Laser-Scan Ltd.

TVES

Reference Manual

Volume I

Issue 2.2 - 22-February-1993

Science Park, Milton Road, Cambridge, England CB4 4FY tel: (01223) 420414

Document "TVES REFERENCE", Category "REFERENCE"
Document Issue 2.2   Various (modified 22-Feb-1993)

Laser-Scan Ltd.

TVES

Reference Manual

Volume II

Issue 2.2 - 22-February-1993

Science Park, Milton Road, Cambridge, England CB4 4FY tel: (01223) 420414

Document "TVES REFERENCE", Category "REFERENCE"
Document Issue 2.2   Various (modified 22-Feb-1993)

Laser-Scan Ltd.

TVES

Reference Manual

Volume III

Issue 2.2 - 22-February-1993

Science Park, Milton Road, Cambridge, England CB4 4FY tel: (01223) 420414

Document "TVES REFERENCE", Category "REFERENCE"
Document Issue 2.2   Various (modified 22-Feb-1993)

CONTENTS

TVES documentation change record

Version 2.1 Jon Barber 21-Aug-1991

Real RANGE values allowed in VECTORISE.

Version 2.2 Jon Barber 22-Feb-1993

ENABLE BILINEAR command added to keep the observer position as specified, using bilinear interpolation to get the matrix height there, and to do any subsequent visibility calculations on these more accurate values. DISABLE BILINEAR will restore the default behaviour of using the nearest pixel height for the whole pixel extent.

CHAPTER 1

INTRODUCTION

Placeholder for missing INTRODUCTION.RNO

CHAPTER 2

MODULE COVER

MODULE COVER

Intervisibility Calculation and Display

Issue 5.2 - 22-February-1993

Science Park, Milton Road, Cambridge, England CB4 4FY tel: (0223) 420414

Document "COVER Reference" Category "Reference"

Document Issue	1.0	D R Catlow	04-November-1985
	2.0	D R Catlow	14-August-1986
	3.0	D R Catlow	18-March-1987
	4.0	D R Catlow	23-November-1987
	5.0	D R Catlow	09-March-1989
	5.1	M W S Reid	20-July-1989
	5.2	J Barber	22-February-1993

MODULE COVER

FUNCTION

COVER determines which areas of the terrain surface are visible or hidden from the view of an observer. The observer may be positioned on or above the terrain surface, and may represent either a human observer, or a device such as a radio transmitter.

The input to the module is a Digital Terrain Model (DTM) and optionally a file with information on the heights of surface features. The output is a grid based intervisibility or cover map, that may be displayed in colour coded form on a graphics device, or manipulated using other TVES modules.

Both input and output data are held in a Laser Scan Digital Terrain Image (DTI) file.

FORMAT

$ COVER

PROMPTS

COVER is an interactive, command driven program. Command input is expected when the following prompt is issued:

COVER >

Commands may be issued at the terminal in response to the prompt, or in the case of a number of commands, by pressing a button on a digitising puck.

DESCRIPTION

COVER determines which areas of the terrain surface are visible or hidden from the view of one or more observers.

Visibility is calculated from user supplied information on the position and field of view of the observers, and using elevation data held in the form of a Digital Terrain Model (DTM).

The user of the module is required to specify the geographical position of an observer and optionally his height above the terrain surface, his cone of vision, his direction and maximum distance of view. The last two parameters may be specified alternatively by supplying the position of a target point.
Observer and target locations may be entered by digitising a series of points on a map located on a digitising table, or via the terminal keyboard. Definitions for up to 8 observers may be supplied.

The DTM defines the height and shape of the terrain surface. Optionally a second file of the same resolution as the DTM and containing information on the height of surface features such as woodland, may be supplied to the module, and used in the visibility calculations.
All the input data is supplied to the program as raster data held in a Laser Scan Digital Terrain Image file.

When calculating visibility along a line of sight, an option to take into account earth curvature and terrestrial refraction may be selected.

The output from the module is a cover or intervisibility map recorded in a Digital Terrain Image file. This map may be displayed in a colour coded form on a graphics device. Hidden and visible areas of ground may be shown for each observer, while those areas that are visible to more than one observer may be distinguished. Alternatively the visibility data may be processed through other TVES modules; for example, the boundaries of all hidden ground may be extracted and plotted in a vector form using the module VECTORISE.

An option to compute visibility along a simple single line of sight is provided in the module.

COVER may be used to generate a cover map, create a new map from an existing intervisibility map, or display a cover map, as separate and independent operations.

CALCULATION OF INTERVISIBILITY

The manual production of an intervisibility map is a time consuming operation. Typically, it involves constructing a series of terrain profiles emanating from an observation point using the contour information on a topographic map, and the determination of the hidden parts of each profile by the graphical construction of lines of sight. Hidden areas are delineated by linking together adjacent hidden profile sections to form continuous areas of hidden ground.
The accuracy of the resulting cover map depends on the scale of the contour map, the vertical interval of the contours, the accuracy of the contours, the number and spacing of the profiles that are constructed, and not least the map interpretation skill of the user.

The increasing availability of Digital Terrain Models and computer algorithms, has overcome many of the problems associated with the manual construction of cover maps.
Using computer methods it is possible to produce a more accurate cover map based on a large number of densely spaced profiles; to interpolate consistently heights between recorded data points; to determine quickly line of sights along profiles, and to produce a digital product that can be plotted, manipulated and combined with other digital information. Use of automated methods has encouraged the production of cover maps that show the complex patterns of visibility generated by a number of observers.

In order to determine the visibility of an area of terrain to an observer, COVER constructs a series of lines of sight or profiles between the observation point, and each DTM node within the defined rectangular area of interest. For an area of interest covering 200 columns and 200 rows, a total of 40000 lines of sight may be potentially constructed; in practice fewer will be required since only DTM nodes that lie within an observer's field of view are considered. Each DTM node is considered in turn, starting with the node defining the SW corner of the area of interest.

In order to produce a reliable picture of visibility over an area of terrain, the construction of this large number of lines of sight is essential, since each line of sight between an observer and a DTM node, will intersect the grid of elevation values at slightly different points. Only in a small number of cases will a line of sight exactly intersect at a DTM node, so interpolation of heights between nodes is required. This fact is essential to an understanding of the intervisibility algorithm used by COVER.

For each DTM node (target point) in the rectangular area of interest, the following algorithmic steps are performed:

1. The height of the target point is read from the DTM, and the distance, direction and angle of elevation of the target point from the observation point is calculated.

2. The position of the target point is tested against the field of view of the observer. Only the visibility of a point that lies inside this 2D or 3D area (depending on the cone of vision), is computed.

3. The intersection of the line of sight drawn between the observation and target point, with intervening columns of the DTM is first considered. Each column intersection point is calculated and considered in turn. For all intersection points, other than those actually on a DTM node, a height value is interpolated linearly using the heights of the DTM nodes above and below the intersection point.

4. Optionally compensation is applied for earth curvature and terrestrial refraction, and the height of any surface feature at the intersection point is derived from a second matrix, and added to the terrain surface height.

5. If the height at the intersection point is equal or below both the observation and target point heights, the point has no influence on visibility, and the program proceeds to calculate the intersection of the line of sight with the next column.

6. If the height at the intersection point is higher than the observation or target points, then the target point cannot be visible to the observer, and no further processing of the line of sight is performed. The non-visibility of the target point is recorded.

7. For other cases, the distance and angle of elevation of the intersection point with respect to the observation point is calculated. If the height at the intersection point is below the observer, the angle of elevation is of course negative.

8. The angle of elevation at the intersection point, is compared against the target point elevation angle. If it is greater, then it indicates that the target point is not visible, and no further processing on the line of sight is performed. If it is equal or less than the target angle, the program proceeds to determine the intersection of the line of sight with the next column, and re-performs the above tests.

9. If after all intervening columns are considered, the target point is still potentially visible to the observer, the program proceeds to consider the intersection of the line of sight with all intervening DTM rows. As before the height at intersection points is interpolated linearly, though this time using the heights of the DTM nodes immediately to the left and right of the intersection point. The height and elevation angle tests are applied as before.

10. If processing continues until all intervening rows have been considered, then the target point must be visible to the observer.

An important point to note about the implementation of the algorithm, is that the DTM nodes immediately surrounding an observer are considered first when computing visibility along a line of sight. The speed with which the visibility of an area of terrain is determined is therefore dependent on the height and form of the terrain surface in the immediate vicinity of an observer. The smaller the area of terrain that is visible to an observer, the faster the cover map will be generated.

ENCODING OF COVER MAP DATA VALUES

A visibility value is recorded in the output cover map for each node that lies within the rectangular area of interest. The recorded value is dependent on how many observers are being considered, and to which observer or observers the node is visible.

The visibility status of a node to each observer is held in a bitwise manner: bit 1 is set for observer 1, bit 2 is set for observer 2 and so on. A bit is unset if the node is not visible to a particular observer. If the node lies outside the field of view of all observers, or is recorded as a NULL value in the input DTM, a value of -1 is written to the cover map.
For the simple case of just observer 1, the values in the output cover map will be in the range -1 to 1; for a cover map generated using observers 1 and 2, the values will be in the range -1 to 3, with a value of 3 indicating that the node is visible to both observers.

The bitwise manner of encoding the data value is interpreted by COVER when the cover map is displayed on a graphics device. Other modules within TVES may not be able to correctly interpret the values, however the WRITE option is provided in COVER to write decoded information to a DTI file for use with modules such as VECTORISE.

EARTH CURVATURE AND TERRESTRIAL REFRACTION

It is important for many applications, and in particular when constructing lines of sight over long distances, to take into account the effect of earth curvature and terrestrial refraction in intervisibility calculations.
Earth curvature causes points to be observed at lower angles as distance increases, and points to disappear below the horizon. The equation

$$e = d*d / 2R$$

is used to compute earth curvature (e), where d is the distance between the observer and target point in kilometres, and R is the radius of the earth (taken as 6378 kms in COVER). For a distance of 10 kilometres, the value of e will be approximately 7.84 metres.

Rays of light passing through the earth's atmosphere in any direction other than vertical are bent from a straight path. This refraction takes place in a direction towards the earth's surface under normal atmospheric and temperature conditions. As a result of refraction, points tend to be observed at a higher elevation as distance from the viewpoint increases. The equation

$$r = (d*d) * k/2R$$

is used to compute the effect of refraction (r), where d is the distance between the observer and target point in kilometres, R is the radius of the earth, and k is the coefficient of refraction. By default COVER uses a coefficient value based on a 4/3 Earth of 0.125, although this value may be redefined by the user. Using the default value, for a distance of 10 kilometres, the value of r is approximately 0.98 metres.

If both earth curvature and refraction are taken into account, the height of a point 10 kilometres from the observer is therefore adjusted downwards by 6.86 metres.

USE OF SURFACE COVER HEIGHT INFORMATION

In most areas of the World, surface features such as vegetation and large buildings, have a considerable influence on visibility along a line of sight. This is particularly the case in the immediate vicinity of an observer, where even a single building may severely restrict an observer's field of view.

Surface cover has generally not been taken into account in line of sight calculations when using computer algorithms, because of the lack of data on the location and height of surface features. Increasingly however, suitable datasets are becoming available. A mechanism is provided within COVER to determine the height of a point using both the terrain elevation data in the DTM, and data on surface feature heights contained in a second grid file.
If this source data is available only in a vector form, the TVES module I2GRID may be used to generate the grid file used by COVER. The data in the second file must be held at the same resolution as the elevation data in the DTM.

If the option to use surface feature heights is selected, then a height value from the second file is added to the terrain surface height at all points where a line of sight intersects the grid of elevation values. It is possible to specify whether this action is carried out at the target point as well as intervening points, or just at the intervening points alone. In the former case it is the visibility of the top of a feature at the target point that is determined; in the second case the visibility at the base of a feature is calculated.

USE OF A DIGITISING TABLE AND PUCK BUTTONS

If your workstation incorporates a digitising table with a 16 button puck, then it is possible to register a source document such as a map to the DTM, and to issue a number of the COVER commands by pressing an appropriate button on the table puck.

If the logical name LSL$AUTO_ENABLE_TABLE is defined with a value of "1" the program will attempt to initialise the Laser Scan Table Monitor Utility on startup. If the logical name is absent or has any other value, initialisation is not performed on startup, however the table can be subsequently initialised using the command ENABLE TABLE.

The Table Monitor utility controls reading from the digitising table. It must be set up on your system if COVER is to take input from the digitising table. If table initialisation fails then the message:

```
*** ERROR *** Initialising table monitor
COVER will assume no table is available
```

will be output before the prompt **COVER>** is displayed on the terminal. If the event of failure, input from the digitising table or puck button is not possible, and the program will accept commands only from the terminal.

Table initialisation will generally fail because no Table Monitor is currently active, or because the Table Monitor is locked by another user. If neither of these reasons appear to apply, you should consult the TABLIB Reference Manual, or seek guidance from your system manager.

A map is registered to the DTM using the command **SETUP MAP**. Before giving this command you should have first selected the input DTM, and have securely attached the source document to the digitising table.

Registration is performed by digitising 4 rectangular registration points. The registration points represent 4 points on the map that correspond to the 4 corners of the DTM. The points are digitised in the order top left (NW), bottom left (SW), bottom right (SE) and top right (NE) using any button on the table puck. A point is digitised in response to an explanatory prompt on the terminal.

An error message is generated if any of the angles of the digitised rectangle are less than 88 degrees, or greater than 92 degrees (ie. if the corner points of the rectangle are more than 2 degrees off rectangular). In this situation you will be asked to redigitise the 4 registration points.

Once a map has been positioned on the table, it is possible to define an area of interest in the DTM by giving the WINDOW command and digitising two points on the map. It is also possible to digitise observer position and target coordinates using appropriate puck buttons.

The buttons on the table puck have the following meanings:

0 POSITION	1 OBSERVER 1	2 OBSERVER 2	3 OBSERVER 3
4 OBSERVER 4	5 OBSERVER 5	6 OBSERVER 6	7 OBSERVER 7
8 OBSERVER 8	9 TARGET	A POINT	B WINDOW
C undefined	D undefined	E undefined	F ABANDON

BUTTON 0 is used to define the position of an observer currently selected by means of the DEFINE OBSERVER command. A single point inside the map area should be digitised.

BUTTONS 1 to 8 are used to define the position of observers 1 to 8 respectively. A single point inside the map area should be digitised. Note that when any of these buttons are pressed an implicit DEFINE OBSERVER command is performed.

BUTTON 9 is used to define the position of the target view point for the observer currently selected by means of the DEFINE OBSERVER command. A single point inside the map area should be digitised. The distance and direction of the target point from the observer is automatically calculated.

BUTTON A is used to define the position of a single target point. The point must be digitised within the map area. The program will immediately calculate whether the point is visible to all currently selected observers.

BUTTON B is used to give a WINDOW command. The button may be pressed either inside or outside the map area. After pressing the button, you will be asked to digitise 2 points inside the map area, defining the SW corner (bottom left) and NE corner (top right) of a rectangular area of interest.

BUTTONS C to E are currently undefined.

BUTTON F may be used to abandon map set up, or window definition. When the button is pressed, control is returned to the terminal and the prompt COVER> is displayed.

COVER COMMANDS

!

Treat all text to the right of the '!' as a comment.

FORMAT: ! [comment text]

Command parameters:

comment text

text that is to be treated as a comment and which will be excluded from command interpretation.

DESCRIPTION:

An exclamation mark is the standard TVES package comment delimiter. All text (and numbers) which lie to the right of a '!' character are excluded from command interpretation. Comments are useful for annotating command procedures used in batch processing etc.

Messages: None.

Examples:

COVER> **FILEIN TEST !open the file<CR>**
COVER> **!Define a window<CR>**
COVER> **WINDOW 1 1 10 10<CR>**
COVER>

CONE

Defines the cone of vision of the observer currently selected by means of the DEFINE OBSERVER command.

FORMAT: CONE angle

Command parameters:

angle

Angle is the cone of vision specified in degrees. It should be supplied as a real (floating point) value in the range 1 to 360.

DESCRIPTION:

CONE is used to define the cone of vision of the observer selected using the DEFINE OBSERVER command.

If the cone angle is less than 180 degrees, the cone of vision is a 3 dimensional area, defining in combination with the distance of view, the x, y and z extremities of an observer's field of view.
For cone angles of 180 degrees or greater, the cone angle defines in combination with the distance of view, the x and y (planimetric) extremities of an observer's field of view.
Only the visibility status of nodes that lie within an observer's field of view are determined.
In the output cover map, those nodes that are outside the field of view of all observers are given a value of -1.

When an observer is first selected, the default cone of vision is 360 degrees (ie. a complete circular scan around the observer), however if either a TARGET or DIRECTION command is subsequently given, the default is redefined to be 90 degrees.
For a 360 degrees field of view, the DISTANCE command defines the radial distance. For cones of vision less than 360, the field of view is bisected by a line defining the direction of view. For example, if the current direction of view is along a line 45 degrees from North, and the cone 60 degrees, then the subtended cone is between 15 degrees and 75 degrees from North (ie 30 degrees either side of the direction of view).

Messages:

The following error messages are specific to the CONE command:

*** ERROR *** Specifying command CONE
No observer has been selected using the DEFINE OBSERVER command

*** ERROR *** Specifying command CONE
Command requires 1 real argument

```
*** ERROR *** Specifying command CONE
Command argument should be in range 1.0 to 360.0
```

Example:

```
COVER>DEFINE OBSERVER 1<CR>
COVER>POSITION 100 100<CR>
COVER>DISTANCE 70<CR>
COVER>SHOW OBSERVERS<CR>

Current observer settings (Units are DTI Matrix Values)

Observer       Position          Height    Direction    Cone    Distance
  1          100      100         50.3        0.0      360.0       70.0

COVER>DIRECTION 45<CR>
COVER>SHOW OBSERVERS<CR>

Current observer settings (Units are DTI Matrix Values)

Observer       Position          Height    Direction    Cone    Distance
  1          100      100         50.3       45.0       90.0       70.0

COVER>CONE 60<CR>
COVER>SHOW OBSERVERS<CR>

Current observer settings (Units are DTI Matrix Values)

Observer       Position          Height    Direction    Cone    Distance
  1          100      100         50.3        0.0       60.0       70.0

COVER>
```

CREATE

Enables a new cover map to be generated using all or part of the intervisibility information contained in an existing cover map.

FORMAT: **CREATE infile-spec outfile-spec**

Command parameters:

infile-spec

The file specification of an input DTI file which holds an existing cover map. This will be a file that has been created during an earlier run of COVER.
Any part of the file specification not supplied will be taken from the default 'LSL$DTI:DTI.VIS'
If no infile-spec is supplied, you will be asked to supply one in response to the prompt **Input cover map filename>**

outfile-spec

The file specification of an output DTI file which will be created to hold the new cover map.
Any part of the file specification not supplied will be taken from the default 'LSL$DTI:DTI.VIS'
If no outfile-spec is supplied, you will be asked to supply one in response to the prompt **Output cover map filename>**

DESCRIPTION:

CREATE is used to generate a new cover map using all or part of the cover information contained in an existing cover map.

The SELECT OBSERVER command controls what information is output to the cover map. Only visibility data relating to a selected observer is extracted from the input cover map, and written to the output cover map.
For example, if the input cover map was generated using observers 1 2 and 3, information relating to only observers 1 and 3 may be selected for output to the new cover map, by giving the command SELECT OBSERVERS 1 3 before using the CREATE command.
Information on the position and field of view of each selected observer read from the input cover map file header, is displayed on the terminal. The information is output in the current units of measurement.

Messages: None

Example:

```
COVER>SELECT OBSERVERS 1 3
COVER>CREATE TEST TEST1<CR>

Observer values (Units are DTI Matrix Values)

Observer       Position           Height    Direction    Cone    Distance
  1          100       100         50.3        0.0      360.0      70.0
  3           45        30         27.3       23.6       90.0     123.6

COVER>
```

DEFINE OBSERVER

Determines which observer is to have his position and field of view defined using the POSITION, DIRECTION, DISTANCE, CONE, HEIGHT and TARGET commands.

FORMAT: **OBSERVER observer_number**

Command parameters:

observer_number

An integer value in the range 1 to 8 that is used to identify the observer.

DESCRIPTION:

The DEFINE OBSERVER command controls to which observer the position and field of view parameters are applied.
The POSITION, DIRECTION, DISTANCE, CONE, HEIGHT and TARGET commands act only on the observer currently selected by means of the DEFINE OBSERVER command.

If a digitising table and 16 button puck is available on the workstation, puck buttons 1 to 8 may be used to give simultaneous DEFINE OBSERVER and POSITION commands. Button 1 is used for observer 1, button 2 for observer 2 and so on.

Messages:

The following error messages are specific to the DEFINE and DEFINE OBSERVER commands:

*** ERROR *** Defining OBSERVER POSITION using puck button
No map set up has been performed

*** ERROR *** Defining OBSERVER POSITION using puck button
Observer position must lie within DTM bounds

*** ERROR *** Specifying command DEFINE
Command qualifier is OBSERVER

*** ERROR *** Specifying command DEFINE OBSERVER
Command requires 1 integer argument

*** ERROR *** Specifying command DEFINE OBSERVER
Command requires an integer argument in the range 1 to 8

Examples:

```
COVER>DEFINE OBSERVER 5<CR>
COVER>
```

DIRECTION

Defines the viewing direction of the observer currently selected by means of the DEFINE OBSERVER command.

FORMAT: **DIRECTION angle**

Command parameters:

angle

Angle is the direction of view specified in degrees measured clockwise from North (Eg. DIRECTION 270 specifies a viewing direction that is due west).
The angle should be supplied as a real (floating point) value in the range 0 to 360.

DESCRIPTION:

DIRECTION is used to specify the direction of view of the observer currently selected by means of the DEFINE OBSERVER command.
The direction of view is a line constructed between the observer and the central target point. It is specified as an angle measured clockwise from North.

Note that no direction need be specified if a 360 degrees scan around the observer is required.

The TARGET command is an alternative way of specifying target distance and direction of view.

Messages:

The following error messages are specific to the DIRECTION command:

```
*** ERROR *** Specifying command DIRECTION
No observer has been selected using the DEFINE OBSERVER command

*** ERROR *** Specifying command DIRECTION
Command requires 1 real argument

*** ERROR *** Specifying command DIRECTION
Command argument should be in range 0.0 to 360.0
```

Example:

COVER>**DEFINE OBSERVER 1<CR>**
COVER>**POSITION 100 100<CR>**
COVER>**DIRECTION 225<CR>**
COVER>**SHOW OBSERVERS<CR>**

```
Current observer settings (Units are DTI Matrix Values)

Observer       Position          Height    Direction    Cone    Distance
  1          100      100         50.3       225.0      90.0       0.0

COVER>
```

DISABLE ABSOLUTE

Cancels a previous ENABLE ABSOLUTE command, and disables the use of absolute coordinate values.

FORMAT: DISABLE ABSOLUTE

Command parameters: None.

DESCRIPTION:

DISABLE ABSOLUTE cancels a previous ENABLE ABSOLUTE command. If DISABLE ABSOLUTE is given, then coordinate values required by the POINT, POSITION, TARGET and WINDOW commands, supplied in metre or projection units, must be specified as an offset from the SW corner of the matrix.

The command also controls the format in which DTM header coordinate values, and observer and target details are displayed, when the SHOW FILEIN, SHOW OBSERVERS and SHOW TARGET commands are given.

By default the use of absolute coordinate values is enabled.

Messages:

The following error message is specific to the DISABLE command:

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, ADD_TARGET, BILINEAR, CULTURE, DIAGNOSTICS,
EARTH_CURVATURE or TABLE
```

Examples:

COVER> **DISABLE ABSOLUTE<CR>**
COVER>

DISABLE ADD_TARGET

DISABLE ADD_TARGET is used to cancel a previous ENABLE ADD_TARGET command.

FORMAT: DISABLE ADD_TARGET

Command parameters: None.

DESCRIPTION:

The DISABLE ADD_TARGET command is used to cancel any previous ENABLE ADD_TARGET command.
The ENABLE ADD_TARGET command controls whether the height of a surface feature is added to the terrain height at a target point. It therefore controls whether the visibility of the top of a feature, or the visibility of the base of a feature at the target point is determined.
By default the ADD_TARGET option is disabled.

Use of the SHOW ENABLE command is recommended to check on the status of the option before generating a cover map or line of sight.

Messages:

The following error message is specific to the DISABLE command:

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, ADD_TARGET, BILINEAR, CULTURE, DIAGNOSTICS,
EARTH_CURVATURE or TABLE
```

Examples:

```
COVER> DISABLE ADD_TARGET<CR>
COVER>
```

DISABLE BILINEAR

Cancels a previous ENABLE BILINEAR command, and disables the use of height interpolation at the observer position.

FORMAT: DISABLE BILINEAR

Command parameters: None.

DESCRIPTION:

DISABLE BILINEAR cancels a previous ENABLE BILINEAR command. If DISABLE BILINEAR is given, the observer height is assumed to be that of the nearest pixel. This is the default action.

Messages:

The following error message is specific to the DISABLE command:

*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, ADD_TARGET, BILINEAR, CULTURE, DIAGNOSTICS, EARTH_CURVATURE or TABLE.

Examples:

COVER> **DISABLE BILINEAR<CR>**
COVER>

DISABLE CULTURE

Disables the use of surface feature height information.

FORMAT: **DISABLE CULTURE**

Command parameters: None.

DESCRIPTION:

The DISABLE CULTURE command is used to disable the use of surface feature height information. It therefore cancels any previous ENABLE CULTURE command.

Use of the SHOW ENABLE command is recommended to check on the status of the option before generating a cover map or line of sight.

Messages:

The following error message is specific to the DISABLE command:

*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, ADD_TARGET, BILINEAR, CULTURE, DIAGNOSTICS, EARTH_CURVATURE or TABLE

Example:

COVER>**DISABLE CULTURE <CR>**
COVER>

DISABLE DIAGNOSTICS

Disables the output of diagnostics messages.

FORMAT: DISABLE DIAGNOSTICS

Command parameters: None.

DESCRIPTION:

DISABLE DIAGNOSTICS disables the output of diagnostic messages, and is therefore used to cancel any previous ENABLE DIAGNOSTICS command.
By default diagnostic printout, which is sent to SYS$OUTPUT, is enabled.

Messages:

The following error message is specific to the DISABLE command:

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, ADD_TARGET, BILINEAR, CULTURE, DIAGNOSTICS,
EARTH_CURVATURE or TABLE
```

Examples:

```
COVER> DISABLE DIAGNOSTICS<CR>
COVER>
```

DISABLE EARTH_CURVATURE

Disables the option to take into account earth curvature and terrestrial refraction when calculating visibility along a line of sight.

FORMAT: **DISABLE EARTH_CURVATURE**

DESCRIPTION:

Disables the option to take into account earth curvature and terrestrial refraction when calculating visibility along a line of sight. The command is therefore used to cancel any previous ENABLE_CURVATURE command.

By default, no compensation is made for earth curvature or refraction in line of sight calculations.

Use of the SHOW ENABLE command is recommended to check on the status of the option before generating a cover map or line of sight.

Messages:

The following error message is specific to the DISABLE command:

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, ADD_TARGET, BILINEAR, CULTURE, DIAGNOSTICS,
EARTH_CURVATURE or TABLE
```

Examples:

```
COVER>DISABLE EARTH_CURVATURE<CR>
COVER>
```

DISABLE TABLE

Turns off command input from the puck button and digitising table.

FORMAT: **DISABLE TABLE**

Command parameters: None

DESCRIPTION:

The DISABLE TABLE command turns off command and coordinate input from the puck button and digitising table. If table input is disabled then COVER will accept commands only from the terminal.

Input from the table can be reselected using the ENABLE TABLE command.

The SHOW ENABLE command may be used to check on the status of table input.

Messages:

The following error messages are specific to the ENABLE command:

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, ADD_TARGET, BILINEAR, CULTURE, DIAGNOSTICS, EARTH_CURVATURE or TABLE

Example:

COVER>**DISABLE TABLE <CR>**
COVER>

DISPLAY

Displays information contained in a cover map in a colour coded form on a graphics display.

FORMAT: **DISPLAY file-spec**

Command parameters:

file-spec

The file specification for a DTI file containing the cover map.
Any part of the file specification not supplied is taken from the default 'LSL$DTI:DTI.VIS'.
If no file-spec is supplied, you will be asked to supply one in response to the prompt **Cover map filename>**

DESCRIPTION:

The DISPLAY command is used to display information contained in the specified cover map in colour coded form on a graphics screen.

Different colours are used to indicate how many observers can see a particular area of terrain. Black is used to indicate those parts of the terrain surface that are outside the field of view of all observers.
The RGB values for the colours are read from the colour table 'LSL$LOOKUP:COVERCOL.DAT'. The colour values may therefore be changed by editing this file.

Only information relating to those observers currently selected is displayed on the graphics screen. Using the SELECT OBSERVERS command it is therefore possible when a cover map has been generated using a number of observers, to view the visibility pattern associated with each observer in turn, or any combination of observers.
For this and other applications where a series of images are being generated from the same cover map, the REDISPLAY command which requires no file specification, should be used.

The image is scaled to fill the area of the screen reserved for cover map display. A legend is written on the left of the screen, and relates the display colours to the number of observers to whom the node is visible.
Text giving details of the position and field of view for each observer is written at the bottom of the screen. The UNITS command controls the units of measurement that is used when writing observer information.
If the cover map is larger than this display area, more of the screen will be used, but no legend or text will be included.

Messages: None

Example:

```
COVER>DISPLAY TEST <CR>
COVER>SELECT OBSERVERS 1 3<CR>
COVER>REDISPLAY <CR>
COVER>
```

DISTANCE

Defines the distance of view of the observer currently selected by means of the DEFINE OBSERVER command.

FORMAT: DISTANCE distance

Command parameters:

distance

Distance is the observer's maximum distance of view along the central line of sight (ie. the distance of the target point from the observer). A real (floating point) value is required, specified in either matrix units if these are the selected units of measurement, or in metres for all other currently selected units of measurement.

DESCRIPTION:

The DISTANCE command is used to specify the distance of the target point from the observer currently selected using the DEFINE OBSERVER command.

The distance is specified in the units of measurement that have been selected with the UNITS command, or set by default when the input DTM was selected. If matrix units are selected, the distance is calculated in terms of columns and rows; for all other units of measurement, the distance is measured in metres.
If a cone angle of 360 degrees has been specified for an observer, then the DISTANCE command is used to specify the radius of the circle surrounding the observer.

NOTE If no distance of view is specified for an observer, infinity is assumed.

The TARGET command is an alternative way of specifying target distance and direction of view.

Messages:

The following error messages are specific to the DISTANCE command:

```
*** ERROR *** Specifying command DISTANCE
No observer has been selected using the DEFINE OBSERVER command

*** ERROR *** Specifying command DISTANCE
Command requires 1 real argument
```

Examples:

```
COVER>DEFINE OBSERVER 1<CR>
COVER>POSITION 100 100<CR>
COVER>DISTANCE 400.32<CR>

Current observer settings (Units are DTI Matrix Values)

Observer       Position          Height    Direction    Cone    Distance
  1          100      100         50.3        0.0       360.0     400.32

COVER>
```

ENABLE ABSOLUTE

Selects the use of absolute coordinates values.

FORMAT: ENABLE ABSOLUTE

Command parameters: None.

DESCRIPTION:

If ENABLE ABSOLUTE is given, then coordinate values required by the POINT, POSITION, TARGET and WINDOW commands, supplied in metre or projection units, must be specified as absolute (rather than relative) coordinate values.

For example if the projection indicates U.K. National Grid, then the position of an observer should be specified as 6 figure National Grid coordinates.

By default values should be specified as absolute coordinates. This option can be disabled using the DISABLE ABSOLUTE command.

Messages:

The following error message is specific to the ENABLE command:

```
*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, ADD_TARGET, CULTURE, DIAGNOSTICS,
EARTH_CURVATURE or TABLE
```

Examples:

COVER> **ENABLE ABSOLUTE<CR>**
COVER>

ENABLE ADD_TARGET

Enables the visibility of the top of a feature at the target point to be determined.

FORMAT: ENABLE ADD_TARGET

Command parameters: None.

DESCRIPTION:

The ENABLE ADD_TARGET command controls whether a surface feature height value is added to the terrain height at a target point.
The command therefore controls whether the visibility of the top of a feature, rather than the visibility of the base of a feature at a target point, is computed.

The command should be used in conjunction with the ENABLE CULTURE command.
If the ENABLE CULTURE command has been given, then unless ENABLE ADD_TARGET is specified, the height of surface features is added to the terrain surface height at all nodes along a line of sight, but **NOT** at the target point itself.

Use of the SHOW ENABLE command is recommended to check on the status of the option before generating a cover map or line of sight.

Messages:

The following messages are specific to the ENABLE and ENABLE ADD_TARGET commands:

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, ADD_TARGET, CULTURE, DIAGNOSTICS,
EARTH_CURVATURE or TABLE

*** WARNING *** CULTURE is currently not enabled

Examples:

COVER> **ENABLE ADD_TARGET<CR>**
COVER>

ENABLE BILINEAR

Selects the use of a bilinearly interpolated height at the observer position.

FORMAT: ENABLE BILINEAR

Command parameters: None.

DESCRIPTION:

If ENABLE BILINEAR is given, the height at the observer position is calculated from a bilinear interpolation performed on the four surrounding pixel heights, doing the subsequent visibility calculations with these more accurate values. (The default action is to set the observer height to that of the nearest pixel.)

Messages:

The following error message is specific to the ENABLE command:

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, ADD_TARGET, BILINEAR, CULTURE, DIAGNOSTICS, EARTH_CURVATURE or TABLE

Examples:

COVER> **ENABLE BILINEAR<CR>**
COVER>

ENABLE CULTURE

Enables the use of surface feature height information when generating a cover map or determining visibility along a line of sight.

FORMAT: ENABLE CULTURE [file-spec]

Command parameters:

file-spec

An optional file specification for a DTI file containing height information on surface features. The file-spec is only required if this file was not opened when the input DTM was specified, or if you wish to specify a new file.
Any part of the file specification not supplied will be taken from the default 'LSL$DTI:DTI.MAT'.
If no file-spec is supplied, and one is required by the program, you will be asked to supply one in response to the prompt **Matrix filename>**

DESCRIPTION:

The ENABLE CULTURE command selects the use of surface feature height information when generating a cover map, or determining visibility along a line of sight.

This additional height information is read from a DTI file, which will typically contain height values relating to surface features such as woodland and large buildings. These features are often referred to as 'culture' in military environments, and 'clutter' when used in radio propagation studies.
If a file specification is supplied with the command, the height data is read from the specified DTI file.
If no file specification is given, the command will simply cancel any previous DISABLE CULTURE command, and height data read from a currently opened file will be used. An error message is generated if no such file is currently opened, and you will be prompted for a filename.

It is important that the x and y grid resolutions of the data in the surface feature DTI file, are the same as the elevation data in the input DTM.

If the ENABLE CULTURE command is given, the program will add a height read from the surface feature file on to the corresponding node in the DTM (ie. the surface feature height on to the terrain elevation value).
The ENABLE ADD_TARGET command controls whether this action is carried out for all nodes along a line of sight, including the target node, or for just those nodes between the observer and the target point.
The SET OFFSET command may be used to control the registration of the two files.

Use of the SHOW ENABLE command is recommended to check on the status of the option before generating a cover map or line of sight.

Messages:

The following error messages are specific to the ENABLE and ENABLE CULTURE commands:

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, ADD_TARGET, CULTURE, DIAGNOSTICS,
EARTH_CURVATURE or TABLE

*** ERROR *** Specifying command ENABLE CULTURE
Matrix filename is missing

Example:

COVER>**ENABLE CULTURE TREES<CR>**
COVER>**DISABLE CULTURE<CR>**
COVER>**ENABLE CULTURE<CR>** Use of height values in file LSL$DTI:TREES.MAT enabled
COVER>

ENABLE DIAGNOSTICS

Selects the output of diagnostic printout.

FORMAT: ENABLE DIAGNOSTICS

Command parameters: None.

DESCRIPTION:

ENABLE DIAGNOSTICS selects output of diagnostic messages.
The diagnostic printout consists of a series of messages indicating which operation is currently being performed by COVER, and the percentage progress.
The messages are sent to SYS$OUTPUT. Note that if you are using a hardcopy terminal no percentage progress figures are generated.

By default diagnostic printout is selected, and may be turned off using the command DISABLE DIAGNOSTICS

Messages:

The following error message is specific to the ENABLE command:

```
*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, ADD_TARGET, CULTURE, DIAGNOSTICS,
EARTH_CURVATURE or TABLE
```

Examples:

```
COVER> ENABLE DIAGNOSTICS<CR>
COVER>
```

ENABLE EARTH_CURVATURE

Selects the option to take into account earth curvature and terrestrial refraction when generating a cover map or determining visibility along a line of sight.

FORMAT: **ENABLE EARTH_CURVATURE**

DESCRIPTION:

Enables the option to take into account earth curvature and terrestrial refraction when calculating visibility along a line of sight.

The effect of earth curvature and terrestrial refraction on a line of sight becomes increasingly significant as the distance between the observer and the target point increases.
Earth curvature causes points to be observed at lower angles as distance increases, and points to disappear below the horizon. The equation

e = d*d / 2R

is used to compute earth curvature (e), where d is the distance between the observer and target point in kilometres, and R is the radius of the earth (taken as 6378 kms). For a distance of 10 kilometres, the value of e will be approximately 7.84 metres.

Rays of light passing through the earth's atmosphere in any direction other than vertical are bent from a straight path. This refraction takes place in a direction towards the earth's surface under normal atmospheric and temperature conditions. As a result of refraction, points tend to be observed at a higher elevation as distance from the viewpoint increases. The equation

r = (d*d) * k/2R

is used to compute the effect of refraction (r), where d is the distance between the observer and target point in kilometres, R is the radius of the earth, and k is the coefficient of refraction. By default COVER uses a coefficient value based on a 4/3 Earth of 0.125 This may be redefined by the user using the command SET REFRACTION_COEFFICIENT. For a distance of 10 kilometres the value of r is approximately 0.98 metres.

If both earth curvature and refraction are taken into account, the height of a point 10 kilometres from the observer is therefore adjusted downwards by 6.86 metres.

By default, no compensation is made for earth curvature or refraction in the line of sight calculations.

Messages:

The following error message is specific to the ENABLE command:

```
*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, ADD_TARGET, CULTURE, DIAGNOSTICS,
EARTH_CURVATURE or TABLE
```

Examples:

COVER>**ENABLE EARTH_CURVATURE<CR>**
COVER>

ENABLE TABLE

Initialises the Table Monitor to allow command input from a table puck and digitising table.

FORMAT: **ENABLE TABLE**

Command parameters: None

DESCRIPTION:

The ENABLE TABLE command initialises the Table Monitor to allow the input of commands from a table puck and digitising table.

The introduction contains details on table initialisation, and on how commands may be input using a table puck.

The DISABLE TABLE command may be used to turn off input from the digitising table.

Messages:

The following error messages are specific to the ENABLE and ENABLE TABLE commands:

```
*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, ADD_TARGET, CULTURE, DIAGNOSTICS,
EARTH_CURVATURE or TABLE
```

```
*** ERROR *** Specifying command ENABLE TABLE
Unable to initialise the table monitor
```

Example:

COVER>**ENABLE TABLE <CR>**
COVER>

EXIT

Terminates the program.

FORMAT: **EXIT**

Command parameters: None.

DESCRIPTION:

The EXIT command is used to exit from COVER.
<CTRL/Z> (pressing the Ctrl and Z keys together) may also be used to exit from the program.

Messages: None.

Examples:

COVER>**EXIT<CR>**
COVER>

FILEIN

Selects and opens the input DTM that contains the terrain elevation data to be used in the intervisibility calculations.

FORMAT: FILEIN file-spec

Command parameters:

file-spec

The file specification for the input DTM.
Any part of the file specification not supplied is taken from the default 'LSL$DTI:DTI.DTI', although if no file-spec is supplied, you will be asked to supply one in response to the prompt **Input DTI filename>**

DESCRIPTION:

This command opens and maps into memory a DTI file, containing the terrain elevation data to be used in the intervisibility calculations. Details derived from the header of the file are displayed on the terminal to confirm that the file has been successfully opened.

If the logical name LSL$MATRIX_DEFAULT_WINDOW is defined with a value of "1", then a default area of interest defined in matrix units of bottom left hand corner 1,1 and top right hand corner 200,200 is set when the file is opened. If the logical name is absent or has any other value, or if the input DTI file has fewer than 200 columns or 200 rows, then a default window equivalent to the total matrix is set.

After opening the DTM, the program looks for a second DTI file with the same name but with the extension MAT. This second file is optional, and is used to hold data on the height of surface features such as woodland that are present in the area covered by the DTM.
If a second DTI file is found, the option to additionally use the height values in this file is automatically enabled. This is equivalent to giving the command ENABLE CULTURE.
If no second DTI file is found, the option to use the height of surface features is disabled. The option may be subsequently enabled by giving an ENABLE CULTURE command with a DTI file specification.

Messages:

The following error message is specific to the FILEIN command:

*** ERROR *** Specifying command FILEIN
Input DTI filename is missing

Example:

```
COVER>FILEIN TEST<CR>

LSL$DTI:TEST.DTI
Header: LSLA     Data: WORD

Units are DTI Matrix Values

Matrix Coverage    SW:    1     1    NE:   301   251
Matrix Window      SW:    1     1    NE:   200   200
Matrix Interval     E:    1           N:     1
Value Range          :      0   to    851

COVER>
```

FILEOUT

Specifies the name of the output cover map.

FORMAT: **FILEOUT file-spec**

Command parameters:

file-spec

The file specification for the output grid file. Any part of the file specification not supplied is taken from a default partly constructed from the main part of the input DTI filename 'LSL$DTI:dti_filename.VIS' If no file-spec is supplied, you will be asked to supply one in response to the prompt **Cover map filename>**
Note that by default the output grid file is given the extension VIS.

DESCRIPTION:

This command is used to specify the name of the output cover map. The input file must have previously been defined with the FILEIN command.

COVER map generation is initiated with the GO command.

Messages:

The following error message is specific to the FILEOUT command:

*** ERROR *** Specifying command FILEOUT.
The input DTM is undefined.

Example:

COVER>**FILEOUT TEST<CR>** File LSL$DTI:TEST.VIS selected for output.
COVER>

GO

Initiates intervisibility calculation.

FORMAT: **GO**

Command parameters: None

DESCRIPTION:

This command is used to initiatate intervisibility calculation.

The viewing parameters for at least one observer must have been previously defined, the FILEIN command must have been given to read in a DTM, and a FILEOUT command must have been used to specify an output visibility file.
When parameters for a number of observers have been defined, the SELECT OBSERVER command may be used to restrict calculation to specific observers.
The areas of the terrain which are visible and hidden, are calculated for each observer in turn. If diagnostics are enabled, a summary of the percentage of the terrain within the area of interest that is either visible, hidden or outside an observer's field of view, is displayed after an observer has been processed.

The COVER> prompt is displayed when all calculations have been completed, and the output cover map containing the encoded intervisibility data is closed.
Observer viewing parameters and observer selections remain current until redefined by the user. Similarly, the input DTM remains open, and current area of interest values in force, until either a new DTM is selected or the program is terminated.

If the ENABLE EARTH_CURVATURE command has been given, compensation for earth curvature and terrestrial refraction will be made. Similarly if the ENABLE CULTURE command has been given, the program will take into account the height of any intervening surface features when determining visibility.

The generated cover map may be displayed on a graphics device as a colour coded image using the DISPLAY command.

Messages:

The following error messages are specific to the GO command:

*** ERROR *** Specifying command GO
The input DTM is undefined

*** ERROR *** Specifying command GO
No observer has been selected using the DEFINE OBSERVER command

*** ERROR *** Specifying command GO
output visibility matrix has not been defined

Example:

```
COVER>ENABLE DIAGNOSTICS<CR>
COVER>GO<CR>

+------------------------------------------------------------+
|                                                            |
|               Determining areas visible to                 |
|                       Observer  1                          |
|                                                            |
+------------------------------------------------------------+

Observer 1   Visible  13%   Not Visible  32%   Outside  55%

COVER>
```

HEIGHT

Defines the height **above the terrain surface** of the observer currently selected by means of the DEFINE OBSERVER command.

FORMAT: **HEIGHT height**

Command parameters:

height

The height of the observer above the ground in metres. A real (floating point) value is required.

DESCRIPTION:

HEIGHT allows the observer currently selected by the DEFINE OBSERVER command to be positioned above the terrain surface.

The command is interpreted differently depending on whether the observer is positioned within or outside the DTM area.
If the observer has been positioned within the geographical bounds of the DTM using the POSITION command, the value supplied with the HEIGHT command is added to the terrain surface height at the observer position.
If the observer is positioned outside the geographical bounds of the DTM, the command defines the height of the observer above sea-level.

The SHOW OBSERVERS command should be used to check that the height of an observer has been correctly defined.

Messages:

The following error messages are specific to the HEIGHT command:

```
*** ERROR *** Specifying command HEIGHT
No observer has been selected using the DEFINE OBSERVER command
```

```
*** ERROR *** Specifying command HEIGHT
Command requires 1 real argument
```

Examples:

COVER>**DEFINE OBSERVER 1<CR>**
COVER>**POSITION 100 100<CR>**
COVER>**SHOW OBSERVERS<CR>**

```
Current observer settings (Units are DTI Matrix Values)

Observer       Position          Height    Direction    Cone    Distance
```

```
  1          100      100          50.3         0.0      360.0        0.0
```

COVER>**HEIGHT 2.6<CR>**
COVER>**SHOW OBSERVERS<CR>**

```
Current observer settings (Units are DTI Matrix Values)

Observer       Position          Height    Direction    Cone    Distance
  1          100      100          52.9         0.0      360.0        0.0
```

COVER>**POSITION -10 -10<CR>**
COVER>**HEIGHT 100.7<CR>**
COVER>**SHOW OBSERVERS<CR>**

```
Current observer settings (Units are DTI Matrix Values)

Observer       Position          Height    Direction    Cone    Distance
  1          -10      -10         100.7         0.0      360.0        0.0

COVER>
```

HELP

Invokes help on COVER commands.

FORMAT: **HELP [command]**

Command parameters:

command

the command on which help is required

DESCRIPTION:

A brief description is given of the function and format of the specified command.
If no parameter is supplied then a list of all commands available is given.

Messages: None.

Examples:

COVER>**HELP DISTANCE**
COVER>

POINT

Defines the position of a single target point and computes the visibility of the point to all currently selected observers.

FORMAT: **POINT x_coordinate y_coordinate**

Command parameters:

x_coordinate

The x coordinate position of the target point.

y_coordinate

The y coordinate position of the target point.

The format of the command parameters is controlled by the UNITS command.

UNITS MATRIX - 2 integer values are required defining the position of the point in terms of a column and row number.

UNITS METRES - 2 real (floating point) values are required defining the position of the point in metre values. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates (eg. U.K. National Grid coordinates); if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

UNITS SECONDS - 2 real (floating point) values are required defining the absolute position of the point in seconds of arc. The values are supplied in the order latitude followed by longitude. A negative latitude value indicates a position in the Southern Hemisphere, and a negative longitude value a point to the west of Greenwich.

UNITS LATLONG - 2 values are required defining the absolute latitude and longitude position of the point in the format DDD MM SS H where DDD is the number of degrees, MM is the number of minutes, SS is the number of seconds, and H is the hemisphere (N,S,E,W). The values are supplied in the order latitude followed by longitude.

UNITS PROJECTION - 2 real (floating point) values defining the point in projection record units. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates; if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

Note that in all cases, the input values are adjusted to the nearest column and row values.

DESCRIPTION:

The POINT command is used to define the position, and compute the visibility of a single target point.

On giving the command, the program immediately computes the visibility of this point to all currently selected observers, and prints a summary on the terminal. The summary tabulates to how many and to which observers the point is visible or hidden.

The target point must lie inside the geographical bounds of the DTM.

If a digitising table is available, and a map has been registered to the DTM using the SETUP MAP command, puck **Button A** may be used to digitise the position of the point.

If the ENABLE EARTH_CURVATURE command has been given, compensation for earth curvature and terrestrial refraction will be made. Similarly if the ENABLE CULTURE command has been given, the program will take into account the height of any intervening surface features when determining the visibility of the point.

The command is used to determine visibility along a simple line of sight. It may also be used to determine visibility along a linear feature, such as a road, if the command is repeatedly given to digitise a series of points along the map feature.

Messages:

The following error messages are specific to the POINT command:

```
*** ERROR *** Specifying command POINT from the puck button
No map set up has been performed

*** ERROR *** Specifying command POINT
The input DTM is undefined

*** ERROR *** Specifying command POINT
Point must lie within DTM bounds

*** ERROR *** Specifying command POINT
No observer has been selected using the DEFINE OBSERVER command

*** ERROR *** Specifying command POINT
Command requires 2 x y coordinate values

*** ERROR *** Specifying command POINT
Latitude and longitude values supplied in wrong format
```

Examples:

```
COVER>DEFINE OBSERVER 1<CR>
COVER>POSITION 10 10<CR>
COVER>DEFINE OBSERVER 2<CR>
COVER>POSITION 70 50<CR>
COVER>POINT 32 45<CR>

Point at position 32 45 (height 345) is:
    Visible to 1 observer (observer: 1)
NOT Visible to 1 observer (observer: 2)

COVER>
```

POSITION

Defines the geographical position of the observer currently selected by means of the DEFINE OBSERVER command.

FORMAT: **POSITION x_coordinate y_coordinate**

Command parameters:

x_coordinate

The x coordinate position of the observer.

y_coordinate

The y coordinate position of the observer.

The format of the command parameters is controlled by the UNITS command.

UNITS MATRIX - 2 integer values are required defining the position of the observer in terms of a column and row number.

UNITS METRES - 2 real (floating point) values are required defining the position of the observer in metre values. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates (eg. U.K. National Grid coordinates); if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

UNITS SECONDS - 2 real (floating point) values are required defining the absolute position of the observer in seconds of arc. The values are supplied in the order latitude followed by longitude. A negative latitude value indicates a position in the Southern Hemisphere, and a negative longitude value a point to the west of Greenwich.

UNITS LATLONG - 2 values are required defining the absolute latitude and longitude position of the observer in the format DDD MM SS H where DDD is the number of degrees, MM is the number of minutes, SS is the number of seconds, and H is the hemisphere (N,S,E,W). The values are supplied in the order latitude followed by longitude.

UNITS PROJECTION - 2 real (floating point) values defining the position of the observer in projection record units. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates; if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

Note that in all cases, the input values are adjusted to the nearest column and row values.

DESCRIPTION:

The POSITION command is used to define the geographical location of the observer currently selected by means of the DEFINE OBSERVER command.

An observer may be positioned inside or outside the geographical bounds of the DTM.
If the observer is positioned inside the DTM, the height of the surface at the observer position is determined from the DTM. The HEIGHT command may be used, if you wish to position the observer a given distance above the terrain surface. If the observer is positioned outside the DTM, the height of the terrain surface cannot be determined, and you should use the HEIGHT command to set the absolute height of the observer.

If a digitising table is available, and a map has been registered to the DTM using the SETUP MAP command, puck **button 0** may be used to digitise the position of the observer. When using this option, the observer may only be positioned within the geographical bounds of the DTM.

Messages:

The following error messages are specific to POSITION command:

*** ERROR *** Specifying command POSITION using puck button
No map set up has been performed

*** ERROR *** Specifying command POSITION using puck button
Digitised point must lie within DTM bounds

*** ERROR *** Specifying command POSITION
The input DTM is undefined

*** ERROR *** Specifying command POSITION
No observer has been selected using the DEFINE OBSERVER command

*** ERROR *** Specifying command POSITION
Command requires 2 x y coordinate values

*** ERROR *** Specifying command POSITION
Latitude and longitude values supplied in wrong format

Examples:

COVER>**DEFINE OBSERVER 1<CR>**
COVER>**POSITION 100 100<CR>**
COVER>**SHOW OBSERVERS<CR>**

Current observer settings (Units are DTI Matrix Values)

Observer	Position		Height	Direction	Cone	Distance
1	100	100	50.3	0.0	360.0	0.0

COVER>**UNITS LATLONG<CR>**

```
COVER>DEFINE OBSERVER 1<CR>
COVER>POSITION 42 00 00N 2 58 40E<CR>
COVER>SHOW OBSERVERS<CR>

Current observer settings (Units are Degrees, Minutes, Seconds)

Observer        Position          Height    Direction    Cone    Distance
  1       42 00 00N  2 58 40E     73.2         0.0      360.0       0.0

COVER>
```

REDISPLAY

Displays information contained in a currently opened cover map in a colour coded form on a graphics display.

FORMAT: **REDISPLAY**

Command parameters: None

DESCRIPTION:

The REDISPLAY command is used to display information contained in a currently opened cover map in colour coded form on a graphics screen.

The command acts in the same way as the DISPLAY command. For example, only information relating to those observers currently selected is displayed on the graphics screen. Unlike the DISPLAY command however, REDISPLAY does not require a cover map file specification. REDISPLAY should therefore be used in situations where a series of images are being generated from the same cover map eg. when displaying the visibility pattern of each observer in turn.

In the example below, DISPLAY is used initially to open and display the information in a cover map, and REDISPLAY is subsequently used to display data from the same cover map.

Messages:

The following error message is specific to the REDISPLAY command:

*** ERROR *** Specifying command REDISPLAY
No cover map is currently opened for display

Example:

COVER>**DISPLAY TEST <CR>**
COVER>**SELECT OBSERVERS 1 3<CR>**
COVER>**REDISPLAY <CR>**
COVER>

SELECT ALL

The command is used to select all observers when generating a cover map or line of sight, or when displaying a cover map on a graphics device.

FORMAT: **SELECT ALL**

Command parameters: None

DESCRIPTION:

The SELECT ALL command is used to select all observers when generating a cover map using the commands GO or CREATE, or when computing a line of sight using the POINT command. The command also ensures that information relating to all observers is included when displaying a cover map on a graphics device using the DISPLAY or REDISPLAY commands, and when decoding information using the WRITE command.
The command overrides any observer selections made using the SELECT OBSERVER command.

Current observer selections may be examined using the command SHOW OBSERVERS.

Messages:

The following error message is specific to the SELECT command:

```
*** ERROR *** Specifying command SELECT
Command qualifiers are OBSERVER or ALL
```

Examples:

```
COVER>SELECT ALL<CR>
COVER>
```

SELECT OBSERVERS

Selects which observers are to be used when generating a cover map or computing visibility along a line of sight, or when displaying a cover map on a graphics device.

FORMAT: **SELECT OBSERVERS observer_number [...]**

Command parameters:

observer_number

A list of observer numbers consisting of between 1 and 7 integer values. The integer values should be in the range 1 to 8.

DESCRIPTION:

The SELECT OBSERVER command controls which observers are used when generating a cover map using the commands GO or CREATE, or computing a line of sight using the POINT command. The command also controls whether intervisibility information relating to a particular observer is included when displaying a cover map on a graphics device using the DISPLAY or REDISPLAY commands, and what information is decoded when the WRITE command is given.

Up to 7 observers may be specified in the list.

The SELECT ALL command should be used if no restrictions are to be placed on which observers are selected.

Current observer selection may be examined using the command SHOW OBSERVERS.

Messages:

The following error messages are specific to the SELECT and SELECT OBSERVERS commands:

```
*** ERROR *** Specifying command SELECT
Command qualifiers are OBSERVER or ALL

*** ERROR *** Specifying command SELECT OBSERVER
Command requires at least 1 integer argument

*** ERROR *** Specifying command SELECT OBSERVER
Integer values should be in the range 1 to 8
```

Examples:

This example assumes viewing parameters for 3 observers (observers 1 2 and 4) have been entered.

COVER>**SELECT OBSERVERS 2 4 <CR>**
COVER>**SHOW OBSERVERS<CR>**

Current observer settings (Units are DTI Matrix Values)

Observer	Position		Height	Direction	Cone	Distance
2	130	140	70.6	30.0	60.0	120.5
4	114	117	63.2	245.0	60.0	145.2

COVER>**SELECT ALL <CR>**
COVER>**SHOW OBSERVERS<CR>**

Current observer settings (Units are DTI Matrix Values)

Observer	Position		Height	Direction	Cone	Distance
1	100	100	50.3	0.0	360.0	0.0
2	130	140	70.6	30.0	60.0	120.5
4	114	117	63.2	245.0	60.0	145.2

COVER>

SET OFFSET

Registers a DTI file containing data on the height of surface features, to the input DTM.

FORMAT: **SET OFFSET x_offset y_offset**

Command parameters:

x_offset

The x offset specified as an integer column value. It defines the column in the input DTM that corresponds to column 1 in the DTI file.

y_offset

The y offset specified as an integer row value. It defines the row in the input DTM that corresponds to row 1 in the DTI file.

DESCRIPTION:

The SET OFFSET command is used to register a DTI file containing data on the height of surface features, to the input DTM.

By default the SW corner of the DTI file is registered to the SW corner of the DTM. The use of the command is therefore only necessary in situations were the SW corner of the two files do not represent the same geographical location.
The SET OFFSET command should only be given after the file containing the surface feature heights has been read in, either when opening the input DTM or by means of the ENABLE CULTURE command.

The SHOW SETTINGS command may be used to examine the current OFFSET values.

Messages:

The following error messages are specific to the SET and SET OFFSET commands:

```
*** ERROR *** Specifying command SET
Command qualifiers are OFFSET or REFRACTION_COEFFICIENT

*** ERROR *** Specifying command SET OFFSET
Command requires 2 integer arguments

*** ERROR *** Specifying command SET OFFSET
Column and row values may not exceed DTM bounds
```

Examples:

In this example, the SW corner of the DTI file containing surface feature height values is registered to column 50 and row 50 of the input DTM.

```
COVER>SET OFFSET 50 50<CR>
COVER>
```

SET REFRACTION_COEFFICIENT

Specifies the refraction coefficient value that will be used by the program, when calculating the effect of terrestrial refraction along a line of sight.

FORMAT: **SET REFRACTION_COEFFICIENT coefficient**

Command parameters:

coefficient

The coefficient of refraction specified as a real (floating point) value.

DESCRIPTION:

This command allows the refraction coefficient value that is used when calculating the effect of terrestrial refraction on a line of sight, to be redefined. The supplied value is substituted in the equation:

r = (d*d) * k/2R

where r is the effect of refraction (r), d is the distance between the observer and the target point in kilometres, R is the radius of the earth, and k is the coefficient of refraction.

By default COVER uses a coefficient value based on a 4/3 Earth of 0.125

The effect of refraction is only considered in intervisibility calculations if the ENABLE EARTH_CURVATURE command has been given.

The current value of the coefficient may be examined using the command SHOW SETTINGS.

Messages:

The following error messages are specific to the SET and SET REFRACTION_COEFFICIENT commands:

*** ERROR *** Specifying command SET
Command qualifiers are OFFSET or REFRACTION_COEFFICIENT

*** ERROR *** Specifying command SET REFRACTION_COEFFICIENT
Command requires 1 real argument

Examples:

```
COVER>SET REFRACTION_COEFFICIENT 0.3<CR>
COVER>
```

SETUP MAP

Registers a map or other source document placed on a digitising table, to the input DTM.

FORMAT: **SETUP MAP**

DESCRIPTION:

The SETUP MAP command allows a source document to be registered to the input DTM.

In order to register the map and a DTM, the DTM should have first been specified, and a source document should have been securely attached to the surface of a digitising table.

On giving the command you will be asked to digitise 4 rectangular registration points. The registration points represent 4 points on the map that correspond to the 4 corners of the DTM. The points are digitised in the order top left (NW), bottom left (SW), bottom right (SE) and top right (NE) using any button on the table puck. A point should be digitised in response to a prompt on the terminal.
An error message is generated if any of the angles of the digitised rectangle are less than 88 degrees, or greater than 92 degrees (ie. if the corner points of the rectangle are more than 2 degrees off rectangular). In this case you will be asked to redigitise the 4 registration points.
Setup of the map may be aborted using <CTRL/Z> (pressing the Ctrl and Z keys together), or by pressing **Button F** on the table puck.

Following the registration of a map to the DTM, coordinate values required by the WINDOW, POSITION, TARGET and POINT commands, may be input using an appropriate button on the table puck.

Messages:

The following error messages are specific to the SETUP and SETUP MAP commands:

*** ERROR *** Specifying command SETUP
Command qualifier is MAP

*** ERROR *** Specifying command SETUP MAP
Command is invalid if the table has not been initialised

*** ERROR *** Specifying command SETUP MAP
The input DTM must be specified before SETUP MAP

*** ERROR *** Setting up MAP
Badly digitised corner points; try again

Examples:

COVER>**SETUP MAP<CR>**

Digitise map NW corner>

Digitise map SW corner>

Digitise map SE corner>

Digitise map NE corner>

COVER>

SHOW ENABLE

Shows the current status of those options that may be enabled by means of the ENABLE command, or disabled using the DISABLE command.

FORMAT: **SHOW ENABLE**

Command parameters: None.

DESCRIPTION:

Displays the current status of all the COVER options that may be enabled or disabled using the ENABLE and DISABLE commands.
The name of the option is shown, followed by either the word ON or OFF to indicate its current status.
If the command SHOW ENABLE is used before any ENABLE or DISABLE commands have been given, the default status of the options is displayed.

Messages:

The following message is specific to the SHOW command:

*** ERROR *** Specifying command SHOW
Command qualifiers are ENABLE, FILES, OBSERVERS, SETTINGS or TARGETS

Examples:

COVER>**SHOW ENABLE<CR>**

Current status:

ABSOLUTE	On	ADD_TARGET	Off	CULTURE	Off
DIAGNOSTICS	On	EARTH_CURVATURE	Off	TABLE	Off

COVER>

SHOW FILES

Displays information extracted from the header of the input files and on the output file.

FORMAT: **SHOW FILES**

Command parameters: None.

DESCRIPTION:

Details extracted from the header of the DTM are displayed on the terminal, along with details of the current window or area of interest.

The header values are shown in the current units of measurement. This is dependent on the header type of the input file, or may be set explicitly using the UNITS command.

Information on the output visibility file selected with the FILEOUT command and the culture file selected with the ENABLE CULTURE command is also output.

Messages:

The following messages are specific to the commands SHOW and SHOW FILES:

```
*** ERROR *** Specifying command SHOW
Command qualifiers are ENABLE, FILES, OBSERVERS, SETTINGS or TARGETS
```

```
*** WARNING *** The input DTM is undefined
```

Examples:

COVER>**SHOW FILES<CR>**

```
LSL$DTI:TEST.DTI
Header: LSLA     Data: WORD

Units are DTI Matrix Values

Matrix Coverage    SW:    1     1    NE:   301   251
Matrix Window      SW:    1     1    NE:   200   200
Matrix Interval     E:    1           N:     1
Value Range          :      0   to    851

File LSL$DTI:COVER_TEST.VIS selected for output.
COVER>
```

SHOW OBSERVERS

Displays information about all observers that are currently selected.

FORMAT: **SHOW OBSERVERS**

Command parameters: None.

DESCRIPTION:

The SHOW OBSERVERS command is used to display information about currently selected observers at the terminal.

Details of the coordinate position, height and field of view of each observer, are displayed.
The information is shown in the current units of measurement. This is dependent on the header type of the input DTM, or may be set explicitly using the UNITS command.

Messages:

The following messages are specific to the commands SHOW and SHOW OBSERVERS

*** ERROR *** Specifying command SHOW
Command qualifiers are ENABLE, FILES, OBSERVERS, SETTINGS or TARGETS

*** WARNING *** No observers are currently selected

Examples:

COVER>**SHOW OBSERVERS<CR>**

```
Current observer settings (Units are DTI Matrix Values)

Observer       Position          Height    Direction    Cone     Distance
  1          100       100        50.3         0.0     360.0          0.0
  2          130       140        70.6        30.0      60.0        120.5
  4          114       117        63.2       245.0      60.0        145.2

COVER>
```

SHOW SETTINGS

Shows the current value of those parameters that may be set using the SET command.

FORMAT: **SHOW SETTINGS**

Command parameters: None.

DESCRIPTION:

Displays the name of all COVER parameters that may be set using the SET command, and their current values.
If the SHOW SETTINGS command is given before using a SET command, the values shown are the default values that are allocated to the parameters by COVER.

Messages:

The following message is specific to the command SHOW:

*** ERROR *** Specifying command SHOW
Command qualifiers are ENABLE, FILES, OBSERVERS, SETTINGS or TARGETS

Examples:

COVER>**SHOW SETTINGS<CR>**

Current values:

OFFSET   1   1   REFRACTION_COEFFICIENT 0.125

COVER>

SHOW TARGETS

Displays the position of the target viewing point associated with each currently selected observer.

FORMAT: **SHOW TARGETS**

Command parameters: None.

DESCRIPTION:

This command is used to display at the terminal, the target view point associated with each currently selected observer.

Details are shown in the current units of measurement. This is dependent on the header type of the input DTM, or may be set explicitly using the UNITS command.

Messages:

The following messages are specific to the commands SHOW and SHOW TARGETS

*** ERROR *** Specifying command SHOW
Command qualifiers are ENABLE, FILES, OBSERVERS, SETTINGS or TARGETS

*** WARNING *** No observers are currently selected

Examples:

COVER>**SHOW TARGETS<CR>**

```
Current Target Points (Units are DTI Matrix Values)

Observer      Target View Point

  1              130      160
  2               70      325

COVER>
```

SPAWN

The SPAWN command enables a subprocess to be created from within COVER.

FORMAT: SPAWN command-line

Command parameters:

command-line

Specifies a DCL command string to be executed as if typed in response to a '$' prompt. When the command completes, the subprocess terminates and control is returned to COVER. The command line cannot exceed 80 characters.

DESCRIPTION:

The SPAWN command enables you to create a subprocess while within COVER. When the subprocess terminates, control is returned to COVER.

Messages:

The following error messages are specific to the SPAWN command:

```
*** ERROR *** Specifying command SPAWN
Command requires a valid DCL command line

*** ERROR *** Unable to spawn command, returning to COVER
```

Examples:

```
COVER> SPAWN DIRECTORY LSL$DTI:*.DTI<CR>

Directory DUA3:[LSL.DTI]

TEST1.DTI;1                 8/8       18-AUG-1987 07:56  [LSL,DAVEC]
TEST2.DTI;1                 7/8       18-AUG-1987 17:17  [LSL,DAVEC]
TEST2.DTI;2                 7/8       18-AUG-1987 17:34  [LSL,DAVEC]

Total of 3 files, 22/24 blocks.

COVER>
```

TARGET

Defines the target view point of the observer currently selected by means of the DEFINE OBSERVER command.

FORMAT: TARGET x_coordinate y_coordinate

Command parameters:

x_coordinate

The x coordinate position of the target point.

y_coordinate

The y coordinate position of the target point.

The format of the command parameters is controlled by the UNITS command.

UNITS MATRIX - 2 integer values are required defining the position of the target point in terms of a column and row number.

UNITS METRES - 2 real (floating point) values are required defining the position of the target point in metre values. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates (eg. U.K. National Grid coordinates); if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

UNITS SECONDS - 2 real (floating point) values are required defining the absolute position of the target point in seconds of arc. The values are supplied in the order latitude followed by longitude. A negative latitude value indicates a position in the Southern Hemisphere, and a negative longitude value a point to the west of Greenwich.

UNITS LATLONG - 2 values are required defining the absolute latitude and longitude position of the target point in the format DDD MM SS H where DDD is the number of degrees, MM is the number of minutes, SS is the number of seconds, and H is the hemisphere (N,S,E,W). The values are supplied in the order latitude followed by longitude.

UNITS PROJECTION - 2 real (floating point) values defining the position of the target point in in projection record units. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates; if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

Note that in all cases, the input values are adjusted to the nearest column and row values.

DESCRIPTION:

Defines the position of the target view point of the observer currently selected by means of the DEFINE OBSERVER command.

The TARGET command provides an alternative means to the DIRECTION and DISTANCE commands, of defining an observer's direction and distance of view.

If a cone of vision for the selected observer has not been previously defined when the TARGET command is given, or the current cone value is 360 degrees, a cone of vision of 90 degrees is applied to the observer.

The target point must be inside the geographical bounds of the DTM, but may lie outside the current area of interest defined by means of the WINDOW command.

Note that once defined, the target position is applied by default to subsequent observers when a further DEFINE OBSERVER command is given.

If a digitising table is available, and a map has been registered to the DTM using the SETUP MAP command, puck **button 9** may be used to digitise the position of the target point.

The use of the commands SHOW TARGETS and SHOW OBSERVERS is recommended to confirm the correct definition of target position, and observer field of view parameters.

Messages:

The following error messages are specific to TARGET command:

```
*** ERROR *** Specifying command TARGET using puck button
No map set up has been performed

*** ERROR *** Specifying command TARGET using puck button
Digitised point must lie within DTM bounds

*** ERROR *** Specifying command TARGET
The input DTM is undefined

*** ERROR *** Specifying command TARGET
No observer has been selected using the DEFINE OBSERVER command

*** ERROR *** Specifying command TARGET
Command requires 2 real arguments

*** ERROR *** Specifying command TARGET
Latitude and longitude values supplied in wrong format
```

Examples:

```
COVER>UNITS MATRIX<CR>
COVER>DEFINE OBSERVER 1<CR>
COVER>POSITION 10 10<CR>
```

```
COVER>TARGET 400 300<CR>
COVER>SHOW OBSERVERS<CR>

Current observer settings (Units are DTI Matrix Values)

Observer       Position           Height    Direction    Cone    Distance
  1          10        10          50.3         56.4     90.0      486.0

COVER>SHOW TARGETS<CR>

Current Target Points (Units are DTI Matrix Values)

Observer      Target View Point

  1             400      300

COVER>
```

UNITS

Specifies the units of measurement that will be used when defining an area of interest in the input DTM, or defining the position of an observer or target point. The command also controls the units of measurement which will be used when displaying file header details or observer viewing parameters.

FORMAT: UNITS units

Command parameters:

units

A keyword defining the measurement units, chosen from:

MATRIX	Matrix grid interval units, i.e rows and columns
METRES	Metres on the ground
LATLONG	Latitude and Longitude (in degrees, minutes and seconds)
SECONDS	Seconds of arc
PROJECTION	Projection Record Units (eg. mms on the source document)

DESCRIPTION:

The UNITS command defines in which units of measurement arguments to the DISTANCE, POINT, POSITION and TARGET commands are specified, and the units of measurement used when defining an area of interest in the input DTM by means of the WINDOW command.
The command also controls in what format details from the header of the DTM, and observer and target details are displayed, when the SHOW FILES, SHOW OBSERVERS and SHOW TARGET commands are given.

The command should be given after defining the input DTM since an appropriate default units of measurement is set up when the file is opened. If the DTM file is of header type TED4 or UHL1 (ie DTED files), then the default is latitude and longitude specified in degrees, minutes and seconds; for all other header types, matrix units (ie. columns and rows) are assumed.

Messages:

The following error messages are specific to the UNITS command:

*** ERROR *** Specifying command UNITS
Command qualifiers are MATRIX, METRES, PROJECTION, SECONDS or LATLONG

*** ERROR *** Specifying command UNITS
Command qualifier is invalid for the input file

Examples:

```
COVER> UNITS MATRIX<CR>
COVER>
```

WAIT

The WAIT command causes processing to be suspended for a specified number of seconds.

FORMAT: WAIT seconds

Command parameters:

seconds

The number of seconds for which processing is to be suspended. A real (floating point) value is required.

DESCRIPTION:

The WAIT command causes processing to be suspended for a specified number of seconds. It is designed for use in software demonstration situations and is of no value in a production flowline.

Messages:

The following error message is specific to the WAIT command:

*** ERROR *** Specifying command WAIT
Command requires a real argument

Examples:

COVER> **WAIT 4.0<CR>**
COVER>

WINDOW

Specifies an area of interest in the input DTM.

FORMAT: **WINDOW xmin ymin xmax ymax**

Command parameters:

xmin ymin
The coordinates of the bottom left hand corner of the defining rectangle.

xmax ymax
The coordinates of top right hand corner of the defining rectangle.

The format of the command parameters is controlled by the UNITS command.

UNITS MATRIX Requires 4 integer values defining the rectangle in terms of column and row numbers

UNITS METRES Requires 4 real (floating point) values defining the rectangle in metre values.

UNITS SECONDS Requires 4 real (floating point) values defining the absolute position of the rectangle in seconds of arc. The values are supplied in the order latitude followed by longitude for the SW corner, and latitude followed by longitude for the NE corner. A negative latitude value indicates a position in the Southern Hemisphere, and a negative longitude value a point to the east of Greenwich.

UNITS LATLONG Requires 4 values defining the absolute latitude and longitude position of the rectangle in the format DDD MM SS H where DDD is the number of degrees, MM is the number of minutes, SS is the number of seconds, and H is the hemisphere (N,S,E,W). The values are supplied in the order latitude followed by longitude for the SW corner, and latitude followed by longitude for the NE corner.

