.8.6. SET MODE GRATICULE

SET MODE GRATICULE turns on plotting of graticule lines at the tic marks of both the horizontal and vertical axes of all subsequent plots. In effect this is equivalent to specifying the GRATICULE qualifier for all plot commands (See p. 379). You may specify line types with SET MODE GRATICULE:(type) where type is color, dash, or thickness

Examples:

```
yes? SET MODE GRATICULE
yes? SET MODE GRATICULE:(color=red)
yes? SET MODE GRATICULE:(dash,color=blue)
yes? SET MODE GRAT:"LARGE(COLOR=blue),SMALL(COLOR=lightblue)"
```

Currently, dash types cannot be customized, as they can for PLOT/DASH. Look for this in a future release of Ferret.

Ref Sec31.8.7. SET MODE IGNORE_ERROR

SET MODE IGNORE_ERROR causes Ferret to continue execution of a command file despite errors encountered. (See command GO, p. 359.)

default state: canceled

Ref Sec31.8.8. SET MODE INTERPOLATE

Note: The transformation @ITP provides the same functionality as MODE INTERPOLATE with a greater level of control.

SET MODE INTERPOLATE affects the interpretation of world coordinate specifiers (/X, /Y, /Z, and /T) in cases where the position is normal to the plane in which the data is being examined. When this mode is SET and a world coordinate is specified which does not lie exactly on a grid point, Ferret automatically interpolates from the surrounding grid point values. When this mode is canceled, the same world coordinate specification is shifted to the grid point of the grid box that contained it before computations were made (see examples).

default state: canceled

Example:

If the grid underlying the variable temp has points defined at Z=5 and at Z=15 (with the grid box boundary at Z=10) and data is requested at Z=12 then

```
yes? SET MODE INTERPOLATE
yes? LIST/T=18249/X=130W:125W/Y=0:3N/Z=12 temp
```

lists temperature data in the X-Y plane obtained by interpolating between the Z=5 and Z=15 planes. Whereas,

```
yes? CANCEL MODE INTERPOLATE
yes? LIST/T=18249/X=130W:125W/Y=0:3N/Z=12 temp
```

lists the data at Z=15. The output documentation always reflects the true location used.

Ref Sec31.8.9. SET MODE LABELS

SET MODE LABELS restores the default behavior of labeling, if CANCEL MODE LABELS has been issued. CANCEL MODE LABELS implements the /NOLABELS qualifier for all plots after it has been set.

Ref Sec31.8.10. SET MODE LOGO

SET MODE LOGO turns on the Ferret logo (three lines at the upper right of plots), if it has been turned off by CANCEL MODE LOGO

Ref Sec31.8.11. SET MODE JOURNAL

SET MODE JOURNAL causes Ferret to record all commands issued in a journal file. Output echoed to this file may be turned on and off via mode JOURNAL at any time.

default state: set

Example:

```
yes? SET MODE JOURNAL:my_journal_file.jnl
```

The optional argument to MODE JOURNAL specifies the name of the output journal file—with no argument, the default name "ferret.jnl" is used. Journal files for successive Ferret sessions are handled by version number. See the chapter "Computing Environment", section "Output file naming" (p. 263).

Ref Sec31.8.12. SET MODE LATIT_LABEL

SET MODE LATIT_LABEL causes Ferret to output latitude coordinate information in degrees N/S format (instead of the internal latitude coordinate). This affects both plotted and listed output.

This mode accepts an optional argument specifying the degree of precision for the output. If the argument is omitted the precision is unchanged from its last value.

Example:

```
yes? SET MODE LAT:2
```

default state: set (argument: 1)

Arguments

See command SET MODE LONG (p. 415) for a detailed description of precision control.

Ref Sec31.8.13. SET MODE LONG_LABEL

SET MODE LONG_LABEL causes Ferret to output longitude coordinate information in degrees E/W format (instead of the internal longitude coordinate). This will affect both plotted and listed output.

This mode accepts an optional argument specifying the degree of precision for the output. If the argument is omitted the precision will be unchanged from its last value.

Example:

```
yes? SET MODE LONG:2
```

default state: set (argument: 1)

Arguments

The argument of SET MODE LONG is an integer specifying the precision. If the argument is positive or zero it specifies the maximum number of decimal places to display. If the argument is negative it specifies the maximum number of significant digits to display.

Examples:

Suppose the longitude to be displayed is 165.23W. Then

```
yes? SET MODE LONG:1    will produce 165.2W
yes? SET MODE LONG:-3   will produce 165W
```

When LONG mode is canceled the argument still determines the output precision.

Ref Sec31.8.14. SET MODE METAFILE

The optional argumenSET MODE METAFILE causes Ferret to capture all graphics in metafiles. These metafiles can later be routed to various devices to obtain hard copy output or postscript files.

The optional argument to MODE METAFILE specifies the name of the output metafile—with no argument, the default name "metafile.plt" is used. Multiple output files (i.e., successive plots) are handled by version number. See the chapter "Computing Environment", section "Output file naming" (p. 263).

See the chapter "Computing Environment", section "Hard copy" (p. 260) for details on generating hard copy.

Example:

```
yes? SET MODE METAFILE:june_sst.plt
```

default state: canceled (default argument when set: "metafile.plt")

Ref Sec31.8.15. SET MODE PPLLIST

Directs listed output from PPLUS commands (e.g., PPL LIST LABS) to the specified file. This mode is useful for creating scripts that customize plots. The user can specify the name of the output file by giving it as an argument, otherwise file name "ppllist.out" is assigned.

Example:

```
yes? SET MODE PPLLIST:plot_symbols.txt
yes? PPL LISTSYM
yes? SPAWN grep "WIDTH" plot_symbols.txt
```

default state: canceled

Ref Sec31.8.16. SET MODE REFRESH

The SET MODE REFRESH command causes Ferret to update windows following "occlusion" events on X-servers that lack a backing store (SGI workstations have been a case in point).

default state: canceled (except on SGI systems)

Ref Sec31.8.17. SET MODE SEGMENTS

SET MODE SEGMENTS causes Ferret to utilize GKS segments ("GKS" is the Graphical Kernel System—an international graphics standard). On some systems MODE SEGMENTS may be necessary to update windows following "occlusion" events or to resize window with the mouse.

Segments, however, make heavy demands on the system's virtual memory. If Ferret crashes during graphics output due to insufficient virtual memory try CANCEL MODE SEGMENTS.

default state: set

Ref Sec31.8.18. SET MODE STUPID

Note: MODE STUPID is included for diagnostic purposes only.

SET MODE STUPID controls the ability of Ferret to reuse results left in memory from previous commands. It also effects its ability to reuse intermediate variables that are referenced multiple times during complex calculations. Given with no argument

```
yes? SET MODE STUPID
```

causes Ferret to forget data cached in memory. The result is that all requests for variables are read from disk and no intermediate calculations are reused. The program will be significantly slower as a result.

A lesser degree of cache limitation occurs with the command

```
yes? SET MODE STUPID: weak_cache
```

which causes Ferret to revert to the cache access strategy that it used previous to Ferret version 5.0. In this mode cache hits are unreliable unless the region of interest is fully specified. (Unspecified limits will typically default to the full range of the relevant axis.)

default state: canceled

Ref Sec31.8.19. SET MODE UPCASE_OUTPUT

By default Ferret changes variable and attribute names to uppercase form, and they are written in uppercase to netCDF files. In order to keep the original lowercase or mixed-case spelling of variable and attribute names when they are written to netCDF files, issue the command

yes? CANCEL MODE UPCASE_OUTPUT

and the original upper- or lower-case spellings to be used.

default state: set

Ref Sec31.8.20. SET MODE VERIFY

SET MODE VERIFY causes commands from a command file ("GO file") to be displayed on the screen as they are executed. Note that if MODE VERIFY is canceled, loop counting in the REPEAT command is turned off.

default state: SET, argument "default"

Note: Many GO files begin with CANCEL MODE VERIFY to inhibit output and end with SET MODE/LAST VERIFY to restore the previous state. Only if an error or interrupt occurs during the execution of such a command file will the state of MODE VERIFY be affected.

SET MODE VERIFY can accept arguments to further refine control over command echoing.

```
yes? SET MODE VERIFY: DEFAULT
```

- This will be the default state if no argument is given
- Ferret echos commands taken from GO scripts

- Ferret echos commands in which symbol substitutions occur or in which embedded expressions are evaluated

```
yes? SET MODE VERIFY: ALL
```

- In addition to the cases above Ferret also displays the individual commands that are generated by repeat loops and semicolon-separated command groups
- Ferret displays a REPEAT loop counter ("!-> REPEAT: ...")

```
yes? SET MODE VERIFY: ALWAYS
```

- Echoing behavior is the same as argument ALL but ALWAYS, in addition, causes CANCEL MODE VERIFY to be ignored when it is encountered in a GO file. This functionality is useful when debugging GO scripts. Entering CANCEL MODE VERIFY or SET MODE VERIFY:DEFAULT from the command line will cancel this state.

Ref Sec31.8.21. SET MODE WAIT

SET MODE WAIT causes Ferret to wait for a keyboard keystroke from the user after each plotted output is completed. This is useful on graphics terminals that do not have a separate graphics plane; on these terminals SET MODE WAIT prevents the graphical output from being wiped off the screen until the user is ready to proceed.

default state: canceled

Ref Sec31.9. SET MOVIE

/COMPRESS /FILE /LASER /START

Designates a file (specified or default) for storing graphical images as movie frames (in HDF Raster-8 format). Note that the FRAME/FILE=filename qualifier is generally preferable to the SET MOVIE command, as it is simpler and more flexible. See the chapter "Animations and GIF Images (p. 175) for further explanation.

```
yes? SET MOVIE[/qualifiers]
```

Command qualifiers for SET MOVIE:

SET MOVIE/COMPRESS=
Turns on or off compression of HDF frames using run length compression.

```
yes? SET MOVIE/COMPRESS=OFF
```

The allowed arguments are "on" and "off" —CANCEL MOVIE does not affect this qualifier.

default state: on

SET MOVIE/FILE
Specify an output file to receive movie frames.

```
!specify a new filename
yes?  SET MOVIE/FILE=filename
      or
!reactivate a previously specified filename after CANCEL MOVIE
yes?  SET MOVIE/FILE
```

The default movie filename extension is ".mgm"

The default movie filename is "ferret.mgm"

SET MOVIE/LASER
Output to Panasonic OMDR. Valid only on older VAX/VMS systems.

SET MOVIE/START
Only valid for use on older VAX/VMS systems with the Panasonic Optical Memory Disk Recorder (OMDR). Only valid with /LASER qualifier.

Ref Sec31.10. SET REGION

/I/J/K/L /X/Y/Z/T /DI/DJ/DK/DL /DX/DY/DZ/DT

Specifies the default space-time region for the evaluation of expressions.

```
yes? SET REGION[/qualifiers] [ reg_name]
```

See the chapter "Grids and Regions", section "Regions" (p. 162) for further information.

Examples:

1) ***yes? SET REGION/X=140E***
   Sets X axis position in the default context.

2) ***yes? SET REGION/@N   !N specifies X and Y but not Z or T***
   Sets only X and Y in the default context (since X and Y are defined in region N but Z and T are not).

3) ***yes? SET REGION N***
   Sets ALL AXES in the default region to be exactly the same as region N. Since Z and T are undefined in region N they will be set undefined in the default context.

4) ***yes? SET REGION/@N/Z=50:250***
Sets X and Y in the default region to be exactly the same as region N and then sets Z to the range 50 to 250.

5) ***yes? SET REGION/DZ=-5***
Set the region along the Z axis to be 5 units less than its current value.

6) ***yes? SET REGION/DJ=-10:10***
Increases the current vertical axis range by 10 units on either end of the axis.

Command qualifiers for SET REGION:

SET REGION/I=/J=/K=/L=/X=/Y=/Z=/T=
Sets region bounds for specified axis subscript (I, J, K, or L) or axis coordinates (X, Y, Z, or T). See examples above.

SET REGION/DI=/DJ=/DK=/DL=/DX=/DY=/DZ=/DT=
Modifies current region information by the specified increment of an axis subscript (I, J, K, or L) or axis coordinate (X, Y, Z, or T). See examples above. Syntax: /D*=val, or /D*=lo:hi.

Ref Sec31.11. SET VARIABLE

/BAD /GRID /TITLE /UNIT /DATASET/NAME/OFFSET/SCALE

Modifies characteristics of a variable defined by DEFINE VARIABLE or SET DATA/EZ. This command permits variables within a single EZ data set to be defined on different grids and it allows the titles and units to be superseded for the duration of a session, only, on netCDF and GT data sets.

```
yes? SET VARIABLE/qualifiers variable_name
```

Parameters

The variable name can be a simple name or a name qualified by a data set.

Example:

```
yes? SET VAR/UNITS="CM" WIDTH[D=snoopy]
```

Command qualifiers for SET VARIABLE:

Ref Sec31.11.1. SET VARIABLE/BAD=

Designates a value to be used as the missing data flag. The qualifier is applicable to EZ data set variables and to netCDF data sets. The bad value which is specified will be used in subsequent outputs and calculations involving this variable. It applies only for the duration of the current Ferret session. It does not alter the data files. It is not applicable to variables defined with DEFINE VARIABLE.

For netCDF files only, if the file contains Fortran NaN (not a number), the user may designate NaN as the bad value flag for a given variable in a netCDF dataset. SET VARIABLE/BAD=NAN.

When the command SET VARIABLE/BAD= is be used to set one of the two missing value flags of a file variable, the bad value which is specified will be used in subsequent outputs and calculations involving this variable.

Ferret is aware of up to two missing value flags for each variable in a netCDF file. Under most circumstances, netCDF file variables use only a single flag. With a command like

```
SET VARIABLE/BAD=-999 my_file_var
```

you can specify -999 as an additional missing value flag. It is this value which will be present in all subsequent SAVEs to files and calculations.

Note that if the netCDF file contains two distinct flag values specified by the netCDF attributes "missing_value" and "_FillValue", then this command will migrate the value specified by missing_value to the position previously occupied by _FillValue and replace the one specified by missing_value. Thus a double usage of this command allows you to control both flags. You can use the command "SHOW DATA/VARIABLES" to see both bad value flags.

Ref Sec31.11.2. SET VARIABLE/GRID=

Sets the defining grid for a variable in an EZ data set. The argument may be an expression.

Example:

```
yes? SET VARIABLE/GRID=my_grid width[D=snoopy]
```

This is the mechanism by which the shape of the data (1D along T axis, 2D in the XY plane, etc.) is specified. By default Ferret will use grid EZ, a line of up to 20480 points oriented along the X axis. The qualifier is not applicable to variables defined with DEFINE VARIABLE.

SET VARIABLE/NAME=
This is effectively a RENAME command -- applies to all classes of variables (but not pseudo-variables). Useful for "repairing" variables whose definitions are inadequate as-is but whose variable names are significant. A common application of this is in creating output netCDF files in which contain modified versions of variables from input files.

```
yes? SET VARIABLE/NAME=new  old
```

Example:

```
yes? SET VARIABLE/NAME=north_vel V[d=1]
```

Ref Sec31.11.3. SET VARIABLE/TITLE=

Associates a title with the variable. This title appears on plotted outputs and listings. The qualifier is applicable to all variables.

```
yes? SET VARIABLE/TITLE="title string" var_name
```

SET VARIABLE/UNITS=
Associates units with the variable. The units appear on plotted outputs and listings. The qualifier is applicable to all variables.

```
yes? SET VARIABLE/UNITS="units string" var_name
```

Ref Sec31.11.4. SET VARIABLE/OFFSET=

For variables in netCDF files only, set an offset which will always be applied when the data is read from the file. This offset is applied after any offset factor specified by an attribute in the file. It is not applied to missing data. For applying a scale and offset to data when writing data to a netCDF file see p. 73.

An application of this is when we want to use a variable to define an axis, but the variations in the data aren't well represented by a single precision variable. This is described in the section on DEFINE VARIABLE/FROM_DATA (p. 343).

Ref Sec31.11.5. SET VARIABLE/SCALE=

For variables in netCDF files only, set a scale factor which will always be applied when the data is read from the file. This factor is applied after any scale factor specified by an attribute in the file. It is not applied to missing data.

When a scale factor and offset are specified, the scale factor is applied first:

```
var = scale*var_in + offset
```

If scale and offset values have been set, with either netCDF attributes or with a SET VARIABLE/SCALE=/OFFSET=, they may be accessed with the RETURN= keyword.

Example:

```
yes? USE coads_climatology
yes? SAY `sst,RETURN=nc_scale`
 !-> MESSAGE/CONTINUE 1
1
yes? SAY `sst,RETURN=nc_offset`
 !-> MESSAGE/CONTINUE 0
0
yes? SET VAR/SCALE=10/OFF=50 sst
yes? SAY `sst,RETURN=user_off`
 !-> MESSAGE/CONTINUE 50
50
yes? SAY `sst,RETURN=user_scale`
 !-> MESSAGE/CONTINUE 10
10
```

Ref Sec31.12. SET VIEWPORT

Sets the rectangular region within the output window where output will be drawn.

```
yes? SET VIEWPORT view_name
```

Issuing the command SET VIEWPORT is best thought of as entering "viewport mode." While in viewport mode all previously drawn viewports remain on the screen until explicitly cleared with either SET WINDOW/CLEAR or CANCEL VIEWPORT. If multiple plots are drawn in a single viewport without the use of /OVERLAY the current plot will erase and replace the previous one; the graphics in other viewports will be affected only if the viewports overlap. If viewports overlap the most recently drawn graphics will always lie on top, possibly obscuring what is underneath. By default, the state of "viewport mode" is canceled.

Pre-defined viewports exist for dividing the window into four quadrants and for dividing the window in half horizontally and vertically. See the chapter "Customizing Plots", section "Pre-defined viewports" (p. 210) for a list.

Ref Sec31.13. SET WINDOW

/ASPECT /CLEAR /LOCATION /NEW /SIZE

Creates, resizes, reshapes or moves graphics output windows.

```
yes? SET WINDOW[/qualifiers]  [window_number]
```

Note: Multiple windows may be simultaneously viewable but only a single window receives output at any time.

See commands SHOW WINDOWS (p. 443) and CANCEL WINDOW (p. 330) for additional information.

Examples:

1) ***yes? SET WINDOW/NEW***
   Creates a new output window and sends subsequent graphics to it.

2) ***yes? SET WINDOW 3***
   Sends subsequent graphics to window 3.

3) ***yes? SET WINDOW/SIZE=.5***
   Resizes current window to ½ of full.

4) ***yes? SET WINDOW/ASPECT=.5***
   Reshapes current window with Y/X equal to 1:2. The effect of this command is not seen until a plot is sent to the window.

5) ***yes? SET WINDOW/LOCATION=0,.5***
   Puts the lower left corner of the current window at the left border of the display and half way up it.

Command qualifiers for SET WINDOW:

SET WINDOW/ASPECT
Sets the aspect ratio of the output window and hard copy. Note the new ratio doesn't take effect until a plot command is issued in the window.

Examples:

1) ***yes? SET WINDOW/ASPECT=y_over_x n***
   Sets the overall aspect ratio of window n.

2) ***yes? SET WINDOW/ASPECT=y_over_x***
   Sets the overall aspect ratio of the current window.

3) ***yes? SET WINDOW/ASPECT=y_over_x:AXIS***
   Sets the axis length aspect ratio of the current window.

The total size (area) of the output window is not changed.
The default value for the overall window ratio is y/x = 8.8/10.2 ~ 0.86.

The default value for the axis length ratio is y/x = 6/8 = 0.75.
Use PPLUS/RESET or SET WINDOW/ASPECT=.75:AXIS to restore defaults.
The aspect ratio specified is a default for future SET WINDOW commands
The origin (lower left) is restored to its default values: 1.2, 1.4

When using "SET WINDOW n" to return to a previous window that differs from the current window in aspect ratio, it is necessary to re-specify its aspect ratio with /ASPECT, otherwise PPLUS will not be properly reset. If you return to a previous window, you cannot expect to make an overlay on the plot that is there. The PPLUS settings for axis lengths and other properties of the plot will have been overwritten.

SET WINDOW/CLEAR
Clears the image(s) in the current or specified window. Useful with viewports.

SET WINDOW/LOCATION
Sets the location for the lower left corner of named (or current) window. The coordinates x and y must be values between 0 and 1 and refer to distances from the lower left corner of the display screen (total length and width of which are each 1).

```
yes? SET WINDOW/LOCATION=x,y [window_number]
```

SET WINDOW/NEW
Causes future graphical output to be directed to a new window. The window will be created at the next graphics output.

```
yes? SET WINDOW/NEW
```

SET WINDOW/SIZE
Resizes a window to r times the area of the standard window. (The length of the sides changes by the square root of r.) If the window number is omitted the command will resize the currently active window. (The default window size is 0.7.)

```
yes? SET WINDOW/SIZE=r [window_number]
```

The actual size of the window is reset to fit on the output device.

Ref Sec32. SHADE

/I/J/K/L /X/Y/Z/T /D /FRAME /KEY /LEVELS /LINE /NOAXIS /NOKEY /NOLABELS /OVERLAY /PALETTE /PATTERN /SET_UP /TITLE /TRANSPOSE /MODULO /HLIMITS /VLIMITS /AXES

Produces a shaded (rectangular raster) plot of a 2-D field. By default a color key is drawn and contour lines are not drawn.

```
SHADE[/qualifiers]  expression
```

In a curvilinear coordinate system (map projections)

```
SHADE [/qualifiers]   expression, xcoords, ycoords (see p. 222)
```

Parameters

The expression may be any valid expression. See the chapter "Variables and Expressions", section "Expressions" (p. 73) for a definition of valid expressions. The expression will be inferred from the current context if omitted from the command line. Multiple expressions are not permitted in a single SHADE command.

Command qualifiers for SHADE:

SHADE/I=/J=/K=/L=/X=/Y=/Z=/T=
Specifies value or range of axis subscripts (I, J, K, or L), or axis coordinates (X, Y, Z, or T) to be used when evaluating the expression being plotted.

SHADE/D=
Specifies the default data set to be used when evaluating the expression being plotted.

This qualifier does not have any impact on the context of the expression being plotted. If data is on a modulo x axis but the arguments of the /HLIMITS qualifier represent a region outside the actual coordinates of the data, only the range including the actual coordinates is shown. Use /X=lo:hi or SET REGION or a context on the variable itself, var[X=lo:hi], to set the context for the expression.

SHADE/FRAME
Causes the graphic image produced by the command to be captured as an animation frame in the file specified by SET MOVIE. In general the FRAME command (p. 358) is more flexible and we recommend its use rather than this qualifier.

SHADE/KEY
Displays a color key for the palette used in the shaded plot. By default a key is drawn unless the /LINE or /NOKEY qualifier is specified. To control the color key position and labeling, see the command SHAKEY in the appendix, "Ferret Enhancements to PPLUS" (p. 561).

SHADE/KEY=CONTINUOUS
Chooses a continous color key for the palette used in a shade plot, without lines separating the colors. This option is particularly good for plots having many levels.

SHADE/LEVELS
Specifies the SHADE levels or how the levels will be determined. If the /LEVELS qualifier is omitted Ferret automatically selects reasonable SHADE levels.

See the chapter "Customizing Plots", section "Contouring" (p. 213) for examples and more documentation on /LEVELS, and also the demonstration "custom_contour_demo.jnl".

SHADE/LINE
Overlays contour lines on a shaded plot. When /LINE is specified the color key is omitted unless specifically requested via /KEY.

SHADE/NOKEY
Suppresses the drawing of a color key for the palette used in the plot.

SHADE/NOAXIS
Suppresses all axis lines, tics and labeling so that no box appears surrounding the contour plot. This is especially useful for map projection plots.

SHADE/NOLABELS
Suppresses all plot labels.

SHADE/OVERLAY
Causes the indicated shaded plot to be overlaid on the existing plot.

Note (SHADE/OVERLAY with time axes):
A restriction in PPLUS requires that if time is an axis of the shaded plot, the overlaid variable must share the same time axis encoding as the base plot variable. If this condition is not met, you may find that the overlaid shaded plot fails to be drawn. The solution is to use the Ferret regridding capability to regrid the base plot variable and the overlaid plot variable onto the same time axis. See the section on overlaying with a time axis (p. 189).

SHADE/PALLETTE=
Specifies a color palette (otherwise, a default rainbow palette is used). Try the Unix command ***% Fpalette '*'*** to see available palettes. The file suffix *.spk is not necessary when specifying a palette. See command PALETTE (p. 371) for more information.

```
Yes? SHADE/PALETTE=land_sea   rose
```

The /PALETTE qualifier changes the current palette for the duration of the plotting command and then restores the previous palette. This behavior is not immediately compatible with the /SET_UP qualifier. See the PALETTE (p. 371) command for further discussion.

SHADE/PATTERN=
Specifies a pattern file (otherwise, the current default pattern specification is used). The file suffix *.pat is not necessary when specifying a pattern. Try the Unix command ***% Fpattern '*'*** to see available patterns. See command PATTERN (p. 372) for more information.

SHADE/SET_UP
Performs all the internal preparations required by program Ferret for a shaded plot but does not actually render output. The command PPL can then be used to make changes to the plot prior to

producing output with the PPL SHADE command. This permits plot customizations that are not possible with Ferret command qualifiers. See the chapter "Customizing Plots" (p. 183).

Example: Customize the colorbar key for a shade plot. By default the color key is positioned to the right of the plot, oriented vertically. PPL SHAKEY offers a means to change its location and size, and to customize the labels. Note that the position of the shade key colorbar is in units of inches from the page origin. These commands put the color key at the top of the plot, and remove the labels that would by default be there. The value -0.12 for the label size tells ferret to put the key labels above the colorbar.

```
yes? use coads_climatology
yes? shade/title="shakey labels above"/lev=(0,32,1)/set sst[l=1]
yes? let x1 = `($ppl$xorg)`
yes? let y1 = `($ppl$yorg)+($ppl$ylen)`
yes? let y2 = `($ppl$yorg)+($ppl$ylen)+0.4`
yes? ppl shakey ,0,-0.12,2,,,`x1`,,`y1`,`y2`
yes? go unlabel 4
yes? go unlabel 5
yes? go unlabel 6
yes? ppl shade
yes? ppl list shakey
```

SHADE/TITLE=
Allows user to specify a plot title (enclosed in quotation marks). Without this qualifier Ferret selects a title based on information about the expression(s).

```
yes? SHADE/TITLE="title string"  expression
```

SHADE/TRANSPOSE
Causes the horizontal and vertical axes to be interchanged. By default the X axis is drawn horizontally on the plot and the Y and Z axes are drawn vertically. For Y-Z plots the Z data axis is vertical.

Note that plots in the YT and ZT planes have /TRANSFORM applied by default in order to achieve a horizontal T axis. See /HLIMITS (below) for further details. Use /TRANSPOSE manually to reverse this effect.

SHADE/HLIMITS=
Specifies the horizontal axis range and tic interval (otherwise, Ferret selects reasonable values).

```
yes? SHADE/HLIMITS=lo:hi:increment
```

The optional "increment" parameter determines tic mark spacing on the axis. If the increment is negative, the axis will be reversed.

SHADE/VLIMITS=
Specifies the vertical axis range and tic interval. See /HLIMITS (above)

SHADE/XLIMITS=/YLIMITS=
Note: XLIMITS and YLIMITS have been deprecated. Please use HLIMITS and VLIMITS instead.

SHADE/AXES[=top,bottom,left,right]

Turns plotting of individual axes off and on. This replaces the use of the "PPL AXSET" command. The syntax is

```
yes? SHADE/AXES[=top,bottom,left,right]  var
```

where the arguments are 1 to turn the axis on and 0 to turn it off. For example:

```
yes? SHADE/AXES=0,1,1,0  sst     ! Draws the bottom and left axes only
```

SHADE/MODULO

For curvilinear (3-argument) shade plots only. Beginning with Ferret v5.81 the argument /MODULO for SHADE plots will draw modulo replications in longitude for curvilinear data in order to fill out the specified extent in the longitude direction. For instance, if the xcoords variable contains longitudes in the range -270:90 we can draw a plot with longitudes 0:360 with the command

```
yes? SHADE/HLIMITS=0:360/MODULO values, xcoords, ycoords
```

SHADE/GRATICULE[=line specifiers]

(Introduced in Ferret version 5.6) Turns on graticule lines for the horizontal and vertical axes. These are lines across the plot at the tic marks. /GRATICULE sets both horizontal and vertical lines; to set each separately see /HGRATICULE and /VGRATICULE, below. The syntax is

```
yes? SHADE/GRATICULE[=line or dash,COLOR=,THICKNESS=]  var
```

where the default is a thin, solid black line. The line colors available are Black, Red, Green, Blue, LightBlue, Purple, and White. The thickness codes are 1, 2, or 3 and as for plot lines, thickness=1 is a thin line, thickness=3 is the thickest, and THICK specified with no value defaults to thickness=2. For clarity the arguments to GRAT may be placed in parentheses

```
yes? SHADE/GRAT  sst     ! default graticules

yes? SHADE/GRAT=(LINE,COLOR=red,THIICK=3) sst

yes? SHADE/GRAT=(DASH,COLOR=lightblue) sst

yes? SHADE/FILL/GRAT=(DASH,COLOR=white) sst
```

The above commands make settings for the large tic marks. If small tic marks are being plotted on the axes, we can make settings for them as well using keywords SMALL and LARGE. Place

all of the arguments for the /GRAT qualifier in double quotes. Note that the PPL AXNMTC command sets the plotting of small tics, and that small tics are used by default for many time axes.

```
yes? ppl axnmtc 2,2
yes? SHADE/GRAT="LARGE(COLOR=blue,thick),SMALL(COLOR=lightblue)" sst
```

SHADE/HGRATICULE[=line specifiers]/VGRATICULE[=line specifiers]

Turns on graticule lines and sets the line characteristics of the graticule for the horizontal or vertical axis separately. You may specify only one of /HGRAT or /VGRAT if desired. These are lines across the plot at the tic marks. The syntax is

```
     yes? SHADE/HGRAT[=line or dash,COLOR=,THICKNESS=] /VGRAT=line or
dash,COLOR=,THICKNESS=]  var
```

where the default is a thin, solid black line. The line colors available are Black, Red, Green, Blue, LightBlue, Purple, and White. The thickness codes are 1, 2, or 3 and as for plot lines, thickness=1 is a thin line, thickness=3 is the thickest, and THICK specified with no value defaults to thickness=2. For clarity the arguments to HGRAT may be placed in parentheses

```
yes? SHADE/HGRAT/VGRAT  sst    !this is equivalent to PLOT/GRAT

yes? SHADE/HGRAT=(LINE,COLOR=red,THIICK=3)/VGRAT=(color=green) sst

yes? SHADE/HGRAT=(DASH,COLOR=lightblue) sst  ! horizontal only
```

The above commands make settings for the large tic marks. If small tic marks are being plotted on the axes, we can make settings for them as well using keywords SMALL and LARGE. Place all of the arguments for the /HGRAT qualifier in double quotes. Note that the PPL AXNMTC command sets the plotting of small tics, and that small tics are used by default for many time axes.

```
yes? ppl axnmtc 2,2
yes? SHADE/HGRAT="LARGE(COLOR=blue,thick),SMALL(COLOR=lightblue)"
/VGRAT="LARGE(COLOR=blue,thick) sst
```

Ref Sec33. SHOW

/ALL

Displays program states and stored values.

Command qualifiers for SHOW:

SHOW/ALL
Executes all SHOW options. This command gives a complete description of the current state,

including information about region, grids, axes, variables, and the state of various modes (default or set with SET MODE).

```
yes? SHOW/ALL
```

Arguments:

The names of variables, data sets, or other definitions can be specified using wildcards. The * wildcard matches any number of characters in the name; the question wildcard matches exactly one character.

Ref Sec33.1. SHOW ALIAS

Lists all command aliases and the full command names for which they stand, or, with an argument, shows a specified command alias.

```
yes? SHOW ALIAS  [alias_name]
```

Ref Sec33.2. SHOW ATTRIBUTE

/ALL /DATASET /OUTPUT

```
yes? SHOW ATTRIBUTE varname.attname
yes? SHOW ATTRIBUTE/ALL varname
yes? SHOW ATTRIBUTE/D=1/ALL varname
```

Shows the attribute(s) for a variable. This command is included for convenience; it does not add any capabilities beyond the LIST command, but is useful especially with the /ALL qualifier for a quick look at what attributes a variable has. (See the section on access to dataset and variable attributes, p. 65).

Ref Sec33.3. SHOW AXIS

Shows a basic description of the named axis.

```
SHOW AXIS[/qualifiers]   axis_name
```

A typical output appears below. The columns are:

name	name of axis (used also in DEFINE AXIS and DEFINE GRID)
# pts	number of points on axis; "r" or "i" for regular or irregular spacing, "m" if the axis is "modulo" (repeating)

axis	the orientation of the axis; "(-)" after the "r" or "i" on a depth axis indicates increasing downward
start	position of first point on the axis
end	position of last point on the axis

The axis span (length of the axis), and for modulo axes, the modulo length are also given.

```
yes? SHOW AXIS PSXT
name       axis              # pts   start                end
PSXT      LONGITUDE          160 r   130.5E               70.5W
  Axis span (to cell edges) = 360 (modulo length = axis span)

yes? SHOW AXIS/I=1:2 COADSX
name       axis              # pts   start                end
COADSX    LONGITUDE          180mr   21E                  19E(379)
  Axis span (to cell edges) = 360 (modulo length = axis span)
       I     X                   XBOX      XBOXLO
       1>  21E                   2         20E
       2>  23E                   2         22E
```

Command qualifiers for SHOW AXIS:

SHOW AXIS/I=/J=/K=/L=/X=/Y=/Z=/T=/XML
Displays the coordinates and grid box sizes for the specified axis. Optionally, low and high limits and a delta value may be specified to restrict the range of values displayed.

```
yes? SHOW AXIS/X[=lo:hi:delta] axis-name
```

Example:

```
yes? SHOW AXIS/L=1:12:3  my_custom_time_axis
```

SHOW AXIS/ALL
Show a brief summary of all axes defined.

```
yes? SHOW AXIS/ALL
```

SHOW AXIS/XML
List the axis information in XML-style

```
yes? SH AXIS/XML fnocx
<axis name="FNOCX">
<units>degrees_east</units>
<length>144</length>
<start>20E</start>
<end>17.5E(377.5)</end>
<point_spacing>even</point_spacing>
<modulo>yes</modulo
</axis>
```

Ref Sec33.4. SHOW COMMANDS

/ALL

Displays commands, subcommands, and qualifiers recognized by program Ferret. This command does not display aliases; use SHOW ALIAS.

```
SHOW COMMAND    [command_name or partial_command]
```

Note: This is the most reliable way to view command qualifiers. The output of this command will be current even when this manual is out of date.

Examples:

```
yes? SHOW COMMAND S       ! show all commands beginning with "S"
yes? SHOW COMMAND         ! show all commands
yes? SHOW COMMAND PLOT    ! shows command PLOT and all its qualifiers
```

Ref Sec33.5. SHOW DATA_SET

/ALL /BRIEF /FILES /FULL /VARIABLE /XML /ATTRIBUTES

Shows information about the data sets which have been SET and indicates the current default data set. By default the variables and their subscript ranges are also listed.

```
yes? SHOW DATA[/qualifiers]   [set_name_or_number1,set2,...]
```

If no data set name or number is specified then all SET data sets are shown.

Command qualifiers for SHOW DATA_SET:

SHOW DATA/ALL
This qualifier has no effect on this command; it exists for compatibility reasons.

SHOW DATA/ATTRIBUTES
Makes an expanded listing which includes the global and variable attributes, the attribute types, sizes, and the output flag for each (See the section on access to dataset and variable attributes, p. 65).

SHOW DATA/BRIEF
Shows only the names of the data sets; does not describe the data contained in them.

SHOW DATA/FILES
Displays the names of the data files for this data set and the ranges of time steps contained in each. Output is formatted as date strings or as time step values depending on the state of MODE CALENDAR.

SHOW DATA/FULL
Equivalent to /VARIABLES and /FILES used together.

SHOW DATA/VARIABLES
In addition to the information given by the SHOW DATA command with no qualifiers, this query also provides the grid name and world coordinate limits for each variable in the data set.

SHOW DATA/XML
For netCDF files, including those accessed via DODS, SHOW DATA/XML and SHOW DATA/VAR/XML list information about the file and variables as xml-style output.

Example: SHOW DATA

SHOW DATA produces a listing similar to the one below. The output begins with the descriptor file name (for TMAP-formatted data) and data set title. The columns I, J, K, and L give the subscript limits for each variable with respect to its defining grid (use SHOW DATA/FULL and SHOW GRID variable_name for more information).

```
yes? SET DATA levitus_climatology
yes? SHOW DATA
     currently SET data sets:
    1> /home/e1/tmap/fer_dsets/descr/levitus_climatology.des  (default)
      name     title                I         J         K         L
      TEMP     TEMPERATURE        1:360     1:180     1:20      1:1
      SALT     SALINITY           1:360     1:180     1:20      1:1
```

Example: SHOW DATA/XML, SHOW DATA/VARIABLES/XML

Example:

```
yes? USE monthly_navy_winds

yes? SHOW DATA/XML
<dataset>
<title> </title>
<vname>UWND</vname>
<vname>VWND</vname>
</dataset>

yes? SHOW DATA/VAR/XML
<var name="UWND">
<units>M/S</units>
<long_name>ZONAL WIND</long_name>
<_FillValue>-99.9</_FillValue>
<missing_value>-99.9</missing_value>
<grid name="GDN1">
<xaxis>FNOCX</xaxis>
<yaxis>FNOCY</yaxis>
<taxis>TIME</taxis>
</grid>
</var>
<var name="VWND">
<units>M/S</units>
<long_name>MERIDIONAL WIND</long_name>
```

```
<_FillValue>-99.9</_FillValue>
<missing_value>-99.9</missing_value>
<grid name="GDN1">
<xaxis>FNOCX</xaxis>
<yaxis>FNOCY</yaxis>
<taxis>TIME</taxis>
</grid>
</var>
```

Ref Sec33.6. SHOW EXPRESSION

Shows the current expression(s) implied or set with SET EXPRESSION. If not explicitly set with this command, the default current context expression is the argument of the most recent "action" command (PLOT, SHADE, CONTOUR, VECTOR, WIRE, etc.) See the chapter "Variables and Expressions", section "Expressions" (p. 73) for an explanation and list of action commands.

```
yes? SHOW EXPRESSION
```

Ref Sec33.7. SHOW FUNCTION

/ALL /BRIEF /EXTERNAL /INTERNAL /DETAILS

Shows a complete list of the functions defined in Ferret including descriptions of the function arguments.

```
yes? SHOW FUNCTION[/qualifiers] [function_name]
```

If no qualifier or function name is given then all functions are listed. SHOW FUNCTION will accept name templates such as

```
yes? SHOW FUNCTION *day*
     DAYS1900 (day, month, year)
     days elapsed since Jan. 1, 1900
```

Parameters

The parameter(s) may be the name of a function, with * replacing part of the string as above.

Command qualifiers for SHOW FUNCTION:

SHOW FUNCTION/ALL
List all functions defined

SHOW FUNCTION/BRIEF
List the functions and their arguments in brief form; no function or argument descriptions.

SHOSHOW FUNCTION/EXTERNAL
List only the available Ferret external functions (p. 267).

SHOW FUNCTION/INTERNAL
List only the internally defined Ferret functions.

SHOW FUNCTION/DETAILS
Lists the axis inheritance for grid-changing functions

Example:.SHOW FUNCTION/DETAILS

```
yes? SHOW FUNCTION/DETAILS SAMPLEXY
SAMPLEXY(DAT_TO_SAMPLE,XPTS,YPTS)
    Returns data sampled at a set of (X,Y) points, using linear
interpolation
        Grid of result:
          X: ABSTRACT (result will occupy indices 1...N)
          Y: NORMAL (no axis)
          Z: inherited from argument(s)
          T: inherited from argument(s)
    DAT_TO_SAMPLE: variable (x,y,z,t) to sample
        Influence on output grid:
          X: no influence (indicate argument limits with "[]")
          Y: no influence (indicate argument limits with "[]")
          Z: passed to result grid
          T: passed to result grid
    XPTS: X indices of sample points
        Influence on output grid:
          X: no influence (indicate argument limits with "[]")
          Y: no influence (indicate argument limits with "[]")
          Z: no influence (indicate argument limits with "[]")
          T: no influence (indicate argument limits with "[]")
    YPTS: Y indices of sample points
        Influence on output grid:
          X: no influence (indicate argument limits with "[]")
          Y: no influence (indicate argument limits with "[]")
          Z: no influence (indicate argument limits with "[]")
          T: no influence (indicate argument limits with "[]")
```

Ref Sec33.8. SHOW GRID

/I/J/K/L /X/Y/Z/T /ALL /DYNAMIC

Shows the name and axis limits of a grid.

```
yes? SHOW GRID[/qualifiers]   [var_or_grid1 var_or_grid2 ...]
```

Example:

(See command SHOW AXIS, p. 432, for an explanation of the columns.)

```
yes? SET DATA levitus_climatology
yes? SHOW GRID salt
```

```
   GRID GLEVITR1
name       axis          # pts   start               end
XAXLEVITR LONGITUDE      360mr   20.5E               19.5E(379.5)
YAXLEVITR LATITUDE       180 r   89.5S               89.5N
ZAXLEVITR DEPTH(-)        20 i   0m                  5000m
```

Parameters

The parameter(s) may be the name of one or more grid(s) or variable(s). If no parameter is given SHOW GRID displays the grid of the last variable accessed. This is the only mechanism to display the grid of an algebraic expression.

Note: To apply SHOW GRID to an algebraic expression it is necessary for Ferret to have evaluated the expression in a previous command. The command LOAD is useful for this purpose in some circumstances.

Command qualifiers for SHOW GRID:

SHOW GRID/I=/J=/K=/L=/X=/Y=/Z=/T=
Displays the coordinates and grid box sizes for the specified axis. Optionally, low and high limits and a delta value may be specified to restrict the range of values displayed. The argument may be an expression.

```
yes? SHOW GRID/X[=lo:hi:delta] [variable_or_grid]
```

Example:

```
yes? SHOW GRID/L=1:12:3  sst[coads_climatology]
```

SHOW GRID/ALL
Shows the names only of all grids defined.

```
yes? SHOW GRID/ALL
```

SHOW GRID/DYNAMIC
Shows the names of dynamic grids that are defined.

```
yes? SHOW GRID/DYNAMIC
```

SHOW GRID/XML gridname
Shows the grid information in XML-style format.

yes? USE monthly_navy_winds
yes? SHOW GRID/XML gdn1
<grid name="GDN1">
<xaxis>FNOCX</xaxis>
<yaxis>FNOCY</yaxis>

<taxis>TIME</taxis>
</grid>

Ref Sec33.9. SHOW LIST

Shows the current states of the LIST command.

```
yes? SHOW LIST
```

The qualifier /ALL may be used with this command but exists merely for compatibility reasons and has no effect.

Ref Sec33.10. SHOW MEMORY

/ALL/FREE/PERMANENT/TEMPORARY

Shows the state of the memory cache.

```
yes? SHOW MEMORY
```

Shows the current size of the cache.

```
yes? SHOW MEMORY[/qualifiers]
```

Command qualifiers for SHOW MEMORY:

SHOW MEMORY/ALL
Shows all variables currently cached in memory—permanent and temporary.

SHOW MEMORY/FREE
Shows cache memory and memory table space that remains unused.

Cache memory is organized into "blocks." One block is the smallest unit that any variable stored in memory may allocate. The total number of variables that may be stored in memory cannot exceed the size of the memory table. The "largest free region" gives an indication of memory fragmentation. A typical SHOW MEMORY/FREE output looks as below:

```
total memory table slots: 150
total memory blocks: 500
memory block size:1600

number of free memory blocks: 439
largest free region: 439
number of free regions:   1
free memory table slots: 149
```

SHOW MEMORY/PERMANENT
Lists the variables cached in memory and cataloged as permanent. These variables will not be deleted even when memory space is needed. They become cataloged in memory as permanent when the LOAD/PERMANENT command is used.

SHOW MEMORY/TEMPORARY
Lists the variables cached in memory and cataloged as temporary (they may be deleted when memory capacity is needed).

Ref Sec33.11. SHOW MODE

Shows the names, states and arguments of the Ferret SET MODE command.

```
SHOW MODE [partial_mode_name1,name2,...]
```

Example:

```
yes? SHOW MODE VERIFY,META
```

The qualifier /ALL may be used with this command but exists merely for compatibility reasons and has no effect.

Ref Sec33.12. SHOW MOVIE

Shows the current state of SET MOVIE. This state affects FRAME and graphics commands specified with the /FRAME qualifier.

```
yes? SHOW MOVIE
```

The qualifier /ALL can be used with this command, but it exists for compatibility purposes only and has no effect.

Ref Sec33.13. SHOW QUERIES

Queries are a vehicle for communication between Ferret and a stand-alone interface program. They are not supported for general use.

Ref Sec33.14. SHOW REGION

Shows the current default region or the named region.

```
yes? SHOW REGION[/ALL]  [region_name]
```

The region displayed is formatted appropriately for the axes of the last data accessed. For example, suppose the region along the Y axis was specified as Y=5S:5N. Then if the Y axis of the last data accessed is in units of degrees-latitude the Y location is shown as Y=5S:5N but if the Y axis of the last data accessed is "ABSTRACT" then the Y location is shown as Y=-5:5.

Ref Sec33.15. SHOW SYMBOL

/ALL

Shows the value of one or more symbols (string variables).

```
yes? SHOW SYMBOL[/qualifier]  [symbol_name]
```

If no qualifier or symbol name is given then all defined symbols are listed. SHOW SYMBOL will accept partial names such as

```
yes? SHOW SYMBOL *lab*
   MY_X_LABEL = "Sample Number"
   LABEL_2 = "Station at 23N"
```

Parameters

The parameter may be the name of a symbol, with * replacing part of the string as above.

Command qualifiers for SHOW SYMBOL:

SHOW SYMBOL/ALL
Lists all symbols that are defined.

Ref Sec33.16. SHOW TRANSFORM

Shows the available transformations, including regridding transformations.

```
yes? SHOW TRANSFORM
```

Note: This is the most reliable way to view transformations. The output of this command will be current even when this manual is out of date.

The qualifier /ALL may be used with this command but exists merely for compatibility reasons and has no effect.

Ref Sec33.17. SHOW VARIABLES

/ALL /DATASET /DIAGNOSTIC /USER

Lists diagnostic or user-defined variables.

```
SHOW VARIABLES[/qualifier]   [partial_name]
```

Examples:

```
yes? SHOW VARIABLES       !all user-defined variables
yes? SHOW VAR/DIAG Q      !all diagnostic vars beginning with Q
```

Command qualifiers for SHOW VARIABLES:

SHOW VARIABLES/ALL
Lists both diagnostic variables (available for the COX/PHILANDER model) and user-defined variables.

SHOW VARIABLES/DATA_SET
Lists variables associated with the named dataset by DEFINE VARIABLE/DATA_SET=

SHOW VARIABLES/DIAGNOSTIC
This is an unsupported (obsolete) qualifier. It lists "diagnostic" variables available for the COX/PHILANDER model.

SHOW VARIABLES/USER
Lists expressions that have been defined by the user as new variables. This is the default behavior of SHOW VARIABLES with no qualifier.

Ref Sec33.18. SHOW VIEWPORT

Shows one or more of the currently defined viewports. Omitting an argument gives information on all viewports.

```
yes? SHOW VIEWPORT [view_name1,view_name2,...]
```

Example:

```
yes? DEFINE VIEWPORT/AXES/XLIM=0:0.5/YLIM=0.3:0.8 leftmid
yes? SHOW VIEWPORT left*

name             text    xlimits     ylimits    mode
 LEFT             1.00   0.00,0.50   0.00,1.00   edges
 LEFTMID          0.50   0.00,0.50   0.30,0.80   axes
   current viewport is NONE
```

This command shows the pre-defined viewport LEFT, and our user-defined viewport named LEFTMID. Under Column 1, text, is the setting for scaling the size of text. The xlimits and ylimits columns are the edges of the viewport. Mode takes the value edges for the default setting where the viewport limits include space for margins around the plot axes, or axes if the viewport was defined with the /axes qualifier, indicating that the limits define the location of the plot axes.

The qualifier /ALL may be used with this command but exists merely for compatibility reasons and has no effect.

Ref Sec33.19. SHOW WINDOWS

Lists open window numbers and indicates which is the active one.

```
yes? SHOW WINDOWS
```

The qualifier /ALL may be used with this command but exists merely for compatibility reasons and has no effect.

Ref Sec34. SPAWN

Executes a command line (Unix shell) command from within Ferret.

```
yes? SPAWN unix_shell_command
```

Example:

```
yes? SPAWN rm -f file.dat
```

Also, "SPAWN shell_name" allows the user to fork into an interactive shell. For example:

```
yes? SPAWN csh
```

enters the user into a c-shell. Use EXIT to return to Ferret.

Ref Sec35. STATISTICS

/I/J/K/L X/Y/Z/T /D /BRIEF

Computes summary statistics about the data specified.

```
yes? STATISTICS[/qualifiers]    expression_1 , expression_2 , ...
```

The statistics include:

- the size and shape of the region
- total number of data values in the region specified
- number of data values flagged as bad data
- minimum value
- maximum value
- mean value (arithmetic mean—not weighted by grid spacing)
- standard deviation (also not weighted by grid spacing)

All values are reported to 5 significant digits.

STATISTICS interacts with the current context exactly as the commands CONTOUR, PLOT and LIST do.

Parameters

Expressions may be anything described under Expressions. If multiple variables or expressions are specified they are treated in sequence. The expression(s) are inferred from the current context if omitted from the command line.

Command qualifiers for STATISTICS:

STATISTICS/I=/J=/K=/L=/X=/Y=/Z=/T=
Specifies value or range of axis subscripts (I, J, K, or L), or axis coordinates (X, Y, Z, or T) to be used when computing statistics about the expression(s).

STATISTICS/D=
Specifies the default data set to be used when computing statistics about the expression(s).

STATISTICS/BRIEF
Produces a shorter listing involving less computation.

Ref Sec36. UNALIAS

Alias for CANCEL ALIAS (p. 323).

Ref Sec37. USE

The USE command is an alias for SET DATA/FORMAT=cdf. (p. 399)

All qualifiers and restrictions are identical to SET DATA/FORMAT=cdf. If no filename extension is given, ".cdf" is assumed.

Example:

```
yes? USE test
```
is equivalent to
```
yes? SET DATA/FORMAT=cdf test
```

Ref Sec38. USER

Executes a user-written extension to the Ferret program.

```
yes? USER[/COMMAND=]    expression_1 , expression_2, ...
```

The USER command is a means of incorporating custom changes in Ferret. It is currently supported only by special request to the Ferret developers. Two special features are currently accessible through the USER command—objective analysis and scattered sampling of grids. These commands are superceded with Version 5.0 by the functionality available through external functions.

We recommend the user access objective analysis via the script objective.jnl. The scattered sampling feature is used in the polar plotting GO tools (try "GO polar_demo" at the Ferret prompt).

Ref Sec38.1. Objective analysis

(Note: see the version 4.4 documentation for an older way of gridding (X,Y, value) triples onto a grid)

To grid a set of (X, Y, value) triples onto a grid of specified resolution, sometimes called Objective analysis, use one of the family of "scat2grid" external functions. See the description of these functions starting at p. 95.

```
yes? SHOW FUNCTION/EXTERNAL scat*
```

The X, Y, and F(X,Y) are lists of locations and a value associated with each location. Define X and Y axes of the desired the grid and call the function to interpolate these points to the grid. Say you have a set of latitudes, longitudes, and samples of a quantity N03 at those points, and that these are in the variables my_lat, my_lon, and n03.

```
yes? DEFINE AXIS/X=170W:120W:5 xax5
yes? DEFINE AXIS/Y=0:40N:5 yax5
yes? LET n03_reg = scat2gridgauss_xy(my_lat, my_lon, n03, xax5, yax5,
2.,2.,2.,2.)
yes? SHADE n03_reg
```

See also the example in the demo script,

```
yes? go objective_analysis_demo
```

Ref Sec38.2. Scattered sampling

Note: there was an older way of doing scattered sampling; see section 33.2 in the version 4.4 documentation)

Ferret functions are available for sampling a gridded data field. See

```
yes? SHOW FUNCTION sample*

SAMPLEI(DAT_TO_SAMPLE,I_INDICES)
SAMPLEJ(DAT_TO_SAMPLE,J_INDICES)  ! These sample a gridded field,
returning
SAMPLEK(DAT_TO_SAMPLE,K_INDICES)  ! data at a set of grid points along
an
SAMPLEL(DAT_TO_SAMPLE,L_INDICES)  ! axis

SAMPLEIJ(DAT_TO_SAMPLE,XPTS,YPTS) ! Returns data sampled at a
2-dimensional
                                  ! subset of its grid points

SAMPLET_DATE(DAT_TO_SAMPLE,YR,MO,DAY,HR,MIN,SEC)  ! Returns data sampled
by
                                  ! interpolating to one or more times

SAMPLEXY(DAT_TO_SAMPLE,XPTS,YPTS) ! Returns data sampled at a set of
(X,Y)
                                  ! points, i.e., a ship track or some
                                  ! other path, using linear
interpolation
```

Examples of the use of these functions are in ef_sort_demo.jnl

Ref Sec39. VECTOR

/I/J/K/L /X/Y/Z/T /D /ASPECT /FRAME /LENGTH /NOAXIS /NOLABELS /OVERLAY /PEN /SET_UP /TITLE /COLOR /TRANSPOSE /HLIMITS /VLIMITS /XSKIP /YSKIP

Produces a vector arrow plot.

```
VECTOR[/qualifiers] x_expr,y_expr
```

In a curvilinear coordinate system (map projections)

VECTOR[/qualifiers] x_expr, y_expr, xcoords, ycoords (see p. 222)

Parameters

x_expr, y_expr
Algebraic expressions (or simple variables) specifying the x components and y components of the vector arrows. The expression pair will be inferred from the current context if omitted from the command line.

Note 1: An alternative method is to plot a vector field with filled polygons drawn in the shape of the vectors. In addition to representing the vectors' length and direction, the vectors may optionally be colored according to another quantity, i.e. wind vectors colored according to the atmospheric pressure. The scripts poly_vectors.jnl and mp_poly_vectors.jnl, included with Ferret beginning with v5.53, set up to make such plots. poly_arrow_key.jnl draws a arrow key. Please run the script poly_vec_demo.jnl for a demonstration of this capability. See the on-line version of this demo. for example figures.

Note 2: A second alternative is the plot_vectors.jnl script, which draws the vector arrows from u, v, lon, and lat, where these are 1-dimensional or 2-dimensional variables representing the vector components and the curvilinear coordinates.

Command qualifiers for VECTOR:

VECTOR/I=/J=/K=/L=/X=/Y=/Z=/T=
Specifies value or range of axis subscripts (I, J, K, or L), or axis coordinates (X, Y, Z, or T) to be used when evaluating the expression being plotted.

VECTOR/D=
Specifies the default data set to be used when evaluating the expression pair being plotted.

VECTOR/ASPECT
Adjusts the direction of the vectors to compensate for differing axis scaling.

```
yes? VECTOR/ASPECT[=aspect_ratio]  x_expr, y_expr...
```

The size of vectors is unchanged—only the direction is modified. Under most circumstances /ASPECT should be specified. The aspect ratio is (Y-scale/X-scale). If the plot lies in the latitude/longitude plane the aspect ratio correction will be adjusted as a function of COS(LATITUDE) on the plot.

For example, in a typical oceanographic XZ plane plot the vertical (Z) axis is in tens of meters while the horizontal (X) axis is in hundreds of kilometers. This means the vertical scale is greatly magnified in comparison to the horizontal. The /ASPECT qualifier correspondingly magnifies the vertical component of the vector relative to the horizontal while preserving the length of the vector. The magnification factor is documented on the plot.

If no aspect ratio is specified by the user (e.g. VECTOR/ASP with no value), then Ferret will plot the vectors so that the two components' relative sizes shows their ratio. (In an XZ plane, then, ocean velocity vectors will nearly always appear horizontal) .

VECTOR/FLOWLINE[/DENSITY]

VECTOR/FLOWLINE (alias FLOWLINE) draws continuous flowlines from the vector components U and V. The qualifier /DENSITY controls the number of lines drawn. The values of /DENSITY are positive integers. /XSKIP and /YSKIP are not valid when the /FLOWLINE qualifier is used. There is also a 4-argument form of this command for drawing flowlines in curvilinear coordinates. Note that Ferret does not compute a stream function, but draws a pathline integration of a 2-dimensional instantaneous flow field. In a 3-dimensional flow field the plots are only useful as a qualitative visualization tool.

The size of the arrows indicates the speed of the flow. Lines are drawn until they intersect a border of the region or another line.

As with the standard VECTOR command, the /ASPECT qualifier adjusts the direction of the vectors to compensate for differing axis scaling. Under most circumstances /ASPECT should be specified.

The underlying algorithm is used with permission from the GrADS program. Our thanks to COLA, the Center for Ocean-Land-Atmosphere Studies, for access to this technique.

Example 1:

```
yes? USE coads_climatology
yes? SET REGION/x=150e:130w/y=40s:50n/l=6
yes? FLOW/ASPECT/DENSITY=4 uwnd,vwnd
```

Example 2:

```
yes? USE coads_climatology
yes? SET REGION/x=0:360/y=70s:70n/l=1
yes? go mp_lambert_cyl
yes? set grid uwnd
yes? go mp_aspect
yes? FLOW/ASPECT/NOAXIS/NOLAB uwnd, vwnd, x_page, y_page
yes? go mp_fland; go mp_graticule
```

VECTOR/FRAME
Causes the graphic image produced to be captured as an animation frame in the file specified by SET MOVIE. In general the FRAME command (p. 358) is more flexible and we recommend its use rather than this qualifier.

VECTOR/LENGTH=
Controls the size of vectors.

```
yes? VECTOR/LENGTH[=value_of_standard]
```

If the /LENGTH qualifier is omitted Ferret automatically selects reasonable vector lengths. To reuse the vector length from the last VECTOR plot use VECTOR/LENGTH.

To specify the vector lengths manually use the value_of_standard argument. This associates the value "val" with the standard vector length, normally ½ inch. Note that the PPLUS command VECSET can be used to modify the length of the standard vector. This is also the length that is displayed in the vector key.

Example:

```
yes? VECTOR/LENGTH=100 U,V
```

Creates a vector arrow plot of velocities with ½ inch vectors for speeds of 100.

VECTOR/NOAXIS
Suppresses all axis lines, tics, and labeling so that no box appears surrounding the contour plot. This is especially useful for map projection plots.

VECTOR/NOLABELS
Suppresses all plot labels.

VECTOR/NOKEY
Suppresses key at the bottom of the plot which shows the vector length. Use in conjunction with /NOLABELS to remove the name of the variables on overlay plots..

VECTOR/OVERLAY
Causes the indicated vector field to be overlaid on the existing plot.

VECTOR/COLOR=
Specifies the line color for the vectors. The available color names are Black, Red, Green, Blue, LightBlue, Purple, and White (not case sensitive), corresponding to the /PEN values 1-6, respectively. (/COLOR also accepts numerical values.). Note that White is only available for the default THICKNESS=1.

```
yes? VECTOR/PEN=green   x_expr, y_expr
```

VECTOR/PEN=
Specifies the line style for the vectors. /PEN= takes the same arguments as the /LINE= qualifier for command PLOT. See command PLOT/LINE= (p. 375). "n" ranges from 1 to 18.

```
yes? VECTOR/PEN=n   x_expr, y_expr
```

VECTOR/SET_UP
Performs all the internal preparations required by program Ferret for vector plots but does not actually render output. The command PPL can then be used to make changes to the plot prior to

producing output with the PPL VECTOR command. This permits plot customizations that are not possible with Ferret command qualifiers. See the chapter "Customizing Plots" (p. 183).

Note that when the /SETUP qualifier is used the /XSKIP and /YSKIP qualifiers are ignored. In this case, use arguments to the PPL VECTOR command to achieve the thinning.

```
PPL VECTOR/qualifiers,xskip,yskip

yes? PPL VECTOR/over/3,4   specifies XSKIP=3 and YSKIP=4
```

VECTOR/TITLE=
Allows user to specify a plot title (enclosed in quotation marks). Without this qualifier Ferret selects a title based on information about x_expr and y_expr.

```
yes? VECTOR/TITLE="title_string"  x_expr, y_expr
```

VECTOR/TRANSPOSE
Causes the horizontal and vertical axes to be interchanged. By default the X axis is always drawn horizontal and the Y and Z axes are drawn vertical. For Y-Z plots the Z data axis is vertical.

VECTOR/HLIMITS=
Specifies horizontal axis limits and tic interval. Without this qualifier, Ferret selects reasonable values.

```
yes? VECTOR/HLIMITS=lo:hi:increment   x_expr, y_expr
```

The optional "increment" parameter determines tic mark spacing on the axis. If the increment is negative, the axis will be reversed.