UNITS PROJECTION Requires 4 real (floating point) values defining the rectangle in projection record units. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates; if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

Note that in all cases, the input values are adjusted to the nearest column and row values.

DESCRIPTION:

The WINDOW command is used limit intervisibility calculations to a particular rectangular geographical area. Only nodes in the DTM that lie within this area are used in line of sight calculations.
The area of interest should lie within the geographical bounds of the DTM.

The command also determines the number of columns and number of rows that will be present in the output cover map.

If the logical name LSL$MATRIX_DEFAULT_WINDOW is defined with a value of "1", then a default area of interest defined in matrix units of bottom left hand corner 1,1 and top right hand corner 200,200 is set when the file is opened. If the logical name is absent or has any other value, or if the input DTI file has fewer than 200 columns or 200 rows, then a default window equivalent to the total matrix is set.

If a digitising table is available, puck **Button B** may be used to give the command WINDOW. When the button is pressed, you will be asked to digitise 2 further points inside the map area, defining the SW (bottom lefthand) and NE (top righthand) corners of the rectangle. Definition of the area of interest may be abandoned by pressing puck **Button F**.

Messages:

The following messages are specific to the WINDOW command:

```
*** ERROR *** Specifying command WINDOW
The input DTM must be specified before WINDOW

*** ERROR *** Specifying command WINDOW using puck button
No map set up has been performed

*** ERROR *** Specifying command WINDOW
Command requires 4 arguments

*** ERROR *** Specifying command WINDOW
NE corner values must exceed SW corner values

*** ERROR *** Specifying command WINDOW
Supplied values exceed matrix extents

*** ERROR *** Specifying command WINDOW
Latitude and longitude values supplied in wrong format
```

Examples:

```
COVER>WINDOW 1 1 200 200<CR>
COVER>UNITS LATLONG<CR>
COVER>WINDOW 42 00 00N 3 00 00E 42 30 00N 2 58 40E
COVER>
```

WRITE

Generates an output DTI file containing visibility data decoded from a cover map.

FORMAT: **WRITE infile-spec outfile-spec**

Command parameters:

infile-spec

The file specification of the input cover map. This will be a file that has been created during an earlier run of COVER.
Any part of the file specification not supplied will be taken from the default 'LSL$DTI:DTI.VIS'
If no infile-spec is supplied, you will be asked to supply one in response to the prompt **Input cover map filename>**

outfile-spec

The file specification of an output DTI file which will be created to hold the decoded visibility information.
Any part of the file specification not supplied will be taken from the default 'LSL$DTI:DTI.DTI'
If no infile-spec os supplied, you will be asked to supply one in response to the prompt **Output DTI filename>**

DESCRIPTION:

The WRITE command creates an output DTI file containing visibility information decoded from a cover map.

Visibility information in a cover map is held in a bitwise manner. Each bit of a data value is related to a single observer; a bit is set or unset depending on whether a node is visible or hidden from view of an observer. The header of a cover map holds position and field of view parameters for each observer.
These data and header values are decoded when the DISPLAY or REDISPLAY commands are given in order to produce a colour coded image showing which parts of the terrain are visible to an observer or observers, and which parts of the terrain lie outside their field of view. The information is read only by the module COVER; other modules within the TVES package are unable to correctly interpret the data values in a cover map.

The WRITE command is provided to allow the visibility data to be stored in a DTI file in a decoded form. In this form, these data can be processed through other modules such as VECTORISE and ROVER.
The data values written to the output DTI file indicate as on the graphics screen, how many observers can see a particular area of terrain. A value of 0 indicates the node is visible to no observers; a value of 1 that it is visible to 1 observer and so on. A value of -1 indicates that the node is outside the cone of vision of all observers.

The SELECT OBSERVERS command is used to control what information is decoded from the cover map. Only information related to the selected observers is used.

Messages: None

Examples:

```
COVER>WRITE TEST_COVER TEST_OUTPUT<CR>
COVER>
```

EXAMPLE COVER SESSION

```
$ COVER

COVER> filein test

LSL$DTI:TEST.DTI
Header: LSLA     Data: WORD

Units are Metres

Matrix Coverage    SW:   440000    80000    NE:   460000   100000
Matrix Window      SW:   440000    80000    NE:   460000   100000
Matrix Interval     E:   50           N:    50
Value Range          :      0   to    190

COVER> units matrix
COVER> window 50 50 150 100
COVER> define observer 1
COVER> position 50 50
COVER> show observers

Current observer settings (Units are DTI Matrix Values)

Observer       Position          Height    Direction    Cone    Distance
  1          50        50         50.3        0.0       360.0      0.0

COVER> define observer 2
COVER> pos -10 -10
COVER> height 32.5
COVER> cone 60
COVER> target 130 130
COVER> show observers

Current observer settings (Units are DTI Matrix Values)

Observer       Position          Height    Direction    Cone    Distance
  1          50        50         50.3        0.0      360.0      0.0
  2         -10       -10         32.5       45.0       60.0    197.9

COVER> fileout test
File LSL$DTI:test.vis  selected for output.
COVER> go

+------------------------------------------------------------+
|                                                            |
|                Determining areas visible to                |
|                        Observer  1                         |
|                                                            |
+------------------------------------------------------------+

Observer 1   Visible  33%   Not Visible  67%   Outside   0%
```

```
+------------------------------------------------------------+
|                                                            |
|                Determining areas visible to                |
|                        Observer  2                         |
|                                                            |
+------------------------------------------------------------+

Observer 2   Visible  14%   Not Visible  22%   Outside  64%

COVER> exit
```

In this example, file LSL$DTI:TEST.DTI is selected as the input DTM. The UNITS MATRIX and WINDOW commands have been used to define an area of interest of 100 columns and 50 rows. Only the visibility of DTM nodes inside the specified rectangle will be computed.

The position and field of view of 2 observers is defined.

In the case of observer 1, only a position command has been given. The default cone of vision (360 degrees) and distance (infinity) values will therefore be used. The observer has been positioned inside the DTM, so that the height of the surface at the observation point is automatically derived from the DTM.

Observer 2 has been placed outside the DTM. The height command is used to set the height at the observation point. The cone command is used in conjunction with the target command to restrict the observer's field of view. Only the visibility of DTM nodes that lie within this field of view will be determined for observer 2.

The FILEOUT command is used to specify the filename of the output cover map, and the GO command is used to initiate intervisibility calculation.

MESSAGES (OTHER)

In addition to messages which are generated by the program itself, other messages may be produced by Laser-Scan libraries. In particular, messages may be generated the DTI library, and by the Laser-Scan I/O library, LSLLIB.

DTI library messages are introduced by '%DTI', and are documented in the DTILIB Reference Manual. In all cases the messages indicate a fatal error, that will cause processing to halt.

LSLLIB messages are introduced by '%LSLLIB' and are generally self-explanatory. Such messages rarely indicate a fatal error, and are generated most frequently by entering a command in an invalid format in response to the COVER prompt.

CHAPTER 3

MODULE DTICHECK

MODULE DTICHECK

REPLACES None. DTICHECK is a new TVES utility.

FUNCTION

DTICHECK is a utility to automatically check a DTI file on a column by column basis, for:

- o critical slopes defined by a user supplied threshold Z difference
- o DTM posts having zero Z
- o DTM posts having a negative Z
- o DTM posts having a null Z

FORMAT

$ DTICHECK

COMMAND QUALIFIERS

None, DTICHECK is command driven.

DTICHECK command defaults.

On program startup, the following command defaults apply:

ABSOLUTE - enabled.

DIAGNOSTICS - disabled.

NULL - once a DTI file is specified the null will have one of the following:-

0 if BYTE type.
-32767 if WORD type.
80000000 Hex if LONG type.
-1.0E-38 if REAL type.

PME - disabled.

UNITS - once a DTI file is specified, appropriate default units are set up for the file.

WINDOW - The entire matrix area.

DESCRIPTION

General

DTICHECK is a utility to automatically check a DTI file on a column by column basis, for:

- o critical slopes defined by a user supplied threshold Z difference
- o DTM posts having zero Z
- o DTM posts having a negative Z
- o DTM posts having a null Z

These checks may be specified singly or in any combination. Output may be directed to three different file types:

- o IFF plot file,
- o LITES2 command file,
- o ROVER command file.

These output file options may be specified singly or in any combination.

If no output file option is selected, all output is directed to SYS$OUTPUT.

If diagnostic printout is enabled, the resulting messages are always sent to SYS$OUTPUT, regardless of output file selections.

DTICHECK reporting characteristics

If no output file option is selected, all checking reports are directed to SYS$OUTPUT. If an output file is selected checking reports will only be directed to SYS$OUTPUT if the ENABLE DIAGNOSTICS command is given.

If more than one check is enabled, DTICHECK scans the input DTI file separately for each check and issues appropriate reports.

The nature of these reports is determined by the check currently selected.

Listed below are the reports issued for ENABLE SLOPE, ENABLE MINUS, ENABLE NULL and ENABLE ZERO:

1. **CHECK SLOPE**

   Threshold Z difference exceeded at X: 'x-position' Y: 'yposition'

2. **CHECK MINUS**

   Negative value at X: 'x-position' Y: 'yposition'

3. **CHECK NULL**

   Null value at X: 'x-position' Y: 'yposition'

4. **CHECK ZERO** -

   Zero value at X: 'x-position' Y: 'yposition'

The incorporation of these reports in LITES2 and ROVER guidance command files is discussed below.

Where errors are reported for adjacent posts within the DTI file, run length encoded messages will be generated:

- **CHECK SLOPE**

  Threshold Z difference exceeded between X: 'x-position' Y: 'yposition' and X: 'x-position' Y: 'yposition'

- **CHECK MINUS**

  Negative values between X: 'x-position' Y: 'yposition' and X: 'x-position' Y: 'yposition'

- **CHECK NULL**

  Null value between X: 'x-position' Y: 'yposition' and X: 'x-position' Y: 'yposition'

- **CHECK ZERO**

  Zero value between X: 'x-position' Y: 'yposition' and X: 'x-position' Y: 'yposition'

As DTICHECK performs all checks on a column by column basis, these run length encoded messages will always have a constant X coordinate and differing start and stop Y coordinates.

DTICHECK and LITES2 command file

Positions within the LITES2 command file are expressed in appropriate units for the input DTI file (eg metres). If the file contains a projection record, or is a UHL1 or DTED type file then the origin offset is added to all the coordinates written to the LITES2 command file. The command file then subtracts the origin offset from the values to obtain the local coordinates of the positions while LITES2 is run.

Entries in the LITES2 command file are typically of the form:

```
%MESSAGE Null post value detected at X = 7650.0  Y = 4560.0
%LET DTICHECK_X_POS = 7650.0 - '$MDOFFSET1'
%LET DTICHECK_Y_POS = 4560.0 - '$MDOFFSET2'
%POSITION 'DTICHECK_X_POS' 'DTICHECK_Y_POS'
%TEST $CURSINWIN
%ELSE %ZOOM 1
%PING
%RESPOND
%ABANDON
%ABANDON
```

Where errors are reported for adjacent posts within the DTI file, a run length encoded message will be generated of the form:

```
%MESSAGE 350 Null post values detected between X = 7650.0  Y = 4560.0 and
X = 7650.0 Y = 5000.0
%LET DTICHECK_X_POS = 7650.0 - '$MDOFFSET1'
%LET DTICHECK_Y_POS = 4560.0 - '$MDOFFSET2'
%POSITION 'DTICHECK_X_POS' 'DTICHECK_Y_POS'
%TEST $CURSINWIN
%ELSE %ZOOM 1
%PING
%RESPOND
%ABANDON
%ABANDON
```

DTICHECK and ROVER command file

Positions within the ROVER command file are expressed in matrix units.

Entries in the ROVER command file are typically of the form:

```
MESSAGE Null post value detected at X = 765  Y = 456
MOVE 765    456
PING
RESPOND
```

Where errors are reported for adjacent posts within the DTI file, a run length encoded message will be generated of the form:

```
MESSAGE 350 Null post values detected between X = 765  Y = 456 and X = 765
Y = 500
MOVE 765    456
PING
RESPOND
```

DTICHECK IFF plot file characteristics

The IFF command enables specification of the IFF file which is to receive features indicating the positions of suspected data errors.

The layer and feature code allocations used in the IFF file may be specified by the user at run time using the SET command.

The IFF file will always contain two layers:

Layer 0, to contain registration marks located at the corners of the rectangle defined by the WINDOW command, or if no WINDOW command is specified, the corners of the whole DTI file area. The registration marks are L-shaped tick features having feature code 0.

The data layer. By default this is layer 1, but the user may specify an alternative data layer number using the SET LAYER command. The features in this layer define boxes which locate the posts for which errors are suspected. Note that a slope exceeding the threshold can be caused by one or both of the posts between which the slope is evaluated. The feature codes used in this layer are defined by the following defaults:

Slope feature code 1
Minus feature code 2
Null feature code 3
Zero feature code 4

Alternatively, the user may specify feature codes using the relevant SET commands.

Where errors are reported for adjacent posts within the DTI file a single IFF feature will be generated, the coordinates within this IFF feature comprise a rectangular box which is located around all posts in error.

The CP (Control Point) entry is set to reflect the rectangle defined by the WINDOW command, or if no WINDOW command is specified, the corners of the whole DTI file area.

The origin offset in the type 2 MD (Map Descriptor) entry will be set to the origin offset extracted from the DTI file header.

DTICHECK and null post handling

The presence of null posts within the DTI file is only reported if the user explicitly requests null post reporting with the ENABLE NULL command.

The null post value is set in accordance with the DTI file data type, unless explicitly set by the user with the NULL command argument.

There may be some conflict with the ENABLE ZERO option if the DTI file type is BYTE type, as the default null value for this is 0 (zero). To overcome this problem, null posts are excluded from all DTICHECK checks except the NULL check.

DTICHECK typical command sequence

Detailed descriptions of the individual commands available are given below.

A typical command sequence is:

```
FILEIN TESTDATA           - input file is LSL$DTI:TESTDATA.DTI
WINDOW 100 100 200 200    - area of interest in input file is defined
                            by SW 100x 100y and NE 200x 200y
SLOPE 8                   - sets threshold inter-post Z difference
ENABLE SLOPE              - report all posts which exceed SLOPE
                            tolerance Z difference
ENABLE NULL               - report all null posts
IFF CHECK_OVERLAY         - output error features to LSL$IF:CHECK_OVERLAY.IFF
SET LAYER 10              - put error features into layer 10 of the IFF file
SET SLOPE_FC 5            - give IFF slope error features, feature code 5
SET NULL_FC 2             - give IFF null post features, feature code 2
GO                        - start to process
```

This series of commands specifies that DTICHECK is to report all posts which have the null value and all posts which have Zs which differ from their neighbours by more than 8 Z units.

Output is to be to an IFF file; LSL$IF:CHECK_OVERLAY.IFF. This file will contain two layers, 0 (for registration marks located at the corners of the rectangle defined by the WINDOW command) and 10, defined by the SET LAYER command. Layer 10 contains the error location features. These have feature code 5 if they relate to slope errors or feature code 2 if null post locations.

DTICHECK commands

@

Take command input from the specified file.

FORMAT: @file-spec<CR>

Command parameters:

file-spec

The file to be opened and used for command input.

Any parts of the file-spec not supplied for the @ command will be taken from the default specification 'SYS$DISK:[].COM;0'.

DESCRIPTION:

DTICHECK offers the facility of command input from an indirect command file. The '@' character preceding a file-spec will cause DTICHECK to open and read commands from the specified file until:

1. a RETURN command is detected and command input is returned to SYS$COMMAND.

2. end-of-file is detected. This provokes an error message and command input is returned to SYS$COMMAND.

Nested command files are not supported (i.e. a command file containing an '@' command), although sequential '@' commands are supported when read from SYS$COMMAND.

As an aid to batch log interpretation DTICHECK will echo all commands read from an indirect command file.

Messages:

The following messages are specific to the @ command:

*** WARNING *** Indirect file error - returning to terminal input

*** ERROR *** Can't open indirect command file 'file-spec'

Examples:

```
$ DTICHECK<CR>
TVES module DTICHECK of 13:30:39 18-OCT-87
DTICHECK> @PRESETS<CR>
DTICHECK> ENABLE DIAGNOSTICS
DTICHECK> ENABLE SLOPE
```

```
DTICHECK> FILEIN OEEPE
DTICHECK> UNITS METRES
DTICHECK> WINDOW 1 1 101 51
DTICHECK> LITES2 DTICHECK
DTICHECK> GO
DTICHECK> RETURN
DTICHECK>
```

!

Treat all text to the right of the '!' as a comment.

FORMAT: ! [comment text]

Command parameters:

comment text

text that is to be treated as a comment and which will be excluded from command interpretation.

DESCRIPTION:

An exclamation mark is the standard DTM package comment delimiter. All text (and numbers) which lie to the right of a '!' character are excluded from command interpretation. Comments are useful for annotating command procedures used in batch processing etc.

Messages: None.

Examples:

DTICHECK> **! a comment for the sake of it<CR>**
DTICHECK> **WINDOW 0.0 0.0 900.0 900.0<CR>**
DTICHECK>

DISABLE ABSOLUTE

Disables a previous ENABLE ABSOLUTE command.

FORMAT: DISABLE ABSOLUTE

Command parameters: None.

DESCRIPTION:

DISABLE ABSOLUTE cancels a previous ENABLE ABSOLUTE command. If DISABLE ABSOLUTE is given, then coordinate values required by the WINDOW command supplied in metre or projection units, must be specified as an offset from the SW corner of the matrix.
By default values should be specified as absolute coordinates.

Messages:

The following error message is specific to the DISABLE command:

```
*** ERROR *** Specifying command DISABLE
Available DISABLE command qualifiers are:
ABSOLUTE    DIAGNOSTICS     MINUS      NULL       PME
SLOPE       ZERO
```

Examples:

DTICHECK> **DISABLE ABSOLUTE<CR>**
DTICHECK>

DISABLE DIAGNOSTICS

Disables a previous ENABLE DIAGNOSTICS command.

FORMAT: DISABLE DIAGNOSTICS

Command parameters: None.

DESCRIPTION:

DISABLE DIAGNOSTICS allows the user to disable a previous ENABLE DIAGNOSTICS command.

Messages:

The following error message is specific to the DISABLE command:

```
*** ERROR *** Specifying command DISABLE
Available DISABLE command qualifiers are:
ABSOLUTE    DIAGNOSTICS    MINUS    NULL    PME
SLOPE       ZERO
```

Examples:

DTICHECK> **DISABLE DIAGNOSTICS<CR>**
DTICHECK>

DISABLE MINUS

Disables a previous ENABLE MINUS command.

FORMAT: DISABLE MINUS

Command parameters: None.

DESCRIPTION:

DISABLE MINUS allows the user to disable a previous ENABLE MINUS command.

Messages:

The following error message is specific to the DISABLE command:

```
*** ERROR *** Specifying command DISABLE
Available DISABLE command qualifiers are:
ABSOLUTE    DIAGNOSTICS     MINUS      NULL       PME
SLOPE       ZERO
```

Examples:

DTICHECK> **DISABLE MINUS<CR>**
DTICHECK>

DISABLE NULL

Disables a previous ENABLE NULL command.

FORMAT: DISABLE NULL

Command parameters: None.

DESCRIPTION:

DISABLE NULL allows the user to disable a previous ENABLE NULL command.

Messages:

The following error message is specific to the DISABLE command:

```
*** ERROR *** Specifying command DISABLE
Available DISABLE command qualifiers are:
ABSOLUTE    DIAGNOSTICS     MINUS      NULL       PME
SLOPE       ZERO
```

Examples:

DTICHECK> **DISABLE NULL<CR>**
DTICHECK>

DISABLE PME

Disables a previous ENABLE PME command.

FORMAT: DISABLE PME

Command parameters: None.

DESCRIPTION:

DISABLE PME allows the user to disable a previous ENABLE PME command.

Messages:

The following error message are specific to the DISABLE and DISABLE PME commands:

```
*** ERROR *** Specifying command DISABLE
Available DISABLE command qualifiers are:
ABSOLUTE    DIAGNOSTICS     MINUS      NULL       PME
SLOPE       ZERO
```

```
*** WARNING *** You are not using PME !
```

Examples:

```
DTICHECK> ENABLE PME<CR>
DTICHECK> DISABLE PME<CR>
DTICHECK>
```

DISABLE SLOPE

Disables a previous ENABLE SLOPE command.

FORMAT: DISABLE SLOPE

Command parameters: None.

DESCRIPTION:

DISABLE SLOPE allows the user to disable a previous ENABLE SLOPE command.

Messages:

The following error message is specific to the DISABLE command:

```
*** ERROR *** Specifying command DISABLE
Available DISABLE command qualifiers are:
ABSOLUTE    DIAGNOSTICS     MINUS      NULL       PME
SLOPE       ZERO
```

Examples:

```
DTICHECK> DISABLE SLOPE<CR>
DTICHECK>
```

DISABLE ZERO

Disables a previous ENABLE ZERO command.

FORMAT: DISABLE ZERO

Command parameters: None.

DESCRIPTION:

DISABLE ZERO allows the user to disable a previous ENABLE ZERO command.

Messages:

The following error message is specific to the DISABLE command:

```
*** ERROR *** Specifying command DISABLE
Available DISABLE command qualifiers are:
ABSOLUTE    DIAGNOSTICS     MINUS      NULL       PME
SLOPE       ZERO
```

Examples:

DTICHECK> **DISABLE ZERO<CR>**
DTICHECK>

ENABLE ABSOLUTE

Selects the use of absolute coordinates values.

FORMAT: ENABLE ABSOLUTE

Command parameters: None.

DESCRIPTION:

If ENABLE ABSOLUTE is given, then coordinate values required by the WINDOW, command, supplied in metre or projection units, must be specified as absolute (rather than relative) coordinate values.

For example if the projection indicates U.K. National Grid, then the WINDOW values may be specified as 6 figure National Grid coordinates.
By default window values should be specified as absolute coordinates.

ENABLE ABSOLUTE also controls whether coordinate information output with the SHOW FILES command, and during a DTICHECK run if the ENABLE DIAGNOSTICS command has been specified, is output in absolute or relative coordinates.

This option can be disabled using the DISABLE ABSOLUTE command.

Messages:

The following error message is specific to the ENABLE command:

```
*** ERROR *** Specifying command ENABLE
Available DISABLE command qualifiers are:
ABSOLUTE    DIAGNOSTICS     MINUS      NULL       PME
SLOPE       ZERO
```

Examples:

DTICHECK> **ENABLE ABSOLUTE<CR>**
DTICHECK>

ENABLE DIAGNOSTICS

ENABLE DIAGNOSTICS allows the user to enable diagnostic printout.

FORMAT: ENABLE DIAGNOSTICS

Command parameters: None.

DESCRIPTION:

ENABLE DIAGNOSTICS allows the user to enable diagnostic printout.

If no output file option is selected, all checking reports are directed to SYS$OUTPUT. If an output file is selected checking reports will only be directed to SYS$OUTPUT if the ENABLE DIAGNOSTICS is given.

Because it is usually used in a batch processing environment to generate a LITES2 or ROVER command file, by default DTICHECK produces minimal diagnostic printout. If however, the user wishes to receive indications of processing progress and of the effect of windowing on data input, diagnostic printout may be selected with the ENABLE DIAGNOSTICS command.

If no output file option is selected, all checking reports are directed to SYS$OUTPUT. If an output file is selected checking reports will only be directed to SYS$OUTPUT if the ENABLE DIAGNOSTICS is given.

On a heavily loaded computer it may be reassuring to ENABLE DIAGNOSTICS to indicate progress through the DTI file.

Messages:

The following error message is specific to the ENABLE command:

```
*** ERROR *** Specifying command ENABLE
Available DISABLE command qualifiers are:
ABSOLUTE    DIAGNOSTICS    MINUS     NULL      PME
SLOPE       ZERO
```

Examples:

DTICHECK> **ENABLE DIAGNOSTICS<CR>**
DTICHECK>

ENABLE MINUS

Perform the check for negative post values on the file specified with the FILEIN command.

FORMAT: ENABLE MINUS

Command parameters: None

DESCRIPTION:

Perform the check for negative post values on the file specified with the FILEIN command.

The ENABLE MINUS command causes DTICHECK to report on the occurrence of all posts within the DTI file which have a negative Z value.

See the main DESCRIPTION section for a description of the report generated.

Posts with negative null values are NOT reported. Use the CHECK NULL command to report on the occurrence of negative null posts.

Where errors are reported for adjacent posts within the DTI file, a run length encoded message will be generated.

By default no check options are active and the user is warned on attempting to issue a GO command if no check options are active.

The ENABLE MINUS command may be specified singly or in combination with any of the other check options.

Messages:

The following error message is specific to the ENABLE command:

```
*** ERROR *** Specifying command ENABLE
Available DISABLE command qualifiers are:
ABSOLUTE    DIAGNOSTICS     MINUS      NULL       PME
SLOPE       ZERO
```

Examples:

DTICHECK> **ENABLE MINUS<CR>**
DTICHECK>

ENABLE NULL

Perform the check for posts having the null value.

FORMAT: ENABLE NULL

Command parameters: None

DESCRIPTION:

The ENABLE NULL command causes DTICHECK to report on the occurrence of all posts within the DTI file which have the null Z value.

See the main DESCRIPTION section for a description of the report generated.

Unless the user specifies a null value with the NULL command, by default the following values are used: -
0 if BYTE type.
-32767 if WORD type.
80000000 Hex if LONG type.
-1.0E-38 if REAL type.

Where errors are reported for adjacent posts within the DTI file, a run length encoded message will be generated.

By default no check options are active and the user is warned on attempting to issue a GO command if no check options are active.

The ENABLE NULL command may be specified singly or in combination with any of the other check options.

Messages:

The following error message is specific to the ENABLE command:

```
*** ERROR *** Specifying command ENABLE
Available DISABLE command qualifiers are:
ABSOLUTE    DIAGNOSTICS    MINUS    NULL    PME
SLOPE       ZERO
```

Examples:

DTICHECK> **ENABLE NULL<CR>**
DTICHECK>

ENABLE PME

ENABLE PME enables the PME performance monitor.

FORMAT: ENABLE PME

Command parameters: None.

DESCRIPTION:

The ENABLE PME and DISABLE PME commands are reserved for Laser-Scan use. PME is a code optimisation tool and should be invoked by LSL software personnel only.

ENABLE PME causes the PME_INIT routine to be invoked.

Message:

The following messages are specific to the ENABLE and ENABLE PME commands:

```
*** ERROR *** Specifying command ENABLE
Available DISABLE command qualifiers are:
ABSOLUTE    DIAGNOSTICS     MINUS      NULL       PME
SLOPE       ZERO
```

```
*** WARNING *** You are already using PME!
```

Examples:

```
$ DTICHECK<CR>
TVES module DTICHECK of 13:30:39  18-JAN-89
DTICHECK> ENABLE PME<CR>
DTICHECK>
```

ENABLE SLOPE

Report where a critical slope defined by a user supplied threshold Z difference is exceeded.

FORMAT: **ENABLE SLOPE**

Command parameters: None

DESCRIPTION:

The ENABLE SLOPE command causes DTICHECK to report where a critical slope defined by a user supplied threshold Z difference is exceeded.

See the main DESCRIPTION section for a description of the report generated.

Posts with null values are NOT reported.

Where errors are reported for adjacent posts within the DTI file, a run length encoded message will be generated.

By default no check options are active and the user is warned on attempting to issue a GO command if no check options are active.

The ENABLE SLOPE command may be specified singly or in combination with any of the other check options.

If ENABLE SLOPE is specified the SLOPE command should also be specified before the GO command is used.

Messages:

The following error message is specific to the ENABLE command:

```
*** ERROR *** Specifying command ENABLE
Available DISABLE command qualifiers are:
ABSOLUTE    DIAGNOSTICS     MINUS      NULL       PME
SLOPE       ZERO
```

Examples:

DTICHECK> **ENABLE SLOPE<CR>**

```
DTICHECK>
```

ENABLE ZERO

Perform the check for zero post values on the file specified with the FILEIN command.

FORMAT: ENABLE ZERO

Command parameters: None

DESCRIPTION:

The ENABLE ZERO command causes DTICHECK to report on the occurrence of all posts within the DTI file which have a zero Z value.

See the main DESCRIPTION section for a description of the report generated.

Care must be exercised when processing BYTE type DTI files which, by default, use zero as the null post value.

Where errors are reported for adjacent posts within the DTI file, a run length encoded message will be generated.

By default no check options are active and the user is warned on attempting to issue a GO command if no check options are active.

The ENABLE ZERO command may be specified singly or in combination with any of the other check options.

Messages:

The following error message is specific to the ENABLE command:

```
*** ERROR *** Specifying command ENABLE
Available DISABLE command qualifiers are:
ABSOLUTE    DIAGNOSTICS     MINUS      NULL       PME
SLOPE       ZERO
```

Examples:

DTICHECK> **ENABLE ZERO<CR>**
DTICHECK>

FILEIN

Specifies a DTI file that is to be opened and used for data input.

FORMAT: FILEIN file-spec

COMMAND PARAMETERS:

file-spec

The specification of the DTI file to be opened for data input.

Any parts of the file-spec not supplied for the FILEIN command will be taken from the default file specification 'LSL$DTI:DTI.DTI;0'.

DESCRIPTION:

The FILEIN command causes the specified file to be opened and used as an input file to DTICHECK. A FILEIN command must be issued before the WINDOW or GO commands will be accepted.

By default the window is the total extent of the input file.

Messages:

The following messages are specific to the FILEIN command:

```
*** WARNING *** You must specify a file-spec argument to the FILEIN command
*** ERROR *** Unable to interpret input file-spec
```

Examples:

```
DTICHECK> FILEIN REAL<CR>
DTI file LSL$DTI:REAL.DTI;0 opened for read

File    :  LSL$DTI:REAL.DTI;0
Header  :  LSLA  Data:  REAL

Units are Metres

Matrix Coverage   SW:       0.00       0.00    NE:      49.00      49.00
Matrix Interval    E:       1.00                N:       1.00
Value Range         :     -5.00 to  326421.00
DTICHECK>
```

GO

Perform the selected checks on the file specified with the FILEIN command.

FORMAT: GO

Command parameters: None

DESCRIPTION:

Perform the selected checks on the file specified with the FILEIN command.

Before starting to process, checks will be carried out to ensure that the input file has been opened and that at least one check option has been selected.

Messages:

*** ERROR *** No CHECKS have been ENABLED
*** ERROR *** DTI file not yet specified DTI file

In addition to these there are the diagnostic printout messages which will be given if diagnostics are enabled.These consist of indications of the progress of checking through the input file.

Examples:

DTICHECK> **GO<CR>**
DTICHECK>

HELP

Give help on a subject

FORMAT: HELP subject

Command parameters:

subject

The subject on which help is required.

Description:

The HELP command looks the rest of the line up in the TVES HELP library. This library contains a brief summary of the operation of each command.

The information is looked up in the DTICHECK section of the TVES help library, LSL$HELP:TVES.HLB.

Messages:

Where required, warning messages are output via the VMS LBR$OUTPUT_HELP utility.

Examples:

```
DTICHECK> HELP ENABLE PME<CR>

DTICHECK

      ENABLE

         PME

   The ENABLE PME and DISABLE PME commands are reserved for Laser-Scan use.  PME
   is  a  code optimisation tool and should be invoked by LSL software personnel
   only.

   ENABLE PME causes the PME_INIT routine to be invoked.
DTICHECK>
```

IFF

Specifies the file-spec of the IFF file which is to receive features indicating the positions of suspected data errors.

FORMAT: IFF file-spec

COMMAND PARAMETERS:

file-spec

The file-spec of the IFF file which is to receive features indicating the positions of suspected data errors.

Any parts of the file-spec not supplied for the IFF command will be taken from the default file specification 'LSL$IFF:IFF.IFF;0'.

DESCRIPTION:

The IFF command enables specification of the IFF file which is to receive features indicating the positions of suspected data errors.

The layer and feature code allocations used in the IFF file may be specified by the user at run time using the SET command. Alternatively, the DTICHECK default layer and feature code allocations may be used. These are:

```
Layer .............................. 1
Slope feature code .......... 1
Minus feature code .......... 2
Null feature code ........... 3
Zero feature code ........... 4
```

The IFF command may be specified in conjunction with the LITES2 and ROVER commands. If none of the IFF, LITES2 and ROVER commands is specified, message output will be to SYS$OUTPUT.

The IFF command must be issued before the GO command.

Messages:

The following messages are specific to the IFF command:

```
*** WARNING *** You must specify a file-spec argument to the IFF command
*** ERROR *** unable to interpret output file-spec
```

Examples:

```
DTICHECK> IFF DUA3:[DEMONSTRATION]IDAHO.IFF<CR>
DTICHECK>
```

LITES2

Specifies the file-spec of the LITES2 command file which is to receive LITES2 commands indicating the positions of suspected data errors.

FORMAT: LITES2 file-spec

COMMAND PARAMETERS:

file-spec

The file-spec of the LITES2 command file.

Any parts of the file-spec not supplied for the LITES2 command will be taken from the default file specification 'LSL$LITES2CMD:DTICHECK.LCM;0'.

DESCRIPTION:

The LITES2 command enables specification of a LITES2 command file to receive LITES2 commands indicating the positions of suspected data errors.

See the main DESCRIPTION section for a description of the LITES2 command used.

The LITES2 command may be specified in conjunction with the IFF and ROVER commands. If none of the IFF, LITES2 and ROVER commands is specified, message output will be to SYS$OUTPUT.

The LITES2 command must be issued before the GO command.

Messages:

The following messages are specific to the LITES2 command:

```
*** WARNING *** You must specify a file-spec argument to the LITES2 command
*** ERROR *** unable to interpret output file-spec
```

Examples:

```
DTICHECK> LITES2 DUA3:[DEMONSTRATION]IDAHO.LCM<CR>
DTICHECK>
```

NULL

Specifies the null value used in the input DTI file for which no calculated value is available.

FORMAT: NULL null_value

Command parameters:

null_value

The null value to be applied. This must lie within a range appropriate to the input DTI file data type:

Minimum	Maximum	DTI data type
0	255	BYTE
-32768	32767	WORD INTEGER
-2E31	2E31-1	LONG INTEGER
-1.0E38	1.0E38	REAL_*4

By default the following null values are used: -

0 if BYTE type.
-32767 if WORD type.
80000000 Hex if LONG type.
-1.0E-38 if REAL type.

DESCRIPTION:

The NULL command specifies the null value used in the input DTI file for which no calculated value is available.

The null value can only be set after a DTI file has been input using the FILEIN command, as the range of valid null values depends on the data type of the file.

Messages:

The following messages are specific to the NULL command:

*** WARNING *** null value out of range for a byte type DTI file

*** ERROR *** null value initially defined upon opening a DTI file

*** WARNING *** null value out of range for a word type DTI file

```
*** WARNING *** null value out of range for a longword type DTI file

*** WARNING *** null value out of range for a real type DTI file
```

Examples:

DTICHECK> **NULL 0<CR>**
DTICHECK>

PAUSE

Pauses DTICHECK execution.

FORMAT: PAUSE

Command parameters: None.

DESCRIPTION:

Pauses DTICHECK execution and issues a prompt for a carriage return to continue execution. This command is designed for use in software demonstration situations.

Messages: None.

Examples:

```
DTICHECK> PAUSE<CR>


Press <RETURN> to continue<CR>
DTICHECK>
```

QUIT

Quit from DTICHECK.

FORMAT: QUIT

Command parameters: None.

Description:

The QUIT command causes DTICHECK to exit immediately, closing the input file and closing and deleting the output file.

<CTRL/Z> (pressing the Ctrl and Z keys together) may also be used to quit from the program.

Messages: None.

Examples:

```
DTICHECK> QUIT<CR>
 ELAPSED:  00:00:20.04 CPU:  0:00:04.71 BUFIO:  281 DIRIO:  46 FAULTS:  263
$
```

RETURN

Restores command input from an indirect file to SYS$COMMAND.

FORMAT: RETURN

Command parameters: None.

DESCRIPTION:

Restores command input from an indirect file to SYS$COMMAND.

A typical application is to allow the user to use an indirect command file to set up those run time defaults which are constant within a flowline and then return to input from the terminal (or batch stream) for the run specific commands. To do this RETURN must be the last command in the indirect command file.

Messages:

The following messages are specific to the RETURN command:

RETURN command detected - returning to terminal input
RETURN command ignored - command input is already from terminal

---

Examples:

```
DTICHECK> @FLOW2<CR>
DTICHECK> ENABLE DIAGNOSTICS
DTICHECK> RETURN
DTICHECK>
```

ROVER

Specifies the file-spec of the ROVER command file which is to receive ROVER commands indicating the positions of suspected data errors.

FORMAT: ROVER file-spec

COMMAND PARAMETERS:

file-spec

The file-spec of the ROVER command file.

Any parts of the file-spec not supplied for the ROVER command will be taken from the default file specification 'LSL$ROVERCMD:DTICHECK.RCM;0'.

DESCRIPTION:

The ROVER command enables specification of a ROVER command file to receive ROVER commands indicating the positions of suspected data errors.

See the main DESCRIPTION section for a description of the ROVER commands used.

The ROVER command may be specified in conjunction with the IFF and LITES2 commands. If none of the IFF, LITES2 and ROVER commands is specified, message output will be to SYS$OUTPUT.

The ROVER command must be issued before the GO command.

Messages:

The following messages are specific to the ROVER command:

```
*** WARNING *** You must specify a file-spec argument to the ROVER command
*** ERROR *** unable to interpret output file-spec
```

Examples:

```
DTICHECK> ROVER DUA3:[DEMONSTRATION]IDAHO.RCM<CR>
DTICHECK>
```

SET LAYER

Specifies the layer number to be used for the error features. This must lie in the range 0 to 32767.

FORMAT: SET LAYER layer-number

Command parameters:

layer-number

The layer number to be used for the error features. This must lie in the range 0 to 32767.

DESCRIPTION:

SET LAYER enables the user to specify the number of the layer which is to contain the error features. By default, layer 1 is created.

Messages:

*** WARNING *** Layer numbers must lie in the range 0 to 32767.

Default layer 1 restored

Examples:

```
DTICHECK> SET LAYER 209<CR>
DTICHECK>
```

SET MINUS_FC

Specifies the feature code to be used for error features representing the location of negative post values. This must lie in the range 0 to 32767.

FORMAT: SET MINUS_FC feature-code

Command parameters:

feature-code

The feature code to be used for error features representing posts having a negative Z value. This must lie in the range 0 to 32767.

DESCRIPTION:

SET MINUS_FC enables the user to specify the feature code used for error features used to represent posts having a negative Z value.

By default feature code 2 is used.

Messages:

*** WARNING *** Feature codes must lie in the range 0 to 32767.

Default minus_FC 2 restored

Examples:

DTICHECK> **SET MINUS_FC 12<CR>**
Minus_FC set to 12
DTICHECK>

SET NULL_FC

Specifies the feature code to be used for error features representing the location of null post values. This must lie in the range 0 to 32767.

FORMAT: SET NULL_FC feature-code

Command parameters:

feature-code

The feature code to be used for error features representing posts having a null Z value. This must lie in the range 0 to 32767.

DESCRIPTION:

SET NULL_FC enables the user to specify the feature code used for error features used to represent posts having a null Z value.

By default feature code 3 is used.

Messages:

*** WARNING *** Feature codes must lie in the range 0 to 32767.

Default Null_FC 3 restored

Examples:

DTICHECK> **SET NULL_FC 13<CR>**
Null_FC set to 13
DTICHECK>

SET SLOPE_FC

Specifies the feature code to be used for error features representing the location of posts with bad slopes. This must lie in the range 0 to 32767.

FORMAT: SET SLOPE_FC feature-code

Command parameters:

feature-code

The feature code to be used for error features representing posts having a bad slope value. This must lie in the range 0 to 32767.

DESCRIPTION:

SET SLOPE_FC enables the user to specify the feature code used for error features used to represent posts having a bad slope value.

By default feature code 1 is used.

Messages:

*** WARNING *** Feature codes must lie in the range 0 to 32767.

Default Slope_FC 1 restored

Examples:

DTICHECK> **SET SLOPE_FC 14<CR>**
Slope_FC set to 14
DTICHECK>

SET ZERO_FC

Specifies the feature code to be used for error features representing the location of zero post values. This must lie in the range 0 to 32767.

FORMAT: SET ZERO_FC feature-code

Command parameters:

feature-code

The feature code to be used for error features representing posts having a zero Z value. This must lie in the range 0 to 32767.

DESCRIPTION:

SET ZERO_FC enables the user to specify the feature code used for error features used to represent posts having a zero Z value.

By default feature code 4 is used.

Messages:

*** WARNING *** Feature codes must lie in the range 0 to 32767.

Default Zero_FC 4 restored

Examples:

DTICHECK> **SET ZERO_FC 15<CR>**
Zero_FC set to 15
DTICHECK>

SHOW ENABLE

Shows the current status of those options that may be enabled by means of the ENABLE command, or disabled using the DISABLE command.

FORMAT: **SHOW ENABLE**

Command parameters: None.

DESCRIPTION:

Displays the current status of all the DTICHECK options that may be enabled or disabled using the ENABLE and DISABLE commands.
The name of the option is shown, followed by either the word ON or OFF to indicate its current status.
If the command SHOW ENABLE is used before any ENABLE or DISABLE commands have been given, the default status of the options is displayed.

Messages:

The following message is specific to the SHOW command:

```
*** ERROR *** Specifying command SHOW
Available SHOW command qualifiers are:
ENABLE  FILES  NULL  SETTINGS   SLOPE
```

Examples:

DTICHECK>**SHOW ENABLE<CR>**

```
Current status:

ABSOLUTE        On       DIAGNOSTICS      Off        MINUS           Off

NULL            Off      PME              Off        SLOPE           Off

ZERO            Off

DTICHECK>
```

SHOW FILES

Extracts information from the headers of the input DTI file and outputs it to the screen along with details of files opened with the IFF, LITES2 and ROVER commands.

FORMAT: **SHOW FILES**

Command parameters: None.

DESCRIPTION:

Details extracted from the header of the input DTI file are displayed on the terminal.

The header values are shown in the current units of measurement. This is dependent on the header type of the input file, or may be set explicitly using the UNITS command. By default or if the ENABLE ABSOLUTE command has been given then metre or projection values are expressed in absolute values. Otherwise they are displayed as offsets from the SW corner of the matrix. Currently selected windows are also displayed.

Details of files opened with the IFF, LITES2 or ROVER commands are also displayed.

Messages:

The following message is specific to the SHOW command:

```
*** ERROR *** Specifying command SHOW
Available SHOW command qualifiers are:
ENABLE  FILES  NULL  SETTINGS   SLOPE
```

Examples:

```
DTICHECK>SHOW FILES<CR>
File    :  LSL$DTI:TST.DTI;0
Header  :  LSLA  Data:  WORD

Units are DTI Matrix Values

Matrix Coverage   SW:     1     1  NE:    71    71
Matrix Window     SW:     1     1  NE:    50    50
Matrix Interval    E:     1         N:     1
Value Range         :   139   to    192

IFF file LSL$IFF:DTICHECK.IFF;0 selected for output
LITES2 output command file not yet specified
```

ROVER output command file not yet specified

SHOW NULL

Displays the currently specified null value.

FORMAT: **SHOW NULL**

Command parameters: None.

DESCRIPTION:

The null value specified with the NULL command is displayed on the terminal.

Messages:

The following message is specific to the SHOW command:

```
*** ERROR *** Specifying command SHOW
Available SHOW command qualifiers are:
ENABLE  FILES  NULL  SETTINGS   SLOPE
```

Examples:

DTICHECK>**SHOW NULL<CR>**
Null value is -32767
DTICHECK>

SHOW SETTINGS

Shows the current value of those parameters that may be set using the SET command.

FORMAT: **SHOW SETTINGS**

Command parameters: None.

DESCRIPTION:

Displays the name of all DTICHECK parameters that may be set using the SET command, and their current values.
If the SHOW SETTINGS command is given before using a SET command, the values shown are the default values that are allocated to the parameters by DTICHECK.

Messages:

The following message is specific to the command SHOW:

```
*** ERROR *** Specifying command SHOW
Available SHOW command qualifiers are:
ENABLE  FILES  NULL  SETTINGS   SLOPE
```

Examples:

DTICHECK>**SHOW SETTINGS<CR>**

```
Current Settings:

LAYER           1       MINUS_FC        2       NULL_FC         3

SLOPE_FC        1       ZERO_FC         4


DTICHECK>
```

SHOW SLOPE

Displays the currently specified slope value.

FORMAT: **SHOW SLOPE**

Command parameters: None.

DESCRIPTION:

The SLOPE value specified with the SLOPE command is displayed on the terminal.

Messages:

The following message is specific to the SHOW command:

```
*** ERROR *** Specifying command SHOW
Available SHOW command qualifiers are:
ENABLE  FILES  NULL  SETTINGS   SLOPE
```

Examples:

DTICHECK>**SHOW SLOPE <CR>**
Slope value is 50
DTICHECK>

SLOPE

Specifies the threshold Z difference allowed between adjacent posts in the DTI file, above which slope is considered unreasonably steep.

FORMAT: SLOPE difference

Command parameters:

difference

The threshold Z difference allowed between adjacent posts in the DTI file, above which slope is considered unreasonably steep.

DESCRIPTION:

The SLOPE argument specifies the threshold Z difference allowed between adjacent posts in the DTI file, above which slope is considered unreasonably steep.

When setting the maximum slope threshold, is is important to be aware of the spacing in ground units of the DTI posts. This is because a Z difference of 20 (metres) will be significant if the post spacing is 10 metres, but not so significant if the spacing is 100 metres.

If ENABLE SLOPE is specified the SLOPE command must also be specified before the GO command will be accepted.

Messages:

*** ERROR *** Missing Z difference argument
Maximum slope for this DTM:

Examples:

DTICHECK> **SLOPE 10.0<CR>**
DTICHECK>

SPAWN

The SPAWN command enables a subprocess to be created from within the DTICHECK utility.

FORMAT: SPAWN command-line

Command parameters:

command-line

Specifies a DCL command string to be executed as if typed in response to a '$' prompt. When the command completes, the subprocess terminates and control is returned to DTICHECK. The command line cannot exceed 80 characters.

DESCRIPTION:

The SPAWN command enables you to create a subprocess while within the DTICHECK utility. When the subprocess terminates, control is returned to DTICHECK.

Messages:

The following error messages are specific to the SPAWN command:

```
*** ERROR *** Specifying command SPAWN
Command requires a valid DCL command line
```

```
*** ERROR *** Unable to spawn command, returning to DTICHECK
```

Examples:

DTICHECK> **SPAWN DIRECTORY LSL$DTI:*.DTI<CR>**

```
Directory DUA3:[LSL.DTI]

TEST1.DTI;1              8/8        18-AUG-1987 07:56  [LSL,DAVEC]
TEST2.DTI;1              7/8        18-AUG-1987 17:17  [LSL,DAVEC]
TEST2.DTI;2              7/8        18-AUG-1987 17:34  [LSL,DAVEC]

Total of 3 files, 22/24 blocks.

DTICHECK>
```

UNITS

Specifies the units of measurement that will be used when defining an area of interest in the input DTI file using the WINDOW command. The command also controls the units of measurement which will be used when displaying file header details.

FORMAT: UNITS units

Command parameters:

units

A keyword defining the measurement units, chosen from:

MATRIX	Matrix grid interval units, i.e rows and columns
METRES	Metres on the ground
LATLONG	Latitude and Longitude (in degrees, minutes and seconds)
SECONDS	Seconds of arc
PROJECTION	Projection Record Units (eg. mms on the source document)

DESCRIPTION:

The UNITS command enables the user to specify in what units of measurement he wishes to define an area of interest in an input DTM using the WINDOW command. The command also controls in what format coordinate data from the header of the DTM are displayed, and if ENABLE DIAGNOSTICS has been specified the coordinates output during the DTICHECK run.

The UNITS command should be given after defining the DTI file containing the input DTM, since an appropriate default units of measurement is set up whenever an input DTI file is opened.

Messages:

The following error messages are specific to the UNITS command:
*** ERROR *** Specifying command UNITS
Command qualifiers are MATRIX, METRES, SECONDS, LATLONG or PROJECTION
UNITS remain set to metres on the ground

*** ERROR *** Specifying command UNITS
Command qualifier is invalid for the input file

Examples:

```
DTICHECK> FILEIN TST11<CR>
DTICHECK> UNITS MATRIX<CR>
DTICHECK> WINDOW 41 41 101 101<CR>
DTICHECK>
```

WINDOW

Specifies the limits of the data area to be extracted from an input file.

FORMAT: WINDOW xmin ymin xmax ymax

Command parameters:

xmin ymin

The coordinates of the bottom left hand corner of the defining rectangle.

xmax ymax

The coordinates of top the right hand corner of the defining rectangle.

The format of the command parameters is controlled by the UNITS command.

UNITS MATRIX - Requires 4 integer values defining the rectangle in terms of column and row numbers

UNITS METRES - Requires 4 real (floating point) values defining the rectangle as metre offsets from the SW corner of the DTM. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates (eg. U.K. National Grid coordinates); if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

UNITS SECONDS - Requires 4 real (floating point) values defining the absolute position of the rectangle in seconds of arc. The values are supplied in the order latitude followed by longitude for the SW corner, and latitude followed by longitude for the NE corner. A negative latitude value indicates a position in the Southern Hemisphere, and a negative longitude value a point to the west of Greenwich.

UNITS LATLONG - Requires 4 values defining the absolute latitude and longitude position of the rectangle in the format DDD MM SS H where DDD is the number of degrees, MM is the number of minutes, SS is the number of seconds, and H is the hemisphere (N,S,E,W). The values are supplied in the order latitude followed by longitude for the SW corner, and latitude followed by longitude for the NE corner.

UNITS PROJECTION Requires 4 real (floating point) values defining the rectangle in projection record units. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates; if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

Note that in all cases, the input values are adjusted to the nearest column and row values.

DESCRIPTION:

The command is used to define the rectangular limits of the area of data which is to be extracted from the input DTI file. The limits must be specified in the order bottom left hand (or south west) corner then top right hand (or north east) corner.

The specified window values should lie within the input DTI file x and y ranges. If necessary the window values will be truncated to fit the input DTI file x and y ranges.

The WINDOW command can be used to clip data from the input DTI file. If input is to be from an entire DTI file then the WINDOW command may be omitted as the WINDOW extents will then, by default, be taken from the DTI file header.

Messages:

The following warning messages are specific to the WINDOW command:

```
*** ERROR **** specifying DTI window
Command requires 2 x,y coordinate pairs ( SW and NE )
*** ERROR **** specifying DTI window
NE coordinates should exceed SW coordinates
```

Examples:

DTICHECK> **WINDOW 0.0 0.0 120.0 120.0<CR>**
DTICHECK>

DTICHECK examples

EXAMPLES

```
$ DTICHECK
TVES module DTICHECK of 12:30:53  17-MAR-88
DTICHECK> FILEIN OEEPE <CR>
 DTI FILE LSL$DTI:OEEPE.DTI;0 opened for read

 File    :  LSL$DTI:OEEPE.DTI;0
 Header  :  LSLA  Data:  WORD

 Units are DTI Matrix Values

 Matrix Coverage   SW:     1     1  NE:    71    71
 Matrix Window     SW:     1     1  NE:    50    50
 Matrix Interval    E:     1         N:     1
 Value Range         :   139   to    192

DTICHECK> IFF DTICHECK <CR>
 %LSLLIB-I-IFFOPENED,LSL$DATAROOT:[LSL.IFF]DTICHECK.IFF;9 opened for write
DTICHECK> ENABLE SLOPE <CR>
DTICHECK> ENABLE MINUS <CR>
DTICHECK> SLOPE 15 <CR>
DTICHECK> SET LAYER 10 <CR>

DTICHECK> GO
ELAPSED: 00:00:20.04  CPU: 0:00:04.71  BUFIO: 281  DIRIO: 46  FAULTS: 263
$
```

This is an example of the sequence of commands used to check a DTI file for slopes exceeding a threshold, and for negative (minus) values.

The input DTI file is specified using the FILEIN command.

Slope and minus value checking are switched on using the ENABLE command and the threshold slope value is specified using the SLOPE command. This represents a difference between matrix values, and the matrix grid interval should be known before SLOPE is specified. The data are output into IFF file layer 10.

before the program is executed using the GO command. After the checks have been performed, the program exits.

DTICHECK messages

MESSAGES (SUCCESS)

These messages are used to indicate that the program has succeeded in performing some action, and do not require any user action.

NORMAL, DTICHECK normal successful completion

> **Explanation:** DTICHECK has terminated successfully, without encountering any errors.
>
> **User action:** None

MESSAGES (INFORMATIONAL)

These messages give information only, and require no immediate action by the user. They are used to provide information on the current state of the program, or to supply explanatory information in support of a warning or error message.

RCMOPEN, ROVER command file 'file-spec' successfully opened

Explanation: DTICHECK has successfully opened the ROVER command file, without encountering any errors.

User action: None

MESSAGES (ERROR)

These messages indicate an error in processing which will cause the program to terminate. The most likely causes are a corrupt or otherwise invalid input file, or an error related to command line processing and file manipulation.

CLDTI, Failed to close DTI file 'file-spec' properly

Explanation: DTICHECK has failed to close the DTI file.

User action: Check the DTI file and its supplied file-spec.

CLIFF, Failed to close IFF file 'file-spec' properly

Explanation: DTICHECK has failed to close the IFF file.

User action: Check the IFF file and its supplied file-spec.

CLIND, Failed to close indirect command file 'file-spec' properly

Explanation: DTICHECK has failed to open the indirect command file.

User action: Check the indirect command file and its supplied file-spec.

RCMFAIL, Failed to open ROVER command file 'file-spec'

Explanation: DTICHECK has failed to open the ROVER command file.

User action: Check the ROVER command file and its supplied file-spec.

MESSAGES (OTHER)

In addition to the above messages which are generated by the program itself, other messages may be produced by the command line interpreter (CLI) and by Laser-Scan libraries. In particular, messages may be generated by the DTILIB library and by the Laser-Scan I/O library, LSLLIB. DTILIB library messages are introduced by '%DTILIB' and are documented in the MATRIX package reference manual. In most cases DTI errors will be due to a corrupt input file, and this should be the first area of investigation. If the cause of the error cannot be traced by the user, and Laser-Scan are consulted, then the output file should be preserved to facilitate diagnosis. LSLLIB messages are introduced by '%LSLLIB' and are generally self-explanatory. They are used to explain the details of program generated errors.

CHAPTER 4

MODULE DTICONTOUR

MODULE DTICONTOUR

DTI Contouring Program

Issue 2.1 - 6-July-1989

Science Park, Milton Road, Cambridge, England CB4 4FY tel: (0223) 420414
Document "DTICONTOUR Reference" Category "REFERENCE"

Document Issue	2.1	Jon Barber	06-July-1989
	2.0	Tim Hartnall	07-March-1989
	1.1	Tim Hartnall	30-November-1988
	1.0	Tim Hartnall	23-March-1988

MODULE **DTICONTOUR**

REPLACES PANACEA module TIMPANI

FUNCTION

DTICONTOUR is a bilinear contouring utility, using data contained in DTI (Digital Terrain Image) files. Contour smoothing may be optionally applied using a least squares filter.

Facilities to define (x,y) and z data windows are provided to enable the user to select sub-sections of the DTI data for contouring.

Output is to IFF file. An option to generate contour labelling is provided. The user may specify the intermediate and index contour interval and the feature coding attributes of the IFF file.

FORMAT

$ DTICONTOUR

COMMAND QUALIFIERS

None, DTICONTOUR is command driven.

DESCRIPTION

General

DTICONTOUR contour construction

For processing speed, DTICONTOUR uses a whole matrix contour lacing algorithm, using linear interpolation to derive the contours from adjacent DTI posts.

DTICONTOUR first maps the input DTI file into virtual memory. It can then treat the file as an array of Z values. Using the user specified x,y (and optionally z) windows, contour interval value, and the minimum and maximum Z values in the file header, the height of the first contour to be constructed can be calculated. Contouring can now begin.

The basic method consists firstly of systematically scanning the boundary edges of the DTI in an anticlockwise manner for the start of the contour. DTICONTOUR then follows the contour throughout the data window and finds the points at which the contour crosses a DTI row or column and the row and column diagonals, until it hits the boundary again. This is repeated for each of the 'open' contours at this height. The interior of the DTI array is then scanned to see if there are any 'closed' contours of this height and these are followed as for the 'open' ones.

When following a contour the convention 'high ground on the right' is adopted.

The output for each contour consists of a set of points where the contour crosses the DTI rows, columns and diagonals together with an indication as to whether a contour is 'open' or 'closed'. These points are then output directly to the IFF output file or they may be buffered for labelling and/or subjected to the smoothing algorithm to give the contours continuous derivatives.

Perhaps the most difficult problem in lacing contours through a DTM using this basic method is to ensure that all contours of a given height are found. This is made considerably easier by only using inverse linear interpolation to find a value of Z, so that there can be at most only one contour of a given height passing between a pair of adjacent points.

Because linear interpolation is used in DTICONTOUR, areas of equal Z value may lead to the formation of contours resembling a "staircase", where the contour makes right angled turns. There is insufficient information for the simple algorithm to lace the contours relative to a more global context.

It is important to note that although cartographically unappealing, the contours produced by DTICONTOUR are "truth". Because linear interpolation is used, the contours represent what is really there in the DTI file, not the expression of a high order surface patch. The linear interpolation used is not only fast, but provides a powerful quality check on the DTI file contents. If the "staircase" effect is undesirable, then the optional smoothing technique may be applied.

Contouring will stop at any regions of null data as defined by the data type specific DTI NULL parameter values, and will resume on the other side, if necessary.

DTICONTOUR contour labelling

DTICONTOUR provides the user with optional contour labelling; the numbers so generated being stored in the IFF file as independent features. It is therefore important to deselect labelling if the IFF file is going to be used for anything other than plot purposes. Heights are stored in the output IFF file type 3 ACs which may be accessed by Laser-Scan LAMPS utilities (eg clipping, merging and plotting routines)

If selected, the contour labelling generated by DTICONTOUR obeys the following cartographic conventions:

1. All labels "read" up-hill.

2. Labels are only written out where there is room for them, without over-writing adjacent contours.

3. The contour line is "broken" before a label, the label written and the line restarted.

4. Labels are not written out where the contour curves are too tight.

5. If drawn on a curve labels are "fitted" to the shape of the contour curve.

In order to ensure that contour labelling is appropriate to the nature of the contours the user is allowed to specify:

- which contours are to be labelled

- the minimum distance along contour lines between successive labels,

- the size of the labels,

- the maximum rate of contour curvature above which no label is output

The algorithm used works as follows:

1. DTICONTOUR tests to see if the current contour is one that is to be labelled.

2. If not, DTICONTOUR simply laces the contour, buffering the coordinates until the contour reaches the edge of the DTM or closes. DTICONTOUR flushes the contour buffer, either directly to the IFF output file or via the optional smoothing routine.

3. If the contour is to be labelled, DTICONTOUR calculates the width of the label, incrementing character by character not forgetting leading and trailing and inter-character spaces and any decimal point.

4. DTICONTOUR then identifies and buffers up the next point on the contour using the lacing algorithm. If the contour reaches the edge of the matrix, or closes, it simply flushes the contour buffer.

5. DTICONTOUR increments the distance travelled from the start of the contour, or the position of the last label. When the incremented distance exceeds the minimum distance between labels specified by the user, DTICONTOUR sets a flag "position suitable for labelling", and notes the point at which the flag was set. Tests for a suitable site for the label can begin.

6. The first test is for contours that are too closely spaced for the label to fit. The characters forming the label should be positioned so that the path of the contour line runs through the centre of the character y-axis. DTICONTOUR tests the approximated inter-contour distance against the half character height.

7. If the half character height is largest, then there is insufficient room. DTICONTOUR clears the "position suitable for labelling" flag and goes back to step 4.

8. DTICONTOUR tests whether the distance travelled since the "position suitable for labelling" flag was set exceeds the calculated label width. If it does not go back to step 4. Otherwise, DTICONTOUR takes each pair of segments buffered up since the "position suitable for labelling" flag was set and calculates the angle between them.

9. If the angle exceeds the user specified curve sharpness value the contour bend is too sharp: DTICONTOUR unsets the "position suitable for labelling" flag and goes back to step 4.

10. If the angles are within tolerance the label can be output. DTICONTOUR creates a gap for the label to lie in by flushing the coordinate buffer only up to the point flagged with the "position suitable for labelling" flag.

11. DTICONTOUR outputs vectors which define the character of the label. As a whole matrix contour lacing algorithm is adopted it is straightforward to make labels lie "uphill". The lacing algorithm always keeps high ground on the right of the contour being threaded through the matrix.

    The curved label is also straightforward as the shape of the contour (ie the length and angles of its segments) can be calculated from what is stored in the contour buffer, which was not flushed. DTICONTOUR remembers that the left hand corner of the characters will be offset from the contour line, which should run through the character y-axis centreline.

12. The last coordinate in the contour buffer is saved.

    DTICONTOUR clears the contour buffer, then makes the saved last coordinate become the first coordinate in the empty buffer. Go back to step 4.

Default settings are assumed for all labelling options and parameters unless otherwise specified. The default status for all parameters may be examined at any time using the SHOW LABELLING command.

DTICONTOUR and changes of measurement system

DTICONTOUR allows the user to contour a DTM which has heights in a different measurement system to that desired for the contours.

Four height modification options are provided for maximum processing flexibility:

o ENABLE TOFEET - results in the application of a scaling factor (3.2808455) to produce imperial (feet) contours from a metric (i.e. metres) DTM. It has the same effect as an explicit ENABLE MULTIPLY 3.2808455 command.

o ENABLE TOMETRES - results in the application of a scaling factor (3.2808455) to produce metric (metres) contours from an imperial (i.e. feet) DTM. It has the same effect as an explicit ENABLE DIVIDEBY 3.2808455 command.

o ENABLE DIVIDEBY - enables the user to select division of input file heights by a specified floating point constant.

o ENABLE MULTIPLYBY - enables the user to select multiplication of input file heights by a specified floating point constant.

DTICONTOUR and input from DTI files

DTICONTOUR can accept for input LSLA, UHL1, TED4 and the obsolete MIKE and ALVY type DTI (Digital Terrain Image) files. The elevation data within these files may be held as byte, word, real*4 and longword values. Bit valued DTI files are not supported by DTICONTOUR.

DTICONTOUR is designed to enable the user to contour DTI files which contain null DTM post values (i.e. post values which have been assigned a height value of -32767 in DTI files of the word, longword and real data types, or which have any of the DTILIB NULL parameter values for word, long or real datatypes). This is achieved by ignoring all posts which have the null height value.

DTICONTOUR reads and interprets the header information from the input DTI file. DTI files of type TED4 and UHL1 and type LSLA (with projection record) contain an (X,Y) origin offset value for the SW corner (row 1, column 1) of the file in the file header information. This origin offset value is used to derive the position of each post within the DTI file in absolute coordinate space. If the user wishes to generate an IFF file of contours whose coordinates are relative to the local origin (0.0, 0.0) of the SW corner (row 1, column 1) of the file, the DISABLE ABSOLUTE command should be issued.

The UNITS command should be given after defining the DTI file containing the input DTM, since appropriate default units of measurement are set up whenever an input DTI file is opened.

The following rules are applied when returning a default UNITS value:

- o MATRIX - LSLA files without a projection record; LSLA files with a projection record with a units entry that indicates the units are unset, or historical MIKE or ALVY files.
- o METRES - LSLA files with a projection record whose units entry indicates metres.
- o LATLONG - UHL1 and TED4 files, LSLA files with a projection record indicating the data is held as geographicals.
- o PROJECTION - LSLA files with a projection record whose units entry indicates the data is held as feet, sheet mms. or thousands of an inch.

For the rules applied when testing the validity of a UNITS argument supplied by the user see the UNITS command below.

It is possible to change the header values in a DTI file by use of the MATRIX package DTIPATCH utility, although care should be exercised to ensure that changes are made that are compatible with other stages of the production flowline. The DTI transformation utility DTITRANS may be used to change the coordinate basis of a DTI file before contouring it. DTITRANS allows, for example, transformation of a DTI file with post spacing in latitude and longitude, to grid, and vice versa.

DTICONTOUR output IFF file characteristics

The characteristics of the IFF file produced by DTICONTOUR is best described by reference to a listing of an example file. The numbers in **bold** type enclosed in round brackets are referred to below.

(1)
```
RA        0.0      1000.0        0.0      1000.0
```
(2)
```
HI
 8-OCT-1987 15:30 TIM         DTICONTOUR Create       00:03:48 00:00:00 00000001
```
(3)
```
MH 174 0
```
(4)
```
MD            2
```
(5)
```
NS Created by DTICONTOUR. Number of smoothing passes = 0
```
(6)
```
CC   .00000000E 000  .00000000E 000
     .10000000E 001  .00000000E 000
     .00000000E 000  .10000000E 001
     .00000000E 000  .00000000E 000
     .00000000E 000  .00000000E 000
     .00000000E 000  .00000000E 000
     .00000000E 000  .00000000E 000
     .00000000E 000  .00000000E 000
```

```
     .00000000E 000  .00000000E 000
     .00000000E 000  .00000000E 000
```
(7)
```
CP        0.0     1000.0        0.0     1000.0
          0.0        0.0        0.0        0.0
       1000.0        0.0     1000.0        0.0
       1000.0     1000.0     1000.0     1000.0
```
(8)
```
NO 1 0
```
(9)
```
NF 1 1
```
(10)
```
FS 0 0 0 0
```
(11)
```
TH 0
ST 3 0
      145.8      23.67
      248.9       26.7
      252.4       28.3
EF
```

(12)
```
Other features
 .
 .
 .
 .
 .

EO
EM
EJ
```

The effect of DTICONTOUR on individual IFF entries is as follows: (Item numbers refer to the bold labelling on the above example).

1. The range entry reflects the (x,y) range of the contour and spot height coordinates. If ENABLE FRAME is selected, it can be guaranteed that the range reflects the WINDOW command arguments.

2. DTICONTOUR produces a HI (HIstory) entry.

3. The MH (Map Header) is always unset.

4. The degree to which the type 2 MD (Map Descriptor) is set depends on the amount of information available in the input DTI file header. If input is from an LSLA, TED4 or UHL1 type DTI file the origin offset field is set; all coordinates within the file are offset such that the south west (bottom left) control point of the plot is (0.0, 0.0). If the input DTI file contains a Projection Record, details from the record are transferred to the MD entry of the IFF file. The scale field in the MD is by default set to 50000, as this information is unavailable in the DTI file header. The user may override this default scale by use of the SET SCALE command.

5. The NS (New Section) entry indicates the number of smoothing passes (if any) employed on the data.

6. The CC (Cubic Coefficients) entry is set to ensure a unit transform, (i.e. to have no effect).

7. The CP (Control Point) entry values reflect the coordinates specified by the DTICONTOUR WINDOW command. In the absence of a WINDOW command the CP entry values reflect the limits of the whole input DTI file.

8. By default all data are placed in layer 1. The DTICONTOUR SET LAYER command must be used to substitute another layer number.

9. Feature serial number (and internal sequence numbers) increase monotonically in steps of one from a base of one.

10. The feature code varies according to the status of the feature. Different feature codes may be specified to differentiate between contours, index contours, spot heights and the optional plot frame. Further feature codes may be specified for label features for contours, index contours, and spot heights.

11. The TH (line THickness) is always 0. The user should use a FRT file to define different line thickness values.

12. There can be up to 65535 features within the file. Bear in mind that labelling is held in the file as IFF features, and so even for a comparatively small plot, a large number of features may be generated.

DTICONTOUR command defaults.

On program startup, the following command defaults apply:

- o ABSOLUTE - enabled
- o DIAGNOSTICS - disabled
- o DIVIDEBY - disabled
- o FRAME - disabled
- o INDEX - enabled
- o INDEX_INTERVAL - 5 times the INTERVAL value
- o INTERVAL - 10% of the DTI file Z range
- o LABELLING - disabled, if enabled all contours are labelled

- LABEL SIZE - 0.5% of the maximum axis of the DTI (x,y) range
- LABEL MAXCURVE - 0.7 (scaled value in range 0.0 to 1.0)
- LABEL MINGAP - 10% of the maximum axis of the DTI file (x,y) range
- LABEL MODULUS - label every contour
- LABEL INDEX_MODULUS - label every index contour
- MULTIPLYBY - disabled
- SPOT_HEIGHTS - disabled
- SMOOTH - disabled
- TOFEET - disabled
- TOMETRES - disabled
- UNITS - dependent of DTI header type

IFF scale, layer number and feature codes:

- SCALE = 50000
- LAYER = 0
- CONTOUR_FC = 0
- FRAME_FC = 0
- INDEX_FC = 0
- SPOT_FC = 1
- CONTOUR_LABEL_FC = 0
- INDEX_LABEL_FC = 0
- SPOT_LABEL_FC = 0

DTICONTOUR commands

@

Take command input from the specified file.

FORMAT: @file-spec<CR>

Command parameters:

file-spec

The file to be opened and used for command input.

Any parts of the file-spec not supplied for the @ command will be taken from the default specification 'SYS$DISK:[].COM;0'.

DESCRIPTION:

DTICONTOUR offers the facility of command input from an indirect command file. The '@' character preceding a file-spec will cause DTICONTOUR to open and read commands from the specified file until:

1. a RETURN command is detected and command input is returned to SYS$COMMAND.

2. a GO command is detected - after completion of the contouring DTICONTOUR exits.

3. end-of-file is detected. This provokes an error message and command input is returned to SYS$COMMAND.

Nested command files are not supported (i.e. a command file containing an '@' command), although sequential '@' commands are supported when read from SYS$COMMAND.

As an aid to batch log interpretation DTICONTOUR will echo all commands read from an indirect command file.

Messages:

The following messages are specific to the @ command:

*** WARNING *** "@" must precede a file-spec

*** WARNING *** Indirect file error - returning to terminal input

*** ERROR *** Can't open indirect command file 'file-spec'

Examples:

```
$ DTICONTOUR<CR>
TVES module DTICONTOUR of 13:30:39 18-OCT-87
DTICONTOUR> @PRESETS<CR>
DTICONTOUR> ENABLE DIAGNOSTICS
DTICONTOUR> INTERVAL 10
DTICONTOUR> INDEX_INTERVAL 100
DTICONTOUR> ENABLE SPOT_HEIGHTS
DTICONTOUR> SET CONTOUR_FC 102
DTICONTOUR> SET INDEX_FC 103
DTICONTOUR> SET SPOT_FC 104
DTICONTOUR> SET LAYER 3
DTICONTOUR> RETURN
DTICONTOUR>
```

!

Treat all text to the right of the '!' as a comment.

FORMAT: ! [comment text]

Command parameters:

comment text

text that is to be treated as a comment and which will be excluded from command interpretation.

DESCRIPTION:

An exclamation mark is the standard DTM package comment delimiter. All text (and numbers) which lie to the right of a '!' character are excluded from command interpretation. Comments are useful for annotating command procedures used in batch processing etc.

Messages: None.

Examples:

```
DTICONTOUR> ! a comment for the sake of it<CR>
DTICONTOUR> WINDOW 0.0 0.0 900.0 900.0<CR>
DTICONTOUR> ZLIMITS 0.0 1290.0 !  limits are in metres<CR>
DTICONTOUR> IFF TEST3<CR>
DTICONTOUR> @PRESETS !  use indirect command file for presets<CR>
DTICONTOUR> INTERVAL 10
DTICONTOUR> INDEX_INTERVAL 100
DTICONTOUR> ENABLE SPOT_HEIGHTS
DTICONTOUR> SET CONTOUR_FC 102
DTICONTOUR> SET INDEX_FC 103
DTICONTOUR> SET SPOTFC 104
DTICONTOUR> SET LAYER 3
DTICONTOUR> RETURN
DTICONTOUR> FILEIN JOE!  get the DTI file<CR>
DTICONTOUR> !  do the contouring<CR>
DTICONTOUR> GO<CR>
 ELAPSED:  00:05:25.84 CPU:  0:00:05.71 BUFIO:  281 DIRIO:  46 FAULTS:  263
$
```

DISABLE ABSOLUTE

Cancels a previous ENABLE ABSOLUTE command.

FORMAT: DISABLE ABSOLUTE

Command parameters: None.

DESCRIPTION:

DISABLE ABSOLUTE cancels a previous ENABLE ABSOLUTE command. If DISABLE ABSOLUTE is given, then coordinate values required by the WINDOW command, supplied in metre or projection units, must be specified as an offset from the SW corner of the matrix.
By default window values should be specified as absolute coordinates.

Messages: None.

Examples:

DTICONTOUR> **DISABLE ABSOLUTE<CR>**
DTICONTOUR>

DISABLE DIAGNOSTICS

Cancels a previous ENABLE DIAGNOSTICS command.

FORMAT: DISABLE DIAGNOSTICS

Command parameters: None.

DESCRIPTION:

Disable a previous ENABLE DIAGNOSTICS command.

Messages: None.

Examples:

DTICONTOUR> **ENABLE DIAGNOSTICS<CR>**
DTICONTOUR> **SHOW ENABLE<CR>**

```
ABSOLUTE ..................................................... On
DIAGNOSTICS .................................................. On
DIVIDEBY ..................................................... Off
FRAME ........................................................ Off
INDEX ........................................................ On
INTEGER_HEIGHT ............................................... Off
(Contour heights will be placed in type 3 AC entries)
LABELLING .................................................... Off
MULTIPLY .....................................................  Off
PME .......................................................... Off
SMOOTHING .................................................... Off
SPOT_HEIGHTS ................................................. Off
(Single point contours to be omitted from output)
TOFEET ....................................................... Off
TOMETRES ..................................................... Off
```

DTICONTOUR> **DISABLE DIAGNOSTICS<CR>**
DTICONTOUR>

DISABLE DIVIDEBY

Disables a previous ENABLE DIVIDEBY command.

FORMAT: DISABLE DIVIDEBY

Command parameters: None.

DESCRIPTION:

DISABLE DIVIDEBY allows the user to disable a previous ENABLE DIVIDEBY command.

Messages: None.

Examples:

DTICONTOUR> **SHOW ENABLE<CR>**

```
ABSOLUTE ..................................................... On
DIAGNOSTICS .................................................. Off
DIVIDEBY ..................................................... Off
FRAME ........................................................ Off
INDEX ........................................................ On
INTEGER_HEIGHT ............................................... Off
(Contour heights will be placed in type 3 AC entries)
LABELLING .................................................... Off
MULTIPLY ....................................................  Off
PME .......................................................... Off
SMOOTHING .................................................... Off
SPOT_HEIGHTS ................................................. Off
(Single point contours to be omitted from output)
TOFEET ....................................................... Off
TOMETRES ..................................................... Off
```

DTICONTOUR> **DISABLE DIVIDEBY<CR>**
DTICONTOUR>

DISABLE FRAME

Disables a previous ENABLE FRAME command.

FORMAT: DISABLE FRAME

Command parameters: None.

DESCRIPTION:

DISABLE FRAME cancels the effect of a previous ENABLE FRAME command.

Messages: None.

Examples:

```
DTICONTOUR> DISABLE FRAME<CR>
DTICONTOUR>
```

DISABLE INDEX

Disable the generation of index contours.

FORMAT: DISABLE INDEX

Command parameters: None.

DESCRIPTION:

DISABLE INDEX enables the user to suppress the generation of index contours. A similar effect may be achieved by specifying an INDEX_INTERVAL of zero.

Messages: None.

Examples:

DTICONTOUR> **DISABLE INDEX<CR>**
DTICONTOUR>

DISABLE LABELLING

Disable the generation of contour and spot height label features.

FORMAT: DISABLE LABELLING

Command parameters: None.

DESCRIPTION:

DISABLE LABELLING cancels the effect of a previous ENABLE LABELLING command.

Messages: None.

Examples:

DTICONTOUR> **DISABLE LABELLING<CR>**
DTICONTOUR>

DISABLE MULTIPLYBY

Disables a previous ENABLE MULTIPLYBY command.

FORMAT: DISABLE MULTIPLYBY

Command parameters: None.

DESCRIPTION:

DISABLE MULTIPLYBY allows the user to disable a previous ENABLE MULTIPLYBY command.

Messages: None.

Examples:

DTICONTOUR> **DISABLE MULTIPLYBY<CR>**
DTICONTOUR>

DISABLE PME

DISABLE PME disables the PME performance monitor.

FORMAT: DISABLE PME

Command parameters: None.

DESCRIPTION:

PME is a code optimisation tool and should be invoked by LSL software personnel only.

The DISABLE PME command causes the PME_EXIT routine to be invoked.

Message:

The following warning message is specific to the DISABLE PME command:

*** WARNING *** You are not using PME!

Examples:

DTICONTOUR> **DISABLE PME<CR>**
DTICONTOUR>

DISABLE SMOOTH

Cancels the effect of a previous ENABLE SMOOTH command.

FORMAT: DISABLE SMOOTH

Command parameters: None.

DESCRIPTION:

DISABLE SMOOTH disables the effect of a previous ENABLE SMOOTH command.

Message: None.

Examples:

DTICONTOUR> **ENABLE SMOOTH 3<CR>** DTICONTOUR> **SHOW ENABLE<CR>**

```
ABSOLUTE ...................................................... On
DIAGNOSTICS ................................................... Off
DIVIDEBY ...................................................... Off
FRAME ......................................................... Off
INDEX ......................................................... On
INTEGER_HEIGHT ................................................ Off
(Contour heights will be placed in type 3 AC entries)
LABELLING ..................................................... Off
MULTIPLY .....................................................  Off
PME ........................................................... Off
SMOOTHING ..................................................... On
Number of smoothing iterations................................. 3
SPOT_HEIGHTS .................................................. Off
(Single point contours to be omitted from output)
TOFEET ........................................................ Off
TOMETRES ...................................................... Off
```

DTICONTOUR> **DISABLE SMOOTH<CR>**
DTICONTOUR> **SHOW ENABLE<CR>**

```
ABSOLUTE ...................................................... On
DIAGNOSTICS ................................................... Off
DIVIDEBY ...................................................... Off
FRAME ......................................................... Off
INDEX ......................................................... On
INTEGER_HEIGHT ................................................ Off
(Contour heights will be placed in type 3 AC entries)
LABELLING ..................................................... Off
MULTIPLY .....................................................  Off
PME ........................................................... Off
SMOOTHING ..................................................... Off
```

```
SPOT_HEIGHTS ............................................... Off
(Single point contours to be omitted from output)
TOFEET ..................................................... Off
TOMETRES ................................................... Off

DTICONTOUR>
```

DISABLE SPOT_HEIGHTS

Cancels the effect of a previous ENABLE SPOT_HEIGHTS command.

FORMAT: DISABLE SPOT_HEIGHTS

Command parameters: None.

DESCRIPTION:

DISABLE SPOT_HEIGHTS disables the effect of a previous ENABLE SPOT_HEIGHTS command.

Message: None.

Examples:

DTICONTOUR> **DISABLE SPOT_HEIGHTS<CR>**
DTICONTOUR>

DISABLE TOFEET

Cancels the effect of a previous ENABLE TOFEET command.

FORMAT: DISABLE TOFEET

Command parameters: None.

DESCRIPTION:

It is possible that the input DTI file may have heights recorded in different measurement systems to that required in the output IFF contour file. Two preset height conversion options are available: ENABLE TOMETRES and ENABLE TOFEET.

The ENABLE TOFEET command enables the conversion of heights held in the DTI file in metres to feet. It has the same effect as an explicit ENABLE MULTIPLYBY 3.2808455 command. The contours will then be produced in feet.

If one of the height modification options is selected using ENABLE MULTIPLYBY, ENABLE TOFEET etc., you must give the contour interval and any ZLIMITS in the target measurement system or height range (ie feet for the ENABLE TOFEET command). Failure to do this will result in unexpected densities of contours!

N.B. DISABLE TOFEET will not cancel an explicit ENABLE MULTIPLYBY 3.2808455 command.

Messages: None.

Examples:

DTICONTOUR> **DISABLE TOFEET<CR>**
DTICONTOUR>

DISABLE TOMETRES

Cancels the effect of a previous ENABLE TOMETRES command.

FORMAT: DISABLE TOMETRES

Command parameters: None.

DESCRIPTION:

It is possible that the input DTI file may have heights recorded in different measurement systems to that required in the IFF output file. The model must be relative to one system only. Two height conversion options are available: ENABLE TOFEET and ENABLE TOMETRES.

The ENABLE TOMETRES command results in the conversion of heights held in the DTI file in feet to metres. It has the same effect as an explicit ENABLE DIVIDEBY 3.2808455 command. The IFF file contours will then be produced in metres.

If one of the height modification options is selected using ENABLE MULTIPLYBY, ENABLE TOMETRES etc., you must give the contour interval and any ZLIMITS in the target measurement system or height range (ie metres for the ENABLE TOMETRES command). Failure to do this may result in an unexpected contour density!

N.B. DISABLE TOMETRES will not cancel an explicit ENABLE DIVIDEBY 3.2808455 command.

Messages: None.

Examples:

DTICONTOUR> **DISABLE TOMETRES<CR>**
DTICONTOUR>

ENABLE ABSOLUTE

Selects the use of absolute coordinate values.

FORMAT: ENABLE ABSOLUTE

Command parameters: None.

DESCRIPTION:

If ENABLE ABSOLUTE is given, then coordinate values required by the WINDOW command, supplied in metre or projection units, must be specified as absolute (rather than relative) coordinate values.

For example if the projection indicates U.K. National Grid, then the WINDOW values may be specified as 6 figure National Grid coordinates.
By default window values should be specified as absolute coordinates. This option can be disabled using the DISABLE ABSOLUTE command.

Messages: None.

Examples:

DTICONTOUR> **ENABLE ABSOLUTE<CR>**
DTICONTOUR>

ENABLE DIAGNOSTICS

Enable diagnostic printout.

FORMAT: ENABLE DIAGNOSTICS

Command parameters: None.

DESCRIPTION:

ENABLE DIAGNOSTICS allows the user to enable diagnostic printout.

Because it is usually used in a batch processing environment, by default DTICONTOUR produces minimal diagnostic printout. If however, the user wishes to receive indications of processing progress and of the effect of windowing on data input, diagnostic printout may be selected with the ENABLE DIAGNOSTICS command.

On a heavily loaded computer it may be reassuring to ENABLE DIAGNOSTICS for the contouring stage of DTICONTOUR processing to indicate progress through the data set. A messages indicating contouring progress is issued as contours of each new contour height are generated.

Messages: None.

Examples:

DTICONTOUR> **ENABLE DIAGNOSTICS<CR>**
DTICONTOUR> **SHOW ENABLE<CR>**

```
ABSOLUTE .................................................... On
DIAGNOSTICS ................................................. On
DIVIDEBY .................................................... Off
FRAME ....................................................... Off
INDEX ....................................................... On
INTEGER_HEIGHT .............................................. Off
(Contour heights will be placed in type 3 AC entries)
LABELLING ................................................... Off
MULTIPLY ...................................................  Off
PME ......................................................... Off
SMOOTHING ................................................... Off
SPOT_HEIGHTS ................................................ Off
(Single point contours to be omitted from output)
TOFEET ...................................................... Off
TOMETRES .................................................... Off

DTICONTOUR>
```

ENABLE DIVIDEBY

Enable division of input file heights by a specified floating point constant.

FORMAT: ENABLE DIVIDEBY denominator

Command parameters:

denominator

The value by which all input file heights are to be divided.

DESCRIPTION:

The ENABLE DIVIDEBY enables the user to divide all incoming DTI heights by a specified (floating point) constant. For example, the command ENABLE DIVIDEBY 2.0 will cause all incoming heights to be divided by 2.0. An ENABLE DIVIDEBY 3.2808455 command has the same effect as an ENABLE TOMETRES command.

Messages:

The following messages are specific to the ENABLE DIVIDEBY command:

```
*** WARNING *** You are already planning to multiply by 'constant'
                ENABLE DIVIDEBY command now overrides ENABLE MULTIPLYBY command
*** WARNING *** You must specify a value for DIVIDEBY
```

Examples:

DTICONTOUR> **ENABLE DIVIDEBY 10.0<CR>**
DTICONTOUR> **SHOW ENABLE<CR>**

```
ABSOLUTE .................................................... On
DIAGNOSTICS ................................................. Off
DIVIDEBY .................................................... On
Matrix heights to be divided by ............................. 10.0
FRAME ....................................................... Off
INDEX ....................................................... On
INTEGER_HEIGHT .............................................. Off
(Contour heights will be placed in type 3 AC entries)
LABELLING ................................................... Off
MULTIPLY ....................................................  Off
PME ......................................................... Off
SMOOTHING ................................................... Off
SPOT_HEIGHTS ................................................ Off
(Single point contours to be omitted from output)
TOFEET ...................................................... Off
TOMETRES .................................................... Off
```

ENABLE FRAME

Causes DTICONTOUR to output a frame feature to the IFF file.

FORMAT: ENABLE FRAME

Command parameters: None.

DESCRIPTION:

Causes DTICONTOUR to output a frame feature to the IFF file which represents the extent of the contour window defined by the WINDOW command.

If no WINDOW command is issued, the frame represents the boundary of the whole area covered by the DTI file.

The frame feature is given the feature code specified with the SET FRAME_FC command. In the absence of a SET FRAME_FC command, feature code 0 is used.

By default, DTICONTOUR does not create a frame feature.

Messages: None.

Examples:

DTICONTOUR> **ENABLE FRAME<CR>**
DTICONTOUR>

ENABLE INDEX

Enable the generation of index contours (default).

FORMAT: ENABLE INDEX

Command parameters: None.

DESCRIPTION:

ENABLE INDEX enables the user to cancel the effect of a previous DISABLE INDEX command. By default DTICONTOUR will always generate index contours.

Messages: None.

Examples:

DTICONTOUR> **ENABLE INDEX<CR>**
DTICONTOUR>

ENABLE LABELLING

Enable the generation of contour and spot height label features.

FORMAT: ENABLE LABELLING

Command parameters:

None

DESCRIPTION:

ENABLE LABELLING enable the user to instruct DTICONTOUR to generate label features for contours, index contours, and if ENABLE SPOT_HEIGHTS is specified, spot heights.

Messages: None.

Examples:

DTICONTOUR> **ENABLE LABELLING<CR>**
DTICONTOUR>

ENABLE MULTIPLYBY

Enable multiplication of input file heights by a specified floating point constant.

FORMAT: ENABLE MULTIPLYBY constant

Command parameters:

constant

The value by which all input file heights are to be multiplied.

DESCRIPTION:

The ENABLE MULTIPLYBY enables the user to multiply all incoming DTI heights by a specified (floating point) constant. For example, the command ENABLE MULTIPLYBY 2.0 will cause all incoming heights to be multiplied by 2.0. An ENABLE MULTIPLYBY 3.2808455 command has the same effect as an ENABLE TOFEET command.

Messages:

The following messages are specific to the ENABLE MULTIPLYBY command:

```
*** WARNING *** You are already planning to divide by 'constant'
                ENABLE MULTIPLY command now overrides ENABLE DIVIDEBY command
*** WARNING *** You must specify a value for MULTIPLYBY
```

Examples:

DTICONTOUR> **ENABLE MULTIPLYBY 10.0<CR>**
DTICONTOUR> **SHOW ENABLE<CR>**

```
ABSOLUTE ................................................... On
DIAGNOSTICS ................................................ Off
DIVIDEBY ................................................... Off
FRAME ...................................................... Off
INDEX ...................................................... On
INTEGER_HEIGHT ............................................. Off
(Contour heights will be placed in type 3 AC entries)
LABELLING .................................................. Off
MULTIPLY ................................................... On
Matrix heights to be multiplied by ......................... 10.0
PME ........................................................ Off
SMOOTHING .................................................. Off
SPOT_HEIGHTS ............................................... Off
(Single point contours to be omitted from output)
TOFEET ..................................................... Off
TOMETRES ................................................... Off
```

```
DTICONTOUR>
```

ENABLE PME

ENABLE PME enables the PME performance monitor.

FORMAT: ENABLE PME

Command parameters: None.

DESCRIPTION:

The ENABLE PME and DISABLE PME commands are reserved for Laser-Scan use. PME is a code optimisation tool and should be invoked by LSL software personnel only.

ENABLE PME causes the PME_INIT routine to be invoked.

Message:

The following warning message is specific to the ENABLE PME command:

*** WARNING *** You are already using PME!

Examples:

```
$ DTICONTOUR<CR>
TVES module DTICONTOUR of 13:30:39  18-OCT-87
DTICONTOUR> ENABLE PME<CR>
DTICONTOUR>
```

ENABLE SMOOTH

Enables the use of smoothing filters within DTICONTOUR.

FORMAT: ENABLE SMOOTH iterations

Command parameter:

iterations

The number of smoothing iterations required.

DESCRIPTION:

ENABLE SMOOTH causes DTICONTOUR to apply a moving least squares filter to the contour strings generated during the matrix lacing process. The number of times that the filter is applied is controlled by the 'iterations' argument.

By default DTICONTOUR does not smooth the contour strings

Because linear interpolation is used in DTICONTOUR, areas of equal Z value may lead to the formation of contours resembling a "staircase", where the contour makes right angled turns. There is insufficient information for the simple algorithm to lace the contours relative to a more global context.

It is important to note that although cartographically unappealing, the contours produced by DTICONTOUR are "truth". Because linear interpolation is used, the contours represent what is really there in the DTI file, not the expression of a high order surface patch. The linear interpolation used is not only fast, but provides a powerful quality check on the DTI file contents.

Message: None.

Examples:

DTICONTOUR> **ENABLE SMOOTH 3<CR>** DTICONTOUR> **SHOW ENABLE<CR>**

```
ABSOLUTE ................................................... On
DIAGNOSTICS ................................................ Off
DIVIDEBY ................................................... Off
FRAME ...................................................... Off
INDEX ...................................................... On
INTEGER_HEIGHT ............................................. Off
(Contour heights will be placed in type 3 AC entries)
LABELLING .................................................. Off
MULTIPLY ...................................................  Off
PME ........................................................ Off
SMOOTHING .................................................. On
Number of smoothing iterations.............................. 3
SPOT_HEIGHTS ............................................... Off
```

```
(Single point contours to be omitted from output)
TOFEET ...................................................... Off
TOMETRES .................................................... Off

DTICONTOUR>
```

ENABLE SPOT_HEIGHTS

Causes DTICONTOUR to output any single point spot height features to the IFF file.

FORMAT: ENABLE SPOT_HEIGHTS

Command parameters: None.

DESCRIPTION:

It is possible that single DTI post values lying in peaks and troughs of the DTI surface, are at exactly the same height as a contour. This would result in the generation of a single point contour. As such features are conventionally represented by spot heights, the ENABLE SPOT_HEIGHTS option is provided to output any single point contours as spot height features to the IFF file.

By default, DTICONTOUR does not create spot height features; the single point features are discarded.

The spot height features are given the feature code specified with the SET SPOT_FC command. In the absence of a SET SPOT_FC command, feature code 1 is used.

Spot height features are always given labels if the ENABLE LABELLING option is selected. The feature code of the spot height label features is specified with the SET SPOT_LABEL_FC command argument. In the absence of a SET SPOT_LABEL_FC command, feature code 0 is used.

Messages: None.

Examples:

DTICONTOUR> **SET SPOT_FC 23<CR>**
DTICONTOUR> **SET SPOT_LABEL_FC 3<CR>**
DTICONTOUR> **ENABLE SPOT_HEIGHTS<CR>**
DTICONTOUR>

ENABLE TOFEET

The ENABLE TOFEET command enables the conversion of input file heights from metres to feet.

FORMAT: ENABLE TOFEET

Command parameters: None.

DESCRIPTION:

It is possible that the DTI input files may have heights recorded in a different measurement system to that required in the IFF output file. Two height conversion options are available: ENABLE TOMETRES and ENABLE TOFEET.

The ENABLE TOFEET command enables the conversion of input DTI file heights from feet to metres. It has the same effect as an explicit ENABLE MULTIPLYBY 3.2808455 command. The IFF file will then be produced in feet.

If one of the height modification options is selected using ENABLE MULTIPLYBY, ENABLE TOFEET etc., you must give the contour interval and any ZLIMITS in the target measurement system or height range (ie feet for the ENABLE TOFEET command). Failure to do this may result in an unexpected density of contours!

Messages: None.

Examples:

DTICONTOUR> **ENABLE TOFEET<CR>**
DTICONTOUR>

ENABLE TOMETRES

The ENABLE TOMETRES command enables the conversion of input DTI file heights from feet to metres.

FORMAT: ENABLE TOMETRES

Command parameters: None.

DESCRIPTION:

It is possible that the input DTI file may have heights recorded in different measurement systems to that required in the IFF output file. Two height conversion options are available: ENABLE TOFEET and ENABLE TOMETRES.

The ENABLE TOMETRES command results in the conversion of heights held in the DTI file in feet to metres. It has the same effect as an explicit ENABLE DIVIDEBY 3.2808455 command. The IFF contours will then be produced in metres.

If one of the height modification options is selected using ENABLE MULTIPLYBY, ENABLE TOMETRES etc., you must give the contour interval and any ZLIMITS in the target measurement system or height range (ie metres for the ENABLE TOMETRES command). Failure to do this may result in an unexpected density of contours!

Messages: None.

Examples:

DTICONTOUR> **ENABLE TOMETRES<CR>**
DTICONTOUR>

FILEIN

Specifies a DTI file that is to be opened and used for data input.

FORMAT: FILEIN file-spec

COMMAND PARAMETERS:

file-spec

The specification of the DTI file to be opened for data input.

Any parts of the file-spec not supplied for the FILEIN command will be taken from the default file specification 'LSL$DTI:DTI.DTI;0'.

DESCRIPTION:

The FILEIN command opens and maps into memory a DTI file, containing the matrix data to be processed. Details derived from the header of the file are displayed on the terminal to confirm that the file has been successfully opened.

A FILEIN command must be issued before the UNITS, WINDOW, ZLIMITS or GO commands will be accepted.
If the logical name LSL$MATRIX_DEFAULT_WINDOW is defined with a value of "1", then a default area of interest defined in matrix units of bottom left hand corner 1,1 and top right hand corner 200,200 is set when the file is opened. If the logical name is absent or has any other value, or if the input DTI file has fewer than 200 columns or 200 rows, then a default window equivalent to the total matrix is set. The area of interest may be redefined using the command WINDOW.

Messages:

The following messages are specific to the FILEIN command:

*** WARNING *** You must specify a file-spec argument to the FILEIN command
*** ERROR *** Unable to interpret input file-spec

Examples:

```
DTICONTOUR> FILEIN 52N09EDTEDUTM<CR>
DTI file LSL$DTI:52N09EDTEDUTM.DTI;0 opened for read

File    :  LSL$DTI:52N09EDTEDUTM.DTI;0
Header  :  LSLA  Data:  WORD

Units are metres

Matrix Coverage   SW:  500083.88 5761241.50    NE:  568683.88 5872941.50
```

```
Matrix Window     SW:  500083.88 5761241.50    NE:  568683.88 5872941.50
Matrix Interval    E:     100.00                N:     100.00
Value Range        :      4   to    434

DTICONTOUR>
```

GO

Perform the contouring using default or user supplied values for contour interval, labelling etc.

FORMAT: GO

Command parameters: None.

DESCRIPTION:

The GO command can only be issued after FILEIN and IFF commands have been used.

The number of contours produced is controlled by the arguments supplied to the INTERVAL and INDEX_INTERVAL commands.

The type of contours produced is determined by using the ENABLE SMOOTH command. The default is to produce simple linear contours, which tend to wander in a "staircase" like manner in areas of low relief. If ENABLE SMOOTH is selected smoothed contours will result with the "staircases" suppressed.

On a heavily loaded computer it may be reassuring to ENABLE DIAGNOSTICS for the contouring stage of DTICONTOUR processing to indicate progress through the data set. A message indicating contouring progress is then issued as contours of each new contour height are generated.

Messages: None.

Examples:

```
DTICONTOUR> ENABLE DIAGNOSTICS<CR>
DTICONTOUR> GO<CR>

=====================  STARTING CONTOUR GENERATION  =====================

Units are metres

Matrix Coverage   SW:  500083.88 5761241.50    NE:  568683.88 5872941.50
Contouring window SW:  500083.88 5761241.50    NE:  568683.88 5872941.50

All heights between     4.000 and   434.000 will be contoured


               +------------------------+
               | Tracing index contours |
               +------------------------+
Tracing index contours with a height of   250.000


            +--------------------------------+
```

```
            | Tracing intermediate contours |
            +-------------------------------+
Tracing intermediate contours with a height of    50.000
Tracing intermediate contours with a height of   100.000
Tracing intermediate contours with a height of   150.000
Tracing intermediate contours with a height of   200.000
Tracing intermediate contours with a height of   300.000
Tracing intermediate contours with a height of   350.000
Tracing intermediate contours with a height of   400.000
 ELAPSED: 00:05:25.84  CPU: 0:00:05.71  BUFIO: 281  DIRIO: 46  FAULTS: 263
$
```

HELP

Give help on a subject

FORMAT: HELP subject

Command parameters:

subject

The subject on which help is required.

Description:

The HELP command looks the rest of the line up in the TVES HELP library. This library contains a brief summary of the operation of each command.

The information is looked up in the DTICONTOUR section of the TVES help library, LSL$HELP:TVES.HLB.

Messages:

Where required, warning messages are output via the VMS LBR$OUTPUT_HELP utility.

Examples:

DTICONTOUR> **HELP ENABLE TOMETRES<CR>**

DTICONTOUR

ENABLE

TOMETRES

It is possible that the input DTI file may have heights recorded in different a measurement systems to that required in the IFF output file. Two height conversion options are available: ENABLE TOFEET and ENABLE TOMETRES.

The ENABLE TOMETRES command results in the conversion of heights held in the DTI file in feet to metres. It has the same effect as an explicit ENABLE DIVIDEBY 3.2808455 command. The IFF file contours will then be produced in metres.

If one of the height modification options is selected using ENABLE MULTIPLYBY, ENABLE TOMETRES etc., you must give the contour interval and any ZLIMITS in the target measurement system (i.e. metres for the ENABLE TOMETRES command). Failure to do this may result in an unexpected contour density!

DTICONTOUR>

IFF

Specifies the file-spec of the IFF file which is to receive the contours.

FORMAT: IFF file-spec

COMMAND PARAMETERS:

file-spec

The file-spec of the IFF file which is to receive the contours.

Any parts of the file-spec not supplied for the IFF command will be taken from the default 'LSL$IF:IFF.IFF;0'.

DESCRIPTION:

The IFF command enables specification of the IFF file which is to receive the contours.

The IFF command must be issued before the GO command.

Messages:

The following messages are specific to the IFF command:

```
*** WARNING *** You must specify a file-spec argument to the IFF command
*** ERROR *** reading output file-spec
```

Examples:

```
DTICONTOUR> IFF DUA3:[DEMONSTRATION]IDAHO<CR>
%LSLLIB-I-IFFOPENED, DUA3:[DEMONSTRATION]IDAHO.IFF;6 opened for write
DTICONTOUR>
```

INTERVAL

Specifies the height interval between successive contours.

FORMAT: INTERVAL interval

Command parameters:

interval

The height interval between successive contours (floating point or integer).

DESCRIPTION:

The INTERVAL argument specifies the height interval between successive contours.

Failure to set the contour interval will result in the application of a default interval of 10% of the height range within the area of the DTI file defined by the WINDOW command. If no WINDOW command is issued then 10% of the height range of the whole DTI file will be used. If ZLIMITS are specified, with or without the WINDOW command, then the default contour interval is 10% of the height range defined by the ZLIMITS arguments.

If you intend to use the TOMETRES, TOFEET, MULTIPLYBY, or DIVIDEBY options, ensure that you specify the contour interval relative to the TARGET measurement system. Failure to do this will result in far more (or less!) contours than you had bargained for!

Messages:

```
*** ERROR *** Missing interval argument
Only one value please !
Taking first value only ('real')
```

Examples:

DTICONTOUR> **INTERVAL 10.0<CR>**
DTICONTOUR>

INDEX_INTERVAL

Specifies the height interval between successive index contours.

FORMAT: INDEX_INTERVAL interval

Command parameters:

interval

The height interval between successive index contours (floating point or integer).

DESCRIPTION:

The INDEX_INTERVAL argument specifies the height interval between successive index contours. Index contours are given a different feature code (specified by the SET INDEX_FC command) in the IFF output file to enable them to be differentiated from other (or "intermediate") contours.

Failure to specify the index contour interval will result in the application of a default index interval of five times the ordinary contour interval.

If you intend to use the TOMETRES, TOFEET, MULTIPLYBY, or DIVIDEBY options, ensure that you specify the index contour interval relative to the TARGET measurement system. Failure to do this will result in far more (or less!) contours than you had bargained for!

Messages:

*** ERROR *** Missing interval argument
Only one value please !
Taking first value only ('real')

Examples:

DTICONTOUR> **INTERVAL 10.0<CR>**
DTICONTOUR> **INDEX_INTERVAL 200.0<CR>**
DTICONTOUR>

LABEL FLOAT

Specifies that contour labels are to be displayed as floating point numbers.

FORMAT: LABEL FLOAT

Command parameters: None.

DESCRIPTION:

LABEL FLOAT enables the user to specify that contour labels are to be displayed as floating point numbers. DTICONTOUR generates labels to 5 significant figures.

A very long floating point number will often fail the contour curve test (specified using LABEL MAXCURVE). If only zeros follow the decimal point, DTICONTOUR will display the label as an integer to increase the likelihood of the label site being acceptable.

Spot height labels are always displayed as floating point values.

By default DTICONTOUR will generate floating point labels. Integer labels are generated only if the LABEL INTEGER command is issued.

Messages: None.

Examples:

DTICONTOUR> **LABEL FLOAT<CR>**
DTICONTOUR>

LABEL INTEGER

Specifies that contour labels are to be displayed as integer numbers.

FORMAT: LABEL INTEGER

Command parameters: None.

DESCRIPTION:

LABEL INTEGER enables the user to specify that contour labels are to be displayed as integer numbers.

Spot height labels are always displayed as floating point values.

By default DTICONTOUR will generate floating point labels. Integer labels are generated only if the LABEL INTEGER command is issued.

Messages: None.

Examples:

DTICONTOUR> **LABEL INTEGER<CR>**
DTICONTOUR>

LABEL INDEX_MODULUS

Specifies the height modulus for index contour labelling.

FORMAT: LABEL INDEX_MODULUS height

Command parameter:

height

A real height value. Index contours with heights that are exactly divisible by this value will be considered for labelling.

DESCRIPTION:

The LABEL INDEX_MODULUS command enables the user to specify a real (floating point) height to determine which index contours are to be considered for annotation. Only index contours which are exactly divisible by this value will be considered for labelling.

For example, if the user specifies that LABEL INDEX_MODULUS 100.0 is to be used in conjunction with an index contour interval of 50.0, every other index contour will be labelled.

By default an index height modulus of 1.0 is used, i.e. all index contours are considered for labelling.

A LABEL INDEX_MODULUS argument of zero will suppress labelling of index contours.

Messages:

```
*** ERROR *** Missing modulus argument
Only one value please !
Taking first value only ('real')
```

Examples:

```
DTICONTOUR> INTERVAL 10.0<CR>
DTICONTOUR> INDEX_INTERVAL 40.0<CR>
DTICONTOUR> LABEL INDEX_MODULUS 80.0<CR>
DTICONTOUR>
```

LABEL MAXCURVE

Specifies the maximum curve on a contour which will be considered as a suitable site for label insertion.

FORMAT: LABEL MAXCURVE curve

Command parameter:

curve

A real value in the range 0.0 to 1.0 which determines the maximum rate of change of curvature along along a line before that section of line is considered to be unsuitable for labelling.

DESCRIPTION:

LABEL MAXCURVE enables the user to alter the default parameter which defines the maximum rate of change of curvature along along a line before that section of line is considered to be unsuitable for labelling.

The acceptable range of values for LABEL MAXCURVE is 0.0 to 1.0. As LABEL MAXCURVE approaches 0.0, less and less labelling will be output because the tolerance of curve sharpness is decreased. A LABEL MAXCURVE value of 1.0 will result in labelling being output regardless of curve sharpness. The default value is 0.7

Messages:

```
*** ERROR *** Missing maximum curvature argument
Only one value please !
Taking first value only ('real')
```

Examples:

```
DTICONTOUR> LABEL MAXCURVE 0.3<CR>
DTICONTOUR>
```

LABEL MINGAP

Specifies the minimum distance travelled along a contour before DTICONTOUR will begin looking for a suitable site for label insertion.

FORMAT: LABEL MINGAP distance

Command parameter:

distance

A real value, expressed in IFF output file units, which defines the minimum distance travelled along a contour before DTICONTOUR will begin looking for a suitable site for label insertion.

DESCRIPTION:

The LABEL MINGAP argument specifies the minimum distance travelled along a contour before DTICONTOUR will begin looking for a suitable site for label insertion. By default DTMCREATE uses 10% of the maximum (x,y) axis of the window defined by the WINDOW command. If no WINDOW command is issued, then 10% of the maximum (x,y) axis of the whole DTI file is used.

The relationship between the LABEL MINGAP argument value and the frequency of labels occurring along a contour line can be subtle. If the LABEL MINGAP argument is too big, DTICONTOUR may be unable to label any contours because the contours are shorter then the LABEL MINGAP distance. Less obvious, however, is the interaction between a dense contour pattern and a large LABEL MINGAP distance. Because the inter-contour gap is smaller in a dense contour map there will be fewer sites suitable for the insertion of labels anyway. If, in addition a large LABEL MINGAP value is specified, many of the possible label sites will be rendered unsuitable. The net result is a contour map with few or no labels. As a general rule of thumb, the LABEL MINGAP value should be lowered for DTI files defining dense contour areas.

If there is wide variation in contour density between sub-areas of the DTI file, the contours should be generated using several DTICONTOUR runs, each with a different window and a LABEL MINGAP value appropriate to the contour density. The windows should not overlap. When all the windows are complete, the IFF contour files can be edge butted together using the IMP (IFF Map Processing) package utility IMERGE.

As linear interpolation is used for contour generation, edge matching should be perfect. Some edge matching imperfections may be introduced, however, by the ENABLE SMOOTH option in areas of little or no relief variation.

Messages:

*** ERROR *** Missing label minimum gap argument
Only one value please !
Taking first value only ('real')

Examples:

DTICONTOUR> **LABEL MINGAP 130.0<CR>**

LABEL MODULUS

Specifies the height modulus for index contour labelling.

FORMAT: LABEL MODULUS height

Command parameter:

height

A real height value. Intermediate contour heights that are exactly divisible by this value will be considered for labelling.

DESCRIPTION:

The LABEL MODULUS command enables the user to specify a real (floating point) height to determine which intermediate (i.e. non-index) contours are to be considered for labelling. Only intermediate contours which are exactly divisible by this value will be considered for labelling.

For example, if the user specifies that LABEL MODULUS 10.0 is to be used in conjunction with a contour interval of 5.0, every other contour will be labelled.

By default a height modulus of 1.0 is used, i.e. all intermediate contours are considered for labelling.

A LABEL MODULUS argument of zero will suppress labelling of intermediate contours.

Messages:

*** ERROR *** Missing modulus argument
Only one value please !
Taking first value only ('real')

Examples:

DTICONTOUR> **INTERVAL 10.0<CR>**
DTICONTOUR> **INDEX_INTERVAL 40.0<CR>**
DTICONTOUR> **LABEL MODULUS 10.0<CR>** ! label all contours

LABEL SIZE

Specifies the size of the contour labels.

FORMAT: LABEL SIZE size

Command parameter:

size

The size of the contour labels in IFF output file units.

DESCRIPTION:

The LABEL SIZE argument enables the user to alter the default size for contour labels. This default is calculated as 0.5% of the DTI maximum (x,y) range. The value displayed by the SHOW LABELLING command is the height of the label characters in IFF output file units. New label sizes should always be expressed in terms of output IFF file units, bearing in mind of course the contour density and legibility requirements.

The LABEL SIZE argument actually specifies the height of the label characters, the character width is calculated to be 0.7 of the character height.

It is most important that the user appreciates the relationship between the contour label size and the frequency of labels occurring along a contour line. If the label size is too big, DTICONTOUR will be able to fit the label into the gap between adjacent contours less often than in the case when the label size is small. Clearly, a dense contour map will offer less opportunities for labelling than a contour map having few contours, and even fewer opportunities if the label size is big.

Messages:

```
*** ERROR *** Missing label size argument
Only one value please !
Taking first value only ('real')
```

Examples:

```
DTICONTOUR> LABEL SIZE 3.0<CR>
DTICONTOUR>
```

PAUSE

Pauses DTICONTOUR execution.

FORMAT: PAUSE

Command parameters: None.

DESCRIPTION:

Pauses DTICONTOUR execution and issues a prompt for a carriage return to continue execution. This command is designed for use in software demonstration situations.

Messages: None.

Examples:

DTICONTOUR> **PAUSE<CR>**

Press <RETURN> to continue**<CR>**
DTICONTOUR>

QUIT

Quit from DTICONTOUR.

FORMAT: QUIT

Command parameters: None.

Description:

The QUIT command causes DTICONTOUR to exit immediately, closing all input files and closing and deleting the output IFF file.

<CTRL/Z> (pressing the Ctrl and Z keys together) may also be used to quit from the program.

Messages: None.

Examples:

```
DTICONTOUR> QUIT<CR>
 ELAPSED:  00:00:20.04 CPU:  0:00:04.71 BUFIO:  281 DIRIO:  46 FAULTS:  263
$
```

RETURN

Restores command input from an indirect file to SYS$COMMAND.

FORMAT: RETURN

Command parameters: None.

DESCRIPTION:

Restores command input from an indirect file to SYS$COMMAND.

A typical application is to allow the user to use an indirect command file to set up those run time defaults which are constant within a flowline and then return to input from the terminal (or batch stream) for the run specific commands. To do this RETURN must be the last command in the indirect command file.

Messages:

The following messages are specific to the RETURN command:

RETURN command detected - returning to terminal input
RETURN command ignored - command input is already from terminal

---

Examples:

DTICONTOUR> **@FLOW2<CR>**
DTICONTOUR> ENABLE DIAGNOSTICS
DTICONTOUR> RETURN
DTICONTOUR>

SET CONTOUR_FC

Set IFF feature code of ordinary contours to the specified value.

FORMAT: SET CONTOUR_FC feature-code

Command parameters:

feature-code

The feature code to be used for ordinary contours. This must lie in the range 0 to 32767.

DESCRIPTION:

SET CONTOUR_FC enables the user to specify the feature code to be used for ordinary contours. By default, a feature code of 0 is assumed for ordinary (or "intermediate") contours and index contours.

When choosing a feature code for contours, ensure that the feature code is in the FRT file which is to be used for plotting the contours and that the feature code has an appropriate graphical type.

Messages:

*** WARNING *** Feature codes must lie in the range 0 to 32767.

Examples:

DTICONTOUR> **SET CONTOUR_FC 209<CR>**
DTICONTOUR>

SET FRAME_FC

Set IFF feature code for the optional plot frame feature.

FORMAT: SET FRAME_FC feature-code

Command parameters:

feature-code

The feature code to be used for the frame. This must lie in the range 0 to 32767.

DESCRIPTION:

SET FRAME_FC enables the user to specify the feature code to be used for the optional plot frame. By default, a feature code of 0 is assumed for the frame.

When choosing a feature code for the frame, ensure that the feature code is in the FRT file which is to be used for plotting the data and that the feature code has an appropriate graphical type.

Messages:

*** WARNING *** Feature codes must lie in the range 0 to 32767.

Examples:

DTICONTOUR> **SET FRAME_FC 209<CR>**
DTICONTOUR>

SET INDEX_FC

Set IFF feature code of index contours to the specified value.

FORMAT: SET INDEX_FC feature-code

Command parameters:

feature-code

The feature code to be used for index contours. This must lie in the range 0 to 32767.

DESCRIPTION:

SET INDEX_FC enables the user to specify the feature code to be used for index contours. By default, a feature code of 0 is assumed for both ordinary (or "intermediate") contours and index contours.

When choosing a feature code for index contours, ensure that the feature code is in the FRT file which is to be used for plotting the contours and that the feature code has an appropriate graphical type.

Messages:

*** WARNING *** Feature codes must lie in the range 0 to 32767.

Examples:

DTICONTOUR> **SET INDEX_FC 209<CR>**
DTICONTOUR>

SET INDEX_LABEL_FC

Set IFF feature code of label features on index contours to the specified value.

FORMAT: SET INDEX_LABEL_FC feature-code

Command parameters:

feature-code

The feature code to be used for label features on index contours. This must lie in the range 0 to 32767.

DESCRIPTION:

SET INDEX_LABEL_FC enables the user to specify the feature code to be used for index contour label features. By default, a feature code of 0 is assumed for label features on index contours.

When choosing a feature code for index contour label features, ensure that the feature code is in the FRT file which is to be used for plotting the contours and that the feature code has an appropriate graphical type.

Messages:

*** WARNING *** Feature codes must lie in the range 0 to 32767.

Examples:

DTICONTOUR> **SET INDEX_LABEL_FC 209<CR>**
DTICONTOUR>

SET LABEL_FC

Set IFF feature code of label features to the specified value.

FORMAT: SET LABEL_FC feature-code

Command parameters:

feature-code

The feature code to be used for label features. This must lie in the range 0 to 32767.

DESCRIPTION:

SET LABEL_FC enables the user to specify the feature code to be used for ordinary (or "intermediate") contour label features. By default, a feature code of 0 is assumed for label features on ordinary contours.

When choosing a feature code for label features, ensure that the feature code is in the FRT file which is to be used for plotting the contours and that the feature code has an appropriate graphical type.

Messages:

*** WARNING *** Feature codes must lie in the range 0 to 32767.

Examples:

DTICONTOUR> **SET LABEL_FC 209<CR>**
DTICONTOUR>

SET LAYER

Set IFF output file layer number to the specified value.

FORMAT: SET LAYER layer-number

Command parameters:

layer-number

The number of the layer to contain the contour features. This must lie in the range 0 to 32767.

DESCRIPTION:

SET LAYER enables the user to specify the number of the layer which is to contain the contour features. By default, layer 1 is created.

Messages:

*** WARNING *** Layer numbers must lie in the range 0 to 32767.

Examples:

DTICONTOUR> **SET LAYER 1008<CR>**
DTICONTOUR>

SET SCALE

Sets the scale in the output IFF file MD (Map Descriptor) entry.

FORMAT: SET SCALE scale

Command parameters:

scale

The scale to be set in the output IFF file MD (Map Descriptor) entry. By default a scale value of 50000, for 1:50000 is used.

DESCRIPTION:

The scale field in the MD is by default set to 50000, as this information is unavailable in the DTI file header. The user may override this default scale by use of the SET SCALE command. The argument to the SET SCALE command should be specified as an integer value to represent the denominator of the scale fraction, thus a desired scale of 1:250000 must be specified as 250000.

Messages:

*** WARNING *** Illegal SCALE value 'number'

Examples:

DTICONTOUR> **SET SCALE 100000<CR>**
DTICONTOUR>

SET SPOT_FC

Set IFF feature code of spot height features to the specified value.

FORMAT: SET SPOT_FC feature-code

Command parameters:

feature-code

The feature code to be used for ordinary contours. This must lie in the range 0 to 32767.

DESCRIPTION:

SET SPOT_FC enables the user to specify the feature code to be used for spot heights. By default, a feature code of 1 is assumed for spot height features.

When choosing a feature code for spot heights, ensure that the feature code is in the FRT file which is to be used for plotting the spot heights and that the feature code has an appropriate graphical type, i.e. unorientated symbol.

Messages:

*** WARNING *** Feature codes must lie in the range 0 to 32767.

Examples:

```
DTICONTOUR> SET SPOT_FC 2<CR>
DTICONTOUR>
```

SET SPOT_LABEL_FC

Set IFF feature code of label features for spot heights to the specified value.

FORMAT: SET SPOT_LABEL_FC feature-code

Command parameters:

feature-code

The feature code to be used for spot height label features. This must lie in the range 0 to 32767.

DESCRIPTION:

SET SPOT_LABEL_FC enables the user to specify the feature code to be used for label features relating to spot heights. By default, a feature code of 0 is assumed for label features on spot heights.

When choosing a feature code for label features, ensure that the feature code is in the FRT file which is to be used for plotting the contours and that the feature code has an appropriate graphical type.

Messages:

*** WARNING *** Feature codes must lie in the range 0 to 32767.

Examples:

DTICONTOUR> **SET SPOT_LABEL_FC 209<CR>**
DTICONTOUR>

SHOW

Shows current status of DTICONTOUR defaults.

FORMAT: SHOW subject

Command parameters:

subject

The subject that is to be displayed, chosen from:

CONTOURS ENABLE FILES LABELLING SETTINGS UNITS

DESCRIPTION:

SHOW enables the user to examine the current status of the DTICONTOUR defaults, e.g. SHOW FILES will give the current status of the contouring window parameters and input and output files; SHOW SETTINGS will show the current status of the settings made for the IFF file with SET commands, etc.

Messages: None.

Examples:

```
$ DTICONTOUR<CR>
DTMCREATE module DTICONTOUR of 09:13:58 18-NOV-87
DTICONTOUR> SHOW ENABLE<CR>

ABSOLUTE ...................................................... On
DIAGNOSTICS ................................................... Off
DIVIDEBY ...................................................... Off
FRAME ......................................................... Off
INDEX ......................................................... On
INTEGER_HEIGHT ................................................ Off
(Contour heights will be placed in type 3 AC entries)
LABELLING ..................................................... Off
MULTIPLY .....................................................  Off
PME ........................................................... Off
SMOOTHING ..................................................... Off
SPOT_HEIGHTS .................................................. Off
(Single point contours to be omitted from output)
TOFEET ........................................................ Off
TOMETRES ...................................................... Off

DTICONTOUR> QUIT<CR>
 ELAPSED:    0 00:00:19.08  CPU: 0:00:04.95  BUFIO: 10  DIRIO: 111  FAULTS: 68
$
```

SPAWN

The SPAWN command enables you to create a subprocess while within DTICONTOUR.

FORMAT: SPAWN command-line

Command parameters:

command-line

Specifies a DCL command string to be executed as if typed in response to a '$' prompt. When the command completes, the subprocess terminates and control is returned to DTICONTOUR. The command string cannot exceed 80 characters.

DESCRIPTION:

The SPAWN command enables you to create a subprocess while within DTICONTOUR. When the subprocess terminates control is returned to DTICONTOUR.

Messages:

The following warning messages are specific to the SPAWN command:

```
*** WARNING *** SPAWN requires a valid DCL command line
 *** ERROR *** Unable to spawn command, returning to DTICONTOUR
```

Examples:

DTICONTOUR> **SPAWN DIRECTORY *.DTI;*<CR>**

```
Directory DUA3:[TVES.ACCEPTANCE_TESTS]

TEST1.DTI;1               8/8       18-Oct-1987 07:56  [LSL,TIM]
TEST2.DTI;2               7/8       18-Oct-1987 17:17  [LSL,TIM]
TEST2.DTI;1               7/8       18-Oct-1987 17:07  [LSL,TIM]

Total of 3 files, 22/24 blocks.

DTICONTOUR>
```

UNITS

Specifies the units of measurement that will be used when defining an area of interest in the input DTM using the WINDOW command.

The command also controls the units of measurement which will be used when displaying file header details and the output coordinate system of the output IFF file.

FORMAT: UNITS units

Command parameters:

units

A keyword defining the measurement units, chosen from:

MATRIX	Matrix grid interval units, i.e rows and columns
METRES	Metres on the ground
LATLONG	Latitude and Longitude (in degrees, minutes and seconds)
SECONDS	Seconds of arc
PROJECTION	Projection units

DESCRIPTION:

The UNITS command enables the user to specify in what units of measurement he wishes to define an area of interest in the input DTI file using the WINDOW command, or in what units of measurement details from the header of the DTI file are displayed using the SHOW FILES command.
The command should be given after defining the input DTI file, since an appropriate default units of measurement is set up whenever an input DTI file is opened, (see general Description).

The command may also be used to define the units of measurement in the output IFF file. If matrix units are currently selected, the coordinates in the IFF file are recorded as real (floating point) column and row values; if metres or projection units are selected before giving the GO command, the IFF data is recorded as metre or projection unit coordinate values.

The following rules are applied when testing the validity of a UNITS command argument supplied by the user:

- o MATRIX - valid for all DTI files

- o METRES - valid for UHL1 and TED4 files; LSLA files without a projection record, and LSLA files with a projection record whose units entry does not indicate unset, feet, sheet mms. or thousands of an inch.

- SECONDS - valid for UHL1 and TED4 files, and LSLA files with a projection record whose units indicate seconds, degrees, radians or .1 seconds of arc.
- LATLONG - valid for UHL1 and TED4 files, and LSLA files with a projection record whose units indicate seconds, degrees, radians or .1 seconds of arc.
- PROJECTION - valid for LSLA files with a projection record.

Messages:

The following error messages are specific to the UNITS command:

```
*** ERROR *** Specifying command UNITS
Command qualifiers are MATRIX,METRES,PROJECTION,SECONDS or LATLONG

*** ERROR *** Specifying command UNITS
Command qualifier is invalid for the input file

*** WARNING  *** Specifying command UNITS
                You must use the FILEIN command to specify the input
                DTI file first
```

Examples:

DTICONTOUR> **UNITS MATRIX<CR>**
DTICONTOUR>

WAIT

Suspend processing for the specified number of seconds.

FORMAT: WAIT seconds

Command parameters:

seconds

The number (floating point) of seconds for which DTICONTOUR processing is to be suspended.

DESCRIPTION:

The WAIT command causes processing to be suspended for a specified number of seconds. It is designed for use in software demonstration situations and is of no value in a production flowline.

Messages:

The following warning message is specific to the WAIT command:

*** WARNING *** You must specify the number of seconds to wait

---

Examples:

DTICONTOUR> **WAIT 4.0<CR>**
DTICONTOUR>

WINDOW

Specifies the limits of the data area to be contoured.

FORMAT: WINDOW xmin ymin xmax ymax

Command parameters:

xmin ymin
: The coordinates of the bottom left hand corner of the defining rectangle.

xmax ymax
: The coordinates of top right hand corner of the defining rectangle.

The format of the command parameters is controlled by the UNITS command.

UNITS MATRIX Requires 4 integer values defining the rectangle in terms of column and row numbers

UNITS METRES Requires 4 real (floating point) values defining the rectangle in metre values. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates (eg. U.K. National Grid coordinates); if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

UNITS SECONDS Requires 4 real (floating point) values defining the absolute position of the rectangle in seconds of arc. The values are supplied in the order latitude followed by longitude for the SW corner, and latitude followed by longitude for the NE corner. A negative latitude value indicates a position in the Southern Hemisphere, and a negative longitude value a point to the east of Greenwich.

UNITS LATLONG Requires 4 values defining the absolute latitude and longitude position of the rectangle in the format DDD MM SS H where DDD is the number of degrees, MM is the number of minutes, SS is the number of seconds, and H is the hemisphere (N,S,E,W). The values are supplied in the order latitude followed by longitude for the SW corner, and latitude followed by longitude for the NE corner.

UNITS PROJECTION Requires 4 real (floating point) values defining the rectangle in projection record units. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates; if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

Note that in all cases, the input values are adjusted to the nearest column and row values.

DESCRIPTION:

The command is used to define rectangular limits to the area of data to be included within the contouring. The limits must be specified in the order bottom left hand (or south west) corner then top right hand (or north east) corner.

The WINDOW command is used to limit the contouring process to a specified area of interest in the DTI file.

The area of interest should lie within the bounds of the DTI file.

If the logical name LSL$MATRIX_DEFAULT_WINDOW is defined with a value of "1", then a default area of interest defined in matrix units of bottom left hand corner 1,1 and top right hand corner 200,200 is set when the file is opened. If the logical name is absent or has any other value, or if the input DTI file has fewer than 200 columns or 200 rows, then a default window equivalent to the total matrix is set.

Messages:

The following warning messages are specific to the WINDOW command:

```
*** WARNING **** Unable to read WINDOW arguments
*** WARNING **** is still unset, please respecify the WINDOW command
*** ERROR **** window values define a zero width window
*** ERROR **** window values define a zero height window
```

Examples:

DTICONTOUR> **UNITS LATLONG<CR>**
DTICONTOUR> **WINDOW 52 10 00.0N 06 00 00.0E 52 30 00.0N 06 30 00.0E<CR>**
DTICONTOUR>

ZLIMITS

Specifies that contouring is only to be done between the specified minimum and maximum Z values. Heights that lie outside the specified limits will not be contoured.

FORMAT: ZLIMITS real1 real2

COMMAND PARAMETERS:

real1

The minimum Z value in the input data

real2

The maximum Z value in the input data

Description:

The ZLIMITS command enables the user to specify that contouring is only to be done between the stated minimum and maximum Z values. Heights that lie outside the specified limits will not be contoured.

IMPORTANT

If one of the height modification options is selected using ENABLE MULTIPLYBY, ENABLE TOFEET etc., you must give the ZLIMITS in the target measurement system or height range (ie feet for the TOFEET command). Failure to do this may result in a lot more (or less) contours than you expected!

Messages:

The following warning messages are specific to the ZLIMITS command:

```
*** WARNING *** You must specify minimum and maximum Z value arguments
                  For example ZLIMITS 80.0 3000.0
```

Examples:

```
DTICONTOUR> SHOW WINDOW<CR>
XMIN = 0.0 XMAX = 100.0 YMIN = 0.0 YMAX = 100.0
Z-limits currently undefined
DTICONTOUR> ZLIMITS 0 850<CR>
DTICONTOUR> SHOW WINDOW<CR>
Xmin = 0.0 Xmax = 100.0 Ymin = 0.0 Ymax = 100.0
Zmin= 0.000 Zmax= 850.000
DTICONTOUR>
```

EXAMPLES

```
$
$ DTICONTOUR<CR>
DTMCREATE module DTICONTOUR of 15:02:26 16-NOV-87
DTICONTOUR> FILEIN 52N09EDTEDUTM<CR>
DTI file LSL$DTI:52N09EDTEDUTM.DTI;0 opened for read

File    :  LSL$DTI:52N09EDTEDUTM.DTI;0
Header  :  LSLA  Data:  WORD

Units are metres

Matrix Coverage   SW:  500083.88 5761241.50    NE:  568683.88 5872941.50
Matrix Window     SW:  500083.88 5761241.50    NE:  568683.88 5872941.50
Matrix Interval    E:     100.00                N:     100.00
Value Range         :      4   to    1104


DTICONTOUR> IFF 5209WORD<CR>
%LSLLIB-I-IFFOPENED, LSL$SOURCEROOT:[TVES.DTICONTOUR]5209WORD.IFF;2 opened for
write
DTICONTOUR> WINDOW 500083.88 5761241.50 538683.88 5802941.50<CR>
DTICONTOUR> ENABLE SMOOTH 2<CR>
DTICONTOUR> ENABLE FRAME<CR>
DTICONTOUR> SET FRAMEFC 2<CR>
DTICONTOUR> SET FC 5<CR>
DTICONTOUR> SET INDEX_FC 3<CR>
DTICONTOUR> SET LABELFC 1<CR>
DTICONTOUR> SET INDEX_LABELFC 1<CR>
DTICONTOUR> ENABLE SPOT<CR>
DTICONTOUR> SET SPOTFC 4<CR>
DTICONTOUR> SET SPOTLABELFC 1<CR>
DTICONTOUR> INTERVAL 50<CR>
DTICONTOUR> INDEX_INTERVAL 250<CR>
DTICONTOUR> ENABLE LABELLING<CR>
DTICONTOUR> LABEL SIZE 120<CR>
DTICONTOUR> LABEL MINGAP 2000<CR>
DTICONTOUR> LABEL MAXCURVE 0.75<CR>
DTICONTOUR> LABEL MODULUS 0             ! don't label intermediate contours<CR>
DTICONTOUR> LABEL INDEX_MODULUS 250     ! but do label all index ones<CR>
DTICONTOUR> LABEL INTEGER<CR>
DTICONTOUR> ENABLE DIAGNOSTICS<CR>
DTICONTOUR> GO                  ! go!<CR>

=====================  STARTING CONTOUR GENERATION  =====================

Units are metres

Matrix Coverage   SW:  500083.88 5761241.50    NE:  568683.88 5872941.50
Contouring window SW:  500083.88 5761241.50    NE:  538683.88 5802941.50

All heights between     4.000 and   1104.000 will be contoured
```

```
              +------------------------+
              | Tracing index contours |
              +------------------------+
Tracing index contours with a height of   250.000
Tracing index contours with a height of   500.000
Tracing index contours with a height of   750.000
Tracing index contours with a height of  1000.000


            +-------------------------------+
            | Tracing intermediate contours |
            +-------------------------------+
Tracing intermediate contours with a height of    50.000
Tracing intermediate contours with a height of   100.000
Tracing intermediate contours with a height of   150.000
Tracing intermediate contours with a height of   200.000
Tracing intermediate contours with a height of   300.000
Tracing intermediate contours with a height of   350.000
Tracing intermediate contours with a height of   400.000
Tracing intermediate contours with a height of   450.000
Tracing intermediate contours with a height of   550.000
Tracing intermediate contours with a height of   600.000
Tracing intermediate contours with a height of   650.000
Tracing intermediate contours with a height of   700.000
Tracing intermediate contours with a height of   800.000
Tracing intermediate contours with a height of   850.000
Tracing intermediate contours with a height of   900.000
Tracing intermediate contours with a height of   950.000
Tracing intermediate contours with a height of  1050.000
Tracing intermediate contours with a height of  1100.000
 ELAPSED:    0 00:16:15.77  CPU: 0:10:05.76  BUFIO: 103  DIRIO: 4683  FAULTS:
36493
$
```

In this example DTICONTOUR has been used to generate contours from an LSLA format DTI file LSL$DTI:52N09EDTEDUTM.DTI;0. This file is of type word (integer) and the projection information indicate that the data are in metres.

The output IFF file is specified using the IFF command. The missing parts of the IFF file specification are taken from the translation of logical name LSL$IF. IFF file LSL$SOURCEROOT:[TVES.DTICONTOUR]5209WORD.IFF;2 is created and opened for write.

Post tracing contour smoothing is specified with the ENABLE SMOOTH command. Two iterations of the smoothing routine will be used.

The user has next specified that the data window shall be defined in the IFF file by a frame feature. This feature is given a feature code of 2 by the SET FRAME_FC command. This enables the frame to be differentiated from other data within the file.

Other IFF feature code assignments are then set up using the SET command. Feature code 5 is assigned for ordinary (or "intermediate") contours and feature code 3 is assigned for index contours. All labels generated for intermediate and index contours and spot heights are assigned feature code 1.

The ENABLE SPOT_HEIGHTS command ensures that any single points which lie at a contour height will be represented by a labelled spot height feature. These spot heights are given feature code 4 using the SET SPOT_FC command.

The intermediate contour interval is set to 50 using the INTERVAL command and the index contour interval to 250 using the INDEX_INTERVAL command.

LABEL generation is enabled with the ENABLE LABEL command. Labelling characteristics are set up via a series of LABEL command arguments. LABEL SIZE sets the label character height in IFF units (in this example metres!). LABEL MINGAP sets the minimum distance (in IFF units) permitted between successive labels along a contour line. The LABEL MAXCURVE argument of 0.75 (on a scale of 0.0 to 1.0) increases DTICONTOURs tolerance to bendy linework (default 0.7) when assessing the suitability of the site for a label. The LABEL MODULUS and LABEL INDEX_MODULUS commands determine the frequency of contour labelling. The label modulus command arguments specify a height. If a contour is exactly divisible by this height it will be labelled. By specifying a LABEL MODULUS argument of zero the user prevents labelling of any intermediate contours. The LABEL INDEX_MODULUS command argument of 250 means that only index contours which are divisible by 250 will be labelled.

The ENABLE DIAGNOSTICS command causes DTICONTOUR to output diagnostic information during processing. This text can be seen in this example after the GO command is used to set the contouring process in motion.

The run has completed successfully. DCL symbol $STATUS is set to SS$_NORMAL, normal successful completion.

DTICONTOUR messages

MESSAGES (ERROR)

These messages indicate an error in processing which will cause the program to terminate. The most likely causes are a corrupt or otherwise invalid input file, or an error related to command line processing and file manipulation.

CLDTI, error closing DTI file

Explanation: DTICONTOUR is unable to close the input DTI file.

User action: The supplementary messages supplied with this message will enable the user to determine the cause of the problem. If the problem persists notify your system manager.

CLIFF, error closing output IFF file

Explanation: DTICONTOUR is unable to close the output IFF file.

User action: The supplementary messages supplied with this message will enable the user to determine the cause of the problem, e.g. insufficient disk space for file allocation. Notify your system manager.

CLIND, error closing indirect command file

Explanation: DTICONTOUR is unable to close the indirect command file.

User action: The supplementary messages supplied with this message will enable the user to determine the cause of the problem. If the problem persists notify your system manager.

SPACELIM, insufficient workspace - max. available is 'integer' bytes

Explanation: DTICONTOUR dynamically allocates the virtual memory it requires to contour the DTI window specified. Unfortunately it has only been able to allocate the specified number of bytes and is unable to continue.

User action: The supplementary message supplied with this message will enable the user to determine the cause of the problem. It may be that you have attempted to exceed your process page file quota (PGFLQUOTA) or that your working set limit is not large enough to accommodate the increased virtual address space. The bigger the DTI window that you want to contour, the heavier the demand on virtual memory will be. As an alternative to increasing your page file quota, contour the DTI file using two or more smaller windows.

ZMAXERR, ZLIMITS upper bound less than DTI data minimum

Explanation: The requested z range in the ZLIMITS qualifier is not within the DTI data values.

User action: Resupply the values given with the ZLIMITS qualifier.

ZMINERR, ZLIMITS lower bound exceeds DTI data maximum

Explanation: The requested z range in the ZLIMITS qualifier is not within the DTI data values.

User action: Resupply the values given with the ZLIMITS qualifier.

MESSAGES (FATAL)

These messages indicate a severe error in processing, or some form of system failure, which has caused the program to terminate.

LOST, failed to locate IFF entry at recorded address - position lost

> **Explanation:** DTICONTOUR has incorrectly stored the address of an entry within the output IFF file and has now attempted to locate that IFF entry. This is a very severe error. DTICONTOUR is irrevocably lost.
>
> **User action:** Try reading the input IFF files into LITES2. If this is successful then the problem lies within DTICONTOUR itself; please make a copy of the input DTI file and the commands used and report the problem to Laser-Scan.

MESSAGES (OTHER)

In addition to the above messages which are generated by the program itself, other messages may be produced by the command line interpreter (CLI) and by Laser-Scan libraries. In particular, messages may be generated by the DTILIB library and by the Laser-Scan I/O library, LSLLIB. DTILIB library messages are introduced by '%DTILIB' and are documented in the MATRIX package reference manual. In most cases DTI errors will be due to a corrupt input file, and this should be the first area of investigation. If the cause of the error cannot be traced by the user, and Laser-Scan are consulted, then the output file should be preserved to facilitate diagnosis. LSLLIB messages are introduced by '%LSLLIB' and are generally self-explanatory. They are used to explain the details of program generated errors.

CHAPTER 5

MODULE I2GRID

MODULE I2GRID

Vector to Raster Conversion

Issue 3.0 - 9 March 1989

Science Park, Milton Road, Cambridge, England CB4 4FY tel: (0223) 420414

Document "I2GRID Reference"			Category "Reference"
Document Issue	3.0	D R Catlow	09-March-1989
Issue	2.0	D R Catlow	21-December-1988
Issue	1.2	D R Catlow	19-August-1988
Issue	1.1	D R Catlow	12-May-1988
Issue	1.0	D R Catlow	17-October-1986

MODULE I2GRID

FUNCTION

I2GRID converts data from Laser-Scan's Internal Feature Format (IFF) to Laser-Scan's Digital Terrain Image (DTI) format. In so doing the data is converted from a VECTOR to a GRID form ie. the data in the input IFF file is rasterised.

FORMAT

$ I2GRID

PROMPTS

I2GRID is an interactive, command driven program. Command input is expected when the following prompt is issued:

I2GRID>

Commands are issued at the terminal in response to the prompt.

DESCRIPTION

I2GRID is a program to convert data from a vector IFF format to a raster DTI format.

DATA CONVERSION

Data conversion involves transforming each IFF coordinate into the coordinate system of the output matrix, and then scan converting on a vector by vector basis.

The program will convert point, line and area information. A point feature is represented in the output matrix by a single encoded matrix point. A linear or area feature is represented by a series of contiguously encoded points. When converting an area feature both the boundary matrix points and the matrix points interior to the feature are encoded. Area features are identified to the program by means of the FRT graphical type entry, or through a list of Feature Codes specified using the AREA command.

```
                                                               B
                                                              B
                                   AAAAAAA                    B
                  C                AAAAAAAA               BBBBB
                                AAAAAAAAA                B
                                 AAAAAAAAAAAA             B
                      C           AAAAAA   AAA           B
                                  AAAAA   AA            B
                                  AAAA                  B
                  C                AAA                   B


              Point Features      Area feature         Line Feature
```

The file containing the input vector data is defined using the IFF command. The user may select which IFF features are converted by means of the SELECT and DESELECT commands. Restrictions may be imposed on the basis of Feature Serial Number, Feature Code or Layer. By default all features in the IFF file will be used.

The IFFWINDOW command may be used to restrict conversion to IFF features that lie within a rectangular geographical area.

OUTPUT DTI MATRIX

Output from I2GRID may be to an existing DTI file specified by means of the OPEN command, or to a new DTI file that will be created when the GO command is issued. The file specification of the new DTI file is given using the CREATE command.

In the former case, the header type, data type and grid interval values are read from the header of the file, and **cannot** be reset. When output is to a new DTI file however, it is possible to specify the DTI header type, data type and grid intervals using the HEADER_TYPE, DATA_TYPE and GRID commands. It is also possible to use the BACKGROUND command to specify the value of any matrix point that is not encoded during the conversion process.

The resolution of the gridded data is controlled by means of the GRID command. This sets both the x and y grid interval values. The representation of the IFF data is controlled largely by these values, and they should be chosen carefully and with consideration to the final application of the gridded data. The grid interval values in conjunction with the IFF range or IFFWINDOW values, determine the dimensions (ie. number of columns and rows) of the output matrix.

OUTPUT ATTRIBUTE SELECTION

The output matrix is encoded during the scan conversion operation with a value derived from an attribute of the IFF feature. The program allows the user to select what attribute information is stored in the DTI file by means of the ATTRIBUTE command. For many applications this information may be the Feature Serial Number (FSN) or Feature Code (FC), while for other applications (eg. for use with COVER) may be a value relating to the height or other characteristics of a feature. Only one attribute value per feature may be stored in the output DTI file. By default the matrix is encoded with the FSN value.

The attribute may be the FSN, FC or an attribute associated with an Ancillary Code AC entry. In the latter case the attribute may be the ancillary code value, or a value associated with a text string in the text field of the AC. For example, the command ATTRIBUTE AC CONTOUR will select the contour height for transfer, while the command ATTRIBUTE AC DFAD_FADT HGT will select a value associated with a DFAD feature text string HGT.

The attribute value may also be a height value derived from a Z coordinate of a 3 dimensional IFF feature. This option is selected using the command ATTRIBUTE Z. When a 3 dimensional linear feature is scan converted, the Z value at each coordinate point will be used, and Z values will be derived for intervening matrix points using linear interpolation. Note however that if the feature is identified as being an area feature, then a constant Z value derived from the first IFF coordinate is used.

As an alternative to transferring an IFF attribute value, the PRESET command may be used to assign particular values to IFF features on the basis of feature code or feature serial number. For example, the command PRESET FSN 1-100 = 100 will assign a value of 100 to FSNs 1 through 100.

OUTPUT DATA TYPE

The DATA_TYPE command controls the format in which the data will be output to a new matrix. By default a WORD data type is assumed. If output is to an existing matrix, the data type is derived from the header of this file and cannot be changed. As well as controlling the format, the data type also determines the valid range of the output value. The following data formats may be selected:

1. BIT - Data is stored as a single bit, and may take the value of 0 or 1

2. BYTE - Data is stored as an 8 bit byte. The byte value is treated as unsigned, and may be in the range 0 through 255

3. WORD - Data is stored as a 16 bit integer. A value may be in range -32768 to 32767

4. LONGWORD - Data is stored as a 32 bit integer. A value may be in the range -2147483648 through 2147483647

5. REAL - Data is stored as a single-precision 32 bit real. A value may be in the approximate range .29 times 10 raised to the power of -38 through 1.7 times 10 to the power of 38.

The ENABLE TRUNCATE command controls whether attribute values that are too large or too small to be stored in the output file are truncated prior to output. For example, if output is to a DTI file holding byte data, an attribute value of 340 will be truncated to 255, and an attribute value of -10 to 0 before encoding. By default values are truncated, and the option may be disabled using the DISABLE TRUNCATE command. In the latter case, no value will be encoded in the output file if the value is either too large or too small, and a warning will be output to the terminal.

OUTPUT PRIORITY

During the vector to grid conversion, situations will arise where a number of IFF coordinates reference the same point in the DTI file. This situation will occur particularly if a coarse grid resolution is selected. The action taken in this event, may be controlled by the user with the PRIORITY command. The various PRIORITY command options are:

1. CURRENT - The current value will replace an existing value. The final matrix value depends on the way features are ordered in the IFF file.

2. FIRST - The current value will NOT replace an existing value. The final matrix value depends on the way features are ordered in the IFF file.

3. LARGEST - The current value will replace an existing value if it is larger.

4. SMALLEST - The current value will replace an existing value if it is smaller.

By default PRIORITY CURRENT is applied.

```
      C    A
       C   A
       C   A
        C  A
         C A
BBBBBBBBBBCBBBBBBBBBB
          C
         A C
        A  C
       A    C
      A     C
     A       C
    A         C
```

Figure 1 PRIORITY CURRENT

```
      C    A
       C   A
       C   A
        C  A
         C A
BBBBBBBBBBABBBBBBBBB
          A
         A C
        A  C
       A    C
      A     C
     A       C
    A         C
```

Figure 2 PRIORITY FIRST

```
      C    A
       C   A
       C   A
        C  A
         C A
BBBBBBBBBBCBBBBBBBBBB
          C
         A C
        A  C
       A    C
      A     C
     A       C
    A         C
```

Figure 3 PRIORITY LARGEST

```
      C    A
       C   A
       C   A
        C  A
         C A
BBBBBBBBBBABBBBBBBBB
          A
         A C
        A  C
       A    C
      A     C
     A       C
    A         C
```

Figure 4 PRIORITY SMALLEST

A = 1 = FSN 1   B = 2 = FSN 2   C = 3 = FSN 3

MATRIX BACKGROUND

A background matrix point is defined as a point in the matrix which is not referenced by any transformed IFF coordinate. By default when output is to a new DTI matrix, background points are assigned a value of 0. This default may be overridden using the BACKGROUND command. For example, the command BACKGROUND 10 will assign a value of 10 to all points not encoded during the vector to raster conversion process. This is achieved by initialising all matrix points to 10 immediately following the creation of the output matrix. The background value must be specified so that it is suitable for the output data type. The BACKGROUND command is not applicable if output is to an existing DTI matrix.

USE WITH DFAD DATA

A special option is provided in I2GRID for DFAD data. This option is enabled using the command ENABLE DFAD. If the command is given then the following rules are applied by default when converting the IFF data:

1. The value associated with the AC DFAD_FADT text string HGT is selected for transfer to the output DTI file. This is equivalent to giving the command ATTRIBUTE AC DFAD_FADT HGT. The ATTRIBUTE command may be used to override this default.

2. IFF features which are to be treated as areas are identified by means of the AC DFAD_FADT text string TYP. A 'TYP' value of 2 indicates an area feature. The AREA FRT or AREA FC commands may be used to override this default.

3. DFAD point features are expanded to their correct planimetric representation using the AC DFAD_FADT text strings WID, LTH and ORI. Depending on these attribute values, and the output DTI file grid intervals, a DFAD point feature may be represented by more than a single encoded pixel, and may take the form of a square, rectangle or circle.

4. DFAD linear features are expanded to their correct planimetric representation using the AC DFAD_FADT text string WID. Depending on the 'WID' value, and the output DTI file grid intervals, a DFAD linear feature may be represented by more than a single line of encoded pixels.

5. During the conversion process, priority is given to the largest attribute value, therefore if the default DFAD attribute selection of HGT is used, a matrix point is encoded with the highest height value. This is equivalent to giving the command PRIORITY LARGEST. The PRIORITY command may be used to override this default.