This qualifier does not have any impact on the context of the expression being plotted. If data is on a modulo x axis but the arguments of the /HLIMITS qualifier represent a region outside the actual coordinates of the data, only the range including the actual coordinates is shown. Use /X=lo:hi or SET REGION or a context on the variable itself, var[X=lo:hi], to set the context for the expression.

The /HLIMITS and /VLIMITS qualifiers will retain their "horizontal" and "vertical" interpretations in the presence of the /TRANSPOSE qualifier. Thus, the addition of /TRANSPOSE to a plotting command mandates the interchange of "H" and "V" on the limits qualifiers

VECTOR/VLIMITS=
Specifies the axis range and tic interval for the vertical axis. See /HLIMITS (above)

VECTOR/XLIMITS=/YLIMITS=
Note: XLIMITS and YLIMITS have been deprecated. Please use HLIMITS and VLIMITS instead.

VECTOR/XSKIP=/YSKIP=
Draws every nth vector along the requested axis beginning with the first vector requested.

```
yes? VECTOR/XSKIP=nx/YSKIP=ny  u,v
```

By default, Ferret "thins" vectors to achieve a clear plot. These qualifiers allow control over thinning.

Note that when the /SETUP qualifier is used the /XSKIP and /YSKIP qualifiers are ignored. In this case, use arguments to the PPL VECTOR command to achieve the thinning.

```
PPL VECTOR/qualifiers,xskip,yskip

yes? PPL VECTOR/over/3,4   specifies XSKIP=3 and YSKIP=4
```

VECTOR/AXES[=top,bottom,left,right]

Turns plotting of individual axes off and on. This replaces the use of the "PPL AXSET" command. The syntax is

```
yes? VECTOR/AXES[=top,bottom,left,right]  u,v
```

where the arguments are 1 to turn the axis on and 0 to turn it off. For example:

```
yes? VECTOR/AXES=0,1,1,0  u,v     ! Draws the bottom and left axes only
```

VECTOR/GRATICULE[=line specifiers]

(Introduced in Ferret version 5.6) Turns on graticule lines for the horizontal and vertical axes. These are lines across the plot at the tic marks. /GRATICULE sets both horizontal and vertical lines; to set each separately see /HGRATICULE and /VGRATICULE, below. The syntax is

```
yes? VECTOR/GRATICULE[=line or dash,COLOR=,THICKNESS=]  u,v
```

where the default is a thin, solid black line. The line colors available are Black, Red, Green, Blue, LightBlue, Purple, and White. The thickness codes are 1, 2, or 3 and as for plot lines, thickness=1 is a thin line, thickness=3 is the thickest, and THICK specified with no value defaults to thickness=2. For clarity the arguments to GRAT may be placed in parentheses

```
yes? VECTOR/GRAT u,v     ! default graticules

yes? VECTOR/GRAT=(LINE,COLOR=red,THIICK=3) u,v

yes? VECTOR/GRAT=(DASH,COLOR=lightblue) u,v

yes? VECTOR/FILL/GRAT=(DASH,COLOR=white) u,v
```

The above commands make settings for the large tic marks. If small tic marks are being plotted on the axes, we can make settings for them as well using keywords SMALL and LARGE. Place all of the arguments for the /GRAT qualifier in double quotes. Note that the PPL AXNMTC command sets the plotting of small tics, and that small tics are used by default for many time axes.

```
yes? ppl axnmtc 2,2
yes? VECTOR/GRAT="LARGE(COLOR=blue,thick),SMALL(COLOR=lightblue)" u,v
```

VECTOR/HGRATICULE[=line specifiers]/VGRATICULE[=line specifiers]

Turns on graticule lines and sets the line characteristics of the graticule for the horizontal or vertical axis separately. You may specify only one of /HGRAT or /VGRAT if desired. These are lines across the plot at the tic marks. The syntax is

```
     yes? VECTOR/HGRATICULE[=line or dash,COLOR=,THICKNESS=]
/VGRATICULE=line or dash,COLOR=,THICKNESS=]  u,v
```

where the default is a thin, solid black line. The line colors available are Black, Red, Green, Blue, LightBlue, Purple, and White. The thickness codes are 1, 2, or 3 and as for plot lines, thickness=1 is a thin line, thickness=3 is the thickest, and THICK specified with no value defaults to thickness=2. For clarity the arguments to HGRAT may be placed in parentheses

```
yes? VECTOR/HGRAT/VGRAT  u,v  !this is equivalent to VECTOR/GRAT

yes? VECTOR/HGRAT=(LINE,COLOR=red,THIICK=3)/VGRAT=(color=green) u,v

yes? VECTOR/HGRAT=(DASH,COLOR=lightblue) u,v  ! horizontal only
```

The above commands make settings for the large tic marks. If small tic marks are being plotted on the axes, we can make settings for them as well using keywords SMALL and LARGE. Place all of the arguments for the /HGRAT qualifier in double quotes. Note that the PPL AXNMTC command sets the plotting of small tics, and that small tics are used by default for many time axes.

```
yes? ppl axnmtc 2,2
yes? VECTOR/HGRAT="LARGE(COLOR=blue,thick),SMALL(COLOR=lightblue)"
/VGRAT="LARGE(COLOR=blue,thick) u,v
```

Ref Sec40. WHERE

The command (alias) WHERE requests mouse input from the user, using the indicated click position to define the symbols XMOUSE and YMOUSE in units of the plotted data. This command works only in Window 1. It does not function in other windows that have been opened with SET WINDOW/NEW. Comments that include the digitized position are also written to the current journal file (if open). The WHERE command can be embedded into scripts to allow interactive positioning of color keys, boxes, lines, and other annotations.

Ref Sec41. WIRE

/I/J/K/L /X/Y/Z/T /D /FRAME /NOLABEL /OVERLAY/SET_UP /TITLE /TRANSPOSE /VIEWPOINT /ZLIMITS /ZSCALE

Produces a wire frame representation of a two-dimensional field.

```
yes? WIRE[/qualifiers] expression
```

Parameters

The expression may be anything described in the chapter "Variables and Expressions", section "Expressions" (p. 73). The expression will be inferred from the current context if omitted from the command line. Multiple expressions are not permitted in a single WIRE command. The indicated region should denote a plane (2D) of data.

Command qualifiers for WIRE:

WIRE/I=/J=/K=/L=/X=/Y=/Z=/T=
Specifies value or range of axis subscripts (I, J, K, or L), or axis coordinates (X, Y, Z, or T) to be used when evaluating the expression being plotted.

Example:

The following commands will create a wire frame representation of a simple mathematical function in two dimensions.

```
yes? SET REGION/I=1:80/J=1:80
yes? WIRE/VIEWPOINT=-4,-10,2 exp(-1*(((I-40)/20)^2 + ((J-40)/20)^2) )
```

WIRE/D=
Specifies the default data set to be used when evaluating the expression being plotted.

WIRE/FRAME
Causes the graphic image produced to be captured as an animation frame in the file specified by SET MOVIE. In general the FRAME command (p. 358) is more flexible and we recommend its use rather than this qualifier.

WIRE/NOLABEL
Suppresses all plot labels.

WIRE/OVERLAY
Causes the indicated wire frame plot to be overlaid on the existing plot.

WIRE/SET_UP
Performs all the internal preparations required by program Ferret for wire frame graphics but

does not actually render output. The command PPL can then be used to make changes to the plot prior to producing output with the PPL WIRE command. This permits plot customizations that are not possible with Ferret command qualifiers. See the chapter "Customizing Plots" (p. 183).

WIRE/TITLE=
Allows user to specify a plot title (enclosed in quotation marks). Without this qualifier Ferret selects a title based on information about the expression.

WIRE/TRANSPOSE
Causes the X and Y axes to be interchanged.

WIRE/VIEWPOINT=
Specifies a viewpoint for viewing the wire frame.

```
yes? WIRE/VIEWPOINT=x,y,z   expression
```

The x,y values are specified as coordinates on the X and Y axes (though they may exceed the axis limits). The z value is in units of the requested variable.

WIRE/ZLIMITS=
Specifies limits of Z axis for wire frame.

```
yes? WIRE/ZLIMITS=zmin,zmax,delta   expression
```

The values given are in units of the requested variable. (The string given as an argument to /ZLIMITS= is passed unmodified to the PPLUS command WIRE as the zmin and zmax parameters.)

WIRE/ZSCALE=
Controls Z axis scaling of the 3-D plot.

```
yes? WIRE/ZSCALE=s  expression
```

The default value is equivalent to (ymax-ymin)/(zmax-zmin) (i.e., the aspect ratio of the Z axis to the Y axis). This qualifier is identical to the PPLUS VIEW command parameter of the same name.

GLOSSARY

ABSTRACT EXPRESSION (or VARIABLE)
An expression which contains no dependencies on any disk-resident data is referred to as "abstract". For example, SIN(x), where x is a pseudo-variable.

AXIS
A line along one of the dimensions of a grid. The line is divided into n points, or more precisely, n grid boxes where each grid box is a length along the axis. Adjacent grid boxes must touch (no gaps along the axis) but need not be uniform in size (points may be unequally spaced). Axes may be oriented (e.g. latitude, depth, ...) or simply abstract values.

COARDS
A profile for the standardization of netCDF files.

CONTEXT
The information needed to obtain values for a variable: the location in space and time (points or ranges), the name of the data set (if a file variable) and an optional grid.

DATA SET
A collection of variables in one or more disk files that may be specified with a single SET DATA command.

DESCRIPTOR
A file containing background data about a GT or TS-formatted data set: variable names, coordinates, units and pointers to the data files. Descriptor file names normally end with ".DES".

DYNAMIC AXIS
An axis that is inferred through the use of lo:hi:delta notation. It is created and destroyed dynamically by Ferret.

DYNAMIC GRID
A grid whose axes are inferred from a regridding operation that does not explicitly specify all of the destination axes or specifies a destination grid that can be rendered conformable with the originating grid only if some axes are removed or substituted.

EXPRESSION
Any valid combination of operators, functions, transformations, variables and pseudo-variables is an expression. For example, "ABS(U)", "TEMP/(-0.03^Z)" or "COS(TEMP[Y=0:40N@LOC:15])".

EZ DATA SET
Any disk data file that is readable by Ferret but is not in GT, TS, or netCDF format.

FILE VARIABLE
A variable made available with the SET DATA command. File variables are data in disk files suitable for plotting, listing, using in user-variable definitions, etc.

GKS
The "Graphical Kernel System" — a graphics programming interface that facilitates the development of device-independent graphics code.

GO FILE or GO SCRIPT
A file of Ferret commands intended to be executed as a single command with the GO command.

GRID
A group of 1 to 4 axes defining a coordinate space. A grid can associate the axes as "outer products" creating a rectangular array of points. Grids may be defined with the DEFINE GRID command or from inside data sets.

GRID BOX
A length along an axis assumed to belong to a single grid point. It is represented by a box "middle", a box upper and a box lower limit. The "middle" need not actually be at the center of the box but the upper limit of box m must always be the lower limit of box m+1. (This concept is needed for integration of variables along an axis.)

GRID FILE
A file containing the definition of grids and axes — part of the GT and TS formats.

GT FORMAT
"grids at time steps" format. A direct access format using a separate descriptor file for descriptive metadata.

METAFILE
A representation of graphics stored in a computer file. Such a file can be processed by an interpreter program (such as Fprint) and sent to a graphics output device.

MODULO AXIS
An axis where the first point of the axis logically follows the last. Examples of this are degrees of longitude or dates in a climatological year.

MODULO REGRIDDING
A regridding operation where the destination axis is modulo and the regridding transform is a modulo operation. Typical usage would be to create a 12-month climatology from a multi-year time series.

NETCDF
Network Common Data Format is an interface to a library of data access routines for storing and retrieving scientific data. NetCDF allows the creation of data sets which are self-de-

scribing and network transparent. As of Ferret version 2.30, NetCDF is the suggested method of data storage.

OPERATOR
A function that is syntactically expressed in-line instead of as a name followed by arguments. The Ferret operators are +, -, *, /, ^, AND, OR, EQ, NE, LT, LE, GT and GE.

PSEUDO-VARIABLE
A special variable whose values are coordinates or coordinate information about a grid. X, I, XBOX, XBOXLO and XBOXHI are the pseudo-variables for the X axis — similarly for the other axes.

QUALIFIER
Commands and variable names may require auxiliary information supplied by qualifiers. In the command "SHOW DATA/FULL," "/FULL" is a qualifier. In the variable "SST[Y=20N]," "Y=20N" is a qualifier.

REGION
The location in space and time (or other axis units) at which a variable is to be evaluated. The locations may be points or ranges. For example, T="1-JAN-1982",Y=12S:12N describes a region in latitude and time.

REGRID
The process of converting the values of a variable from one grid to another. By default this is done through multi-linear interpolation along all axes from the old grid to the new. Other methods are also supported.

SUBSCRIPT
A coordinate system for referring to grid locations in which the points along an axis are regarded as integers from 1 to the number of points on the axis. The qualifiers I, J, K, and L are provided to specify locations by subscript.

TRANSFORMATION
An operation performed on a variable along a particular axis and specified via the syntax "@trn". Some transformations, such as averaging (e.g. U[Z=@AVE]), reduce the range of the variable along the axis to a single point. Others, such as taking a derivative (e.g., V[T=@DDC]) do not.

TMAP-FORMAT
Special formats created by the Thermal Modeling and Analysis Project (TMAP). These formats use descriptor files to store information about the variables, units, titles, and grids for the data. Separate formats allow optimized access as time series (TS format) or as geographical regions (GT format). As of Ferret version 2.30, netCDF is the suggested method of data storage.

TS FORMAT
"time step" format. A direct access format using a separate descriptor file for descriptive metadata.

USER-DEFINED VARIABLE
A variable created with DEFINE VARIABLE (alias LET).

VARIABLE
Value defined on a grid.

VARIABLE NAME
The name by which a variable will be indicated in commands and expressions. Names begin with letters and may include letters, digits, dollar signs, and underscores.

VARIABLE TITLE
A title string used to label plots and listed outputs of a variable.

VIEWPORT
A graphical display region which may be any subrectangle of a window. Graphical commands (PLOT, CONTOUR, etc.) take complete control of a viewport, clearing it as needed. A window may contain several viewports — possibly overlapping. Viewports are defined with DEFINE VIEWPORT and controlled with SET and CANCEL VIEWPORT.

WINDOW
A rectangular graphical display region. On a graphics terminal the terminal screen is the one and only window available. On a graphics workstation there may be many output windows.

WORLD COORDINATE
A coordinate system for referring to grid locations in which the points along an axis are regarded as continuous values in some particular units (e.g., meters of depth, degrees of latitude). The qualifiers X, Y, Z, and T are provided to specify locations by world coordinate.

Appendix A: EXTERNAL FUNCTIONS

A number of external functions are included with the Ferret distribution. This number is expected to grow as Ferret developers and users contribute more functions. See the chapter "Writing External Functions" (p. 293) for how to adapt your Fortran code to a Ferret external function. Send your contributions to the Users Guide editor at ferret_ug@pmel.noaa.gov.

Many of these functions, originally developed as external functions, are now linked into the Ferret executable as static functions. The code for them is still available on the Ferret external functions web page.

The functions are listed in the following sections. To see what functions are available to you, type

```
yes? SHOW FUNCTION/EXTERNAL
```

or

```
yes? SHOW FUNCTIONS/DETAILS/EXTERNAL function_name
```

gives further details on how the arguments influence the grid for the function's result.

Appendix A Sec1. COMPRESSI

COMPRESSI(DAT) Returns data, compressed along the I axis: Missing points moved to the end

Arguments:	DAT	DAT: variable to compress in I
Result Axes:	X	ABSTRACT, same length as DAT x-axis
	Y	Inherited from DAT
	Z	Inherited from DAT
	T	Inherited from DAT

Note:
It is generally advisable to include explicit limits when working with functions that replace axes. for example, consider the function compressi(v). The expression

```
list/i=6:10 compressi(v)
```

is not equivalent to

```
list compressi(v[i=6:10])
```

The former will list the 6th through 10th compressed indices from the entire i range of variable v. the latter will list all of the indices that result from compressing v[i=6:10].

Appendix A Sec2. COMPRESSJ

COMPRESSJ(DAT) Returns data, compressed along the J axis: Missing points moved to the end

Arguments:	DAT	DAT: variable to compress in J
Result Axes:	X	Inherited from DAT
	Y	ABSTRACT, same length as DAT y-axis
	Z	Inherited from DAT
	T	Inherited from DAT

Note: see the note under COMPRESSI on specifying axis limits (p. 75)

Appendix A Sec3. COMPRESSK

COMPRESSK(DAT) Returns data, compressed along the I axis: Missing points moved to the end

Arguments:	DAT	DAT: variable to compress in K
Result Axes:	X	Inherited from DAT
	Y	Inherited from DAT
	Z	ABSTRACT, same length as DAT z-axis
	T	Inherited from DAT

Note: see the note under COMPRESSI on specifying axis limits (p. 75)

Appendix A Sec4. COMPRESSL

COMPRESSL(DAT) Returns data, compressed along the L axis: Missing points moved to the end

Arguments:	DAT	DAT: variable to compress in L
Result Axes:	X	Inherited from DAT
	Y	Inherited from DAT

	Z	Inherited from DAT
	T	ABSTRACT, same length as DAT t-axis

Note: see the note under COMPRESSI on specifying axis limits (p. 75)

Appendix A Sec5. COMPRESSI_BY

COMPRESSI_BY (var, mask), Compress data according to a mask

Arguments:	VAR	Variable to compress according to MASK
	MASK	mask to use in compressing the data
Result Axes:	X	Abstract
	Y	Inherited from VAR and MASK
	Z	Inherited from VAR and MASK
	T	Inherited from VAR and MASK

Compress variable "dat" along its I axis using the (multi-dimensional) mask supplied in the second argument.

For example:

```
yes? LET mask = {1,,1,,1} + 0*L[l=101:102] + 0*K[k=10:11]
yes? LIST mask
                {1,,1,,1} + 0*L[L=101:102] + 0*K[K=10:11]
                    1      2      3      4      5
                    1      2      3      4      5
      ---- L:101 T:   101
      10   / 10:  1.000   ....  1.000   ....  1.000
      11   / 11:  1.000   ....  1.000   ....  1.000
      ---- L:102 T:   102
      10   / 10:  1.000   ....  1.000   ....  1.000
      11   / 11:  1.000   ....  1.000   ....  1.000

yes?  LIST compressi_by({11,22,33,44,55},mask)
                  COMPRESSI_BY({11,22,33,44,55},MASK)
                    1      2      3      4      5
                    1      2      3      4      5
      ---- L:101 T:   101
      10   / 10:  11.00  33.00  55.00   ....   ....
      11   / 11:  11.00  33.00  55.00   ....   ....
      ---- L:102 T:   102
      10   / 10:  11.00  33.00  55.00   ....   ....
      11   / 11:  11.00  33.00  55.00   ....   ....
```

Appendix A Sec6. COMPRESSJ_BY

COMPRESSJ_BY (var, mask), Compress data according to a mask

Arguments:	VAR	Variable to compress according to MASK
	MASK	mask to use in compressing the data
Result Axes:	X	Inherited from VAR and MASK
	Y	Abstract
	Z	Inherited from VAR and MASK
	T	Inherited from VAR and MASK

Compress variable "dat" along its J axis using the (multi-dimensional) mask supplied in the second argument. See the example under COMPRESSI_by.

Appendix A Sec7. COMPRESSK_BY

COMPRESSK_BY (var, mask), Compress data according to a mask

Arguments:	VAR	Variable to compress according to MASK
	MASK	mask to use in compressing the data
Result Axes:	X	Inherited from VAR and MASK
	Y	Inherited from VAR and MASK
	Z	Abstract
	T	Inherited from VAR and MASK

Compress variable "dat" along its K axis using the (multi-dimensional) mask supplied in the second argument. See the example under COMPRESSI_by.

Appendix A Sec8. COMPRESSL_BY

COMPRESSL_BY (var, mask), Compress data according to a mask

Arguments:	VAR	Variable to compress according to MASK
	MASK	mask to use in compressing the data
Result Axes:	X	Inherited from VAR and MASK
	Y	Inherited from VAR and MASK
	Z	Inherited from VAR and MASK
	T	Abstract

Compress variable "dat" along its L axis using the (multi-dimensional) mask supplied in the second argument. See the example under COMPRESSI_by.

Appendix A Sec9. CONVOLVEI

CONVOLVEI (VAR, WEIGHT), CONVOLVEJ (VAR, WEIGHT) ,
CONVOLVEK (VAR, WEIGHT), CONVOLVE L (VAR, WEIGHT)
Convolve I (J,K,or L)component of variable with weight function

Arguments:	VAR	COM: variable to convolve
	WEIGHT	Weight function
Result Axes:	X	Inherited from VAR
	Y	Inherited from VAR
	Z	Inherited from VAR
	T	Inherited from VAR

This function (and likewise CONVOLVEJ, CONVOLVEK, and CONVOLVEL) convolves the variable VAR, with the weight function, wt along the X axis. Note that the variable's context may not be of adequate size for the full calculation. Missing data flags will be inserted where computation is impossible.

When bad data points are encountered in the component data all result data depending on it are flagged as bad, too.

The weight function is applied at each point from i-hlen to i+hlen, where hlen is half the length of the weight function. If the function is of even length, a zero weight is used at the upper end. Thus if the weights were {0.1, 0.4, 0.4, 0.1} the result at point I would be computed as the sum 0.1* COM(i-2) + 0.4* com(i-1) + 0.4* COM(i) + 0.1* COM(i+1) + 0.* COM(i+2)

Example:

Use the function to smooth a variable.

```
yes? LET weight = {0.25, 0.5, 0.25}
yes? LET c = SIN(x[x=0:10:.1]) + RANDU(X[X=0:10:.1])/5
yes? PLOT c
yes? PLOT/OVER/TITLE="convolvei" CONVOLVEI(c,weight)
```

Appendix A Sec10. CURV_TO_RECT_MAP

CURV_TO_RECT_MAP(lon_in, lat_in, grid_out, radius)

Computes mapping parameters for regridding from a curvilinear grid (see p. 252) to a rectilinear latitude-longitude grid. The mapping is applied to data with the function CURV_TO_RECT which interpolates the data to the rectilinear grid. This computation uses a spherical interpolation code written at GFDL; the Ferret developers are responsible for its implementation as an external function.

The output of this function is a set of mapping parameters; this mapping may be saved and applied to interpolate any field on the curvilinear grid onto the output rectangular grid.

Arguments:	lon_in	Source grid longitudes, in degrees (2-D field of longitudes describing the curvilinear grid)
	lat_in	Source grid latitudes, in degrees (2-D field of latitudes describing the curvilinear grid)
	grid_out	Any variable on the output recangular lon-lat grid. This grid may have irregularly spaced longitude and/or latitude axes
	radius	Source points falling within radius (in degrees) of destination point are included in the mapping to the destination point. Described further below.
Result Axes:	X	Inherited from grid_out
	Y	Inherited from grid_out
	Z	Abstract
	T	Abstract

This function does large amounts of calculation, and so runs slowly. It is recommended that you compute mappings from the curvilinear grid to desired rectilinear grid(s) and save them for use with your data fields. For an example of a call to this function see the documentation for the function CURV_TO_RECT, below (p. 465).

The following figure illustrates a destination grid location (*) with a radius of influence R. Valid source curvilinear grid locations (o) which fall within the radius of influence of the destination point are used in the mapping. Missing source points (x) do not contribute to the mapping. In this case, 13 valid source grid points fall within the radius of influence.

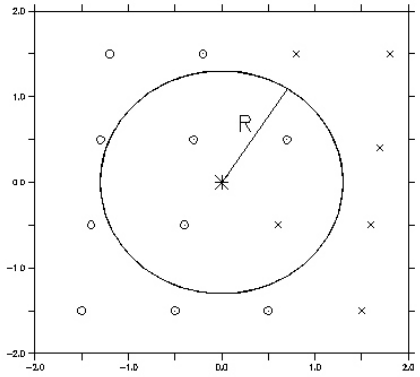


Figure A-1

The radius parameter should be chosen to be somewhat larger than the size of the input curvilinear grid cells so that any output grid location will have some input data contributing to its value.

e the value.The variable MAP, the result of a call to function CURV_TO_RECT_MAP, contains weights and the indices of the longitudes and latitudes of the curvilinear grid that correspond to the coordinates of the output grid. Below, in_curv_lon and in_curv_lat are the index value from the input curvilinear grid that correspond to the longitude and latitude of the rectangular output grid. The parameter num_neighbors is 4; the code looks around at four neighboring grid cells.

The weights are based on the distance from the source to the result grid points. The output variable map, then contains:

```
for each  m = 1, nlon_out
for each  n = 1, nlat_out

    for each k=1, num_neighbors

       MAP(m,n,k,1) = weight(m,n,k)
       MAP(m,n,k,2) = in_curv_lon(m,n,k)
       MAP(m,n,k,3) = in_curv_lat(m,n,k)
```

This may be used to determine the range of indices required in order to specify a subset of data on a curvilinear grid. See the FAQ, Plotting subsets of data on a curvilinear grid http://www.ferret.noaa.gov/Ferret/FAQ/custom_plots/subsetting_curvi_data.html, for an outline of the procedure.

Appendix A Sec11. CURV_TO_RECT

CURV_TO_RECT (V, mapping)

Applies the mapping computed by function CURV_TO_RECT_MAP to interpolate data from a curvilinear to a rectilinear latitude-longitude grid. This computation employs a spherical interpolation code written at GFDL; the Ferret developers are responsible for its implementation as an external function.

Arguments:	V	Variable on the source curvilinear grid
	mapping	mapping from the source grid to a rectilinear grid
Result Axes:	X	Inherited from mapping
	Y	Inherited from mapping
	Z	Inherited from V
	T	Inherited from V

This is a grid-changing function. Indicate limitson the argument axes with square brackets []. See the note on grid-changing functions in Chapter 3 (p. 77)

Example of calls to CURV_TO_RECT_MAP and CURV_TO_RECT:

```
yes? use my_curvilinear_data.nc
yes? show data
     currently SET data sets:
    1> /.my_curvilinear_data.nc  (default)
 name     title                          I      J      K    L
 GEOLON  geographic longitude of grid  1:180  1:173   ...   ...
 GEOLAT  geographic latitude of grid   1:180  1:173   ...   ...
 SALT    salinity                      1:180  1:173  1:30  1:12
 AIRT  air temperature                 1:180  1:173  1:30  1:12

yes? ! For convenience define variables with the input grid
yes? let lonin = geolon[d=1]
yes? let latin = geolat[d=1]

yes? ! Define output grid and a variable on the output grid
yes? define axis/x=0:360:5/modulo/units=degrees xax
yes? def axis/y=0:85:5/units=degrees yax
yes? let lonlatout = y[gy=yax] + x[gx=xax]

yes? ! Compute the mapping to the rectangular grid, save to a file
yes? let map = curv_to_rect_map ( lonin,latin,lonlatout,10)
yes? save/clobber/file=curv_map.nc  map

yes? ! Apply the mapping to the data fields
yes? cancel var/all
yes? use curv_map.nc
yes? let out_salt = curv_to_rect(salt[d=1,K=1,L=2], map[d=2])
yes? shade out_salt

yes? let out_airt = curv_to_rect(airt[d=1], map[d=2])
yes? save/file=air_on_rect.nc out_airt
```

Appendix A Sec12. RECT_TO_CURV

RECT_TO_CURV (V, lon_bounds_out, lat_bounds_out, missing_allowed) Uses Bilinear Interpolation to regrid data on a recilinear grid to a curvilinear grid.

Arguments:	V	Variable on the source rectilinear grid
	lon_bounds_out	Destination grid longitudes, in degrees (2-D field of longitudes describing the curvilinear grid)
	lat_bounds_out	Destination grid latitudes, in degrees (2-D field of latitudes describing the curvilinear grid)
	missing_allowed	Number of missing values allowed among the 4 surrounding source cells: 0 to 3
Result Axes:	X	Inherited from lon_bounds_out
	Y	Inherited from lat_bounds_out
	Z	Inherited from V

	T	Inherited from V

Example: put a rectilinear data set on a curvilinear grid for comparison.

```
yes? use my_curvilinear_data.nc
yes? use levitus_climatology

yes? show data
     currently SET data sets:
    1> /.my_curvilinear_data.nc
 name     title                          I      J      K    L
 GEOLON  geographic longitude of grid  1:180  1:173   ...   ...
 GEOLAT  geographic latitude of grid   1:180  1:173   ...   ...
 SALT    salinity                      1:180  1:173  1:30  1:12
 AIRT  air temperature                 1:180  1:173  1:30  1:12

    2> /home/ja9/tmap/fer_dsets/data/levitus_climatology.cdf  (default)
 name     title                          I      J      K    L
 TEMP     TEMPERATURE                   1:360  1:180  1:20   ...
 SALT     SALINITY                      1:360  1:180  1:20   ...

yes? ! For convenience, define variables for arguments to RECT_TO_CURV
yes? let lonout = geolon[d=1]
yes? let latout = geolat[d=1]

yes? let a = rect_to_curv(temp[d=2,k=1], lonout, latout, 2)
yes? shade a, lonout, latout

yes? ! Compare to a variable on the curvilinear grid
yes? shade a-airt[d=1], lonout, latout

yes? ! save the new variable for all depths, with the curvilinear
yes? ! latitude and longitude variables

yes? let temp_curv = rect_to_curv(temp[d=2], lonout, latout, 2)
yes? save/file=tcurv.nc temp_curv, lonout, latout
 LISTing to file tcurv.nc
```

Appendix A Sec13. DATE1900

DATE1900(formatted date) Scalar function: converts a formatted date into Julian days since 1-Jan-1900.

Argument:	formatted date	dd-mmm-yyyy or dd-mmm-yyyy, in quotes
Result Axes:	X	NORMAL (no axis)
	Y	NORMAL (no axis)
	Z	NORMAL (no axis)
	T	NORMAL (no axis)

Examples:

```
yes?  list date1900("23-feb-2003")
            VARIABLE : DATE1900("23-feb-2003")
          37673.
yes? list date1900("2-jan-1900")
            VARIABLE : DATE1900("2-jan-1900")
          1.000
```

Appendix A Sec14. DAYS1900TOYMDHMS

DAYS1900TOYMDHMS(day1900) Converts Julian days since 1-Jan-1900 to values year, month, day, hour, minute, second.