I2GRID COMMANDS

COMMAND LIST

The following commands are defined :

```
@               !               AREA FC         AREA FRT
ATTRIBUTE AC    ATTRIBUTE FC    ATTRIBUTE FSN   ATTRIBUTE Z
BACKGROUND      CREATE          DATA_TYPE       DESELECT FC
DESELECT FSN    DESELECT LAYER  DISABLE ABSOLUTE
DISABLE DFAD    DISABLE LOG     DISABLE PRESET  DISABLE THICK
DISABLE TRUNCATE                ENABLE ABSOLUTE ENABLE DFAD
ENABLE LOG      ENABLE PRESET   ENABLE THICK    ENABLE TRUNCATE
EXIT            FRT             GO              GRID
HEADER_TYPE     HELP            IFF             IFFWINDOW
OPEN            PRESET FC       PRESET FSN      PRIORITY CURRENT
PRIORITY FIRST  PRIORITY LARGEST                PRIORITY SMALLEST
RETURN          SELECT ALL      SELECT FC       SELECT FSN
SELECT LAYER    SHOW ENABLE     SHOW IFF        SHOW PARAMETERS
SHOW PRESET     SHOW SELECTIONS SPAWN           WAIT
WIDTH
```

@

Take command input from the specified file.

FORMAT: @ file-spec

Command parameters:

file-spec

The file to be opened and used for command input.

Any parts of the file-spec not supplied will be taken from the default specification 'SYS$DISK:[].COM;0'.

DESCRIPTION:

I2GRID offers the facility of command input from an indirect command file. The '@' character preceding a file-spec will cause I2GRID to open and read commands from the specified file until:

1. a RETURN command is detected and command input is returned to SYS$COMMAND.

2. the end of file is detected. This provokes an error message and command input is returned to SYS$COMMAND.

Nested command files are not supported (i.e. a command file containing an '@' command), although sequential '@' commands are supported when read from SYS$COMMAND.

As an aid to batch log interpretation I2GRID will echo all commands read from an indirect command file.

Messages:

The following messages are specific to the @ command:

*** ERROR *** Specifying command @
Command file specification is missing

*** ERROR *** Specifying command @
Unable to open indirect command file 'file-spec'

*** ERROR *** Specifying command @
Nested command files not supported

Examples:

```
I2GRID> @I2GRID_EDITS<CR>
I2GRID> ENABLE LOG
I2GRID> ENABLE TRUNCATE
I2GRID> PRESET FC 1-50 = 100
I2GRID> RETURN
I2GRID>
```

!

Treat all text to the right of the '!' as a comment.

FORMAT: ! [comment text]

Command parameters:

comment text

text that is to be treated as a comment and which will be excluded from command interpretation.

DESCRIPTION:

An exclamation mark is the standard TVES package comment delimiter. All text (and numbers) which lie to the right of a '!' character are excluded from command interpretation. Comments are useful for annotating command procedures used in batch processing etc.

Messages: None.

Examples:

I2GRID> **ENABLE TRUNCATE !truncate attribute values<CR>**
I2GRID> **!enable the log option<CR>**
I2GRID> **ENABLE LOG<CR>**
I2GRID>

AREA FC

Identifies features in the IFF file that are to be treated as area features, on the basis of feature code.

FORMAT: **AREA FC feature_code [,...]**

Command parameters:

feature_code

An integer feature code in the range 0 to 32767. Multiple feature codes may be specified separated by commas or spaces, while a range of feature codes may be specified by means of a '-'. eg. AREA FC 10-13 defines feature codes 10,11,12 and 13 as area features.

DESCRIPTION:

The AREA FC command is used to identify features in the IFF file that are to be treated as area features by I2GRID. Using this command area features are defined on the basis of their feature code.

When scan converting area features, both the boundary cells and the cells interior to the feature are coded with the selected attribute value.

Messages:

The following error messages are specific to the AREA and AREA FC commands:

*** ERROR *** Specifying command AREA
Command qualifiers are FC or FRT

*** ERROR *** Specifying command AREA
Command requires a list of integer feature code values

*** ERROR *** Specifying command AREA
Illegal feature code value

*** ERROR *** Specifying command AREA
Second part of feature code range is missing

Examples:

I2GRID>**AREA FC 10,20-30 40 <CR>**
I2GRID>

AREA FRT

Identifies features in the IFF file that are to be treated as area features, on the basis of FRT graphical type.

FORMAT: **AREA FRT**

Command parameters: None

DESCRIPTION:

The AREA FRT command is used to identify features in the IFF file that are to be treated as area features by I2GRID. Using this command, area features are identified on the basis of the graphical type entry in a FRT file. Graphical Type 12 indicates an area feature.

When scan converting area features, both the boundary cells and the cells interior to the feature are coded with the selected attribute value.

If this option is selected, a FRT must have been previously defined using the FRT command. It should be noted though that on giving the FRT command, AREA FRT is assumed by default.

Messages:

The following error messages are specific to the AREA and AREA FRT commands:

*** ERROR *** Specifying command AREA
Command qualifiers are FC or FRT

*** ERROR *** Specifying command AREA
No FRT file has been specified

Examples:

I2GRID>**AREA FRT <CR>**
I2GRID>

ATTRIBUTE AC

Selects the option of encoding the output DTI matrix with the ancillary code value of an IFF feature.

FORMAT: **ATTRIBUTE AC ac_type [text]**

Command parameters:

ac_type

An integer AC type value in the range 0 to 32767, or a valid AC name.

text

An optional AC text field string. If this string is present then a value associated with the text field string is encoded in the output matrix.

DESCRIPTION:

The ATTRIBUTE command is used to define which IFF feature attribute is transferred to the output DTI matrix when the GO command is given. Only a single attribute value may be output per feature.

The command ATTRIBUTE AC specifies that the attribute is the value associated with an AC of the specified type. Eg. the command ATTRIBUTE AC 2, or alternatively ATTRIBUTE AC CONTOUR, will transfer the height value associated with a contour AC. The AC may be an AC predefined by LSL, or may be an AC defined by the user in an ACD table in an FRT file. In the latter case, the command FRT must have been previously given to read in an FRT.

The ACD data type determines the format of the AC value. I2GRID will transfer only integer or real AC data values.

If the optional text string is given, then a value associated with that text string in the text field of the AC, is transferred to the output matrix. Eg. the command ATTRIBUTE AC DFAD_FADT HGT will select the output of a value from the AC text field, associated with the text string HGT.

Messages:

The following error messages are specific to the ATTRIBUTE and ATTRIBUTE AC commands:

```
*** ERROR *** Specifying command ATTRIBUTE
Valid qualifiers are AC, FC, FSN or Z

*** ERROR *** Specifying command ATTRIBUTE AC
```

Command requires an integer AC type number or AC name

*** ERROR *** Specifying command ATTRIBUTE AC
AC <ac_name> not found in ACD table

*** ERROR *** Specifying command ATTRIBUTE AC
AC <ac_type> not found in ACD table

*** ERROR *** Specifying command ATTRIBUTE AC
Unable to use ACD <ac_name> <ac_type>
Associated value is not integer or real

*** ERROR *** Specifying command ATTRIBUTE AC
Unable to read the associated text string

Output of AC <ac_name> <ac_type> value selected

Output of AC <ac_name> <ac_type> <ac_text_string> value selected

---

Examples:

I2GRID>**ATTRIBUTE AC CONTOUR<CR>**
Output of AC Contour(2) value selected
I2GRID>**ATTRIBUTE AC DFAD_FADT HGT<CR>**
Output of AC DFAD_FADT(7) HGT value selected

ATTRIBUTE FC

Selects the option of encoding the output DTI matrix with the feature code value of an IFF feature.

FORMAT: **ATTRIBUTE FC**

Command parameters: None

DESCRIPTION:

The ATTRIBUTE command is used to define which IFF feature attribute is transferred to the output DTI matrix when the GO command is given. Only a single attribute value may be output per feature.

The command ATTRIBUTE FC specifies that the attribute is the feature code. Eg. if the FC is 10, then a value of 10 will be recorded in the output DTI file.

Messages:

The following messages are specific to the ATTRIBUTE command:

```
*** ERROR *** Specifying command ATTRIBUTE
Valid qualifiers are AC, FC, FSN or Z

Output of FC value selected
```

Examples:

```
I2GRID>ATTRIBUTE FC<CR>
Output of FC value selected
I2GRID>
```

ATTRIBUTE FSN

Selects the option of encoding the output DTI matrix with the feature serial number of an IFF feature.

FORMAT: **ATTRIBUTE FSN**

Command parameters: None

DESCRIPTION:

The ATTRIBUTE command is used to define which IFF feature attribute is transferred to the output DTI matrix when the GO command is given. Only a single attribute value may be output for a feature.

The command ATTRIBUTE FSN specifies that the attribute is the feature serial number. Eg. if the FSN is 10, then a value of 10 will be recorded in the output DTI file.

Messages:

The following messages are specific to the ATTRIBUTE command:

```
*** ERROR *** Specifying command ATTRIBUTE
Valid qualifiers are AC, FC, FSN or Z
```

```
Output of FSN value selected
```

Examples:

```
I2GRID>ATTRIBUTE FSN<CR>
Output of FSN value selected
I2GRID>
```

ATTRIBUTE Z

Selects the option of encoding the output DTI matrix with a height value derived from the Z coordinates of a 3-D IFF feature.

FORMAT: ATTRIBUTE Z

Command parameters: None

DESCRIPTION:

The ATTRIBUTE command is used to define which IFF feature attribute is transferred to the output DTI matrix when the GO command is given. Only a single attribute value may be output per feature.

The command ATTRIBUTE Z specifies that the attribute is a height value derived from the Z coordinates of an IFF feature. For an area feature, a height associated with the first coordinate point is transferred to the output matrix. For a linear feature, the height value associated with each coordinate point is used, and the heights of any intermediary matrix points are derived using linear interpolation.

Messages:

The following messages are specific to the ATTRIBUTE command:

*** ERROR *** Specifying command ATTRIBUTE
Valid qualifiers are AC, FC, FSN or Z

Output of Z coordinate values selected

Examples:

I2GRID>**ATTRIBUTE Z<CR>**
Output of Z coordinate values selected
I2GRID>

BACKGROUND

Assigns a value to the background matrix points in the output DTI file.

FORMAT: BACKGROUND value

Command parameters:

value

The background data value. Depending on the data type of the output DTI file, an integer or real data value is required. The value should be selected such that it falls within the valid range for the data type.

DESCRIPTION:

The BACKGROUND command allows a value to be assigned to the background matrix points in the output DTI file. A background matrix point is a point that is not encoded during the vector to raster conversion process. By default the matrix background is given a value of 0.

A background value may only be assigned if conversion is to a new output DTI file specified using the CREATE command. The background value of an existing DTI file cannot be changed.

The background value defined using the BACKGROUND command is validated against the output DTI file data type when the GO command is given. An error message is generated if the background value is too large or too small.

Messages:

The following error message is specific to the BACKGROUND command:

```
*** ERROR *** Specifying command BACKGROUND
Command requires an integer or real argument
```

Example:

```
I2GRID>BACKGROUND 23<CR>
I2GRID>
```

CREATE

Creates a new matrix to receive the gridded attribute information when the GO command is given (compare OPEN command).

FORMAT: **CREATE file-spec**

Command parameters:

file-spec

The file specification for the output DTI file.
Any part of the file specification not supplied is taken from the default 'LSL$DTI:DTI.DTI', although if no file-spec is supplied an error message will be generated.

DESCRIPTION:

The CREATE command specifies the name of a DTI file that will be created when the GO command is given, and into which the gridded attribute data will be encoded.

The grid interval values for this file are specified using the GRID command, while the DATA_TYPE and HEADER_TYPE commands may be used to specify the format of the data values and DTI header. The dimensions of the output DTI file (ie. the number of columns and rows), are calculated using the IFF range or IFFWINDOW values, and the matrix grid interval values.

If data is to be encoded into an existing matrix, the OPEN command should be used.

Messages:

The following error message is specific to the CREATE command:

```
*** ERROR *** Specifying command CREATE
DTI filename is missing
```

Example:

I2GRID>**CREATE FORESTS<CR>**
I2GRID>

DATA_TYPE

Specifies in what format the encoded data will be held in the output DTI file.

FORMAT: DATA_TYPE format

Command parameters:

format

A keyword defining the data type, chosen from:

BIT	- Data will be stored as a single bit, and may take the value of 0 or 1
BYTE	- Data will be stored as an 8 bit byte. The byte value is treated as unsigned, and may have a value in the range 0 through 255
WORD	- Data will be stored as a 16 bit integer. A value may be in the range -32768 to 32767
LONGWORD	- Data will be stored as a 32 bit integer. A value may be in the range -2147483648 through 2147483647
REAL	- Data will be stored as a single-precision 32 bit real. A value may be in the approximate range .29 times 10 raised to the power of -38 through 1.7 times 10 to the power of 38

DESCRIPTION:

The DATA_TYPE command may be used to control the format in which the encoded attribute data is held in the output DTI file. The command is only applicable if output is to a new DTI file specified using the CREATE command. If output is to an existing file using the OPEN command, the data type is derived from the header of this file, and cannot be redefined.

By default the attribute values are output as a word in the range -32768 to 32767.

The ENABLE TRUNCATE command controls whether an attribute value that is too small or too large to be stored in the data type, is truncated to fit, or is rejected during the vector to raster conversion process.

Messages:

The following error message is specific to the DATA_TYPE command:

*** ERROR *** Specifying command DATA_TYPE
Valid qualifiers are BIT, BYTE, LONGWORD, WORD or REAL

Example:

```
I2GRID>DATA_TYPE BYTE<CR>
I2GRID>
```

DESELECT FC

Excludes an IFF feature for conversion on the basis of feature code.

FORMAT: DESELECT FC feature_code [,...]

Command parameters:

feature_code

An integer feature code in the range 0 to 32767. Multiple feature codes may be specified separated by commas or spaces, while a range of feature codes may be specified by means of a '-'. eg. DESELECT FC 10-13 excludes feature codes 10,11,12 and 13.
Alternatively a valid FRT group name may be specified. eg. DESELECT RIVERS

DESCRIPTION:

The DESELECT FC command is used to exclude IFF features from conversion. Features are excluded on the basis of their feature code.

By default I2GRID will convert all features within an IFF file.

Use of the SHOW SELECTIONS command is recommended to display feature selections before the GO command is given.

Deselection on the basis of group name is valid only if the FRT command has been used to previously read a Feature Representation Table.

Messages:

The following error messages are specific to the DESELECT and DESELECT FC commands:

*** ERROR *** Specifying command DESELECT
Command qualifiers are FC, FSN, or LAYER

*** ERROR *** Specifying command DESELECT
Illegal feature code

*** ERROR *** Specifying command DESELECT FC
No FRT has been read

*** ERROR *** Specifying command DESELECT FC
No groups have been defined in the FRT

Examples:

```
I2GRID>DESELECT FC 1<CR>
I2GRID>DESELECT FC RAILWAYS<CR>
I2GRID>DESELECT FC RIVERS 7-10,56-78
I2GRID>
```

DESELECT FSN

Excludes an IFF feature from conversion on the basis of feature serial number.

FORMAT: DESELECT FSN fsn [,...]

Command parameters:

fsn

An integer feature serial number in the range 0 to 65534. Multiple feature serial numbers may be specified separated by commas or spaces, while a range of numbers may be specified by means of a '-'. eg. DESELECT FSN 10-13 excludes feature serial numbers 10,11,12 and 13.

DESCRIPTION:

The DESELECT FSN command is used to exclude IFF features from conversion. Features are excluded on the basis of their feature serial number.

By default I2GRID will convert all features within an IFF file.

Use of the SHOW SELECTIONS command is recommended to display feature selections before the GO command is given.

Messages:

The following error messages are specific to the DESELECT and DESELECT FSN commands:

*** ERROR *** Specifying command DESELECT
Command qualifiers are FC, FSN, or LAYER

*** ERROR *** Specifying command DESELECT FSN
Illegal FSN number

Examples:

I2GRID>**DESELECT FSN 4**
I2GRID>**DESELECT FSN 7-10,56-78**
I2GRID>

DESELECT LAYER

Excludes an IFF feature from conversion on the basis of layer number.

FORMAT: **DESELECT LAYER layer [,...]**

Command parameters:

layer

An integer layer number in the range 1 to 32767. Multiple layer numbers may be specified separated by commas or spaces, while a range of layer numbers may be specified by means of a '-'. eg. DESELECT LAYER 10-13 excludes all features in layers 10,11,12 and 13.

DESCRIPTION:

The DESELECT LAYER command is used to exclude IFF features from conversion. Features are excluded on the basis of IFF layer.

By default I2GRID will convert all features within an IFF file.

Use of the SHOW SELECTIONS command is recommended to display feature selections before the GO command is given.

Messages:

The following error messages are specific to the DESELECT and DESELECT LAYER commands:

```
*** ERROR *** Specifying command DESELECT
Command qualifiers are FC, FSN, or LAYER
```

```
*** ERROR *** Specifying command DESELECT
Illegal layer number
```

Examples:

I2GRID>**DESELECT LAYER 7**
I2GRID>**DESELECT LAYER 1,7-10**
I2GRID>

DISABLE ABSOLUTE

Cancels a previous ENABLE ABSOLUTE command, and disables the use of absolute coordinates values with the IFFWINDOW command.

FORMAT: DISABLE ABSOLUTE

Command parameters: None

DESCRIPTION:

The DISABLE ABSOLUTE cancels a previous ENABLE ABSOLUTE command. If DISABLE ABSOLUTE is specified, then IFF coordinates supplied with the IFFWINDOW command are interpreted as values relative to the IFF local origin.

The command also controls how IFF coordinates are displayed when the SHOW IFF command is given.

By default the use of absolute coordinate values is enabled.

Messages:

The following error message is specific to the DISABLE command:

*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, DFAD, LOG, PRESET, THICK or TRUNCATE

Examples:

I2GRID>**DISABLE ABSOLUTE<CR>**
I2GRID>

DISABLE DFAD

Cancels a previous ENABLE DFAD command, and deselects the options that are appropriate to the conversion of DLMS DFAD data.

FORMAT: DISABLE DFAD

Command parameters: None

DESCRIPTION:

The DISABLE DFAD command cancels a previous ENABLE DFAD command, and deselects the options that are appropriate to the conversion of DLMS DFAD data. If the command is given, then the options and rules that are applied when converting IFF features, are set to program startup defaults.

The defaults are:

1. The IFF feature attribute transferred to the output DTI file is the Feature Serial Number. This is equivalent to giving the command ATTRIBUTE FSN.

2. Area IFF features are identified by means of a user specified feature code list, unless the FRT command has been previously given, in which case area features will be identified using the FRT Graphical Type value. If the IFF file contains DFAD features, the DFAD_FADT text string attribute TYP is not used.

3. DFAD point and linear features are not expanded to their correct planimetric representation prior to conversion.

4. During the conversion process, priority is given to the attribute associated with the IFF feature currently being converted. This is equivalent to giving the command PRIORITY CURRENT.

Use of the SHOW PARAMETERS command is recommended to check on the status of these various options before the GO command is given.

Messages:

The following error message is specific to the DISABLE command:

*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, DFAD, LOG, PRESET, THICK or TRUNCATE

Examples:

```
I2GRID>DISABLE DFAD<CR>
I2GRID>
```

DISABLE LOG

Disables the output of informational and warning messages to the terminal as each IFF feature is converted.

FORMAT: DISABLE LOG

Command parameters: None

DESCRIPTION:

DISABLE LOG disables the output of informational and warning messages to the terminal as each IFF feature is converted.

By default the LOG messages are enabled.

Messages:

The following error message is specific to the DISABLE command:

*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, DFAD, LOG, PRESET, THICK or TRUNCATE

Examples:

I2GRID>**DISABLE LOG<CR>**
I2GRID>

DISABLE PRESET

Disables the use of values pre-assigned to features by means of the PRESET command.

FORMAT: DISABLE PRESET

Command parameters: None

DESCRIPTION:

The DISABLE PRESET command disables the use of the values pre-assigned to features by means of the PRESET command. When the GO command is given to initiate the vector to raster conversion all PRESET values will be ignored, and the output matrix will be encoded with a value determined by means of the ATTRIBUTE command.

By default any pre-assigned values defined using the PRESET command will be used.

Messages:

The following error message is specific to the DISABLE command:

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, DFAD, LOG, PRESET, THICK or TRUNCATE
```

Examples:

I2GRID>**DISABLE PRESET<CR>**
I2GRID>

DISABLE THICK

Cancels a previous ENABLE THICK command, and deselects the option of representing a linear feature as a thickened line.

FORMAT: **DISABLE THICK**

Command parameters: None

DESCRIPTION:

The DISABLE THICK command controls whether a thickness value is applied to a linear feature before conversion.

If DISABLE THICK is specified then all linear features are represented by a line of minimum thickness (single pixel wide) in the output matrix.

By default THICK is disabled.

Messages:

The following error messages is specific to the DISABLE command:

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, DFAD, LOG, PRESET, THICK or TRUNCATE

Examples:

I2GRID>**DISABLE THICK<CR>**
I2GRID>

DISABLE TRUNCATE

Deselects the option of truncating attribute values if they are either too small or too large to be stored in the output DTI file.

FORMAT: DISABLE TRUNCATE

Command parameters: None

DESCRIPTION:

The DISABLE TRUNCATE command deselects the option of truncating attribute values if they are either too small or too large to be stored in the output DTI file. Any IFF feature whose attribute value is too small or too large will be rejected, and will not be scan converted.

If the ENABLE LOG command has been given a warning message will be output.

By default values will be truncated.

Messages:

The following error message is specific to the DISABLE command:

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, DFAD, LOG, PRESET, THICK or TRUNCATE
```

Examples:

```
I2GRID>DISABLE TRUNCATE<CR>
I2GRID>
```

ENABLE ABSOLUTE

Enables the use of absolute coordinates values with the IFFWINDOW command.

FORMAT: **ENABLE ABSOLUTE**

Command parameters: None

DESCRIPTION:

The ENABLE ABSOLUTE command selects the use of absolute IFF coordinate values. If ENABLE ABSOLUTE is specified, then values supplied with the IFFWINDOW command are interpreted as absolute IFF coordinates.

The command also controls how IFF coordinates are displayed when the SHOW IFF command is given.

By default the use of absolute coordinate values is enabled.

Messages:

The following error message is specific to the ENABLE command:

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, DFAD, LOG, PRESET, THICK or TRUNCATE

Examples:

I2GRID>**DISABLE ABSOLUTE<CR>**
I2GRID>

ENABLE DFAD

Selects various options that are appropriate to the conversion of DLMS DFAD data.

FORMAT: **ENABLE DFAD**

Command parameters: None

DESCRIPTION:

The ENABLE DFAD command selects various options that are appropriate to the conversion of DLMS DFAD data. If the command is given then the following rules are applied by default, when converting the IFF data:

1. The value associated with the AC DFAD_FADT text string HGT is selected for transfer to the output DTI file. This is equivalent to giving the command ATTRIBUTE AC DFAD_FADT HGT. The ATTRIBUTE command may be used to override this default.

2. Area IFF features are identified by means of the AC DFAD_FADT text string TYP. A 'TYP' value of 2 indicates an area feature. The AREA FRT or AREA FC commands may be used to override this default.

3. DFAD point features are expanded to their correct planimetric representation using the AC DFAD_FADT text strings WID, LTH and ORI. Depending on these values, and the output DTI file grid intervals, a DFAD point feature may be represented by more than a single encoded pixel, and may take the form of a square, rectangle or circle.

4. DFAD linear features are expanded to their correct planimetric representation using the AC DFAD_FADT text string WID. Depending on the 'WID' value, and the output DTI file grid intervals, a DFAD linear feature may be represented by more than a single line of encoded pixels.

5. During the conversion process, priority is given to the largest attribute value. This is equivalent to giving the command PRIORITY LARGEST. The PRIORITY command may be used to override this default.

Use of the SHOW PARAMETERS command is recommended to check on the status of these various options before the GO command is given.

Messages:

The following error message is specific to the ENABLE command:

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, DFAD, LOG, PRESET, THICK or TRUNCATE

Examples:

```
I2GRID>ENABLE DFAD<CR>
I2GRID>
```

ENABLE LOG

Enables the output of informational and warning messages to the terminal as each IFF feature is converted.

FORMAT: ENABLE LOG

Command parameters: None

DESCRIPTION:

ENABLE LOG enables the output of informational and warning messages to the terminal as each IFF feature is converted. These messages indicate whether a feature has been successfully converted, and the value of its associated attribute.

By default the output of LOG messages is enabled.

Messages:

The following error message is specific to the ENABLE command:

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, DFAD, LOG, PRESET, THICK or TRUNCATE

Examples:

I2GRID>**ENABLE LOG<CR>**
I2GRID>

ENABLE PRESET

Enables the use of values pre-assigned to features by means of the PRESET command.

FORMAT: **ENABLE PRESET**

Command parameters: None

DESCRIPTION:

The ENABLE PRESET command enables the use of the values pre-assigned to features by means of the PRESET command. By default, or if this command is given, any value preassigned to a feature on the basis of feature code of FSN, is used rather than the attribute value.

Messages:

The following error message is specific to the ENABLE command:

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, DFAD, LOG, PRESET, THICK or TRUNCATE

Examples:

I2GRID>**ENABLE PRESET<CR>**
I2GRID>

ENABLE THICK

Selects the option of representing a linear features as a thickened line.

FORMAT: **ENABLE THICK**

Command parameters: None

DESCRIPTION:

The ENABLE THICK command controls whether a thickness value is applied to a linear feature before conversion.

If ENABLE THICK is specified then all linear features are thickened by an amount specified using the WIDTH command. The width is specified in IFF units. The actual representation of a thickened line in the output matrix depends upon the size of this value, and on the output matrix grid interval.

By default the option to thicken lines is disabled.

Messages:

The following error messages are specific to the ENABLE command:

```
*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, DFAD, LOG, PRESET, THICK or TRUNCATE
```

Examples:

I2GRID>**ENABLE THICK<CR>**
I2GRID>

ENABLE TRUNCATE

Selects the option of truncating attribute values so that they may be stored in the output DTI file, if they are either too small or too large.

FORMAT: **ENABLE TRUNCATE**

Command parameters: None

DESCRIPTION:

The ENABLE TRUNCATE command controls whether attribute values are truncated so that they may be stored in the output DTI file, if they are either too small or too large.

If ENABLE TRUNCATE is specified the attribute values will be truncated. For example, if output is to a DTI file holding byte data, an attribute value of 340 will be truncated to 255, and an attribute value of -10 to 0, before encoding.

When outputting to a BIT, BYTE, WORD or LONGWORD data file, a real attribute value is converted to the nearest integer, before truncating.

By default values will be truncated. The option may be disabled using the DISABLE TRUNCATE command.

Messages:

The following error messages are specific to the ENABLE command:

```
*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, DFAD, LOG, PRESET, THICK or TRUNCATE
```

Examples:

I2GRID>**ENABLE TRUNCATE<CR>**
I2GRID>

EXIT

Terminates the program.

FORMAT: **EXIT**

Command parameters: None.

DESCRIPTION:

The EXIT command is used to exit from I2GRID.
<CTRL/Z> (pressing the Ctrl and Z keys together) may also be used to exit from the program.

Messages: None.

Examples:

```
I2GRID>EXIT<CR>
$
```

FRT

Reads a FRT (Feature Representation Table) file.

FORMAT: **FRT file-spec**

Command parameters:

file-spec

The file specification for the FRT file. Any part of the file specification not supplied will be taken from the default 'LSL$FRT:FRT.FRT', although if no file-spec is supplied an error message will be generated.

DESCRIPTION:

This command reads the specified Feature Representation Table. The FRT is used to determine which IFF features are to be treated as areas during the conversion process (a graphical type of 12 indicates an area feature), and to define group names and user specified ACDs.

The commands SELECT FC 'group', AREA FRT and ATTRIBUTE AC 'user ACD' require a FRT to have been previously selected.

If the FRT command is given, then I2GRID assumes by default that areas are to be defined on the basis of FRT graphical type. This is equivalent to giving the command AREA FRT. The command AREA FC may be used to override this default.

Messages:

The following error message is specific to the FRT command:

```
*** ERROR *** Specifying command FRT
Input FRT filename is missing
```

Examples:

```
I2GRID>FRT LSL.FRT<CR>
I2GRID>
```

GO

Initiates the IFF to DTI conversion process.

FORMAT: GO

Command parameters: None

DESCRIPTION:

This command initiates the vector to raster conversion process.

An input IFF file must have been previously defined using the IFF command, and an output DTI file using the OPEN or CREATE commands. If the CREATE command has been used, then the x and y grid interval of the output file must also have been previously defined by means of the GRID command.

If the BACKGROUND command has been given, then a check is made before conversion proceeds, that the background value is valid for the output DTI data type.

Successful conversion is indicated by the display of DTI header details (including the minimum and maximum data values), and by the I2GRID> prompt on the terminal.

Messages:

The following messages are specific to the GO command:

```
*** ERROR *** Specifying command GO
No output DTI file has been defined
Use the command OPEN or CREATE
```

```
*** ERROR *** Specifying command GO
No input IFF file has been defined
Use the command IFF
```

```
*** ERROR *** Specifying command GO
Matrix grid intervals are not defined
Use the command GRID
```

```
*** ERROR *** Specifying command GO
Background value invalid for output data type
Redefine the value using the BACKGROUND command
```

Example:

I2GRID>**GO<CR>**
I2GRID>

GRID

Defines the column and row grid interval of the output matrix

---

FORMAT: GRID column_interval row_interval

Command parameters:

column_interval

The column grid interval. This is the distance in the x axis between successive matrix points. The grid interval is specified in IFF units. A positive real (floating point) value is required.

row_interval

The row grid interval. This is the distance in the y axis between successive matrix points. The grid interval is specified in IFF units. A positive real (floating point) value is required.

DESCRIPTION:

The GRID command defines the output matrix column and row grid interval values. The command is only required if conversion is to a new output DTI file specified using the CREATE command; if conversion is to an existing DTI file the grid interval values in the header of this file are used.

The grid interval values are used to determine in conjunction with the IFF range or IFFWINDOW values, the dimensions (ie. number of columns and rows) of the output matrix. It is important that grid values are chosen carefully to avoid excessively large output DTI files being produced, and according to the application of the gridded data.

Messages:

The following error messages are specific to the GRID command:

```
*** ERROR *** Specifying command GRID
Command requires 2 arguments

*** ERROR *** Specifying command GRID
Command requires 2 real numbers greater than 0
```

Examples:

I2GRID>**GRID 2.5 3.5<CR>**
I2GRID>

HEADER_TYPE

Specifies the format of the output DTI file header.

FORMAT: **HEADER_TYPE format**

Command parameters:

format

A keyword defining the DTI file header format, chosen from:

LSLA - Create a DTI file with a LSLA style header. Any map projection data will be transferred from the IFF file to the DTI projection record.

UHL1 - Create a DMA Change 2 DTI file with a blank UHL1 record

TED4 - Create a DMA Change 3 or 4 DTI file with blank DSI and ACC records

---

DESCRIPTION:

The HEADER_TYPE command may be used to define the header format of the output DTI file. The command is only applicable if output is to a new DTI file specified using the CREATE command. If output is to an existing file using the OPEN command, the header type is derived from this file, and cannot be changed.

If output is to a DTI file with a LSLA header, projection information is transferred from the IFF Map Descriptor to the DTI Projection Record. If output is to a UHL1 DTI file, a blank UHL1 header record is created. If output is to a TED4 DTI file, blank DSI and ACC records are inserted into the file header.

By default a DTI file with a LSLA header is created.

Messages:

The following error message is specific to the HEADER_TYPE command:

*** ERROR *** Specifying command HEADER_TYPE
Valid qualifiers are LSLA, UHL1 or TED4

Example:

```
I2GRID>HEADER_TYPE TED4<CR>
I2GRID>
```

HELP

Invokes help on I2GRID commands.

FORMAT: **HELP [command]**

Command parameters:

command

the command on which help is required

DESCRIPTION:

A brief description is given of the function and format of the specified command.
If no parameter is supplied then a list of all commands available is given.

Messages: None.

Examples:

I2GRID>**HELP CREATE**
I2GRID>

IFF

Opens an IFF (Internal Feature Format) file.

FORMAT: **IFF file-spec**

Command parameters:

file-spec

The file specification of the input IFF file. Any part of the file specification not supplied will be taken from the default 'LSL$IF:IFF.IFF although if no file specification is supplied an error message will be generated.

DESCRIPTION:

The IFF command specifies the name of an Internal Feature Format file, containing the vector data that is to be converted to a grid representation.

On opening the IFF file, a default area of interest equivalent to the IFF coordinate range is set. This may be subsequently altered using the IFFWINDOW command. The IFF range or IFFWINDOW values, in conjunction with the output matrix grid interval values, determine the dimensions (number of columns and rows) of the output DTI file.

The IFF filename, file range and current IFF window may be examined using the command SHOW IFF.

Messages:

The following error message is specific to the IFF command:

*** ERROR *** Specifying command IFF
Input IFF filename is missing

Examples:

I2GRID>**IFF TEST<CR>**
I2GRID>

IFFWINDOW

Defines an area of interest in the IFF file.

FORMAT: **IFFWINDOW swx swy nex ney**

Command parameters:

swx swy nex ney

The IFF coordinates of the south-west (bottom lefthand) and north-west (top righthand) corners of the window. The coordinates are specified in IFF units, as real (floating point) values.

DESCRIPTION:

IFFWINDOW is used to define a rectangular area of interest within the IFF file. Only IFF features that lie within this area are converted.

By default or if the ENABLE ABSOLUTE command has been given then window values should be supplied in absolute IFF units. Otherwise coordinates should be specified as values relative to the IFF file local origin.

The IFFWINDOW values, in conjunction with the output matrix grid interval values, determine the dimensions (number of columns and rows) of the output DTI file.

The IFF filename, file range and current IFF window may be examined using the command SHOW IFF.

Messages:

The following error messages are specific to the IFFWINDOW command:

*** ERROR *** Specifying command IFFWINDOW
No IFF file is open

*** ERROR *** Specifying command IFFWINDOW
NE corner should exceed SW corner

Examples:

I2GRID>**IFFWINDOW 310000 220000 270000 180000 <CR>**
I2GRID>

OPEN

Opens an existing matrix to receive the gridded attribute information when the GO command is given (compare CREATE command).

FORMAT: **OPEN file-spec**

Command parameters:

file-spec

The file specification for the DTI file.
Any part of the file specification not supplied is taken from the default 'LSL$DTI:DTI.DTI', although if no file-spec is supplied an error message will be generated.

DESCRIPTION:

The OPEN command specifies the name of an existing DTI file that will be opened when the GO command is given, and into which the gridded attribute data will be encoded.

The header type, data type and grid interval values are read from the header of this file, and **cannot** be reset using the commands HEADER_TYPE, DATA_TYPE and GRID. Similarly the BACKGROUND command cannot be used to change the file's existing background data value.

Messages:

The following error message is specific to the OPEN command:

*** ERROR *** Specifying command OPEN
DTI filename is missing

Example:

I2GRID>**OPEN FORESTS<CR>**
I2GRID>

PRESET FC

Assigns a preset value to features on the basis of feature code.

FORMAT: PRESET FC feature_code [,...] = data_value

Command parameters:

feature_code

An integer feature code in the range 0 to 32767. Multiple feature codes may be specified separated by commas or spaces, while a range of feature codes may be specified by means of a '-'

=

This sign separates a feature code or list of feature code values from the preset data_value. It must always be present.

data_value

The preset value. Depending on the data type of the output DTI file, an integer or real data value is required. The value should be selected such that it falls within the valid range for the data type.

DESCRIPTION:

The PRESET FC command is used to assign a preset value to particular IFF features. Features are assigned a preset value on the basis of feature code. The preset data value is output in preference to any value associated with the currently selected attribute.

The PRESET FC command may be given a number of times to assign different preset data values to a different set of feature codes.

Use of the PRESET values may be enabled or disabled using the ENABLE and DISABLE PRESET commands. By default use of the PRESET values is enabled.

The current settings may be examined by means of the SHOW PRESET command.

Messages:

The following error messages are specific to the PRESET and PRESET FC command:

*** ERROR *** Specifying command PRESET
Valid qualifiers are FC or FSN

*** ERROR *** Specifying command PRESET
Unexpected end of line
Missing = delimiter

```
*** ERROR *** Specifying command PRESET
Unexpected end of line
Missing preset value

*** ERROR *** Specifying command PRESET
Unexpected end of line
Second part of range missing

*** ERROR *** Specifying command PRESET
Unable to read preset value

*** ERROR *** Specifying command PRESET
Missing FC or FSN values

*** ERROR *** Specifying command PRESET
Unable to read FC or FSN value
```

Example:

I2GRID>**PRESET FC 10-20,34 = 10<CR>**
I2GRID>**PRESET FC 1,2,3 = 64<CR>**
I2GRID>

PRESET FSN

Assigns a preset value to features on the basis of feature serial number.

FORMAT: **PRESET FSN fsn [,...] = data_value**

Command parameters:

fsn

An integer feature serial number in the range 0 to 65534. Multiple feature serial numbers may be specified separated by commas or spaces, while a range of feature serial numbers may be specified by means of a '-'

=

This sign separates a feature serial number or list of feature serial numbers from the preset data_value. It must always be present.

data_value

The preset value. Depending on the data type of the output DTI file, an integer or real data value is required. The value should be selected such that it falls within the valid range for the data type.

DESCRIPTION:

The PRESET FSN command is used to assign a preset value to particular IFF features. Features are assigned a preset value on the basis of feature serial number. The preset data value is output in preference to any value associated with the currently selected attribute.

The PRESET FSN command may be given a number of times to assign different preset data values to a different IFF features.

Use of the PRESET values may be enabled or disabled using the ENABLE and DISABLE PRESET commands. By default use of the PRESET values is enabled.

The current settings may be examined by means of the SHOW PARAMETERS command.

Messages:

The following error messages are specific to the PRESET and PRESET FSN command:

*** ERROR *** Specifying command PRESET
Valid qualifiers are FC or FSN

*** ERROR *** Specifying command PRESET

```
Unexpected end of line
Missing = delimiter

*** ERROR *** Specifying command PRESET
Unexpected end of line
Missing preset value

*** ERROR *** Specifying command PRESET
Unexpected end of line
Second part of range missing

*** ERROR *** Specifying command PRESET
Unable to read preset value

*** ERROR *** Specifying command PRESET
Missing FC or FSN values

*** ERROR *** Specifying command PRESET
Unable to read FC or FSN value
```

Example:

```
I2GRID>PRESET FSN 10-20,34 = 10<CR>
I2GRID>PRESET FSN 1,2,3 = 64<CR>
I2GRID>
```

PRIORITY CURRENT

The PRIORITY CURRENT command determines what action will be taken if more than one IFF coordinate references the same point in the output matrix.

FORMAT: PRIORITY CURRENT

Command parameters: None.

DESCRIPTION:

The PRIORITY command determines what action will be taken if more than one IFF coordinate references the same point in the output matrix.

If PRIORITY CURRENT is specified then the value recorded at the matrix position is always the attribute associated with the current IFF feature. A previous matrix value may therefore be overwritten during conversion, and the final matrix value depends on the order in which features are arranged in the IFF file.

Use of the SHOW PARAMETERS command is recommended to check on the current PRIORITY option before the GO command is given.

The introduction contains further explanation and guidance on the use of the PRIORITY command.

Messages:

The following error message is specific to the PRIORITY command:

*** ERROR *** Specifying command PRIORITY
Valid qualifiers are CURRENT, FIRST, LARGEST or SMALLEST

Example:

I2GRID>**PRIORITY CURRENT<CR>**
I2GRID>

PRIORITY FIRST

The PRIORITY FIRST command determines what action will be taken if more than one IFF coordinate references the same point in the output matrix.

FORMAT: **PRIORITY FIRST**

Command parameters: None.

DESCRIPTION:

The PRIORITY command determines what action will be taken if more than one IFF coordinate references the same point in the output matrix.

If PRIORITY FIRST is specified then the attribute associated with the first feature in the IFF file to reference the point, is used. An existing matrix value will not be overwritten by any other value.

Use of the SHOW PARAMETERS command is recommended to check on the current PRIORITY option before the GO command is given.

The introduction contains further explanation and guidance on the use of the PRIORITY command.

Messages:

The following error message is specific to the PRIORITY command:

```
*** ERROR *** Specifying command PRIORITY
Valid qualifiers are CURRENT, FIRST, LARGEST or SMALLEST
```

Example:

I2GRID>**PRIORITY FIRST<CR>**
I2GRID>

PRIORITY LARGEST

The PRIORITY LARGEST command determines what action will be taken if more than one IFF coordinate references the same point in the output matrix.

FORMAT: **PRIORITY LARGEST**

Command parameters: None.

DESCRIPTION:

The PRIORITY command determines what action will be taken if more than one IFF coordinate references the same point in the output matrix.

If PRIORITY LARGEST is specified then the value recorded at the matrix position is the largest attribute value encountered. A previous matrix value may therefore be overwritten during conversion.

Use of the SHOW PARAMETERS command is recommended to check on the current PRIORITY option before the GO command is given.

The introduction contains further explanation and guidance on the use of the PRIORITY command.

Messages:

The following error message is specific to the PRIORITY command:

```
*** ERROR *** Specifying command PRIORITY
Valid qualifiers are CURRENT, FIRST, LARGEST or SMALLEST
```

Example:

I2GRID>**PRIORITY LARGEST<CR>**
I2GRID>

PRIORITY SMALLEST

The PRIORITY SMALLEST command determines what action will be taken if more than one IFF coordinate references the same point in the output matrix.

FORMAT: **PRIORITY SMALLEST**

Command parameters: None.

DESCRIPTION:

The PRIORITY command determines what action will be taken if more than one IFF coordinate references the same point in the output matrix.

If PRIORITY SMALLEST is specified then the value recorded at the matrix position is the smallest attribute value encountered. A previous matrix value may therefore be overwritten during conversion.

Use of the SHOW PARAMETERS command is recommended to check on the current PRIORITY option before the GO command is given.

The introduction contains further explanation and guidance on the use of the PRIORITY command.

Messages:

The following error message is specific to the PRIORITY command:

*** ERROR *** Specifying command PRIORITY
Valid qualifiers are CURRENT, FIRST, LARGEST or SMALLEST

Example:

I2GRID>**PRIORITY SMALLEST<CR>**
I2GRID>

RETURN

Restores command input from an indirect command file to SYS$COMMAND.

FORMAT: RETURN

Command parameters: None.

DESCRIPTION:

Restores command input from an indirect command file to SYS$COMMAND.

A typical application is to use an indirect command file to set up a number of run time defaults for a flowline, and then return to input from the terminal for the run specific commands. To do this RETURN must be the last command in the indirect command file.

A RETURN command is only valid if present in a command file.

Messages:

The following messages are specific to the RETURN command:

*** ERROR *** Specifying command RETURN
The RETURN command is only valid in a command file

Returning to terminal input

Examples:

```
I2GRID> @SETUP<CR>
I2GRID> CREATE TEST
I2GRID> GRID 10 10
I2GRID> RETURN
Returning to terminal input
I2GRID>
```

SELECT ALL

Resets all feature selections

FORMAT: **SELECT ALL**

Command parameters: None

DESCRIPTION:

SELECT ALL resets all IFF feature selections made using any of the SELECT or DESELECT commands.

Message:

The following error messages are specific to the SELECT command:

```
*** ERROR *** Specifying command SELECT
Command qualifiers are ALL, FC, FSN or LAYER
```

Examples:

```
I2GRID>SELECT ALL <CR>
I2GRID>SELECT FSN 7-10,56-78 <CR>
I2GRID>SHOW SELECTIONS <CR>
I2GRID>SELECT ALL <CR>
I2GRID>SHOW SELECTIONS <CR>
I2GRID>
```

SELECT FC

Includes an IFF feature for conversion on the basis of feature code.

FORMAT: **SELECT FC feature_code [,...]**

Command parameters:

feature_code

An integer feature code in the range 0 to 32767. Multiple feature codes may be specified separated by commas or spaces, while a range of feature codes may be specified by means of a '-'. eg. SELECT FC 10-13 includes feature codes 10,11,12 and 13.
Alternatively a valid FRT group name may be specified eg. SELECT FC RIVERS

DESCRIPTION:

The SELECT FC command is used to include an IFF feature for conversion. Features are included on the basis of their feature code.

By default I2GRID will convert all features within an IFF file.

Use of the SHOW SELECTIONS command is recommended to check on feature selections before the GO command is given.

Selection on the basis of group name is valid only if the FRT command has been used to previously read a Feature Representation Table.

Messages:

The following error messages are specific to the SELECT command:

```
*** ERROR *** Specifying command SELECT
Command qualifiers are ALL, FC, FSN, or LAYER
```

Examples:

```
I2GRID>SELECT FC 1<CR>
I2GRID>SELECT FC RAILWAYS<CR>
I2GRID>SELECT FC RIVERS 7-10,56-78<CR>
I2GRID>
```

SELECT FSN

Includes an IFF feature for conversion on the basis of feature serial number.

FORMAT: SELECT FSN fsn [,...]

Command parameters:

fsn

An integer feature serial number in the range 0 to 65534. Multiple feature serial numbers may be specified separated by commas or spaces, while a range of numbers may be specified by means of a '-'. eg. SELECT FSN 10-13 includes feature serial numbers 10,11,12 and 13.

DESCRIPTION:

The SELECT FSN command is used to include IFF features for conversion. Features are included on the basis of their feature serial number.

By default I2GRID will use all features within an IFF file.

Use of the SHOW SELECTIONS command is recommended to display feature selections before the GO command is given.

Messages:

The following error messages are specific to the SELECT command:

*** ERROR *** Specifying command SELECT
Command qualifiers are ALL, FC, FSN, or LAYER

Examples:

I2GRID>**SELECT FSN 4<CR>**
I2GRID>**SELECT FSN 7-10,56-78<CR>**
I2GRID>

SELECT LAYER

Includes an IFF feature for conversion on the basis of layer number.

FORMAT: **SELECT LAYER layer [,...]**

Command parameters:

layer

An integer layer number in the range 1 to 32767. Multiple layer numbers may be specified separated by commas or spaces, while a range of layer numbers may be specified by means of a '-'. eg. SELECT LAYER 10-13 includes all features in layers 10,11,12 and 13.

DESCRIPTION:

The SELECT LAYER command is used to include IFF features for conversion. Features are included on the basis of IFF layer.

By default I2GRID will use all features within an IFF file.

Use of the SHOW SELECTIONS command is recommended to display feature selections before the GO command is given.

Messages:

The following error messages are specific to the SELECT command:

*** ERROR *** Specifying command SELECT
Command qualifiers are ALL, FC, FSN, or LAYER

Examples:

I2GRID>**SELECT LAYER 15<CR>**
I2GRID>**SELECT LAYER 1,7-10<CR>**
I2GRID>

SHOW ENABLE

Shows the current status of those options that may be enabled by means of the ENABLE command, or disabled using the DISABLE command.

FORMAT: **SHOW ENABLE**

Command parameters: None.

DESCRIPTION:

Displays the current status of all the I2GRID options that may be enabled or disabled using the ENABLE and DISABLE commands.

The name of the option is shown, followed by either the word ON or OFF to indicate its current status.

If the command SHOW ENABLE is used before any ENABLE or DISABLE commands have been given, the default status of the options is displayed.

Messages:

The following message is specific to the SHOW command:

```
*** ERROR *** Specifying command SHOW
Command qualifiers are ENABLE, IFF, PARAMETERS, PRESET or SELECTIONS
```

Examples:

I2GRID>**SHOW ENABLE<CR>**

```
Current status:

ABSOLUTE        On        DFAD              Off        LOG            On

PRESET          On        THICK             Off        TRUNCATE       On

I2GRID>
```

SHOW IFF

Displays information on the IFF file coordinate range, and current IFFWINDOW values.

FORMAT: **SHOW IFF**

Command parameters: None.

DESCRIPTION:

Displays information on the currently selected IFF file.

The name of the IFF file, and the IFF coordinate range and window settings are shown. By default or if the ENABLE ABSOLUTE command has been given, the coordinates are expressed as absolute values. If the DISABLE ABSOLUTE command has been given, the values are displayed relative to the IFF local origin.

Messages:

The following messages are specific to the SHOW and SHOW IFF commands:

*** ERROR *** Specifying command SHOW
Command qualifiers are ENABLE, IFF, PARAMETERS, PRESET or SELECTIONS

*** WARNING *** The input IFF file is undefined

Examples:

```
I2GRID>SHOW IFF<CR>

IFF file:       LSL$IF:TEST.IFF

IFF file range:
SW: 270000.00 180000.00   NE: 310000.00  220000.00
IFF file window :
SW: 280000.00 190000.00   NE: 320000.00  190000.00

I2GRID>
```

SHOW PARAMETERS

Displays information on those I2GRID parameters controlling the format of the output DTI file, and how an attribute value is encoded in the matrix.

FORMAT: **SHOW PARAMETERS**

Command parameters: None.

DESCRIPTION:

The SHOW PARAMETERS command displays information on those parameters controlling the format of the output DTI file, and how an attribute value is encoded in the matrix.

Information is given on the output DTI filename, header and data type and matrix grid interval. Details are also provided on the selected IFF attribute and priority, and on how area features are to be identified.

Messages:

The following message is specific to the SHOW command:

```
*** ERROR *** Specifying command SHOW
Command qualifiers are ENABLE, IFF, PARAMETERS, PRESET or SELECTIONS
```

Examples:

I2GRID>**SHOW PARAMETERS<CR>**

I2GRID>

SHOW PRESET

Displays a list of feature codes or feature serial numbers that have been preassigned a value using the PRESET command.

FORMAT: **SHOW PRESET**

Command parameters: None.

DESCRIPTION:

The SHOW PRESET command displays a list of feature codes or feature serial numbers that have been preassigned a value using the PRESET command.

Messages:

The following message is specific to the SHOW command:

```
*** ERROR *** Specifying command SHOW
Command qualifiers are ENABLE, IFF, PARAMETERS, PRESET or SELECTIONS
```

Examples:

I2GRID>**PRESET FSN 1-3 = 100<CR>**
I2GRID>**SHOW PRESET<CR>**

```
Features with the following codes have pre-assigned values:

Features with the following FSNs  have pre-assigned values:
FSN         :      1 - 3      Value:   100.000

I2GRID>
```

SHOW SELECTIONS

Displays information about current IFF feature selections.

Command parameters: None

FORMAT: **SHOW SELECTIONS**

DESCRIPTION:

The SHOW SELECTIONS command displays information about current IFF feature selections made with the SELECT and DESELECT commands.

Messages:

The following message is specific to the SHOW command:

*** ERROR *** Specifying command SHOW
Command qualifiers are ENABLE, IFF, PARAMETERS, PRESET or SELECTIONS

Examples:

I2GRID>**SHOW SELECTIONS<CR>**
All Layers selected
All Feature Codes selected
All FSNs selected
I2GRID>

SPAWN

The SPAWN command enables a subprocess to be created from within I2GRID.

FORMAT: SPAWN command-line

Command parameters:

command-line

Specifies a DCL command string to be executed as if typed in response to a '$' prompt. When the command completes, the subprocess terminates and control is returned to I2GRID. The command line cannot exceed 80 characters.

DESCRIPTION:

The SPAWN command enables you to create a subprocess while within I2GRID. When the subprocess terminates, control is returned to I2GRID.

Messages:

The following error messages are specific to the SPAWN command:

```
*** ERROR *** Specifying command SPAWN
Command requires a valid DCL command line

*** ERROR *** Specifying command SPAWN
DCL command line cannot exceed 80 characters

*** ERROR *** Unable to spawn command, returning to I2GRID
```

Examples:

I2GRID> **SPAWN DIRECTORY LSL$DTI:*.DTI<CR>**

```
Directory DUA3:[LSL.DTI]

TEST1.DTI;1               8/8       18-AUG-1987 07:56  [LSL,DAVEC]
TEST2.DTI;1               7/8       18-AUG-1987 17:17  [LSL,DAVEC]
TEST2.DTI;2               7/8       18-AUG-1987 17:34  [LSL,DAVEC]

Total of 3 files, 22/24 blocks.

I2GRID>
```

WAIT

The WAIT command causes processing to be suspended for a specified number of seconds.

FORMAT: WAIT seconds

Command parameters:

seconds

The number of seconds for which processing is to be suspended. A real (floating point) value is required.

DESCRIPTION:

The WAIT command causes processing to be suspended for a specified number of seconds. It is designed for use in software demonstration situations and is of no value in a production flowline.

Messages:

The following error message is specific to the WAIT command:

```
*** ERROR *** Specifying command WAIT
Command requires a real argument
```

Examples:

I2GRID> **WAIT 4.0<CR>**
I2GRID>

WIDTH

Defines a line width value that will be associated with all IFF linear features.

FORMAT: **WIDTH width**

Command parameters:

width

The width value to be applied to linear IFF features. The width is specified in IFF units. A positive real (floating point) value is required.

DESCRIPTION:

The WIDTH command is used to specify a width or line thickness value that is applied to linear IFF features prior to scan conversion.

The width is specified in IFF units, and results in a thicker representation of the feature in the output matrix. The exact representation is dependent on the output matrix grid intervals.

To produce a thickened representation of a line, I2GRID converts a linear feature into an area feature with the required thickness prior to scan conversion. Care should be taken that an excessively large width value is not applied to sinuous line features since a poor representation of the thickened line in the output matrix may result.

The option to apply the width value to linear features may be disabled using the DISABLE THICK command. It may be enabled again using the ENABLE THICK command.

Messages:

The following error message is specific to the WIDTH command:

*** ERROR *** Specifying command WIDTH
Command requires a positive real argument

Examples:

I2GRID>**WIDTH 4<CR>**
I2GRID>

EXAMPLE I2GRID SESSION

```
$ I2GRID
TVES module I2GRID of 12:49:09 23-DEC-88
I2GRID> IFF TEST
%LSLLIB-I-IFFOPENED, LSL$IF:TEST.IFF;1 opened for read

LSL$IF:TEST.IFF;0

IFF file range:
SW:       0.0       0.0  NE:      20.0      20.0
IFF file window :
SW:       0.0       0.0  NE:      20.0      20.0

I2GRID> GRID 1 1
I2GRID> DATA BYTE
I2GRID> CREATE TEST
I2GRID> ATTRIBUTE FC
Output of Feature Code selected
I2GRID> GO
Feature      1 converted   value      5
Feature      2 converted   value      5
Feature      3 converted   value      3

File    :  LSL$DTI:TEST.DTI
Header  :  LSLA  Data:  BYTE

Units are DTI matrix values

Matrix Coverage   SW:     1     1  NE:    21    21
Matrix Interval    E:     1         N:     1
Value Range         :     0   to      5

I2GRID> EXIT
 ELAPSED:    0 00:01:39.78  CPU: 0:00:01.18  BUFIO: 56  DIRIO: 17  FAULTS: 196
```

In this example, the IFF file LSL$IF:TEST.IFF is selected as the input IFF containing the vector data to be rasterised. Output is to a new DTI file with the file specification LSL$DTI:TEST.DTI. This file will be created when the GO command is given.
The DATA_TYPE command is used to specify that the attribute data will be transferred to the output file in byte format, in the range 0 through 255. Since DISABLE TRUNCATE has not been specified, any data values greater than this limit will be truncated to fit. The output DTI file will be created by default with a LSLA header structure, and any projection information will be transferred from the IFF file to the DTI file.
The GRID command controls the resolution of the rasterised data, and is used by the program to calculate the number of columns and rows in the output DTI file. If output is to a new DTI file this command must always be given before the GO command.
The ATTRIBUTE FC command has been used to select the transfer the output of the IFF feature code to the DTI file.

```
$I2GRID
TVES module I2GRID of 12:49:09 23-DEC-88
I2GRID> IFF TEST
%LSLLIB-I-IFFOPENED, LSL$IF:TEST.IFF;1 opened for read

LSL$IF:TEST.IFF;0

IFF file range:
SW:       0.0       0.0  NE:      20.0      20.0
IFF file window :
SW:       0.0       0.0  NE:      20.0      20.0

I2GRID> GRID 1 1
I2GRID> DATA BYTE
I2GRID> BACKGROUND 10
I2GRID> CREATE TEST
I2GRID> PRESET FSN 1-3 = 100
I2GRID> SHOW PRESET

Features with the following codes have pre-assigned values:

Features with the following FSNs  have pre-assigned values:
FSN         :      1 - 3      Value:   100.000

I2GRID> GO
Feature      1 converted   value    100
Feature      2 converted   value    100
Feature      3 converted   value    100

File    :  LSL$DTI:TEST.DTI
Header  :  LSLA  Data:  BYTE

Units are DTI matrix values

Matrix Coverage   SW:     1     1  NE:    21    21
Matrix Interval    E:     1         N:     1
Value Range         :    10   to    100

I2GRID> EXIT
 ELAPSED:    0 00:01:02.23  CPU: 0:00:01.53  BUFIO: 53  DIRIO: 12  FAULTS: 332
```

In this example, the IFF file LSL$IF:TEST.IFF is selected as the input IFF containing the vector data to be rasterised. Output is to a new DTI file with the file specification LSL$DTI:TEST.DTI. This file will be created when the GO command is given. The DATA_TYPE command is used to specify that the attribute data will be transferred to the output file in byte format, in the range 0 through 255. The background command has been used to initialise all the matrix points to 10 before the encoding operation begins. The output DTI file will be created by default with a LSLA header structure, and any projection information will be transferred from the IFF file to the DTI file.
The GRID command controls the resolution of the rasterised data, and is used by the program to calculate the number of columns and rows in the output DTI file. If output is to a new DTI file this command must always be given before the GO

command.
The PRESET FSN command has been used to assign a value of 100 with FSNs 1, 2 and 3. This value will be transferred to the DTI file rather than an attribute value when these features are processed.

MESSAGES (OTHER)

In addition to messages which are generated by the program itself, other messages may be produced by Laser-Scan libraries. In particular, messages may be generated the IFF Library, DTI library, and by the Laser-Scan I/O library, LSLLIB.

IFF library messages are introduced by '%IFF', and are documented in the IFF library users' guide.DTI library messages are introduced by '%DTI', and are documented in the DTILIB Reference Manual. In all cases the messages indicate a fatal error, that will cause processing to halt.

LSLLIB messages are introduced by '%LSLLIB' and are generally self-explanatory. Such messages rarely indicate a fatal error, and are generated most frequently by entering a command in an invalid format in response to the I2GRID prompt.

CHAPTER 6

MODULE PROFILE

MODULE PROFILE

Vector Profile Generation from a DTM

Issue 2.0 - 9 March 1989

Science Park, Milton Road, Cambridge, England CB4 4FY tel: (0223) 420414

Document "PROFILE Reference"			Category "Reference"
Document Issue	2.0	D R Catlow	09-March-1989
	1.1	F J Brown	30-November-1988
	1.0	D R Catlow	13-May-1988

MODULE PROFILE

FUNCTION

PROFILE is a utility to construct a series of path profiles lines through a Digital Terrain Model (DTM). Typically, the origin of the profiles will be a base station, and the profile data will be used for radio frequency propagation analysis. Profiles are constructed either radially around the base station, or between each DTM node and the base station. Options are provided in the program to restrict profile generation to a rectangular area of interest or to a sector of interest.

Input is a DTM, and optionally a series of clutter grid files, held in a Laser Scan Digital Terrain Image (DTI) format. Output is to a Laser Scan vector Internal Feature Format (IFF) file.

Each profile is written to the IFF file as a separate feature. The bearing and length of the profile is stored in the IFF file, along with the x y and z coordinates of the profile sample points. Additionally a clutter index value is stored for each sample point.

FORMAT

$ PROFILE

PROMPTS

PROFILE is an interactive, command driven program. Command input is expected when the following prompt is issued:

PROFILE >

Commands may be issued at the terminal in response to the prompt, or in the case of a number of commands, by pressing a button on a digitising puck.

DESCRIPTION

GENERAL

PROFILE constructs a series of path profiles lines through a Digital Terrain Model (DTM).

The input to the program is a DTM, and optionally a series of clutter grid files, held in a Laser Scan Digital Terrain Image (DTI) format. Utilities to convert data held in other formats (eg. DTED, NTF) to a DTI format are available.

The DTM defines the elevation of the terrain surface. It consists of a set of elevation values recorded on a regular rectangular grid. The DTM is specified using the FILEIN command.

A clutter grid file contains height information relating to surface features such as buildings, vegetation or water. Information on each group of features must be contained in a separate file. The specification for a clutter file is given by means of the BUILDING, VEGETATION and WATER commands. The use of a clutter file is optional.

Output from the program is to a Laser Scan vector Internal Feature Format (IFF) file. Each profile is written to the IFF file as a separate feature. The bearing and length of the profile is held, along with the x y and z coordinates of the profile sample points. Additionally a clutter index value is stored for each profile sample point.

Typically, the origin of the profiles will be a base station. The profiles are constructed either radially around the base station position, or between each DTM node and the base station. Options are provided in the program to restrict profile generation to a rectangular area of interest or to a sector of interest.

THE OUTPUT IFF FILE

The profiles calculated by PROFILE are output to a Laser Scan Internal Feature Format (IFF) file.

The Map Descriptor entry of the IFF file is taken from any projection details held in the header of the DTI file.

Each profile is output to the IFF file as a separate feature. Attributes associated with a profile are held either in an Ancillary Code (AC) entry, or on a per point basis with the coordinate information in a Coordinate Block (CB) entry. The bearing and length of the profile is stored, along with the x y and z coordinates of the profile sample points. Additionally a clutter index value is stored for each sample point.

A profile feature will have the following structure in the IFF file:

NF
FS
AC
AC
CB
EF

The NF indicates the start of a feature, and defines the feature serial number (FSN). The FSN is incremented by 1 for each new feature up to a maximum value of 65535, at which point the FSN is reset to 1. Depending on the number of features in the file, the FSN may therefore be used to uniquely identify a profile.

The Feature Status (FS) entry holds the feature code of a profile. PROFILE assigns the same feature code to all features in the IFF file. By default features are given a value of 1. An alternative feature code may be specified using the command SET FC.

The first Ancillary Code (AC) entry in a feature, holds the length of the profile. This is the distance of the profile end point from the profile origin (base station). The distance is stored in metres as a real (floating point) number. By default, PROFILE uses an AC type 1002 to hold the distance value. An alternative AC type may be specified using the command SET AC_LENGTH.

The second Ancillary Code (AC) entry in a feature, holds the bearing of the profile from the profile origin (base station). The bearing is specified in degrees from north eg. 90.0 indicates a bearing that is due east. A real (floating point) value is held in the AC. By default PROFILE uses an AC type 1001 to hold the bearing value. An alternative AC type may be specified using the command SET AC_BEARING.

The Coordinate Block (CB) entry stores coordinate and attribute information associated with profile sample points.
The x and y coordinates are held as real (floating point) values in the first two columns of the coordinate block. They define the position of the sample points in metres, relative to the SW corner of the DTM rectangular area of interest. The true (absolute) position of a sample point may be determined by adding the x and y local origin values held in the IFF map descriptor (MD) record to the CB coordinate values.
The height (z value) of a sample point is held as a real (floating point) value in column 3 of the coordinate block. The z value is the height of the terrain surface (above sea level), plus the height of any clutter feature present at the sample point.
Column 4 of the coordinate block holds a clutter index, and is used to determine the type of clutter present at the sample point. By default, PROFILE assigns an attribute code of 1003 to the clutter index attribute code. An alternative attribute code may be specified using the command SET AC_CLUTTER. The clutter index value should be interpreted as a longword integer number, and has the following meaning:

Clutter Index	0	No clutter feature is present at the sample point. In this case the Z value represents a

		terrain surface height derived from the DTM.
Clutter Index	1	A building is present at the sample point.
Clutter Index	2	A wood or some other type of significant vegetation is present at the sample point.
Clutter Index	3	A water feature is present at the sample point.

Depending on the distance of the profile, and the rate of sampling along the profile, a feature may contain multiple CB entries. Coordinate data for a maximum of 200 sample points are stored in a single coordinate block.

Note that in order to reduce the amount of space occupied by a coordinate block, the coordinate block is compressed by PROFILE before it is written to the output IFF file. This means that if either the Z value or the clutter index value is constant for all the points in the CB, it will be written as a fixed attribute, rather than as an attribute on a per point basis.

The End of Feature (EF) entry indicates the end of an IFF feature ie. the end of a profile.

AREA OF INTEREST

It is possible to restrict profile generation both to a rectangular area of interest, and to a sector of interest in the DTM.

A rectangular area of interest is defined using the **WINDOW** command. The coordinates of the SW (bottom left) corner and NE (top right) corner of the rectangle are required, and may be entered via the terminal keyboard in the current units of measurement, or from the digitising table. The window coordinates are converted to the nearest DTM column and row positions on input. Using this command for example, calculation of profiles may be restricted to a 10km square area.

In addition to a rectangular area, it is also possible to define a sector of interest. A sector of interest is specified using the **SECTOR** command, and is defined in terms of a sector radius, and optionally 2 bearing values that define the angular limits of the sector. The origin of the sector is the profile origin.

If no bearing values are supplied a 360 degrees scan around the base station position is carried out. The sector radius determines the outer limit of the circular scan.

The order in which the bearing values are specified determines whether the area of interest is inclusive or exclusive to the angular limits. A clockwise convention is used. For example if the first bearing is defined as 40 degrees and the second bearing as 80 degrees, the angular width of the sector of interest will be 40 degrees rather than 320 degrees. If the values are supplied in the order 80 and 40 degrees, the angular width of the sector of interest will be 320 degrees.

Note that a sector of interest will be clipped to any rectangular area of interest that may have been defined (see Figure 1).

SAMPLE INTERVAL

The sample interval determines the separation of profile sample points, and therefore controls the rate at which the DTM height values are sampled.

A sample interval that is constant along the length of a profile may be specified using the command **SAMPLE_INTERVAL**. Alternatively, a sample interval that varies with the distance of the profile sample point from the profile origin (base station) may be selected, using one or more **BAND** commands.

The **BAND** command is used to specify the width of a concentric band surrounding the base station, and the sampling rate within the band. Up to 20 bands may be specified. Band 1 is the band immediately surrounding the profile origin; band 2 surrounds band 1 and so on (Figure 2). The bands must be defined in ascending sequence. Typically this mechanism is used to sample at a lower rate in the immediate vicinity of a base station, and at a higher rate towards the edge of the area of interest.

If the band width is not exactly divisible by the band sample interval, the program will ensure that a sample point is generated exactly on the outer edge of the band. If the total width of all the bands is less than the sector distance, or is insufficient to extend to the corners of the rectangular area of interest, the width of the highest band will be extended as necessary.

CALCULATION OF THE HEIGHT AT A PROFILE SAMPLE POINT

The height of the terrain surface at a sample point is calculated from the DTM. If the sample point does not lie exactly on a DTM node, bilinear interpolation, using the height of the surrounding 4 nodes, is applied to derive a height value.

If one or more clutter grid files have been specified, any clutter height value at the sample point is added to the terrain height. If the sample point does not lie exactly on a grid node, the height value associated with the nearest node in the clutter grid file is used.

A clutter value indicating whether the terrain height has been adjusted using a building, vegetation or water value, is stored in the IFF file for each sample point.

If a value of -1 is found in the clutter matrix, it indicates to the program that no feature is present at the sample point. In this situation a clutter index of 0 will be recorded for the sample point in the coordinate block.

Note that a warning message will be generated if a height for a sample point is recorded in more than one clutter file. In this situation the user will be informed which clutter value has been used.

PROFILE INTERVAL

The **PROFILE_INTERVAL** command may be used to determine both how the profiles are constructed, and how many profiles are constructed. Profiles may either be generated radially around the base station at a fixed angular interval, or alternatively between the base station and each DTM node in the area of interest.

If the value supplied with the **PROFILE_INTERVAL** command is greater than 0, then profiles are constructed radially around the base station. The angular separation (ie. the profile interval) is specified by means of the command parameter. The SECTOR command may be used in conjunction with this command to control the number of profiles that are constructed. The radial profiles are constructed in clockwise order.

If the profile interval is specified as 0, then PROFILE will construct a path profile between the base station and each DTM node that lies within the rectangular or sector areas of interest. For example if a rectangular area of interest of 100 columns and 100 rows has been defined, then a total of 9999 separate profiles will be generated. (A profile between the base station and the base station is not constructed.) **Note that this is the default action of the program.**

USE OF A DIGITISING TABLE AND PUCK BUTTONS

If your workstation incorporates a digitising table with a 16 button puck, then it is possible to register a source document to the DTM, and to issue a number of the PROFILE commands by pressing an appropriate button on the table puck.

The Laser Scan Table Monitor Utility controls reading from the digitising table. It must be set up on your system if PROFILE is to take input from the digitising table. The automatic initialisation of the table monitor utility is optionally invoked by means of the logical name:

```
LSL$AUTO_ENABLE_TABLE
```

If the logical name is defined as "1", initialisation should be carried out on program startup. If the logical name is absent, or is defined to anything other than "1", the table monitor system is not automatically invoked. To cause automatic initialisation of the table monitor utility on program startup issue the following command before running PROFILE:

```
ASSIGN 1 LSL$AUTO_ENABLE_TABLE
```

After program startup initialisation of the table monitor system can be invoked by the command **ENABLE TABLE** and subsequently halted by the command **DISABLE TABLE**.

If table monitor initialisation fails, either at program startup or after an **ENABLE TABLE** command, then the message:

```
*** WARNING *** Unable to initialise the table monitor
PROFILE will assume no table is available
```

will be output before the prompt **PROFILE>** is displayed on the terminal. If the event of failure, input from the digitising table or puck button is not possible, and the program will accept commands only from the terminal.

Table initialisation failure is generally because no Table Monitor is currently active, or because the Table Monitor is locked by another user. If neither of these reasons appear to apply, you should consult the TABLIB Reference Manual, or seek guidance from your system manager.

A map is registered to the DTM using the command **SETUP MAP**. Before giving this command you should have first selected the input DTM, and have securely attached the source document to the digitising table.

Registration is performed by digitising 4 registration points. The registration points represent 4 points on the map that correspond to the 4 corners of the DTM. The points are digitised in the order top left (NW), bottom left (SW), bottom right (SE) and top right (NE) using any button on the table puck. A point is digitised in response to an explanatory prompt on the terminal.

An error message is generated if any of the angles of the digitised rectangle are less than 88 degrees, or greater than 92 degrees (ie. if the corner points of the rectangle are more than 2 degrees off rectangular). In this situation you will be asked to redigitise the 4 registration points.

Once a map has been positioned on the table, it is possible to define an area of interest in the DTM by giving the WINDOW command and digitising two points on the map. It is also possible to digitise the position of the base station (profile origin), a sector radius and angular limits, and a profile end point.

The buttons on the table puck have the following meanings.

0 POSITION	1 WINDOW	2 SECTOR	3 POINT
4 undefined	5 undefined	6 undefined	7 undefined
8 undefined	9 undefined	A undefined	B undefined
C undefined	D undefined	E undefined	F ABANDON

BUTTON 0 is used to define the position of the base station ie. the profile origin point. A single point inside the map area should be digitised.

BUTTON 1 is used to give a WINDOW command. The button may be pressed either inside or outside the map area. After pressing the button, you will be asked to digitise 2 points inside the map area, defining the SW corner (bottom left) and NE corner (top right) of a rectangular area of interest.

BUTTON 2 is used to give a SECTOR command. A single point inside the map area should be digitised. The coordinates of this point are used to calculate the radius of the sector (ie. the distance from the profile origin to the sector arc). The position of the base station must have been previously defined.
In addition you will be asked to digitise two points that define the angular limits of the sector. The points are digitised in response to prompts on the terminal. Any button on the table puck may be used. If **Button F** is used to abandon the command, the program will assume a 360 degrees scan around the base station is required.

BUTTON 3 is used to define the position of a profile end point. A single point inside the map area should be digitised. The program will immediately calculate the profile, and write the coordinate and

attribute values to the output IFF file. Any number of profiles may be defined in this way.
The position of the base station, a sampling interval and the name of an output IFF file must have been defined before the command is issued for the first time.

BUTTONS 4 to E are currently undefined.

BUTTON F may be used to abandon map set up, window or sector definition. When the button is pressed, control is returned to the terminal and the prompt **PROFILE>** is displayed.

PROFILE COMMANDS

@

Take command input from the specified file.

FORMAT: @ file-spec

Command parameters:

file-spec

The file to be opened and used for command input.

Any parts of the file-spec not supplied will be taken from the default specification 'SYS$DISK:[].COM;0'.

DESCRIPTION:

PROFILE offers the facility of command input from an indirect command file. The '@' character preceding a file-spec will cause PROFILE to open and read commands from the specified file until:

1. a RETURN command is detected and command input is returned to SYS$COMMAND.

2. the end of file is detected. This provokes an error message and command input is returned to SYS$COMMAND.

Nested command files are not supported (i.e. a command file containing an '@' command), although sequential '@' commands are supported when read from SYS$COMMAND.

As an aid to batch log interpretation PROFILE will echo all commands read from an indirect command file.

Typically the commands contained in the file will define frequently used values, such as default BAND and SECTOR parameters.

Messages:

The following messages are specific to the @ command:

*** ERROR *** Specifying command @
Command file specification is missing

*** ERROR *** Specifying command @
Unable to open indirect command file 'file-spec'

*** ERROR *** Specifying command @
Nested command files not supported

Examples:

```
PROFILE> @DEFAULT_BANDS<CR>
PROFILE> BAND 1 500.0 50.0
PROFILE> BAND 2 1000.0 100.0
PROFILE> BAND 3 2000.0 200.0
PROFILE> RETURN
Returning to terminal input
PROFILE>
```

!

Treat all text to the right of the '!' as a comment.

FORMAT: ! [comment text]

Command parameters:

comment text

text that is to be treated as a comment and which will be excluded from command interpretation.

DESCRIPTION:

All text (and numbers) which lie to the right of a '!' character are excluded from command interpretation. Comments are useful for annotating command procedures used in batch processing etc.

Messages: None.

Examples:

PROFILE> **FILEIN TEST !open the file<CR>**
PROFILE> **!Define a window<CR>**
PROFILE> **WINDOW 1 1 10 10<CR>**
PROFILE>

BAND

BAND is used to define the width of a concentric band surrounding the base station (profile origin), and the rate at which the DTM is sampled within the band. A series of BAND commands may be used to specify a sample interval along a profile, that varies with distance from the profile origin.

FORMAT: **BAND band_number width sample_interval**

Command parameters:

band_number

Band_number is the band identifier. The band immediately surrounding the base station is band 1; band 2 surrounds band 1 and so on. An integer value in the range 1 to 20 is required.

width

Width defines the width of the band. The width is specified in metres, as a real (floating point) value.

sample_interval

Sample_interval defines the rate at which the DTM is sampled within the band. The sample interval is specified in metres as a real (floating point) value.

DESCRIPTION:

BAND is used to define the rate at which the DTM is sampled along a profile. Up to 20 different bands may be specified, each with a different width and sample interval. This allows a profile to be generated using a sample interval that varies with distance from the base station. It is generally used to sample the DTM at a lower rate in the immediate vicinity of the base station, and at a higher rate towards the edge of the area of interest.
If BAND is specified, any constant sample interval defined using the SAMPLE_INTERVAL command is ignored.

It is a requirement of the program that bands are defined in ascending sequence. For example, if an attempt to define parameters for band 3 is made, before bands 1 or 2 have been defined, an error message will be generated.

The command SHOW PARAMETERS may be used to output details of band parameters to the terminal.

A sample interval or a series of sample intervals must be specified using the SAMPLE_INTERVAL or BAND commands, before the command GO is given to generate the profiles.

Messages:

The following error messages are specific to the BAND command:

*** ERROR *** Specifying command BAND
Command requires 3 arguments

*** ERROR *** Specifying command BAND
Band number must be in the range 1 to 20

*** ERROR *** Specifying command BAND
Parameters for lower band <number> must be defined first

*** ERROR *** Specifying command BAND
Band width must be specified as a positive real number

*** ERROR *** Specifying command BAND
Sample interval must be specified as a positive real number

Example:

PROFILE>**BAND 1 1000.5 50.0<CR>**
PROFILE>**BAND 2 2500.0 100.0<CR>**
PROFILE>

BUILDING

Selects and opens a DTI file that contains clutter information relating to buildings.

FORMAT: **BUILDING file-spec**

Command parameters:

file-spec

The file specification for the input grid file containing clutter information relating to buildings.
Any part of the file specification not supplied is taken from the default 'LSL$DTI:DTI.DTI', although if no file-spec is supplied, you will be asked to supply one in response to the prompt **Input DTI filename>**

DESCRIPTION:

This command opens and maps into memory a DTI file, containing height information relating to buildings. Details derived from the header of the file are displayed on the terminal to confirm that the file has been successfully opened.

A clutter file may only be specified, after a DTM has been selected using the FILEIN command.

If the clutter file contains a DTI Projection Record, the position of the SW corner of the grid may be determined. This enables the clutter data to be correctly positioned with respect to the DTM. If the origin is unset or the file contains no projection record, the SW corner of the clutter file is assumed to register to the SW corner of the DTM.

Messages:

The following error messages are specific to the BUILDING command:

*** ERROR *** Specifying command BUILDING
Input DTI filename is missing

*** ERROR *** Specifying command BUILDING
The input DTM must be specified first

Example:

```
PROFILE>BUILDING TEST_BUILDING<CR>

LSL$DTI:TEST_BUILDING.DTI
Header: MIKE     Data: WORD

Units are Metres

Matrix coverage    SW:    1     1    NE:   301   251
Matrix Window      SW:    1     1    NE:   301   251
Matrix Interval     E:    1           N:     1
Value Range          :      0   to   25

PROFILE>
```

DISABLE ABSOLUTE

Disables the use of absolute coordinates.

FORMAT: DISABLE ABSOLUTE

Command parameters: None.

DESCRIPTION:

DISABLE ABSOLUTE disables the use of absolute coordinates, and is therefore used to cancel an ENABLE ABSOLUTE command. DISABLE ABSOLUTE causes the program to work in relative coordinates. The coordinates displayed in a SHOW FILEIN or SHOW PARAMETERS output will be relative, and those entered in a WINDOW, POINT, or POSITION command are expected also to be relative to the SW corner of the data.
By default coordinates are entered and displayed in absolute form.

Messages:

The following error message is specific to the DISABLE command:

```
*** ERROR *** Specifying command DISABLE
Valid command qualifiers are ABSOLUTE, DIAGNOSTICS or TABLE
```

Examples:

PROFILE> **DISABLE ABSOLUTE<CR>**
PROFILE>

DISABLE DIAGNOSTICS

Disables the output of diagnostics messages.

FORMAT: DISABLE DIAGNOSTICS

Command parameters: None.

DESCRIPTION:

DISABLE DIAGNOSTICS disables the output of diagnostic messages, and is therefore used to cancel an ENABLE DIAGNOSTICS command.
By default diagnostic printout, which is sent to SYS$OUTPUT, is enabled.

Messages:

The following error message is specific to the DISABLE command:

```
*** ERROR *** Specifying command DISABLE
Valid command qualifiers are ABSOLUTE, DIAGNOSTICS or TABLE
```

Examples:

```
PROFILE> DISABLE DIAGNOSTICS<CR>
PROFILE>
```

DISABLE TABLE

Disables the use of a digitising table.

FORMAT: DISABLE TABLE

Command parameters: None.

DESCRIPTION:

DISABLE TABLE disables the use of a digitising table, and is therefore used to cancel an ENABLE TABLE command.
By default the use of a digitising table is determined on program startup by the value assigned to the logical name LSL$AUTOENABLETABLE.

Messages:

The following error message is specific to the DISABLE command:

*** ERROR *** Specifying command DISABLE
Valid command qualifiers are ABSOLUTE, DIAGNOSTICS or TABLE

Examples:

PROFILE> **DISABLE TABLE<CR>**
PROFILE>

ENABLE ABSOLUTE

Causes the program to work in absolute coordinates.

FORMAT: ENABLE ABSOLUTE

Command parameters: None.

DESCRIPTION:

ENABLE ABSOLUTE enables the use of absolute coordinates, and is cancelled by a DISABLE ABSOLUTE command. ENABLE ABSOLUTE causes the program to work in absolute coordinates. The coordinates displayed in a SHOW FILEIN or SHOW PARAMETERS output will be absolute, and those entered in a WINDOW, POINT, or POSITION command are expected also to be absolute
By default coordinates are entered and displayed in absolute form.

Messages:

The following error message is specific to the ENABLE command:

*** ERROR *** Specifying command ENABLE
Valid command qualifiers are ABSOLUTE, DIAGNOSTICS or TABLE

Examples:

PROFILE> **ENABLE ABSOLUTE<CR>**
PROFILE>

ENABLE DIAGNOSTICS

Selects the output of diagnostic printout.

FORMAT: ENABLE DIAGNOSTICS

Command parameters: None.

DESCRIPTION:

ENABLE DIAGNOSTICS selects output of diagnostic messages. The messages are sent to SYS$OUTPUT.
The diagnostic printout consists of messages describing the process being performed, the percentage complete, and the number of profiles generated.

Note that if you are using a hardcopy terminal no percentage complete figures are generated.

By default diagnostic printout is selected, and may be turned off using the command DISABLE DIAGNOSTICS

Messages:

The following error message is specific to the ENABLE command:

```
*** ERROR *** Specifying command ENABLE
Valid command qualifiers are ABSOLUTE, DIAGNOSTICS or TABLE
```

Examples:

PROFILE> **ENABLE DIAGNOSTICS<CR>**
PROFILE>

ENABLE TABLE

Enables the use of a digitising table.

FORMAT: ENABLE TABLE

Command parameters: None.

DESCRIPTION:

ENABLE TABLE allows the use of a digitising table. Its effect is cancelled be the DISABLE TABLE command.
By default the use of a digitising table is determined on program startup by the value assigned to the logical name LSL$AUTOENABLETABLE.

Messages:

The following error message is specific to the ENABLE command:

```
*** ERROR *** Specifying command ENABLE
Valid command qualifiers are ABSOLUTE, DIAGNOSTICS or TABLE
```

Examples:

PROFILE> **ENABLE TABLE<CR>**
PROFILE>

EXIT

Terminates the program.

FORMAT: **EXIT**

Command parameters: None.

DESCRIPTION:

The EXIT command is used to exit from PROFILE.
<CTRL/Z> (pressing the Ctrl and Z keys together) may also be used to exit from the program.

Messages: None.

Examples:

PROFILE>**EXIT<CR>**

$

FILEIN

Selects and opens a DTI file that contains the terrain elevation data to be used in profile generation.

FORMAT: **FILEIN file-spec**

Command parameters:

file-spec

The file specification for the input DTM.
Any part of the file specification not supplied is taken from the default 'LSL$DTI:DTI.DTI', although if no file-spec is supplied, you will be asked to supply one in response to the prompt **Input DTI filename>**

DESCRIPTION:

This command opens and maps into memory a DTI file, containing the terrain elevation data to be used for profile generation. Details derived from the header of the file are displayed on the terminal to confirm that the file has been successfully opened.

On opening the DTM, a default area of interest covering the whole of the DTM area is defined. A smaller area of interest may be specified using the WINDOW command.

If the input file contains a DTI Projection Record, the absolute position of the SW corner of the DTM is determined. If the file contains no Projection Record, an origin of 0,0 is assumed, and therefore any coordinate values represent DTM offset values.

The input DTM must be specified before the commands BUILDING, GO, POSITION, SETUP MAP, VEGETATION, WATER and WINDOW.

Messages:

The following error message is specific to the FILEIN command:

*** ERROR *** Specifying command FILEIN
Input DTI filename is missing

Example:

```
PROFILE>FILEIN TEST<CR>

LSL$DTI:TEST.DTI
Header: MIKE     Data: WORD

Units are Metres

Matrix coverage    SW:    1     1    NE:   301   251
Matrix Window      SW:    1     1    NE:   301   251
Matrix Interval     E:    1           N:     1
Value Range          :      0   to    851

PROFILE>
```

GO

The command is used to create the profile features in the output IFF file.

FORMAT: **GO**

Command parameters: None

DESCRIPTION:

GO is used to initiate the generation of the path profiles, and the output of the profiles to the IFF file.

Before the command is given, the input DTM and output IFF file, the position of the profile origin (base station), and a sample interval using either the SAMPLE_INTERVAL or BAND commands, must have been defined. All other optional commands such as WINDOW, PROFILE_INTERVAL and SECTOR should be first specified, if required.

If diagnostics are enabled, details on the progress of the processing operation are output to the terminal. Successful completion of processing is indicated by the display of the prompt **PROFILE>** on the terminal.

<CTRL/C> (pressing the Ctrl and C key together) may be used to abort processing. If <CTRL/C> is used, the output IFF file will be correctly closed and all features created so far will be preserved.

Messages:

The following error message is specific to the GO command:

```
*** ERROR *** Specifying command GO
The input DTM must be specified before GO

*** ERROR *** Specifying command GO
The output IFF file must be specified before GO

*** ERROR *** Specifying command GO
Profile origin position has not been defined

*** ERROR *** Specifying command GO
Profile sample interval has not been defined

*** ERROR *** Specifying command GO
Profile origin is not within the current area of interest
Redefine the position, or window before issuing GO
```

Examples:

```
PROFILE>GO
PROFILE>
```

HELP

Invokes help on PROFILE commands.

FORMAT: **HELP [command]**

Command parameters:

command

the command on which help is required

DESCRIPTION:

A brief description is given of the function and format of the specified command.
If no parameter is supplied then a list of all commands available is given.

Messages: None.

Examples:

PROFILE>**HELP FILEIN**

FILEIN

Opens and maps into memory a DTI file, containing the terrain elevation data to be used for profile generation. Details derived from the header are displayed on the terminal to confirm that the file has been successfully opened

PROFILE>

IFF

Specifies the name of the output IFF file.

FORMAT: **IFF file-spec**

Command parameters:

file-spec

The file specification for the output IFF file.
Any part of the file specification not supplied is taken from the default 'LSL$IF:IFF.IFF', although if no file-spec is supplied, you will be asked to supply one in response to the prompt **Output IFF filename>**

DESCRIPTION:

IFF specifies the name of the output IFF file. This is the file which will contain the terrain profiles.

The commands SET FC and SET LAYER may be used to control the feature code assigned to each profile feature, and define the layer in which the features will be held. The SET AC_BEARING, SET AC_CLUTTER, and SET AC_DISTANCE commands may be used to control the AC type used to hold the profile bearing, a sample point clutter index, and the profile distance value respectively.

An IFF file must be specified before the command GO is issued.

If the input DTM contains a DTI Projection Record, details from the record are transferred to the Map Descriptor (MD) entry of the IFF file.

Messages:

The following message is specific to the IFF command:

```
*** ERROR *** Specifying command IFF
Output IFF filename is missing
```

Examples:

```
PROFILE>IFF TEST_PROFILES<CR>
PROFILE
```

POINT

Defines the geographical position of a profile end point, and writes the path profile data for a single profile to the output IFF file.

FORMAT: **POINT x_coordinate y_coordinate**

Command parameters:

x_coordinate

The x coordinate position of the profile end point.

y_coordinate

The y coordinate position of the profile end point.

The format of the command parameters is controlled by the UNITS command.

UNITS MATRIX - 2 real values are required defining the position in terms of a DTM column and row number.

UNITS METRES Requires 2 real (floating point) values defining the position in metre values. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates (eg. 6 figure U.K. National Grid coordinates); if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

UNITS SECONDS - Requires 2 real (floating point) values defining the absolute point in seconds of arc. The values are supplied in the order latitude followed by longitude for the SW corner, and latitude followed by longitude for the NE corner. A negative latitude value indicates a position in the Southern Hemisphere, and a negative longitude value a point to the west of Greenwich.

UNITS LATLONG - Requires 2 values defining the absolute latitude and longitude position in the format DDD MM SS H where DDD is the number of degrees, MM is the number of minutes, SS is the number of seconds, and H is the hemisphere (N,S,E,W). The values are supplied in the order latitude followed by longitude for the SW corner, and latitude followed by longitude for the NE corner.

UNITS PROJECTION Requires 2 real (floating point) values defining the position in projection record units. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates; if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

DESCRIPTION:

The POINT command is used to define the geographical position of a profile end point. The point must lie within the geographical bounds of the DTM, but may be outside any currently defined area of interest.

On giving the command the program will immediately calculate the profile, and write the coordinate and attribute values to the IFF file. Any number of profiles may be defined in this way.

If a digitising table is available, and a map has been registered to the DTM using the SETUP MAP command, puck **button 4** may be used to digitise the position of a profile end point.

The position of the base station (profile origin), a sampling interval and the name of an output IFF file must have been defined before the POINT command is given for the first time.

Messages:

The following error messages are specific to the POINT command:

*** ERROR *** Specifying command POINT using puck button
No map set up has been performed

*** ERROR *** Specifying command POINT
The input DTM is undefined

*** ERROR *** Specifying command POINT
Point must lie within DTM bounds

*** ERROR *** Specifying command POINT
Command requires 2 x y coordinate values

*** ERROR *** Specifying command POINT
Latitude and longitude values supplied in wrong format

*** ERROR *** Specifying command POINT
Profile origin position has not been defined

*** ERROR *** Specifying command POINT
The sampling interval has not been defined

*** ERROR *** Specifying command POINT
No output IFF file has been specified

Examples:

PROFILE>**POINT 4500000 85000<CR>**
PROFILE>

POSITION

Defines the geographical position of the base station. This is the point from which all profiles are generated.

FORMAT: **POSITION x_coordinate y_coordinate**

Command parameters:

x_coordinate

The x coordinate position of the base station.

y_coordinate

The y coordinate position of the base station.

The format of the command parameters is controlled by the UNITS command.

UNITS MATRIX - 2 real values are required defining the position in terms of a DTM column and row number.

UNITS METRES Requires 2 real (floating point) values defining the position in metre values. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates (eg. 6 figure U.K. National Grid coordinates); if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

UNITS SECONDS - Requires 2 real (floating point) values defining the absolute point in seconds of arc. The values are supplied in the order latitude followed by longitude for the SW corner, and latitude followed by longitude for the NE corner. A negative latitude value indicates a position in the Southern Hemisphere, and a negative longitude value a point to the west of Greenwich.

UNITS LATLONG - Requires 2 values defining the absolute latitude and longitude position in the format DDD MM SS H where DDD is the number of degrees, MM is the number of minutes, SS is the number of seconds, and H is the hemisphere (N,S,E,W). The values are supplied in the order latitude followed by longitude for the SW corner, and latitude followed by longitude for the NE corner.

UNITS PROJECTION Requires 2 real (floating point) values defining the position in projection record units. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates; if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

DESCRIPTION:

The POSITION command is used to define the geographical position of the base station. This is the point from which all profiles are constructed. The base station must be located within the geographical bounds of the DTM.

It is not necessary for the base station to be positioned exactly on a DTM node. If located between nodes, the terrain height at the profile origin will be interpolated from the DTM.

The height of the terrain surface at the base station position is displayed on the terminal after the POSITION command is given.

If a digitising table is available, and a map has been registered to the DTM using the SETUP MAP command, puck **button 0** may be used to digitise the position of the base station.

The base station position must be defined before the GO command is issued.

Messages:

The following error messages are specific to the POSITION command:

```
*** ERROR *** Specifying command POSITION using puck button
No map set up has been performed

*** ERROR *** Specifying command POSITION
The input DTM is undefined

*** ERROR *** Specifying command POSITION
Point must lie within DTM bounds

*** ERROR *** Specifying command POSITION
Command requires 2 x y coordinate values

*** ERROR *** Specifying command POSITION
Latitude and longitude values supplied in wrong format
```

Examples:

```
PROFILE>POSITION 4500000 85000<CR>
Height of terrain surface at base station is 106.00
PROFILE>
```

PROFILE_INTERVAL

PROFILE_INTERVAL is used to define the angular separation of radial profiles.

FORMAT: **PROFILE_INTERVAL angle**

Command parameters:

angle

Angle is the angular separation of the radial profiles. Angle is specified in degrees as a real (floating point) number. A value in the range 0 to 180 is required.

DESCRIPTION:

The command PROFILE_INTERVAL defines the angular separation of profiles generated radially around the base station position.

In conjunction with the SECTOR command it determines the number of profiles that are generated. For example, if a 360 degree area of interest is defined, and the profile interval is 1 degree, 360 profiles will be generated in the IFF file.

A profile interval of 0 may be specified. This causes the program to construct a profile between every node in the DTM area of interest and the base station. On program startup, the profile interval will be set to 0.

The current profile interval value may be examined using the command SHOW PARAMETERS.

Messages:

The following error messages are specific to the PROFILE_INTERVAL command:

```
*** ERROR *** Specifying command PROFILE_INTERVAL
Command requires 1 real argument
```

```
*** ERROR *** Specifying command PROFILE_INTERVAL
Command requires a real argument in the range 0 to 180
```

Example:

PROFILE>**PROFILE_INTERVAL 15.0<CR>**
PROFILE>

RESET BAND

Cancels all previous BAND commands, causing the program to revert to the use of a constant sample interval specified by means of the SAMPLE_INTERVAL command.

FORMAT: RESET BAND

Command parameters: None.

DESCRIPTION:

Cancels all previous BAND commands, causing the program to revert to the use of a constant sample interval specified by means of the SAMPLE_INTERVAL command.

Messages:

The following message is specific to the RESET BAND command:

```
*** ERROR *** Specifying command RESET
Command qualifier is BAND
```

Examples:

```
PROFILE> RESET BAND
PROFILE>
```

RETURN

Restores command input from an indirect command file to SYS$COMMAND.

FORMAT: RETURN

Command parameters: None.

DESCRIPTION:

Restores command input from an indirect command file to SYS$COMMAND.

A typical application is to use an indirect command file to set up a number of run time defaults, and then return to input from the terminal for the run specific commands. To do this, RETURN must be the last command in the indirect command file.

A RETURN command is only valid if present in a command file.

Messages:

The following messages are specific to the RETURN command:

```
*** ERROR *** Specifying command RETURN
The RETURN command is only valid in a command file

Returning to terminal input
```

Examples:

```
PROFILE> @DEFAULT_BANDS<CR>
PROFILE> BAND 1 1000.0 50.0
PROFILE> BAND 2 2000.0 100.0
PROFILE> RETURN
Returning to terminal input
PROFILE>
```

SAMPLE_INTERVAL

SAMPLE_INTERVAL is used to define a constant DTM sampling interval.

FORMAT: **SAMPLE_INTERVAL sample_interval**

Command parameters:

sample_interval

Sample_interval is the rate at which the DTM is sampled when generating a profile. The sample interval is specified in metres as a positive real (floating point) number.

DESCRIPTION:

The command SAMPLE_INTERVAL defines the rate at which the DTM is sampled when generating a profile. It therefore defines the separation of successive profile sample point coordinates.

The sample interval specified with this command is used for all profiles, and is constant along the profile irrespective of the distance of the sample point from the profile origin (base station). If a sample rate that varies with distance from the profile origin (base station) is required, one or more BAND commands should be used.

A sample interval or a series of sample intervals must be specified using the SAMPLE_INTERVAL or BAND commands, before the command GO is given to generate the terrain profiles.

Messages:

The following error messages are specific to the SAMPLE_INTERVAL command:

*** ERROR *** Specifying command SAMPLE__INTERVAL
Command requires 1 real argument

*** ERROR *** Specifying command SAMPLE__INTERVAL
Command requires a positive value

Example:

PROFILE>**SAMPLE_INTERVAL 10.5<CR>**
PROFILE>

SECTOR

SECTOR is used to restrict profile calculation to a sector of interest in the DTM. The sector is defined in terms of a sector radius, and 2 sector bearings.

FORMAT: **SECTOR radius [bearing_1] [bearing_2]**

Command parameters:

radius

Radius is the distance from the base station (sector origin) to the sector arc. A positive real (floating point) value is required.

bearing_1

Bearing_1 is the bearing of the first sector radius. A real (floating point) value in the range 0 to 360 degrees is required.

bearing_2

Bearing_2 is the bearing of the first sector radius. A real (floating point) value in the range 0 to 360 degrees is required.

DESCRIPTION:

SECTOR is used to define a sector of interest in the DTM. Profiles are only constructed between the profile origin (base station) and DTM nodes that lie within the sector of interest.

The sector of interest is defined in terms of a sector radius, and optionally 2 bearing values defining the angular limits of the sector.

If only a sector radius is supplied, a full circular scan around the base station will be performed, and the bearing values will be set to 0 and 360 degrees.

The order in which the bearing values are specified determines whether the area of interest is inclusive or exclusive to the angular limits. A clockwise convention is used. For example if the first bearing is defined as 40 degrees and the second bearing as 80 degrees, the angular width of the sector of interest will be 40 degrees rather than 320 degrees. If the values are supplied in the order 80 and 40 degrees, the angular width of the sector of interest will be 320 degrees.

Note that a sector of interest will be clipped to any rectangular area of interest that may have been defined.

Messages:

The following error messages are specific to the SECTOR command:

*** ERROR *** Specifying command SECTOR
Command requires either 1 or 3 real arguments

*** ERROR *** Specifying command SECTOR
Sector distance must be specified as a positive real number

*** ERROR *** Specifying command SECTOR
Sector bearing values must be in the range 0 to 360

Example:

PROFILE>**POSITION 100.5 450.3<CR>**
PROFILE>**SECTOR 1000.0 40.5 90.0<CR>**
PROFILE>

SET AC_BEARING

Specifies the AC type that will hold the profile bearing value in the output IFF file.

FORMAT: **SET AC_BEARING ac_type**

Command parameters:

ac_type

The AC type of the Ancillary Code entry that will hold the profile bearing value. An integer value in the range 1 to 32767 is required.

DESCRIPTION:

The SET AC_BEARING command is used to specify the AC type value of the AC entry in the output IFF file that will hold the bearing of the profile.

By default an AC type 1001 will be used.

Use of the SHOW SETTINGS command is recommended to check on the current setting.

Messages:

The following error messages are specific to the SET and SET AC_BEARING commands:

```
*** ERROR *** Specifying command SET
Command qualifiers are AC_BEARING, AC_CLUTTER, AC_LENGTH, FC, LAYER or SCALE

*** ERROR *** Specifying command SET AC_BEARING
Command requires an integer argument

*** ERROR *** Specifying command SET AC_BEARING
Command requires a feature code in the range 1 to 32767
```

Examples:

PROFILE>**SET AC_BEARING 2005 <CR>**
PROFILE>

SET AC_CLUTTER

Specifies the attribute code that will be used to used to identify the clutter index in an IFF Coordinate Block (CB) entry.

FORMAT: SET AC_CLUTTER ac_type

Command parameters:

ac_type

The attribute code of the column in the CB entry that holds the clutter index. An integer value in the range 1 to 32767 is required.

DESCRIPTION:

Specifies the attribute code that will be used to used to identify the clutter index in an IFF Coordinate Block (CB) entry.

By default a value of 1003 is used.

Use of the SHOW SETTINGS command is recommended to check on the current setting.

Messages:

The following error messages are specific to the SET and SET AC_CLUTTER commands:

```
*** ERROR *** Specifying command SET
Command qualifiers are AC_BEARING, AC_CLUTTER, AC_LENGTH, FC, LAYER or SCALE

*** ERROR *** Specifying command SET AC_CLUTTER
Command requires an integer argument

*** ERROR *** Specifying command SET AC_CLUTTER
Command requires a feature code in the range 1 to 32767
```

Examples:

PROFILE>**SET AC_CLUTTER 2003 <CR>**
PROFILE>

SET AC_LENGTH

Specifies the AC type that will hold the profile length value in the output IFF file.

FORMAT: **SET AC_LENGTH ac_type**

Command parameters:

ac_type

The AC type of the the Ancillary Code entry that will hold the profile length value. An integer value in the range 1 to 32767 is required.

DESCRIPTION:

The SET AC_LENGTH command is used to specify the AC type value of the AC entry in the output IFF file that will hold the length of the profile.

By default an AC type 1002 will be used.

Use of the SHOW SETTINGS command is recommended to check on the current setting.

Messages:

The following error messages are specific to the SET and SET AC_LENGTH commands:

*** ERROR *** Specifying command SET
Command qualifiers are AC_BEARING, AC_CLUTTER, AC_LENGTH, FC, LAYER or SCALE

*** ERROR *** Specifying command SET AC_LENGTH
Command requires an integer argument

*** ERROR *** Specifying command SET AC_LENGTH
Command requires a feature code in the range 1 to 32767

Examples:

PROFILE>**SET AC_LENGTH 2004 <CR>**
PROFILE>

SET FC

Specifies the feature code that will be given to all features in the output IFF file.

FORMAT: SET FC feature_code

Command parameters:

feature_code

The feature code value. An integer number in the range 1 to 32767 is required.

DESCRIPTION:

The SET FC command is used to specify the feature code value that is given to all features in the output IFF file.

By default all features are given a feature code value of 1.

Use of the SHOW SETTINGS command is recommended to check on the current setting.

Messages:

The following error messages are specific to the SET and SET FC commands:

```
*** ERROR *** Specifying command SET
Command qualifiers are AC_BEARING, AC_CLUTTER, AC_LENGTH, FC, LAYER or SCALE

*** ERROR *** Specifying command SET FC
Command requires 1 integer argument

*** ERROR *** Specifying command SET FC
Command requires a feature code in the range 1 to 32767
```

Examples:

PROFILE>**SET FC 4 <CR>**
PROFILE>

SET LAYER

Specifies the layer into which features are placed in the output IFF file.

FORMAT: **SET LAYER layer**

Command parameters:

layer

The IFF layer number. An integer value in the range 1 to 32767 is required.

DESCRIPTION:

The SET LAYER command is used to specify the layer into which features are placed in the output IFF file.

By default features are written to layer 1 in the IFF file.

Use of the SHOW SETTINGS command is recommended to check on the current setting.

Messages:

The following error messages are specific to the SET and SET LAYER commands:

```
*** ERROR *** Specifying command SET
Command qualifiers are AC_BEARING, AC_CLUTTER, AC_LENGTH, FC, LAYER or SCALE

*** ERROR *** Specifying command SET LAYER
Command requires 1 integer argument

*** ERROR *** Specifying command SET LAYER
Command requires a feature code in the range 1 to 32767
```

Examples:

PROFILE>**SET LAYER 4 <CR>**
PROFILE>

SET SCALE

Sets the scale in the output IFF file MD (Map Descriptor) entry.

FORMAT: SET SCALE scale

Command parameters:

scale

The scale denominator to be set in the output IFF file MD (Map Descriptor) entry. A real (floating point) value is required eg. 10000.0

DESCRIPTION:

The SET SCALE command is used to specify the scale of the output IFF file. The scale value is stored in the MD (Map Descriptor) entry of the IFF file. The command is required since a scale value is not available from the DTI Projection Record.

By default a scale value of 50000 (ie. 1:50000) is written to the output IFF file.

Use of the SHOW SETTINGS command is recommended to check on the current setting.

Messages:

The following error messages are specific to the SET and SET SCALE commands:

```
*** ERROR *** Specifying command SET
Command qualifiers are AC_BEARING, AC_CLUTTER, AC_LENGTH, FC, LAYER or SCALE
```

```
*** ERROR *** Specifying command SET SCALE
Command requires a real argument
```

Examples:

PROFILE>**SET SCALE 10000 <CR>**
PROFILE>

SETUP MAP

Registers a map or other source document placed on a digitising table, to the input DTM.

FORMAT: **SETUP MAP**

DESCRIPTION:

The SETUP MAP command allows a source document to be registered to the input DTM.

In order to register the map and a DTM, the DTM should have first been specified, and a source document should have been securely attached to the surface of a digitising table.

On giving the command you will be asked to digitise 4 rectangular registration points. The registration points represent 4 points on the map that correspond to the 4 corners of the DTM. The points are digitised in the order top left (NW), bottom left (SW), bottom right (SE) and top right (NE) using any button on the table puck. A point should be digitised in response to a prompt on the terminal.
An error message is generated if any of the angles of the digitised rectangle are less than 88 degrees, or greater than 92 degrees (ie. if the corner points of the rectangle are more than 2 degrees off rectangular). In this case you will be asked to redigitise the 4 registration points.
Setup of the map may be aborted using <CTRL/Z> (pressing the Ctrl and Z keys together), or by pressing **Button F** on the table puck.

Following the registration of a map to the DTM, coordinate values required by the POINT, POSITION, SECTOR and WINDOW commands, may be input using the table puck.

Messages:

The following error messages are specific to the SETUP and SETUP MAP commands:

*** ERROR *** Specifying command SETUP
Command qualifier is MAP

*** ERROR *** Specifying command SETUP MAP
Command is invalid if the table has not been initialised

*** ERROR *** Specifying command SETUP MAP
The input DTM must be specified before SETUP MAP

*** ERROR *** Setting up MAP
Badly digitised corner points; try again

Examples:

PROFILE>**SETUP MAP<CR>**

Digitise map NW corner>

Digitise map SW corner>

Digitise map SE corner>

Digitise map NE corner>

PROFILE>

SHOW CLUTTER

Displays the file specification of all selected clutter grid files.

FORMAT: **SHOW CLUTTER**

Command parameters: None.

DESCRIPTION:

SHOW CLUTTER displays the file specification of all currently defined clutter grid files. These are the names of the DTI files selected using the BUILDING, VEGETATION and WATER commands.

If a clutter file has not been specified, the file will be shown as undefined.

Messages:

The following message is specific to the SHOW command:

```
*** ERROR *** Specifying command SHOW
Command qualifiers are CLUTTER, ENABLE, FILEIN, PARAMETERS or SETTINGS
```

Examples:

PROFILE>**SHOW CLUTTER<CR>**

```
Current status:

BUILDING   File      : Undefined
VEGETATION File      : LSL$DTI:VEGETATION_TEST.DTI;1
WATER      File      : Undefined

PROFILE>
```

SHOW ENABLE

Shows the current status of those options that may be enabled by means of the ENABLE command, or disabled using the DISABLE command.

FORMAT: **SHOW ENABLE**

Command parameters: None.

DESCRIPTION:

Displays the current status of all the PROFILE options that may be enabled or disabled using the ENABLE and DISABLE commands.
The name of the option is shown, followed by either the word ON or OFF to indicate its current status.
If the command SHOW ENABLE is used before any ENABLE or DISABLE commands have been given, the default status of the options is displayed.

Messages:

The following message is specific to the SHOW command:

*** ERROR *** Specifying command SHOW
Command qualifiers are CLUTTER, ENABLE, FILEIN, PARAMETERS or SETTINGS

Examples:

PROFILE>**SHOW ENABLE<CR>**

Current status:

```
ABSOLUTE          On        DIAGNOSTICS      On          TABLE          Off


PROFILE>
```

SHOW FILEIN

Displays information extracted from the header of the input DTM.

FORMAT: **SHOW FILEIN**

Command parameters: None.

DESCRIPTION:

Details extracted from the header of the DTM are displayed on the terminal, along with details of the current window (rectangular area of interest).

The header values are shown in the current units of measurement. This is dependent on the header type of the input file, or may be set explicitly using the UNITS command.

Messages:

The following messages are specific to the commands SHOW and SHOW FILEIN:

```
*** ERROR *** Specifying command SHOW
Command qualifiers are CLUTTER, ENABLE, FILEIN, PARAMETERS or SETTINGS
```

```
*** WARNING *** The input DTM is undefined
```

Examples:

PROFILE>**SHOW FILEIN<CR>**

```
LSL$DTI:TEST.DTI
Header: MIKE     Data: WORD

Units are DTI Matrix Values

Matrix coverage    SW:    1     1    NE:   301   251
Matrix Window      SW:    1     1    NE:   200   200
Matrix Interval     E:    1           N:     1
Value Range          :      0   to    851

PROFILE>
```

SHOW PARAMETERS

Displays information on the profile origin (base station position), sample and profile intervals, sample bands and sector of interest.

FORMAT: **SHOW PARAMETERS**

Command parameters: None.

DESCRIPTION:

The SHOW PARAMETERS command is used to display information about the profile origin (base station), sample and profile intervals, sample bands and sector of interest.

Messages:

The following message is specific to the SHOW command:

```
*** ERROR *** Specifying command SHOW
Command qualifiers are CLUTTER, ENABLE, FILEIN, PARAMETERS or SETTINGS
```

Examples:

PROFILE>**SHOW PARAMETERS<CR>**

```
Base Station       :
 X Y Z (Metres)    : 443500.00  82550.00   43.00
       (Matrix)    :     69.00     50.00   43.00

Sector of Interest :

  RADIUS    BEARING 1    BEARING 2      WIDTH
  6000.00      0.00       360.00       360.00

Profile Interval   :    0.00 degrees (i.e. use all DTM nodes)

Sampling Interval  :   Variable (see Band Information)

Sample Bands       :   (in metres)

BAND       WIDTH      OUTER DISTANCE      SAMPLE INTERVAL
  1       3000.00        3000.00             200.00
  2       1000.00        4000.00             100.00
  3       1000.00        5000.00              50.00

PROFILE>
```

SHOW SETTINGS

Shows the current value of those parameters that may be set using the SET command.

FORMAT: **SHOW SETTINGS**

Command parameters: None.

DESCRIPTION:

Displays the name of all PROFILE parameters that may be set using the SET command, and their current values.
If the SHOW SETTINGS command is given before using a SET command, the values shown are the default values that are allocated to the parameters by PROFILE.

Messages:

The following message is specific to the SHOW command:

*** ERROR *** Specifying command SHOW
Command qualifiers are CLUTTER, ENABLE, FILEIN, PARAMETERS or SETTINGS

Examples:

PROFILE>**SHOW SETTINGS<CR>**

```
Current values:

AC_BEARING       1001       AC_CLUTTER       1003       AC_LENGTH       1002

FC                  1       LAYER               1       SCALE        50000.0

PROFILE>
```

SUB_SAMPLE

Defines the interval at which nodes will be sampled along the columns and rows of the DTM, when using PROFILE with a profile interval of 0.

FORMAT: SUB_SAMPLE column_interval row_interval

Command parameters:

column_interval

The sample interval along the columns of the DTM. A positive integer value is required.

row_interval

The sample interval along the rows of the DTM. A positive integer value is required.

DESCRIPTION:

SUB_SAMPLE is used to define the interval at which PROFILE samples the columns and rows of the DTM with a profile interval of 0.

By default with a profile interval of 0, PROFILE constructs a path profile between the base station and each DTM node that lies within the rectangular or sector areas of interest using SUB_SAMPLE values of 1 and 1. If SUB_SAMPLE values of 2 and 2 are used, then only profiles between the base station and every second node along a DTM column and row, are constructed.

Note that if the number of columns and rows in the rectangular area of interest is not exactly divisible by the sub-sample values, profiles between the last column or row nodes are generated.

The SUB_SAMPLE command is used in situations where it is necessary to reduce the total number of profiles generated for subsequent analysis. The current sub-sampling intervals may be examined using the SHOW PARAMETERS command.

Message:

The following error messages are specific to the SUB_SAMPLE command:

*** ERROR *** Specifying command SUB_SAMPLE
Command requires 2 integer arguments

*** ERROR *** Specifying command SUB_SAMPLE
Command requires 2 positive integer arguments

Examples:

```
PROFILE>SUB_SAMPLE 2 2 <CR>
PROFILE>
```

SPAWN

The SPAWN command enables a subprocess to be created from within PROFILE.

FORMAT: SPAWN command-line

Command parameters:

command-line

Specifies a DCL command string to be executed as if typed in response to a '$' prompt. The command line cannot exceed 80 characters.

DESCRIPTION:

The SPAWN command enables you to create a subprocess while within PROFILE. When the subprocess terminates, control is returned to PROFILE.

The command is generally used for file management purposes.

Messages:

The following error messages are specific to the SPAWN command:

```
*** ERROR *** Specifying command SPAWN
Command requires a valid DCL command line

*** ERROR *** Unable to spawn command, returning to PROFILE
```

Examples:

PROFILE> **SPAWN DIRECTORY LSL$DTI:*.DTI<CR>**

```
Directory DUA3:[LSL.DTI]

TEST1.DTI;1             8/8       18-AUG-1987 07:56  [LSL,DAVEC]
TEST2.DTI;1             7/8       18-AUG-1987 17:17  [LSL,DAVEC]
TEST2.DTI;2             7/8       18-AUG-1987 17:34  [LSL,DAVEC]

Total of 3 files, 22/24 blocks.

PROFILE>
```

UNITS

Specifies the units of measurement that will be used when defining an area of interest in the input DTM, defining the position of a base station, or a profile end point. The command also controls the units of measurement which will be used when displaying file header details.

FORMAT: UNITS units

Command parameters:

units

A keyword defining the measurement units, chosen from:

MATRIX	Matrix grid interval units, i.e rows and columns
METRES	Metre values
SECONDS	Latitude and Longitude in seconds of arc
LATLONG	Latitude and Longitude in degrees, minutes and seconds
PROJECTION	Projection Record Units (eg. mms on the source document)

DESCRIPTION:

The UNITS command defines in which units of measurement arguments to the POSITION and POINT commands are specified, and the units of measurement used when defining an area of interest in the input DTM by means of the WINDOW command.
The command also controls in what format details from the header of the DTM are displayed when the SHOW FILEIN command is given.
The command also controls in what format details of the current parameters are displayed when the SHOW PARAMETERS command is given.

The command should be given after defining the input DTM since an appropriate default units of measurement is set up when the file is opened. If the DTM file is of header type TED4 or UHL1 (ie DTED files), then the default is latitude and longitude specified in degrees, minutes and seconds; for all other header types metre units are assumed.

Messages:

The following error messages are specific to the UNITS command:

*** ERROR *** Specifying command UNITS
Command qualifiers are MATRIX, METRES, PROJECTION, SECONDS or LATLONG

*** ERROR *** Specifying command UNITS
Command qualifiers LATLONG or SECONDS are only valid for DTED input DTI files

---

Examples:

```
PROFILE> UNITS MATRIX<CR>
PROFILE>
```

VEGETATION

Selects and opens a DTI file that contains clutter information relating to vegetation.

FORMAT: VEGETATION file-spec

Command parameters:

file-spec

The file specification for the input grid file containing clutter information relating to vegetation.
Any part of the file specification not supplied is taken from the default 'LSL$DTI:DTI.DTI', although if no file-spec is supplied, you will be asked to supply one in response to the prompt **Input DTI filename>**

DESCRIPTION:

This command opens and maps into memory a DTI file, containing height information relating to vegetation. Details derived from the header of the file are displayed on the terminal to confirm that the file has been successfully opened.

A clutter file may only be specified after a DTM has been selected using the FILEIN command.

If the clutter file contains a DTI Projection Record, the position of the SW corner of the grid may be determined. This enables the clutter data to be correctly positioned with respect to the DTM. If the origin is unset or the file contains no projection record, the SW corner of the clutter file is assumed to register to the SW corner of the DTM.

Messages:

The following error messages are specific to the VEGETATION command:

*** ERROR *** Specifying command VEGETATION
Input DTI filename is missing

*** ERROR *** Specifying command VEGETATION
Input DTM must be specified first

Example:

```
PROFILE>VEGETATION TEST_VEGETATION<CR>

LSL$DTI:TEST_VEGETATION.DTI
Header: MIKE     Data: WORD

Units are Metres

Matrix coverage    SW:    1     1    NE:   301   251
Matrix Window      SW:    1     1    NE:   301   251
Matrix Interval     E:    1           N:     1
Value Range          :      0   to   10

PROFILE>
```

WAIT

The WAIT command causes processing to be suspended for a specified number of seconds.

FORMAT: WAIT seconds

Command parameters:

seconds

The number of seconds for which processing is to be suspended. A real (floating point) value is required.

DESCRIPTION:

The WAIT command causes processing to be suspended for a specified number of seconds. It is designed for use in software demonstration situations and is of no value in a production flowline.

Messages:

The following error message is specific to the WAIT command:

*** ERROR *** Specifying command WAIT
Command requires a real argument

Examples:

PROFILE> **WAIT 4.0<CR>**
PROFILE>

WATER

Selects and opens a DTI file that contains clutter information relating to water.

FORMAT: **WATER file-spec**

Command parameters:

file-spec

The file specification for the input grid file containing clutter information relating to water.
Any part of the file specification not supplied is taken from the default 'LSL$DTI:DTI.DTI', although if no file-spec is supplied, you will be asked to supply one in response to the prompt **Input DTI filename>**

DESCRIPTION:

This command opens and maps into memory a DTI file, containing information relating to water. Details derived from the header of the file are displayed on the terminal to confirm that the file has been successfully opened.

A clutter file may only be specified, after a DTM has been selected using the FILEIN command.

If the clutter file contains a DTI Projection Record, the position of the SW corner of the grid may be determined. This enables the clutter data to be correctly positioned with respect to the DTM. If the origin is unset or the file contains no projection record, the SW corner of the clutter file is assumed to register to the SW corner of the DTM.

Messages:

The following error messages are specific to the WATER command:

```
*** ERROR *** Specifying command WATER
Input DTI filename is missing

*** ERROR *** Specifying command WATER
Input DTM must be specified first
```

Example:

```
PROFILE>WATER TEST_WATER<CR>

LSL$DTI:TEST_WATER.DTI
Header: MIKE     Data: WORD

Units are Metres

Matrix coverage    SW:    1     1    NE:   301   251
Matrix Window      SW:    1     1    NE:   301   251
Matrix Interval     E:    1           N:     1
Value Range          :      -1   to    0

PROFILE>
```

WINDOW

Specifies a rectangular area of interest in the input DTM.

FORMAT: WINDOW xmin ymin xmax ymax

Command parameters:

xmin ymin
The coordinates of the bottom left hand corner of the defining rectangle.

xmax ymax
The coordinates of the top right hand corner of the defining rectangle.

The format of the command parameters is controlled by the UNITS command.

UNITS MATRIX - Requires 4 integer values defining the rectangle in terms of column and row numbers

UNITS METRES Requires 4 real (floating point) values defining the rectangle in metre values. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates (eg. 6 figure U.K. National Grid coordinates); if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

UNITS SECONDS - Requires 4 real (floating point) values defining the absolute position of the rectangle in seconds of arc. The values are supplied in the order latitude followed by longitude for the SW corner, and latitude followed by longitude for the NE corner. A negative latitude value indicates a position in the Southern Hemisphere, and a negative longitude value a point to the west of Greenwich.

UNITS LATLONG - Requires 4 values defining the absolute latitude and longitude position of the rectangle in the format DDD MM SS H where DDD is the number of degrees, MM is the number of minutes, SS is the number of seconds, and H is the hemisphere (N,S,E,W). The values are supplied in the order latitude followed by longitude for the SW corner, and latitude followed by longitude for the NE corner.

UNITS PROJECTION Requires 4 real (floating point) values defining the rectangle in projection record units. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates; if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

Note that in all cases, the input values are adjusted to the nearest column and row values.

DESCRIPTION:

The WINDOW command is used to limit profile calculation to a rectangular geographical area. Only profiles between the base station and DTM nodes that lie within the rectangle are generated when the GO command is given. The rectangular window is ignored when the POINT command is used.
The area of interest should lie within the geographical bounds of the DTM.

If a digitising table is available, puck **Button 1** may be used to give the command WINDOW. When the button is pressed, you will be asked to digitise 2 points inside the map area, defining the SW (bottom lefthand) and NE (top righthand) corners of the rectangle. Definition of the area of interest may be abandoned by pressing puck **Button F**.

Note, that when the DTM is first opened, a default area of interest is set which corresponds to the whole of the DTM.

Messages:

The following messages are specific to the WINDOW command:

```
*** ERROR *** Specifying command WINDOW
The input DTM must be specified before WINDOW

*** ERROR *** Specifying command WINDOW using puck button
No map set up has been performed

*** ERROR *** Specifying command WINDOW
Command requires 4 arguments

*** ERROR *** Specifying command WINDOW
NE corner values must exceed SW corner values

*** ERROR *** Specifying command WINDOW
Supplied values exceed matrix extents

*** ERROR *** Specifying command WINDOW
Latitude and longitude values supplied in wrong format
```

Examples:

PROFILE>**WINDOW 1 1 200 200<CR>**
PROFILE>**UNITS LATLONG<CR>**
PROFILE>**WINDOW 42 00 00N 3 00 00E 42 30 00N 2 58 40E**
PROFILE>

PROFILE examples

EXAMPLES

```
$ PROFILE

*** WARNING ***  Unable to initialise the table monitor
PROFILE will assume no table is available
PROFILE>FILEIN OS44000800<CR>

File    :  LSL$DTI:OS44000800.DTI
Header  :  ALVY  Data:  WORD

Units are Metres

Matrix Coverage   SW:  440000.00   80000.00    NE:  460000.00  100000.00
Matrix Window     SW:  440000.00   80000.00    NE:  460000.00  100000.00
Matrix Interval    E:      50.00                N:      50.00
Value Range         :     0   to    235

PROFILE>WINDOW 440000 80000 450000 90000<CR>

File    :  LSL$DTI:os44000800.DTI
Header  :  ALVY  Data:  WORD

Units are Metres

Matrix Coverage   SW:  440000.00   80000.00    NE:  460000.00  100000.00
Matrix Window     SW:  440000.00   80000.00    NE:  450000.00   90000.00
Matrix Interval    E:      50.00                N:      50.00
Value Range         :     0   to    235

PROFILE>POSITION 445000 85000<CR>
Height of terrain surface at base station is   145.00

PROFILE>SECTOR 6000<CR>
PROFILE>PROFILE_INTERVAL 3<CR>
PROFILE>SAMPLE_INTERVAL 100<CR>
PROFILE>SHOW PARAMETERS<CR>

Base Station       :
 X Y Z (Metres)    : 445000.00  85000.00  145.00
       (Matrix)    :     99.00     99.00  145.00

Sector of Interest :

  RADIUS    BEARING 1    BEARING 2      WIDTH
  6000.00      0.00       360.00       360.00

Profile Interval   :    3.00 degrees

Sampling Interval  :  100.00

Sample Bands       :      Undefined
```

```
PROFILE>IFF DAVEC<CR>
PROFILE>GO<CR>
%LSLLIB-I-IFFOPENED, LSL$DATA_ROOT:[LSL.IFF]DAVEC.IFF;1 opened for write

+------------------------------------------------------------------+
|                                                                  |
|              Generating the terrain profiles                     |
|                                                                  |
+------------------------------------------------------------------+

Total of 121 profiles written to IFF file
PROFILE>EXIT<CR>
```

In this example an area of interest in the DTM is defined using the WINDOW command. The base station (profile origin) is positioned in the centre of the area of interest. The SECTOR command has been used to define the radius of a circle surrounding the base station. The PROFILE_INTERVAL command is used to define the angular separation of the radial profiles, while the SAMPLE_INTERVAL command has been used to define a constant rate of sampling along the profiles. Output is to the file 'LSL$IF:DAVEC.IFF'. Diagnostics are enabled by default.

```
$PROFILE

WARNING ***  Unable to initialise the table monitor
PROFILE will assume no table is available
PROFILE>FILE OS44000800<CR>

File    :  LSL$DTI:OS44000800.DTI
Header  :  ALVY  Data:  WORD

Units are Metres

Matrix Coverage   SW:  440000.00   80000.00    NE:  460000.00  100000.00
Matrix Window     SW:  440000.00   80000.00    NE:  460000.00  100000.00
Matrix Interval    E:      50.00                N:      50.00
Value Range         :      0   to    235

PROFILE>WINDOW 440000 80000 445000 85000<CR>

File    :  LSL$DTI:OS44000800.DTI
Header  :  ALVY  Data:  WORD

Units are Metres

Matrix Coverage   SW:  440000.00   80000.00    NE:  460000.00  100000.00
Matrix Window     SW:  440000.00   80000.00    NE:  445000.00   85000.00
Matrix Interval    E:      50.00                N:      50.00
Value Range         :      0   to    235

PROFILE>POSITION 443500 82550<CR>
Height of terrain surface at base station is    43.00

PROFILE>SECTOR 6000<CR>
PROFILE>BAND 1 3000 200<CR>
PROFILE>BAND 2 1000 100<CR>
PROFILE>BAND 3 1000 50<CR>
PROFILE>IFF DAVEC<CR>
PROFILE>SHOW PARAMETERS<CR>

Base Station       :
 X Y Z (Metres)    : 443500.00  82550.00   43.00
       (Matrix)    :     69.00     50.00   43.00

Sector of Interest :

  RADIUS    BEARING 1    BEARING 2      WIDTH
  6000.00      0.00       360.00       360.00

Profile Interval   :    0.00 degrees (i.e. use all DTM nodes)

Sampling Interval  :   Variable (see Band Information)

Sample Bands       :   (in metres)

BAND       WIDTH      OUTER DISTANCE      SAMPLE INTERVAL
  1       3000.00        3000.00              200.00
  2       1000.00        4000.00              100.00
```

```
  3       1000.00        5000.00              50.00

PROFILE>GO<CR>
%LSLLIB-I-IFFOPENED, LSL$DATA_ROOT:[LSL.IFF]DAVEC.IFF;3 opened for write

Total distance of BANDS less than sector distance
Width of BAND 3 extended to   2000.00

+-----------------------------------------------------------------+
|                                                                 |
|              Generating the terrain profiles                    |
|                                                                 |
+-----------------------------------------------------------------+

Total of 10200 profiles written to IFF file

PROFILE>EXIT<CR>
```

In this example a profile is constructed between the base station and each DTM node in the sector of interest. A series of BAND commands have been given to define a sample interval that varies with distance along the profile. Since the total width of the bands, is less than the sector distance, the outer band has been extended.

```
ELAPSED:    0 00:47:55.38  CPU: 0:09:00.06  BUFIO: 284  DIRIO: 7160  FAULTS: 527
```

MESSAGES (OTHER)

In addition to the above messages which are generated by the program itself, other messages may be produced by by Laser-Scan libraries. In particular, messages may be generated by the IFF and DTI libraries or by the Laser-Scan I/O library, LSLLIB. IFF library messages are introduced by '%IFF' and are documented in the IFF library users' guide. DTI library messages are introduced by %DTI and are documented in the MATRIX Reference manual. LSLLIB messages are introduced by '%LSLLIB' and are generally self-explanatory. They are often used to explain the details of program generated errors.

If the cause of the error cannot be traced by the user, and Laser-Scan are consulted, then the output file should be preserved to facilitate diagnosis.

CHAPTER 7

MODULE SLOPES

MODULE SLOPES

Slope and Aspect Calculation

Issue 7.0   13-March-1989

Science Park, Milton Road, Cambridge, England CB4 4FY tel: (0223) 420414

Document "SLOPES Reference"   Category "Reference"

Document	Issue 7.0	M W S Reid	13-March-1989
	Issue 6.0	D R Catlow	08-September-1988
	Issue 5.0	D R Catlow	25-January -1988
	Issue 4.0	D R Catlow	17-October -1986
	Issue 3.0	D R Catlow	24-February-1986

MODULE   SLOPES

FUNCTION

SLOPES is a program to produce a slope, aspect, height difference or shaded relief map from a Digital Terrain Model (DTM). The various maps may be output either to a colour graphics display, or to a Laser Scan DTI (Digital Terrain Image) disk file.

In the slope and height difference options, the user may specify the value step interval and the value and height range within which the information is displayed. In the shaded image option, the user has control over the sun position and sun angle. For aspect maps the user may vary the size of the angular step, and may display the information either as a grey scale or colour image.

FORMAT

```
$ SLOPES
```

PROMPTS

SLOPES is an interactive, command driven program. Command input is expected when the following prompt is issued:

```
SLOPES>
```

Commands may be issued at the terminal in response to the prompt, or in most cases by digitising a point inside a SLOPES menu box.

DESCRIPTION

SLOPES is a program to produce a slope, aspect, height difference or shaded relief map from a Digital Terrain Model (DTM). The various images may be output either to a colour graphics display, or to a DTI disk file.

SLOPE AND ASPECT CALCULATION

The two components of the local gradient are slope and aspect (the direction of slope). Both derivatives may be calculated and displayed using SLOPES.

The user may select from 5 different slope algorithms. 4 algorithms provide a measure of average slope based on the slope of adjacent parts of the terrain, while the fifth algorithm provides a measure of maximum slope. The algorithm is selected using the integer parameter associated with the **SLOPE** command.

The default slope algorithm is number 1. This algorithm employs a 3 X 3 operator that is passed over the matrix column by column and row by row. The operator calculates the slope at the centre point of the matrix. This is done by calculating the change in slope in the x and y direction for the surrounding matrix points, and combining the x and y components to provide an average measure of slope.

```
Z-+    Zo+    Z++

Z-o    Zoo    Z+o

z--    Zo-    Z+-
```

The algorithm used is:

P = [(Z++ + 2Z+o + Z+-) - (Z-+ + 2Z-o + Z--)] / 8dx

Q = [(Z++ + 2Zo+ + Z-+) - (Z+- + 2Zo- + Z--)] / 8dy

Where:

P	slope in the x direction (west to east)
Q	slope in the y direction (south to north)
dx	grid interval in the x direction
dy	grid interval in the y direction

The remaining algorithms are based on the division of the terrain matrix into a number of regular triangular facets, and the calculation of the surface normal vector for each facet. The slope of a facet is the angle between the surface normal and the vertical.

Algorithm 2 calculates an average slope value for a grid cell using the elevation values for the matrix points at the corner of the cell.

Z-+	Z++
Z--	Z+-

The cell is divided into 2 triangular facets defined by the nodes Z--, Z+-, Z++ and Z++, Z-+, Z--. The average slope value for the cell is produced by meaning the slope for the 2 facets.

Algorithm 3 uses the same technique but attempts to remove the directional bias inherent in the previous algorithm by dividing the cell into 4 triangular facets. The 4 facets are defined by the nodes: Z--, Z++, Z-+ ; Z--, Z+-, Z++ ; Z-+, Z--, Z+- ; Z-+, Z-+, Z++. The average slope value for the cell is produced by meaning the slope for the 4 facets.

Algorithm 4 is based on a 3 X 3 matrix and involves the calculation of 4 surface normal vectors.

Z-+	Zo+	Z++
Z-o	Zoo	Z+o
z--	Zo-	Z+-

An average slope value for the matrix node Zoo is derived by meaning the slope values for the 4 triangles: Zoo, Zo+, Z-o ; Zoo, Z+o, Zo+ ; Zoo, Zo-, Z+- ; Zoo, Z-o, Z--.

Algorithm 5 is identical to the previous algorithm, except that the maximum of the 4 facet slopes is output rather than the mean value.

The slope values are calculated by default as degrees of slope, however using the command **ENABLE PERCENT** it is possible to output the derived slope value as a percentage slope. In the former case the slope values will be in the range 0 to 90 degrees; in the second case the slope values will be output in the range 0 to 100%.

SHADE CALCULATION

A greyscale shaded overlay or relief map is an effective way of visualising elevation data, enabling the viewer to readily perceive the form and trend of surface relief.

The shaded overlay is produced by modelling the terrain surface as though lit by a light source. The grey levels in the shaded overlay reflect the apparent brightness of a surface element. This in turn is dependent upon its orientation with respect to the light source. A surface normal to the light source will appear brighter than a surface facing away from the light source. The orientation of a surface is a product of the local gradient (ie. its direction and slope).

It should be noted that the apparent brightness is a function only of the local gradient. No account is taken of effects dependent on the position of the surface element with respect to other elements of the DTM eg. mutual illumination or shadowing. Taking into account such factors can detract from the visual quality of the shaded image, and make interpretation more difficult.

The appearance of the final shaded image is dependent on the position of the light source. The position of the light source is defined in terms of an elevation angle (zenith angle), and an azimuth angle. The two angles are defined using the commands **SUN_ANGLE** and **SUN_POSITION**. The azimuth angle is measured clockwise from north.

By default a zenith angle of 45 degrees, and an azimuth angle of 315 degrees (north-west) are assumed. This is the light source position that is most frequently used when hill shading is manually generated by a cartographer.

Incorporated within SLOPES are 3 reflectance maps, which describe how a surface reflects and absorbs light. The different reflectance maps may be selected using the **REFLECTANCE_MAP** command. The default reflectance map (number 2) uses a Lambertian surface model, in which the apparent brightness of a surface facet is directly related to the cosine of the incident angle.

Since the computation of the grey level using the cosine of the incident angle is expensive, a piecewise linear approximation developed by Peucker is also incorporated into SLOPES. The approximation is computed using an elevation angle of 45 degrees and an azimuth angle of 315 degrees, and may be selected the command REFLECTANCE_MAP 1. The third reflectance map developed by Wiechel also assumes a Lambertian surface, but utilises an approximation to the cosine of the incident angle, again in order to improve performance.

The range of slopes, in other than a particularly mountainous region, is often so small as to produce a disappointing shaded image with little contrast. Exaggerating the height values, using the **ZSCALE** command, prior to calculating the components of the local gradient, has the effect of exaggerating the slope component, and leads to a image with more contrast in areas of low relief. A similar effect can be generated by decreasing the elevation of the light source.

HEIGHT DIFFERENCE CALCULATION

A height difference map may be output using the command **HEIGHT_DIFFERENCE**. The map may be used to give an indication of terrain roughness, or may be used to validate the height matrix (eg. to detect abnormally large or small matrix values).

The algorithm uses a 3 X 3 operator that is passed over the matrix column by column and row by row. The operator calculates a height difference for the centre point of the matrix by calculating the absolute difference between

Z-+	Zo+	Z++
Z-o	Zoo	Z+o
z--	Zo-	Z+-

point Zoo and the 8 surrounding points. By default, an average height value is output, derived by meaning the 8 height difference values, however if the **DISABLE AVERAGE** command has been given, then the maximum of the 8 values is output.

OUTPUT MAP PARAMETERS

The content of the SLOPE, ASPECT and HEIGHT DIFFERENCE maps is controlled using the commands **ANGLE_INTERVAL**, **STEP**, **VALUE_RANGE** and **ZLIMITS**.

ANGLE_INTERVAL is used to define the angular step value for ASPECT calculation. For example, if an angular step value of 90 degrees is used, then 4 different classes will be distinguished, with those parts of the terrain which have a direction of 0 to 90 degrees measured clockwise from north (ie. between north and east) being assigned to class 1. The default angular interval is 30 degrees.

VALUE_RANGE is used to define lower and upper limits of output slope or height difference values. For example, using the command VALUE_RANGE 0 30 it is possible to restrict slope display to those slopes that lie between 0 and 30 degrees. When output is to a graphics device, matrix points outside the value range will appear in black; when output is to a DTI file, matrix points outside the value range are assigned a null value.

The STEP command is used to define the slope or height difference class interval. If a single integer value is supplied with the command, a constant class interval is defined. For example, the command STEP 10 when applied to a slope map, will classify the slopes into 10 degree classes. Alternatively, a variable class interval may be defined by supplying a number of integer arguments with the STEP command. For example, the command STEP 5 8 10 when applied to a slope map, and assuming a value range of 0 to 40, will result in 5 classes of slope being distinguished with class upper limits of 5 13 23 33 40. Note the final step value is used to fill out the remainder of the value range. When output is to a graphics device, each class will appear in a different colour.

In addition to restricting the display of slope or height difference values to a particular value range, it is also possible using the ZLIMITS command, to restrict calculation to matrix points within a particular height range. For example, if the command ZLIMITS 0 200 is given, then only matrix points that fall within this height range, will be used in the slope, aspect or height difference calculations. When output is to a graphics device, matrix points outside the height range will appear in white.

The default value of these parameters may be examined using the commands **SHOW ASPECT**, **SHOW HEIGHT_DIFFERENCE**, **SHOW SHADE** and **SHOW SLOPE**, while the default values may be selected using the commands **DEFAULT ASPECT**, **DEFAULT HEIGHT_DIFFERENCE**, **DEFAULT SHADE** and **DEFAULT SLOPE**.

DTI FILE OUTPUT AND GRAPHICS DISPLAY

It is possible to output a SLOPE, ASPECT, HEIGHT DIFFERENCE or SHADE map either directly to a colour graphics device or to an output DTI file. The latter option is usually selected if the derived information is to be displayed as a raster backdrop within LITES2, or if further processing such as the vector extraction of slope boundaries is required. Note that it is possible to output both to a graphics device and to a DTI file simultaneously.

The command **FILEOUT** is used to specify the name of the output DTI file. The output file is created with the same header as the input DTM, and any header or projection information is transferred to the output file. It is possible to specify the format of the data values in the output DTI file using the command **DATA_TYPE**. By default an output value is held as a 16 bit word.

The derived values are by default output to the DTI file in an unclassified form, and any class values that may have been defined are not used. It is however possible to output the class value, rather than the actual derived value, by giving the command **ENABLE CLASSIFY** prior to generating the output map. If this command is given, then it is the colour index that would be used to display the value on a graphics device that is output. For example, a slope value of 15 degrees which lies in the second class, would be output as 2 with CLASSIFY enabled, and as 15 if CLASSIFY is disabled. The COLOURS command may be used to control the index value that is output to the file.

The command **ENABLE GRAPHICS** is used to direct output to the workstation graphics device. If this option is selected then the derived slope, aspect, height difference or shade data is displayed on the screen, in a classified colour coded or greyscale image, as it is calculated. The way in which the derived data is classified and appears to the user on the screen, is controlled using the output map parameters, along with the commands **LUT**, and **COLOURS**.

The colours used for the display of the classified information are defined in a colour lookup table. 2 colour tables are supplied with the program, and may be found in a directory which forms part of the search list LSL$LOOKUP. By default the colour table 'SLOPES.DAT' is used for slope, height difference and colour aspect maps, and the colour table 'GREY.DAT' for shade or aspect greyscale maps.

The contents of these files may be edited to change the RGB definitions of particular colour indices, alternatively the LUT command may be used to read a user specified colour table. The optional qualifier on the LUT command is used to associate a colour table with a particular type of map eg. the command LUT SLOPES MYTABLE.DAT will cause this colour table to be used in preference to the default colour table 'SLOPES.DAT'.

The COLOURS command may be used to assign a colour index to a particular class. For example the command COLOUR 1 5 6 will assign colours 1 5 6 to classes 1 2 and 3.

By default the screen is cleared of any existing image whenever an ASPECT, SLOPE, HEIGHT_DIFFERENCE or SHADE command is given. The command **DISABLE CLEAR** may be used to disable the clearing of the screen between image generation. It is often useful to do this to demonstrate the relationship between SLOPE, ASPECT and SHADE, and to see the effect of visualising the terrain as a shaded relief map with different light source parameters.

DISPLAY ANNOTATION

When using a graphics device, the display is by default annotated with a legend and text. The legend allows the different colours used in the map to be related to the value class interval. The text displayed on the lower part of the screen provides details on the input DTM, and the output map parameters. The text will vary depending on the type of derived information contained in the map.

Display of a legend is controlled by means of the commands **DISABLE / ENABLE LEGEND** and **LEGEND POSITION**. The **DISABLE / ENABLE TEXT** commands may be used to control whether informational text is output to the screen.

In addition to the annotation described above, the map may be annotated with user information by means of the command **DRAW LABEL**. There is no restriction on the number of labels that may be drawn, however a single text string cannot consist of more than 80 characters. The **LABEL POSITION** command determines the position of the label on the screen; a label is drawn so that the first character is located at the current label position. The **LABEL SIZE** command may be used to select from 4 different character sizes. The command **CLEAR LABEL** may be used to delete user annotation from the screen.

USE OF A DIGITISING TABLE, TABLE PUCK AND MENU

If your workstation incorporates a digitising table, then it is possible to register a source document such as a map to the DTM, and to issue many of the SLOPES commands from a table menu.

If the logical name LSL$AUTO_ENABLE_TABLE is defined with a value of "1" the program will attempt to initialise the Laser Scan Table Monitor Utility on startup. If the logical name is absent or has any other value, initialisation is not performed on startup, however the table can be subsequently initialised using the command ENABLE TABLE.

The Table Monitor controls reading from the digitising table. It must be set up on your system if SLOPES is to take input from the digitising table. If table initialisation fails then the message:

```
*** WARNING *** Initialising table monitor
SLOPES will assume no table is available
```

will be output before the prompt **SLOPES>** is displayed on the terminal. In the event of failure, input from the digitising table or puck button is not possible, and the program will accept commands only from the terminal.

Table initialisation will generally fail because no Table Monitor is currently active, or because the Table Monitor is locked by another user. If neither of these reasons appear to apply, you should consult the TABLIB Reference Manual, or seek guidance from your system manager.

A map is registered to the DTM using the command **SETUP MAP**. Before giving this command you should have first selected the input DTM, and have securely attached the source document to the digitising table.

Registration is performed by digitising 4 rectangular registration points. The registration points represent 4 points on the map that correspond to the 4 corners of the DTM. The points are digitised in the order top left (NW), bottom left (SW), bottom right (SE) and top right (NE) using any button on the table puck. A point is digitised in response to an explanatory prompt on the terminal.

An error message is generated if any of the angles of the digitised rectangle are less than 88 degrees, or greater than 92 degrees (ie. if the corner points of the rectangle are more than 2 degrees off rectangular). In this situation you will be asked to redigitise the 4 registration points.

Once a map has been positioned on the table, it is possible to define an area of interest in the DTM by giving the WINDOW command and digitising two points on the map. If the workstation incorporates a graphics device that supports hardware pixel replication, it is also possible to centre the displayed image around a DTM point using puck button 1. Note that puck buttons 2 and 3 are defined as ENLARGE and REDUCE.

If the table has been successfully initialised, then a command menu may be positioned on the table. The majority of the SLOPES commands may be issued from the menu.

A menu is positioned on the table using the command **SETUP MENU**. As with map setup, registration is performed by digitising 4 rectangular registration points. The registration points represent the 4 corners of the menu. The points are digitised in the order top left, bottom left, bottom right and top right using any button on the table puck. A point is digitised in response to an explanatory prompt on the terminal.

A menu command is issued by digitising a point in the relevant menu box. For commands that require a parameter argument, you should enter the parameter in response to a prompt on the terminal, using the keyboard. The ENABLE commands when given from the menu act as 'flip-flops'; if the enable option is currently disabled it will be enabled when the menu command is given, and vice versa.

SLOPES COMMANDS

COMMAND LIST

The following commands are defined :

```
@               !               ANGLE_INTERVAL  ASPECT
CENTRE          CLEAR           CLEAR LABELS    COLOUR
DATA_TYPE       DEFAULT ASPECT  DEFAULT HEIGHT_DIFFERENCE
DEFAULT SHADE   DEFAULT SLOPE   DISABLE ABSOLUTE
DISABLE AVERAGE DISABLE CLASSIFY                DISABLE CLEAR
DISABLE GRAPHICS                DISABLE GREYSCALE
DISABLE LEGEND  DISABLE NULL_BORDER             DISABLE PERCENT
DISABLE RECENTRE                DISABLE TABLE   DISABLE TEXT
DRAW LABEL      ENABLE ABSOLUTE
ENABLE AVERAGE  ENABLE CLASSIFY ENABLE CLEAR    ENABLE GRAPHICS
ENABLE GREYSCALE                ENABLE LEGEND   ENABLE NULL_BORDER
ENABLE PERCENT  ENABLE RECENTRE ENABLE TABLE    ENABLE TEXT
ENLARGE         EXIT            FILEIN          FILEOUT
HEIGHT_DIFFERENCE               HELP            LABEL POSITION
LABEL SIZE      LEGEND POSITION LUT             REDUCE
REFLECTANCE_MAP SETUP MAP       SETUP MENU      SHADE
SHOW ANNOTATION SHOW ASPECT     SHOW ENABLE     SHOW FILES
SHOW HEIGHT_DIFFERENCE          SHOW SHADE      SHOW SLOPE
SLOPES          SPAWN           STEP            SUN_ANGLE
SUN_POSITION    UNITS           VALUE_RANGE     WAIT
WINDOW          ZLIMITS         ZSCALE
```

@

Take command input from the specified file.

FORMAT: @ file-spec

Command parameters:

file-spec

The file to be opened and used for command input.

Any parts of the file-spec not supplied will be taken from the default specification 'SYS$DISK:[].COM;0'.

DESCRIPTION:

SLOPES offers the facility of command input from an indirect command file. The '@' character preceding a file-spec will cause SLOPES to open and read commands from the specified file until:

1. a RETURN command is detected and command input is returned to SYS$COMMAND.

2. the end of file is detected. This provokes an error message and command input is returned to SYS$COMMAND.

Nested command files are not supported (i.e. a command file containing an '@' command), although sequential '@' commands are supported when read from SYS$COMMAND.

As an aid to batch log interpretation SLOPES will echo all commands read from an indirect command file.

Messages:

The following messages are specific to the @ command:

*** ERROR *** Specifying command @
Command file specification is missing

*** ERROR *** Specifying command @
Unable to open indirect command file 'file-spec'

*** ERROR *** Specifying command @
Nested command files not supported

Examples:

```
SLOPES> @SLOPES_TEST<CR>
SLOPES> STEP 7
SLOPES> VALUE_RANGE 10 20
SLOPES>
```

!

Treat all text to the right of the '!' as a comment.

FORMAT: ! [comment text]

Command parameters:

comment text

text that is to be treated as a comment and which will be excluded from command interpretation.

DESCRIPTION:

An exclamation mark is the standard TVES package comment delimiter. All text (and numbers) which lie to the right of a '!' character are excluded from command interpretation. Comments are useful for annotating command procedures used in batch processing etc.

Messages: None.

Examples:

SLOPES> **ENABLE AVERAGE !calculate average height differences<CR>**
SLOPES> **ENABLE ABSOLUTE !use absolute coordinates<CR>**
SLOPES>

ANGLE_INTERVAL

Sets the angular step value for aspect calculation.

FORMAT: **ANGLE_INTERVAL angular_interval**

Command parameters:

angular_interval

An integer value specifying a whole number of degrees. The default value is 30 degrees.

DESCRIPTION:

ANGLE_INTERVAL is used to define the angular step value for ASPECT calculation. If output to a graphics device has been selected with the ENABLE GRAPHICS command then each angular step is displayed in a different colour.

If output is to a DTI file then if the ENABLE CLASSIFY command has been given, each classified index is output to the DTI file. eg. if an angular step of 10 degrees is specified then aspect values of 1 to 10 degrees would be output to the DTI file with a value of 1, 11 to 20 degrees with a value of 2 etc.