Argument:	day1900	Julian day counted from 1-jan-1900
Result Axes:	X	Inherited from argument
	Y	Inherited from argument
	Z	ABSTRACT (results occupy indices 1...6)
	T	Inherited from argument

This function applies only to time in the standard, Gregorian calendar.

Example:

Create a variable based on a time axis. List the result of DAYS1900TOYMDHMS.

```
yes? DEF AXIS/T=28-JAN-2003:3-FEB-2003:1/UNITS=days/T0=1-JAN-1900 timeax
yes? LET tday = T[gt=timeax]
yes? LIST days1900toymdhms(tday)
            VARIABLE : DAYS1900TOYMDHMS(TDAY)
            SUBSET   : 6 by 9 points (Z-TIME)
                         1      2      3      4      5      6
                         1      2      3      4      5      6
 28-JAN-2003 00 / 1:  2003.     1.    28.     0.     0.     0.
 29-JAN-2003 00 / 2:  2003.     1.    29.     0.     0.     0.
 30-JAN-2003 00 / 3:  2003.     1.    30.     0.     0.     0.
 31-JAN-2003 00 / 4:  2003.     1.    31.     0.     0.     0.
 01-FEB-2003 00 / 5:  2003.     2.     1.     0.     0.     0.
 02-FEB-2003 00 / 6:  2003.     2.     2.     0.     0.     0.
 03-FEB-2003 00 / 7:  2003.     2.     3.     0.     0.     0.
```

Appendix A Sec15. ELEMENT_INDEX

ELEMENT_INDEX(A, B) Returns index value in B for each point in A

Arguments:	A	data to mask
	B	list of values to match
Result Axes:	X	Inherited from argument A
	Y	Inherited from argument A
	Z	Inherited from argument A
	T	Inherited from argument A

Example

```
yes? LIST  ELEMENT_INDEX({6,5,4,3,2,1}, {1,2,3})
             VARIABLE : ELEMENT_INDEX ({6,5,4,3,2,1}, {1,2,3})
             SUBSET   : 6 points (X)
 1   / 1:   ....
 2   / 2:   ....
 3   / 3:   ....
 4   / 4:  3.000
 5   / 5:  2.000
 6   / 6:  1.000
```

Appendix A Sec16. ELEMENT_INDEX_STRING

ELEMENT_INDEX_STRING(A, B) Returns index value in B for each string in A. The string comparisons are case-sensitive.

Arguments:	A	string data to mask
	B	list of strings to match
Result Axes:	X	Inherited from argument A
	Y	Inherited from argument A
	Z	Inherited from argument A
	T	Inherited from argument A

Example

(See ELEMENT_INDEX) above for a simple example of the numeric case.)

Say we have a dataset that contains a string variable called institute for a collection of stations, and station locations xloc, yloc. Use ELEMENT_INDEX_STR to draw a polygon plot where the ! data stations belonging to each institute is shown by a dot with a unique color for each institution.

```
yes? USE obsdata.nc   ! contains a variable called institute

yes? LET inst_names = {"NOAA", "WHOI", "FSU", "BOM"}
yes? LET code = IS_ELEMENT_STR(institute, inst_names)

yes? LIST institute, code
       X: 0.5 to 53.5
    Column 1: INSTITUTE
    Column 2: CODE is ELEMENT_INDEX_STR(institute, {"NOAA", "FSU",
"BOM", "WHOI"})
    INSTITUTE CODE
     1 / 1: "NOAA" 1.000
     2 / 2: "WHOI" 4.000
     3 / 3: "FSU" 2.000
     4 / 4: "BOM" 3.000
     5 / 5: "NOAA" 1.000
     6 / 6: "BOM" 3.000
     ...
yes? GO basemap X=120:250 Y=25S:65N 20
yes? GO polymark POLY/OVER/PALETTE=rainbow_by_levels/LEV=(1,5,1) \ xloc,
yloc, code, square, 0.2
```

Appendix A Sec17. EOF_SPACE

EOF_SPACE(A, FRAC_TIMESER) Returns EOF (Empirical Orthogonal Function) spacial fields(eigenfunctions) from x-y-z-time field

Arguments:	A	Variable in any spatial dimensions, and time
	FRAC_TIMESER	Use only those time series with this fraction valid data, e.g. 0.8 to require that 80% of the data be present to use the data at a location.
Result Axes:	X	Inherited from A
	Y	Inherited from A
	Z	Inherited from A
	T	ABSTRACT 1 to NEOF

The EOF functions all make the same computations, returning different portions of the results. EOF_SPACE returns the eigenfunctions, normalized so that they have the units of data, while time amplitude functions (TAF's) are dimensionless. Thus the sum of the values of a given EOF = sqrt eigenvalue), and the mean of a given TAF = 1. EOF_STAT returns some useful statistics: the number of EOF's which were computed and normalized for the parameters given; the %variation explained for each eigenfunction, and the eigenvalues.

Specifying the context of the input variable explicitly e.g.

```
EOF_SPACE(A[x=20:40,y=2s:40n,l=1:58],FRAC_TIMESER)
```

will prevent any confusion about the region. See the note in chapter 3 (p.77)on the context of variables passed to functions.

The method is an implementation of Chelton's '82 method for finding EOFs of gappy time series. If there are no gaps, it reduces to ordinary EOFs.

The EOF analysis solves a matrix problem where the matrix is dimensioned (NX*NY*NZ) by NT, which can quickly become quite large. The EOF functions use other workspace as well which demands even more memory, and often memory must be increased with the SET MEMORY command. Regridding to a coarser grid or restricting the region may be necessary.

See the example under EOF_STAT for more on the input parameters, and see the demonstration ef_eof_demo.jnl for examples of this function.

Note: Earlier versions of the EOF functions had one more parameter. Check the version you have by saying

```
yes? SHOW FUNCTION eof*
```

Appendix A Sec18. EOF_STAT

EOF_STAT(A,FRAC_TIMESER) Used with EOF_SPACE and/or EOF_TFUNC. Return statistics related to an EOF solution for a given set of parameters. Results are on the x-axis j = 1: # EOFscomputed and scaled, j = 2: % percentage of total variance accounted for by each eigenvector, j = 3: the eigenvalues.

Arguments:	A	Variable in any spatial dimensions, and time
	FRAC_TIMESER	Use only those time series with this fraction valid data, e.g. 0.8 to require that 80% of the data be present to use the data at a location.
Result Axes:	X	ABSTRACT: 1 to NEOF
	Y	ABSTRACT: 1 through 3 as outlined in the description.
	Z	NORMAL (no axis)
	T	NORMAL (no axis)

Please see the discussion under EOF_SPACE, and see the demonstration ef_eof_demo.jnl for examples of this function.

Example results:

For a simple sample function, eof_stat called to decompose it into eigenfunctions. We allow data to be used if the time series at the point has at least 80% valid data.

Request the number of eigenvalues computed for this choice of parameters.

```
yes? list/i=1/j=1 eofstat
             VARIABLE : EOF_STAT(SST[X=67W:1W,Y=11S:11N], 0.8)0
             DATA SET : COADS Monthly Climatology (1946-1989)
             FILENAME : coads_climatology.des
             FILEPATH : /home/ja9/tmap/fer_dsets/descr/
             X        : 1
             Y        : 1
          284.0
```

Now get the percent variance explained by the eigenfunctions which were computed.

```
yes? list/i=1:10/j=2 eofstat
             VARIABLE : EOF_STAT(SST[X=67W:1W,Y=11S:11N], 0.8)
             DATA SET : COADS Monthly Climatology (1946-1989)
             FILENAME : coads_climatology.des
             FILEPATH : /home/ja9/tmap/fer_dsets/descr/
             SUBSET   : 10 points (X)
             Y        : 2
               2
               2
 1    /  1:  86.95
 2    /  2:   5.82
 3    /  3:   3.87
 4    /  4:   1.51
 5    /  5:   0.56
 6    /  6:   0.38
 7    /  7:   0.31
 8    /  8:   0.23
 9    /  9:   0.15
 10   / 10:   0.11
```

And finally the eigenvalues associated with these eigenfunctions.

```
yes? list/i=1:10/j=3 eofstat
             VARIABLE : EOF_STAT(SST[X=67W:1W,Y=11S:11N], 0.8)
             DATA SET : COADS Monthly Climatology (1946-1989)
             FILENAME : coads_climatology.des
             FILEPATH : /home/ja9/tmap/fer_dsets/descr/
             SUBSET   : 10 points (X)
             Y        : 3
               3
               3
 1    /  1:  249.4
 2    /  2:   16.7
 3    /  3:   11.1
 4    /  4:    4.3
 5    /  5:    1.6
 6    /  6:    1.1
 7    /  7:    0.9
 8    /  8:    0.7
 9    /  9:    0.4
 10   / 10:    0.3
```

Appendix A Sec19. EOF_TFUNC

EOF_TFUNC(A, FRAC_TIMESER) Compute EOF time amplitude functions from x-y-z-time field w/gaps.

Arguments:	A	Variable in any spatial dimensions, and time
	FRAC_TIMESER	Use only those time series with this fraction valid data, e.g. 0.8 to require that 80% of the data be present to use the data at a location.
Result Axes:	X	ABSTRACT: 1 to NEOF
	Y	NORMAL (no axis)
	Z	NORMAL (no axis)
	T	Inherited from A

Please see the discussion under EOF_SPACE, and see the demonstration ef_eof_demo.jnl for examples of this function.

The time amplitude functions (TAF's) are dimension less; and the mean of a given TAF = 1. They are returned as follows: For x=1, time amplitude function corresponding to the first eigenfunction is the time series with t=1:NT.

Appendix A Sec20. FINDHI

FINDHI(A,XRANGE,YRANGE) Find local maxima of a variable.

Arguments:	A	Variable in x and y, may be a function of z and/or t
	XRANGE	Range in data units of the X radius in which the function looks for maxima
	YRANGE	Range in data units of the Y radius in which the function looks for maxima
Result Axes:	X	ABSTRACT
	Y	ABSTRACT: j=1:3
	Z	Inherited from A
	T	Inherited from A

The maxima are listed along the X axis: j=1 contains the X locations of the points, j=2 contains the Y coordinates of the points, and j=3 contains the function values at the maxima.

This function looks for the maximumm gridded value in the neighborhood x+/- XRANGE, Y+/- YRANGE. It returns only values in the interior of the region, not on boundaries. It is an implementaion of the NCAR graphics routine "minmax"

The GO script label_hi_lo.jnl makes it easy to call this function and label and label low's and high's with either their numerical value or the letters L and H. See the demonstration script minmax_label_demo.jnl

Also see the script bullseye.jnl which locates and marks a "bullseye", i.e. a local minimum or maximum in a 2-D field within a user-specified region.

Appendix A Sec21. FINDLO

FINDLO(A,XRANGE,YRANGE) Find local minima of a variable.

Arguments:	A	Variable in x and y, may be a function of z and/or t
	XRANGE	Range in data units of the X radius in which the function looks for minima
	YRANGE	Range in data units of the Y radius in which the function looks for minima
Result Axes:	X	ABSTRACT
	Y	ABSTRACT: j=1:3
	Z	Inherited from A
	T	Inherited from A

The minima are listed along the X axis: j=1 contains the X locations of the points, j=2 contains the Y coordinates of the points, and j=3 contains the function values at the minima.

This function looks for the minimumm gridded value in the neighborhood x+/- XRANGE, Y+/- YRANGE. It returns only values in the interior of the region, not on boundaries. It is an implementaion of the NCAR graphics routine "minmax".

The GO script label_hi_lo.jnl makes it easy to call this function and label and label low's and high's with either their numerical value or the letters L and H. See the demonstration script minmax_label_demo.jnl

Also see the script bullseye.jnl which locates and marks a "bullseye", i.e. a local minimum or maximum in a 2-D field within a user-specified region.

Appendix A Sec22. FFT_IM

FFT_IM(A) computes the imaginary part of Fast Fourier Transform of time series in variable A

Arguments:	A	Variable with a regular time axis; may be a function of x, y, and/or z
Result Axes:	X	Inherited from A
	Y	Inherited from A

	Z	Inherited from A
	T	Generated by the function: frequency in cyc/(time units from A)

The units of the returned time axis are "cycles/ t" where t is the time increment.

Even and odd N's are allowed. N need not be a power of 2. FFT_RE and FFTP_IM assume f(1)=f(N+1), and the user gives the routines the first N pts.

Specifying the context of the input variable explicitly e.g.

```
LIST FFT_IM(A[l=1:58])
```

will prevent any confusion about the region. See the note in chapter 3 (p. 77)on the context of variables passed to functions.

The code is based on the FFT routines in Swarztrauber's FFTPACK available at www.netlib.org. See the section on FFTA for more discussion (p. 86). For further discussion of the FFTPACK code, please see the document, Notes on FFTPACK - A Package of Fast Fourier Transform Programs at http://ferret.pmel.noaa.gov/Ferret/Documentation/FFTpack_notes/FFTPACK_notes.html

Appendix A Sec23. FFT_RE

FFT_RE(A) computes the real part of Fast Fourier Transform of time series in variable A

Arguments:	A	Variable with a regular time axis; may be a function of x, y, and/or z
Result Axes:	X	Inherited from A
	Y	Inherited from A
	Z	Inherited from A
	T	Generated by the function: frequency in cyc/(time units from A)

The units of the returned time axis are "cycles/ t" where t is the time increment.

Even and odd N's are allowed. N need not be a power of 2. FFT_RE and FFT_IM assume f(1)=f(N+1), and the user gives the routines the first N pts.

Specifying the context of the input variable explicitly e.g.

```
LIST FFT_RE(A[l=1:58])
```

will prevent any confusion about the region. See the note in chapter 3 (p. 77)on the context of variables passed to functions.

The code is based on the FFT routines in Swarztrauber's FFTPACK available at www.netlib.org. See the section on FFTA for more discussion (p. 86). For further discussion of the FFTPACK code, please see the document, Notes on FFTPACK - A Package of Fast Fourier Transform Programs at http://ferret.pmel.noaa.gov/Ferret/Documentation/FFTpack_notes/FFTPACK_notes.html

Appendix A Sec24. FFT_INVERSE

FFT_INVERSE(AR, AI) computes the inverse Fast Fourier Transform of the two frequency series AR and AI

Arguments:	AR	Real part of an FFT transform. Variable with a frequency axis; may be a function of x, y, and/or z
	AI	Imaginary part of an FFT transform. Variable with a frequency axis; may be a function of x, y, and/or z
Result Axes:	X	Inherited from AR, AI
	Y	Inherited from AR, AI
	Z	Inherited from AR, AI
	T	Abstract axis: 2*length of input frequency axes of AR and AI

The returned time axis is abstract; the user will need to regrid it to the appropriate time axis.

The code is based on the FFT routines in Swarztrauber's FFTPACK available at www.netlib.org. See the section on FFTA for more discussion (p. 86). For further discussion of the FFTPACK code, please see the document, Notes on FFTPACK - A Package of Fast Fourier Transform Programs at http://ferret.pmel.noaa.gov/Ferret/Documentation/FFTpack_notes/FFTPACK_notes.html

Appendix A Sec25. IS_ELEMENT_OF

IS_ELEMENT_OF(A, B) returns 1 if any element of B matches any element of A, and 0 if there is no match

Arguments:	A	Numeric variable to search

	B	Numeric variableto match
Result Axes:	X	Abstract, 1 element long
	Y	Normal
	Z	Normal
	T	Normal

Example:

```
yes? list/nohead IS_ELEMENT_OF ({44,55,66}, {55}) 1.00
```

Appendix A Sec26. IS_ELEMENT_OF_STR

IS_ELEMENT_OF_STR(A, B) returns 1 if any element of string variable B matches any element of string variable A, and 0 if there is no match. The comparisons are case-sensitive.

Arguments:	A	String variable to search
	B	String variable to match
Result Axes:	X	Abstract, 1 element long
	Y	Normal
	Z	Normal
	T	Normal

Example:

```
yes? list/nohead IS_ELEMENT_OF_STR ({"HELLO", "hello", "Friend"},
{"Friend", "bye"})
1.00
yes? list/nohead IS_ELEMENT_OF_STR ({"HELLO", "hello", "Friend"},
{"friend", "heLLo"})
0.00
```

Appendix A Sec27. LANCZOS

LANCZOS(A, F1, F2, N) Returns the argument, bandpass-filtered in time using a Lanczos filter.

Arguments:	A	Variable with a regular time axis; may be a function of x, y, and/or z
	F1	Low frequency cutoff
	F2	High frequency cutoff
	N	Number of weights (must be odd)

Result Axes:	X	Inherited from A
	Y	Inherited from A
	Z	Inherited from A
	T	Inherited from A

For details see: Duchon, C. E., 1979: Lanczos filtering in one and two dimensions. J. of App. Met., 18, 1016-1022. This function is based on code written by Bill Gustafson at UC Davis.

Example: apply filter to series after removing its mean.

```
yes? use monthly_navy_winds
yes? let aa = uwnd[i=50,j=20]
yes? let az = aa - aa[t=@ave]
yes? plot az
yes? plot/over lanczos(az, 0.05, 0.1, 11)
yes? plot/over lanczos(az, 0.1, 0.2, 11)
```

Appendix A Sec28. LSL_LOWPASS

LSL_LOWPASS(A, cutoff_period, filter_span) Returns the argument filtered with Least Squares Lanzcos filter in time.

Arguments:	A	Variable with a regular time axis; may be a function of x, y, and/or z
	cutoff_period	Cutoff period (the period at which the filter attains 1/2 amplitude)
	filter_span	number of input data points used in each filtered output point.
Result Axes:	X	Inherited from A
	Y	Inherited from A
	Z	Inherited from A
	T	Inherited from A

This function low-pass filters an equally spaced time series using least-squares approximation to the ideal low-pass filter of Bloomfield with Lanczos convergence factors. It is very similar to subroutine LOPASS in Chapter 6, p. 149, of Bloomfield, P., 1976, Fourier Analysis of Time Series: An Introduction, John Wiley & Sons, New York, 258 pp.

The main difference is that the present routine takes account of missing values in the input time series Values near the ends and near gaps are filled with the missing value flag.

The cutoff period is the period at which the filter attains ½ amplitude or 1/4 "energy", measured in units of delta T. The cutoff_period must be less than or equal to N, the length of the time axis.

The filter span is the number of input data points used in each filtered output point. A wide filter gives a narrow frequency response transition band, but leads to ringing near data discontinuities and loss of filtered values at the end points and surrounding missing values. A narrow filter reduces ringing and output data loss, but gives a wider frequency transition width, i.e. it falls off less rapidly at frequencies higher than the cutoff. The filter transition region lies in the period range between N*cutoff_period/(N + cutoff_period) and N*cutoff_period/(N - cutoff_period). The filter span should be an odd integer. It is set to the next lower odd number if the input is even.

Note on tidal filtering: For hourly time series containing tidal signals, some investigators use this filter with a 35-hour cutoff period and a filter span of xxx hours to remove at least 99.5 % of the energy for periods less than 25 hours.

Adapted from Bloomfield by E. D. Cokelet, NOAA/PMEL, 3 Dec 1999

Appendix A Sec29. MINUTES24

MINUTES24(formatted time) Scalar function: converts a formatted time of day into minutes since 00:00, with fractions of minutes.

Argument:	formatted time	hours, minutes, seconds in the form hh:mm:ss, in quotes.
Result Axes:	X	NORMAL (no axis)
	Y	NORMAL (no axis)
	Z	NORMAL (no axis)
	T	NORMAL (no axis)

Examples:

```
yes? list minutes24("12:24:13")
            VARIABLE : MINUTES24("12:24:13")
           744.2
yes? list minutes24("00:30:30")
            VARIABLE : MINUTES24("00:30:30")
           30.50
```

Appendix A Sec30. WRITEV5D

WRITEV5D(V1,V2,V3,V4,V5,V6,V7,V8,FILENAME) Write up to 8 variables to a Vis5D-formatted file

Arguments:	V1	
	V2	
	V3	Up to 8 variables to write to the file
	V4	
	V5	
	V6	
	V7	
	V8	
	FILENAME	Name of the file to write: file type for Vis5d files is .v5d
Result Axes:	X	Inherited from variables: all variables must have the same x and y axes
	Y	Inherited from variables: all variables must have the same x and y axes
	Z	Inherited from variables; the result grid will contain the union of all the levels that are present in the variables.
	T	Inherited from variables: all variables must have the same time axis

This function calls utility functions from the Vis5D distribution to write a Vis5D-formatted file containing Ferret variables. TheVis5D tool is a system for interactive visualization of large 5-D gridded data sets. It was developed by Bill Hibbard and others at the University of Wisconsin, and can be found at

http://www.ssec.wisc.edu/~billh/vis5d.html

There are limits in Vis5D on the size of the grid and the number of timesteps. The function will issue an error if these limits are exceeded.

To make it more convenient to call the writev5d function, to open Vis5D from Ferret, and to append to a Vis5D file, GO tools are available: vis5d_write.jnl, vis5d_start.jnl, and vis5d_append.jnl. These have the filename first in their argument lists, and do not require the user to specify all 9 arguments to the function.

Example:

Write 3 variables to a file, then append timesteps to some of the variables. There is a gap between the times first written to the file and the times written when we call vis5d_append; this will show up in the Vis5d tool as a gap in time. Last, start Vis5d and open the file.

```
Yes? SET REGION/I=55:180/J=30:60
yes? GO vis5d_write myfile.v5d sst[L=20:30], airt[L=20:30], fcn_1
yes? GO vis5d_append myfile.v5d sst[l=34,50], airt[l=34,50]
yes? GO vis5d_start myfile.v5d
```

See the demonstration ef_wv5d_demo.jnl for examples of this function.

Appendix A Sec31. XCAT

XCAT (A, B)

Concatenates the values of two variables into one list on an abstract X axis.

Arguments:	A	variables to concatenate in X
	B	
Result Axes:	X	Abstract, with length the sum of the length of the X axes of A and B
	Y	Inherited from variables A and B
	Z	Inherited from variables A and B
	T	Inherited from variables A and B

Note:
This is a grid-changing function. It is generally advisable to include explicit limits when working with functions that replace axes. for example, consider the function xcat(a,b). Look at the expressions

```
list/i=10:20 xcat(a,b)
```

and

```
list xcat(a[i=16:20],b[i=1:5])
```

Both will list 10 values in the X direction. The former will list the 10th through 20th data values indices from the entire I range of both variables. The latter will list all of the data that results from concatenating b[i=1:5] onto a[i=16:20].

Appendix A Sec32. YCAT

YCAT (A, B)

Concatenates the values of two variables into one list on an abstract Y axis.

Arguments:	A	variables to concatenate in Y
	B	
Result Axes:	X	Inherited from variables A and B
	Y	Abstract, with length the sum of the length of the Y axes of A and B
	Z	Inherited from variables A and B
	T	Inherited from variables A and B

Note: This is a grid-changing function. Please see the discussion under the function XCAT (p. 481) about specifying regions.

Appendix A Sec33. ZCAT

ZCAT (A, B)

Concatenates the values of two variables into one list on an abstract Z axis.

Arguments:	A	variables to concatenate in Z
	B	
Result Axes:	X	Inherited from variables A and B
	Y	Inherited from variables A and B
	Z	Abstract, with length the sum of the length of the Z axes of A and B
	T	Inherited from variables A and B

Note: This is a grid-changing function. Please see the discussion under the function XCAT (p. 481) about specifying regions.

Appendix A Sec34. TCAT

TCAT (A, B)

Concatenates the values of two variables into one list on an abstract T axis.

Arguments:	A	variables to concatenate in T
	B	
Result Axes:	X	Inherited from variables A and B
	Y	Inherited from variables A and B
	Z	Inherited from variables A and B
	T	Abstract, with length the sum of the length of the T axes of A and B

Note: This is a grid-changing function. Please see the discussion under the function XCAT (p. 481) about specifying regions.

Appendix A Sec35. ZAXREPLACE_AVG

ZAXREPLACE_AVG(V,ZVALS,ZAX)

Use weighted averaging to convert between alternative monotonic Zaxes. The weighting is done according to the portion of the source box that lies within the destination grid cell.

The mapping between the source and destination Z axes is a function of X,Y, and or T. Typical applications in the field of oceanography include converting from a Z axis of layer number to a Z axis in units of depth (e.g., for sigma coordinate fields) and converting from a Z axes of depth to one of density (for a stably stratified fluid).

Argument 1, V, is the field of data values, say temperature on the "source" Z-axis, say, layer number. The second argument, ZVALS, contains values in units of the desired destination Z axis (ZAX) on the same Z axis as V — for example, depth values associated with each vertical layer. The third argument, ZAX, is any variable defined on the destination Z axis, often "Z[gz=zaxis_name]" is used. For an example of the ZAXREPLACE family of functions see ZAXREPLACE (p. 85)

Arguments:	V	A function of depth and perhaps, x, y, and time.
	ZVALS	Destination Z axis values as a fcn of source Z axis
	ZAX	Variable with desired z (depth) axis points
Result Axes:	X	Inherited from V
	Y	Inherited from V
	Z	Inherited from ZAX
	T	Inherited from V

Appendix A Sec36. ZAXREPLACE_BIN

ZAXREPLACE_BIN(V,ZVALS,ZAX)

Use unweighted averaging to convert between alternative monotonic Zaxes. The function finds the source points within each destination box and averages them.

The mapping between the source and destination Z axes is a function of X,Y, and or T. Typical applications in the field of oceanography include converting from a Z axis of layer number to a Z axis in units of depth (e.g., for sigma coordinate fields) and converting from a Z axes of depth to one of density (for a stably stratified fluid).

Argument 1, V, is the field of data values, say temperature on the "source" Z-axis, say, layer number. The second argument, ZVALS, contains values in units of the desired destination Z axis (ZAX) on the same Z axis as V — for example, depth values associated with each vertical layer. The third argument, ZAX, is any variable defined on the destination Z axis, often "Z[gz=zaxis_name]" is used. For an example of the ZAXREPLACE family of functions see ZAXREPLACE (p. 85)

Arguments:	V	A function of depth and perhaps, x, y, and time.
	ZVALS	Destination Z axis values as a fcn of source Z axis
	ZAX	Variable with desired z (depth) axis points
Result Axes:	X	Inherited from V
	Y	Inherited from V
	Z	Inherited from ZAX
	T	Inherited from V

Appendix B: PPLUS Users Guide

Note: This is the Users Guide for PPLUS, also called Plot Plus, a Scientific Graphics System written by Donald W. Denbo April 8, 1987. Its graphics calls are the basis for Ferret's graphics. In this appendix the PPLUS Users Guide is included unchanged, except for formatting changes and without its table of contents or index to avoid confusion. Note that some of the information is not relevant to the purpose of making PPLUS calls from Ferret. If there are differences, adhere to the information in the main Ferret Users Guide. See particularly the chapter "Customizing Plots" in the Ferret Users Guide (p. 183) for discussion of how Ferret interacts with PPLUS

Appendix B Sec1 INTRODUCTION

Plot Plus (PPLUS) is an interactive, command-driven general-purpose program for plotting user supplied data. PPLUS recognizes data in standard Fortran formatted, unformatted and free format files as well as some specialized formats (see the section on Data Formats). Data can also be entered from the keyboard.

The major use of PPLUS is the plotting of contour data and X-Y pairs. A very small number of commands are required to generate a plot, making use of the many defaults available. However, it is also possible to control almost every aspect of the plot and to generate a final product which looks as though it were professionally drafted. Over thirty character sets are available, including special Greek and Math symbols. It is possible to make a composite of several plots of different kinds (or the same kind) on a single page and to add text information anywhere on the plot.

PPLUS commands can be entered interactively from the keyboard or from a command file much like a VAX/VMS command file. PPLUS command files support parameter passing, symbol substitution, and logic structures such as WHILE loops and block IF statements. The PPLUS command files are simple ASCII disk files which are easily edited with any VAX/VMS editor.

Interactive help is available with the VAX/VMS command HELP PPLUS. (First, PPLUS definitions must have been established as indicated in the Getting Started chapter.)

Appendix B Sec2 GETTING STARTED

Appendix B Sec2.1 VAX/VMS

To get a copy of this manual, type the following lines on your terminal in response to the VAX/VMS prompt:

```
$ @DISK1:[OC.SYMBOLS]PLOT5
$ PPLUS_MANUAL
$ PPLUS_FONTS
```

The manual will be printed on the laser printer, and the PPLUS character fonts will be plotted on the Versatec plotter.

Appendix B Sec2.2 Required Definitions

PPLUS requires several assignments and definitions to execute under VMS. The following should be included in your LOGIN.COM file prior to running PPLUS:

```
$ @DISK1:[OC.SYMBOLS]PLOT5.COM
$ GRAPHTERM :== xxxx,
```

where xxxx describes your graphics terminal and has the following allowed values:

VT240
GVT+
ZENITH
TEK4010
MAC
TEK41XX
TEK4105
TAB

In order to provide automatic entry and exit into and out of graphics mode you should use the GRAPHTERM that corresponds to your terminal. If your terminal is a TEK4010 or TEK4014 compatible, but not one of the above, then place your terminal into graphics mode before plotting and use GRAPHTERM :== TEK4010. The execution of PLOT5.COM will define any other symbols needed by PPLUS.

PPLUS is entered interactively by typing PPLUS (or just PPL) in response to the VAX/VMS prompt.

Interactive help is available by typing HELP PPLUS in response to the VAX/VMS prompt. If you are in PPLUS, help is available by typing HLP.

Appendix B Sec2.2.1 Optional Definitions

In addition to the above, the following VAX/VMS symbols and logicals may optionally be defined by the user:

PPL$RESET	The "SAVE" file to be used by the PPLUS RESET command (logical). Default is PPL$EXE:PPL$RESET.DAT
ECHO	Defines the file to be used to echo PPLUS commands (logical). Default is ECHO.DAT.
PPL$STARTUP	Defines an initialization or startup command command file that will be executed each time PPLUS is entered (symbol). Default is no startup command file.

Example definitions:

```
DEFINE PPL$RESET DISK1:[your-directory]your-reset.file
DEFINE ECHO your-echo.file
PPL$STARTUP :== DISK1:[your-directory]your-startup.file
```

Appendix B Sec3 COMMAND FORMAT

Appendix B Sec3.1 THE COMMANDS

The basic format for PPLUS commands is:

COMM[/Q1/Q2 ...][,arg1,arg2,arg3...][,sarg1,sarg2...]

where COMM is the PPL command. The numeric arguments arg1,arg2,... may be numbers in any fortran format (e.g. 1.E-5, -6, 10.23) or blank. The character string arguments sarg1,sarg2,... must begin with a non-numeric character string or be enclosed in quotes ("), i.e., "100". If the numeric or character string arguments are blank, the input is considered null and the default is used. Where all numeric arguments are to be defaulted, they may be omitted entirely (i.e., blank entries need not be made).