If the ENABLE CLASSIFY command is not specified then the ANGLE_INTERVAL command has no effect on the output to a DTI file.

Messages:

The following error messages are specific to the ANGLE_INTERVAL command.

*** ERROR *** Specifying command ANGLE_INTERVAL
Command requires an integer value between 3 and 360

Examples:

SLOPES>**ANGLE_INTERVAL 20<CR>**
SLOPES>

ASPECT

Calculates and displays an ASPECT map.

FORMAT: **ASPECT**

Command parameters: None

DESCRIPTION:

This command calculates an ASPECT map indicating the direction of SLOPE at each node in the DTM. Aspect is measured in degrees clockwise from north (ie east = 90 degrees, west = 270 degrees).

Further information on the ASPECT algorithm used is given in the Description section.

The aspect information is either displayed on the graphics screen, if the ENABLE GRAPHICS command has been given and/or to a DTI file if the FILEOUT command has been issued.

If output to a graphics device is selected then either a grey or colour image is produced depending on whether the ENABLE GREY command has been given. By default a colour image is produced. The displayed step intervals are controlled by the ANGLE_INTERVAL command.

If output is to a DTI file then if the ENABLE CLASSIFY has been given each classified index, as defined by the ANGLE_INTERVAL command is output to the DTI file. For example, if an angular step of 10 degrees is specified then aspect values of 1 to 10 degrees are output to the DTI file with a value of 1, 11 to 20 degrees with a value of 2 etc. Otherwise the actual calculated aspect values are output.

ASPECT information is only calculated for matrix nodes lying within the currently defined WINDOW.

Typing <CTRL/C> (Pressing the Ctrl and C keys together) aborts the calculation and display of ASPECT.

Messages:

The following error messages are specific to the ASPECT command.

```
*** ERROR *** Specifying command ASPECT
Output medium not specified
Use ENABLE GRAPHICS or FILEOUT commands
```

```
*** ERROR *** Specifying command ASPECT
No input DTI file had been specified
```

```
*** ERROR *** Specifying command ASPECT
```

DATA_TYPE BYTE illegal for ASPECT output

Examples:

SLOPES>**ASPECT<CR>**
SLOPES>

CENTRE

Centres the current display around the specified DTI matrix point.

FORMAT: CENTRE x_coord y_coord

Command parameters:

x_coord y_coord

The matrix coordinates specified as a column and row value. 2 integer values are required.

DESCRIPTION:

The displayed image is centred around the screen position corresponding to the supplied matrix x y coordinate point provided they lie within the geographical bounds of the file.

This command has effect only on a graphics device which supports hardware pixel replication.

Messages:

The following error messages are specific to the CENTRE command.

*** ERROR *** Specifying command CENTRE
A graphics device is not selected

*** ERROR *** Specifying command CENTRE
Command requires 1 DTI x,y coord pair

Examples:

SLOPES>**CENTRE 100 100<CR>**
SLOPES>

CLEAR

Clears the screen.

FORMAT: **CLEAR**

Command Parameters: None

DESCRIPTION:

This command, when issued with no qualifiers, clears the entire screen of displayed information.

Messages:

The following messages are specific to the CLEAR command.

```
*** ERROR *** specifying command CLEAR
Valid qualifier is LABEL
```

Example:

SLOPES>**CLEAR<CR>**
SLOPES>

CLEAR LABEL

Clears user annotation.

FORMAT: **CLEAR LABEL**

Command Parameters: None

DESCRIPTION:

User annotation which has been specified with the DRAW LABEL command is cleared from the screen. Any underlying image is preserved.

Messages:

The following message is specific to the CLEAR command.

*** ERROR *** specifying command CLEAR
Valid qualifier is LABEL

Example:

SLOPES>**CLEAR LABEL<CR>**
SLOPES>

COLOURS

Defines the colour entry that is to be associated with the value steps.

FORMAT: COLOURS I1 I2 ...I7

Command parameters:

I1 I2 ...I7

The colour index values to be associated with the value steps. Up to seven integer values may be specified in the range 1 to 127.

DESCRIPTION:

The COLOUR command allows the colour indices associated with step values to be reassigned.
eg.

```
COLOURS 3 12 7
```

allocates colours 3,12 and 7 in the colour table to the first three gradient steps in a slope map.

The command applies only to SLOPE and HEIGHT_DIFFERENCE maps. If the ENABLE CLASSIFY command has been given, then the colour index values are output to the DTI file.

Messages:

The following error messages are specific to the COLOURS command.

```
*** ERROR *** Specifying command COLOURS
Command requires between 1 and 7 colour index values

*** ERROR *** Specifying command COLOURS
Colour values should be in the range 1 to 127
```

Examples:

SLOPES>**COLOURS 2 5 20<CR>**
SLOPES>

DATA_TYPE

Sets the data type for the output file.

FORMAT: DATA_TYPE keyword

Command Parameters:

keyword

The data type chosen from one of the following:

BYTE
WORD
LONGWORD
REAL

The default data type is WORD.

DESCRIPTION:

The DATA_TYPE command is used to set the data type of the output DTI file specified using the FILEOUT command.

Note that ASPECT maps may not be output if BYTE data type has been selected, unless ENABLE CLASSIFY has been specified.

The default data type is WORD.

Messages: None
The following message is specific to the DATA_TYPE command.

*** ERROR *** specifying command DATA_TYPE
Valid qualifier is BYTE, LONGWORD, REAL or WORD.

Example:

SLOPES>**DATA_TYPE WORD<CR>**
SLOPES>

DEFAULT ASPECT

Resets the default ASPECT parameters

FORMAT: **DEFAULT ASPECT**

Command Parameters: None

DESCRIPTION:

This command resets the default program parameters for the calculation and display of aspect maps. The following values are set: -

ANGLE_INTERVAL	30 degrees
ZLIMITS	DTI file minimum and maximum values
LUT ASPECT	LSL$LOOKUP:SLOPES.DAT
Colour Image Selected	

Messages:

The following message is specific to the DEFAULT command.

*** ERROR *** specifying command DEFAULT
Valid qualifier is SLOPE ,ASPECT, SHADE or HEIGHT_DIFFERENCE

Example:

SLOPES>**DEFAULT ASPECT<CR>**
SLOPES>

DEFAULT HEIGHT_DIFFERENCE

Resets the default HEIGHT_DIFFERENCE parameters

FORMAT: **DEFAULT HEIGHT_DIFFERENCE**

Command Parameters: None

DESCRIPTION:

This command resets the default program parameters for the calculation and display of height difference maps. The following values are set: -

```
STEP                    5
VALUE_RANGE             0 - 60
ZLIMITS                 DTI file minimum and maximum values
LUT HEIGHT__DIFFERENCE  LSL$LOOKUP:SLOPES.DAT
Calculation of Average Height difference selected
```

Messages:

The following message is specific to the DEFAULT command.

*** ERROR *** specifying command DEFAULT
Valid qualifier is SLOPE ,ASPECT, SHADE or HEIGHT_DIFFERENCE

Example:

SLOPES>**DEFAULT HEIGHT_DIFFERENCE<CR>**
SLOPES>

DEFAULT SHADE

Resets the default SHADE parameters

FORMAT: **DEFAULT SHADE**

Command Parameters: None

DESCRIPTION:

This command resets the default program parameters for the calculation and display of shaded maps. The following values are set: -

```
SUN_ANGLE               45 degrees
SUN_POSITION            315 degrees (North-West)
REFLECTANCE_MAP          2
LUT SHADE               LSL$LOOKUP:GREY.DAT
```

Messages:

The following message is specific to the DEFAULT command.

*** ERROR *** specifying command DEFAULT
Valid qualifier is SLOPE ,ASPECT, SHADE or HEIGHT_DIFFERENCE

Example:

SLOPES>**DEFAULT SHADE<CR>**
SLOPES>

DEFAULT SLOPE

Resets the default SLOPE parameters

FORMAT: **DEFAULT SLOPE**

Command Parameters: None

DESCRIPTION:

This command resets the default program parameters for the calculation and display of slope maps. The following values are set: -

STEP	5 degrees
VALUE_RANGE	0 - 60 degrees
ZLIMITS	DTI file minimum and maximum values
LUT SLOPE	LSL$LOOKUP:SLOPES.DAT
Slope data in degrees	

Messages:

The following message is specific to the DEFAULT command.

*** ERROR *** specifying command DEFAULT
Valid qualifier is SLOPE ,ASPECT, SHADE or HEIGHT_DIFFERENCE

Example:

SLOPES>**DEFAULT SLOPE<CR>**
SLOPES>

DISABLE ABSOLUTE

Disables a previous ENABLE ABSOLUTE command.

FORMAT: **DISABLE ABSOLUTE**

Command Parameters: None

DESCRIPTION:

DISABLE ABSOLUTE cancels a previous ENABLE ABSOLUTE command. If DISABLE ABSOLUTE is given, then coordinate values required by the WINDOW command, supplied in metre or projection units, must be specified as an offset from the SW corner of the matrix.
By default window values should be specified as absolute coordinates.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the DISABLE command.

*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT

Examples:

SLOPES>**DISABLE ABSOLUTE<CR>**
SLOPES>

DISABLE AVERAGE

Disables a previous ENABLE AVERAGE command.

FORMAT: **DISABLE AVERAGE**

Command Parameters: None

DESCRIPTION:

DISABLE AVERAGE cancels a previous ENABLE AVERAGE command.
If DISABLE AVERAGE is given, then maximum height difference is calculated when the HEIGHT_DIFFERENCE command is given.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the DISABLE command.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT
```

Examples:

```
SLOPES>DISABLE AVERAGE<CR>
SLOPES>
```

DISABLE CLASSIFY

Disables a previous ENABLE CLASSIFY command.

FORMAT: **DISABLE CLASSIFY**

Command Parameters: None

DESCRIPTION:

DISABLE CLASSIFY cancels a previous ENABLE CLASSIFY command. If DISABLE CLASSIFY is given, then the calculated SLOPE, HEIGHT_DIFFERENCE or ASPECT values are written to the output DTI file.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the DISABLE command.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT
```

Examples:

SLOPES>**DISABLE CLASSIFY<CR>**
SLOPES>

DISABLE CLEAR

Disables a previous ENABLE CLEAR command.

FORMAT: **DISABLE CLEAR**

Command Parameters: None

DESCRIPTION:

DISABLE CLEAR cancels a previous ENABLE CLEAR command. If DISABLE CLEAR is given, then the screen is not automatically cleared before each new image is generated.

The DISABLE CLEAR command is useful to allow differences caused by changing output map parameters to be examined.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the DISABLE command.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT
```

Examples:

SLOPES>**DISABLE CLEAR<CR>**
SLOPES>

DISABLE GRAPHICS

Disables a previous ENABLE GRAPHICS command.

FORMAT: **DISABLE GRAPHICS**

Command Parameters: None

DESCRIPTION:

DISABLE GRAPHICS cancels a previous ENABLE GRAPHICS command. If DISABLE GRAPHICS is given, then SLOPE, ASPECT, HEIGHT_DIFFERENCE or SHADE information is not output to the graphics device.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the DISABLE command.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT
```

Examples:

SLOPES>**DISABLE GRAPHICS<CR>**
SLOPES>

DISABLE GREYSCALE

Disables a previous ENABLE GREYSCALE command.

FORMAT: **DISABLE GREYSCALE**

Command Parameters: None

DESCRIPTION:

DISABLE GREYSCALE cancels a previous ENABLE GREYSCALE command. The command is used in conjunction with the ASPECT command to specify that the aspect map should be displayed as a colour image.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the DISABLE command.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT
```

Examples:

SLOPES>**DISABLE GREYSCALE<CR>**
SLOPES>

DISABLE LEGEND

Disables a previous ENABLE LEGEND command.

FORMAT: **DISABLE LEGEND**

Command Parameters: None

DESCRIPTION:

DISABLE LEGEND cancels a previous ENABLE LEGEND command. If DISABLE LEGEND is given, then no colour value guide is generated before an ASPECT, SLOPE, or HEIGHT_DIFFERENCE image is generated.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the DISABLE command.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT
```

Examples:

```
SLOPES>DISABLE LEGEND<CR>
SLOPES>
```

DISABLE NULL_BORDER

Disables a previous ENABLE NULL_BORDER command.

FORMAT: **DISABLE NULL_BORDER**

Command Parameters: None

DESCRIPTION:

DISABLE NULL_BORDER cancels a previous ENABLE NULL_BORDER command. It disables null values being written to the first and last rows and columns in the output DTI file.

The DTM is conceptually expanded by 2 columns and 2 rows, and heights are interpolated linearly for the new points, to allow an output slope or aspect value to be calculated for the first and last column and row.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the DISABLE command.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT
```

Examples:

SLOPES>**DISABLE NULL_BORDER<CR>**
SLOPES>

DISABLE PERCENT

Disables a previous ENABLE PERCENT command.

FORMAT: **DISABLE PERCENT**

Command Parameters: None

DESCRIPTION:

DISABLE PERCENT cancels a previous ENABLE PERCENT command. Gradient values are subsequently calculated in degrees of slope.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the DISABLE command.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT
```

Examples:

```
SLOPES>DISABLE PERCENT<CR>
SLOPES>
```

DISABLE RECENTRE

Disables a previous ENABLE RECENTRE command.

FORMAT: **DISABLE RECENTRE**

Command Parameters: None

DESCRIPTION:

DISABLE RECENTRE cancels a previous ENABLE RECENTRE command.

The display remains at the current magnification factor, and is not recentred before the drawing of a new image.

To avoid undesirable effects it is advisable to DISABLE LEGEND and TEXT display when using DISABLE RECENTRE.

This command has effect only on a graphics device which supports hardware pixel replication. The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the DISABLE command.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT
```

Examples:

SLOPES>**DISABLE RECENTRE<CR>**
SLOPES>

DISABLE TABLE

Turns off input from the digitising table.

FORMAT: **DISABLE TABLE**

Command Parameters: None

DESCRIPTION:

The DISABLE TABLE command turns off input from the digitising table. Input from the table can be reselected using the ENABLE TABLE command.

The SHOW ENABLE command may be used to check on the status of table input.

Messages:

The following message is specific to the DISABLE command.

*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT

Examples:

SLOPES>**DISABLE TABLE<CR>**
SLOPES>

DISABLE TEXT

Disables a previous ENABLE TEXT command.

FORMAT: **DISABLE TEXT**

Command Parameters: None

DESCRIPTION:

DISABLE TEXT disables the output of border information when an image is displayed. By default the DTI file name, area of interest, value range and current parameter settings are output to the screen.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the DISABLE command.

*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT

Examples:

SLOPES>**DISABLE TEXT<CR>**
SLOPES>

DRAW LABEL

Annotates the display

FORMAT: DRAW LABEL text

Command parameters:

text

The text to be written to the display. Up to 80 characters of text may be specified.

DESCRIPTION:

The specified text is written to the display.

By default the position of the text is along the bottom line of the current plotting area.

Both the size and position of the text may be changed by means of the LABEL SIZE and LABEL POSITION commands.

Messages:

The following message is specific to the DRAW LABEL command.

```
*** ERROR *** Specifying command DRAW
Command qualifier is LABEL
```

Examples:

SLOPES> **DRAW LABEL A Very Pretty Picture<CR>**
SLOPES>

ENABLE ABSOLUTE

Selects the use of absolute coordinates values.

FORMAT: **ENABLE ABSOLUTE**

Command Parameters: None

DESCRIPTION:

If ENABLE ABSOLUTE is given, then coordinate values required by the WINDOW command, supplied in metre or projection units, must be specified as absolute (rather than relative) coordinate values.

For example if the projection indicates U.K. National Grid, then the WINDOW values may be specified as 6 figure National Grid coordinates.
By default window values should be specified as absolute coordinates.

This option can be disabled using the DISABLE ABSOLUTE command. The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the ENABLE command.

```
*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT
```

Examples:

SLOPES>**ENABLE ABSOLUTE<CR>**
SLOPES>

ENABLE AVERAGE

Selects average height difference calculation.

FORMAT: **ENABLE AVERAGE**

Command Parameters: None

DESCRIPTION:

If the ENABLE AVERAGE command is given, then average height difference is calculated when a HEIGHT_DIFFERENCE map is generated. The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the ENABLE command.

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT

Examples:

SLOPES>**ENABLE AVERAGE<CR>**
SLOPES>

ENABLE CLASSIFY

FORMAT: **ENABLE CLASSIFY**

Command Parameters: None

DESCRIPTION:

If ENABLE CLASSIFY is given, then SLOPE, HEIGHT_DIFFERENCE or ASPECT values are output as classified indices as defined by the STEP and COLOUR, or ANGLE_INTERVAL commands. The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the ENABLE command.

```
*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT
```

Examples:

SLOPES>**ENABLE CLASSIFY**
SLOPES>**FILEOUT TEST**
SLOPES>**STEP 10<CR>**
SLOPES>**COLOURS 5 10 15 20 30**
SLOPES>SLOPE 1
File LSL$DTI:TEST.SLP opened for output

In this example a slope value of 15 degrees lying in the second step interval is output as colour index 10 to the DTI file.

SLOPES>

ENABLE CLEAR

Selects the clearing of the graphics screen between image displays.

FORMAT: **ENABLE CLEAR**

Command Parameters: None

DESCRIPTION:

If the ENABLE CLEAR command is given, then the graphics screen is automatically cleared prior to display of ASPECT, SLOPES, SHADE or HEIGHT_DIFFERENCE images. The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the ENABLE command.

```
*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT
```

Examples:

SLOPES>**ENABLE CLEAR<CR>**
SLOPES>

ENABLE GRAPHICS

Selects output to a workstation graphics device.

FORMAT: **ENABLE GRAPHICS**

Command Parameters: None

DESCRIPTION:

The ENABLE GRAPHICS command initialises the graphics workstation. Subsequent SLOPES output generated by the SLOPE, ASPECT, HEIGHT_DIFFERENCE and SHADE commands is directed to the graphics display.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the ENABLE command.

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT

Examples:

SLOPES>**ENABLE GRAPHICS<CR>**
SLOPES>

ENABLE GREYSCALE

Selects the use of a greyscale for output of ASPECT information.

FORMAT: **ENABLE GREYSCALE**

Command Parameters: None

DESCRIPTION:

The command is used in conjunction with the ASPECT command to specify that the aspect map should be displayed as a greyscale image.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the ENABLE command.

```
*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT
```

Examples:

SLOPES>**ENABLE GREYSCALE<CR>**
SLOPES>

ENABLE LEGEND

Enables the output of a display legend

FORMAT: **ENABLE LEGEND**

Command Parameters: None

DESCRIPTION:

ENABLE LEGEND enables the display of a colour/value guide when an ASPECT, SLOPE or HEIGHT_DIFFERENCE command is given.

The position of the LEGEND on the screen may be changed with the LEGEND POSITION command.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the ENABLE command.

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT

Examples:
SLOPES>**ENABLE LEGEND**
SLOPES>

ENABLE NULL_BORDER

Selects the output of null border values to the DTI file.

FORMAT: **ENABLE NULL_BORDER**

Command Parameters: None

DESCRIPTION:

The ENABLE NULL_BORDER command specifies that null values are written to the first and last rows and columns in the output DTI file.

The null values output are dependent on the output data type specified with the DATA TYPE command. These values are specified in the DTILIB reference manual.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the ENABLE command.

```
*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT
```

Examples:

SLOPES>**ENABLE NULL_BORDER<CR>**
SLOPES>

ENABLE PERCENT

Selects the calculation of gradient as percentage slope values.

FORMAT: **ENABLE PERCENT**

Command Parameters: None

DESCRIPTION:

The ENABLE PERCENT command is used to select the calculation of percentage slope. Gradients for subsequent SLOPE commands are output as percentage slope. ie:-

45° = 100%

Slopes of greater than 45° are output as 100%.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the ENABLE command.

```
*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT
```

Examples:

SLOPES>**ENABLE PERCENT<CR>**
SLOPES>

ENABLE RECENTRE

Selects the recentring of an image before display of an image.

FORMAT: **ENABLE RECENTRE**

Command Parameters: None

DESCRIPTION:

The ENABLE RECENTRE command specifies that the display is recentred and set to a magnification factor of one before the drawing of a new image.

This command has effect only on a graphics device which supports hardware pixel replication.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the ENABLE command.

```
*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT
```

Examples:

SLOPES>**ENABLE RECENTRE<CR>**
SLOPES>

ENABLE TABLE

Initialises the Table Monitor to allow input from a digitising table.

FORMAT: **ENABLE TABLE**

Command Parameters: None

DESCRIPTION:

The ENABLE TABLE command initialises the table monitor to allow input from a digitising table. The command SETUP MAP may be used to register a map to the DTI file. The SETUP MENU command allows commands to be issued using the SLOPES table menu. In the event of failure, input from the digitising table or puck button is not possible, and the program will accept commands only from the terminal.

Table initialisation will generally fail because no Table Monitor is currently active, or because the Table Monitor is locked by another user. If neither of these reasons appear to apply, you should consult the TABLIB Reference Manual, or seek guidance from your system manager.

If the logical name LSL$AUTO_ENABLE_TABLE is defined with a value of "1", then the table is initialised on program startup.

The DISABLE TABLE command may be used to turn off input from the digitising table.

The SHOW ENABLE command may be used to check on the status of table input.

Messages:

The following message is specific to the ENABLE command.

```
*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT
```

Examples:

SLOPES>**ENABLE TABLE<CR>**
SLOPES>

ENABLE TEXT

Enables the output of border information.

FORMAT: **ENABLE TEXT**

Command Parameters: None

DESCRIPTION:

ENABLE TEXT enables the output of border information when an image is displayed. The DTI filename, area of interest, selected value range and current parameter settings are output to the screen.

The SHOW ENABLE command may be used to check on the status of the option.

Messages:

The following message is specific to the ENABLE command.

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, AVERAGE, CLEAR,
CLASSIFY, GRAPHICS, GREYSCALE, LEGEND, NULL_BORDER, PERCENT,
RECENTRE, TABLE or TEXT

Examples:

SLOPES>**ENABLE TEXT<CR>**
SLOPES>

ENLARGE

Magnifies the displayed image by a factor of two.

FORMAT: **ENLARGE**

Command Parameters: None

DESCRIPTION:

The screen image is magnified by a factor of 2 using pixel replication. If the command is repeatedly given the display will continue to be enlarged until the maximum magnification (x 16) is reached.

This command has effect only on a graphics device which supports hardware pixel replication.

Messages

Display is at maximum magnification

Examples:

SLOPES>**ENLARGE<CR>**
SLOPES>

EXIT

Terminates program execution.

FORMAT: **EXIT**

Command Parameters: None

DESCRIPTION:

The program is terminated, and any currently opened DTI files will be unmapped, along with any input or output IFF files.

<CTRL/Z> (pressing the Ctrl and Z keys together) may also be used to exit from the program.

Messages None

Example:

SLOPES>**EXIT<CR>**
$

FILEIN

Opens and maps into memory a DTI file.

FORMAT: FILEIN file-spec

Command parameters:

file-spec

The file specification for the input DTI file. Any part of the file specification not supplied is taken from the default 'LSL$DTI:DTI.DTI'

DESCRIPTION:

This command opens and maps into memory a DTI file.
Details derived from the header of the file are displayed on the terminal to confirm that the file has been successfully opened.

If the logical name LSL$MATRIX_DEFAULT_WINDOW is defined with a value of "1", then a default area of interest defined in matrix units of bottom left hand corner 1,1 and top right hand corner 200,200 is set when the file is opened. If the logical name is absent or has any other value, or if the input DTI file has fewer than 200 columns or 200 rows, then a default window equivalent to the total matrix is set.

The area of interest may be altered at any time by use of the WINDOW command.

If the rotation value (DTI_ORDER_CORNER) held in the header of the DTI file is not the default, (south west) then a warning message is output when the file is opened.

Information on the mapped DTI file may be obtained at any time by typing 'SHOW FILES'.

Messages:

The following messages are specific to the FILEIN command.

*** ERROR *** Specifying command FILEIN
Filename is missing

Example:

```
SLOPES>FILEIN TEST<CR>

File    :  LSL$DTI:TEST.DTI
Header  :  LSLA Data:  WORD

Units are DTI Matrix Value

Matrix Coverage   SW:      1      1  NE:   301   251
Matrix Window     SW:      1      1  NE:   301   251
Matrix Interval    E:      1          N:     1
Value Range         :      0    to    851
SLOPES>
```

FILEOUT

Specifies the file-spec of the DTI file is to receive SLOPE, ASPECT, HEIGHT_DIFFERENCE or SHADE DTI data.

FORMAT: FILEOUT file-spec

COMMAND PARAMETERS:

file-spec

The file-spec of the DTI file which is to receive the derived data.

Any parts of the file-spec not supplied for the FILEOUT command will be taken from the default file specification 'LSL$DTI:DTI.SLP;0'.

DESCRIPTION:

The FILEOUT command specifies the name of the DTI file which is to receive the calculated SLOPE, ASPECT, HEIGHT_DIFFERENCE or SHADE data. The file is closed after calculations have taken place.

Note that output may take place both to the DTI file and the graphics screen if the ENABLE GRAPHICS command has also been given.

Messages:

The following messages are specific to the FILEOUT command:

*** ERROR *** Specifying command FILEOUT
Filename is missing

Examples:

SLOPES> **FILEOUT DUA3:[DEMONSTRATION]IDAHO<CR>**
SLOPES>

HEIGHT_DIFFERENCE

Calculates and displays a height difference map.

FORMAT: HEIGHT_DIFFERENCE

Command parameters: None

DESCRIPTION:

This command calculates a height difference map indicating the difference in height between each DTM node and its neighbours. If the ENABLE AVERAGE command has been given then the average height difference is output. Otherwise the maximum height difference is calculated.

Output is displayed on the graphics screen if ENABLE GRAPHICS command has been given and/or to a DTI file if the FILEOUT command has been issued.

HEIGHT_DIFFERENCE information is only calculated matrix points that lie within the currently defined WINDOW.

Typing <CTRL/C> (Pressing the Ctrl and C keys together) aborts the calculation and display of height difference.

Messages:

The following error messages are specific to the HEIGHT_DIFFERENCE command.

```
*** ERROR *** Specifying command HEIGHT_DIFFERENCE
Output medium not specified
Use ENABLE GRAPHICS or FILEOUT commands

*** ERROR *** Specifying command HEIGHT_DIFFERENCE
No input DTI file had been specified
```

Examples:

```
SLOPES>HEIGHT_DIFFERENCE<CR>
SLOPES>
```

HELP

Invokes on-line help

FORMAT: **HELP [command]**

Command parameters:

command

the command for which help is required

DESCRIPTION:

A brief description is given of the function and format of the specified command. If no parameter is specified then a complete list of available commands are output.

Messages: None

Examples:

```
SLOPES>HELP SLOPE<CR>
SLOPES>
```

LABEL POSITION

Positions the user annotation.

FORMAT: LABEL POSITION screenx screeny

Command parameters:

screenx, screeny

The screen coordinates in pixel units. Two integer values are required.

DESCRIPTION:

The position of the user annotation is defined

The position is specified in screen pixel units, with respect to the SW corner of the screen.

Messages:

The following messages are specific to the LABEL and LABEL POSITION commands.

*** ERROR *** Specifying command LABEL
Command qualifiers are POSITION or SIZE

Examples:

SLOPES>**LABEL POSITION 400 200<CR>**
SLOPES>

LABEL SIZE

Selects the size of user annotation

FORMAT: LABEL SIZE font-size

Command parameters:

font-size

The size of the user text. An integer in the range 1 (smallest) to 4 (largest) is required. The default size is 2.

DESCRIPTION:

The command LABEL SIZE is used to define the size of any text output to a graphics screen using the DRAW LABEL command.

The font_size parameter controls the relative size of the text; size 1 is the smallest, and size 4 is the largest. The actual size of the output text screen will vary with the size of the graphics device screen, though the relative sizes of the text will remain constant.

Messages:

The following messages are specific to the LABEL and LABEL SIZE commands.

*** ERROR *** Specifying command LABEL
Command qualifiers are POSITION, or SIZE

*** ERROR *** Specifying command LABEL SIZE
Command requires an integer in the range 1 to 4

Examples:

SLOPES>**LABEL SIZE 3<CR>**
SLOPES>

LEGEND POSITION

Defines the position of the legend on the screen.

FORMAT: **LEGEND POSITION screenx screeny**

Command parameters:

screenx, screeny

The screen coordinates in pixel units. Two integer values are required.

DESCRIPTION:

The position of the legend is defined

The position is specified in screen pixel units, with respect to the bottom left corner of the screen.

Messages:

The following messages are specific to the LEGEND and LEGEND POSITION commands.

*** ERROR *** Specifying command LEGEND
Command qualifier is POSITION

Examples:

SLOPES>**LEGEND POSITION 400 200<CR>**
SLOPES>

LUT

Selects and makes current the specified colour table.

FORMAT: LUT [keyword] file-spec

Command parameters:

keyword

The SLOPES option to which the look up table refers. Valid options are:-

ASPECT
HEIGHT_DIFFERENCE
SLOPE
SHADE

file-spec

The file specification for the new colour table. Any part of the file specification not supplied will be taken from the default LSL$LOOKUP:.DAT.

DESCRIPTION:

This command is used to read in a new look up colour table. The colour table is a file containing the RGB definitions for a series of colour values. These values are used when images are displayed on the screen. If the optional keyword argument is omitted, then the specified colour table is read in immediately.

The keyword argument is used to to specify the colour table to be used when subsequent SLOPE, ASPECT, SHADE or HEIGHT_DIFFERENCE maps are generated on the graphics screen.

Messages:

The following message is specific to the LUT command.

*** ERROR *** Specifying command LUT
Filename is missing

*** ERROR *** Specifying command LUT
A graphics device is not selected

Examples:

SLOPES>**LUT SHADES YELLOW.DAT<CR>**
SLOPES>**LUT YELLOW<CR>**

REDUCE

Reduces the size of the displayed image by a factor of two.

FORMAT: **REDUCE**

Command Parameters: None

DESCRIPTION:

The screen image is reduced by a factor of 2 using pixel replication. If the command is repeatedly given the display will continue to be enlarged until the maximum magnification (x 16) is reached.

This command has effect only on a graphics device which supports hardware pixel replication.

Messages

Display is at minimum magnification

Examples:

SLOPES>**REDUCE<CR>**
SLOPES>

REFLECTANCE_MAP

Selects a reflectance map for the generation of a shaded relief map.

FORMAT: **REFLECTANCE__MAP index**

Command Parameters:

index

The index number of the reflectance map. An integer between 1 and 3 is required.

DESCRIPTION:

This command is used to specify the reflectance map to be used in the generation of a shaded relief map. The three reflectance maps are as follows:-

1	= Peucker's piecewise linear approximation to a Lambertian surface.
2 (default)	= Reflectance from a Lambertian surface.
3	= Wiechel's projected incident angle algorithm.

Note: Reflectance Map 1 assumes the source of illumination to be in the standard 'hill-shading' position ie. in the North West at 45°, and current sun angle and sun position settings are not used by this algorithm.

Messages

The following message is specific to the REFLECTANCE_MAP command.

```
*** ERROR *** Specifying command REFLECTANCE_MAP
Command requires an integer between 1 and 3
```

Examples:

SLOPES>**REFLECTANCE_MAP 1 <CR>**
SLOPES>

SETUP MAP

Registers a map or other source document placed on a digitising table, to the input DTM.

FORMAT: **SETUP MAP**

DESCRIPTION:

The SETUP MAP command allows a source document to be registered to the input DTM.

In order to register the map and a DTM, the DTM should have first been specified, and a source document should have been securely attached to the surface of a digitising table.

On giving the command you will be asked to digitise 4 rectangular registration points. The registration points represent 4 points on the map that correspond to the 4 corners of the DTM. The points are digitised in the order top left (NW), bottom left (SW), bottom right (SE) and top right (NE) using any button on the table puck. A point should be digitised in response to a prompt on the terminal.
An error message is generated if any of the angles of the digitised rectangle are less than 88 degrees, or greater than 92 degrees (ie. if the corner points of the rectangle are more than 2 degrees off rectangular). In this case you will be asked to redigitise the 4 registration points.
Setup of the map may be aborted using <CTRL/Z> (pressing the Ctrl and Z keys together).

Following the registration of a map to the DTM, coordinate values required by the WINDOW command, may be input using the table puck.

Messages:

The following error messages are specific to the SETUP and SETUP MAP commands:

*** ERROR *** Specifying command SETUP
Command qualifiers are MAP or MENU

*** ERROR *** Specifying command SETUP MAP
Command is invalid if the table has not been initialised

*** ERROR *** Specifying command SETUP MAP
The input DTM must be specified before SETUP MAP

*** ERROR *** Setting up MAP
Badly digitised corner points; try again

Examples:

SLOPES>**SETUP MAP<CR>**

Digitise map NW corner>

Digitise map SW corner>

Digitise map SE corner>

Digitise map NE corner>

SLOPES>

SETUP MENU

Positions the SLOPES menu on the digitising table.

FORMAT: **SETUP MENU**

DESCRIPTION:

The SETUP MENU command allows the SLOPES menu to be positioned on the digitising table.

On giving the command you will be asked to digitise the 4 corners points of the menu. The corners are digitised in the order top left (NW), bottom left (SW), bottom right (SE) and top right (NE) using any button on the table puck. A point should be digitised in response to a prompt on the terminal.
An error message is generated if any of the angles of the digitised rectangle are less than 88 degrees, or greater than 92 degrees (ie. if the corner points of the rectangle are more than 2 degrees off rectangular). In this case you will be asked to redigitise the 4 menu corner points.
Setup of the menu may be aborted using <CTRL/Z> (pressing the Ctrl and Z keys together).

Once a menu has been positioned on the table, it is possible to enter many of the SLOPES commands by digitising a point in the appropriate menu box. If a command requires a parameter argument, you should enter the parameter in response to the prompt on the terminal, using the keyboard.

Messages:

The following error messages are specific to the SETUP and SETUP MENU commands:

*** ERROR *** Specifying command SETUP
Command qualifiers are MAP or MENU

*** ERROR *** Specifying command SETUP MENU
Command is invalid if the table has not been initialised

*** ERROR *** Setting up MENU
Badly digitised corner points; try again

Examples:

SLOPES>**SETUP MENU<CR>**

Digitise menu NW corner>

Digitise menu SW corner>

Digitise menu SE corner>

Digitise menu NE corner>

SLOPES>

SHADE

Calculates and displays a shaded relief map.

FORMAT: **SHADE**

Command parameters: None

DESCRIPTION:

This command calculates a shaded relief map.

Output is displayed on the graphics screen if ENABLE GRAPHICS command has been given and/or to a DTI file if the FILEOUT command has been issued.

The description section contains more information on the algorithms used in the generation of a shaded relief map.

SHADE information is only calculated for matrix points that lie within the currently defined WINDOW.

Typing <CTRL/C> (Pressing the Ctrl and C keys together) aborts the calculation and display of a shaded relief map.

Messages:

The following error messages are specific to the SHADE command.
*** ERROR *** Specifying command SHADE
Output medium not specified
Use ENABLE GRAPHICS or FILEOUT commands

*** ERROR *** Specifying command SHADE
No input DTI file had been specified

Examples:

SLOPES>**SHADE<CR>**
SLOPES>

SHOW ANNOTATION

Shows information about legend and user annotation information.

FORMAT: **SHOW ANNOTATION**

Command parameters: None

DESCRIPTION:

Outputs details of the label and legend positions, and label size to the terminal.

Messages:

The following error messages are specific to the SHOW command.

*** ERROR *** specifying command SHOW
SHOW qualifiers are ANNOTATION, ASPECT, ENABLE, FILES,
HEIGHT_DIFFERENCE, SHADE or SLOPE

Examples:

SLOPES>**SHOW ANNOTATION<CR>**

Current label setting:
Position: X 150 Y 75 Size 2
Current legend setting:
Position: X 50 Y 150

SLOPES>

SHOW ASPECT

Displays information about the current aspect parameter settings.

FORMAT: **SHOW ASPECT**

Command parameters: None

DESCRIPTION:

Information about the current aspect parameter settings is displayed on the terminal.

Messages:

The following error messages are specific to the SHOW command.

```
*** ERROR *** specifying command SHOW
SHOW qualifiers are ANNOTATION, ASPECT, ENABLE, FILES,
HEIGHT_DIFFERENCE, SHADE or SLOPE
```

Examples:

SLOPES>**SHOW ASPECT<CR>**

```
Z limits          : -32766 to 32767
Angular step      : 30
Look up table     : LSL$LOOKUP:SLOPES.DAT
Aspect colour image selected

SLOPES>
```

SHOW ENABLE

Shows the current status of those options that may be enabled by means of the ENABLE command, or disabled using the DISABLE command.

FORMAT: **SHOW ENABLE**

Command parameters: None

DESCRIPTION:

Displays the current status of all the SLOPES options that may be enabled or disabled using the ENABLE and DISABLE commands.

The name of the option is shown, followed by either the word ON or OFF to indicate its current status.

If the command SHOW ENABLE is used before any ENABLE or DISABLE commands have been given, the default status of the options is displayed.

Messages:

The following error messages are specific to the SHOW command.
SHOW qualifiers are ANNOTATION, ASPECT, ENABLE, FILES,
HEIGHT_DIFFERENCE, SHADE or SLOPE

Examples:

SLOPES>**SHOW ENABLE<CR>**

Current status:

ABSOLUTE	On	AVERAGE	On	CLASSIFY	Off
CLEAR	On	GRAPHICS	Off	GREYSCALE	Off
LEGEND	On	NULL_BORDER	Off	PERCENT	Off
RECENTRE	On	TABLE	Off	TEXT	On

SLOPES>

SHOW FILES

Displays information about the currently selected input and output files.

FORMAT: **SHOW FILES**

Command parameters: None

DESCRIPTION:

This command displays information about the currently selected input and output DTI files.

Details extracted from the header of the input DTM are displayed on the terminal, along with details of the current window or area of interest.

The header values are shown in the current units of measurement. This is dependent on the header type of the input file, or may be set explicitly using the UNITS command. By default or if the ENABLE ABSOLUTE command has been given then metre or projection values are expressed in absolute values.

Messages:

The following error messages are specific to the SHOW command.

*** ERROR *** specifying command SHOW
SHOW qualifiers are ANNOTATION, ASPECT, ENABLE, FILES,
HEIGHT_DIFFERENCE, SHADE or SLOPE

Examples:

SLOPES>**SHOW FILES<CR>**

```
File    :  LSL$DTI:matrix_test.DTI
Header  :  LSLA  Data:  WORD

Units are metres

Matrix Coverage   SW:  450000.00   80000.00    NE:  460000.00   90000.00
Matrix Window     SW:  450000.00   80000.00    NE:  460000.00   90000.00
Matrix Interval    E:      50.00                N:     100.00
Value Range         :-32766   to  32767

File LSL$DTI:MATRIX_TEST.SLP selected for output
SLOPES>
```

SHOW HEIGHT_DIFFERENCE

Displays information about the current height difference parameter settings.

FORMAT: **SHOW HEIGHT_DIFFERENCE**

Command parameters: None

DESCRIPTION:

Information about the current height difference parameter settings is displayed on the terminal.

Messages:

The following error messages are specific to the SHOW command.

```
*** ERROR *** specifying command SHOW
SHOW qualifiers are ANNOTATION, ASPECT, ENABLE, FILES,
HEIGHT_DIFFERENCE, SHADE or SLOPE
```

Examples:

SLOPES>**SHOW HEIGHT_DIFFERENCE<CR>**

```
Calculation of average height difference selected
Z limits          : -32766 to 32767
Value range       : 0 to 60
Z Scale           : 1
Number of steps   : 12
Look up table     : LSL$LOOKUP:SLOPES.DAT
Step upper limits :  5 10 15 20 25 30 35 40 45 50
                  : 55 60
Colour indices    :  2  3  4  5  6  7  8  9 10 11
                  : 12 13

SLOPES>
```

SHOW SHADE

Displays information about the current shade parameter settings.

FORMAT: **SHOW SHADE**

Command parameters: None

DESCRIPTION:

Information about the current shade parameter settings is displayed on the terminal.

Messages:

The following error messages are specific to the SHOW command.

```
*** ERROR *** specifying command SHOW
SHOW qualifiers are ANNOTATION, ASPECT, ENABLE, FILES,
HEIGHT_DIFFERENCE, SHADE or SLOPE
```

Examples:

SLOPES>**SHOW SHADE<CR>**

```
Sun position   : 315.0
Sun angle      : 45.0
Reflectance map: 2
Z Scale        : 1
Look up table  : LSL$LOOKUP:GREY.DAT

SLOPES>
```

SHOW SLOPE

Displays information about the current slope parameter settings.

FORMAT: **SHOW SLOPE**

Command parameters: None

DESCRIPTION:

Information about the current slope parameter settings is displayed on the terminal.

Messages:

The following error message is specific to the SHOW command.

```
*** ERROR *** specifying command SHOW
SHOW qualifiers are ANNOTATION, ASPECT, ENABLE, FILES,
HEIGHT_DIFFERENCE, SHADE or SLOPE
```

Examples:

SLOPES>**SHOW SLOPE<CR>**

```
Gradient algorithm: 1
Z limits          : -32766 to 32767
Z Scale           : 1
Gradient range    : 0 to 60
Gradient units    : Degrees of Slope
Number of steps   : 12
Look up table     : LSL$LOOKUP:SLOPES.DAT
Step upper limits :  5 10 15 20 25 30 35 40 45 50
                  : 55 60
Colour indices    :  2  3  4  5  6  7  8  9 10 11
                  : 12 13

SLOPES>
```

SLOPE

Calculates and displays a slope map.

FORMAT: **SLOPE [algorithm]**

Command parameters:

algorithm

The index number of the slope algorithm to be used. A value in the range 1 to 5 is required. The 5 slope algorithms are as follows:-

```
1              = 3 by 3 local operator (default)
2              = Average of 2 facet normals for 2 by 2 matrix
3              = Average of 4 facet normals for 2 by 2 matrix
4              = Average of 4 facet normals for 3 by 3 matrix
5              = Maximum slope computed using 4 facet normals
                 for 3 by 3 matrix
```

If the algorithm is omitted then the last selected algorithm is used.

DESCRIPTION:

This command calculates a slope map indicating the gradient at each DTM node and using the specified algorithm. Further details of the SLOPE algorithms is contained in the description section.

Output is displayed on the graphics screen if ENABLE GRAPHICS command has been given and/or to a DTI file if the FILEOUT command has been issued.

If output is to a DTI file then if the ENABLE CLASSIFY has been given each classified index, as defined by the STEP and COLOUR commands are output to the DTI file. Otherwise the actual calculated slope values are output. On completion of SLOPE calculation, details of the DTI file produced are output to the terminal.

Values are calculated in degrees unless the ENABLE PERCENT command has been given, in which case values are output as percentage slope. The ZSCALE command may be used to increase the slope values by the specified factor.

SLOPE information is only calculated for the currently defined WINDOW.

Typing <CTRL/C> (Pressing the Ctrl and C keys together) aborts the display of and calculation of slope.

Messages:

The following error messages are specific to the SLOPE command.

*** ERROR *** Specifying command SLOPE

Output medium not specified
Use ENABLE GRAPHICS or FILEOUT commands

*** ERROR *** Specifying command SLOPE
No input DTI file had been specified

*** WARNING *** Specifying command SLOPE
Gradient algorithm should be in the range 1 to 5
Current gradient algorithm will be used

Examples:

SLOPES>**SLOPE 1<CR>**
SLOPES>

SPAWN

Allows a DCL sub-command to be spawned within the SLOPES program.

FORMAT: **SPAWN command**

Command parameters:

command

The DCL command which is to be spawned. A Valid DCL command is required.

DESCRIPTION:

SPAWN allows a DCL command to be obeyed from within the SLOPES program. This is to allow file management operations such as DIRECTORY, DELETE and RENAME to be carried out during program execution.

Messages:

The following message is specific to the SPAWN command.

*** ERROR *** Specifying command SPAWN
SPAWN requires a valid DCL command

Examples:

SLOPES>**SPAWN DIRECTORY LSL$DTI<CR>**
Directory DUA0:[DTI]

AFRICA.DTI;2 LS41.DTI;1 WALES.DTI;3 TEST.DTI;13

Total of 3 files

SLOPES>**SPAWN RENAME AFRICA.DTI SOUTHAMERICA.DTI<CR>**
SLOPES>

STEP

Defines a series of value step intervals.

FORMAT: STEP values[...]

Command parameters:

values

One or more step interval values. Up to 7 integer values may be specified. At least one value is required.

DESCRIPTION:

This command divides the calculated values for the SLOPE and HEIGHT_DIFFERENCE commands, into a series of steps which are displayed in different colours. A single value denotes a constant step interval, while a variable step interval may be defined by entering up to 7 values. A particular colour index may be associated with each step by means of the COLOUR command.

The default value is a step of 5.

The colour index that will be associated with each step may by examined by means of the command SHOW SLOPE or SHOW HEIGHT_DIFFERENCE.

Messages:

The following messages are specific to the STEP command.

```
*** ERROR *** Specifying command STEP
Command requires between 1 and 7 integer values

*** ERROR *** Specifying command STEP
Command requires integer values in the range 1 to "upper-value"

*** WARNING *** Value range exceeded
step "stepnumber" is truncated to "DTI-value"
```

Examples:

SLOPES>**STEP 10<CR>**
SLOPES>**STEP 1 2 3 5 10 10 10 <CR>**
SLOPES>

SUN_ANGLE

Sets the sun angle for SHADE calculations.

FORMAT: **SUN_ANGLE angle**

Command parameters:

angle

The angle of elevation in degrees of the sun. A real value in the range 0 to 90 degrees is required. The default angle is 45 degrees.

DESCRIPTION:

This command is used to set the angle of elevation of the light source for SHADE calculations.

Reducing the sun angle tends to increase the slope component, and is therefore useful for emphasising slope in less mountainous areas.

The current value of the SUN_ANGLE may be shown by using the SHOW SHADE command.

Messages:

The following messages are specific to the SUN_ANGLE command.

```
*** ERROR *** Specifying command SUN_ANGLE
Command requires a real value between 0 and 90
```

Examples:

SLOPES>**SUN_ANGLE 30<CR>**
SLOPES>

SUN_POSITION

Sets the sun position for SHADE calculations.

FORMAT: SUN_POSITION angle

Command parameters:

angle

The sun position or azimuth measured in degrees clockwise from north. A real value in the range 0 to 360 degrees is required. The default angle is 315 degrees (North-West).

DESCRIPTION:

This command is used to set the position of the light source for SHADE calculations. The default value of 315 degrees is chosen because it represents normal visual perception of the terrain.

It may be useful to change the sun position to show more clearly terrain features lying in particular direction.

The current value of the SUN_POSITION may be shown by using the SHOW SHADE command.

Messages:

The following messages are specific to the SUN_POSITION command.

*** ERROR *** Specifying command SUN_POSITION
Command requires a real value between 0 and 360

Examples:

SLOPES>**SUN_POSITION 45<CR>**
SLOPES>

UNITS

Specifies the units of measurement that will be used when defining an area of interest in the input DTI file.

FORMAT: UNITS units

Command parameters:

units

A keyword defining the measurement units, chosen from:

MATRIX	Matrix grid interval units, i.e rows and columns
METRES	Metre offsets from the south west corner of the DTI file
SECONDS	Latitude and Longitude in seconds of arc
LATLONG	Latitude and Longitude in degrees, minutes and seconds
PROJECTION	Projection Record Units (eg. mms on the source document)

DESCRIPTION:

The UNITS command defines the units of measurement that will be used when defining an area of interest in the currently selected DTI file by means of the WINDOW command.
The command also controls in what format details from the header of the DTI file are displayed, when the SHOW FILES command is given.

The command should be given after defining the an DTI file since an appropriate default units of measurement is set up when the file is opened.

Messages:

The following messages are specific to the UNITS command.

```
*** ERROR *** Specifying command UNITS
Command should be followed by MATRIX, METRES, LATLONG, SECONDS or PROJECTION
Current setting is "units"
```

```
*** ERROR *** Specifying command UNITS
Command qualifier is invalid for the input file
```

Examples:

SLOPES> **UNITS MATRIX<CR>**
SLOPES>

VALUE_RANGE

Defines the range of values that will be displayed.

FORMAT: **VALUE_RANGE lower_value upper_value**

Command parameters:

lower_value

The lower value of the required range of values.

upper_value

The upper value of the required range of values.

Two integer arguments are required.

DESCRIPTION:

The VALUE_RANGE command is used to define the range of values output when the SLOPES or HEIGHT_DIFFERENCE commands are given.

Cells with values outside the range appear in black on the displayed image. They are output as null values in the output DTI file.

Messages:

The following message is specific to the VALUE_RANGE command.

*** ERROR *** Specifying command VALUE_RANGE Lower value exceeds upper value

---

Examples:

SLOPES> **VALUE_RANGE 5 15<CR>**
SLOPES>

WAIT

Suspends the program for a specified number of seconds.

FORMAT: WAIT interval

Command parameters:

interval

The time interval in seconds. A real value is required.

DESCRIPTION:

This command suspends program execution for the specified time interval. It is useful for demonstration purposes if SLOPES is being run from a command file.

Messages:

The following messages are specific to the WAIT command.

```
*** ERROR *** Specifying command WAIT
Command requires a real in the range 1 to 1000
```

Examples:

SLOPES>**WAIT 5.0<CR>**
SLOPES>

WINDOW

Specifies an area of interest in the currently selected DTI file.

FORMAT: **WINDOW xmin ymin xmax ymax**

Command parameters:

xmin ymin
: The coordinates of the bottom left hand corner of the defining rectangle.

xmax ymax
: The coordinates of top right hand corner of the defining rectangle.

The format of the command parameters is controlled by the UNITS command.

UNITS MATRIX - Requires 4 integer values defining the rectangle in terms of column and row numbers

UNITS METRES - Requires 4 real (floating point) values defining the rectangle as metre offsets from the SW corner of the DTI file. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates (eg. U.K. National Grid coordinates); if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

UNITS SECONDS - Requires 4 real (floating point) values defining the absolute position of the rectangle in seconds of arc. The values are supplied in the order latitude followed by longitude for the SW corner, and latitude followed by longitude for the NE corner. A negative latitude value indicates a position in the Southern Hemisphere, and a negative longitude value a point to the west of Greenwich.

UNITS LATLONG - Requires 4 values defining the absolute latitude and longitude position of the rectangle in the format DDD MM SS H where DDD is the number of degrees, MM is the number of minutes, SS is the number of seconds, and H is the hemisphere (N,S,E,W). The values are supplied in the order latitude followed by longitude for the SW corner, and latitude followed by longitude for the NE corner.

UNITS PROJECTION Requires 4 real (floating point) values defining the rectangle in projection record units. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates; if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

Note that in all cases, the input values are adjusted to the nearest column and row values.

DESCRIPTION:

The WINDOW command is used to limit display to a particular rectangular geographical area. Only nodes in the DTI file that lie within this area are drawn when the DISPLAY command is given.
The area of interest should lie within the geographical bounds of the DTI file.

If the logical name LSL$MATRIX_DEFAULT_WINDOW is defined with a value of "1", then a default area of interest defined in matrix units of bottom left hand corner 1,1 and top right hand corner 200,200 is set when the file is opened. If the logical name is absent or has any other value, or if the input DTI file has fewer than 200 columns or 200 rows, then a default window equivalent to the total matrix is set.

If a digitising table is available, and a map has been registered to the DTI file, the coordinates of the window may be defined using any button on the table puck.

Messages:

The following messages are specific to the WINDOW command:

*** ERROR *** Specifying command WINDOW
Command not available during editing operation

*** ERROR *** Specifying command WINDOW
Command requires 4 coordinate values

*** ERROR *** Specifying command WINDOW
DTI file has not yet been opened

*** ERROR *** Specifying command WINDOW
NE coordinates should exceed SW coordinates

*** ERROR *** Specifying command WINDOW
Unable to use supplied LAT LONG values

*** WARNING *** Specified window exceeds matrix limits
Window will be truncated to fit

Examples:

SLOPES>**WINDOW 1 1 200 200<CR>**
SLOPES>**UNITS LATLONG<CR>**
SLOPES>**WINDOW 42 00 00N 3 00 00E 42 30 00N 2 58 40E**
SLOPES>

ZLIMITS

Sets height limits on SLOPE, ASPECT and HEIGHT_DIFFERENCE calculation.

FORMAT: **ZLIMITS lower_height upper_height**

Command parameters:

lower_height

The lower height limit. An integer value is required.

upper_height

The upper height limit. An integer value is required.

DESCRIPTION:

The ZLIMITS command restricts the calculation of SLOPE, ASPECT and HEIGHT_DIFFERENCE information to those DTM nodes that lie within the specified height limits.

Cells with heights outside the range appear in white on the displayed image. They are output as null values in the output DTI file.

The default range is the minimum and maximum values in the current in the current DTI file.

Messages:

The following error messages are specific to the ZLIMITS command:

*** ERROR *** Specifying command ZLIMITS
Command requires 2 integer arguments

*** ERROR *** Specifying command ZLIMITS
Upper height value must exceed the lower height value

Examples:

SLOPES>**ZLIMITS 50 250CR>**
SLOPES>

ZSCALE

Defines the vertical exaggeration that is applied to all DTM heights when generating a slope, height difference or shaded relief map.

FORMAT: **ZSCALE scale_factor**

Command parameters:

scale_factor

Scale_factor defines a z scaling factor that is applied to all DTM height values. A real value is required.

DESCRIPTION:

The ZSCALE command defines a vertical exaggeration (z scale) factor. This is a value by which all DTM height values are multiplied before a slope, aspect or shaded map is generated.

By default no vertical exaggeration is applied ie. a z scale factor of 1 is used.

Messages:

The following error messages are specific to the ZSCALE command:

```
*** ERROR *** Specifying command ZSCALE
Command requires 1 real argument
```

Examples:

SLOPES>**ZSCALE 2.5**
SLOPES>

MESSAGES (OTHER)

In addition to messages which are generated by the program itself, other messages may be produced by Laser-Scan libraries. In particular, messages may be generated the DTI library, and by the Laser-Scan I/O library, LSLLIB.

DTI library messages are introduced by '%DTI', and are documented in the DTILIB Reference Manual. In all cases the messages indicate a fatal error, that will cause processing to halt.

LSLLIB messages are introduced by '%LSLLIB' and are generally self-explanatory. Such messages rarely indicate a fatal error, and are generated most frequently by entering a command in an invalid format in response to the SLOPES> prompt.

CHAPTER 8

MODULE ROVER

MODULE ROVER

Combined Raster and Vector Display Program

Issue 3.0 - 16-March-1989

Science Park, Milton Road, Cambridge, England CB4 4FY tel: (0223) 420414
Document "ROVER Reference" Category "REFERENCE"

Document Issue 3.0	M W S Reid	16-March-1989
Document Issue 2.1	M W S Reid	23-March-1988
Document Issue 2.0	M W S Reid	28-January-1988
Document Issue 1.3	M W S Reid	13-January-1987
Document Issue 1.2	M W S Reid	21-November-1986
Document Issue 1.1	D R Catlow	24-September-1986
Document Issue 1.0	D R Catlow	20-August-1986

MODULE ROVER

FUNCTION

ROVER displays both grid based and vector geographic data, and allows a user to interact with the displayed information. It forms part of the Laser-Scan TVES Package.

The grid and vector data may be displayed independently, but more importantly they may also be displayed together using the program. The user is able to derive considerable benefits from this integrated display:

1. in the areas of data validation and visualisation

2. through the derivation of other geographic information from the combined datasets.

Four modes of operation are available in ROVER:

1. Normal Display Mode.

2. Vector Digitising Mode.

3. Raster Editing Mode

4. Registration Mode

FORMAT

$ ROVER

PROMPTS

ROVER is an interactive, command driven program. Command input is expected when certain prompts are issued:

The main prompts reflect the four primary modes of operation. In the main Display mode the prompt is:

    Rover>

In EDIT mode:

    Edit>

In DIGITISE mode:

Digitise>

In REGISTRATION mode:

Registration>

At other times the prompt may change depending on program status. Certain commands may also prompt for values - the prompt used depends upon the answer required.

DESCRIPTION

INTRODUCTION

ROVER is a versatile program which allows both grid based raster and vector geographic data to be displayed and manipulated. It forms part of the Laser-Scan TVES package. Four main modes of operation are available:

1. Display Mode

   This is the normal mode of Rover operation. Both raster and vector datasets may be displayed singularly or in combination.

2. Digitise Mode

   Digitise Mode is activated with the command ENABLE DIGITISE. It is only available if a digitising table is connected to the TVES workstation.

   This mode is useful for digitising information which is derived from the displayed raster data, or from a combination of the raster and vector datasets. See the DIGITISING COMMANDS section for further information on Digitise mode.

3. Edit Mode

   Edit mode is activated with the command ENABLE EDIT. The raster DTI files may be edited. Various methods of editing are available, together with an interactive flooding facility. Editing may take place with reference to the overlaid vectors. See the RASTER EDITING COMMANDS section for further information on Edit mode.

4. Registration Mode

   Registration mode is activated with the command ENABLE REGISTRATION. Options are provided in this mode to allow Registration Control Point (RCP) files to be created and modified. The RCP files created in ROVER may be used for map/image registration. See REGISTRATION COMMANDS for further information on Registration mode facilities.

If your workstation incorporates a digitising table and button puck, then it is possible to register a source document such as a map to the raster and vector datasets, and to invoke the DIGITISE mode of operation.

If the logical name LSL$AUTO_ENABLE_TABLE is defined with a value of "1" the program will attempt to initialise the Laser Scan Table Monitor Utility on startup. If the logical name is absent or has any other value, no attempt is made to initialise the table monitor on startup, however the table can be subsequently initialised using the command ENABLE TABLE.

The Table Monitor controls reading from the digitising table. It must be set up on your system if ROVER is to take input from the digitising table. If table initialisation fails then the message:

```
*** WARNING *** Failed to Initialise table monitor
ROVER will assume no table is available
```

will be output before the prompt **ROVER>** is displayed on the terminal. In the event of failure, input from the digitising table or puck button is not possible, and the program will accept commands only from the terminal.

Table initialisation will generally fail because no Table Monitor is currently active, or because the Table Monitor is locked by another user. If neither of these reasons appear to apply, you should consult the TABLIB Reference Manual, or seek guidance from your system manager.

Note the buttons on the table puck may have different meanings depending on the mode of operation. The following sections describe the operation of the program, and each mode of operation. Each individual command is described in detail in the COMMANDS section.

DISPLAY CONCEPTS

- **Pictures and Planes**

ROVER uses an 8 bit-planes raster display device. In normal operation these planes are divided into two contiguous sets of planes known as Pictures. The first set of planes forms Picture 1, and is used for the display of raster or grid data. The second set of planes forms Picture 2, and is used for the drawing of vector information. The two Pictures may be selected and cleared independently.

You may select or deselect either of the Pictures using the DISABLE/ENABLE PICTURE commands.

eg. DISABLE PICTURE 1

will deselect Picture 1. This causes any image displayed in Picture 1 to be set 'invisible'. It may be set 'visible' again by giving the command ENABLE PICTURE 1. Picture 2 may be manipulated in a similar way. You should note that deselecting one Picture, will cause the other Picture to be selected. It is therefore not possible (and presumably not very useful), to deselect both Pictures.

Both Picture 1 and 2 may be cleared selectively by means of the CLEAR PICTURE command.

eg. CLEAR PICTURE 2

will clear Picture 2 only. Clearing erases the contents of the Picture, so that the image in the Picture can only be recovered by replotting.

It is possible to clear both Pictures together using the command CLEAR with no qualifier.

By default ROVER will allocate 5 planes to Picture 1 and the remaining 3 planes to Picture 2. You may vary the allocation of planes between Pictures by using the PLANES command. PLANES allows you to define the number of bit-planes that will be used for Picture 1. The remaining planes are always allocated to Picture 2.

The number of bit-planes allocated to each Picture determines the number of available raster display and vector overlay colours.

	Picture 1	Picture 2
PLANES 5	32 colours	7 overlay colours
PLANES 6	64 colours	3 overlay colours
PLANES 7	128 colours	1 overlay colours
PLANES 8	256 colours	0 overlay colours

You are limited to a maximum of 7 overlay colours, so consequently a PLANES setting with a value less than 5 is not allowed.

The number of colours you may require for Picture 1, will vary with the nature and application of the grid data. For elevation data 32 or 64 colours may be adequate to produce a segmented or layered image, while for a single channel of satellite imagery you may require 128 or perhaps the maximum 256 colours. If 256 colours are selected no overlay colours are available. It is always necessary to strike a balance between the grid display and vector overlay requirements.

For a number of applications you may require less than 32 display colours, or may wish to use less than the number of colours available with the current PLANES setting. The number of colours used for display in Picture 1 may be set using the COLOURS command.

    eg. COLOURS 16

tells the program to display the information in Picture 1 using just the first 16 colours in the currently selected colour table. The value supplied with the COLOURS command may not exceed the number of possible colours available with the current PLANES setting. It should also be noted, that giving the PLANES command will always set the number of display colours to the maximum available for the number of Picture 1 planes.

You may control which group of colours in the colour table are used for display, using the FIRST_COLOUR command. By default the colours defined at the start of the colour table are used, thus COLOURS 16 will select the first 16 colours defined in the colour table. If however you give the command

    FIRST_COLOUR 4

then the 16 colours starting at entry 4 in the colour table will be selected.

COLOURS and FIRST_COLOUR may be controlled for each open DTI file independently. By default FIRST_COLOUR is set to 3, which is the first entry in the default colour tables following black and white.

The COLOURS and FIRST_COLOUR commands may be used to good effect in quartered screen mode, allowing for example a colour image to be displayed alongside a monochrome image.

o **Colour Tables**

6 colour tables are supplied with the program. Four of these tables are designed to be utilised with the four available PLANES setting, and the appropriate colour table is read in when the PLANES command is given. The colour tables associated with each PLANES setting are listed below:

	Colour Table Name	Description
PLANES 5	ROVER5	32 display colours 7 overlay colours (white, red, green, blue, brown, orange, yellow)
PLANES 6	ROVER6	64 display colours 3 overlay colours (white, red, blue)
PLANES 7	ROVER7	128 grey levels 1 overlay colour (white)
PLANES 8	ROVER8	256 grey levels

2 special colour tables that incorporate both a set of colours and grey-levels, ROVERG5 and ROVERG6 are also provided. These enable the simultaneous display of a colour and monochrome image in quartered screen mode, and provide 7 and 3 overlay colours respectively.

You may edit the colour representations in these tables using the system text editor. Alternatively, you may wish to define your own special colour tables, and select these from within ROVER. Colour table selection is achieved using the LUT command.

eg. LUT MYTABLE

will select and read the colour table 'MYTABLE.' The supplied filename is parsed against the default 'LSL$LOOKUP: .DAT'. The effect of reading a new set of colour tables on any displayed information will take effect immediately.

It should be noted that if the PLANES command is now given, then the definitions in 'MYTABLE' will be overwritten by one of the default tables supplied above. The LUT command therefore to be effective, should be issued after a PLANES command.

In the supplied colour tables the following colour definition is present:

	Red	Green	Blue	Blink	
Colour 1	0	0	0	0	!black

ROVER assumes that colour 1 is always defined in this way. Selective quadrant and text clearing are achieved by use of colour 1. Legend and border text are usually displayed in Colour 2 of the table. It is recommended that this colour should be defined to white for the text to show clearly:

	Red	Green	Blue	Blink	
Colour 2	FF	FF	FF	0	!white

User annotation is always written to Picture 2, and by default uses the first overlay colour defined in the colour table.

The default colour table 'ROVER5' is listed in Appendix 1. This illustrates how you should set up a colour table in order to obtain the correct number of overlay colours. Note that the first 32 colours in the table are the colours used for display in Picture 1, and how these are followed by the definitions for 7 overlay colours, with each overlay colour being defined by a block of 32 values. The table below defines the structure of tables appropriate to different PLANES settings.

	Picture 1 (Display Colours)	Picture 2 (Overlay Colours)
PLANES 5	32	32,32,32,32,32,32,32
PLANES 6	64	64,64,64
PLANES 7	128	128
PLANES 8	256	

o **Full and Quartered Screen Display**

By default, ROVER will assume you wish to display a single image on the screen using the full screen. In full screen mode an area of the screen is reserved for image display. The remaining areas of the screen are used for legend and border text.

```
 _________________________________
|       ....................... |
|       :                     : |
|       :                     : |
| leg-  :                     : |
| end   :        IMAGE        : |
|       :                     : |
|       :       DISPLAY       : |
|       :                     : |
|       :                     : |
|       :                     : |
|       :.....................: |
|                               |
|           Border Text         |
 ---------------------------------
```

The display image is automatically scaled to fill as much of the image display area as possible. If you select an area of interest in the grid file that is larger than the display area, then the image is still displayed but the output of a legend, border and border text will be disabled, and the image will be centred within the full screen area. Unless the SAMPLE and ENABLE SAMPLE commands are used, a raster image larger than the screen size may not be displayed in full with a single DISPLAY command.

It is possible to select quartered screen mode by using the ENABLE DIVIDE command. In quartered screen mode, the screen is divided into 4 quadrants, allowing the display of 4 images. The quartered screen is arranged as shown below.

```
 ________________________________
|               :               |
|               :               |
|       1       :       2       |
|               :               |
|               :               |
|               :               |
|...............:...............|
|               :               |
|               :               |
|               :               |
|       4       :       3       |
|               :               |
|               :               |
 --------------------------------
```

As in full screen mode, each quadrant is divided into an image display, legend and border text area. These are arranged as for the full screen, but are all half the size.

Each quadrant is identified by means of its quadrant number. On selecting quartered screen mode, the current quadrant is 1 (ie. the top left). If the DISPLAY command is now given the image will be drawn in this quadrant. The current quadrant is incremented by 1 each time you give the DISPLAY command. After displaying in quadrant 4, the current quadrant number becomes 1 again, and on giving the DISPLAY command quadrant 1 is first cleared. You are therefore able to move clockwise around the quadrants without the need to issue the CLEAR QUADRANT command between DISPLAY commands.

The vector information which is displayed by means of the OVERLAY command, is by default drawn in the quadrant that last received a raster image. Using the OVERLAY command, unlike the DISPLAY command, does not update the current quadrant number, so issuing the OVERLAY command a second time will draw the information in the same quadrant.

You may select which quadrant to receive either the raster image or vector information using the QUADRANT command.

    Eg. QUADRANT 3

will select the bottom right quadrant.

It is possible to clear a single quadrant using the CLEAR QUADRANT command. This command requires a quadrant number defining which quadrant is to be cleared.

Eg. CLEAR QUADRANT 4

will clear the bottom left quadrant.

Quartered screen mode is useful for displaying the same grid data in 4 different ways or with 4 different overlays, or for displaying a number of different data alongside each other.

- **Zoom and Roam**

In both full and quartered screen mode the raster image, and consequently any vector information overlayed on the image, is automatically scaled to fill as much of the image display area as possible. In the case of the raster image this is achieved by writing a single DTI cell of information as a number of cells or pixels on the screen.

In addition to this however, you may also enlarge the screen image using commands that invoke the hardware pixel replication functions of certain devices. Using the command ENLARGE, you are able to enlarge the screen image by up to a factor of 16. Full magnification of the display image is achieved by repeatedly giving the ENLARGE command.

In full screen mode the image is always enlarged around the centre of the display image. In quartered screen mode, the display is enlarged around the centre of the image displayed in the currently selected quadrant, and the display is also recentred, so that the enlarged quartered screen image will always appear in the centre of the 'enlarged' display. By this means it is possible to alternate between 4 quartered screen images on the 'enlarge' display, by simply giving a series of QUADRANT commands.

REDUCE has the effect of cancelling an ENLARGE command, and may be given repeatedly until the original display is regained. In quartered screen mode a reduction back to the minimum magnification will result in the 4 quadrants being displayed in their original form.

In order to change the point around which the display is enlarged or reduced, you should use the CENTRE command. The CENTRE command requires a matrix x and y coordinate value, and results in the displayed image being recentred around the screen pixel that corresponds to this coordinate position. By repeatedly issuing the CENTRE command with different matrix coordinate values, it is possible to roam around the display image at the current enlargement factor. The command CENTRE is generally used in combination with source document registration (see below).

On issuing the DISPLAY command the screen is always recentred, ie. the effect of any CENTRE command is negated.

The ENLARGE, REDUCE and CENTRE commands may be given from the terminal, but in order to make them easier to use they have also been defined as puck button commands.

Button 1	CENTRE
Button 2	ENLARGE
Button 3	REDUCE

These commands may be given using a puck button only if the digitising table has been successfully initialised by the program on start up or with the ENABLE TABLE command. If the table has been initialised, the commands ENLARGE and REDUCE are always available from the puck as well as from the terminal. CENTRE requires that a source document is registered to the current DTI file using the SETUP MAP command. Following map registration, pressing puck button 3 inside the map area and inside the area corresponding to the current area of interest (ie. DTI WINDOW), will recentre the display around the screen point corresponding to the puck DTI coordinate position.

It should be noted that when using the ENLARGE, REDUCE and CENTRE commands, that all the information drawn on the display, including the raster image, vector overlay, legend, border text and any user annotation will be affected.

A ZOOM command, which does not require hardware pixel replication facilities, is also provided. This operates by resetting the display window according to the specified zoom factor and automatically redisplaying the image.

- **Image Sub-Sampling**

  In order that a raster image which has more columns or rows than pixels on the screen may be displayed, ROVER provides an image sub-sampling option. The two commands associated with this option are **SAMPLE** and **ENABLE SAMPLE**.

  SAMPLE allows the user to define the rate at which nodes will be sampled along the columns and rows of the DTI file, while ENABLE SAMPLE enables the use of the defined sample intervals. Using this command it is possible for the user to control whether sub-sampling is applied, and for ROVER itself to enable or disable sub-sampling according to the mode of operation. The command ENABLE REGISTRATION automatically enables sub-sampling, while ENABLE EDIT will disable it, since it is not possible to edit a sub-sampled matrix.

  If sub-sampling is selected, ROVER will reset the sample intervals automatically if the rectangular display window, specified explicitly using the WINDOW command or implicitly using the ZOOM command, is to large to be displayed with the current sample values, in the area of the screen reserved for raster display.

  Use of the SAMPLE and ENABLE SAMPLE commands is recommended when an overall view of a large image is required, or in order to generate a quick look view of a particular image.

RASTER DISPLAY

- **Introduction**

  ROVER is able to display in colour coded form any grid information held within a Laser-Scan DTI file format. The gridded data may consist of a variety of geographic values eg. elevation, slope or radiometric information.

  The grid information is always drawn to Picture 1. It may be selected and redrawn independently from any vector information output to Picture 2. The number of colours available for display may be selected by means of the COLOURS command, or by default is the maximum number of colours possible with the current plane allocation. The number of planes allocated to Picture 1 is controlled using the PLANES command. Upon program initialisation 5 planes (32 colours) are allocated to Picture 1.

- **File and Window selection**

  Before any grid data may be displayed, a DTI file must be specified and successfully mapped into memory. This is achieved by means of the FILEIN command. FILEIN requires a filename, which is parsed against the default 'LSL$DTI:DTI.DTI'.

  eg. FILEIN TEST

  opens the file LSL$DTI:TEST.DTI for read-only

  If editing operations are to be carried out on the file then the command UPDATE_FILEIN should be used to open the file for write access.

  Up to 4 DTI files may be opened. A different number is assigned to each file. The first file specified is assigned number 1; the second number 2, and so on. This number is the means by which you may select which file is current, using the SELECT FILEIN command. By default the file opened with the latest FILEIN command is the current file.

  If 4 DTI files are opened, then it is necessary to close at least one of the files using the CLOSE FILEIN command, before specifying a further DTI file.

  The SELECT FILEIN command may be used to make a DTI file current. The command requires a number in the range 1-4.

  eg. SELECT FILEIN 1

  selects the DTI file assigned with the number 1

  The commands DISPLAY, UNITS, WINDOW, STEP, RANGE, COLOURS and FIRST_COLOUR operate only on the currently selected DTI file.

A UNITS setting appropriate to the DTI type is set for each open file. The UNITS setting associated with the currently selected DTI file, may be changed using the UNITS command.

ROVER handles byte, word, longword or real datatype DTI files, and files with TED4, UHL1, or LSLA headers. The DTILIB Reference Manual in Package MATRIX contains full details on the structure of DTI files.

An area of interest (DTI WINDOW) may be defined using the WINDOW command. The command takes 4 values specifying the SW x and y coordinate values, and NE x and y coordinate values. The NE coordinate values must exceed the SW coordinate values. If the specified window exceeds the DTI file area, a warning is generated, and the window values are truncated. The window settings for each DTI file may be examined by using SHOW FILEIN

If a source document has been registered to the DTI file, the window values may be defined using any puck button.
If the logical name LSL$MATRIX_DEFAULT_WINDOW is defined with a value of "1", then a suitable default area of interest is calculated for the available display area. If the logical name is absent or has any other value, then a default window equivalent to the total matrix is set. If the total matrix is too big for the graphics area then the largest WINDOW which may be displayed is set.

o **Display Commands**

The basic raster drawing command is DISPLAY.

Giving the DISPLAY command will display information extracted from the current selected DTI file. Only data lying within the current area of interest is displayed. The raster image is automatically scaled to fill as much of the display area as possible.

Typing <CTRL/C> (pressing the Ctrl and C keys together) after giving the DISPLAY command will abort the display of the grid information.

By default a legend consisting of a colour / value guide, and overlay colour guide is generated to the left of the screen; a border is drawn around the image, and information about the image and the current display parameters is output. The legend and border may be suppressed using the DISABLE LEGEND command, and the text information by giving the DISABLE TEXT command, prior to issuing DISPLAY.

The colour representation of the image, and the matrix values displayed are controlled by the STEP and RANGE commands, and by the commands PLANES, COLOURS and FIRST_COLOUR.

By default the value range in the DTI file is divided by the number of display colours to determine a colour/value step interval. For many applications you may wish to set a specific step interval or intervals, or reduce the number of intervals in order to increase the contrast between successive colour steps. This may be achieved by using the STEP command.

eg. STEP 100

sets a constant step interval of 100.

The STEP command may also be used to set a number of different step intervals.

eg. STEP 100 200 50

In this example the first step is defined as a 100; the second as 200, and remaining steps up to the maximum matrix value as 50. Up to 7 such values may be supplied with the STEP command.

If the step interval or intervals given with the command are of such a size that the resulting number of steps exceed the number of display colours, a warning is output on the terminal and the upper step values will appear in white.

The RANGE command defines a band within which matrix values are displayed.

eg. RANGE 100 400

In this example only matrix values that fall inclusively within the range 100 - 400 will be displayed in colour on the display screen. Values that fall outside the range are drawn in the background colour.

- **Fast Display**

If a byte data file is input then a fast display option is available. Because the data values are already in a suitable range for display (0 to 255) , these may be used directly without any intermediate calculation. This option may be particularly suitable for fast display of large areas of satellite imagery which is normally stored in byte format.

Prior to display with the FAST command the PLANES command should be given. 256 grey levels are available if PLANES 8 is selected. If the byte file contains values which are all less than 127 then PLANES 7 may be selected leaving 1 colour for vector display.

The command can be used in a similar way to the DISPLAY command but because no calculation is involved,the current STEP, RANGE, COLOURS and FIRST_COLOUR values are ignored.

- **Colour Mapping**

If many windowing and zooming commands are to be carried out during a Rover session then it is useful to specify the command ENABLE COLOUR_MAP. A temporary Rover DTI file is opened to store the colour matrix values of the latest screen display. Subsequent displays within the same area of the original window will be faster because colour lookup calculations are not required. Issue of any commands which may change the colours of the display (STEP, COLOURS, FIRST_COLOUR or RANGE) will again cause colour lookup calculations to be initially required.

Exit of the program will take longer if ENABLE COLOUR_MAP has been given because the temporary file requires to be unmapped from memory.

VECTOR DISPLAY

o **Introduction**

ROVER is able to display any vector information held within a Laser-Scan IFF file. The way in which the information is displayed is dependent upon the graphical type of the feature. The graphical type is looked up in a Feature Representation Table (FRT), and is determined on the basis of feature code. The various graphical types, and the means by which the FRT and associated Symbol Representation IFF (SRI) and Text Representation IFF (TRI) files are set up, are detailed in the FRTLIB Reference Manual. Using these lookup tables you are able to determine the graphical representation including colour, line style, area fill, symbol and text form, of overlaid features.

The vector information is drawn by default to Picture 2, which may be selected and redrawn independently from any raster image output to Picture 1. The number of available overlay colours is determined by the PLANES command. By default 7 overlay colours and the background colour are available.

o **File and Window Selection**

The IFF file containing the vector information to be drawn to Picture 2, should be selected using the IFF command. IFF requires a filename, which is parsed against the default 'LSL$IF:IFF.IFF'.

    eg. IFF EXAMPLE

opens the IFF file LSL$IF:EXAMPLE.IFF

Only 1 IFF file may be opened at any one time. If the command IFF is given a second time, the currently opened IFF file will be closed. Alternatively you may close the IFF file using the command CLOSE IFF.

The command IFFWINDOW is used to define a rectangular geographical area in the IFF file. Registration of the overlay data with the raster data is achieved using this command in combination with the WINDOW command; the corners of the IFF rectangular area are registered to the rectangular corners of the DTI current area of interest. The default IFF window is set up to achieve registration automatically provided header records, containing absolute file origin values, are present in both the DTI and IFF files. (See the MATRIX Module DTILIB for more information on DTI header records and the IFF user guide for an explanation of the IFF Map Descriptor entry.) Resetting the DTI WINDOW causes automatic recalculation of the IFFWINDOW to maintain registration. The IFF corner and window values may be examined by typing SHOW IFF.

On opening an IFF file it is assumed you are interested in all the features contained in the file. Features selected for overlay may be set using the SELECT command.

- **Feature Selection**

  You may determine which IFF features are drawn on the display by using the SELECT and DESELECT commands. Selection may be on the basis of Feature Serial Number (FSN), Feature Code (FC), Layer Number or FRT group definitions. A single value or range or values (indicated by use of '-') may be given, while delimiters may be commas or spaces. If no selection qualifier is present then FC is assumed.

  eg. SELECT FC 10-20, RIVERS 90

  In this example features with feature code 10-20, 90 or with a feature code defined in the FRT as belonging to the group 'RIVERS' are selected for overlay.

  eg. SELECT LAYER 10

  In this example only features within IFF layer 10 are selected for overlay.

  If the previous commands are both given before issuing the OVERLAY command, then an IFF feature must satisfy both criteria to be selected for overlay.

  Current selections may be examined by using SHOW SELECTIONS. A special command

  eg. SELECT ALL

  will reset any feature selections.

  DESELECT, which has the same syntax as SELECT, will allow you to negate feature selection made by means of the SELECT command, or deselect the drawing of specific features.

- **The OVERLAY command**

  The basic vector drawing command is OVERLAY. OVERLAY will draw any features currently selected and lying with the IFF area of interest (IFF WINDOW) to Picture 2. If raster information has already been displayed in Picture 1, the vector image will be scaled and registered to this image. If no information has been output to Picture 1 using the DISPLAY command, the vector image will be scaled to fill the image area.

  Typing <CTRL/C> (pressing the Ctrl and C keys together) after giving the OVERLAY command will abort the drawing of vector information.

- **Representation Tables**

  The program determines how to graphically represent the IFF feature using a Feature Representation Table (FRT) and depending on the graphical type, a Symbol Representation IFF (SRI) file and a Text Representation IFF (TRI) file. These look-up files should be defined using the FRT, SRI and TRI commands before the OVERLAY command is given.

If only a FRT is supplied before giving an OVERLAY command, the program will look for an SRI and TRI of the same name. If these files are not found then an error is generated and the OVERLAY command will be abandoned. The program will search the directory LSL$FRT for the required look up tables.

- **Overlay Colours**

The number of colours available for the overlaid vector information is controlled by the PLANES command. PLANES defines the number of bit-planes allocated to the raster background (Picture 1), and consequently the number of bit-planes allocated to Picture 2. By default 5 bit-planes are allocated to Picture 1 and 3 to Picture 2. This division of the 8 planes means that 7 overlay colours are available by default.

Other divisions of the bit-planes will give rise to a different number of overlay colours:

PLANES 5	Overlay Colours 7
PLANES 6	Overlay Colours 3
PLANES 7	Overlay Colours 1
PLANES 8	Overlay Colours 0

The maximum number of overlay colours is 7.

The exact nature (red, green, blue and blink) of the overlay colours is defined in the colour look up table (LUT). You may vary these by editing the colour table .

By default the colour in which a particular IFF feature is drawn is determined from the FRT. The feature code of the feature determines the colour index. It should be noted that if the colour index specified in the FRT is larger than the number of available overlay colours, the colour is set to the first overlay colour defined in the colour table.

Alternatively you may select an overlay colour, and override the FRT colour value. The available overlay colours are displayed in the upper part of the legend.

eg. SET COLOUR 1

This will select overlay colour 1 as defined in the colour table. If an OVERLAY command is now issued, all currently selected IFF features will be drawn in that colour. The colour index supplied with the SET COLOUR command must be valid for the current PLANES settings eg. if 2 planes are allocated to Picture 2, then the colour index must not exceed 3.

To enable the FRT colour values, you should use

SET COLOUR FRT

The command

SET COLOUR 0

has a special meaning. It sets the overlay colour to the background colour, and may be used to selectively erase overlaid features in Picture 2. You should note that if no information has been overlaid setting the colour to 0 will result in no information being drawn.

Normally the IFF information is drawn in Picture 2. A special command is available

ENABLE MASK

which forces the information to be written to picture 1 in the background colour. Using ENABLE MASK the raster background information underlying the IFF information is overwritten. In other words, the IFF information punches a hole into any raster image displayed in Picture 1.

This command is useful when an application requires that only part of the raster image eg. only terrain that is not covered by woodland, be displayed. Drawing of the IFF information to Picture 2 may be enabled again by means of DISABLE MASK.

Other commands that influence the representation of the drawn vector information are INTERPOLATE, ENABLE HATCH, ENABLE HEIGHT, ENABLE POSITIONING, SCALE and ENABLE OVERRIDE. These commands are described in the COMMANDS section.

DISPLAY ANNOTATION

- **Introduction**

  You may annotate the ROVER display with 3 types of information:

  1. Legend (Display Colour/Value Guide and Overlay Colour Guide)

  2. Border Text

  3. User Labelling

  It most cases it is possible to control the position of the annotation, and in the case of user labelling select the colour and size of labels. It is possible to suppress the different types of annotation.

- **Legend**

  The image may be annotated with a legend. By default the legend option is selected, and a legend will be generated automatically before the output of a raster image using the DISPLAY command. Output of the legend may be disabled by typing DISABLE LEGEND, and may be reselected by giving the ENABLE LEGEND command.

  The legend is in two parts:

  The lower part of the legend is a display colour/value guide, which allows the colours used in the raster display to be related to the range of values they represent. A maximum of 12 colour steps are displayed. If the number of steps defined for an image is greater than 12, the colour/value guide will be sub-sampled accordingly.

  The upper part of the legend displays the overlay colours available with the current planes setting and selected colour table. The overlay colours are labelled with their index value. This index value may be used in conjunction with the SET COLOUR and LABEL COLOUR commands to set a current overlay or annotation colour.

  For a number of applications you may wish to change the position of the LEGEND on the display screen. Using the LEGEND POSITION command, which requires a screen x and y position, you may change the position of the legend.

  It should be noted that the output of the border generated around the raster image, is also controlled by the ENABLE LEGEND command.

  The LEGEND is automatically disabled when a display window larger than can be displayed in the default display area is specified.

- **Border Text**

By default information about the displayed DTI file and current display parameters, is written below the image following output using the DISPLAY command. The output of the border text may be disabled using DISABLE TEXT and may be enabled again using ENABLE TEXT.

When selected, the name of the DTI file used to generate the image is output, along with the DTI area of interest (DTI WINDOW). The window values are output in the units currently selected for the displayed file. Details of the DTI display range set using the RANGE command, and the number of steps or colour levels, is also output.

In EDIT mode of operation output of border text is automatically disabled to allow space for editing status information to be displayed. Border text or edit status information are disabled if a DTI window larger than may be displayed in the default display area is specified.

- **User Labelling**

It is possible to annotate the display at any time with user labels. There is no restriction on the number of labels that may be generated, while the only restriction on the content of the labels is that they should not exceed 68 characters.

To output a label you should type DRAW LABELS followed by the required text string.

eg. DRAW LABEL EXAMPLE TEXT

will annotate the display with the the label 'EXAMPLE TEXT'. The position, size and colour of the label are determined by the current label settings, which may be examined using the command SHOW ANNOTATION. It is possible to change all of these label parameters.

You are able to select the position of a label using the LABEL POSITION command. This command requires an x,y screen position supplied in pixel units.

eg. LABEL POSITION 512 512

will set the current label position to the screen position 512 512.

The size of labelling is controlled using the LABEL SIZE command. Four text sizes are currently available ranging in size from 1 (smallest) to 4 (largest). The default label size is 2. The font style cannot be varied as the characters are derived not from a Text Representation IFF (TRI) file, but from a font supplied in the SIGMA firmware.

The colour of any label may be set using the LABEL COLOUR command. The number of colours available for labelling, since all labels are drawn into Picture 2, is the number of overlay colours currently available. By default labels will be drawn in the first overlay colour specified in the current colour table. In the default colour tables supplied with the program, this is white. The command

LABEL COLOUR 3

will select overlay colour 3 for subsequent labels. The supplied value should be valid for the current PLANES setting.

Labelling may be cleared independently from an image displayed in Picture 1, using the command CLEAR LABEL. This command has the same effect as giving the command CLEAR PICTURE 2, so that unfortunately any vector information previously output to Picture 2 will also be lost. You should always be aware of this limitation, and in a situation where you have a large amount of overlaid vector information and a large amount of annotation, it is advisable to experiment with user labelling before giving the OVERLAY command.

DIGITISING COMMANDS

o **Introduction**

ROVER is a display program that brings together grid based and vector information. For a number of applications, this combined information may represent a basis from which other information may be derived. For example, you may wish to record the points where a vector feature (eg. a road) crosses the edge of a raster feature (eg. a slope boundary). For these applications a set of simple commands that allow you to digitise a point or a line, and save the information in an output IFF file, are provided.

To enter DIGITISE mode the command ENABLE DIGITISE should be issued. This command is only valid if the digitising table has been successfully initialised during program start up and if the SETUP MAP command has been given in order to register a source document to the currently selected DTI file, and optionally to an input IFF file. In DIGITISE MODE the prompt 'Digitise>' is displayed on the terminal.

The set of digitising commands include OUTPUT, SET LAYER, SET FSN, SET FC, START, END, and CLOSE OUTPUT.

o **Output File Selection**

An output IFF file to receive the digitised information is specified using the OUTPUT command. OUTPUT requires a filename, which is parsed against the default 'LSL$IF:IFF.IFF'.

    eg. OUTPUT DIGIT

will open the file 'LSL$IF:DIGIT.IFF' to receive any digitised information.

OUTPUT will always create a new IFF file. It is not possible to digitise to an existing file. The corners of the output file and map descriptor projection information are determined from either the input IFF file specified using the command IFF, or if no input IFF file has been selected, from the DTI file currently selected and registered to the source document using the SELECT FILEIN and SETUP MAP commands. All digitised (table) coordinates are transformed to these corners, and consequently the coordinate information in the output IFF file will be in either IFF or in appropriate units related to the currently selected DTI file if no input IFF file has been specified. These units will be in tenths of seconds for TED4 or UHL1 files. They will be in projection units for those files containing a projection record, and in DTI matrix units for other files.

The output IFF file may be closed using the command CLOSE OUTPUT, in order to terminate the digitising session, or in order to specify a new OUTPUT file.

- **Feature and Layer Set Up**

The SET LAYER command will allow you to specify an IFF layer in the IFF file to receive the digitised information.

eg. SET LAYER 5

will create layer 5 in the output IFF file. By default the information is written to layer 1. Only 1 layer per output IFF file is allowed.

The features serial number (FSN) of a digitised feature may be set using SET FSN.

eg. SET FSN 90

gives the next feature to be digitised, a FSN of 90. Subsequent IFF features will be given a FSN of 91, 92, 93 and so on, unless the command SET FSN is again specified. By default, the first feature digitised in the output IFF feature will be given a FSN of 1, and this value will be incremented by 1 when a new feature is started.

The feature code (FC) of a digitised feature may be specifically set using the SET FC command.

eg. SET FC 999

will set the feature code of the digitised feature to 999. All subsequent digitised features will also be given a feature code of 999, unless a new output feature code is set using SET FC. By default the output feature code is 0.

- **Feature Construction**

In order to start the construction of a feature you should give the command START. START may be issued either from the terminal, or using button 4 on the table puck. In the second case a point is recorded at the current cursor position. A crosshair cursor will appear on the screen to indicate that you are currently constructing a feature, and the prompt 'Digitise Point>' is displayed on the terminal.

The cursor may be moved around the screen without recording a point by repeatedly pressing button 0 on the puck.

In order to digitise a point you should use button 4. Button F is defined as ABANDON, and will allow you to abort the construction of a feature. There is no limit on the number of points that may be digitised, but it should be noted that ABANDON is effective only before the first ST entry has been output for the current feature. The first ST entry is output when 200 points have been recorded.

The construction of a feature is terminated by using the END command. This command may be issued either from the terminal or by using button 5 on the table puck. In the second case a point is recorded at the current cursor position before closing the feature.

During the construction of a feature it is still possible to use the ENLARGE and REDUCE commands if given from the puck. Buttons 2 and 3 are defined as ENLARGE and REDUCE respectively, and operate by centring the enlarged display around the current cursor position.

o **Puck Button Commands**

The puck buttons are defined as follows in DIGITISE mode:

Button Number	Meaning
0	Move the crosshair cursor but do not record a point
1	Not valid when digitising
2	ENLARGE display around cursor
3	REDUCE display around cursor
4	START
5	END
F	ABANDON

RASTER EDITING COMMANDS

o **Introduction**

Rover provides an EDIT mode of operation for the editing of the raster DTI files. During editing, the vector overlay may remain present, thereby realising the powerful ability to edit the raster file with direct reference to the overlaid vectors.

Rover enters EDIT mode when the command ENABLE EDIT is issued, and the prompt 'Edit>' is displayed. A default black (Colour 1 in the colour table) pixel-wide cursor is then visible on the display area. The cursor may be moved by means of mouse or trackerball movement, by pressing button 0 on the puck if a map has been registered, or by explicit MOVE commands issued at the terminal. The UP, DOWN, LEFT and RIGHT commands may be used to move the cursor by a single DTI pixel.

The SET DELAY command may be used to control the delay time between successive cursor updates on the screen. By default this is set to 100 milliseconds. If the cursor lags behind the movement of the mouse or trackerball, then the value should be increased. A suitable delay setting my depend on the size of the window displayed and also on the amount of graphics activity taking place on the workstation.

The Cursor colour may be changed to any of the available Picture 1 colours by means of the SET CURSOR_COLOUR command. The cursor will also change colour automatically if it is currently over a pixel which is the same colour as itself. Alternatively a crosshair cursor may be selected by using the ENABLE CROSS_CURSOR command. This cursor remains a constant size regardless of the DTI-pixel size and is therefore useful when large windows are displayed in EDIT mode. The DISABLE CROSS_CURSOR command reselects the square pixel-sized cursor.

The cursor position and DTI value is shown at the bottom of the screen. The DTI pixel colour at the current cursor position is also displayed.

All files may be examined in this way regardless of whether they have been opened for read-only access with the FILEIN command or for write access with the UPDATE_FILEIN command. If pixel values are to be changed, then the file must be opened with the UPDATE_FILEIN command.

WARNING: Edits on DTI files are carried out in-situ. Thus edited DTI pixels are changed in the disk file immediately. To safeguard against possible file corruption, it is recommended that copies of files are made prior to a Rover editing session.

Certain commands are only valid in EDIT mode of operation. Other commands may have a slightly different effect from the normal display mode. The WINDOW, RANGE, COLOURS, STEP, FIRST_COLOUR, and SELECT FILEIN commands all cause automatic redisplay of the currently selected DTI file. If this is not required then it is a simple operation to DISABLE EDIT mode and enable again when necessary.

The SELECT FILEIN command selects the file which may be edited. In quartered screen mode this allows the user to switch easily from one file to another. If the file selected has already been displayed in one of the quadrants, then ROVER remembers which, and switches the editing cursor and status lines without redisplay.

On issue of the DISABLE EDIT command the program will check the minimum and maximum values in the DTI file. If they have changed then the new values will be set in the DTI file header. This operation may be carried out explicitly by issue of the RESET_MINMAX command.

Three forms of editing may be carried out.

- **Line Editing**

  A line of DTI pixels may be edited using a sequence of EDIT and MOVE commands followed by an END command. The END command takes an optional argument of a DTI value to be assigned to the pixels currently highlighted. If no argument is supplied, which will be the case if the END command is issued with a button press, then the user is prompted for the DTI value. When a valid DTI value has been specified, the edit pixels are set to the defined value in the DTI file and the display reflects their new colour appropriately.

  During line editing a 'Line>' prompt is displayed on the terminal. At any time during the editing sequence ABANDON or <CTRL/Z> (pressing the Ctrl and Z keys together) may be used to abandon the current edit. The highlighted 'edit' pixels are replaced in their original colour. The REMOVE command may also be given before END is specified. This unsets the effect of the latest EDIT command.

- **Area Editing**

  An area of pixels to be edited may be specified using the AREA_EDIT command. The perimeter of the area is defined using the same method as line editing. On issue of the END command the area polygon is closed and the supplied DTI value is assigned to all pixels inside the defined perimeter.

  During area editing a 'Area>' prompt is displayed on the terminal.

- **Smooth Editing**

  The command SMOOTH_EDIT requires the subsequent definition of two edit lines. The lines are defined using the same method as line editing. Appropriate messages on the terminal indicate the required sequence of commands. A DTI value is assigned to each line, and values are linearly interpolated between them. Alternatively, if no DTI value is specified by typing <CR> only after the Dti Value> prompt then the existing DTI values for the specified line are used as the end values of the interpolation.

This form of editing is particularly useful for the correction of Digital Elevation Models. For example a hill which has been modelled incorrectly using an incorrect spot height may be corrected with SMOOTH_EDIT.

During smooth editing a 'Smooth>' prompt is displayed on the terminal.

o **Interactive Flooding**

If the command ENABLE FLOODING is given then the program enters FLOODING mode and the 'Flood>' prompt is displayed on the terminal. Movement of the mouse or trackerball in the Y direction or the table puck (pressing button 0) causes the display to be "flooded" or "unflooded".

The effect of Flooding is achieved by dynamically rewriting the colour table to be blue. The current flood level is shown at the bottom of the display.

Flooding is a particularly powerful tool for the validation of Digital Elevation Models.

Flood mode is exited by specifying DISABLE FLOOD.

o **Puck Button Commands**

If a table has been successfully set up during program initialisation and and a map has been registered to the currently selected DTI file then edit commands may be specified by means of the table puck. The buttons on the table puck have the following meanings in EDIT mode.

0 MOVE	1 CENTRE	2 ENLARGE	3 REDUCE
4 EDIT	5 END	6 REMOVE	7 ENABLE/ DISABLE CROSS_CURSOR
8 LEFT	9 UP	A RIGHT	B DOWN
C AREA_EDIT	D SMOOTH_EDIT	E ENABLE/ DISABLE	F ABANDON

```
|             |             |  FLOODING   |             |
|_____________|_____________|_____________|_____________|
```

The three mouse or function buttons are defined to be EDIT, END and REMOVE

REGISTRATION COMMANDS

o **Introduction**

ROVER provides facilities to allow Registration Control Point (RCP) files to be created and modified. The RCP files may be used for map/image registration.

The ROVER registration options and commands are only available in registration mode. Registration mode is activated using the command **ENABLE REGISTRATION**, and is signified by the prompt 'Registration>' The command DISABLE REGISTRATION is used to exit from Registration mode.

The specific registration commands are briefly described below, and in more detail in the COMMANDS section.

o **RCP File Commands**

Registration Control Points may be added either to a new RCP file, or appended to the contents of an existing file. A new RCP file is created using the command **CREATE RCP_FILE**, while an existing RCP file, probably generated in an earlier session of ROVER, is read using the command **OPEN RCP_FILE**. By default RCP files are assumed to be found in a directory defined with the logical name **LSL$RCP**

Up to 4 RCP files may be opened. A different number is assigned to each file. The first file to be opened or created is assigned the number 1; the second number 2, and so on. The number is used to select which file is current, using the **SELECT RCP_FILE** command. By default the file most recently opened or created with the OPEN RCP_FILE and CREATE RCP_FILE commands, is the current file. The commands described below in the section RCP COMMANDS operate only on the currently selected RCP file.

The contents of the current RCP file may be examined using the command **SHOW RCP_FILE**, while a RCP file may be closed by giving the command **CLOSE RCP_FILE**. The latter command is only required if more than 4 RCP files are to be manipulated, since an implicit CLOSE RCP_FILE ALL command is carried out when DISABLE REGISTRATION is specified to exit from registration mode.

o **RCP Commands**

Commands are provided to add, delete and edit individual Registration Control Points contained in the currently selected RCP file. In addition a command is provided to position the screen cursor on to an existing RCP.

For each RCP, an identifier, type value, x y z coordinate and x y z coordinate weight values are recorded.
The RCP identifier is a character string of up to 10 characters that is used to uniquely identifier a RCP.
The RCP type value is a character string of up to 2 characters that

may be used to identifier the type of RCP (eg. an image control point or map control point).
The x and y coordinates define the geographical position of the RCP.
The optional Z coordinate defines the height at the RCP position.
The x y and z coordinate weight values are used to assign a nominal degree of confidence to the control points.

The command **RCP ADD** is used to add a new RCP to the RCP file. The x y coordinates of the RCP may be supplied as command parameters, or entered from the workstation mouse or table puck. The centre button of the mouse, and button 4 on the table puck are defined as RCP ADD. The various DISABLE and ENABLE DEFAULT RCP commands control whether the user is prompted for the additional RCP information. For example, if the command DISABLE DEFAULT RCP_ID has been given, then the user is prompted to enter a RCP identifier. The SET DEFAULT RCP commands control the default value that is supplied with a prompt, and which may be accepted using carriage return. By default, no prompting for additional RCP information occurs, and the RCP fields are assigned their default values.

It is possible to edit an existing RCP using the **RCP EDIT** command. The RCP to be edited is identified by means of the RCP identifier. When editing a RCP the user is prompted for each RCP attribute in turn. The defaults supplied with the prompts are the current RCP values. As with the RCP ADD command, the new x y coordinate information may be supplied from the terminal, mouse or table.

A RCP may be deleted using the **RCP DELETE** command. The RCP to be deleted is identified by means of the RCP identifier.

The **RCP MOVE** command is provided to position the screen cursor on to the location of an existing RCP.

o **Use of the Workstation Mouse or Tracker Ball in Registration Mode**

If the workstation incorporates a mouse or tracker ball then it is possible to interactively move the registration cross cursor over the image displayed on the screen. As the cursor is moved, the coordinates of the image at the cursor position, and the associated image value, are written to the status area at the bottom of the screen. Note that in comparison to Edit mode, in Registration mode, the coordinates values are updated as the cursor is moved between matrix nodes.

The image coordinates are written to the status line in the current units of measurement defined using the UNITS command, and as either absolute coordinates or coordinates relative to the origin of the DTI file, depending on whether the ENABLE ABSOLUTE command has been given.

Function button 2 (the centre button of the mouse) is defined as the command RCP_ADD. On pressing the button a RCP will be created in the currently selected RCP file, with the x y image coordinates associated with the current cursor position. The RCP coordinates are recorded in the RCP file in the current units of measurement, and as either relative or absolute values depending on whether the ENABLE

ABSOLUTE command has been given. Note that if UNITS LATLONG is current, then the coordinates are held in the RCP file in .1 seconds of arc.

- **Use of a Digitising Table in Registration Mode**

  If the workstation incorporates a digitising table, and the Table Monitor has been successfully initialised either on program startup or following an ENABLE TABLE command, it is possible to enter the coordinates of an RCP using button 4 on the table puck.

  To enter RCP coordinates from the table it is necessary to position a source document on the table by means of the SETUP MAP command. This command operates differently if given in Registration mode, than in other ROVER modes of operation. In other modes of operation, the source document is registered to the current DTI file - this means that a table coordinate is converted to a DTI coordinate. If given in registration mode however, the source document is treated independently of any input DTI file, and a table coordinate will be converted into the coordinate system of the source document. In order to perform this conversion, the user will be prompted to enter the source document coordinates that correspond to the bottom left and top right rectangular registration points, digitising during map setup.

  The **TABLE RCP_FILE** command may be used to associate an RCP file with the table. For example, if the command TABLE RCP_FILE 2 is given, then an RCP entered from the table using button 4, will automatically be recorded in RCP_FILE 2 irrespective of whether this is the currently selected RCP file. This mechanism allows an RCP relating to an image to be recorded in one file, and an RCP relating to a map to be recorded in a separate file. The user is able to easily alternate between the two.

In Registration mode the buttons of the table puck have the following meanings:

0 MOVE	1 CENTRE	2 ENLARGE	3 REDUCE
4 RCP_ADD	5 Undefined	6 Undefined	7 Undefined
8 LEFT	9 UP	A RIGHT	B DOWN
C Undefined	D Undefined	E Undefined	F DISABLE REGIS- TRATION

ROVER COMMANDS

!

Treat all text to the right of the '!' as a comment.

FORMAT: ! [comment text]

Command parameters:

comment text

text that is to be treated as a comment and which will be excluded from command interpretation. Up to 80 characters may be specified.

DESCRIPTION:

An exclamation mark is the standard TVES package comment delimiter. All text (and numbers) which lie to the right of a '!' character are excluded from command interpretation. Comments are useful for annotating command procedures used in batch processing etc.

Messages: None

Examples:

Rover> **FILEIN TEST !open the file<CR>**
Rover> **!Define a window<CR>**
Rover> **WINDOW 1 1 10 10<CR>**
Rover>

@

Execute a Rover command file.

FORMAT: @file-spec

Command parameters:

file-spec

The file specification of the command file to be obeyed. Any part of the file specification which is not supplied is taken from the default 'LSL$ROVERCMD:ROVER.RCM'

DESCRIPTION:

The @ command is used to specify that further commands are to be read from the supplied Rover Command File (RCM). Input is taken from the command file until the end of file is reached.

The RCM file may contain RESPOND commands which return temporary control to the terminal. The CONTINUE command issued at the terminal resumes RCM file execution.

Only one RCM file may be open at a time and command file execution may be abandoned using <CTRL/C> (pressing the Ctrl and C keys together).

Messages:

The following messages are specific to the @ command

```
*** ERROR *** Specifying command @
Command invalid while command file already open
```

```
*** ERROR *** Specifying command @
Command File name missing
```

Examples:

```
Rover> @DTIERRORS.RCM<CR>
Reading input from command file LSL$ROVERCMD:DTIERRORS.RCM
Rover>
```

ABANDON

Abandons current feature construction or raster editing sequence.

Only available in EDIT or DIGITISE modes of operation.

FORMAT: ABANDON

Command parameters: None

DESCRIPTION:

In the DIGITISE mode the ABANDON command abandons the construction of a feature.

In EDIT mode ABANDON cancels the current raster editing sequence.

The command is only available in EDIT or DIGITISE modes of operation. ABANDON is defined as puck button F. In EDIT mode typing <CTRL/Z> (pressing the Ctrl and Z keys together) also produces the effect of an ABANDON command.

Messages:

The following messages are specific to the ABANDON command.

```
*** ERROR *** Specifying command ABANDON
Command only available in EDIT or DIGITISE modes
```

Examples:

```
Rover> ENABLE EDIT<CR>
Edit> MOVE 10 10<CR>
Edit> EDIT<CR>
Line> MOVE 20 20<CR>
Line> ABANDON<CR>
Edit>
```

```
Rover> ENABLE DIGITISE<CR>
Digitise> START<CR>
Digitise Point> ABANDON<CR>
Digitise>
```

AREA_EDIT

Starts an area editing operation

Only available in EDIT mode of operation.

FORMAT: **AREA_EDIT**

Command parameters: None

DESCRIPTION:

The AREA_EDIT command starts an area editing operation.

A subsequent sequence of EDIT and MOVE commands are used to define the perimeter of the area to be edited. The END command completes the operation and the specified DTI value is assigned to pixels which lie within the area.

The command is only available in EDIT mode of operation.

To indicate that area editing is taking place the program prompt becomes 'Area>' while the area is being defined.

ABANDON or <CTRL/Z> (pressing the Ctrl and Z keys together) may be used to abort an AREA editing operation.

Puck button C is defined to be AREA_EDIT.

Messages:

The following messages are specific to the AREA_EDIT command.

*** ERROR *** specifying command AREA_EDIT
Command only available in EDIT mode

*** ERROR *** specifying command AREA_EDIT
Selected DTI file is opened read-only

*** ERROR *** specifying command AREA_EDIT
Command not available during edit operation

Examples:

Edit>**AREA_EDIT<CR>**
Area>**MOVE 50 50<CR>**
Area>**EDIT<CR>**
Area>**MOVE 50 100<CR>**
Area>**EDIT<CR>**
Area>**MOVE 100 100<CR>**
Area>**EDIT<CR>**
Area>**MOVE 100 50<CR>**
Area>**END<CR>**
Area> DTI Value> **500<CR>**
Edit>

In this example a square area is defined and all the pixels within it are assigned the value 500. The commands could, of course, be more easily specified with the mouse, trackerball or table puck buttons.

CENTRE

Centres the displayed image around the given DTI coordinates.

FORMAT: **CENTRE xcoord, ycoord**

Command parameters:

xcoord, ycoord

The matrix coordinates specified as a column and row value. 2 integer values are required.

DESCRIPTION:

The displayed image is centred around the screen position corresponding to the supplied matrix x y coordinate point provided they lie within the geographical bounds of the file.

If a source document has been registered to the current DTI using the SETUP MAP command, CENTRE may be given using Puck Button 1.

This command has effect only on a graphics device which supports hardware pixel replication.

Messages:

The following message are specific to the CENTRE command.

*** ERROR *** specifying command CENTRE
Command requires 2 integer arguments

Examples:

Rover>**CENTRE 100 100<CR>**
Rover>

In this example the display will be centred around the screen position corresponding to the point with the given matrix coordinates

---

CLEAR

Clears the screen.

FORMAT: **CLEAR**

Command Parameters: None

DESCRIPTION:

This command, when issued with no qualifiers, clears the entire screen of both raster and vector information.

In EDIT mode the effect of clearing results in movement of the pixel cursor in the previously displayed area "repainting" the image. It is therefore recommended that a DISPLAY command should be issued before the cursor is moved.

Messages:

The following messages are specific to the CLEAR command.

*** ERROR *** specifying command CLEAR
Command not available during edit operation

*** ERROR *** specifying command CLEAR
CLEAR qualifier is QUADRANT, PICTURE or LABEL

Example:

Rover>**CLEAR <CR>**
Rover>

CLEAR LABEL

Clears user annotation.

FORMAT: **CLEAR LABEL**

Command Parameters: None

DESCRIPTION:

User annotation which has been specified with the DRAW LABEL command is cleared from the screen.

Note that because annotation is held in picture 2, any overlaid vector information is also cleared using this command.

Messages:

The following messages are specific to the CLEAR command.

```
*** ERROR *** specifying command CLEAR
CLEAR qualifier is QUADRANT, PICTURE or LABEL
```

```
*** ERROR *** specifying command CLEAR
Command not available during edit operation
```

Example:

Rover>**CLEAR LABEL<CR>**
Rover>

CLEAR PICTURE

Clears the specified Picture.

FORMAT: CLEAR PICTURE picture-number

Command parameters:

picture-number

The picture that is to be cleared. An integer value in the range 1 to 2 is required.

DESCRIPTION:

The CLEAR PICTURE command is used to clear either picture 1 or picture 2.

The number of bit-planes allocated to picture 1 and picture 2 is determined using the PLANES command. By default 5 planes (bit-planes 1 to 5) are used for Picture 1, and the remaining 3 planes (bit-planes 6 to 8) are used for Picture 2. A raster DTI image is drawn into Picture 1, while any vector overlay information or annotation drawn using the OVERLAY and DRAW LABEL commands, is displayed in Picture 2.
A raster image may therefore be cleared independently of any overlay using the command CLEAR PICTURE 1, while any overlay may be cleared independently from the raster base using the command CLEAR PICTURE 2. To clear both pictures (ie. all bit-planes of a graphics display), the command CLEAR should be used.

The CLEAR PICTURE command erases the information held in the memory of the graphics device and hence further DISPLAY or OVERLAY commands are required to reproduce an image on the screen. Note that ENABLE/DISABLE PICTURE commands allow instantaneous selection and deselection of raster and vector information already produced with the DISPLAY or OVERLAY commands.

Messages:

The following messages are specific to the CLEAR and CLEAR PICTURE commands.

```
*** ERROR *** specifying command CLEAR
Command not available during edit operation

*** ERROR *** specifying command CLEAR
CLEAR qualifier is QUADRANT, PICTURE or LABEL

*** ERROR *** specifying command CLEAR PICTURE
Command requires an integer in the range 1 to 2
```

Example:

```
Rover>CLEAR PICTURE 2<CR>
Rover>
```

CLEAR QUADRANT

Clears a particular quadrant.

FORMAT: **CLEAR QUADRANT quadrant [1..4]**

Command parameters:

quadrant number(s)

Up to 4 integer quadrant numbers may be specified

The quadrant numbers are defined as follows.

Quadrant 1	Top Left Quadrant
Quadrant 2	Top Right Quadrant
Quadrant 3	Bottom Right Quadrant
Quadrant 4	Bottom Left Quadrant

DESCRIPTION:

This command clears both picture 1 (raster) and picture 2 (vector) in the specified quadrant. More than one quadrant may be specified for clearing simultaneously.

The command is only valid if quartered screen mode has been enabled with the ENABLE DIVIDE command.

Messages:

The following messages are specific to the CLEAR and CLEAR QUADRANT commands.

*** ERROR *** specifying command CLEAR
Command not available during edit operation

*** ERROR *** Specifying command CLEAR
CLEAR qualifier is QUADRANT, PICTURE or LABEL

*** ERROR *** Specifying command CLEAR QUADRANT
Command only valid if ENABLE DIVIDE is specified

*** ERROR *** Specifying command CLEAR QUADRANT
Command requires up to 4 integers in the range 1 to 4

Examples:

Rover>**CLEAR QUADRANT 1<CR>**
Rover>

In this example the upper left quadrant is cleared.

Rover>**CLEAR QUADRANT 2 3 4<CR>**
Rover>

Here the remaining part of the screen is also cleared.

CLOSE FILEIN

Closes DTI file(s)

FORMAT: **CLOSE FILEIN DTI_number[s]**
CLOSE FILEIN ALL

Command parameters:

DTI_number[s]

A list of up to 4 DTI file numbers to which the open DTI files have been assigned. The values should be in the range 1 to 4. Alternatively the ALL qualifier may be specified.

DESCRIPTION:

This command is used to close one or more DTI files which may be open. As only four files may be open, the parameters should be in the range 1 to 4, while CLOSE FILEIN ALL will close all currently open DTI files.

Messages:

The following messages are specific to the CLOSE and CLOSE FILEIN commands.

```
*** ERROR *** Specifying command CLOSE
Command not available during edit operation

*** ERROR *** Specifying command CLOSE
Command qualifiers are FILEIN, IFF, OUTPUT or RCP_FILE

*** ERROR *** Specifying command CLOSE FILEIN
Command requires up to 4 integers in the range 1 to 4 or the ALL qualifier

*** ERROR *** Specifying command CLOSE FILEIN
File "DTI_number" is not open
```

Examples:

```
Rover>CLOSE FILEIN 1<CR>
DTI File 1 Closed

Rover>CLOSE FILEIN ALL<CR>
DTI File 3 Closed
DTI File 4 Closed
Rover>
```

CLOSE IFF

Closes the input IFF file.

FORMAT: **CLOSE IFF**

Command Parameters: None

DESCRIPTION:

The input IFF file holding vector information is closed. This command is required before another IFF file may be opened.

Messages:

The following messages are specific to the CLOSE and CLOSE IFF commands.

*** ERROR *** Specifying command CLOSE
Command not available during edit operation

*** ERROR *** Specifying command CLOSE
Command qualifiers are FILEIN, IFF, OUTPUT or RCP_FILE

*** ERROR *** Specifying command CLOSE IFF
No IFF file is open

Examples:

Rover>**CLOSE IFF<CR>**
Input IFF file closed
Rover>

CLOSE OUTPUT

Closes the output IFF file.

FORMAT: **CLOSE OUTPUT**

Command Parameters: None

DESCRIPTION:

The output IFF file which has been previously opened to receive digitised coordinates is closed.

Messages:

The following messages are specific to the CLOSE and CLOSE OUTPUT commands.

*** ERROR *** Specifying command CLOSE
Command not available during edit operation

*** ERROR *** Specifying command CLOSE
Command qualifiers are FILEIN, IFF, OUTPUT or RCP_FILE

*** ERROR *** Specifying command CLOSE IFF
No Output IFF file is open

Examples:

Rover>**CLOSE OUTPUT<CR>**
Output IFF file closed
Rover>

CLOSE RCP_FILE

Closes a Registration Control Point (RCP) file.

The command CLOSE RCP_FILE is only available in registration mode.

FORMAT: **CLOSE RCP_FILE number**
CLOSE RCP_FILE ALL

Command Parameters:

number

A list of up to 4 integer RCP file numbers. The values should be in the range 1 to 4. Alternatively the ALL qualifier may be specified.

DESCRIPTION:

Closes a Registration Control Point (RCP) file.

Messages:

The following messages are specific to the CLOSE and CLOSE RCP_FILE commands.

```
*** ERROR *** Specifying command CLOSE
Command qualifiers are FILEIN, IFF, OUTPUT or RCP_FILE

*** ERROR *** Specifying command CLOSE RCP_FILE
Command only valid in REGISTRATION mode

*** ERROR *** Specifying command CLOSE RCP_FILE
Command requires a RCP file number or the qualifier ALL

*** ERROR *** Specifying command CLOSE RCP_FILE
Command requires a RCP file number in the range 1 to 4

Writing RCP entries to RCP file number <FILE_NUMBER>

Updating RCP entries in RCP file number <FILE_NUMBER>
```

Examples:

```
Registration>CLOSE RCP_FILE 1<CR>
Updating RCP entries in RCP file number <FILE_NUMBER>
Registration>
```

COLOURS

Assigns the number of colours used for raster display for the currently selected DTI file.

FORMAT: **COLOURS num_colours**

Command parameters:

num_colours

The number of colours used for raster display. This should be a positive integer in the range 1 to 2 raised to the power of n - 3 where n is the current planes setting. eg. If PLANES 6 has been set then any number between 1 and 61 may be specified.

By default the number of display colours is the maximum number of colours available with the current PLANES setting.

DESCRIPTION:

The number of colours that will be used for the display of the raster image for the currently selected DTI file is specified by this command. The number of colours which may be specified depends on the current PLANES setting. If, for example, the PLANES setting is 5 then up to 29 colours may be specified. Colour indices 1 and 2 are reserved for black and white and a further colour (the first colour as specified by the FIRST_COLOUR command) is reserved for the display of zero values in the input matrix (normally blue for digital elevation models).

This command can be used in conjunction with the FIRST_COLOUR command to enable different DTI files to be displayed using different parts of the colour table. Thus a colour image may be displayed alongside a grey scale image.

Note that redefining the number of colours, may change the step value interval value previously set by the STEP command, and that the PLANES command will reset the number of colours to the maximum allowable for that bit-plane setting.

In EDIT mode of operation this command causes automatic redisplay of the currently selected DTI file.

Messages:

The following messages are specific to the COLOURS command

*** ERROR *** Specifying command COLOURS Command requires an integer in the range 1 to 253

*** ERROR *** Specifying command COLOURS
Command not available during edit operation

*** ERROR *** Specifying command COLOURS

Use of "num_colours" colours not possible with current PLANES setting

---

Example:

```
Rover>PLANES 6<CR>
Rover>COLOURS 63<CR>
*** ERROR specifying command COLOURS
Use of 63 colours not possible with current PLANES setting
Rover>
Rover> COLOURS 61<CR>
Rover>
```

CONTINUE

Resumes Rover Command File execution.

FORMAT: **CONTINUE**

Command parameters: None

DESCRIPTION:

CONTINUE is used to resume the execution of a Rover Command File which has been specified with the @ command.

The command is only valid when the command file has passed control back to the terminal with a RESPOND command.

Messages:

The following messages are specific to the CONTINUE command

*** ERROR *** Specifying command CONTINUE
Command only valid if Rover Command File open

Example:

Rover>**CONTINUE <CR>**
Rover>

CREATE RCP_FILE

Creates a new Registration Control Point (RCP) file.

The command CREATE RCP_FILE is only available in registration mode.

FORMAT: CREATE RCP_FILE file-spec

Command parameters:

file-spec

The RCP file specification. Any part of the file specification not supplied will be taken from the default 'LSL$RCP:RCP.RCP'.

DESCRIPTION:

The CREATE RCP_FILE command is used to create a new Registration Control Point file.

Up to 4 RCP files may be created. The SELECT RCP command controls which RCP file is current. By default, the current file is the most recent RCP file to be created.

Entries may be added to the current file using the RCP ADD command.

Messages:

The following messages are specific to the CREATE and CREATE RCP_FILE commands:

*** ERROR *** Specifying command CREATE
Command qualifier is RCP_FILE

*** ERROR *** Specifying command CREATE RCP_FILE
Command only valid in REGISTRATION mode

*** ERROR *** Specifying command CREATE RCP_FILE
RCP Filename is missing

*** ERROR *** Specifying command CREATE RCP_FILE
Maximum number of RCP files already open
Use the CLOSE RCP_FILE command to close a file

File <FILE_SPEC> created as RCP File Number <RCP_FILE_NUMBER>

Examples:

Registration>**CREATE RCP_FILE CONTROL<CR>**
File LSL$RCP:CONTROL.RCP created as RCP File Number 1
Registration>

DESELECT FC

Excludes an IFF feature from overlay on the basis of feature code.

FORMAT: DESELECT FC feature_code [,...]

Command parameters:

feature_code

An integer feature code in the range 0 to 32767. Multiple feature codes may be specified separated by commas or spaces, while a range of feature codes may be specified by means of a '-'. eg. DESELECT FC 10-13 excludes feature codes 10,11,12 and 13.
Alternatively a valid FRT group name may be specified.

DESCRIPTION:

The DESELECT FC command is used to exclude IFF features from vector overlay. Features are excluded on the basis of their feature code.

By default ROVER will use all features within an IFF file.

Use of the SHOW SELECTIONS command is recommended to display feature selections before the OVERLAY command is given.

The DESELECT FC command is only valid if an FRT command has been used to select a Feature Representation Table.

Messages:

The following messages are specific to the DESELECT and DESELECT FC commands

*** ERROR *** Specifying command DESELECT
Command qualifiers are FC, FSN or LAYER

*** ERROR *** Specifying command DESELECT FC
Command invalid before FRT has been read

*** ERROR *** Specifying command DESELECT FC
No groups have been defined in the FRT

*** ERROR *** Specifying command DESELECT FC
Max number of SEL/DES commands allowed exceeded

*** ERROR *** Specifying command DESELECT FC
Illegal feature code "featurecode"

*** ERROR *** Specifying command DESELECT FC
Bad group name "group"

Examples:

Rover>**DESELECT FC 1<CR>**
Rover>**DESELECT FC RAILWAYS<CR>**
Rover>**DESELECT FC RAIL,7-10,56-78<CR>**
Rover>

DESELECT FSN

Excludes an IFF feature from overlay on the basis of feature serial number.

FORMAT: **DESELECT FSN fsn [,...]**

Command parameters:

fsn

An integer feature code in the range 0 to 65534. Multiple feature serial numbers may be specified separated by commas or spaces, while a range of numbers may be specified by means of a '-'. eg. DESELECT FSN 10-13 excludes feature serial numbers 10,11,12 and 13.

DESCRIPTION:

The DESELECT FSN command is used to exclude IFF features from vector overlay. Features are excluded on the basis of their feature serial number.

By default ROVER will use all features within an IFF file.

Use of the SHOW SELECTIONS command is recommended to display feature selections before the OVERLAY command is given.

The DESELECT FSN command is only valid if an FRT file has been specified.

Messages:

The following messages are specific to the DESELECT and DESELECT FSN commands

```
*** ERROR *** Specifying command DESELECT
Command qualifiers are FC, FSN or LAYER
```

```
*** ERROR *** Specifying command DESELECT FSN
Command invalid before FRT has been read
```

```
*** ERROR *** Specifying command DESELECT FSN
Illegal FSN number "fsn"
```

Examples:

```
Rover>DESELECT FSN 7<CR>
Rover>DESELECT FSN,7-10,56-78<CR>
Rover>
```

DESELECT LAYER

Excludes an IFF feature from overlay on the basis of layer number.

FORMAT: **DESELECT LAYER layer [,...]**

Command parameters:

layer

An integer layer number in the range 0 to 32767. Multiple layer numbers may be specified separated by commas or spaces, while a range of layer numbers may be specified by means of a '-'. eg. DESELECT LAYER 10-13 excludes all features in layers 10,11,12 and 13.

DESCRIPTION:

The DESELECT LAYER command is used to exclude IFF features from vector overlay. Features are excluded on the basis of IFF layer.

By default ROVER will use all features within an IFF file.

Use of the SHOW SELECTIONS command is recommended to display feature selections before the OVERLAY command is given.

The DESELECT LAYER command is only valid if an FRT file has been specified.

Messages:

The following messages are specific to the DESELECT and DESELECT LAYER commands

```
*** ERROR *** Specifying command DESELECT
Command qualifiers are FC, FSN or LAYER

*** ERROR *** Specifying command DESELECT LAYER
Command invalid before FRT has been read

*** ERROR *** Specifying command DESELECT LAYER
Illegal Layer number "layer"

*** ERROR *** Specifying command DESELECT LAYER
Command invalid before FRT has been read
```

Examples:

Rover>**DESELECT LAYER 7<CR>**
Rover>**DESELECT LAYER 1,7-10<CR>**
Rover>

DISABLE ABSOLUTE

Disables a previous ENABLE ABSOLUTE command.

FORMAT: DISABLE ABSOLUTE

Command Parameters: None

DESCRIPTION:

DISABLE ABSOLUTE cancels a previous ENABLE ABSOLUTE command. If DISABLE ABSOLUTE is given, then coordinate values required by the WINDOW and MOVE, commands, supplied in metre or projection units, must be specified as an offset from the SW corner of the matrix. Coordinates supplied with the IFFWINDOW command should similarly be specified as an offset from the SW corner of the IFF file area.
By default window values should be specified as absolute coordinates.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the DISABLE command.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE,
TABLE or TEXT
```

Examples:

Rover>**DISABLE ABSOLUTE<CR>**
Rover>

DISABLE COLOUR_MAP

Cancels the effect of a previous ENABLE COLOUR_MAP command

FORMAT: **DISABLE COLOUR_MAP**

Command Parameters: None

DESCRIPTION:

DISABLE COLOUR_MAP disables the use of a temporary mapped file used to make frequent redisplay, windowing and zooming more efficient. The command therefore cancels the effect of any previous ENABLE COLOUR_MAP command.

By default no temporary mapped file is used.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following messages are specific to the DISABLE and DISABLE COLOUR_MAP commands.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE,
TABLE or TEXT
```

```
*** ERROR *** Specifying command DISABLE COLOUR_MAP
Command not valid during editing operation
```

Examples:

```
Rover>DISABLE COLOUR_MAP<CR>
Rover>
```

DISABLE CROSS_CURSOR

Disables the use of the raster editing cross cursor.

Only valid in EDIT mode of operation

FORMAT: **DISABLE CROSS_CURSOR**

Command Parameters: None

DESCRIPTION:

DISABLE CROSS_CURSOR disables the use of a vector cross cursor during raster editing and reselects the square DTI-pixel sized cursor. The command therefore cancels the effect of a previous ENABLE CROSS_CURSOR command. By default the square DTI-pixel sized cursor is used.

This command is only available in EDIT mode of operation.

The SHOW ENABLE command may be used to check the status of the option.

Puck button 7 alternately ENABLEs and DISABLEs CROSS_CURSOR.

Messages:

The following messages are specific to the DISABLE and DISABLE CROSS_CURSOR commands.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE,
TABLE or TEXT
```

```
*** ERROR *** Specifying command DISABLE CROSS_CURSOR
Command only available in EDIT mode
```

Examples:

Rover>**DISABLE CROSS_CURSOR<CR>**
Rover>

DISABLE DEFAULT

Controls for what RCP information the user is prompted, when the RCP ADD command is given.

The command DISABLE DEFAULT is only valid in REGISTRATION mode

---

FORMAT: DISABLE DEFAULT rcp_qualifier

Command Parameters:

rcp_qualifier

The name of a Registration Control Point attribute, chosen from:

ALL	All RCP attributes
RCP_ID	The RCP identifier
RCP_TYPE	The RCP type identifier.
RCP_WXY	The RCP XY coordinate weighting value
RCP_WZ	The RCP Z coordinate weighting value
RCP_Z	The RCP Z coordinate value

DESCRIPTION:

The DISABLE/ENABLE DEFAULT RCP command controls for what information the user is prompted, when the RCP ADD command is given.

If DISABLE DEFAULT is specified then the user is prompted for the specified RCP attribute. For example, if the command DISABLE DEFAULT RCP_ID has been given, the user is prompted to enter an RCP identifier. The SET DEFAULT RCP commands control the default value that is supplied with a prompt, and which may be accepted using carriage return. By default no prompting for additional RCP information occurs, and the RCP fields are assigned their default values.

The SHOW ENABLE command may be used to check on the status of the options when in Registration mode.

Messages:

The following messages are specific to the DISABLE and DISABLE DEFAULT commands:
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE,
TABLE or TEXT

*** ERROR *** Specifying command DISABLE DEFAULT
Command only valid in REGISTRATION mode

*** ERROR *** Specifying command DISABLE DEFAULT

Command qualifiers are ALL, RCP_ID, RCP_TYPE, RCP_WXY, RCP_WY or RCP_Z

Examples:

```
Registration>DISABLE DEFAULT RCP_TYPE<CR>
Registration>RCP ADD 1.4 2.3
Enter RCP TYPE (Default = AA)>BB<CR>
Registration>
```

DISABLE DIGITISE

Disables the ROVER digitising option.

FORMAT: **DISABLE DIGITISE**

Command Parameters: None

DESCRIPTION:

DISABLE DIGITISE ends the digitising mode of operation. The program returns to the main display mode, and the 'Rover>' prompt returns.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following messages are specific to the DISABLE command.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE,
TABLE or TEXT
```

```
*** ERROR *** Specifying command DISABLE DIGITISE
Command not available in EDIT mode
```

Examples:

Digitise>**DISABLE DIGITISE<CR>**
Rover>

DISABLE DIVIDE

Disables divided screen mode of display.

FORMAT: **DISABLE DIVIDE**

Command Parameters: None

DESCRIPTION:

DISABLE DIVIDE deselects the divided screen display. Subsequent display commands operate on the full screen display. The command therefore cancels the effect of any previous ENABLE DIVIDE command.

In EDIT mode the command causes automatic redisplay of the currently selected DTI file.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following messages are specific to the DISABLE and DISABLE DIVIDE commands.

*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE,
TABLE or TEXT

*** ERROR *** Specifying command DISABLE DIVIDE
Command not available during EDITING operation.

Examples:

Rover>**DISABLE DIVIDE<CR>**
Rover>

DISABLE EDIT

Disables the ROVER raster editing option.

FORMAT: **DISABLE EDIT**

Command Parameters: None

DESCRIPTION:

DISABLE EDIT ends the editing mode of operation. The program returns to the main Display mode, signified by the 'Rover>' prompt.

The editing status lines at the bottom of the display are replaced by the border text if the TEXT option is enabled. An implicit RESET_MINMAX command is also applied to update the value ranges stored in the header of any edited files which have been opened for write access with the UPDATE_FILEIN command.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following messages are specific to the DISABLE and DISABLE EDIT commands.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE,
TABLE or TEXT
```

```
*** ERROR *** Specifying command DISABLE EDIT
Command not available in DIGITISE mode.
```

Examples:

Edit>**DISABLE EDIT<CR>**
Rover>

DISABLE FLOODING

Cancels a previous ENABLE FLOODING command.

FORMAT: **DISABLE FLOODING**

Command Parameters: None

DESCRIPTION:

DISABLE FLOODING ends interactive flooding using the mouse, trackerball or table. The current flood level is, however, remembered if a subsequent ENABLE FLOODING command is given.

Interactive flooding is only available in EDIT mode of operation.

The SHOW ENABLE command may be used to check the status of the option.

Puck button E alternately disables and enables flooding.

Messages:

The following messages are specific to the DISABLE and DISABLE FLOODING commands.

*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE,
TABLE or TEXT

*** ERROR *** Specifying command DISABLE FLOODING
Command only available in EDIT mode.

Examples:

Flood>**DISABLE FLOODING<CR>**
Rover>

DISABLE HATCH

Cancels a previous ENABLE HATCH command.

FORMAT: **DISABLE HATCH**

Command Parameters: None

DESCRIPTION:

DISABLE HATCH returns to the default mode of using hatching parameters specified in the FRT file for overlay.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the DISABLE command.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE,
TABLE or TEXT
```

Examples:

Rover>**DISABLE HATCH<CR>**
Rover>

DISABLE HEIGHT

Cancels the effect of a previous ENABLE HEIGHT command

FORMAT: **DISABLE HEIGHT**

Command Parameters: None

DESCRIPTION:

DISABLE HEIGHT returns to the default mode of operation where text heights are taken from the FRT file for vector overlay.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the DISABLE command.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE,
TABLE or TEXT
```

Examples:

Rover>**DISABLE HEIGHT<CR>**
Rover>

DISABLE INTERPOLATION

Disables curve interpolation.

FORMAT: **DISABLE INTERPOLATION**

Command Parameters: None

DESCRIPTION:

DISABLE INTERPOLATION deselects curve Interpolation of graphical type 6 features output with the OVERLAY command.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the DISABLE command.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE,
TABLE or TEXT
```

Examples:

Rover>**DISABLE INTERPOLATION<CR>**
Rover>

DISABLE LEGEND

Disables the output of a display legend

FORMAT: **DISABLE LEGEND**

Command Parameters: None

DESCRIPTION:

DISABLE LEGEND disables the display of a colour/value guide and an overlay colour guide when a DISPLAY command is given.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the DISABLE command.

*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE,
TABLE or TEXT

Examples:

Rover>**DISABLE LEGEND<CR>**
Rover>

DISABLE MASK

Cancels the effect of a previous ENABLE MASK command

FORMAT: **DISABLE MASK**

Command Parameters: None

DESCRIPTION:

DISABLE MASK deselects the option to draw vectors in PICTURE 1. Subsequent vectors are drawn in PICTURE 2 when the OVERLAY command is given.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the DISABLE command.

*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE,
TABLE or TEXT

Examples:

Rover>**DISABLE MASK<CR>**
Rover>

DISABLE OVERRIDE

Cancels the effect of a previous ENABLE OVERRIDE command

FORMAT: **DISABLE OVERRIDE**

Command Parameters: None

DESCRIPTION:

DISABLE OVERRIDE cancels feature codes set by the ENABLE OVERRIDE command. The actual feature codes in the file will be used to determine their representation on the screen when output with the OVERLAY command.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the DISABLE command.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE,
TABLE or TEXT
```

Examples:

Rover>**DISABLE OVERRIDE<CR>**
Rover>

DISABLE PATTERN

Cancels the effect of a previous ENABLE PATTERN command

FORMAT: **DISABLE PATTERN**

Command Parameters: None

DESCRIPTION:

DISABLE PATTERN suppresses the the drawing of patterned lines. Lines specified as patterned in the FRT file will be drawn as solid when the OVERLAY command is issued.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the DISABLE command.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE,
TABLE or TEXT
```

Examples:

```
Rover>DISABLE PATTERN<CR>
Rover>
```

DISABLE PICTURE

Deselects the specified picture.

FORMAT: **DISABLE PICTURE picnum**

Command parameters:

picnum

The picture number. An integer value in the range 1 to 2 is required.

DESCRIPTION:

Picture 1 contains the raster information output using the DISPLAY command.

Picture 2 contains any vector information drawn using the OVERLAY command, and any user annotation drawn using the DRAW LABEL command.

DISABLE PICTURE makes the specified picture 'invisible'. The image may be made visible again with the ENABLE PICTURE command. (cf CLEAR PICTURE).

The SHOW ENABLE command may be used to check the status of the 2 pictures.

Messages:

The following messages are specific to the DISABLE and DISABLE PICTURE commands.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE,
TABLE or TEXT
```

```
*** ERROR *** Specifying command DISABLE PICTURE
Command requires an integer in the range 1 to 2
```

Examples:

```
Rover> ENABLE PICTURE 1<CR>
Rover> DISABLE PICTURE 1<CR>
Rover>
```

DISABLE POSITIONING

Cancels the effect of a previous ENABLE POSITIONING command

FORMAT: **DISABLE POSITIONING**

Command Parameters: None

DESCRIPTION:

DISABLE POSITIONING disables the use of the justification code (0-8) stored in the FS entry for text positioning. The text is positioned by its lower left corner.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the DISABLE command.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE,
TABLE or TEXT
```

Examples:

Rover>**DISABLE POSITIONING<CR>**
Rover>

DISABLE REGISTRATION

Disables the ROVER registration option

The command DISABLE REGISTRATION is only valid in REGISTRATION mode.

FORMAT: DISABLE REGISTRATION

Command Parameters: None

DESCRIPTION:

The DISABLE REGISTRATION command is used to exit from REGISTRATION mode, and to return to Display mode signified by the prompt 'ROVER>'

On giving the command, the registration status line at the bottom of the display is replaced by the border text (if the TEXT option is enabled), and the cross cursor is deleted from the screen. The command also causes all currently opened RCP files to be closed.

CTRL/Z (pressing the Ctrl and Z keys together) may also be used to exit from REGISTRATION mode.

Messages:

The following message is specific to the DISABLE command.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE,
TABLE or TEXT
```

Examples:

Registration>**DISABLE REGISTRATION<CR>**
Rover>

DISABLE SAMPLE

Disables sub-sampling during display.

FORMAT: **DISABLE SAMPLE**

Command Parameters: None

DESCRIPTION:

DISABLE SAMPLE disables the use of sub-sampling during the display of DTI files. Every DTI pixel in the current window is therefore displayed, regardless of the current sampling values set with the SAMPLE command.

Note that the command ENABLE REGISTRATION automatically enables sampling, while ENABLE EDIT automatically disables sampling.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the DISABLE command.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE,
TABLE or TEXT
```

Examples:

Rover>**DISABLE SAMPLE<CR>**
Rover>

DISABLE TABLE

Turns off input from the digitising table.

FORMAT: **DISABLE TABLE**

Command Parameters: None

DESCRIPTION:

The DISABLE TABLE command turns off input from the digitising table.

Input from the table can be reselected using the ENABLE TABLE command.

The SHOW ENABLE command may be used to check on the status of table input.

Messages:

The following message is specific to the DISABLE command.

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE,
TABLE or TEXT
```

Examples:

Rover>**DISABLE TABLE<CR>**
Rover>

DISABLE TEXT

Disables the output of border information.

FORMAT: **DISABLE TEXT**

Command Parameters: None

DESCRIPTION:

DISABLE TEXT disables the output of border information when a DISPLAY command is given. By default the DTI file name, area of interest, selected value range and number of colour values used in the display image are output to the screen.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the DISABLE command.

*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT, DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND, MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE, TABLE or TEXT

Examples:

Rover>**DISABLE TEXT<CR>**
Rover>

DISPLAY

Displays the currently selected DTI file

FORMAT: **DISPLAY**

Command parameters: None

DESCRIPTION:

DISPLAY is the raster drawing command, that will output grid information contained in a mapped DTI file, to the graphics screen in colour coded form.

The currently selected DTI file, selected by means of the SELECT FILEIN command, is displayed. RANGE, STEP, COLOURS, and FIRST_COLOUR settings for each DTI file are used to determine the exact nature of the displayed image, while only the information in the current area of interest set by the WINDOW command is displayed. The displayed image is automatically scaled to fill as much of the full screen or currently selected quadrant as possible, and is drawn in the direction specified by the ROTATE command. The default display direction is from the South. It should be noted that on certain devices DISPLAY is faster in some directions than others.

The DISPLAY command is also invoked implicitly by the ZOOM command and in EDIT mode by the STEP, RANGE, WINDOW, SELECT FILEIN, COLOURS, FIRST_COLOUR, FILEIN, and UPDATE_FILEIN commands.

See also the FAST command as an alternative method for output of byte data files.

Typing <CTRL/C> (Pressing the Ctrl and C keys together) aborts the display of the current file.

Messages:

The following messages are specific to the DISPLAY command.

*** ERROR *** Specifying command DISPLAY
Command not available during editing operation

*** ERROR *** Specifying command DISPLAY
Input DTI file "DTI-num" not yet selected

Examples:

Rover>**DISPLAY<CR>**
Rover>

DOWN

Moves the raster editing cursor down one pixel

Only available in EDIT and REGISTRATION mode

FORMAT: **DOWN**

Command parameters: None

DESCRIPTION:

DOWN moves the raster editing cursor down by one DTI pixel.

The command is only available in EDIT and REGISTRATION modes of operation.

Puck button B is defined as DOWN.

Messages:

The following message is specific to the DOWN command.

```
*** ERROR *** Specifying command DOWN
Command only available in EDIT or REGISTRATION mode
```

Examples:

Edit>**DOWN<CR>**
Edit>

DRAW LABEL

Annotates the display

FORMAT: **DRAW LABEL text**

Command parameters:

text

The text to be written to the display. Up to 80 characters of text may be specified.

DESCRIPTION:

The specified text is written to the display.

By default the position of the text is along the bottom line of the current plotting area, and is output in overlay colour 1.

Both the size and position of the text may be changed by means of the LABEL SIZE and LABEL POSITION commands, while the colour of annotation may be set using the LABEL COLOUR command.

Messages:

The following message is specific to the DRAW LABEL command.

*** ERROR *** Specifying command DRAW
Command qualifier is LABEL

Examples:

Rover> **DRAW LABEL A Very Pretty Picture<CR>**
Rover>

EDIT

Adds a point to the current line of editing pixels.

Only valid in the EDIT mode of operation

FORMAT: **EDIT**

Command parameters: None

DESCRIPTION:

The EDIT command is used to store the current matrix coordinates of the raster editing cursor, and all those pixels which join the current position to the previous position given with the latest EDIT command. Thus a series of EDIT and MOVE commands creates a line of pixels which may then be changed to a specific value on issue of the END command. Pixels which will be changed are highlighted in the current editing cursor colour.

During editing the program prompt is displayed as 'Line>' to indicate line editing is taking place. Line editing is the default mode of raster editing, but if the AREA_EDIT or SMOOTH_EDIT commands are used to initialise an editing operation then the EDIT command is also utilised.

This command is only available in EDIT mode of operation.

ABANDON or <CTRL/Z> (pressing the Ctrl and Z keys together) will abort the edit operation. Puck button 4 and mouse or function button 1 are defined to be EDIT.

Messages:

The following messages are specific to the EDIT command.

*** ERROR *** Specifying command EDIT
Command only available in EDIT mode

*** ERROR *** Specifying command EDIT
Selected DTI file is opened read-only

Examples:

Edit>**MOVE 10 10<CR>**
Edit>**EDIT<CR>**
Line>**MOVE 20 20<CR>**
Line>**EDIT<CR>**
Line>**END<CR>**
Line> DTI Value> **50<CR>**
Edit>

In this example a line of pixels joining matrix positions (10,10) and (20,20) are assigned the value of 50

Edit>**MOVE 50 50<CR>**
Edit>**EDIT<CR>**
Line>**END<CR>**
Line> DTI Value> **100<CR>**
Edit>

In this example a single pixel at matrix position (50,50) is assigned the value of 100.

ENABLE ABSOLUTE

Selects the use of absolute coordinates values.

FORMAT: **ENABLE ABSOLUTE**

Command Parameters: None

DESCRIPTION:

If ENABLE ABSOLUTE is given, then coordinate values required by the WINDOW or MOVE commands, supplied in metre or projection units, must be specified as absolute (rather than relative) coordinate values. Coordinates for the IFFWINDOW command should also be supplied as absolute values.

For example if the projection indicates U.K. National Grid, then the WINDOW values may be specified as 6 figure National Grid coordinates.
By default window values should be specified as absolute coordinates.

This option can be disabled using the DISABLE ABSOLUTE command. The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the ENABLE ABSOLUTE command.

```
*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE, TABLE or
TEXT
```

Examples:

Rover>**ENABLE ABSOLUTE<CR>**
Rover>

ENABLE COLOUR_MAP

Enables the use of a temporary colour mapped file for redisplay

---

FORMAT: **ENABLE COLOUR_MAP**

Command Parameters: None

DESCRIPTION:

This command enables the use of a temporary mapped DTI file to speed up raster display. It is recommended that this option is enabled if many ZOOM or WINDOW commands are to be given during the course of a ROVER session. After initial draw of a large window, the subsequent WINDOW and ZOOM commands make use of the temporary colour map for fast redisplay.

It should be noted that the use of the temporary colour map increases the time ROVER takes to exit because the file has to be unmapped. By default no temporary mapped file is used.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following messages are specific to the ENABLE and ENABLE COLOUR_MAP commands.

```
*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE, TABLE or
TEXT
```

```
*** ERROR *** Specifying command ENABLE COLOUR_MAP
Command not valid during editing operation
```

```
*** ERROR *** Specifying command ENABLE COLOUR_MAP
Command not valid during sampling
```

Examples:

Rover>**ENABLE COLOUR_MAP<CR>**
Rover>

ENABLE CROSS_CURSOR

Enables the use of the raster editing cross cursor.

Only valid in EDIT OR REGISTRATION modes of operation

FORMAT: **ENABLE CROSS_CURSOR**

Command Parameters: None

DESCRIPTION:

ENABLE CROSS_CURSOR selects the use of a vector cross cursor in EDIT mode of operation. The cross cursor is the same size regardless of the dimensions of the DTI window displayed. Thus it is useful if a large window is displayed and consequently the default square DTI-pixel sized cursor appears very small.

This command is only available in EDIT or REGISTRATION modes of operation.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following messages are specific to the ENABLE and ENABLE CROSS_CURSOR commands.

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT, DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND, MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE, TABLE or TEXT

*** ERROR *** Specifying command ENABLE CROSS_CURSOR
Command only available in EDIT or REGISTRATION modes

Examples:

Rover>**ENABLE CROSS_CURSOR<CR>**
Rover>

ENABLE DEFAULT

Controls for what RCP information the user is prompted, when the RCP ADD command is given.

The command ENABLE DEFAULT is only valid in REGISTRATION mode

---

FORMAT: ENABLE DEFAULT rcp_qualifier

Command Parameters:

rcp_qualifier

The name of a Registration Control Point attribute, chosen from:

ALL	All RCP attributes
RCP_ID	The RCP identifier
RCP_TYPE	The RCP type identifier.
RCP_WXY	The RCP XY coordinate weighting value
RCP_WZ	The RCP Z coordinate weighting value
RCP_Z	The RCP Z coordinate value

DESCRIPTION:

The DISABLE/ENABLE DEFAULT RCP commands controls for what information the user is prompted, when the RCP ADD command is given

If ENABLE DEFAULT is specified then the user is **NOT** prompted for the specified RCP attribute, and the default attribute value will be used. For example, if the command ENABLE DEFAULT RCP_TYPE has been given, then the user will not be prompted for a RCP type identifier, and the default RCP type identifier will be used. The SET DEFAULT RCP commands control the default value that will be used.

By default no prompting for additional RCP information occurs, and the RCP fields are assigned their default values.

The SHOW ENABLE command may be used to check on the status of the option when in Registration mode.

Messages:

The following messages are specific to the ENABLE and ENABLE DEFAULT commands:
*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT, DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND, MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE, TABLE or TEXT

*** ERROR *** Specifying command ENABLE DEFAULT
Command only valid in REGISTRATION mode

```
*** ERROR *** Specifying command ENABLE DEFAULT
Command qualifiers are ALL, RCP_ID, RCP_TYPE, RCP_WXY, RCP_WY or RCP_Z
```

Examples:

Registration>**ENABLE DEFAULT ALL<CR>**
Registration>**RCP ADD 1.2 3.4<CR>**
Registration>

ENABLE DIGITISE

Enables the ROVER digitising mode of operation.

FORMAT: **ENABLE DIGITISE**

Command Parameters: None

DESCRIPTION:

ENABLE DIGITISE enables the ROVER digitising mode of operation. Features may be digitised using the current settings with the SET FC, SET FSN and SET FC commands. A cursor is displayed which may be moved around the screen by pressing puck button 0. Commands START and END, which are issued from the cursor button, are used to create feature in the output file.

ENABLE DIGITISE may only be specified if a map has been setup using the SETUP MAP command.

Once this command has been issued the program is in DIGITISE mode until DISABLE DIGITISE is specified. The program prompt becomes 'Digitise>' until DIGITISE mode is exited.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following messages are specific to the ENABLE and ENABLE DIGITISE commands.