PPLUS commands may have optional qualifiers (Q1, Q2 etc...). The format for qualifiers is "/value" or "/novalue" for true or false, respectively.

All parameters must be separated by commas or blanks, except null entries which must have separating commas. Null entries are allowed except where noted in the specific command description.

Note that if you use commas, a blank followed by a comma will be interpreted as a null entry. e.g.

```
PPL AXLEN  2 , 1          ! Is interpreted as   PPL AXLEN 2(, null)
PPL AXLEN  2, 1           ! Is interpreted as   PPL AXLEN 2,1
PPL AXLEN  2  1           ! Is interpreted as   PPL AXLEN 2,1
```

Commands can be continued on sequential lines by inserting a "-" (minus sign) at the end of the line to be continued.

All commands/parameters may be entered upper or lower case. Conversion to upper case is performed automatically when required.

Appendix B Sec4 COMMAND SYNOPSIS

This is intended as a brief overview of the PPLUS commands. Commands are fully described in the Command Description chapter. Examples illustrating their use are in the Beginners Guide section.

Appendix B Sec4.1 FILES

Appendix B Sec4.1.1 Data Files

These commands are used to extract the information from a file containing the data to be plotted.

RD	Reads/identifies file containing data to be plotted.
SKP	Skips/identifies records on the data file being read.
RWD	Rewinds/identifies the data file.
FORMAT	Describes the format of the data file.
VARS	Locates the data to be plotted in the records of the data file.
EVAR	Locates the data to be plotted in the records of the EPIC data file.
AUTOLAB	Controls automatic labeling of EPIC and BIBO data plots.

Appendix B Sec4.1.2 Other Data Entry

The following commands allow data entry from a souce other than a file.

ENTER Allows data entry from the keyboard.

LINFIT Does a linear least squares fit on data already in a line and inserts the least squares line into the next available line.

Appendix B Sec4.1.3 PPLUS Output Files

ECHO	Controls echoing of PPLUS commands to a PPLUS echo file.
DEBUG	Controls PPLUS debug mode (echos after symbol substitution)
PLTNME	Names the output plot file.
PLTYPE	Controls the format of the output plot file

Appendix B Sec4.1.4 PPLUS Command Files

@	Initiates reading of commands from a PPLUS command file.
ECHO	Controls echoing of PPLUS commands to a PPLUS echo file.
DEBUG	Controls PPLUS debug mode (echos after symbol substitution)

Appendix B Sec4.2 AXIS

The following commands control axis labelling and appearance.

Appendix B Sec4.2.1 X- And Y-axis

XAXIS Controls numeric labeling and tics on the x-axis.

YAXIS Controls numeric labeling and tics on the y-axis.

AXATIC Sets number of large tics automatically for x and y.

AXLABP Locates axis labels at top/bottom or left/right of plot.

AXLEN Sets axis lengths.

AXLINT Sets label interval for axes.

AXLSZE Sets axis label heights.

AXNMTC Sets number of small tics between large tics on axes.

AXNSIG Sets no. significant digits in numeric axis labels (auto only).

AXSET Allows omission of plotting of any axis.

AXTYPE Sets axis type for x- and y-axis.

TICS Sets axis tic characteristics

XFOR Sets format of x-axis numeric labels.

YFOR Sets format of y-axis numeric labels.

XLAB Sets label of x-axis.

YLAB Sets label of y-axis.

Appendix B Sec4.2.2 Time Axis

TIME Sets start and end of time axis, start time of data.

TAXIS Sets time axis on, sets time series delta-t (minutes),orients axis.

TXLABP Establishes time axis label position (or absence).

TXLINT Specifies which tics will be labeled.

TXLSZE Sets height of time axis labels.

TXNMTC Sets number of small tics between large tics.

TXTYPE Sets type and style of time axis.

Appendix B Sec4.3 LABELS

LABS Makes a moveable label (up to 25 labels allowed).

HLABS Sets height of each moveable label.

RLABS Sets angle for each moveable label.

LABSET Sets character heights for labels.

LLABS Sets start position for a line to location of each moveable label. Draws a line from the label to a point.

CONPRE Sets prefix for contour labels (characters, color, font).

CONPST Set suffix for contour labels (characters, color, font).

TITLE Sets and clears main plot label (without making a plot).

XLAB Sets label of x-axis.

YLAB Sets label of y-axis.

Appendix B Sec4.4 COMMAND PROCEDURES

@ Initiates reading of commands from a PPLUS command file.

DEC Decrements a counter.

INC Increments a counter.

IF Block IF statement.

ELSE Block IF statement.

ENDIF Block IF statement.

WHILE WHILE loop construct.

ENDW WHILE loop construct.

SET Sets the value of a PPLUS symbol.

SHOW Shows the value of a PPLUS symbol.

LISTSYM Lists values of defined PPLUS symbols.

Appendix B Sec4.5 COLOR AND FONTS

Commands to change the pen number or the character font can be embedded in any labels character string. See the preceding section for label commands and the chapter on LABELS.

@Pn Sets pen number "n" when embedded in a label, where n is less than 10

@Cnnn Sets color to number "nnn" when embedded in a label.

PEN Sets pen number for each data line.

DFLTFNT Sets default character font for all labeling.

LEV Sets pen numbers (colors) for contour plots.

Appendix B Sec4.6 PLOT APPEARANCE

The following commands control various aspects of the plot's appearance.

ORIGIN Sets distance of plot origin from lower left corner of the box.

SIZE Sets size of entire plotting region.

BOX Controls drawing of a box around the entire plotting region.

CROSS Controls drawing of lines through the point x=0, y=0 on graph.

LINE Sets characteristics for each X-Y plot line.

MARKH Sets character size for each X-Y plot line marks.

MULTPLT Allows a composite of several plots (all kinds) on onepage.

ROTATE Rotates plot by 90 degrees on screen and plotter.

Appendix B Sec4.7 PLOT GENERATION

The following commands select the plot type and generate the plot.

PLOT Plots x-y pairs for all lines of data.

PLOTUV Makes stick plot of vector data for U,V pairs in line1.

PLOTV Makes stick plot of vector data for U in line1 and V in line2.

CONTOUR Makes contour plot.

VIEW Makes a 3-D surface plot.

VPOINT Sets the viewpoint for a 3-D surface plot.

VECTOR Makes a plot of a vector field

VELVCT Makes vector plot of U,V pairs located at X,Y locations.

MULTPLT Allows a composite of several plots (all kinds) on one page.

Appendix B Sec4.8 DATA MANIPULATION

LINFIT Does a linear least squares fit on data already in a line and inserts the least squares line into the next available line.

TRANSXY Applies a linear transformation to variables x and y.

SMOOTH Controls smoothing of contour type data.

LIMITS Sets testing values for good data points.

WINDOW Controls windowing of data within axis bounds.

Appendix B Sec4.9 HELP

HELP VAX/VMS on-line help for PPLUS.

HLP Access on-line help from within PPLUS.

Appendix B Sec5 BEGINNERS GUIDE

To use PPLUS a minimum of preparation is required. See the chapter on Getting Started for the symbol definitions that are required. Once this has been done PPLUS can be entered by typing PPLUS (or just PPL) in response to the VAX/VMS prompt.

The minimum number of commands needed to read in data and then plot the data are: FORMAT (sets the input format), SKP (a command to position the file to a given record). VARS (tells PPLUS how the data is arranged in each data record), RD (reads the data) and PLOT (create the plot) or CONTOUR (create a contour plot). The name of the file containing the data can be specified with the RD or SKP commands. Following are discussions of these commands and some examples of how these commands are used. For more information see the Command Description chapter.

Appendix B Sec5.1 FORMAT

FORMAT informs PPLUS the type of the data file and the format the data has within this file. Valid formats are:

UNF -- the data is unformatted (data type REAL)

FREE -- the data is formatted and in free form

(xxx) -- the data is formatted with a format of xxx, where xxx is a legal FORTRAN format, i.e., (3F10.2)

Appendix B Sec5.2 5.2 VARS

The next command you need to know about is VARS. VARS is a complicated command because it allows great flexibility in the organization of the data within each file record. Position of the characters 1, 2, and 3 within the command line indicates the position of the X, Y, and Z variables within the data record. The format of the command is:

VARS,NGRP,A1, ... ,Ai

where i is the number of data values per data group

NGRP = number of groups per record. For example, if the data file has Depth,Temperature pairs packed 3 pairs per record with a format of 3(F6.1,F6.2) then NGRP=3.

Aj = 1, 2, 3 or blank to indicate that the variable in this position within the group is to be plotted as X (Aj = 1), Y (Aj = 2), Z (Aj = 3), or is not to be read at all (Aj = blank). An example will make this clearer.

EXAMPLE: VARS,1,,2,1

First arg is 1 --> there is only 1 group per record (e.g. 1 scan per line of data) in the data file

Second arg is blank --> Variable 1 in the data record is not to be read. (A1 = blank)

Third arg is 2 --> Variable 2 in the data record is to be plotted as Y (A2 = 2)

Fourth arg is 1 --> Variable 3 in the data record is to be plotted as X (A3 = 1)

No variable is to be read as Z.

The default is VARS,1,1,2 (i.e. one group per record, first variable is X, second is Y)

The following are examples of the VARS command.

VARS,1,,,1 tells PPLUS that there is one group of data per>record and to read the third number in the record as the X variable. Since no Y variable location has been specified the Y variable will contain the sequence number. VARS,5,1,2 lets PPLUS know that there are five groups of data pairs per record. Again the X variable is first and the Ysecond.

VARS,1,1,2,3 informs PPLUS that the data is X,Y,Z triplets with one group per record. The fact that X,Y, and Z appears tells PPLUS that the data is not on a regular grid and PPLUS should place it on an even grid. The method used to place the data on a regular grid and the grid itself are determined by the RD and CONSET commands.

VARS,1,,,,2,1 tells PPLUS that there is one group of data per record where the Y variable is the fourth number and the X the fifth number in the record.

VARS,1,3 tells PPLUS that there is one group of data per record and Z is the only variable in the group. This is for contour data which is already gridded. The RD command defines how the data is stored, i.e., which index varies fastest.

Appendix B Sec5.3 SKP AND RD

The name of the file containing the data to be plotted can be specified with either the SKP or the RD command. The SKP command tells PPLUS to skip records in the data file (e.g., header records or data which should not be plotted). Its format is SKP,N,FILE_NAME where N is the number or records to skip, and FILE_NAME is the name of the data file and is an optional parameter. If the name of the data file is included, the data file will be rewound before skipping. If the data file name is omitted, the file will not be rewound before skipping.

The RD command informs PPLUS how many records to read and what file to read them from. If you are not making a contour plot, the format of the command is RD,NX,FILE_NAME where NX is the number of points to read from the data file and FILE_NAME is the name of the data file and is an optional parameter. If the data file name is included, the data file will be rewound before the data is read. If the data file name is omitted, the file will not be rewound before reading.

If you are making a contour plot, the RD command format is somewhat different. If Z is being read (a 3 in the VARS command), RD defines the size of the plotting grid and prompts the user for the minimum and maximum values of X and Y to be used for the plotting grid. The format for RD is

RD,NX,NY,TYPE,FILE_NAME where NX and NY set the size of the grid for contour data read. Specifically, when X,Y,Z triplets are being read for contouring, the grid on which the data is plotted can be either coarser or finer or the same as the input data. If NX=50 and NY=21, then the data will be plotted on a grid which is 50 x 21 points (regardless of input data limits or gridding). TYPE tells PPLUS whether the data is stored by rows (X varies fastest) or columns (Y varies fastest) if the data is already-gridded contour data. Finally, FILE_NAME is the data file name. If the data file name is included, the data file will be rewound before the data is read. If the data file name is omitted, the file will not be rewound before reading.

Appendix B Sec5.4 PLOT AND CONTOUR

PLOT or CONTOUR initiates plotting. An optional label can be included and this label will be used to title the plot. The label must start with a non-numeric character. See following section on labels.

Appendix B Sec5.5 EXAMPLES

All the examples in this section can be typed in while running PPLUS interactively after typing PPLUS in response to the VAX/VMS prompt. Just be sure you have first defined the PPLUS symbols according to the Getting Started chapter before you try to do this. Once the plot appears on your terminal, enter <CR> to exit from graphics mode and continue. To exit from PPLUS, type EXIT.

Appendix B Sec5.5.1 Unformatted Data, X-Y Plot

The following example reads in data from an unformatted filewith one group of data per record. The data to be plotted has Xin the second position and Y in the first. The data file has

296data points in it but we will read only 100 at a time. The datafile also has an 8 record header that contains character data andmust be skipped.

```
ppl>FORMAT UNF
ppl>VARS,1,2,1
ppl>SKP,8,PPL$EXAMPLES:DEEP3000.AVG
ppl>RD,100
ppl>PLOT,The first 100 data points
ppl>RD,100
ppl>PLOT,The second 100 data points
```

Appendix B Sec5.5.2 Pre-gridded Data, Contour Plot

The next example illustrates reading in data to be contoured. The data file is unformatted and does not have any header. The data is already gridded with 1 value of Z per record. Since only Z is read from the data file, the input grid and the plotting grid must be identical, and are specified by the RD command. The grid is 34 points in the x-direction and 5 points in the y-direction. The PPLUS RD command prompts for the minimum and maximum for the X-Y contouring grid. In this example, the grid is 34 points in the x-direction from 10 to -6.5 units and is 51 points in the y-direction from 0 to -500 units. PPLUS will read Z values from the data file assuming x varies fastest. This means that the Z values on the data file correspond to the following x,y pairs:

```
  x     y
 10.0    0
  9.5    0
  9.0    0
   .
   .
 -6.5    0
 10.0  -10
  9.5  -10
  9.0  -10
   .
   .
 -6.5  -10
   .
   .
```

```
ppl>FORMAT,UNF
ppl>VARS,1,,3
ppl>RD,34,51,1,PPL$EXAMPLES:CTDDAT.DAT
ENTER XMIN,XMAX,YMIN,YMAX
rd>10,-6.5,0,-500
ppl>CONTOUR,A test plot for contouring
```

Appendix B Sec5.5.3 Ungridded Data, Contour Plot

This example shows the reading in of ungridded contour data. The data is unformatted with Y,X,Z the order of the triplets. We define the grid for plotting to be 22 x 11 with X and Y limits of 1,22 and -.033,.0576. Although the data file contains less than 1000 points, we can give PPLUS a much larger number to read, and it will stop at the end-of-file without error.

```
ppl>FORMAT,UNF
ppl>VARS,1,2,1,3
ppl>RD,22,11,PPL$EXAMPLES:GRIDWI.FMT
ENTER NUMBER PTS TO READ
rd>1000
ENTER XMIN,XMAX,YMIN,YMAX
rd>1,22,.033,.567
ppl>CONTOUR,An example of contouring with ungridded data
```

Appendix B Sec5.5.4 Time Series Plot

This example demonstrates the reading in of time series data and setting up the x axis to be a time axis. The data file contains a sequence number, which is the day of the year or Julian Day and temperature. Since the sequence number increments by 1 for 1 day, and delta-time is 1 day by default in PPLUS, there is no need to include the delta-time in the TAXIS command. The TAXIS command tells PPLUS that the time series has a delta-time of 1440 minutes (the default) and that the time axis is to be turned on. (The alternate form of the TAXIS command would be "taxis,1440,on".) The TIME command tells PPLUS that the time axis will start at 0000 1 Jul 85, end at 0000 1 Dec 85, and that a sequence number of 1 corresponds to a time of 1200 1 Jan 85. The YLAB command sets the y-axis label. The LIMITS command tells PPLUS to omit data where Y = 0. The VARS command is not needed since the data is formatted with one group of data per record, with the X variable first and the Y variable second, which is the VARS command default. The CROSS command suppresses the drawing of a solid line through x=0, y=0 on the plot. The BOX command suppresses the drawing of a box around the entire plotting region. The SKP command names the data file and skips past the 5 header records at the beginning of the data file. The RD command reads the data. The PLTYPE command sets the plotting medium to be both Tektronix compatible and binary suitable for routing to hardcopy devices. The PLTNME sets the name of the output plot file. The PLOT command generates the plot. See the Command Description chapter for a full description of all PPLUS commands.

```
ppl>format (17x,f3.0,7x,f5.0)
ppl>taxis,on
ppl>time,W8507010000,W8512010000,W8501011200
ppl>ylab,Air Temperature
ppl>limits,0,yeq,on
ppl>cross,0
ppl>box,off
ppl>skp,5,ppl$examples:atlas.dat
ppl>rd
ppl>pltype,2
ppl>pltnme,atlas.plt
ppl>plot,ATLAS Air Temperature at 2N 165E
```

Additional examples are in the directory PPL$EXAMPLES in the form of PPLUS command files, which are the files with extension .PPC. Use the VAX/VMS command "DIR PPL$EXAMPLES:*.PPC" to see what the file names are. You can run these command files with the VAX/VMS command "PPLUS PPL$EXAMPLES:xxx.PPC", where xxx is the name of the PPLUS command file. The file will generate a plot on your terminal. Enter a <CR> to exit from graphics mode and return to the VAX/VMS prompt. (Be sure that you have first defined the PPLUS symbols according to the Getting Started chapter before you do this.) See the chapter on Command Files for more information about using PPLUS command files.

You can copy these PPLUS command files to your own directory with the VAX/VMS command "COPY PPL$EXAMPLES:*.PPC []". Then you can run them with the VAX/VMS command "PPLUS xxx.PPC", where xxx is the name of the PPLUS command file. You can experiment with PPLUS commands by editing the PPLUS command file to change the appearance of the plot, and then run PPLUS again with your new command file.

Appendix B Sec6 ROUTING PLOT FILES

Appendix B Sec6.1 VAX/VMS

Appendix B Sec6.1.1 Plot Files And Mom

PPLUS plot files are named ZETA.PLT by default (this can be changed with the PLTNME command). A graphics postprocessor called MOM is available to reformat these binary plot files and route them to a graphics device. MOM submits a batch job to BETA$LOPRI or BETA$BATCH. When the batch job has finished, the original plot files will have been renamed from file.ext to file.PLT_HHMMSS, and the plots queued to the appropriate device. A log file with the name MOM_HHMMSS.LOG is placed in the original directory when the MOM option /LOG is selected.

The command is (brackets [] enclose optional information):

MOM [arg1 [arg2 ...]]

The arguments for MOM are order independant and are separated by spaces. The arguments are:

[F[ILE]=]file name (default ZETA.PLT)

[D[EVICE]=]device (e.g. TEK, VER etc, default VER)

S[CALE]=scale factor (default 1)

G[RACE]=grace distance (inches, default = 0.25)

W[IDTH]=width (paper width CAL only, default = 11.5)

C[PLOT]="cplot arguments" (CPLOT parameters CAL only, default=NULL)

[NO]ROT[ATE] (rotate the plot, default NOROT)

[NO]CEN[TER] (center the plot, default CENTER)

/[NO]SAVE (save the input file, default /SAVE)

/[NO]LOG (create a batch log file, default /NOLOG)

/SMALL, /LARGE or /TRANS (type of hard copy made, default /SMALL)

File names which are the same as a legal device name (e.g. VER, TEK, etc.) are not allowed. The file name can contain any wild carding that is valid with the VAX/VMS rename command. The default file extension is .PLT.

Appendix B Sec6.1.2 Plotting Devices

VER Batch plot on Versetec V80 printer/plotter

TEK Interactive plot on Tekronix compatible terminal

CPY Batch plot on Tekronix 4691 hardcopy unit

CAL Batch plot on CALCOMP plotter

HP Batch plot on HP7550A plotter

LN03 Batch plot on TMAP1:: LN03 printer/plotter

HPT Batch plot on TMAP1:: HP7475

Appendix B Sec6.1.3 Examples

1) $MOM question

Will cause MOM to prompt for inputs. If the CPLOT argumentis a ? you are then prompted for the CPLOT inputs.

2) $MOM CTD110W VER SCALE=1.25 ROTATE

Will instruct MOM to create a VERSATEC plot from the metafile CTD110W.PLT, rotate the plot 90 degrees on the paper and rescale the plot by a factor of 1.25.

3) $MOM CAL CPLOT=""

Will have MOM create a CALCOMP plot using ZETA.PLT and cal lCPLOT with the default parameters. If CPLOT is omitted then MOM will prompt for the CPLOT command line (omitting CCFILE).

4) $MOM TEMP.PLT;* CAL CPLOT="/P1=BLK:.3"

Will cause MOM to send all the versions of TEMP.PLT to the CALCOMP with operator instructions to have pen 1 be black ink pen of 0.3 mm width.

5) $MOM HP *.MYPLOT;* /TRANS

Will send all plots with extension .MYPLOT to the HP7550 plotter with operator instructions to plot on transparencies.

Appendix B Sec7 PPLUS COMMAND FILES

Appendix B Sec7.1 INTRODUCTION

PPLUS can be run using a PPLUS command file that contains the same commands used by PPLUS interactively. The file can have any name or extension, but the default extension is .PPC. To run a PPLUS command file named CMD.PPC, you can enter PPLUS by typing PPLUS CMD.PPC in response to the VAX/VMS prompt, or you can enter PPLUS in the usual way and give the PPLUS command @CMD.PPC. (See @ in the chapter on Command Description.)

Each time PPLUS is used, an echo file (named ECHO.DAT by default) is generated. This file can be edited (it should be renamed) with any VAX text editor and used as a PPLUS command file in subsequent PPLUS sessions.

Appendix B Sec7.2 SYMBOL SUBSTITUTION

PPLUS allows symbol substitution in a manner similar to VAX/VMS symbols. Global and local symbols are supported in conjunction with nested command files and parameter passing. The SET and SHOW commands create, modify and list the symbols. When initially entering PPLUS (i.e., at the first command level) the symbols are global and available to all command levels. At each subsequent command level, local symbols are created and used by default. Global symbols are used when no local symbol exists. If the symbol name is preceded by a star (*), the global symbol will be created, modified or substituted.

Parameters passed via the @ command line are named P1, P2, P3, etc... just as they are in VAX/VMS. Symbols are recognized by PPLUS by being enclosed by single quotes. Character strings can be enclosed in double quotes. For example:

```
SET TEMP "This is a test label"
XLAB 'temp'
```

will have the same effect as:

```
XLAB This is a test label
```

Several symbols are predefined. 'DATE' and 'TIME' contain the current date and time. Date and time formats are dd-mmm-yy and hh:mm:ss. In addition, P1 through Pn are also predefined if the corresponding argument was passed via the @ command. For example, the command procedure PLOTIT.PPC could be executed in PPLUS by typing @PLOTIT 110W Temperature. Then in the procedure PLOTIT, the symbol P1 will have the value "110W" and the symbol P2 will have the value "Temperature".

Symbols can also be defined and used in an array format, i.e., 'P(3)' will get symbol P3 and 'label(12)' will access symbol LABEL12.

To have a single quote (') in the symbol or command line two single quotes must be used ('').
To have a double quote (") in the command line two double quotes ("") are required.

Here is a sample PPLUS command file which demonstrates some of the new, powerful PPLUS features. In this example, the symbol P1 has the value 110W.

```
pltnme,'p1'.plt
format,(f5.0,15x,f15.0)
vars,1,1,2
skp,1,'p1'.dat
rd,60
debug,on
show p1
debug,off
plot,@TRMonthly data 1979-83 at 'P1' ('date' 'time')
```

The proceeding PPLUS command file (named PLOTIT.PPC) could be called repeatedly in PPLUS for different data files named 110W.DAT, 140W.DAT, etc. by entering the PPLUS

commands @PLOTIT 110W, @PLOTIT 140W, etc. The resulting plot files, ECHO.DAT files and graphs would be identified by the data file names of 110W, 140W, etc. The graph title will also include the time and date when the graph was made.

Appendix B Sec7.3 GENERAL GLOBAL SYMBOLS

The global symbols set by PPLUS to allow information to be available in the command procedure are:

command	SYMBOL	COMMAND	DESCRIPTION
DATE			The current date dd-mmm-yy
PPL$COMMAND_FILE	@		The current command file name.
PPL$EOF	RD,RWD,SKP		"YES" if an EOF was read.
PPL$FORMAT	FORMAT		The current format.
PPL$HEIGHT	SIZE		Height of the box.
PPL$INPUT_FILE	RD,SKP,RWD		The current input file.
PPL$LF_A	LINFIT		Constant from fit y= a + b*x
PPL$LF_A_STDEV	LINFIT		Standard error of A.
PPL$LF_B	LINFIT		Constant from fit.
PPL$LF_B_STDEV	LINFIT		Standard error of B.
PPL$LF_R2	LINFIT		Regression coefficient squared.
PPL$LF_RES_VAR	LINFIT		Residual variance.
PPL$LF_VAR	LINFIT		Total variance.
PPL$LINE_COUNT	-		The number of the last line read.
PPL$PLTNME	PLTNME		The name of the plot file.
PPL$RANGE_INC	%RANGE		See Advanced Commands Chapter
PPL$RANGE_HIGH	%RANGE		See Advanced Commands Chapter
PPL$RANGE_LOW	%RANGE		See Advanced Commands Chapter
PPL$TEKNME	TEKNME		The name of the tektronix file.
PPL$VIEW_X	VPOINT		X viewpoint
PPL$VIEW_Y	VPOINT		Y viewpoint
PPL$VIEW_Z	VPOINT		Z viewpoint
PPL$WIDTH	SIZE		Width of the box.
PPL$XFACT(n)	TRANSXY		Xfact for line n.
PPL$XLEN	AXLEN		Length of X axis.
PPL$XOFF(n)	TRANSXY		Xoff for line n.
PPL$XORG	ORIGIN		Distance between origin and left edge.
PPL$XFIRST(n)	-		X value for first data point in line n.
PPL$XLAST(n)	-		X value for last data point in line n.
PPL$XMAX	RD		Xmax of contour grid
PPL$XMIN	RD		Xmin of contour grid
PPL$XMAX(n)	-		Xmax for valid data in line n.
PPL$XMIN(n)	-		Xmin for valid data in line n.
PPL$YFACT(n)	TRANSXY		Yfact for line n.
PPL$YLEN	AXLEN		Length of Y axis.
PPL$YOFF(n)	TRANSXY		Yoff for line n.
PPL$YORG	ORIGIN		Distance between origin and bottom edge.
PPL$YFIRST(n)	-		Y value for first data point in line n.
PPL$YLAST(n)	-		Y value for last data point in line n.
PPL$YMAX	RD		Ymax of contour grid
PPL$YMIN	RD		Ymin of contour grid

```
PPL$YMAX(n)          -          Ymax for valid data in line n.
PPL$YMIN(n)          -          Ymin for valid data in line n.
PPL$ZMAX             -          Zmax for valid contour data.
PPL$ZMIN             -          Zmin for valid contour data.
TIME                 -          The current time hh:mm:ss
```

Appendix B Sec7.4 EPIC GLOBAL SYMBOLS

The following global symbols set by PPLUS contain information from EPIC time series data headers:

```
 SYMBOL                      COMMAND DESCRIPTION

PPL$EPIC_COMMENT_DATA(n)     RD      Data comment from header.
PPL$EPIC_COMMENT_FIRST(n)    RD      Data comment from header.
PPL$EPIC_COMMENT_SECOND(n)   RD      Data comment from header.
PPL$EPIC_DEPTH(n)            RD      Depth of measurement.
PPL$EPIC_DESCRIPT(n)         RD      EPIC series descriptor.
PPL$EPIC_EXPERIMENT(n)       RD      Experiment identifier.
PPL$EPIC_LATITUDE(n)         RD      Latitude.
PPL$EPIC_LONGITUDE(n)        RD      Longitude.
PPL$EPIC_MOORING(n)          RD      Mooring identifier.
PPL$EPIC_PROJECT(n)          RD      Project identifier.
PPL$EPIC_XLAB(n)             RD      X-axis label.
PPL$EPIC_YLAB(n)             RD      Y-axis label.
```

The following global symbols set by PPLUS contain information from EPIC CTD data headers:

```
 SYMBOL                      COMMAND DESCRIPTION

PPL$EPIC_CAST(n)             RD      CTD Cruise and Cast identifier
PPL$EPIC_COMMENT_FIRST(n)    RD      Data comment from header.
PPL$EPIC_COMMENT_SECOND(n)   RD      Data comment from header.
PPL$EPIC_DATE(n)             RD      CTD Cast Date (GMT)
PPL$EPIC_LATITUDE(n)         RD      Latitude.
PPL$EPIC_LONGITUDE(n)        RD      Longitude.
PPL$EPIC_XLAB(n)             RD      X-axis label.
PPL$EPIC_YLAB(n)             RD      Y-axis label.
```

The following global symbol set by PPLUS contains information about the EPIC data file:

```
SYMBOL                     COMMAND DESCRIPTION
  PPL$EPIC_DATAFILE(n)        RD      Data file name
  PPL$INPUT_FILE                      RD EPIC/pointer file
```

Appendix B Sec7.5 COMMAND FILE LOGIC

There are several commands that enable the user to make command files more like small programs. These commands are similar to FORTRAN's block IF and C's WHILE loops. Commands have been introduced that enable the user to increment and decrement a counter stored in a symbol by one. In order to make command files more readable leading blanks and tabs are ignored.

The syntax for the PPLUS commands is given in the Command Description chapter.

EXAMPLES:

In this example, PPLUS is exited when an end-of-file is encountered by the RD command. This illustrates both the block IF and the use of the global PPLUS symbol PPL$EOF.

```
RD
IF PPL$EOF .EQ. "YES" THEN
    EXIT
ENDIF
```

In the following example, the size of the plot is set to val by val inches if the value of the symbol val is less than or equal to 13 otherwise the size is set to 13 x 13.

```
IF VAL .LE. 13 THEN
    SIZE 'VAL' 'VAL'
ELSE
    SIZE 13 13
ENDIF
```

In the next example, if P1 is null then P1 is set to TEMPORARY.PLT and then the plot name is set to the value of the symbol P1.

```
IF P1 .EQ. "" THEN
    SET P1 TEMPORARY.PLT
ENDIF
PLTNME 'P1'
```

This WHILE loop results in 10 plots of 100 points each from data file DLDK1039.DAT. (PPL$LINE_COUNT is a PPLUS defined symbol for the sequence number of the last data line read.)

```
SKP,DLKD1039.DAT
WHILE PPL$LINE_COUNT .LE. 10 THEN
    RD,100
    PLOT
ENDW
```

Appendix B Sec7.6 ARITHMETIC

Simple arithmetic can be performed using PPLUS symbols. The commands that perform these function are SET, INC and DEC. The INC and DEC functions are primarily used to increment and decrement counters in WHILE loops. The following WHILE loop uses the counter to set the line type to a solid line for each line to be plotted (PPL$LINE_COUNT is a PPLUS defined symbol for the number of the last data line read):

```
SET COUNT 1
WHILE COUNT .LE. PPL$LINE_COUNT THEN
    LINE,'COUNT',,0
    INC COUNT
ENDW
```

The SET command can be used to perform simple arithmetic on PPLUS symbols. The syntax for these arithmetic expressions have the form:

num1 op num2,

where op is +, -, * or / (addition, subtraction, multiplication or division) and num1 and num2 are numbers. The numeric values must be separated from the operator op by spaces. The string will be used exactly as it appears if enclosed by double quotes ("). The following example centers a moveable label 0.5 inches above the top axis (PPL$XLEN and PPL$YLEN are PPLUS symbols for the X and Y axis lengths):

```
SET XPOS 'PPL$XLEN' / 2.0
SET YPOS 'PPL$YLEN' + 0.5
LABS/NOUSER,1,'XPOS','YPOS',0,"A centered label"
```

Appendix B Sec7.7 SYMBOL ARRAYS

As described in the SYMBOL SUBSTITUTION section, PPLUS symbols can be defined and used as arrays. There are several general PPLUS global symbols which are also defined as arrays, such as PPL$XLAST(n) and PPL$YLAST(n), the last x and y values for data line n. The array index, in parentheses, can be either a number or a PPLUS symbol. Examples will illustrate this.

The following piece of a PPLUS command file uses moveable lables to write the line number to the right of the last point plotted for the last line read in. It uses the global PPLUS symbols PPL$XLAST(n), PPL$YLAST(n) and PPL$LINE_COUNT.

```
SET XPOS 'PPL$XLAST(PPL$LINE_COUNT)'
SET YPOS 'PPL$YLAST(PPL$LINE_COUNT)'
LABS 'PPL$LINE_COUNT','XPOS','YPOS',-1,'PPL$LINE_COUNT'
```

The array index can also be a user defined symbol. In the following example, the array MON contains the names of the first 3 months of the year. The graph title will be "Daily Values for the Month of FEBRUARY".

```
set mon(1) "JANUARY"
set mon(2) "FEBRUARY
set mon(3) "MARCH"
.
.
.
set count 2
.
.
.
plot,"Daily Values for the Month of 'mon(count)'
```

The index of an array (inside parentheses) will be interpreted according to the following rules: 1) if it is a number, that number will be used as the array index, 2) if it is not a number, it will be interpreted as a symbol, 3) if it is in single quotes, it will be interpreted as a symbol.

Appendix B Sec7.8 SPECIAL FUNCTIONS

The functions described in this sections are all accessed with the SET command. They can be accessed only with the SET command. The functions enable string manipulation and formatting within PPLUS symbol values. The PPLUS functions are similar to some of the VAX/VMS lexical functions.

The general syntax is :

SET sym $function (arg1, arg2,...),

where "sym" is the symbol set by the function and "function" is the name of the PPLUS function. PPLUS functions and their arguments are described in the following sections. Where function arguments are indicated as symbols, they must be PPLUS symbols and cannot be strings. Where function arguments are indicated as strings, they can be enclosed in double quotes.

Appendix B Sec7.8.1 $EDIT

The command is :

```
SET sym_out $EDIT (sym_in, arg1 [ arg2 arg3...] )
```

where:

sym_out = symbol set by the function

sym_in = symbol on which function is to work

arg1 = UPCASE - changes string in sym_in to upper case
= TRIM - trims leading and trailing blanks from sym_in
= COMPRESS - removes extra blanks from sym_in (reduces each group of blanks to a single blank)
= COLLAPSE - removes all blanks from sym_in

If multiple arguments are used, they can be separated by blanks, e.g., SET sym $EDIT(sym_in,UPCASE COLLAPSE). If commas are used as separaters, the entire set of arguments must be enclosed in quotes, e.g.,
SET sym $EDIT(sym_in,"UPCASE,COLLAPSE").

Example:

```
SET S1 "depth"
SET S2 $EDIT (S1,UPCASE)
```

This results in S2 having the value "DEPTH".

Example:

```
SET S1 "  depth    "
SET S2 $EDIT (S2,UPCASE TRIM)
```

This results in S2 having the value "DEPTH".

Appendix B Sec7.8.2 $EXTRACT

This function extracts selected characters from the input string. The first character in the string is in position 1. The command is :

```
SET sym_out $EXTRACT (start,length,sym_in)
```

where:

sym_out = symbol set by the function

start = starting character position

length = length of character string to be extracted

sym_in = symbol on which function is to work

Example:

```
SET S1 "February"
SET S2 $EXTRACT(1,3,S1)
```

This results in S2 having the value "Feb".

Appendix B Sec7.8.3 $INTEGER

This function converts a number to integer format. The command is :

```
SET sym_out $INTEGER (sym_in)
```

where:

sym_out = symbol set by the function

sym_in = symbol on which function is to work

Example:

```
SET MON 1
.
.
INC MON
SET INT_MON $INTEGER(MON)
```

In this example, the symbol MON has been incremented, and will have the value "2.00". The symbol INT_MON will have the value "2".

Appendix B Sec7.8.4 $LENGTH

This function returns the length of the input string. The command is :

```
SET sym_out $LENGTH (sym_in)
```

where:

sym_out = symbol set by the function

sym_in = symbol on which function is to work

Example:

```
SET S1 "February"
SET S2 $LENGTH(S1)
```

This results in S2 having the value "8".

Appendix B Sec7.8.5 $LOCATE

[This function is deprecated; see the Ferret function SUBSTRING] This function locates a substring in the input string. The first character in the string is in position 1. The command is :

```
SET sym_out $LOCATE (substrg,sym_in)
```

where:

sym_out = symbol set by the function

substrg = string to be located. The first 30 characters of this string are compared with sym_in.

sym_in = symbol function on which function is to work

Example:

```
SET S1 "JAN 21,1987"
SET S2 $LOCATE(",",S1)
```

This results in S2 having the value "7".

Appendix B Sec7.8.6 $ELEMENT

This function extracts an element from an input string in which the elements are separated by a specified delimiter. The command is :

```
SET sym_out $ELEMENT (pos,delim,sym_in)
```

where:

sym_out = symbol set by the function

pos = position of element to be extracted

delim = delimiter

sym_in = symbol on which function is to work

Example:

```
SET MONTH "JAN/FEB/MAR/APR/MAY/JUN/JUL"
SET MON $ELEMENT(3,"/",MONTH)
```

This results in MON having the value "MAR".

Example:

```
SET MONTH "JAN/FEB/MAR/APR/MAY/JUN/JUL"
SET COUNT 1
WHILE COUNT .LE. 7 THEN
   SET MON(COUNT) $ELEMENT('COUNT',"/",MONTH)
   INC COUNT
ENDW
```

This results in MON(1) = "JAN", MON(2) = "FEB", MON(3) = "MAR", MON(4) = "APR", MON(5) = "MAY", MON(6) = "JUN", MON(7) = "JUL".

Appendix B Sec7.9 LABELS

Appendix B Sec7.9.1 AXIS LABELING

Commands affecting the labeling of the axes are:

XAXIS	Controls numeric labeling and tics on the x-axis.
YAXIS	Controls numeric labeling and tics on the y-axis.
AXATIC	Sets number of large tics automatically for x and y.
AXLABP	Locates axis labels at top/bottom or left/right of plot.
AXLEN	Sets axis lengths.
AXLINT	Sets label interval for axes.
AXLSIG	Sets axis label heights.
AXNMTC	Sets number of small tics between large tics on axes.
AXNSIG	Sets no. significant digits in numeric axis labels (auto only).
AXSET	Allows omission of plotting of any axis.
AXTYPE	Sets axis type for x- and y-axis.
XFOR	Sets format of x-axis numeric labels.

YFOR	Sets format of y-axis numeric labels.
XLAB	Sets label of x-axis.
YLAB	Sets label of y-axis.

The numeric axis labels are drawn such that zero will be labelled if it occurs between the low and high axis limits. If zero does not occur, then the first large tic (from the bottom or left) will be labelled. The large tics are forced to occur at integer multiples of the tic interval.

Appendix B Sec7.9.2 EMBEDDED STRING COMMANDS

Fonts

All labels in PPLUS can be plotted using any one of 21 character fonts and 11 symbol fonts. The default font is SR (Simplex Roman) and other fonts are called by preceding their two letter abbreviation by an @, i.e., @CI for complex itallic.

Symbol fonts are called by using the symbol number, i.e., @MA01 plots the first symbol in MATH and @MA12 will plot the twelfth symbol. Font changes (of the form @XX) can be imbedded in any label string (e.g., XLAB, YLAB, PLOT commands).

@font selects "font" as the character or symbol font to be used, where the font abbreviations are listed below.

Character Fonts

Tables showing these fonts are linked to the web page:

http://ferret.pmel.noaa.gov/Ferret/Documentation/Users_Guide/pplus_char_fonts.html

SR	Simplex Roman (default)
DR	Duplex Roman
TR	Triplex Roman
CR	Complex Roman
AS	ASCII Simplex Roman
AC	ASCII Complex Roman
CS	Complex Script
TI	Triplex Italic
GE	Gothic English
IR	Indexical complex Roman
SS	Simplex Script

CI	Complex Italic
II	Indexical complex Italic
SG	Simplex Greek
CG	Complex Greek
IG	Indexical complex Greek
GG	Gothic German
GI	Gothic Italian
CC	Complex Cyrillic
AR	Cartographic Roman
AG	Cartographic Greek

Symbol Fonts

Tables showing these fonts are linked to the web page:

http://ferret.pmel.noaa.gov/Ferret/Documentation/Users_Guide/pplus_symbol_fonts.html

ZO	Zodiac
MU	Music
EL	Electrical
WE	Weather
MA	Math
SM	Simplex Math
MP	Map
LM	Large Math
IZ	Indexical Zodiac
IM	Indexical Math
CA	Cartographic

A clear font command @CL is available to change the default font. The next font called after a @CL becomes the new default font. The font is reset to the default at the start of each label. The command DFLTFNT can also be used to change the default font to one of your choice.

Tables showing the symbol fonts are at

http://ferret.pmel.noaa.gov/Ferret/Documentation/Users_Guide/current/ppl_symbol_fonts.htm

Control characters for the two ASCII fonts AS and AC must be preceded by an <ESC> (ascii code=27). For example, to superscript while using the ASCII fonts you must have <ESC> in the label preceed the character to superscript.

Appendix B Sec7.9.3 Pen Selection

The pen may also be selected by giving the change pen command @Pn, where n is the character 1-9 and A-G. This allows the selection of up to 16 pens/colors. The color and font is reset to the default font and previous color after the character string is drawn. The PEN command can be used to change the default color by typing PEN,0,default_color.

If you need to select a color index beyond the range of P1 through PG, you can use the change color command @Cnnn where "nnn" is a 3-digit color index. (It must be 3 digits.)

Appendix B Sec7.9.4 Character Slant

The slant used in drawing the fonts may be changed by using the command @Zn, where n is the character 0-9 and A-G. This allows the selection of slant angles from 0 to 45 in 16 increments. The slant is reset to zero after the character string is drawn.

Appendix B Sec7.9.5 Subscripting, Superscripting And Back Spacing

An ^ (up arrow) imbedded in any label string will cause the next character to be drawn superscripted, an _ (underscore) will draw it subscripted, and a \ (backslash) backspaces over the last character drawn. The control characters ^, _ and \ are available in the two ASCII fonts AS and AC by preceding the control character by an <ESC> (ASCII code=27). For example, to subscript while using the ASCII fonts you must have <ESC>_ in the label preceed the character to subscript.

Appendix B Sec7.10 DATA FORMATS

Appendix B Sec7.10.1 SEQUENTIAL FORMATS

The format to be used in reading from a sequential file is defined by the commands FORMAT, VARS, and RD. Some definitions are useful:

NVAR - the number of variables per group

NGRP - the number of groups per record

NREC - the total number of records

For example, if the data consists of depth, u, v, t and the format is 8F10.2 (the format statement must be for an entire record) with two groups per record, the data would look like

D U V T D U V T

and NGRP=2, NVAR=4.

If you wanted to plot D as the Y variable, T as the X then, FORMAT (8F10.2) would be the correct FORMAT command and VARS,2,2,,,1 would be the correct VARS command. (U and V are not read or plotted.)

However, if the format was F10.2,30X,2F10.2,30X,F10.2 then FORMAT (F10.2,30X,2F10.2,30X,F10.2) and VARS,2,2,1 would be appropriate.

If the data is unformatted the meanings of NVAR and NGRP are unchanged. Unformatted data is specified by the FORMAT command FORMAT,UNF.

Reading will automatically stop at the end of the file and properly store the data.

Appendix B Sec7.10.2 BIBO FORMAT

The BIBO data format consists of data files created using the DSF routines and a 145 word header in the BIBO format. This data format is in the standard dsf file format for data storage.

Appendix B Sec7.10.3 EPIC FORMAT

This is the standard format for data from the EPIC data base. The data files are binary sequential files with at least one header of 8 80-character lines followed by data records with 1 data scan per record. When the FORMAT,EPIC command is used, the file name specified with the RD, SKP and RWD commands refers to the EPIC or pointer file. Variables to be read are specified with the EVAR command. Both time series EPIC data files and CTD EPIC data files are recognized by PPLUS. The /CTD qualifier on the FORMAT command tells PPLUS which type of EPIC data is being read.

Appendix B Sec7.10.4 DSF FORMAT

This data format is that produced by the DSF routines with the header and data in PPLUS format. The format must be followed to ensure that PPLUS can interpret the data file read correctly.

A single data file consists of a single header record and any number of data records followed by an EOF. The header must be either an array or other sequentially organized data set of 38 real variables. Below is the expected format.

INT WORD	DESCRIPTION
1	XPTS
3	ZMIN first four created by CLSDSF
5	ZMAX
7	ZMEAN
9	XMIN minimum x value (real)
11	XMAX maximum x value (real)
13	KX number of x grid points (integer*4)
15	YMIN minimum y value (real)
17	YMAX maximum y value (real)
19	KY number of y grid points (integer*4)
21	ITYPE data type 0= 2-d set, 1= 1-d set (integer*4)
23-38	LAB(16) main label hollerith (integer*2)
39	NCH number of characters is LAB (integer*4)
41-56	IXLAB(16) x axis label hollerith (integer*2)
57	NXLB number of characters in IXLAB (integer*4)
59-74	IYLAB(16) y axis label hollerith (integer*2)

75 NYLB number of characters in IYLAB (integer*4)

All labels use SYMBEL to generate the plotted characters. The labels are optional, but if not used they should contain blanks.

ITYPE=0

Data must be stored in a linear array as:

Z(1,1),Z(2,1),...,Z(KX,1),Z(1,2),...,...,Z(KX,KY)

or as a 2-d array where the array is dimensioned as KX,KY.

Assuming the following arrays exist, ITYPE=0 data can be created as follows: HEAD(38),Z(25,50) NOTE: use EQUIVALENCE to set the integers in the real array.

```
CALL OPNDSF(file_name,'WR',ILUN)

CALL WRHDSF(ILUN,38,HEAD)

CALL WRDDSF(ILUN,1250,Z)

CALL CLSDSF(ILUN)
```

where file_name is the file name and ILUN is the logical unit to be used.

ITYPE=1

Data must be stored as a linear array as:

X(1),X(2),...,X(KX),Y(1),Y(2),...,Y(KX)

in this case KX= length of the series and KY must be set to 1, there must be KX of each X and Y in the data set. Given,

HEAD(38),X(200),Y(200) KX=100 then,

```
CALL OPNDSF(file_name,'WR',ILUN)

CALL WRHDSF(ILUN,38,HEAD)

CALL WRDDSF(ILUN,KX,X)

CALL WRDDSF(ILUN,KX,Y)
```

CALL CLSDSF(ILUN)

where KX is the number of pairs. The DSF routines are available in a user library by Task building with

DISK1:[DENBO.PPL]OURLIB/LIB.

Appendix B Sec7.11 ADVANCED COMMANDS

This section describes PPLUS primitive plot commands. With these commands, the user can make a plot with several x- or y-axes. The location of each axis can be specified. To distinguish them from the standard PPLUS commands, these commands all begin with "%".

These % commands can be entered only from a PPLUS command file, and can not be entered interactively from the keyboard. Each command is implemented as it is read from the command file.

Specifically, when the %XAXIS command is read from a command file, an x-axis is immediately drawn on the graph. By contrast, the standard PPLUS XAXIS command simply sets x-axis parameters and the x-axis is not drawn on the graph until a plotting command such as PLOT is issued. The % commands give the user great control over the graphics display, but must be used carefully. No PPLUS error messages are issued for illegal % commands. The % commands can not be used with the MULTPLT command. See the notes with each command description and the example at the end of this chapter.

Command descriptions follow.

Appendix B Sec7.11.1 %OPNPLT/qualifier

Opens the plot by putting the terminal into and out of graphics mode and setting /QUIET.

Valid qualifiers are:

/[NO]OVERLAY Controls whether PPLUS overlays the plot on the preceeding plot. The default is /OVERLAY which causes the plot to be overlaid without erasing the last plot.

Appendix B Sec7.11.2 %CLSPLT/qualifiers

Closes the plot by putting the terminal out of graphics mode and restoring /QUIET or /NOQUIET, whichever was in effect when the %OPNPLT command was issued.

Valid qualifiers are:

/[NO]WAIT

Controls whether PPLUS pauses after plot completion. Pause is signaled by a tone and terminated by typing a character. If an <ESC> is typed PPLUS will return from the current command level to the lowest command level. Default = WAIT.

Appendix B Sec7.11.3 %PLTLIN,n

Plots the n-th data line. Each RD command increments the data line count by 1. Use of the standard plotting commands (PLOT, PLOTUV, PLOTV, CONTOUR, VECTOR, and VIEW) resets the data line count. The %PLTLIN command does not reset the data line count. (WINDOW works.)

n Plot line n using current scale factors.

Appendix B Sec7.11.4 %LABEL/qualifier,x,y,ipos,ang,chsiz,label

Draws a label similar to a moveable label (LABS command). There is no label number and the label is drawn as soon as the command is read from the command file. Any number of labels may be drawn.

x	x position user or inches
y	y position user or inches
ipos	-1 left, 0 center, +1 right justify
ang	Angle at which lable is to o be drawn. (0 degrees is at 3 o'clock and positive rotation is counter clockwise.)
chsize	character size (inches)
label	character string to draw

Valid qualifiers are:

/[NO]USER determines units of x and y positions. Default is /USER. If /NOUSER units are inches from the ORIGIN. (see the ORIGIN command)

Appendix B Sec7.11.5 %RANGE,min,max,ntic

Finds axis limits for use with the %XAXIS and %YAXIS commands given the data extrema of min and max. The axis limits and tic interval are returned in the PPLUS symbols PPL$RANGE_LOW, PPL$RANGE_HIGH, and PPL$RANGE_INC.

min	minimum value of data to be ranged. Can use PPL$XMIN(n) or PPL$YMIN(n).
max	maximum value of data Can use PPL$XMAX(n) or PPL$YMAX(n).
ntic	number of large increments

PPL$RANGE_LOW	new minimum range value
PPL$RANGE_HIGH	new maximum range value
PPL$RANGE_INC	new increment

Appendix B Sec7.11.6 %XAXIS/qualifier,xlow,xhigh,xtic,y[,nmstc][,lint][,xunit][,ipos][,csize][,frmt]

This command draws an x-axis and redefines scaling for the x-direction. The arguments xlow, xhigh, xtic and y should not be omitted. See the %RANGE command to get default values for axis limits and increments. If you have used %RANGE, then you can use

PPL$RANGE_LOW, PPL$RANGE_HIGH, PPL$RANGE_INC for xlow, xhigh and xtic.

xlow	min value of x user
xhigh	max value of x user
xtic	large tic increment user
yy	position user or inches
nmstc	number of small tics
lint	label interval (large tics)
xunit	divisor for axis label
ipos	-1 bottom, 0 none, +1 top of label
csize	character size inches
frmt	axis format char*20

Valid qualifiers are:

/[NO]USER determines units of y position. Default is /USER. If /NOUSER units are inches from the ORIGIN. (see the ORIGIN command)

Appendix B Sec7.11.7 %YAXIS/qualifier,ylow,yhigh,ytic,x[,nmstc][,lint] [,yunit][,ipos][,csize][,frmt]

This command draws an y-axis and redefines scaling for the y direction. The arguments ylow, yhigh, ytic and x should not be omitted. See the %RANGE command to get default values for axis limits and increments. If you have used %RANGE, then you can use PPL$RANGE_LOW, PPL$RANGE_HIGH, PPL$RANGE_INC for ylow, yhigh and ytic.

ylow	min value of y user
yhigh	max value of y user
ytic	large tic increment user
xx	position user or inches
nmstc	number of small tics
lint	label interval (large tics)
yunit	divisor for axis label
ipos	-1 left, 0 none, +1 right of label
csize	character size inches
frmt	axis format char*20

Valid qualifiers are:

/[NO]USER determines units of y position. Default is /USER. If /NOUSER units are inches from the ORIGIN. (see the ORIGIN command)

Example:

Here is a PPLUS command file which uses all the % routines described above. It can be found in the directory ppl$examples (PPL$EXAMPLES:CTD4.PPC), and can be executed in PPLUS to generate a plot.

```
c
c  PPLUS command file to plot EPIC CTD data demonstrating multiple axis
c  capability.
c
c  It plots  Pressure vs Temperature, Salinity, Sigma_t, Oxygen.
c
box,off
```

```
window,on
size,8,10.5
origin,,2.3
format/ctd,epic
axlint,1,1
c pltnme,ctd4.plt
c
c  First plot P vs T with T axis at top. Supress bottom x axis.
c
evar,t,p
rd,ppl$examples:ctd4
%opnplt
%range/nouser 'ppl$ymin(1)','ppl$ymax(1)',5
yfor,(i7)
yaxis,'ppl$range_high','ppl$range_low','ppl$range_inc'
title

axlabp,1
axset,,0
plot
c
c  Plot P vs Salinity with S axis at top above T axis.
c
evar/next sal,p
rd
set ypos 'ppl$ylen' + .7
%range/nouser 'ppl$xmin(1)','ppl$xmax(1)',4
%xaxis/nouser,'ppl$range_low','ppl$range_high','ppl$range_inc',-
'ypos',,,,+1
%pltlin,1
c
c  Plot P vs Sigma_t with S_t axis at bottom
c
evar/next sig,p
rd
set ypos 0.
%range/nouser 'ppl$xmin(2)','ppl$xmax(2)',4
%xaxis/nouser,'ppl$range_low','ppl$range_high','ppl$range_inc',-
'ypos',,,,-1
%pltlin 2
c
c  Plot P vs Oxygen with O axis at bottom below S_t axis.
c
evar/next ox,p
rd
set ypos 'YPOS' - .7
%range/nouser 'ppl$xmin(3)','ppl$xmax(3)',4
%xaxis/nouser,'ppl$range_low','ppl$range_high','ppl$range_inc',-
'ypos',,,,-1
%pltlin 3
c
c  Now use PPLUS EPIC symbols in moveable labels for graph titles
c
set ypost 'ppl$ylen' + 1.9
%label/nouser 0,'ypost',-1,0.,.16,'ppl$epic_latitude1'
'ppl$epic_longitude1'
set ypos 'ypost' + .3
%label/nouser 0,'ypos',-1,0.,.16,'ppl$epic_cast1'
'ppl$epic_date1'
%clsplt
```

Appendix B Sec8 PLOT5, PPLUS DIFFERENCES

PPLUS is a greatly enhanced replacement to PLOT5. Most PLOT5 syntax and commands are identical to PPLUS usage. However, there are the following differences and incompatabilities.

RDCOM command has been replaced by the @ command.

The LEV command replaces the LEVEL and CLINE commands.

In format statements and labels single quotes (') must be replaced by two single quotes ('). The same applies to double quotes ("). See the chapter on labels.

The LIMITS command is enhanced.

IF / ELSE / ENDIF and WHILE / ENDW logic are available in command files. The INC and DEC commands are available to increment and decrement symbols.

The TXLINT, TXLABP, TXLSZE, TXNMTC and TXTYPE commands should be used instead of using the corresponding arguments in the TAXIS command.

The TIME command should be used instead of the TMIN, TMAX and TSTART commands.

NOTE : The following commands are not supported in this and future versions of PPLUS:

TMIN, TMAX and TSTART

LEVEL and CLINE

RWDSEQ, READSEQ and SKPSEQ

TAXIS will not support the obsolete arguments.

Appendix B Sec9 COMMAND DESCRIPTION

Appendix B Sec9.1 @file_name/qualifier arg1 arg2 arg3 ...

Reads commands from the file file_name until an EOF, blank line, a RETURN command is executed or the file ends, then reverts to the previous command level for input. Default device is SY:. Default extension is '.PPC'. The current command file name is placed in global symbol PPL$COMMAND_FILE.

PPLUS can be started with a command file specified by typing $PPL file_name, where file_name is the command file name. PPLUS will produce no screen output if called from a BATCH file. PPLUS will terminate and not pass control back to the SYS$INPUT file.

The arguments may be any legal string. The arguments arg1,arg2,etc are SET to the local symbols P1, P2, etc. For example:

@command_file your_file "A label" "PLTYPE 2"

The local symbols will be:

P1 = your_file

P2 = A label

P3 = PLTYPE 2

These symbols can then be substituted into the command file.

Qualifiers are (default in parenthesis):

/[NO]ECHO Controls echoing to the file echo.dat during execution. (NOECHO)

/[NO]DEBUG Sets DEBUG mode during execution. In debug mode the commands are written to the echo file after symbol substitution has occurred. (NODEBUG)

/[NO]QUIET Turns off messages to the terminal. (NOQUIET)

/[NO]LOG Echos commands to terminal. (NOLOG)

/[NO]LATCH Causes the current qualifiers to be the new default for all command levels. (NOLATCH)

Appendix B Sec9.2 AUTO,ON/OFF

Turns on and off the automatic copying of plots while at a TEK terminal. Default=OFF

Appendix B Sec9.3 AUTOLAB,ON/OFF

ON (default for BIBO and EPIC data) to get graph labels from data file headers. OFF (default for other data formats) for manual entry of graph labels. Default=OFF

Appendix B Sec9.4 AXATIC,ATICX,ATICY

Sets the number of large tics in auto mode for X and Y axes. Default=5

Appendix B Sec9.5 AXLABP,LABX,LABY

Sets the numeric and character label position for X and Y axes. -1=bottom/left of plot, 0=no label, +1=top/right of plot. Default=-1

Appendix B Sec9.6 AXLEN,XLEN,YLEN

Sets the X and Y axes length in inches. XLEN is also used as the length in inches of the time axis. Default=5.5,4.0 The values of xlen and ylen are placed in global symbols PPL$XLEN and PPL$YLEN.

Appendix B Sec9.7 AXLINT,LINTX,LINTY

Sets the label interval for X and Y axes. Labels are only drawn for large tics. Default=2, i.e. every other large tic.

Appendix B Sec9.8 AXLSZE,HGTX,HGTY

Sets the label height for X and Y axes in inches. Default=0.10 If HGTX or HGTY is negative the numeric axis labels are multiplied by -1 before plotting.

Appendix B Sec9.9 AXNMTC,NMTCX,NMTCY

Sets the number of small tics between large tics for X and Y axes. Default=0

Appendix B Sec9.10 AXNSIG,NSIGX,NSIGY

Sets the number of significant digits in labels for auto labelling. Default=2

Appendix B Sec9.11 AXSET,TOP,BOT,LEFT,RIGHT

Sets the flags controlling the plotting of the four axes. If =1 axis is ON, =0 axis is OFF. The default for all axes is ON.

Appendix B Sec9.12 AXTYPE,TYPEX,TYPEY

Sets the axis type for X and Y axes. 1 - normal, 2 - log, 3 - inv-log. Type 3 axis draws the top/right axis inverse and the bottom/left normal. Default=1

Appendix B Sec9.13 BAUD,IB

Sets baud rate. Null entry not allowed.

B= Baud rate default=110

Appendix B Sec9.14 BOX,ON/OFF

Turns on and off the box that is drawn around the entire plotting region. Default is ON.

Appendix B Sec9.15 C

Comment. This command can be used to comment your @ files. No action is done when this command is processed. The C must be followed by at least one blank space.

Appendix B Sec9.16 CLSPLT

Closes the metacode file. Not to be confused with %CLSPLT, which is documented in the Advanced Commands Chapter.

Appendix B Sec9.17 CONPRE,prefix

Sets a prefix string for the numeric contour labels of up to 10 characters. For example, CONPRE,@P2@TR will give labels using pen 2 and triplex roman font. Default = spaces.

Appendix B Sec9.18 CONPST,postfix

As CONPRE but sets up to 10 characters following the contour numeric label. For example, CONPST,cm/sec will give contour labels like "10 cm/sec". Default = spaces.

Appendix B Sec9.19 CONSET,HGT,NSIG,NARC,DASHLN,SPACLN,CAY,NRNG,DSLAB

Sets parameters for contouring and placing random data on a grid. Must be issued before the RD command.

HGT = height of contour labels. Default=.08 inches

NSIG = no. of significant digits in contour labels. Default=2

NARC = number of line segments to use to connect contour points. Default=1

DASHLN = dash length of dashes mode. Default=.04 inches

SPACLN = space length of dashes mode. Default=.04 inches

CAY = is the interpolation scheme. If CAY=0.0, Laplacian interpolation is used. The resulting surface tends to have rather sharp peaks and dips at the data points (like a tent with poles pushed up into it). There is no chance of spurious peaks appearing. As CAY is increased, Spline interpolation predominates over the Laplacian, and the surface passes through the data points more smoothly. The possibility of spurious peaks increases with CAY. CAY= infinity is pure Spline interpolation. An over relaxation process in used to perform the interpolation. A value of CAY=5.0 (the default) often gives a good surface.

NRNG = Any grid points farther than NRNG away from the nearest data point will be set to "undefined" (1.0E35). Default=5

DSLAB= nominal distance between labels on a contour line. Default = 5.0 inches

CONTOUR/qualifier,vcomp,label

Does a contour plot of data in buffer. Label will replace that in the current main label buffer. Label is optional. If either axis is log that index must be equally spaced in log-space (i.e. 10**(xmin+dx)). Contour does not take the log of the coordinate. The contour lines will be plotted with the pen selected for line 1. The label cannot begin with a numeric character, i.e., 95W. You can plot a number by specifying a font, e.g., @SR100 meters.

Vcomp indicates which vector component to contour. Default is 1. Vcomp is to be used when a vector field has been read in. See the VECSET and VECTOR commands.

Valid qualifiers are:

/[NO]WAIT Controls whether PPLUS pauses after plot completion. Pause is signaled by a tone and terminated by typing a character. If an <ESC> is typed PPLUS will return from the current command level to the lowest command level. Default = WAIT.

/[NO]OVERLAY Controls whether PPLUS overlays the plot on the preceeding plot. The default is /NOOVERLAY which causes the plot to be a new plot. The axes and their labels are not redrawn. Moveable labels (LABS command) will redraw.

Appendix B Sec9.20 CROSS,ICODE

Turns on and off the drawing of a solid line through(0,0) on a plot. Optionally can draw vertical and horizontal lines. Draws line through (XOFF,YOFF) when either TRANSXY or LINE command is used to apply a transformation to the data.

ICODE = 0 cross off
= 1 draw through (0,0) (default)
= 2 horizontal line through each YOFF
= 3 vertical line through each XOFF
= 4 horizontal and vertical through each XOFF, YOFF

Appendix B Sec9.21 DATPT,type,mark

Controls the drawing of marks on a contour plot along the x and/or y axis on a grid at the points where the raw ungridded X,Y,Z triplets are located.

type = 0 no points drawn (default)
= 1 points drawn along the x axis
= 2 points drawn along the y axis
= 3 points drawn at each raw input value

mark = 0 use the default mark (default)
= other use the specified mark to denote the location.

The default mark is down arrow for x axis, left arrow for y axis, and pluses for type=3. (also see MARKH)

Appendix B Sec9.22 DEBUG on/off

Turns on and off the debugging mode. In debug mode theinput lines are echoed to the ECHO.DAT file after symbol substitution. Default = off.

Appendix B Sec9.23 DEC symbol

Decrements the value stored in symbol by one. If symbol does not exist it is created and given a value of zero.

Appendix B Sec9.24 DELETE symbol

Deletes "symbol" from the symbol table.

Appendix B Sec9.25 DFLTFNT,font

Sets the default font used for all labelling. PPLUS initially uses Simplex Roman (SR) as the default font. Fonts are still selectable using the font command @xx, where xx is the two letter font code. NOTE: This command also replaces the string set by the CONPRE command with the selected font. The default font is not saved with MULTPLT.

This command changes the environment and can only be changed back with another DFLTFNT command or using the @CL command.

font = the new default font (no default)

Appendix B Sec9.26 DIR,arg

Prints a listing of files with names or extensions that match "arg".

Appendix B Sec9.27 ECHO,on/off

Turns on/off echoing of PPLUS commands in the echo file ECHO. Default is ON. ECHO is a logical that can be defined prior to entering PPLUS (e.g., DEFINE ECHO echo_file.echo). Default is for echoing to go into the file ECHO.DAT.

Appendix B Sec9.28 ENGLISH

Sets the internal conversion factors in COMPLOT to inches. This is the default condition. (see the METRIC command)

Appendix B Sec9.29 ENTER

Allows the input of X,Y pairs from the terminal. PPLUS prompts the user with 'enter>'. Type END to stop.

Appendix B Sec9.30 EVAR/qualifier,x-var,y-var

Specifies which EPIC variables are to be plotted as x and y when FORMAT,EPIC command has been given. The EPIC/pointer file is named with the RD command, and each call to RD results in reading another EPIC data file as indicated by the EPIC/pointer file. PPLUS can extract axis labels and a plot title from the data file headers. Use FORMAT/CTD,EPIC to tell PPLUS that EPIC CTD data is being read. Use FORMAT,EPIC to tell PPLUS that EPIC time series data is being read. See FORMAT command description for all the EPIC defaults.

x-var = Variable to be plotted as x
y-var = Variable to be plotted as y

EVAR ? displays a list of variables possible for x-var and y-var.

Examples of variables are TIM (time), U (zonal velocity), V (meridional velocity), etc. If you want to plot x=time and y=zonal velocity, the command would be EVAR,TIM,U. If the variable you want to plot is not in this list, you can specify the column number of the variable in the EPIC data file. For example, EPIC current meter data files generally have variables DATE,TIME,U,V,SPEED,DIRECTION. To plot x=time and y=speed, the command would be EVAR,TIM,5. If the x variable is specified by column number, the EVAR argument list must be enclosed in double quotes, (e.g., EVAR,"3,4" will plot the variable in column 3 as x and the variable in column 4 as y).

EVAR (without arguments) will yield a plot with x=date/time and y=the first variable following date/time on the data file for time series data. For CTD data, EVAR (without arguments) will yield a plot with x=variable in column 2 and y=variable in column 1 (usually pressure).

Valid Qualifiers are:

/[NO]OFFSET For time series data. Controls whether PPLUS offsets the time word so that data points are plotted in the center of each time interval. The default is OFFSET, which is appropriate for most EPIC time series. (EPIC time words represent the start of the time interval in

most cases, such as average data.) Use /NOOFFSET to force PPLUS to plot data points at the start of each time interval (e.g., this would be appropriate for subsampled data). Default is OFFSET.

/[NO]TIME For time series data. Controls whether PPLUS reads the time word from the time series data file. The default is /NOTIME, which means that the data is evenly spaced in time, making it unnecessary to read the time words. Use /TIME to make PPLUS read the time word for data which is unevenly spaced in time. Default is /NOTIME (unless dt is negative, in which case the default is /TIME).

/[NO]NEXT /NEXT indicates that the next variable is to be read from the same data file. When /NEXT is used, no new data file name will be read from the EPIC file. The variables indicated by the EVAR command will be read from the last data file. This option permits overplotting several variables from the same data file, and can be used with the commands described in the ADVANCED COMMANDS chapter to produce a plot with multiple axes. When /NEXT is used, both x and y variables must be specified with the EVAR command. Default is /NONEXT.

The above qualifiers will also work with the VARS command when EPIC data is being read.

EXIT Causes all output buffers to be flushed and exits the program.

FORMAT/qualifier,frmt

Allows the input of a user supplied format for formatted sequential data files. Null entry is not allowed. The current format is in global symbol PPL$FORMAT.

frmt = a format default=(3F10.2)

FREE	for free form
DSF	for DSF files
BIBO	for DSF files without a PPLUS header
EPIC	for EPIC time series data
UNF	for UNFORMATTED files.

Valid qualifier (for EPIC data only) is:

/[NO]CTD Controls whether EPIC data is read as time series data or as CTD data. If the data is EPIC CTD data, then the /CTD switch must be used. Default is /NOCTD.

Appendix B Sec9.31 GET,file_name

Restores options to those in effect at the time SAVE,file_name was called. file_name must be specified.

Appendix B Sec9.32 GRID[,LINEAR]

If the argument LINEAR is omitted (default), normal gridding is used. Otherwise, if LINEAR is included, gridding is done by linear interpolation with the following restrictions on the data:

1. Data must be on a grid. The grid may have irregular spacing.

2. There cannot be gaps in the middle of the grid. Every grid point in the middle of the grid must be specified.

3. The grid may have ragged edges.

Must be issued before the RD command. Note that if the grid is coarser than the data, it is possible that some of the data will not be used in the gridding process. It is best to make the grid as fine as or finer than the data rather than coarser.

Appendix B Sec9.33 HELP,arg

Give access to the VMS help files on topic "arg".

Appendix B Sec9.34 HLABS,n,height

Sets the height in inches of the nth moveable label. The height is reset to the default (specified by the LABSET command) by omitting the height value or clearing the labels with a LABS command. (also see LABS, RLABS, LLABS, LABSET)

Appendix B Sec9.35 HLP,arg

Gives help on the PPLUS topic "arg".

Appendix B Sec9.36 F expression THEN

The first element of a BLOCK IF statement; the other two elements are ELSE and ENDIF. ELSE and ENDIF are not valid in any other context. expression = argument operator argument

argument = symbol name, number or a string enclosed by quotes

operator = .EQ., .NE., .LT., .GT., .LE. or .GE.

The symbol name can be undefined and its value is then "" (i.e., null string).

Appendix B Sec9.37 INC sym

Increments the value stored in the symbol sym by one. If sym does not exist it is created and given a value of one.

Appendix B Sec9.38 LABS/qualifier,n,X,Y,JST,label

Defines the nth movable label for all plots. When plotting is done, the cross hairs will come on if no X and Y position has been specified. Typing a C will center the label at the cross hairs or typing a R will position the label to the right of the cross hairs. By typing L or F then repositioning the cross hairs and then typing another character a line will be drawn from the first point to the second and the label will be drawn at the second point (if F was specified an arrow will be drawn). Any character other than L, F, R or C will cause the the label to be drawn at the cross hairs. Null entries are not allowed for n or label. A comment will be inserted into the ECHO.DAT file giving the coordinates when cross hairs are used. If n is omitted LABS is reset and all moveable labels are cleared. (also see LABSET, HLABS, RLABS, LLABS)

n =	label number (up to 25 allowed)
X =	X position of label in user units (optional)
Y =	Y position of label in user units (must exist if X is present)
JST =	justification of label. -1 left (default), 0 center, +1 right
label =	any SYMBEL compatible string

/[NO]USER determines units of x and y positions. Default is /USER. If /NOUSER units are inches from the ORIGIN. (see the ORIGIN command)

NOTE: Units specified by the /user qualifier are also used in the LLABS command. If your terminal does not have cross hairs, you must specify X and Y.

Appendix B Sec9.39 LABSET,HLAB1,HXLAB,HYLAB,HLABS

Sets character heights for labels. (also see LABS, RLABS, LLABS)

HLAB1 = main label default=.16 inches
HXLAB = x - label default=.12 inches
HYLAB = y - label default=.12 inches
HLABS = movable labels default=.12 inches

Appendix B Sec9.40 LEV,arg,arg,arg ...

Sets the contour levels, the contour line type, the contour line label characteristics and lets the user edit (insert/delete) levels. Any duplicate levels will be deleted, however, each LEV command edits the existing levels and unless requested the levels are not cleared. Maximum number of levels is 500.

arg = () clear levels, number of automatic levels to 10.

arg = (min,max,inc,idig) specifies the contour levels and abel type

min = starting value for levels creation
max = ending value for levels creation (if omitted
only the starting level will be created)

inc = increment used to create levels. (if omitted
only the starting and ending levels will be
created, if 0 the starting and ending levels
are deleted)

idig = 0 through 9 Number of digits after the
decimal point in the label
= -1 contour label plotted as an integer
= -3 no contour label will be drawn

arg = type(min,max,inc,ipen) sets the contour lines specified to "type"

type = DASH sets the line type to dash
= DARK sets the line type to dark (heavy)
= DEL deletes the indicated levels.
= LINE sets the line type to line (normal)
= PEN sets the pen used for a contour line to
"ipen". ipen=0 to use default pen.

For example, "LEV,(),(9,20,1,-1),DASH(8,20,2)" will clear the previous levels and create contours at every integral value from 9 to 20 with the labels drawn as integers, all even valued contours from 8 to 20 will be drawn with dashed lines.

Appendix B Sec9.41 LIMITS,value,comparison,flag

This command sets the testing value and type of test for bad data points. X, y and z are checked and the point will not be plotted if the test is true.

value = test value for the test

comparison = XLE test for x .le. value, default off, 0.0
XEQ test for x .eq. value, default off, 0.0
XGE test for x .ge. value, default on, 1.E35
YLE test for y .le. value, default off, 0.0
YEQ test for y .eq. value, default off, 0.0
YGE test for y .ge. value, default on, 1.E35
ZLE test for z .le. value, default off, 0.0
ZEQ test for z .eq. value, default off, 0.0
ZGE test for z .ge. value, default on, 1.E35

flag = OFF the test is disabled, otherwise the test is enabled.

If your are reading data to be contoured with ZGRID, the limits are checked only after interpolation. If you arE using GRID,LINEAR, limits are checked before and after interpolation.

Appendix B Sec9.42 LINE,n,MARK,TYPE,XOFF,YOFF,DN1,UP1,DN2,UP2

Sets the characteristics for each of the 50 possible X-Y plot lines.

n = line number

MARK = data mark (see list at end of manual, e.g. 1 for x, 3 for +)

TYPE = type of line
0 - line connecting points and no mark at each point
1 - line connecting points and mark at each data point
2 - mark end points only
3 - only mark (no line)
4 - dashes
5 - dashes with mark at end points

XOFF = X offset default=0.0

YOFF = Y offset default=0.0

DN,UP = dash characteristics in inches.

Default TYPE=0 for n=1, TYPE=4 otherwise.

Appendix B Sec9.43 LINFIT,n,XIMIN,XIMAX,XOMIN,XOMAX

A linear least squares fit is performed on the data in line n and the resulting fitting line is placed in the next available line buffer.

Example:

```
RD,data.fil   LINFIT,1
```

will place the fitting line from the regression of line 1 into buffer 2.

n = line number (no default)

XIMIN = min x value for the regression domain

XIMAX = max x value for the regression domain

XOMIN = min x value for the fitting line (default=XIMIN)

XOMAX = max x value for the fitting line (default=XOMIN)

XIMIN and XIMAX default to the minimum and the maximum of the data. XOMIN and XOMAX default to XIMIN and XIMAX, respectively. An alternate form for the command may be used when TAXIS is ON and TSTART has been set. It is:

LINFIT,n,TIMIN,TIMAX,TOMIN,TOMAX

Where the arguments are the beginning and ending times in Woods Hole format WYYMMDDHHMM, i.e., W8101121800 is 12-JAN-1981 18:00. The arguments have the same meanings and defaults as above.

The following global symbols are defined by LINFIT:

PPL$LF_R2 = regression coefficient squared

PPL$LF_A = constant for fit (y = a + b*x)

PPL$LF_A_STDEV = standard error of A

PPL$LF_B = constant for fit

PPL$LF_B_STDEV = standard error of B

PPL$LF_VAR = total variance

PPL$LF_RES_VAR = residual variance after fit

Appendix B Sec9.44 LIST,IMIN,IMAX,JMIN,JMAX,VCOMP,arg

List on the terminal the appropriate information. Null entry is not allowed if arg is not DATA. IMIN, IMAX, JMIN, JMAX only valid if arg=DATA. Defaults are to print the total plot buffer.

IMIN= min I for CONTOUR , start pt for X-Y

IMAX= max I for CONTOUR , stop pt for X-Y

JMIN= min J for CONTOUR , start line for X-Y

JMAX= max J for CONTOUR , stop line for X-Y

VCOMP= vector component to be listed (VECTOR command)

arg= LEVELS contour levels and weights
CONSET contour information
DATA data currently in buffer
DATPT contour data location before gridding
LABELS prints the labels at the terminal
LABSET LABSET parameters
LINES current LINE and PEN values
LIMITS the current values set/reset by the limits command
PLOT gives plot information and plot file name
READ sequential read information
STATS min and max plus sizes of last read
TAXIS T-axis attributes
TICS Tic sizes and options
TRANSXY X and Y transform values
VECTOR Vector plotting attributes (VECTOR command)

XAXIS X-axis attributes
YAXIS Y-axis attributes

Appendix B Sec9.45 LISTSYM

Lists the symbols currently defined.

Appendix B Sec9.46 LLABS,n,X,Y,TYPE

Defines the starting position in user units for a line associated with the moveable labels. The end of the line is determined from the LABS command. This command has no effect if the label is to be positioned with the cross-hairs. If the command is issued without coordinates the TYPE is set to none. Fancy has an arrow head at the starting position. (also see LABS, RLABS, HLABS, LABSET)

n = label number less than 11

X = X position of line in user units

Y = Y position of line in user units

TYPE = line type. 0 no line, 1 normal line, 2 fancy line

NOTE: Units of x and y positions are determined by the /USER qualifier in the LABS command.

Appendix B Sec9.47 MARKH,n,SIZE

Sets the mark size used for plotting line number n. The mark size for line 1 is used for the marks in the DATPT command (contouring).

n = line number (no default)

SIZE = size of mark in inches (default= 0.08)

Appendix B Sec9.48 METRIC

Sets the internal conversion factors in COMPLOT to millimeters. Default condition is inches.

MULTPLT,NX,NY

This command allows the user to plot several plots together. The individual plots are arranged in rows and columns. The X axis length of each plot in the same column and the Y axis length of each plot in the same row are identical. The axis lengths are specified in rows and columns. The spacings between the rows and columns are also user controlled. If the spacing is zero the plots are placed together without axis labels if appropriate. There are prompts for all additional information needed.

NX = number of columns
NY = number of rows

The prompts will be:

```
ENTER XLEN FOR COLS 1,2,...,NX
multplt>
ENTER YLEN FOR ROWS 1,2,...,NY
multplt>
ENTER PLOT SPACINGS
LEFT BNDRY TO COL1, COL1 TO COL2,ETC...
multplt>
ROW1 TO ROW2,...,ROW NY TO BOTTOM
multplt>
```

Axis length and ORIGIN are reset after plotting is finished.

Appendix B Sec9.49 NLINES

Resets the the input buffer so that the next data line read will be line 1. The input buffer is normally reset when a plot is made.

Appendix B Sec9.50 ORIGIN,XORG,YORG

Sets the distance the lower left hand corner of the plotting area is from the lower left corner of the box. The values of xorg and yorg are placed in the global symbols PPL$XORG and PPL$YORG.

XORG = x-distance (in) default=1.4
YORG = y-distance (in) default=1.2

Appendix B Sec9.51 PEN,n,ipen

Sets the pen to be used for line n. ipen should be in the range 1-6, subject to the limitations of the plotting device. On the VERSATEC, pen 2 is thicker than pen 1, pen 3 is thicker than pen 4, etc. The pen selected for line 1 will be used to draw the contour lines. (also see LEV)

n = line number. If n=0 sets the pen used to plot the axes and labels.
ipen = pen number. Default=1

Appendix B Sec9.52 PLOT/qualifiers,label

Does an X-Y plot of data in the plot buffer (all lines). The plot label "label" is optional. The plot label can be blanked with the TITLE command. If either x-axis or y-axis is log PLOT will take the logarithm of the appropriate coordinate as it is plotted. This will not affect the data buffer.

Valid qualifiers are:

/[NO]WAIT Controls whether PPLUS pauses after plot completion. Pause is signaled by a tone and terminated by typing a character. If an <ESC> is typed PPLUS will return from the current command level to the lowest command level. Default = WAIT.

/[NO]OVERLAY Controls whether PPLUS overlays the plot on the preceeding plot. The default is /NOOVERLAY which causes the plot to be a new plot. The axes and their labels are not redrawn. Moveable labels (LABS command) will redraw.

Appendix B Sec9.53 PLOTV/qualifiers,VANG,INC,label

Does a stick plot for U,V pairs stored in X,Y, respectively. May be used with or without TAXIS option ON.

VANG = rotation angle of vectors default=0.0
INC = plots every inc vector (subsamples)
label = plot label

Valid qualifiers are:

/[NO]WAIT Controls whether PPLUS pauses after plot completion. Pause is signaled by a tone and terminated by typing a character. If an <ESC> is typed PPLUS will return from the current command level to the lowest command level. Default = WAIT.

/[NO]OVERLAY Controls whether PPLUS overlays the plot on the preceeding plot. The default is /NOOVERLAY which causes the plot to be a new plot. The axes and their labels are not redrawn. Moveable labels (LABS command) will redraw.

Appendix B Sec9.54 PLOTUV/qualifiers,VANG,INC,label

Similar to PLOTV except U and V are in alternate pairs, where X1= count, Y1= U component, X2= count, Y2= V component, etc. NLINES must be set to an even number and first series read will be U second V etc.

Valid qualifiers are:

/[NO]WAIT Controls whether PPLUS pauses after plot completion. Pause is signaled by a tone and terminated by typing a character. If an <ESC> is typed PPLUS will return from the current command level to the lowest command level. Default = WAIT.

/[NO]OVERLAY Controls whether PPLUS overlays the plot on the preceeding plot. The default is /NOOVERLAY which causes the plot to be a new plot. The axes and their labels are not redrawn. Moveable labels (LABS command) will redraw.

Appendix B Sec9.55 PLTNME,fname

Specifies the file name to be used for plots. File name is available in the global symbol PPL$PLTNME. fname = the file name (default = ZETA.PLT)

Appendix B Sec9.56 PLTYPE,ICODE

Sets plotting medium. Null entry is not allowed. The binary file is converted into device specific code using a post processor. The plot file name can be specified using the PLTNME command.

ICODE = device code for plotting
-2 = HP and TEK
-1 = HP
0 = Binary file
1 = TEK
2 = TEK and Binary file
3 = GKS (valid on MicroVAX only)
4 = GKS and Binary file (valid on MicroVAX only)

default=1

Appendix B Sec9.57 RD/qualifier,NX,NY,TYPE,n,file_name

Read formatted or unformatted data from a sequential file according to FORMAT and VARS or EVARS. The input file name is available in the global symbol PPL$INPUT_FILE.

NX and NY define the grid on which data will be plotted. If X,Y,Z triplets are being read the grid can be coarser or finer than the input data. Thus, when reading triplets NX, NY of 50, 21 indicates the grid used for contouring will be 50 x 21 and not that the input data is on this grid. When the input data are values of Z only the input grid and the plotting grid must be identical. Maximum number of points for a single read is 100,000 pairs, 200,000 grid points or 50,000 triplets. Default number of points read is the remaining buffer space. File_name may be omitted if previously defined. Null entries are not allowed.

NX = no. of columns on the plotting grid for contouring or
no. of points to read if not contouring. See NY for explanation.

NY = no. of rows if data is on a grid for contouring. Omitted otherwise.

The meaning of NX and NY change depending on whether you're reading data for contouring or not. If you're reading contour data NX is the number of columns and NY is the number of rows.

If the data is not contour data NX is the number of points to be read and NY is not required. The default for NX is the space remaining in the buffer. Reading will stop automatically at the EOF without any error.

TYPE = method by which grid data is to be read (contour data only)
0 by rows (1st subscript varies fastest)
1 by columns (2nd subscript varies fastest)
N = number of data sets to be read (on same file).
file_name = file name. Default device is SY:.

If the file name is explicitly given the file will be read after rewinding the file. If the file name is not given no rewind takes place.

If the data is EPIC, the file name given with the RD command is the name of the EPIC/pointer file for the data file. Otherwise, the file name is the name of the data file itself

Valid qualifier (use only with VECTOR, VECSET, VECKEY commands):

/[NO]VECTOR /VECTOR reads the second component grid using the old xmin,xmax,ymin,ymax. This is done after the first vector component has been read in the usual fashion. See the VECTOR command Default is /NOVECTOR.

If you are reading triplets PPLUS prompts for total number of points to be read in with 'rd>'. If you are readingtriplets or grid data PPLUS will also prompt for xmin,xmax,ymin,ymax. (limits)

Appendix B Sec9.58 RESET

Uses the logical PPL$RESET as the input file to the GET command.

Appendix B Sec9.59 RETURN

Return from current command level to the previous command level. If executed at the top level PPLUS will exit.

Appendix B Sec9.60 RLABS,n,ANG

Specifies the angle to rotate the moveable labels. (The labels defined by the LABS command.)

n = number of the label (no default)
ANG = angle in degrees. Default = 0.0

Appendix B Sec9.61 ROTATE,ON/OFF

Rotates the plot 90 degrees on the screen and plotter.

Default = OFF

Appendix B Sec9.62 RWD,file_name

Rewinds the current data file. File_name may be omitted if previously defined. Files are also rewound by explicitly including the file name in the SKP and RD commands. Rewinds the EPIC pointer file. The input file name is available in the global symbol PPL$INPUT_FILE.

If the data is EPIC, the file name given with the RWD command is the name of the EPIC/pointer file for time series data. Otherwise, the file name is the name of the data file itself

Appendix B Sec9.63 SAVE,file_name

Saves the options currently in effect on file file_name in a binary format. File_name must be specified.

Appendix B Sec9.64 SET sym arg

Creates/modifies the symbol sym and sets it to arg. The argument arg can be either a legal character string, a simple arithmetic expression, or a special function. A simple arithmetic expression is of the form num1 op num2, where op is +, -, * or / (addition, subtraction, multiplication or division) and num1 and num2 are numbers. The numeric values must be separated from the operator op by spaces. The string will be used exactly as it appears if enclosed by double quotes ("). For example:

SET XPOS 4.4 + 2 results in XPOS = 6.200E00
SET A_LABEL "4.4 + 2" results in A_LABEL = 4.4 + 2

The special functions manipulate and reformat character strings. They are:

$EDIT(symbol,argument)
$EXTRACT(start,length,symbol)
$INTEGER(symbol)
$LENGTH(symbol)
$LOCATE(substring,symbol)
$ELEMENT(position,delimiter,symbol)

The general format is SET sym $function(arg1, arg2,...). These functions are described in the SPECIAL FUNCTIONS section. (p. 507)

Appendix B Sec9.65 SHOW symbol

Prints the current value of "symbol".

Appendix B Sec9.66 SIZE,width,height

Sets total plotting size in inches of the plotting region. Null entries are not allowed. The width and height should be about 2 and 1.5 inches greater than the respective axis lengths. The displacement specified by ORIGIN must be considered when values for SIZE and AXLEN are being chosen. The maximum allowed size for Versatec plots (to keep the plot on a single page) is 8 by 10.5. The values of width and height are placed in the global symbols PPL$WIDTH and PPL$HEIGHT.

width = plotting area total width (default = 7.5)
height = plotting area total height (default = 5.625)

Appendix B Sec9.67 SKP,n,file_name

Skip n sequential or unformatted records. File_name may be omitted if previously defined. If the file name is explicitly given the records will be skipped after rewinding the file. If the file name is not given no rewind takes place. The input file name is available in the global symbol PPL$INPUT_FILE.

If the data is EPIC, the file name given with the SKP command is the name of the EPIC/pointer file for time series data. Otherwise, the file name is the name of the data file itself.

Appendix B Sec9.68 SMOOTH,n

Does n laplacian smoothings on contour type data. Null entry is not allowed.

Appendix B Sec9.69 SPAWN

Creates a sub-process and passes control to this process. When finished with the spawned process type LOGOUT to return to PPLUS.

Appendix B Sec9.70 TAXIS/qualifier,DT,arg

Sets the time axis characteristics. The axis length is specified with AXLEN for this style axis. When TAXIS is turned on and BIBO or EPIC formatted data is read, the time series are automatically adjusted properly relative to TMIN. NOTE: DT and TSTART (set with the TIME command) are needed only when BIBO or EPIC data is not being used.

DT = sampling rate in minutes (default=1440 ,ie, daily)
arg = ON/OFF turns TAXIS option on and off (default=OFF)

/[NO]YAXIS if yaxis draw a vertical time axis in place of the yaxis. (NOYAXIS)

Appendix B Sec9.71 TEKNME[,fname]

Stores the Tektronix plot in file fname if specified. Terminal must have NOWRAP to dump the plot back to the screen with the TYPE command. The current Tektronix plot file name is available in global symbol PPL$TEKNME.

Appendix B Sec9.72 TICS,SMX,LGX,SMY,LGY,IX,IY

Sets the sizes in inches of the small and large tics on the X and Y axis. The tic style may also be set for both axes.

SMX = small X axis tic size default=0.125
LGX = large X axis tic size default=0.25
SMY = small Y axis tic size default=0.125
LGY = large Y axis tic size default=0.25
IX = 1 X tics on the inside
0 X tics on both sides
-1 X tics on the outside (default)
IY = 1 Y tics on the inside
0 Y tics on both sides
-1 Y tics on the outside (default)

Appendix B Sec9.73 TIME,TMIN,TMAX,TSTART

Specifies time axis limits and starting time of time series data. See TAXIS command for restrictions. (Default is auto-scaling for BIBO and EPIC formatted data)

Note: If you read time as a sequence number andspecify DT (set with the TAXIS command) and TSTART, then the TSTART time/date must correspond to a sequence number of 1.

TMIN and DT (see TAXIS command) must be specified before TSTART. TSTART must be re-entered whenever DT is changed.

TMIN = Start date/time of time axis (WHOI format = Wyymmddhhmm)
TMAX = End date/time of time axis
TSTART = Start time of time series data (optional)

Appendix B Sec9.74 TITLE,HLAB,label

Sets the main plot title to "label" without generating a plot. If "label" is omitted the main plot title is cleared. Optionally the size of the title can also be specified.

HLAB = the height of the title in inches. (default = .16 inches)

Appendix B Sec9.75 TKTYPE,TYPE

Sets the type of TEK terminal. Null entry is not allowed. Valid values are: 4010, 4014, 4107, 4115, 4051, 4052 and 4662.

TYPE = model no. of TEK terminal default=4010

Appendix B Sec9.76 TRANSXY,n,XFACT,XOFF,YFACT,YOFF

Lets you define a linear transformation for the X and Y variables in each line, i.e., XT(i)= XFACT*X(i) + XOFF. TRANSXY does not affect the data. The translation is only applied as the data is plotted.

n = line number (no default)
XFACT = multiplicative factor for X (default=1.0)
XOFF = offset for X (default=0.0)
YFACT = multiplicative factor for Y (default=1.0)
YOFF = offset for Y (default=0.0)

The transformation factors are available in the global symbols PPL$XFACT(n), PPL$XOFF(n), PPL$YFACT(n) and PPL$YOFF(n), where "n" is the line number. Initially only the first 10 lines will have these symbols defined.

If the value being scaled is time and TAXIS is on, XOFF or YOFF is in units of DT. Unless DT is changed with the TAXIS command, it will have the default value of 1 day.

Appendix B Sec9.77 TXLABP,n

Specifies time axis label position (-1 for below plot, 0 for no label, or +1 for above plot).

Appendix B Sec9.78 TXLINT,low_int,hi_int

Specifies which time axis tics will be labeled.

Low_int = labeling interval for lowest level of tics (e.g. mon on mon/yr axis)
Hi_int = labeling interval for highest level of tics (e.g. yr on mon/yr axis)

Appendix B Sec9.79 TXLSZE,ht

Specifies height of time axis labels (inches).

Appendix B Sec9.80 TXNMTC,n

Specifies number of small tics between large tics on time axis. If NMTCT is -1 the major divisions are denoted by large tics and the minor divisions by small tics, otherwise they are denoted by thick tics and large tics, respectively.

Appendix B Sec9.81 TXTYPE,type,style

Specifies type and style of time series axis.

type = DAYS
    style = HR (hour,day on 2 lines) (default)
        = HRDAY (on 1 line)
  = MON
    style = DAY (day,mon on 2 lines) (default)
        = DAYMON (day,mon on 1 line)
  = YR (default)
    style = MON1 (1-char month)
        = MON3 (3-char month) (default)
        = MONYR (month,yr on 1 line)

Appendix B Sec9.82 VARS,NGRP,A1,A2,A3,...,Ai

Defines the location of variables within a record of a sequential data file. If only a single variable is specified and it is either X or Y the other is automatically filled with the data point number. If only Z (gridded data) is given the program expects data to be grid points in one of two formats, by rows or by columns. If X, Y, and Z (triplets) are given the program uses ZGRID to put the data on a evenly spaced grid. See the chapters Getting Started and Data Formats for more information on VARS.

NGRP = no. of groups per record
Aj = 1,2, or 3 The position of Aj in VARS command indicates which variable
  is to be read as an x, y or z.
    1 = X variable
    2 = Y variable
    3 = Z variable
i = NVAR no. of variables per group. Default=VARS,1,1,2

(i.e. one group per record, first variable is X, second is Y). If left blank indicates a number not to be read, but a variable is present and expected by the FORMAT.

Appendix B Sec9.83 VECKEY/qualifier,x,y,ipos,format

VECKEY sets where the scaling key for the vectors is plotted. See VECTOR and VECSET commands.

x = x position of vector key
y = y position of vector key (default is no key at all)
ipos = relative position of key (not implemented)
format = format to draw the numeric part of the key default = (1pg10.3)

Valid qualifiers are:

/[NO]USER determines units of x and y positions. Default is /USER. If /NOUSER units are inches from the ORIGIN. (see the ORIGIN command)

Appendix B Sec9.84 VECSET,length,scale

VECSET sets the scaling for the vectors plotted. See the VECTOR and VECKEY commands.

length = length of standard vector in inches. this is also the length of the scale vector. Default is 0.5.

scale = length of standard vector in user units. This is also the length of the scale vector is user units. Default is the twice the mean length of the vectors.

Appendix B Sec9.85 VECTOR/qual,skipx,skipy,label

VECTOR draws a field of vectors from two component grids. See the VECKEY and VECSET commands.

skipx = plot every skipx column (default is 1)
skipy = plot every skipy row (default is 1)
label = title of plot

Valid qualifiers are:

/[NO]WAIT Controls whether PPLUS pauses after plot completion. Pause is signaled by a tone and terminated by typing a character. If an <ESC> is typed PPLUS will return from the current command level to the lowest command level. Default = WAIT.

/[NO]OVERLAY Controls whether PPLUS overlays the plot on the preceeding plot. The default is /NOOVERLAY which causes the plot to be a new plot. The axes and their labels are not redrawn. Moveable labels (LABS command) will redraw.

Appendix B Sec9.86 VELVCT,rlenfact,inc

Does a vector plot of u,v pairs located at x,y locations. This plot is done on a two dimensional field (compared to PLOTV and PLOTUV which are one dimensional). To use VELVCT the data must be stored as two lines. Line 1 containing u,v data pairs, and line 2 containing the corresponding x,y location pairs. The lines are loaded with data in the ordinary manner. Default length scaling is set to the minimum inches/user_unit along the x and y axis.

rlenfact = scaling factor for vector length (default = 1.0)
> 0 scale = rlenfact * inches/user_unit on x-axis
< 0 scale = rlenfact * inches/user_unit on y-axis
inc = plots every inc vectors (subsamples)

Example:

```
xaxis,0,4,1
yaxis,1,8,1
nlines,2
enter
2.2,3.3
5.0,6.0
1.3,2.0
3.0,0.0
0.5,7.3
1.3,4.4
1.1,4.2
end
enter
1,2
3,3
2,2
3,5
2,6
2,7
3,2
end
velvct,-.3,2
```

reads 7 x,y and u,v pairs storing them as lines then plots every other vector scaled .3 * inches/user_unit on y-axis.

Appendix B Sec9.87
VIEW/qualifiers,ZSCALE,IC,ZMIN,ZMAX,VCOMP,label

Does a 3 dimensional surface plot. Label is optional.

ZSCALE = scale of the z data default=(YMAX - YMIN)/
(ZMAX - ZMIN)
IC = 0 set Xscal = Yscale, =1 no effect. Default=0
ZMIN = set the base of the surface plot to ZMIN. Default:
use ZMIN from the data
ZMAX = set the top of the surface plot to ZMAX. Default:
use ZMAX from the data
VCOMP = Vector component to use for plotting (see the
VECTOR command). Default is 1.

Valid qualifiers are:

/[NO]WAIT Controls whether PPLUS pauses after plot completion. Pause is signaled by a tone and terminated by typing a character. If an <ESC> is typed PPLUS will return from the current command level to the lowest command level. Default = WAIT.

/[NO]OVERLAY Controls whether PPLUS overlays the plot on the preceeding plot. The default is /NOOVERLAY which causes the plot to be a new plot. The axes and their labels are not redrawn. Moveable labels (LABS command) will redraw.

Best results are normally obtained by using defaults. Using scales does not change the data buffer.

VPOINT,X,Y,Z

Sets the viewpoint coordinates for surface plotting. To create a surface plot use the VIEW command. The viewpoint coordinates are available in the global symbols PPL$VIEW_X, PPL$VIEW_Y and PPL$VIEW_Z. X, Y and Z form a right handed coordinate system with the Z axis up and Y axis into the page.

X = x coordinate of viewpoint
Y = y coordinate of viewpoint
Z = z coordinate of viewpoint

Appendix B Sec9.88 WHILE expression THEN

The first element of a WHILE statement the other element is ENDW. ENDW is not valid in any other context.

expression = argument operator argument
argument = symbol name, number or a string enclosed by quotes
operator = .EQ., .NE., .LT., .GT., .LE. or .GE.

The symbol name can be undefined and its value is then "" (i.e., null string).

Appendix B Sec9.89 WINDOW,ON/OFF

Windows the data to within the axes. Default=OFF

Appendix B Sec9.90 XAXIS,XLO,XHI,XTIC

Sets the x-axis characteristics. If TYPEX is not 1, then XLO and XHI must be the log of the minimum and maximum (must be integral values). XAXIS without arguments resets the auto scaling. Auto scaling does consider LIMITS and does not consider WINDOW,ON.

XLO = axis minimum (beginning of axis)
XHI = axis maximum (end of axis)
XTIC = dx distance between large tics

Appendix B Sec9.91 XFOR,frmt

Sets the format for the x axis label.

frmt = 0 or (a format) default=0 (auto label)

Starting with Ferret v6.0 there is an option to label with degrees, minutes and optionally, seconds, rather than the default of degrees.decimal_degrees:

XFOR (dm) for degrees,minutes
XFOR (dms) for degrees,minutes,seconds

To create an integer numeric label the format must begin as "(I" or "(i". A latitude or longitude axis can be created by specifying "LAT"), "LON"), "LONE") or "LONW") in the format, where the punctuation surrounding LAT, LATW, etc is two single quotatino marks. The single quotes are required because PPLUS symbol substitution will occur with 1 single quote. The hemisphere designation will be inserted. Longitude must be continuous across the dateline with west positive for "LON" or "LONW", i.e., 135 is 135W and 190 is 170E. For "LONE" longitude is continuous across the dateline with east positive, i.e., 135 is 135E and 190 is 170W.

Example:

```
yes? use coads_climatology
yes? SHADE/L=17/SET sst
yes? PPL XFOR (i5, ''LONW'')
yes? PPL shade

! To label a longitude axis with degrees, minutes use XFOR (DM)

yes? USE my_detailed_data.nc
yes? FILL/SET/x=120:123/y=20:22 var
yes? PPL XFOR (DM)
yes? PPL YFOR (DM)
yes? ppl fill

! or (DMS) may be used to label with seconds as well.
```

Appendix B Sec9.92 XLAB,label

Enters the x-axis label. Label is ignored if TAXIS is on.

Appendix B Sec9.93 YAXIS,YLO,YHI,YTIC

See XAXIS.

Appendix B Sec9.94 YFOR,frmt

See XFOR.

Font tables

Appendix B Sec9.95 YLAB,label

Enters the y-axis label.

Appendix B Sec10 FONT TABLES

Following are the Character and Symbol fontsavailable with PPLUS. Choose the font by its 2-letter code, e.g. PLOT/TITLE=@CITemperature for the title "Temperature" in complex itallic. See "Embedded String Commands" (512) in this appendix for use of the PPLUS fonts.

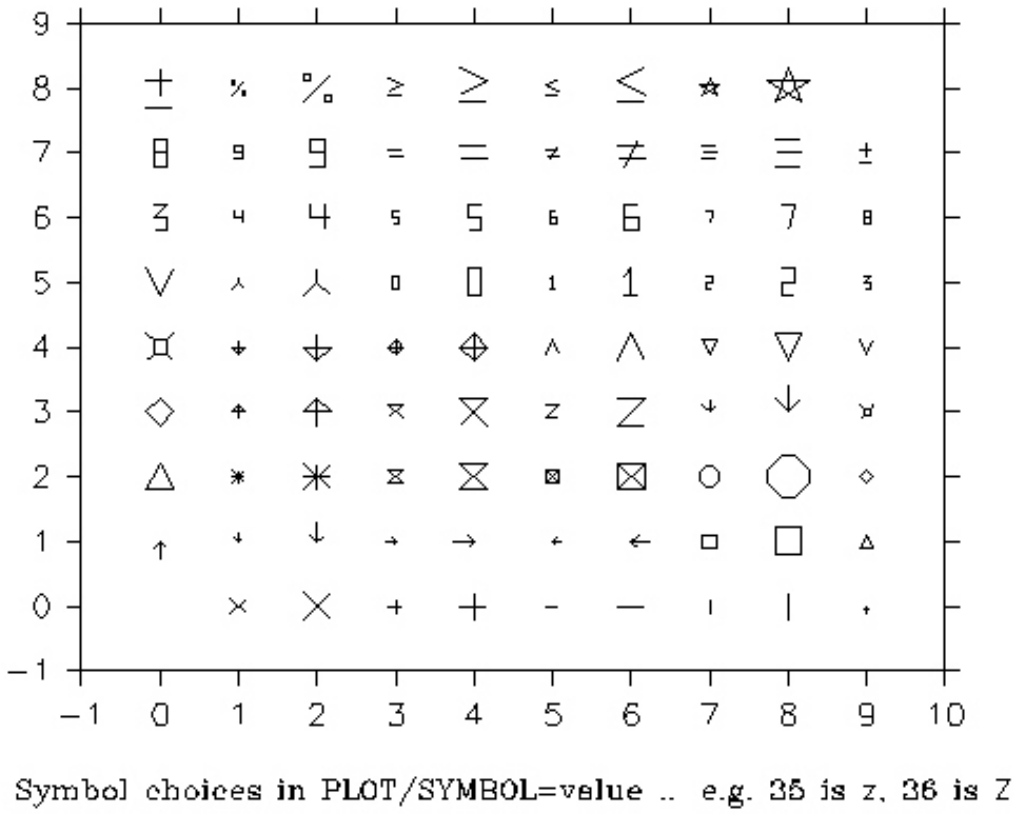
Tables showing the character fonts are linked to the web page:

http://ferret.pmel.noaa.gov/Ferret/Documentation/Users_Guide/pplus_char_fonts.html

Tables showing the symbol fonts are linked to the web page:

http://ferret.pmel.noaa.gov/Ferret/Documentation/Users_Guide/pplus_symbol_fonts.html

The symbols used for PLOT/SYMBOL= are shown below. For example, PLOT/symbol=22 yields a *, and PLOT/SYMBOL=35 yields a z.



Appendix B Fig 01

Appendix C: PLOTPLUS PLUS: Ferret Enhancements to PLOTPLUS

A User's Guide to the TMAP Modifications of the Plotplus Graphics Package

Jerry Davison

NOAA/PMEL/TMAP

April 1994

Note: this document also exists at
http://ferret.pmel.noaa.gov/Ferret/Documentation/PPLUS_Users_Guide/pplus_enhance_user_guide.html

It is included here with changes only to the formatting. Appendices 1 and 2 of the original document are renumbered here as sections 3 and 4. Note that section 4, which describes generating postscript plots with gksm2ps is duplicated in the main Ferret Users Guide in the section "Metafile Translation" (p.262) .

Appendix C Sec1 PLOTPLUS HISTORY, EVOLUTION

Plotplus is a scientific graphics package with a long history. I have traced it only a small distance, and what I know is sketchy. My present understanding is that a number of users at the Oregon State University department of Oceanography contributed over a number of year to a graphics package with both original and pre-existing algorithms and code; PLOT1, PLOT2, PLOT3 and PLOT4 successively became the current standard. Don Denbo took a strong interest in improving the package; from his work evolved PLOT5. He came to PMEL, improved PLOT5 further, and Plotplus was born. While here he made modifications for the TMAP group to that code to support the Graphics Kernel System, GKS, an international standard for programming computer graphics applications. This user's guide describes modifications I made to Plotplus to extend the use of GKS within it; this version will no doubt evolve as well.

The guide addresses itself only to the TMAP modifications to Plotplus. The Plotplus manual describes all other aspects of the current version as supported by its author and should be consulted for information about using Plotplus.

The TMAP GKS enhancements of Plotplus include modification and addition of several PPL commands, including:

ALINE,
CLSPLT,
COLOR,
CONSET,

FILL,
LINE,
LIST,
PEN,
PLTNME,
PLTYPE,
SHADE,
SHAKEY,
SHASET.

This version of PPL+ is modified to be used with the public domain GKS library XGKS. X Windows is the only supported device type. There is in addition a utility, gksm2ps, to generate monochrome and color hard copy of PPL+ plots. `gksm2ps' was written by Larry Oolman at the University of Wyoming; I modified it for use with PPL+. Please see the Ferret Users Guide for information on using gksm2ps.

Appendix C Sec2 ENHANCED COMMANDS DESCRIPTION

Appendix C Sec2.1 ALINE/qualifier line#, minx, miny, maxx, maxy, set

Draws the line associated with the specified line number between 2 points (see PEN for more on this). Two modes are available. In immediate mode the line is drawn when the command is given. Deferred mode permits setting of several lines (with individual endpoints) to be drawn whenever the PLOT command is given. Deferred mode is included so that examples of each linetype used in a plot can be provided as part of a key. The ALINE command does not modify data in the plot buffer; lines drawn can be considered as labels. The ALINE command given with no arguments resets all set lines to OFF.

NOTE: ALINE does not work well when the plot has a time axis, and it fails in some other circumstances. It is simpler to draw a line on a plot with a PLOT/OVERLAY command, such as

```
yes? LET xpts={4,8}
yes? LET ypts={-3,0}
yes? PLOT/VS/LINE/OVER/NOAX/NOLAB/COLOR=/THICK=/DASH=(...) xpts,ypts
```

If you want the line to go outside the plot box:

```
yes? PLOT/VS/OVER/SET....
yes? PPL window,off
yes? PPL plot/over
```

When you have a time axis on the plot, you will need time coordinates to use in assigning point locations. Use the ideas in the FAQ "how can I overlay symbols on a plot with a time axis?" at http://ferret.pmel.noaa.gov/Ferret/FAQ/custom_plots/time_series_symbol_overlay.html

ALINE Arguments:

line# The line to be drawn will be of the type, thickness, and color associated with this line number.

minx X-component of the first endpoint.

miny Y-component of the first endpoint.

maxx X-component of the second endpoint.

maxy Y-component of the second endpoint.

set is optional. If omitted, execution mode is immediate. If ON, sets deferred mode for the specified line number. If OFF, drawing the line is canceled; specification of the endpoints may be omitted when canceling ALINE for a line. Execute LIST ALINE to find which lines are set, and their coordinates.

Valid qualifier:

/[NO]USER determines whether user coordinates or inches will be used in locating the line. Default is /user.

Appendix C Sec2.2 CLSPLT

Modified to be compatible with GKS metafile use. Closes the currently open plot file.

Appendix C Sec2.3 COLOR n, red, green, blue

Sets the color of a single color using the RGB color model. Specify with no arguments to reset colors 1 through 6 to their default values. These six colors are used as the line colors in line and contour plots. See "GKS line bundles" for more on line color and bundles.

n color index

red The intensity of red, with a value from 0 to 100%

green The intensity of green

blue The intensity of blue

In the present version of PPL+, colors of indices 0 and 1, corresponding to the plot window background and foreground, are restricted to black (intensities all 0) or white (intensities all 100).

Appendix C Sec2.4 CONSET hgt, nsig, narc, dashln, spacln, cay, nrng, dslab, spline_tension, draftsman

Two new parameters have been added, spline_tension and draftsman. Spline_tension controls a spline fitting routine for contour lines and is used primarily in conjunction with the narc parameter to alter the way contour lines are drawn. The new parameter draftsman enables the user to specify either horizontally oriented contour labels (draftsman style) or the default, contour labels running along contour lines.

hgt height of contour labels (default=.08 inches)

nsig no. of significant digits in contour label (default=2)

narc number of line segments to use to connect contour points (default=1)

dashln dash length of dashes mode (default=.04 inches)

spacln space length of dashes mode (default=.04 inches)

cay is the interpolation scheme. If CAY=0.0, Laplacian interpolation is used.

The resulting surface tends to have rather sharp peaks and dips at the data points (like a tent with poles pushed up into it). There is no chance of spurious peaks appearing. As CAY is increased, Spline interpolation predominates over the Laplacian, and the surface passes through the data points more smoothly. The possibility of spurious peaks increases with CAY. CAY = infinity is pure Spline interpolation. An over relaxation process in used to perform the interpolation. A value of CAY=5.0 (the default) often gives a good surface.

nrng Any grid points farther than NRNG away from the nearest data point will be set to "undefined" [1.0E35] (default=5)

dslab nominal distance between labels on a contour line (default = 5.0 inches)

spline_tension a real value that affects the fit of the contour line. sometimes it's called the tension factor. This value indicates the curviness desired. If abs(spline_tension) is nearly zero (e.g. .01) the resulting curve is approximately a cubic spline. If abs(spline_tension) is large (e. g. 10) the resulting curve is nearly a polygonal line. If spline_tension equals zero, the resulting curve will be calculated by the original algorithm in PPL. This will result in a cubic polynomial fit. This parameter is only applied if narc is greater than 1. Otherwise, straight lines are drawn between data points and no interpolated points are contoured. A typical value for spline_ten-

sion is 1, and the typical useful range of values is .01 to 10. The spline interpolation used in this calculation will result in erroneous plots for certain large values of spline_tension (about 20 or greater). It is up to the user to choose an appropriate value of spline_tension to avoid this error. No error checking is conducted in the interpolation routine for this condition because the error depends highly on the data being interpolated. The default for spline_tension is zero, so the standard contour line fit is used unless something else is input. NOTE: While it may seem that this feature somewhat overlaps the feature documented in the parameter cay above, this is not true. The parameter cay and the associated feature is only implemented if both the grid and data are read in directly using the RD command. In that case, CONSET must be input before RD. On the other hand, when using the spline_tension feature described here, this is not needed as interpolation is carried out at the time the contour lines are drawn.

draftsman a real value that controls the label format. If draftsman is set to zero, the original label style is used. This style writes the labels along contour lines at varied angles. If draftsman is set to anything other than zero, all the labels will be oriented horizontally on the page (a.k.a. draftsman style). At this time the magnitude of the non-zero value has no bearing on the plot. The default is zero.

Appendix C Sec2.5 FILL/qualifier

FILL is a modification of the PPL AREA command. FILL generates a "smooth-bordered" area-filled contour plot of a 2-d field. As with the SHADE command, the SHAKEY and SHASET commands can be used to control the appearance of the plot. The /[NO]WAIT and /[NO]OVERLAY qualifiers are valid, used in the same way as with PLOT, CONTOUR and SHADE.

Appendix C Sec2.6 LINE n, mark, use

The original PPL command has been modified. PPL+ uses GKS line bundles to specify line type, thickness, and color. A number of line bundles are defined for each device type and their characteristics depend on the capabilities of the device. See "GKS line bundles" for this information. The LINE command use argument no longer specifies the line type -- whether the line is to be dashed or solid. Specification of the use of marks remains the same.

n line number

mark data mark

use line/mark use specification
0 - line connecting points and no mark at points
1 - mark data points
2 - mark end points only
3 - only marks (no line)

Appendix C Sec2.7 LIST arg

New arguments are available to the LIST command to request information about the settings of the added features.

arg New arguments are ALINE, SHAKEY, and SHASET.

Appendix C Sec2.8 PEN n, ndx

This command has been modified with the use of GKS line bundles in PPL+. It now specifies the line bundle index associated with each line. See section "GKS line bundles" (p. 563) for the type, thickness, and color representation for each line bundle.

n The line number. If n is 0, sets the pen used to plot the axes and labels.

ndx sets the line bundle index to be used for line n. Default is 1.

Appendix C Sec2.9 PLTNME metafile_name

Modified to be compatible with GKS metafile use. Specifies the name to be used when a metafile is being made with each plot. The default is `metafile.plt'.

Appendix C Sec2.10 PLTYPE icode META

PLTYPE 3, GKS output, currently supports only X Windows output. Hardcopy can be generated using the gksm2ps command.

icode must be 3 to use the new features.

A GKS metafile with the default name `metafile.plt' (with sequential version numbers for subsequent plots) is produced with each plot when META is specified.

After the metafile is set, if you wish to deactivate the metafile output, reenter the PLTYPE command without entering META, e.g., PLTYPE 3. To reactivate, reenter PLTYPE including the META specification, .e.g., PLTYPE 3 META. The gksm2ps command translates the metafiles and generates PostScript output from them. See the Ferret Users Guidefor information on gkem2ps.

Appendix C Sec2.11 SHADE/qualifier

Generates a fill area plot of a 2-d field. A rectangular grid is defined when visualizing 2-d fields in PPL; a grid cell is associated with each point. The SHADE command fills in each grid cell with a color determined by the field value at the grid points.

The LEV and LIMITS commands can be used, in a way identical to their use with CONTOUR, to determine the levels shaded, and specify intervals. The SHAKEY and SHASET commands also control the appearance of the plot; default colors (or patterns) and key attributes will be used if not specified. The /[NO]WAIT and /[NO]OVERLAY qualifiers are valid, used in the same way as with PLOT and CONTOUR.

Appendix C Sec2.12 SHAKEY do_key, orient, klab_siz, klab_inc, klab_dig, klab_len, kx_lo, kx_hi, ky_lo, ky_hi

This command controls the attributes of the key generated by the SHADE command. The key associates the colors or patterns used in the plot with the field values; its use is optional. LIST SHAKEY will list current settings of the key.

- do_key If 0 the key will not be displayed; if 1 the key will be displayed. Default is 1.
- orient If 0 the key is horizontal (by default on top of the figure); if 1 the key is vertical (by default on the right). Default value is 1. To locate thekey on the left or bottom, or anywhere else on the page, use the last four arguments to SHAKEY; kx_lo, kx_hi, ky_lo, ky_hi.
- klab_siz If non-zero, klab_siz is the height of key label characters in inches. If 0, SHADE selects a reasonable height; default is 0. By default the labels are on the right side of a vertical colorbar key and above a horizontal key. To put the labels on the opposite side of the colorbar, send a negative value for klab_size , e.g. -0.1 for labels of size 0.1 on the left of the colorbar.
- klab_inc If non-zero every klab_inc key level is labeled; if 0, SHADE selects a suitable value. Default value is 0.
- klab_dig is the number of significant digits (klab_dig > 0) or decimal places (klab_dig < 0) in the key. Default is 3.
- klab_len is the maximum number of characters in a key label. Default is 9.
- kx_lo X-coordinate of the left side of the key, in inches. (Prior to V5.53 of Ferret, to change any of kx_lo, kx_hi, ky_lo, ky_hi, you needed to set all four of them.)
- kx_hi X-coordinate of the right side.

- ky_lo Y-coordinate of the bottom of the key, in inches.
- ky_hi Y-coordinate of the top.

Example:

```
SHAKEY 1, 1, 0, 3, 4, 8, 9.4, 10.2, 1.4, 7.4
```

Specifies that the SHADE command draw a vertical key with label size selected automatically and every third color of the key labeled. Labels will contain 4 significant digits to a maximum of 8 digits. The entire key will occupy a rectangle from (x,y) = (9.4,1.4) to (10.2,7.4) in inches.

Appendix C Sec2.13 SHASET

SHASET set_pt, red, green, blue

SHASET SAVE=spknme

SHASET SPECTRUM=spknme

SHASET DEFAULT

SHASET PROTECT

SHASET RESET

This command sets the colors used in the plot generated by the SHADE and FILL commands, and takes several forms. By default the shaded plots use a spectrum of color from blue to red. A user-defined selection of colors can be chosen, saved for future use, and recalled by name.

SHASET uses the following approach in defining a spectrum. The levels set by the LEV command have lower and upper bounds; SHASET defines a scale from 0 to 100 which spans the interval defined by these bounds. With SHASET you may specify a color at any point along the scale by setting a control point on the scale, along with the red, green, and blue fractions (between 0 and 100% of maximum intensity) defining the color at that point.

A spectrum is built up by setting colors at a number of points. The SHADE and FILL routines linearly interpolate each red, green, and blue fraction between set points to achieve a smooth transition from point to point.

When you are interested in creating a custom spectrum to use in SHADE and FILL plots, first execute SHASET with no arguments. This clears all set points except the bottom and top of the scale; the bottom, at zero, is set to black (red, green, and blue fractions all 0% of maximum intensity). The top, at 100, is set to white (with red, green, and blue all 100%). SHASET can then be used to set any point in the scale to any color.

set_pt defines a point in a scale, from 0 to 100, that spans the levels as set in the LEV command. Set_pt can be negative; when set negative, SHASET deletes this set point from the current spectrum, if present. RGB values need not be specified in this case.

red The intensity of red in the color at the set point, with a value between 0 and 100%.

green The intensity of green.

blue The intensity of blue.

You may save a spectrum for later use using the SAVE form of the command, where you give the present spectrum a name. A spectrum can be recalled using the SPECTRUM form of the command. One spectrum, `rainbow', the PPL+ default spectrum, is always available for recall.

spknme A name to be associated with a particular spectrum. The spectrum is stored in the current directory as spknme.spk, is an ASCII file, and can be edited. One spectrum may be saved in memory by omitting the spknme qualifier (specify SHASET SAVE) ; that spectrum is recalled from memory with SHASET SPECTRUM.

The DEFAULT form will set the spectrum to the default colors associated with the workstation in use. This is not the PPL+ default but is defined for the device by GKS.

If you plan to overlay more than one SHADE or FILL in a single plot , specify SHASET PROTECT after each use of either to protect the colors already used in the plot. SHASET RESET resets the internally kept pointer protecting previously used colors, permitting their reassignment. Use the RESET option when you are ready to begin a fresh non-overlay plot.

Appendix C Sec3 GKS LINE BUNDLES

GKS employees the concept of line bundles, where lines may be referred to by an index; the index determines the three characteristics of plotted lines, namely, line type, thickness, and color. Line type means whether the line is solid, dotted, dashed, dashed-dotted, etc. Thickness is measured in units beginning with one; two is twice as thick as one, followed by three, three times as thick. Colors are selected using a color index; the color associated with an index can be set using the PPL+ COLOR command.

The values of these characteristics together determine the line representation. One result of bundle use is that the same line bundle index may be defined to have different representations on different devices.

On X Windows devices color is usually used to distinguish lines; in that case the line type is always solid. When Postscript hard copy is made from PPL+ metafiles, lines may be rendered as color, but on monochrome printers the only available color is black. Consequently, when output is to a monochrome printer, gksm2ps uses differing line types to distinguish lines instead

of color. Lines of differing thickness are available in both PPL+ X Window output and hard copy.

The table below presents the line thickness, color index and associated default color for color devices (assuming a white background), and line type for monochrome devices, for the 19 line bundles in PPL+. On color devices, the default color of line index 1 is black if the background is white, and white if the background is black; line bundle index 19 is the background color (the color index is 0).

Bundle index	Thickness	Color	Line type (Monochrome devices)
1	1	1, black	solid
2	1	2, red	dashed
3	1	3, green	dotted
4	1	4 , blue	dashed-dotted
5	1	5, cyan	long dashed
6	1	6, magenta	dashed and double-dotted
7	2	1, black	solid
8	2	2, red	dashed
9	2	3, green	dotted
10	2	4, blue	dashed-dotted
11	2	5, cyan	long dashed
12	2	6, magenta	dashed and double-dotted
13	3	1, black	solid
14	3	2, red	dashed
15	3	3, green	dotted
16	3	4, blue	dash-dotted
17	3	5, cyan	long dashed
18	3	6, magenta	dashed and double-dotted
19	1	0, white	

Appendix C Sec4 HARD COPY

PostScript formatted files suitable for printing can be generated from PPL+ metafiles using the gksm2ps command. Several command line arguments permit the tailoring of the output. The command and its arguments are:

gksm2ps:Send PostScript translation of GKS metafiles to a file

```
usage:    gksm2ps [-h] [-p landscape||portrait] [-l ps||cps] [-d cps||phaser] \
          [-X || -o <ps_output_file>] [-R] [-a] [-g WxH+X+Y] [-v] file(s)

   -h   print this help message
   -p   page orientation, landscape or portrait (default fits to page)
   -l   line styles,  ps == monochrome (default), cps == color
   -d   device type, cps == Postscript (default), phaser == TEK phaser PS
   -X   Send output to your Xwindow for preview instead of a file
   -o   output file name (default name is 'gksm2ps_output.ps')
   -R   do not rename files with a date stamp appended (default is to stamp)
   -a   make hard copy the size of the original plot (default fits to page)
   -g   WxH+X+Y  WIDTH, HEIGHT, XOFFSET, & YOFFSET in points
   -v   list version number of gksm2ps and do nothing else
   file(s)   The specific metafile(s) to be translated.
```

More about the arguments and their effects:

-h Simply print the above help message.

-p Specifies the orientation of the plot on the page to be landscape (with the long side of the page horizontal), or portrait (the short side horizontal). The default fits the plot on the page in the orientaion that best fits.

-l Specifies line styles that will be used in the PS output. Monochrome is the default but color may be more appropriate on color devices.

-d Specifies the device type. Phaser printers using transfer sheets are PostScript, but the available plotting area is reduced. The phaser option reduces the size of the plot slightly.

-X This option lets you preview plots on your workstation screen.

-o Specifies the output file name. All metafiles translated in a single execution of the gksm2ps command are writen to a single file.

-R The default renaming of the metafiles to be translated is intended to help distinguish metafiles that have been printed from those newly made. This option turns off that renaming.

-a The original size of the PPL plots is captured in the metafile; use this option to create the hard copy that size. The default fits the plot to the available page size.

-g Specify the hard copy plot size and offset in points (72 points = 1 inch).

-v Just lists the version number.

file(s) Name the metafiles to be translated; separate the file names with a space. Wild card specification can be used.

Index

!

A

B

C

D

E

F

G

H

I

J

K

L

M

N

O

P

Q

R

S

T

U

V

W

X

Y

Z