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT, DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND, MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE, TABLE or TEXT

*** ERROR *** Specifying command ENABLE DIGITISE
Command not available in EDIT mode.

*** ERROR *** Specifying command ENABLE DIGITISE
Command invalid without a map registered

Examples:

Rover>**ENABLE DIGITISE<CR>**
Digitise>

ENABLE DIVIDE

Enables divided screen mode of display.

FORMAT: **ENABLE DIVIDE**

Command Parameters: None

DESCRIPTION:

ENABLE DIVIDE selects the use of quartered screen display mode. Each quadrant is treated by the program as though it were a separate graphics screen. This allows four different files to be displayed simultaneously, or alternatively different representations of the same file may be shown.

The 4 quadrants are identified as follows:

1 = Top lefthand quadrant

2 = Top righthand quadrant

3 = Bottom righthand quadrant

4 = Bottom lefthand quadrant

Subsequent display commands display the files in the current quadrant specified with the QUADRANT command. The default quadrant is the top left corner (quadrant 1).

In EDIT mode this command causes automatic redisplay of the currently selected DTI file.

Messages:

The following messages are specific to the ENABLE and ENABLE DIVIDE commands.

```
*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE, TABLE or
TEXT
```

```
*** ERROR *** Specifying command ENABLE DIVIDE
Command not available during editing operation
```

Examples:

```
Rover>ENABLE DIVIDE<CR>
Rover>
```

ENABLE EDIT

Enables the ROVER raster editing mode of operation

FORMAT: **ENABLE EDIT**

Command Parameters: None

DESCRIPTION:

ENABLE EDIT enables the raster editing mode of operation. After this command has been specified then the various raster editing commands are available. The currently selected DTI file may be edited. A status display of the current editing cursor position and DTI value are shown. Thus this command is useful to allow interrogation of the values in the DTI file even if the file has only been opened for read with the FILEIN command. Actual editing commands may only be specified, however if the currently selected DTI file has been opened for write with the UPDATE_FILEIN command.

WARNING: Edits on DTI files are carried out in-situ. Thus edited DTI pixels are changed in the disk file immediately. To safeguard against possible file corruption, it is recommended that copies of files are made prior to a Rover editing session.

Once this command has been issued the program is in EDIT mode until DISABLE EDIT has been specified. The program prompt becomes 'Edit>' until EDIT mode is exited or an editing operation is carried out.

If the currently selected DTI file has not yet been displayed, then this command causes its automatic display.

Messages:

The following messages are specific to the ENABLE and ENABLE EDIT commands.

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT, DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND, MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE, TABLE or TEXT

*** ERROR *** Specifying command ENABLE EDIT
Command not available in REGISTRATION mode.

*** ERROR *** Specifying command ENABLE EDIT
Command not available in DIGITISE mode.

*** ERROR *** Specifying command ENABLE EDIT
Command requires that a DTI file is open

Examples:

```
Rover>ENABLE EDIT<CR>
Edit>
```

ENABLE FLOODING

Enables interactive flooding

Only valid in EDIT mode of operation

FORMAT: **ENABLE FLOODING**

Command Parameters: None

DESCRIPTION:

ENABLE FLOODING turns on the interactive flooding control from the mouse, trackerball or table. The effect of "flooding" is achieved by dynamically rewriting the colour table to blue. The extent of the flooding may be varied by moving the mouse or table puck in the Y direction. In the latter case a map should have been registered to provide limits for the flooding. The flooding takes place within the current RANGE settings and according to the current STEP intervals. The current flood level is displayed at the bottom of the screen.

During flooding the program prompt is displayed as 'Flood>'

ENABLE FLOODING is only available in EDIT mode.

Puck button E alternately enables and disables flooding.

Messages:

The following messages are specific to the ENABLE and ENABLE FLOODING commands.

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT, DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND, MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE, TABLE or TEXT

*** ERROR *** Specifying command ENABLE FLOODING
Command command only available in EDIT mode.

Examples:

Rover>**ENABLE FLOODING<CR>**
Flood>

ENABLE HATCH

Allows the user to set hatching parameters.

FORMAT: **ENABLE HATCH [line-width, line-separation]**

Command Parameters

line-width

the width of the hatched lines.A valid real number is required

line-separation

the separation of the hatched lines. A valid real number is required

---

DESCRIPTION:

ENABLE HATCH allows hatching parameters specified in the width and size entries in the FRT file to be overriden for vector display with the OVERLAY command.

The SHOW ENABLE command may be used to check on the status of the option.

Messages:

The following messages is specific to the ENABLE command.

```
*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE, TABLE or
TEXT
```

```
*** ERROR *** Specifying command ENABLE HATCH
Arguments should be positive real numbers
```

Examples:

Rover>**ENABLE HATCH 2.0 2.0<CR>**
Rover>

ENABLE HEIGHT

Allows vector text sizes to be taken from the TH entry of the IFF file.

FORMAT: **ENABLE HEIGHT**

Command Parameters: None

DESCRIPTION:

The ENABLE HEIGHT command specifies that text sizes are taken from the TH entry of the IFF file rather than from the FRT file.

The SHOW ENABLE command may be used to check on the status of the option.

Messages:

The following message is specific to the ENABLE command.

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT, DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND, MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE, TABLE or TEXT

Examples:

Rover>**ENABLE HEIGHT<CR>**
Rover>

ENABLE INTERPOLATION

Selects curve interpolation.

FORMAT: **ENABLE INTERPOLATION**

Command Parameters: None

DESCRIPTION:

ENABLE INTERPOLATION selects curve Interpolation of graphical type 6 features in the IFF file, using the current interpolation method specified with the INTERPOLATE command for vector overlay.

The SHOW ENABLE command may be used to check on the status of the option.

Messages:

The following message is specific to the ENABLE command.

```
*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE, TABLE or
TEXT
```

Examples:

```
Rover>ENABLE INTERPOLATION<CR>
Rover>
```

ENABLE LEGEND

Enables the output of a display legend

FORMAT: **ENABLE LEGEND**

Command Parameters: None

DESCRIPTION:

ENABLE LEGEND enables the display of a colour/value guide and an overlay colour guide when a DISPLAY command is given.

The position of the LEGEND on the screen may be changed with the LEGEND POSITION command.

Legend display is automatically disabled if a large window which cannot fit in the default display area is specified.

The SHOW ENABLE command may be used to check on the status of the option.

Messages:

The following message is specific to the ENABLE command.

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT, DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND, MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE, TABLE or TEXT

Examples:

Rover>**ENABLE LEGEND<CR>**
Rover>

ENABLE MASK

Enables vector drawing in Picture 1

FORMAT: **ENABLE MASK**

Command Parameters: None

DESCRIPTION:

By default the vector information is drawn to Picture 2 in one of the available overlay colours.

Using ENABLE MASK the vector information may be drawn in Picture 1 in the background colour. This allows the underlying image to be masked.

The SHOW ENABLE command may be used to check on the status of the option.

Messages:

The following message is specific to the ENABLE command.

```
*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE, TABLE or
TEXT
```

Examples:

Rover>**ENABLE MASK<CR>**
Rover>

ENABLE OVERRIDE

Enables features in a specified layer to be set to a supplied feature code.

FORMAT: **ENABLE OVERRIDE layer FC**

Command parameters:

layer

The layer number in which overriding is to take place. An integer value in the range 0 to 32767 is required

FC

The supplied feature code. An integer value in the range 0 to 32767 is required.

DESCRIPTION:

ENABLE OVERRIDE causes all feature codes for all features in the specified layer to be set to the supplied feature code value. This allows all features within a layer to be treated as identical for overlay purposes.

Up to 256 layer/feature code equations may be set up.

DISABLE OVERRIDE will cancel the settings.

The SHOW ENABLE command may be used to check on the status of the option.

Messages:

The following messages are specific to the ENABLE and ENABLE OVERRIDE commands.

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT, DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND, MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE, TABLE or TEXT

*** ERROR *** Specifying command ENABLE OVERRIDE
Command requires two integer arguments in the range 0 to 32767

*** ERROR *** Specifying command ENABLE OVERRIDE
Too many overrides specified

Examples:

```
Rover>ENABLE OVERRIDE 3 5<CR>
Rover>
```

ENABLE PATTERN

Cancels the effect of a previous DISABLE PATTERN command.

FORMAT: **ENABLE PATTERN**

Command Parameters

DESCRIPTION:

ENABLE PATTERN specifies that all lines specified as patterned in the FRT file will be drawn as patterned when the OVERLAY command is issued.

This is the default mode of operation

The SHOW ENABLE command may be used to check on the status of the option.

Messages:

The following message is specific to the ENABLE command.

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT, DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND, MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE, TABLE or TEXT

Examples:

Rover>**ENABLE PATTERN<CR>**
Rover>

ENABLE PICTURE

Selects the specified picture.

FORMAT: **ENABLE PICTURE picnum**

Command parameters:

picnum

The picture number. An integer value in the range 1 to 2 is required.

DESCRIPTION:

Picture 1 contains the raster information output using the DISPLAY command.

Picture 2 contains any vector information drawn using the OVERLAY command, and any user annotation drawn using the DRAW LABEL command.

ENABLE PICTURE makes the specified picture visible, after a previous DISABLE PICTURE command had made it invisible.

The SHOW ENABLE command may be used to check on the status of the 2 pictures.

Messages:

The following messages are specific to the ENABLE and ENABLE PICTURE commands.

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT, DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND, MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE, TABLE or TEXT

*** ERROR *** Specifying command ENABLE PICTURE
Command requires an integer in the range 1 to 2

Examples:

Rover>**ENABLE PICTURE 1<CR>**
Rover>

ENABLE POSITIONING

Specifies that vector text is positioned by the justification code.

FORMAT: **ENABLE POSITIONING**

Command Parameters: None

DESCRIPTION:

ENABLE POSITIONING enables the use of the justification code (0-8) stored in the FS entry for text positioning for vector overlay.

The SHOW ENABLE command may be used to check the status of the option.

Messages:

The following message is specific to the ENABLE command.

```
*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION,  SAMPLE,  TABLE  or
TEXT
```

Examples:

```
Rover>ENABLE POSITIONING<CR>
Rover>
```

ENABLE REGISTRATION

Enables the ROVER registration options and mode of operation.

FORMAT: **ENABLE REGISTRATION**

Command Parameters: None

DESCRIPTION:

The ENABLE REGISTRATION command is used to enter ROVER REGISTRATION mode. Registration mode is signified by the prompt 'Registration>' on the terminal.

After giving the command all registration related commands such as OPEN RCP_FILE and RCP ADD, are available.

The description section contains details on the purpose and operation of REGISTRATION mode.

Messages:

The following message is specific to the ENABLE command.

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE, TABLE or
TEXT

*** ERROR *** Specifying command ENABLE EDIT
Command requires that a DTI file is open

Examples:

Rover>**ENABLE REGISTRATION<CR>**
Registration>

ENABLE SAMPLE

Enables the use of sub-sampling during display.

FORMAT: **ENABLE SAMPLE**

Command Parameters: None

DESCRIPTION:

The ENABLE SAMPLE command enables sub-sampling of the DTI values during display. The column and row sample intervals may be set by the SAMPLE command.

The command also enables Rover to reset the sampling intervals automatically if the window, specified explicitly with the WINDOW command or implicitly with the ZOOM command, is too large to be displayed with the current sample values in the display area.

While sampling is enabled window values may be adjusted slightly to ensure that the the current window extents consist of a whole number of sample intervals. A warning message is issued when this occurs.

Note that the command ENABLE REGISTRATION automatically enables sampling, while ENABLE EDIT automatically disables sampling.

The SHOW ENABLE command may be used to check on the status of this option.

Messages:

The following messages are specific to the ENABLE and ENABLE SAMPLE commands.

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT, DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND, MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE, TABLE or TEXT

*** ERROR *** Specifying command ENABLE SAMPLE
Command invalid while COLOUR_MAP is enabled.

*** ERROR *** Specifying command ENABLE SAMPLE
Command not available in EDIT mode.

Examples:

Rover>**ENABLE SAMPLE<CR>**
Rover>

ENABLE TABLE

Initialises the Table Monitor to allow input from a digitising table.

FORMAT: **ENABLE TABLE**

Command Parameters: None

DESCRIPTION:

The ENABLE TABLE command initialises the table monitor to allow input from a digitising table. The command SETUP MAP may be used to register a map to the DTI file. In the event of failure, input from the digitising table or puck button is not possible, and the program will accept commands only from the terminal.

Table initialisation will generally fail because no Table Monitor is currently active, or because the Table Monitor is locked by another user. If neither of these reasons appear to apply, you should consult the TABLIB Reference Manual, or seek guidance from your system manager.

If the logical name LSL$AUTO_ENABLE_TABLE is defined with a value of "1", then the table is initialised on program startup.

The DISABLE TABLE command may be used to turn off input from the digitising table.

The SHOW ENABLE command may be used to check on the status of table input.

Messages:

The following message is specific to the ENABLE command.

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT,
DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND,
MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE, TABLE or
TEXT

Examples:

```
Rover>ENABLE TABLE<CR>
Rover>
```

ENABLE TEXT

Enables the output of border information.

FORMAT: **ENABLE TEXT**

Command Parameters: None

DESCRIPTION:

ENABLE TEXT enables the output of border information when a DISPLAY command is given. The DTI filename, area of interest, selected value range and number of colour levels used in the display image are output.

The SHOW ENABLE command may be used to check on the status of the option.

Messages:

The following message is specific to the ENABLE command.

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, COLOUR_MAP, CROSS_CURSOR, DEFAULT, DIGITISE, DIVIDE, EDIT, FLOODING, HATCH, HEIGHT, INTERPOLATION, LEGEND, MASK, OVERRIDE, PATTERN, PICTURE, POSITIONING, REGISTRATION, SAMPLE, TABLE or TEXT

Examples:

Rover>**ENABLE TEXT<CR>**
Rover>

END

Terminates the construction of a feature or ends the raster editing sequence.

Only available in EDIT or DIGITISE modes.

FORMAT: END [DTI-value]

Command parameters:

[DTI-value]

The DTI value to be assigned to the pixels which have been edited.

The range of the value which may be specified is dependent on the data type of the file which is being edited. A byte file requires an integer value in the range 0 to 255, while a word file requires an integer in the range -32767 to 32767.

Note that the command parameter is only valid in EDIT mode.

DESCRIPTION:

In the DIGITISE mode the END command ends the construction of the current feature. The digitised line is then stored in the output IFF file.

In EDIT mode END finishes the current editing sequence. The DTI-value supplied is assigned to the pixels which have been specified. If no parameter is supplied then the user is prompted for a DTI value.

The END command is available only if ENABLE EDIT or ENABLE DIGITISE commands have been specified.

END is defined as puck button 5

In EDIT mode mouse or function button 2 is defined as end.

Messages:

The following messages are specific to the END command.

*** ERROR *** Specifying command END
Command only available in EDIT or DIGITISE modes

*** ERROR *** Specifying command END
Command requires a word value in the range -32767 to 32766

*** ERROR *** Specifying command END
Command requires a byte value in the range 0 to 255

Examples:

Edit>**MOVE 10 10<CR>**
Edit>**EDIT<CR>**
Line>**MOVE 20 20<CR>**
Line>**EDIT<CR>**
Line>**END<CR>**
Line> DTI Value> **50<CR>**
Edit>

Digitise Point> **END<CR>**
Digitise>

ENLARGE

Magnifies the displayed image by a factor of two.

FORMAT: **ENLARGE**

Command Parameters: None

DESCRIPTION:

The screen image is magnified by a factor of 2 using pixel replication. If the command is repeatedly given the display will continue to be enlarged until the maximum magnification (x 16) is reached.

Puck button 2 is defined as ENLARGE.

This command has no effect on a graphics device which does not support hardware pixel replication. The ZOOM command offers an alternative way of enlarging the image on such a device.

Messages None
Display is at maximum magnification

Examples:

Rover>**ENLARGE<CR>**
Rover>

EXIT

Terminates program execution.

FORMAT: **EXIT**

Command Parameters: None

DESCRIPTION:

The program is terminated, and any currently opened DTI files will be unmapped, along with any input or output IFF files.

Messages None

Example:

Rover>**EXIT<CR>** $

FAST

Displays in FAST mode the currently selected DTI file.

FORMAT: **FAST**

Command Parameters: None

DESCRIPTION:

The FAST command may be used as an alternative to the DISPLAY command to output data more quickly to the graphics screen. The DTI file file values are output directly to the graphics device with no colour lookup calculations. Thus the current RANGE, STEP, COLOURS and FIRST_COLOUR values are not used. Because there is a limit of 255 colours which may be displayed, only files with byte datatype may be displayed with the FAST command.

The command requires that enough planes are set for the maximum value in the currently selected file to be output. The maximum value should be less than 2 raised to the power of (n) - 1, where n is the current planes setting. Thus a file with a maximum value of 64 requires a PLANES setting of 7, while a file with a maximum value of 63 requires only 6 planes to be used for the raster drawing.

The file is drawn in the direction specified by the latest ROTATE command. The default display direction is from the South. It should be noted that on some devices display speeds improve in certain directions.

Messages:

The following messages are specific to the FAST command.

*** WARNING *** FAST command ignored with non-byte data
Normal display taking place

*** WARNING *** Too few planes set for fast display
Normal display taking place

Examples:

Rover>**FAST<CR>**
Rover>

FILEIN

Opens a DTI file for read access

FORMAT: FILEIN file-spec

Command parameters:

file-spec

The file specification for the input DTI file. Any part of the file specification not supplied is taken from the default 'LSL$DTI:DTI.DTI'

DESCRIPTION:

This command opens and maps into memory a DTI file for read access. The file is opened on the lowest available DTI file number. The DTI file number is displayed on the terminal, and is the means by which the file is selected when a number of DTI files are opened.
Details derived from the header of the file are displayed on the terminal to confirm that the file has been successfully opened.

Up to 4 DTI files may be opened at one time. By default the DTI file opened with the last FILEIN or UPDATE_FILEIN command is the current DTI file.

If the logical name LSL$MATRIX_DEFAULT_WINDOW is defined with a value of "1", then a default area of interest is set for the file based on the maximum window which may be displayed in the default display area on the graphics device at four times magnification. Otherwise, if ENABLE SAMPLE has not been specified, a window corresponding to the entire file extents is set unless this is too large to be displayed in which case truncation takes place. The area of interest may be altered at any time by use of the WINDOW command.

If the rotation value (DTI_ORDER_CORNER) held in the header of the DTI file is not the default, (south west) then a warning message is output when the file is opened. Rover however, sets up the viewing direction taking into account the rotation value. Thus, provided the rotation value in the header is correct, Rover will display the file in the correct orientation. The ROTATE command be used to change the default viewing direction.

Information on the mapped DTI file may be obtained at any time by typing 'SHOW FILEIN'.

Note that editing operations may not be performed on a file that has been opened with FILEIN (cf UPDATE_FILEIN).

In EDIT and REGISTRATION modes issue of the FILEIN command results in automatic display of the file.

Messages:

The following messages are specific to the FILEIN command.

```
*** ERROR *** Specifying command FILEIN
Four DTI are currently opened
CLOSE command should be used

*** ERROR *** Specifying command FILEIN
Filename is missing
```

Example:

Rover>**FILEIN TEST<CR>**

```
*** File LSL$DTI:TEST.DTI opened as DTI file 1

File    :  LSL$DTI:TEST.DTI
Header  :  LSLA Data:  WORD

Units are DTI Matrix Value

Matrix Coverage   SW:     1      1  NE:   301   251
Matrix Window     SW:     1      1  NE:   301   251
Matrix Interval    E:     1          N:     1
Value Range        :     0    to    851
Rover>
```

FIRST_COLOUR

Sets the base colour for raster display for the currently selected DTI file.

FORMAT: **FIRST_COLOUR base-colour**

Command parameters:

base-colour

The base colours for raster display. An integer in the range 1 to 2 raised to the power n (where n is the current planes setting) should be supplied. The default value is 3.

DESCRIPTION:

FIRST_COLOUR defines the starting colour index for raster display of the currently selected DTI file. It enables, in conjunction with the COLOURS command, different parts of the colour table to be used to display different DTI files. (eg. a colour image alongside a grey scale image). The base colour itself is used for the display of zero values in the DTI matrix (normally blue for digital elevation models) and is not included in the actual colour indices. The current base colour setting may be examined using the SHOW PARAMETERS command.

Note that redefining the first colour, may change the step interval values previously set by the STEP command.

In EDIT mode of operation this command causes automatic redisplay of the currently selected DTI file

---

Messages:

The following messages are specific to the FIRST_COLOUR command.

*** ERROR *** Specifying command FIRST_COLOUR
Command requires an integer argument in range 2 to 64

*** ERROR *** Specifying command FIRST_COLOUR
Command not available during editing operation.

Examples:

```
Rover>SELECT FILE 1
Rover>PLANES 6
Rover>FIRST_COLOUR 3
Rover>COLOURS 29
Rover>DISPLAY
Rover>SELECT FILE 2
Rover>FIRST_COLOUR 33
Rover>COLOURS 31
Rover>DISPLAY
Rover>
```

In this example the PLANES 6 command allows 64 colours for raster display. The first DTI file is displayed using colours 3 to 32 in the colour table, while the second uses colours 33 to 64. Colours indices 1 and 2 (normally black and white) are reserved for specific use by ROVER.

FLOOD

Floods display image.

FORMAT: **FLOOD wait_time**

Command parameters:

wait_time

The time to wait in seconds between the flood steps. The default time is one second.

DESCRIPTION:

This is the Rover non-interactive flooding facility (cf ENABLE FLOODING). Each colour index is dynamically respecified to be blue. The effect is the successive recolouring of the displayed image until it becomes entirely blue. The time interval specified is the wait interval between the rewriting of each colour in the colour table.

To clear the effect of flooding a lookup table should be respecified with the LUT command.

Messages: None

Examples:

Rover>**FLOOD 3<CR>**
Rover>

FRT

Opens an FRT (Feature Representation Table) file

FORMAT: **FRT file-spec**

Command parameters:

file-spec

The file specification for the FRT file. Any part of the file specification not supplied will be taken from the default 'LSL$FRT:FRT.FRT'

DESCRIPTION:

This command selects and reads the contents of the specified Feature Representation Table.

The FRT is used by ROVER to look up the graphical type of a feature, and to consequently determine the graphical representation of the feature when drawn.

The table also defines feature code group names that may be supplied as parameters to the SELECT FC or DESELECT FC commands.

The command must be given before the OVERLAY command.

Messages:

The following message are specific to the FRT command.

*** ERROR *** Specifying command FRT
A system message will follow.

Examples:

Rover>**FRT TPCROV<CR>**
Rover>

HELP

Invokes on-line help

FORMAT: **HELP [command]**

Command parameters:

command

the command for which help is required

DESCRIPTION:

A brief description is given of the function and format of the specified command. If no parameter is specified then a complete list of available commands are output.

Messages: None

Examples:

```
Rover>HELP DISPLAY<CR>
Rover>
```

IFF

Opens an IFF (Internal Feature Format) file

FORMAT: IFF file-spec

Command parameters:

file-spec

The file specification of the input IFF file. Any part of the file specification not supplied will be taken from the default 'LSL$IF:IFF.IFF'

DESCRIPTION:

Selects and opens the IFF file containing the vector data to be drawn using the OVERLAY command. A default area of interest within the IFF file is set when the file is opened. This may be subsequently altered using the IFFWINDOW command. If origin offset values are present in the headers of the DTI and IFF files, then the IFF window is set with respect to the current DTI window. This allows registration between the raster image and the vector overlay to take place automatically. If no such header information is present, or if the DISABLE ABSOLUTE command has been given, then the default area of interest in the IFF file that corresponds to the IFF range entry (RA) values is set.

The IFF file range and IFF window may be examined using the command SHOW IFF.

Messages:

The following message are specific to the IFF command.

*** ERROR *** Specifying command IFF
Filename is missing

Examples:

Rover>**IFF ROADS_VECTOR<CR>**
Rover>

IFFWINDOW

Defines an area of interest in the IFF file.

FORMAT: **IFFWINDOW swx swy nex ney**

Command parameters:

swx swy nex ney

The IFF coordinates of the south-west (bottom lefthand) and north-west (top righthand) corners of the window. The coordinates are specified in IFF units, as real (floating point) values.

DESCRIPTION:

IFFWINDOW is used to define a rectangular area of interest within the IFF file. By default or if the ENABLE ABSOLUTE command has been given then window values should be supplied in absolute IFF units. Otherwise values should be specified as an offset from the SW corner of the IFF file area.

The IFFWINDOW command enables the IFF file to be registered to the current area of interest in the DTI file.
When the command is given, a transformation matrix for IFF to DTI coordinate conversion is calculated, based on the coordinates of the IFF and DTI windows.

The command SHOW IFF may be used to examine the IFF range and window coordinate values.

Before giving the command an IFF file must have been selected using the IFF command.

Messages:

The following error messages are specific to the IFFWINDOW command:

*** ERROR *** Specifying command IFFWINDOW
Command requires that an IFF file is open

*** ERROR *** Specifying command IFFWINDOW
IFF Window command requires 4 values

*** ERROR *** Specifying command IFFWINDOW
NE corner should exceed SW corner

Examples:

Rover>**IFFWINDOW 310000.0 220000.0 330000.0 240000.0 <CR>**
Rover>

INTERPOLATION AKIMA

Specifies the AKIMA interpolation method to be used for overlay of graphical type 6 curve features.

FORMAT: **INTERPOLATION AKIMA**

Command parameters: None

DESCRIPTION:

This command specifies the AKIMA interpolation method for the overlay of graphical type 6 curve features.

This is the default method of interpolation.

Messages:

The following message is specific to the INTERPOLATION command.

*** ERROR *** Specifying command INTERPOLATION
Command qualifiers are AKIMA or MCCONALOGUE

Examples:

Rover>**INTERPOLATION AKIMA <CR>**
Rover>

INTERPOLATION MCCONALOGUE

Specifies the MCCONALOGUE interpolation method to be used for overlay of graphical type 6 curve features.

FORMAT: **INTERPOLATION MCCONALOGUE**

Command parameters: None

DESCRIPTION:

This command specifies the MCCONALOGUE interpolation method for the overlay of graphical type 6 curve features.

The default method of interpolation is INTERPOLATION AKIMA.

Messages:

The following message is specific to the INTERPOLATION command.

*** ERROR *** Specifying command INTERPOLATION
Command qualifiers are AKIMA or MCCONALOGUE

Examples:

Rover>**INTERPOLATION MCCONALOGUE<CR>**
Rover>

LABEL COLOUR

Selects the colour of user labelling.

FORMAT: LABEL COLOUR overlay-colour

Command parameters:

overlay-colour

The colour value of the labels. An integer value in the range of 1 up to 7 is required depending on the current PLANES setting.

DESCRIPTION:

The colour of user labelling is selected. By default user labels will be drawn in overlay colour 1. If more than 1 overlay colour is available (dependent upon the current PLANES setting), another overlay colour may be selected using this command. The available colours are displayed in the legend.

Messages:

The following messages are specific to the LABEL and LABEL COLOUR commands.

```
*** ERROR *** Specifying command LABEL
Command qualifiers are POSITION, COLOUR, or SIZE
```

```
*** ERROR *** Specifying command LABEL COLOUR
Command requires an integer in the range 1 to "numcol"
```

Examples:

Rover>**LABEL COLOUR 3<CR>**
Rover>

LABEL POSITION

Positions the user annotation.

FORMAT: **LABEL POSITION screenx screeny**

Command parameters:

screenx, screeny

The screen coordinates in pixel units. Two real values are required.

DESCRIPTION:

The position of the user annotation is defined

The position is specified in screen pixel units, with respect to the SW corner of the screen.

Messages:

The following messages are specific to the LABEL and LABEL POSITION commands.

*** ERROR *** Specifying command LABEL
Command qualifiers are POSITION, COLOUR, or SIZE

*** ERROR *** Specifying command LABEL POSITION
Command requires 2 reals in the range 0 to <xscreensize> and 0 to <yscreensize>

Examples:

Rover>**LABEL POSITION 400 200<CR>**
Rover>

LABEL SIZE

Selects the size of user annotation

FORMAT: LABEL SIZE font-size

Command parameters:

font-size

The size of the user text. An integer in the range 1 (smallest) to 4 (largest) is required. The default size is 2.

DESCRIPTION:

The command LABEL SIZE is used to define the size of any text output to a graphics screen using the DRAW LABEL command.

The font_size parameter controls the relative size of the text; size 1 is the smallest, and size 4 is the largest. The actual size of the output text screen will vary with the size of the graphics device screen, though the relative sizes of the text will remain constant.

Messages:

The following messages are specific to the LABEL and LABEL SIZE commands.

*** ERROR *** Specifying command LABEL
Command qualifiers are POSITION, COLOUR, or SIZE

*** ERROR *** Specifying command LABEL SIZE
Command requires an integer in the range 1 to 4

Examples:

Rover>**LABEL SIZE 3<CR>**
Rover>

LEFT

Moves the raster editing cursor to the left by one pixel

Only available in EDIT or REGISTRATION modes of operation.

FORMAT: **LEFT**

Command parameters: None

DESCRIPTION:

LEFT moves the raster editing cursor to the left by one DTI pixel.

The command is only available in edit or registration modes of operation

Puck button 8 is defined as LEFT.

Messages:

The following message is specific to the LEFT command.

*** ERROR *** Specifying command LEFT
Command only available in EDIT or REGISTRATION modes

Examples:

Edit>**LEFT<CR>**
Edit>

LEGEND POSITION

Defines the position of the legend on the screen.

FORMAT: LEGEND POSITION screenx screeny

Command parameters:

screenx, screeny

The screen coordinates in pixel units. Two real values are required.

DESCRIPTION:

The position of the legend is defined

The position is specified in screen pixel units, with respect to the bottom left corner of the screen.

Messages:

The following messages are specific to the LEGEND and LEGEND POSITION commands.

*** ERROR *** Specifying command LEGEND
Command qualifier is POSITION

*** ERROR *** Specifying command LEGEND POSITION
Command requires 2 reals in the range 0 to <xscreensize> and 0 to <yscreensize>

Examples:

Rover>**LEGEND POSITION 400 200<CR>**
Rover>

LUT

Selects and makes current the specified colour table.

FORMAT: LUT file-spec

Command parameters:

file-spec

The file specification for the new colour table. Any part of the file specification not supplied will be taken from the default LSL$LOOKUP:.DAT.

DESCRIPTION:

This command is used to read in a new look up colour table. The colour table is a file containing the RGB definitions for a series of colour values. These values are used when images are displayed on the screen.

The default colour table which is read in on program initialisation is ROVER5.DAT, but these colour values may be subsequently changed by use of the LUT command.

The colour table that is read in should be appropriate for the current PLANES settings, in order to achieve correct display of both raster and vector information.

See the introductory description section for further information on the compilation of colour tables. An example lookup table is shown in Appendix A.

Messages:

The following message is specific to the LUT command.

*** ERROR *** Specifying command LUT
Filename is missing

Examples:

Rover>**LUT ROVERG6<CR>**
Rover>

MODTEN

Displays the currently selected DTI file using a Modulus of 10 function.

FORMAT: **MODTEN**

Command parameters: None

DESCRIPTION:

MODTEN causes redisplay of the currently selected DTI file using a remainder of a division by 10 function. The resulting image displays the DTI values in one of 10 shades of grey depending on the value of the last digit. Thus 53 and 73 are both displayed with the same grey value. Because local differences in DTI value are emphasised, MODTEN is a useful tool for terrain model validation.

MODTEN makes use of one of the standard greyscale ROVER lookup tables depending on the current PLANES setting. Subsequent displays after a MODTEN display revert to the original lookup table.

Messages: None

Examples:

Rover>**MODTEN<CR>**
Rover>

MESSAGE

Outputs the specified text to the terminal.

FORMAT: **MESSAGE text**

Command parameters:

text

The text to be output to the terminal. Up to 80 characters of text may be specified.

DESCRIPTION:

This command outputs the specified text to the terminal.

This command is useful to use in Rover Command Files to pass information before a RESPOND command returns control to the user.

Messages:

The following message is specific to the MESSAGE command.

*** ERROR *** Specifying command MESSAGE
"System error message"

Examples:

Rover>**MESSAGE ERROR HERE<CR>**
Rover>

MOVE

Moves the editing cursor.

Only available in EDIT and REGISTRATION modes of operation.

FORMAT: MOVE x_position y_position

Command parameters:

x_position y_position

The x and y coordinates of the position to move the cursor.

The format of the command parameters is controlled by the UNITS command.

UNITS MATRIX - 2 integer values are required defining the position of the target position in terms of row and column values.

UNITS METRES - 2 real (floating point) values are required defining the position as x and y metre offsets from the SW corner of the matrix. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates (eg. U.K. National Grid coordinates); if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

UNITS SECONDS - 2 real (floating point) values are required defining the absolute target position in seconds of arc. The values are supplied in the order latitude followed by longitude. A negative latitude value indicates a position in the Southern Hemisphere, and a negative longitude value a point to the west of Greenwich.

UNITS LATLONG - 2 values are required defining the absolute latitude and longitude position in the format DDD MM SS H where DDD is the number of degrees, MM is the number of minutes, SS is the number of seconds, and H is the hemisphere (N,S,E,W). The values are supplied in the order latitude followed by longitude.

UNITS PROJECTION Requires 2 real (floating point) values defining the target position in projection record units. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates; if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

Note that in all cases, the input values are adjusted to the nearest column and row values.

DESCRIPTION:

This command moves the DTI editing cursor to the specified coordinates. The status values on the screen reflect the current cursor position in DTI matrix coordinates and underlying colour. The SHOW POSITION command may be used to examine the position in the current UNITS of measurement.

If a position outside the current DTI window is specified then an implicit ZOOM 1 command around the new position is obeyed.

This command may also be issued by moving the mouse or trackerball within the displayed area.

This command is only available in EDIT and REGISTRATION modes of operation.

Puck button 0 is defined to be move if a map has been registered with the SETUP MAP command.

Messages:

The following messages are specific to the MOVE command.

```
*** ERROR *** Specifying command MOVE
Command requires 2 integer arguments

*** ERROR *** Specifying command MOVE
Supplied coordinates exceed file extents

*** ERROR *** Specifying command MOVE
Command only available in EDIT and REGISTRATION modes
```

Examples:

Rover>**MOVE 20 20<CR>**
Rover>

OPEN RCP_FILE

Opens an existing Registration Control Point (RCP) file.

The command OPEN RCP_FILE is only available in registration mode.

FORMAT: **OPEN RCP_FILE file-spec**

Command parameters:

file-spec

The RCP file specification. Any part of the file specification not supplied will be taken from the default 'LSL$RCP:RCP.RCP'.

DESCRIPTION:

The OPEN RCP_FILE command is used to open and read the contents of an existing Registration Control Point file. The file will most likely have been created during an earlier ROVER session.

Up to 4 RCP files may be opened. The SELECT RCP command controls which RCP file is current. By default, the current file is the most recent RCP file to be opened or created using the OPEN RCP_FILE and CREATE RCP_FILE commands. The contents of the current RCP file may be examined using the SHOW RCP command.

Entries in the current RCP file may be deleted or edited using the RCP DELETE and RCP EDIT commands, and new entries may be added to the file using the RCP ADD command.

Messages:

The following messages are specific to the OPEN and OPEN RCP_FILE commands:

```
*** ERROR *** Specifying command OPEN RCP_FILE
Command only valid in REGISTRATION mode

*** ERROR *** Specifying command OPEN RCP_FILE
RCP Filename is missing

*** ERROR *** Specifying command OPEN RCP_FILE
Maximum number of RCP files already open
Use the CLOSE RCP_FILE command to close a file

*** ERROR ***  Reading RCP file
File contains more than 200 RCPs

*** ERROR ***  Reading RCP file
RCP identifier missing for RCP number <RCP_NUMBER>

*** ERROR ***  Reading RCP file
```

```
RCP Type missing for RCP number <RCP_NUMBER>

*** ERROR ***  Reading RCP file
X value for RCP number <RCP_NUMBER> has invalid format

*** ERROR ***  Reading RCP file
Y value for RCP number <RCP_NUMBER> has invalid format

*** ERROR ***  Reading RCP file
Z value for RCP number <RCP_NUMBER> has invalid format

*** ERROR ***  Reading RCP file
XY weight value for RCP number <RCP_NUMBER> has invalid format

*** ERROR ***  Reading RCP file
Z weight value for RCP number <RCP_NUMBER> has invalid format

File <FILE_SPEC> opened as RCP File Number <RCP_FILE_NUMBER>
```

Examples:

```
Registration>OPEN RCP_FILE CONTROL<CR>
Registration>
```

OUTPUT

Specifies an output IFF file.

Only available in DIGITISE mode of operation.

FORMAT: OUTPUT file-spec

Command parameters:

file-spec

The output IFF file specification. Any part of the file specification not supplied will be taken from the default 'LSL$IF:IFF.IFF'.

DESCRIPTION:

An output IFF file can be specified to receive data generated using the program's digitising option using this command. A new output file is always opened. It is not possible to digitise to an existing file.

The output file uses the corner values and map descriptor information from the input IFF file if one is open. Digitised coordinates will also be in the units of the input IFF file. Otherwise, header information is copied from the projection record if one is present in the currently selected DTI file. Appropriate units are used depending on the type of the DTI file. If the input file is a TED4 or UHL1 file then units are in tenths of seconds. Projection units are used for DTI files containing a projection record. Otherwise DTI matrix units are used.

This command is only valid if a map has been registered to the DTI file. If an output IFF file is already open then it is first closed before the new file is open.

Messages:

The following messages are specific to the OUTPUT command.

*** ERROR *** Specifying command OUTPUT
Command invalid without a map registered

*** ERROR *** Specifying command OUTPUT
Command only available in DIGITISE mode

*** ERROR *** Specifying command OUTPUT
Filename is missing

Examples:

```
Rover>OUTPUT DIGSTUFF.IFF<CR>
Rover>
```

OVERLAY

Draws vector features to PICTURE 2

FORMAT: **OVERLAY**

Command parameters: None

DESCRIPTION:

OVERLAY causes any IFF features currently selected in the IFF file to be drawn in Picture 2 in the current quadrant position.

The graphical representation of features is determined by FRT lookup on the basis of their feature code. The colour used to draw the data may be set using the SET COLOUR command. The representation of features may also be changed by the ENABLE POSITIONING, ENABLE HEIGHT, ENABLE HATCH, ENABLE INTERPOLATION, ENABLE OVERRIDE, ENABLE TEXT and SCALE commands.

The displayed map is scaled so that it occupies the same area on the screen as the previously displayed raster information. Registration with the raster image is achieved by selecting an IFFWINDOW to cover the same area as the DTI window. This is achieved automatically if files contain absolute origin offset information in their headers.

Typing <CTRL/C> (Pressing the Ctrl and C keys together) causes the current OVERLAY operation to be aborted.

Messages:

The following messages are specific to the OVERLAY command.

*** ERROR *** Specifying command OVERLAY
Command requires that an IFF file is open

Examples:

Rover>**OVERLAY<CR>**
Rover>

PING

Outputs a warning sound at the terminal.

FORMAT: **PING**

Command parameters: None

DESCRIPTION:

This command causes a warning bell sound to be made at the terminal.

PING may be a useful command in ROVER Command Files to indicate that control has returned to the user after a RESPOND command has been issued.

Messages: None

Examples:

```
Rover>PING<CR>
Rover>
```

PLANES

Sets the number of planes assigned to picture 1.

FORMAT: PLANES numplanes

Command parameters:

numplanes

The number of bit planes to be assigned to picture 1. An integer value in the range 5 to 8 is required.

DESCRIPTION:

PLANES defines the number of bit-planes on a graphics device, and therefore the number of colours, that will be used to display the raster information in Picture 1.

By default a set of 5 contiguous planes are allocated to Picture 1. If 5 planes are defined, then 32 colours are available for the picture 1 although it should be noted that only 29 colours will actually be used for display, since the first 3 colours have a special use.

The remaining planes on an 8 bit-plane graphics device are used for the display of any vector overlay drawn using the OVERLAY command, and for any annotation generated using the DRAW LABEL command. With a planes setting of 5, 7 overlay and annotation colours are available. The LABEL COLOUR command may be used to select the annotation colour, while the colour entry in the FRT determines in what colour, different overlay features are shown.

It should be noted that after giving the PLANES command, a default colour table appropriate to the number of planes, is read. The effect of this will either be seen immediately if a DTI file is already displayed.

Further information on the default colour tables, and the use of the PLANES command may be found in the introductory description section.

Messages:

The following messages are specific to the PLANES command.

*** ERROR *** Specifying command PLANES
Command not available during an edit operation.

*** ERROR *** Specifying command PLANES
Command requires 1 integer argument in the range 5 to 8

Examples:

```
Rover>PLANES 5<CR>
Rover>PLANES 8<CR>
Rover>
```

QUADRANT

Makes the specified quadrant current.

FORMAT: QUADRANT quadrant_number

Command parameters:

quadrant_number

An integer value in the range 1 to 4 is required.

where	quadrant_number 1	=	Top lefthand quadrant
	quadrant_number 2	=	Top righthand quadrant
	quadrant_number 3	=	Bottom righthand quadrant
	quadrant_number 4	=	Bottom lefthand quadrant

DESCRIPTION:

This command enables the user to control which quadrant becomes current for display purposes.

The SHOW DEVICE command outputs information about which DTI file is currently associated with each quadrant.

This command is not valid in EDIT or REGISTRATION modes and is only valid if use of a quartered screen has been enabled using the ENABLE DIVIDE command.

Messages:

The following messages are specific to the QUADRANT command.

*** ERROR *** Specifying command QUADRANT
Command not available in EDIT or REGISTRATION modes.

*** ERROR *** Specifying command QUADRANT
Command requires an integer in the range 1 to 4

*** ERROR *** Specifying command QUADRANT
Command not valid unless the ENABLE DIVIDE command has been given

---

Examples:

Rover>**QUADRANT 4<CR>**
Rover>**ENLARGE <CR>**
Rover>**QUADRANT 2<CR>**
Rover>**DISPLAY<CR>**
Rover>

RANGE

Specifies a lower and upper range of DTI values for the current DTI file.

FORMAT: **RANGE lowval upval**

Command parameters:

lowval

The lower value of the required range of values.

upval

The upper value of the required range of values.

Two integer arguments are required within the header range of the IFF file. If the input file has real, or longword, datatype then the range is restricted between the values -32767 and 32766 even if the header values lie outside these limits.

DESCRIPTION:

The command is used to specify a lower and upper range of DTI values for the currently selected DTI file. Values outside this range will be displayed in black on the screen when the DISPLAY command is given.

Issue of the RANGE command always preserves the current STEP settings. If there are by consequence too few colours to show all the steps in the range, then the upper steps are displayed in white.

The current range values, along with the current range values stored in the file header and in the current DTI window, may be shown using the SHOW RANGES command.

In EDIT or REGISTRATION modes the RANGE command causes automatic redisplay of the currently selected DTI file.

Messages:

The following messages are specific to the RANGE command.

*** ERROR *** Specifying command RANGE
Command requires two integer arguments

*** ERROR *** Specifying command RANGE
Lower value exceeds upper value

*** ERROR *** Specifying command RANGE
Command not available during editing operation.

*** WARNING *** specified values exceed header range
Values will be truncated

*** WARNING *** specified values exceed program range limits of
-32766 to 32766
Values will be truncated

Examples:

Rover>**RANGE 400 700<CR>**
Rover>

RCP ADD

Adds a Registration Control Point to a RCP file.

The command RCP ADD is only available in registration mode.

FORMAT: RCP ADD x_coord y_coord

Command parameters:

x_coord

The x coordinate of the RCP.

y_coord

The y coordinate of the RCP.

The format of the command parameters is controlled by the UNITS command.

UNITS MATRIX - 2 real (floating point) values are required defining the position of the RCP in terms of columns and rows.

UNITS METRES - 2 real (floating point) values are required defining the position of the RCP in metre values. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates (eg. U.K. National Grid coordinates); if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

UNITS SECONDS - 2 real (floating point) values are required defining the absolute position of the RCP in seconds of arc. The values are supplied in the order latitude followed by longitude. A negative latitude value indicates a position in the Southern Hemisphere, and a negative longitude value a point to the west of Greenwich.

UNITS LATLONG - 2 values are required defining the absolute latitude and longitude position of the RCP in the format DDD MM SS H where DDD is the number of degrees, MM is the number of minutes, SS is the number of seconds, and H is the hemisphere (N,S,E,W). The values are supplied in the order latitude followed by longitude.

UNITS PROJECTION - 2 real (floating point) values defining the position of the RCP in DTI file projection record units (eg. mms on the map sheet). By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates; if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

DESCRIPTION:

The RCP ADD command is used to add a Registration Control Point to the currently selected RCP file.

The command parameters define the x and y position of the RCP. The parameters may be supplied on the command line, or may be recorded by pressing the middle button on the workstation mouse, or button 4 on the table puck if a map has been set up on the table. The coordinates of the RCP must lie within the geographical bounds of the DTM, or the corners of the map positioned on the table.

The DISABLE/ENABLE DEFAULT RCP commands control whether the user is prompted for additional RCP information. For example, if the command DISABLE DEFAULT RCP_ID has been given, then the user is prompted to enter an RCP identifier. The SET DEFAULT RCP commands control the default value that is supplied with a prompt, and which may be accepted using carriage return. By default no prompting for additional RCP information occurs, and the RCP fields are assigned their default values.

Messages:

The following messages are specific to the RCP and RCP ADD commands

```
*** ERROR *** Specifying command RCP
Command only valid in REGISTRATION mode

*** ERROR *** Specifying command RCP
No RCP files are currently open

*** ERROR *** Specifying command RCP
Currently selected RCP file is not open

*** ERROR *** Specifying command RCP
Command qualifiers are ADD, DELETE, EDIT or MOVE

*** ERROR *** Specifying command RCP ADD
Command requires 2 x y coordinates

*** ERROR *** Specifying command RCP ADD
RCP coordinates outside geographical bounds of DTI file

RCP with coordinates <X_COORDINATE> <Y_COORDINATE>
and ID <RCP_IDENTIFIER> added to RCP File Number <RCP_FILE>
```

Examples:

```
Registration>RCP ADD 1.2 1.3<CR>
RCP with coordinates 1.2 1.3
and ID 1          added to RCP File Number 1
Registration>DISABLE DEFAULT RCP_ID<CR>
Registration>RCP ADD 1.32 4.56<CR>
```

```
Enter RCP ID (Default = 2) TEST
RCP with coordinates 1.32 4.56
and ID TEST       added to RCP File Number 1
Registration>
```

RCP DELETE

Deletes a Registration Control Point in a RCP file.

The command RCP DELETE is only available in registration mode.

FORMAT: RCP DELETE rcp_identifier

Command parameters:

rcp_identifier

The RCP identifier. A string of up to 10 characters is required.

DESCRIPTION:

The RCP DELETE command is used to remove a Registration Control Point from the currently selected RCP file. The rcp_identifier determines which RCP is deleted.

Note that once deleted from the file, a RCP cannot be recovered.

Messages:

The following messages are specific to the RCP and RCP DELETE commands

*** ERROR *** Specifying command RCP
Command only valid in REGISTRATION mode

*** ERROR *** Specifying command RCP
No RCP files are currently open

*** ERROR *** Specifying command RCP
Currently selected RCP file is not open

*** ERROR *** Specifying command RCP
Command qualifiers are ADD, DELETE, EDIT or MOVE

*** ERROR *** Specifying command RCP DELETE
Command requires a RCP Identifier

*** ERROR *** Specifying command RCP DELETE
No RCP with identifier <RCP_IDENTIFIER> found in RCP file number <NUMBER>

RCP with ID <RCP_IDENTIFIER> deleted from RCP file number <NUMBER>

Examples:

```
Registration>RCP ADD 1.2 1.3<CR>
RCP with coordinates 1.2 1.3
and ID 1            added to RCP File Number 1
Registration>RCP DELETE 1<CR>
RCP with ID 1            deleted from RCP File number 1
Registration>
```

RCP EDIT

Edits an existing Registration Control Point.

The command RCP EDIT is only available in registration mode.

FORMAT: **RCP EDIT rcp_identifier**

Command parameters:

rcp_identifier

The RCP identifier. A string of up to 10 characters is required.

DESCRIPTION:

The RCP EDIT command is used to edit an existing Registration Control Point in the currently selected RCP file. The rcp_identifier determines which RCP is edited.

On giving the command, the screen cross cursor is positioned on the RCP location. (If the location is outside the current screen window, then a zoom with a factor of 1 is carried out around the RCP location). The user is then prompted for all RCP attributes, including a RCP identifier and RCP x y coordinates. The existing value or values are supplied as the default with each prompt, and may be accepted by pressing carriage return.

New RCP coordinates may be entered from the terminal, or by pressing the middle button on the workstation mouse, or button 4 on the table puck if a map has been set up on the table.

The new values (including the RCP identifier) replace the existing RCP values.

Messages:

The following messages are specific to the RCP and RCP EDIT commands

*** ERROR *** Specifying command RCP
Command only valid in REGISTRATION mode

*** ERROR *** Specifying command RCP
No RCP files are currently open

*** ERROR *** Specifying command RCP
Currently selected RCP file is not open

*** ERROR *** Specifying command RCP
Command qualifiers are ADD, DELETE, EDIT or MOVE

*** ERROR *** Specifying command RCP EDIT
Command requires a RCP Identifier

```
*** ERROR *** Specifying command RCP EDIT
No RCP with identifier <RCP_IDENTIFIER> found in RCP file number <NUMBER>

RCP with ID <RCP_IDENTIFIER> selected for edit
```

Examples:

```
Registration>RCP ADD 1.2 1.3<CR>
RCP with coordinates 1.2 1.3
and ID 1            added to RCP File Number 1
Registration>RCP EDIT 1<CR>
Enter RCP ID (Default = 1) ><CR>
Enter RCP Type (Default = AA) ><CR>
Enter X Y Coordinates (Default = 1.2 1.3) >1.45 4.6<CR>
Enter Z value (Default = 0.0) ><CR>
Enter X Y weight (Default = 1.0) ><CR>
Enter Z weight (Default = 0.0) ><CR>
Registration>
```

RCP MOVE

Moves the screen cursor to the specified Registration Control Point.

The command RCP MOVE is only available in registration mode.

FORMAT: RCP MOVE rcp_identifier

Command parameters:

rcp_identifier

The RCP identifier. A string of up to 10 characters is required.

DESCRIPTION:

The RCP MOVE command is used to position the screen cursor on a Registration Control Point recorded in the currently selected RCP file. For example, the command RCP MOVE 1 will move the cursor to the location of an RCP with an identifier of 1.

If the coordinates of the RCP define a point outside the current screen window, then a zoom with a factor of 1 is carried out around the RCP position.

Messages:

The following messages are specific to the RCP and RCP MOVE commands

*** ERROR *** Specifying command RCP
Command only valid in REGISTRATION mode

*** ERROR *** Specifying command RCP
No RCP files are currently open

*** ERROR *** Specifying command RCP
Currently selected RCP file is not open

*** ERROR *** Specifying command RCP
Command qualifiers are ADD, DELETE, EDIT or MOVE

*** ERROR *** Specifying command RCP MOVE
Command requires a RCP Identifier

*** ERROR *** Specifying command RCP MOVE
RCP lies outside the geographical bounds of the DTM

*** ERROR *** Specifying command RCP MOVE
No RCP with identifier <RCP_IDENTIFIER> found in RCP file number <NUMBER>

RCP with ID <RCP_IDENTIFIER> selected for edit

Examples:

```
Registration>RCP ADD 1.2 1.3<CR>
RCP with coordinates 1.2 1.3
and ID 1            added to RCP File Number 1
Registration>RCP MOVE 1<CR>
Registration>
```

REDUCE

Reduces the displayed image by a factor of two.

FORMAT: **REDUCE**

Command parameters: None

DESCRIPTION:

The screen image is reduced by a factor of 2 using pixel replication. If the command is repeatedly given the display will continue to be reduced until the minimum magnification is reached.

Puck button 3 is defined as REDUCE.

This command has no effect on a graphics device which does not support hardware pixel replication. The ZOOM command offers an alternative way of reducing the image on such a device.

Messages:
Display is at maximum magnification

Examples:

Rover>**REDUCE<CR>**
Rover>

REMOVE

Removes latest point from raster editing line.

Only available in EDIT mode of operation

FORMAT: **REMOVE**

Command parameters: None

DESCRIPTION:

Remove is the raster editing command to remove the latest edited point from the current line being edited. Thus it cancels the effect of the previous EDIT command, and all pixels on the line joining the current point to the previous point are redrawn in their original colour.

A series of REMOVE commands may be used to cancel several previous EDIT commands.

This command is only available in EDIT mode of operation.

Mouse or function button 3 and puck button 7 are defined to be REMOVE.

Messages:

The following message is specific to the REMOVE command.

*** ERROR *** Specifying command REMOVE
Command only available in EDIT mode

Examples:

Line>**REMOVE<CR>**
Line>

RESET_MINMAX

Updates the minimum and maximum heights in edited files.

FORMAT: **RESET_MINMAX**

Command parameters: None

DESCRIPTION:

RESET_MINMAX updates the minimum and maximum values in the headers of those files which have been opened for write access by the UPDATE_FILEIN command.

An Implicit RESET_MINMAX command is carried out by ROVER on issue of the DISABLE EDIT command, to ensure that any edits which have been performed on the file are reflected in the file header.

RESET_MINMAX also shows the current values of window, program and file ranges.

Messages:

The following message is specific to the RESET_MINMAX command.

```
*** ERROR *** Specifying command RESET_MINMAX
Command not available during editing operation
```

Examples:

```
Edit>RESET_MINMAX<CR>
Updating minimum and maximum values - Please wait
File Range          : 0   to 1344
Window Range        : 25  to 681
Current Range       : 400 to 700
Edit>
```

RESPOND

Returns program control to the terminal in a Rover Command File

---

FORMAT: **RESPOND**

Command parameters: None

DESCRIPTION:

RESPOND is used within a Rover Command File to pass control back to the terminal. The user may again type commands in response to the program prompt. Command file execution may be resumed on issue of the CONTINUE command. RESPOND is only valid within a Rover command file.

Messages:

The following message is specific to the RESPOND command

*** ERROR *** Specifying command RESPOND
Command only valid in Rover Command File

Example:

Rover>**RESPOND <CR>**
Rover>

RIGHT

Moves the raster editing cursor to the right by one pixel

Only available in EDIT or REGISTRATION modes of operation

FORMAT: **RIGHT**

Command parameters: None

DESCRIPTION:

RIGHT moves the raster editing cursor to the right by one DTI pixel.

The command is only available in EDIT or REGISTRATION modes of operation.

Puck button A is defined as RIGHT.

Messages:

The following message is specific to the RIGHT command.

*** ERROR *** Specifying command RIGHT
Command only available in EDIT or REGISTRATION modes

Examples:

Edit>**RIGHT<CR>**
Edit>

ROTATE

Rotate the raster display

FORMAT: ROTATE direction

Command parameters:

direction

A keyword describing the display direction. Valid keywords are NORTH, SOUTH, EAST or WEST.

DESCRIPTION:

This command allows the user to specify the direction from which the currently selected DTI file is to be displayed using the FAST and DISPLAY commands .

The default direction is determined from the rotation value (DTI_ORDER_CORNER) held in the header of an LSLA DTI file.

On some display devices display speeds may be better in certain directions than others.

Messages:

The following message is specific to the ROTATE command

*** ERROR *** Specifying command ROTATE
Command not available during edit operation.

*** ERROR *** Specifying command ROTATE
Command Keywords are NORTH, SOUTH, EAST or WEST

Examples:

Rover>**ROTATE EAST<CR>**
Rover>**FAST <CR>**
Rover>

SAMPLE

Defines the interval at which nodes will be sampled along the columns and rows of the DTI file.

FORMAT: **SAMPLE column_interval row_interval**

Command parameters:

column_interval

The sample interval along the columns of the DTI file. A positive integer value is required.

row_interval

The sample interval along the rows of the DTI file. A positive integer value is required. If this parameter is omitted then it is assumed to be equal to column_interval.

DESCRIPTION:

SAMPLE is used to define the intervals at which columns and rows of the currently selected DTI file are sampled during display.

By default all nodes in the current window are displayed. (ie. a sample interval of 1 along both the columns and rows).

The ENABLE SAMPLE command must be given before the defined sample values take effect.

Note that the sampling intervals may be automatically reset if a window, specified explicitly with the WINDOW command or implicitly with the ZOOM command, is too large to be displayed with the current sample values in the display area. A warning message is output when this occurs.

The current sample values may be viewed using the SHOW FILES command.

Messages:

The following message is specific to the SAMPLE command:

*** ERROR *** Specifying command SAMPLE
Command requires at least one integer argument

*** ERROR *** Specifying command SAMPLE
X sample interval should be in the range 1 to N

*** ERROR *** Specifying command SAMPLE
Y sample interval should be in the range 1 to N

*** ERROR *** Specifying command SAMPLE

Command requires that a DTI file is open

*** ERROR *** Specifying command SAMPLE
Command not available in EDIT mode.

Examples:

Rover>**SAMPLE 2 <CR>**
Rover>

SCALE

Sets the scaling factor for symbols and text

FORMAT: **SCALE factor**

Command parameters:

factor

The text and symbol scale factor. A positive real value is required.

DESCRIPTION:

This command is used to scale the symbols, text and hatching parameters for output of vector information with the OVERLAY command. The values specified in the FRT table are enlarged by the specified factor.

Messages:

The following message is specific to the SCALE command:

```
*** ERROR *** Specifying command SCALE
Command requires a positive real number
```

Examples:

```
Rover>SCALE 2.5<CR>
Rover>
```

SELECT ALL

Resets all feature selections

FORMAT: **SELECT ALL**

Command parameters: None

DESCRIPTION:

This command resets all features to be selected for overlay. If features are subsequently selected using the other SELECT commands then all features are first implicitly deselected.

This command is only valid after a FRT file has been specified.

Messages:

The following message is specific to the SELECT command.

*** ERROR *** Specifying command SELECT
Command qualifiers are ALL, FC, FILEIN, FSN, LAYER or RCP_FILE

*** ERROR *** Specifying command SELECT ALL
Command invalid before FRT has been read

Examples:

Rover>**SELECT ALL <CR>**

At this point all features are selected.

Rover>**SELECT FSN 7-10,56-78**

Here only the specified features are selected

Rover>**SELECT FSN 11-20**

At this point the the specified features are added to the currently selected features. ie features with FSN 7-20 and 56-78 are now selected.

Rover>

SELECT FC

Defines which IFF features are overlaid on the basis of feature code.

FORMAT: SELECT FC feature_code [,...]

Command parameters:

feature_code

An integer feature code in the range 0 to 32767. Multiple feature codes may be specified separated by commas or spaces, while a range of feature codes may be specified by means of a '-'. eg. SELECT FC 10-13 selects feature codes 10,11,12 and 13.
Alternatively a valid FRT group name may be specified.

DESCRIPTION:

The SELECT FC command is used to include IFF features for vector overlay. Features are included on the basis of their feature code.

By default ROVER will use all features within an IFF file.

Use of the SHOW SELECTIONS command is recommended to display feature selections before the OVERLAY command is given.

The SELECT FC command is only valid if the FRT command has been used to select a Feature Representation Table.

Messages:

The following messages are specific to the SELECT and SELECT FC commands.
*** ERROR *** Specifying command SELECT
Command qualifiers are ALL, FC, FILEIN, FSN, LAYER or RCP_FILE

*** ERROR *** Specifying command SELECT FC
Command invalid before FRT has been read

*** ERROR *** Specifying command SELECT FC
No groups have been defined in the FRT

*** ERROR *** Specifying command SELECT FC
Max number of SEL/DES commands allowed exceeded

*** ERROR *** Specifying command SELECT FC
Illegal feature code "feature_code"

*** ERROR *** Specifying command SELECT FC
Bad group name "group"

Examples:

Rover>**SELECT FC 1<CR>**
Rover>**SELECT FC RAILWAYS<CR>**
Rover>**SELECT FC RAIL 7-10,56-78**
Rover>

SELECT FILEIN

Selects the DTI file with the assigned number.

FORMAT: SELECT FILEIN file-number

Command parameters:

file-number

The assigned DTI file number. An integer in the range 1 to 4 is required.

DESCRIPTION:

This command sets the DTI file with the assigned number to be the current DTI file. Commands such as WINDOW, UNITS, STEP, RANGE, FIRST and COLOURS are applied to the DTI file currently selected using the SELECT FILEIN command .

If EDIT mode is enabled then editing may only be performed on the currently selected DTI file. In quartered screen mode, this command allows up to 4 different files to be edited on the same display. The currently selected file has the Editing status information displayed at the bottom of the quadrant. If the currently selected DTI file is not displayed then issue of this command causes automatic DISPLAY.

By default the current DTI file is the file last opened using the FILEIN command.

Messages:

The following messages are specific to the SELECT and SELECT FILEIN commands.

*** ERROR *** Specifying command SELECT
Command qualifiers are ALL, FC, FILEIN, FSN, LAYER or RCP_FILE

*** ERROR *** Specifying command SELECT FILEIN
Command requires an integer argument in the range 1 to 4

*** ERROR *** Specifying command SELECT FILEIN
Command not valid during edit operation

*** ERROR *** Specifying command SELECT FILEIN
Input DTI file not yet opened

Examples:

Rover>**SELECT FILEIN 3<CR>**
Rover>**SELECT FILEIN 2<CR>**
Rover>

SELECT FSN

Defines which IFF features are overlaid on the basis of feature serial number.

FORMAT: **SELECT FSN fsn [,...]**

Command parameters:

fsn

An integer feature serial number in the range 0 to 65534. Multiple feature serial numbers may be specified separated by commas or spaces, while a range of feature serial numbers may be specified by means of a '-'. eg. SELECT FSN 10-13 selects feature serial numbers 10,11,12 and 13.

DESCRIPTION:

The SELECT FSN command is used to include IFF features for vector overlay. Features are included on the basis of their feature serial number.

By default ROVER will use all features within an IFF file.

Use of the SHOW SELECTIONS command is recommended to display feature selections before the OVERLAY command is given.

The SELECT FSN command is only valid if an IFF file has been specified and the FRT command has been used to select a Feature Representation Table.

Messages:

The following messages are specific to the SELECT and SELECT FSN commands.

```
*** ERROR *** Specifying command SELECT
Command qualifiers are ALL, FC, FILEIN, FSN, LAYER or RCP_FILE
```

```
*** ERROR *** Specifying command SELECT FSN
Command invalid before FRT has been read
```

```
*** ERROR *** Specifying command SELECT FSN
Illegal FSN number "fsn"
```

Examples:

Rover>**SELECT FSN 1<CR>**
Rover>**SELECT FSN 7-10,56-78**
Rover>

SELECT LAYER

Defines which IFF features are overlaid on the basis of layer number.

FORMAT: SELECT LAYER layer [,...]

Command parameters:

layer

An integer feature serial number in the range 0 to 32767. Multiple layer numbers may be specified separated by commas or spaces, while a range of layer numbers may be specified by means of a '-'. eg. SELECT LAYER 10-13 selects feature serial numbers 10,11,12 and 13.

DESCRIPTION:

The SELECT LAYER command is used to include IFF features for vector overlay. Features are included on the basis of their layer number.

By default ROVER will use all features within an IFF file.

Use of the SHOW SELECTIONS command is recommended to display feature selections before the OVERLAY command is given.

The SELECT LAYER command is only valid if the FRT command has been used to select a Feature Representation Table.

Messages:

The following messages are specific to the SELECT and SELECT LAYER commands.

*** ERROR *** Specifying command SELECT
Command qualifiers are ALL, FC, FILEIN, FSN, LAYER or RCP_FILE

*** ERROR *** Specifying command SELECT LAYER
Command invalid before FRT has been read

*** ERROR *** Specifying command SELECT LAYER
Illegal LAYER number "layer"

Examples:

Rover>**SELECT LAYER 1<CR>**
Rover>**SELECT LAYER 2-5,7-9**
Rover>

SELECT RCP_FILE

Selects and makes current a Registration Control Point (RCP) file.

The command SELECT RCP_FILE is only available in registration mode.

FORMAT: **SELECT RCP_FILE number**

Command parameters:

number

The number of the RCP file. An integer value in the range 1 to 4 is required.

DESCRIPTION:

The SELECT RCP_FILE command is used to select and make current a Registration Control Point (RCP) file.

The commands RCP ADD, RCP DELETE, RCP EDIT and RCP MOVE operate on entries only in the currently selected RCP file.

By default, the RCP file most recently created or opened using the CREATE RCP_FILE or OPEN RCP_FILE commands is selected as the current RCP file.

Messages:

The following messages are specific to the SELECT and SELECT RCP_FILE commands.

*** ERROR *** Specifying command SELECT
Command qualifiers are ALL, FC, FILEIN, FSN, LAYER or RCP_FILE

*** ERROR *** Specifying command SELECT RCP_FILE
Command only valid in REGISTRATION mode

*** ERROR *** Specifying command SELECT RCP_FILE
Command requires a RCP file number

*** ERROR *** Specifying command SELECT RCP_FILE
Command requires a RCP file number in the range 1 to 4

*** ERROR *** Specifying command SELECT RCP_FILE
RCP file number <RCP_FILE_NUMBER> is not open

Examples:

```
Registration>SELECT RCP_FILE 3<CR>
Registration>
```

SET COLOUR

Sets the colour for vector overlay.

FORMAT: SET COLOUR overlay_colour

Command parameters:

overlay_colour

The overlay colour. An integer value in the range of 0 up to 7 is required depending on the current PLANES setting. Alternatively the FRT keyword may be specified.

DESCRIPTION:

SET COLOUR determines the colour of any vector information subsequently output to Picture 2. The available colours are displayed in the legend. SET COLOUR FRT bases colour selection on the FRT lookup and is the default mode of operation.

The current overlay colour is displayed with the SHOW PARAMETERS command.

SET COLOUR 0 sets the overlay to the background colour, and may be used to selectively erase overlaid features in Picture 2. Note that if no information has been overlaid, setting the colour to 0 will result in no information being drawn.

Messages:

The following messages are specific to the SET and SET COLOUR commands.

*** ERROR *** Specifying command SET
Command qualifiers are COLOUR, CURSOR_COLOUR, DEFAULT, DELAY, FC,
FSN, or LAYER

*** ERROR *** Specifying command SET COLOUR
Command requires an integer in the range 0 to "numcol" or the FRT qualifier

---

Examples:

Rover>**SET COLOUR 4<CR>**
Rover>**SET COLOUR FRT<CR>**
Rover>

SET CURSOR_COLOUR

Sets the colour of the square raster editing cursor.

The command is only available in EDIT mode of operation

FORMAT: **SET CURSOR_COLOUR colour**

Command parameters:

colour

The colour of the cursor. An integer in the range of 1 to 2 raised to the power n (where n is the current planes setting) should be supplied. eg. If PLANES 6 has been specified then a value between 1 and 61 may be specified. The default value is 1.

DESCRIPTION:

This command allows the square raster editing cursor to be changed in colour. By default the cursor is drawn in colour 1 of the colour table which is black in the standard ROVER colour tables. Any of the current PICTURE 1 colours may however be selected using this command.

Messages:

The following messages are specific to the SET and SET CURSOR_COLOUR commands.

*** ERROR *** Specifying command SET
Command qualifiers are COLOUR, CURSOR_COLOUR, DEFAULT, DELAY, FC,
FSN, or LAYER

*** ERROR *** Specifying command SET CURSOR_COLOUR
Command requires an integer argument in range 1 to"numcol"

*** ERROR *** Specifying command SET CURSOR_COLOUR
Command only available in EDIT mode

Examples:

Rover>**SET CURSOR_COLOUR 20<CR>**
Rover>

SET DEFAULT RCP_TYPE

Sets the default RCP_TYPE identifier.

The command SET DEFAULT RCP_TYPE is only available in REGISTRATION mode.

FORMAT: SET DEFAULT RCP_TYPE identifier

Command parameters:

identifier

The RCP Type identifier. A character string of 1 or 2 characters is required.

DESCRIPTION:

The SET DEFAULT RCP_TYPE command allows a user to define the default RCP Type identifier.

This identifier will be automatically attached to a Registration Control Point when the RCP ADD is given, unless DISABLE DEFAULT RCP_TYPE has been specified. If the DISABLE DEFAULT RCP_TYPE command has been given, the RCP Type identifier will be supplied as a default value that may be accepted by pressing carriage return in response to the prompt 'Enter RCP TYPE>'

The default RCP_TYPE identifier is XX.

The current RCP_TYPE value may be examined using the SHOW SETTINGS command when in REGISTRATION mode.

Messages:

The following messages are specific to the SET, SET DEFAULT and SET DEFAULT RCP_TYPE commands:

*** ERROR *** Specifying command SET
Command qualifiers are COLOUR, CURSOR__COLOUR, DEFAULT, DELAY, FC,
FSN or LAYER

*** ERROR *** Specifying command SET DEFAULT
Command qualifiers are RCP_TYPE, RCP_Z, RCP_WZ, or RCP_WXY

*** ERROR *** Specifying command SET DEFAULT
The command is only valid in REGISTRATION mode

*** ERROR *** Specifying command SET DEFAULT RCP_TYPE
Command requires a RCP Type value

Examples:

```
Registration>SET DEFAULT RCP_TYPE AA<CR>
Registration>
```

SET DEFAULT RCP_WXY

Sets the default RCP XY coordinate weighting value.

The command SET DEFAULT RCP_WXY is only available in REGISTRATION mode.

FORMAT: SET DEFAULT RCP_WXY weight

Command parameters:

weight

The RCP XY coordinate weight. A real (floating) point value is required.

DESCRIPTION:

The SET DEFAULT RCP_WXY command allows a user to define the default RCP XY coordinate weight. The weight value is used to assign a confidence factor to the RCP x and y coordinates.

The weight value will be automatically attached to a Registration Control Point when the RCP ADD is given, unless DISABLE DEFAULT RCP_WXY has been specified. If the DISABLE DEFAULT RCP_WXY command has been given, the weight value will be supplied as a default value that may be accepted by pressing carriage return in response to the prompt 'Enter XY weight >'

The default RCP_WXY value is 1.0

The current RCP_WXY value may be examined using the SHOW SETTINGS command when in REGISTRATION mode.

Messages:

The following messages are specific to the SET, SET DEFAULT and SET DEFAULT RCP_WXY commands:

*** ERROR *** Specifying command SET
Command qualifiers are COLOUR, CURSOR__COLOUR, DEFAULT, DELAY, FC,
FSN or LAYER

*** ERROR *** Specifying command SET DEFAULT
Command qualifiers are RCP_TYPE, RCP_Z, RCP_WZ, or RCP_WXY

*** ERROR *** Specifying command SET DEFAULT
The command is only valid in REGISTRATION mode

*** ERROR *** Specifying command SET DEFAULT RCP_WXY
Command requires an XY weight value

Examples:

```
Registration>SET DEFAULT RCP_WXY 0.5<CR>
Registration>
```

SET DEFAULT RCP_WZ

Sets the default RCP Z coordinate weighting value.

The command SET DEFAULT RCP_WZ is only available in REGISTRATION mode.

FORMAT: **SET DEFAULT RCP_WZ weight**

Command parameters:

weight

The RCP Z coordinate weight. A real (floating) point value is required.

DESCRIPTION:

The SET DEFAULT RCP_WZ command allows a user to define the default RCP Z coordinate weight. The weight value is used to assign a confidence factor to the RCP Z coordinate.

The weight value will be automatically attached to a Registration Control Point when the RCP ADD is given, unless DISABLE DEFAULT RCP_Z has been specified. If the DISABLE DEFAULT RCP_Z command has been given, the weight value will be supplied as a default value that may be accepted by pressing carriage return in response to the prompt 'Enter Z weight >'

The default RCP_WZ value is 0, unless a RCP Z coordinate has been specified, when a weighting value of 1.0 is assumed.

The current RCP_WZ value may be examined using the SHOW SETTINGS command when in REGISTRATION mode.

Messages:

The following messages are specific to the SET, SET DEFAULT and SET DEFAULT RCP_WZ commands:

```
*** ERROR *** Specifying command SET
Command qualifiers are COLOUR, CURSOR__COLOUR, DEFAULT, DELAY, FC,
FSN or LAYER

*** ERROR *** Specifying command SET DEFAULT
Command qualifiers are RCP_TYPE, RCP_Z, RCP_WZ, or RCP_WXY

*** ERROR *** Specifying command SET DEFAULT
The command is only valid in REGISTRATION mode

*** ERROR *** Specifying command SET DEFAULT RCP_WZ
Command requires a Z weight value
```

Examples:

```
Registration>SET DEFAULT RCP_WZ 0.5<CR>
Registration>
```

SET DEFAULT RCP_Z

Sets the default RCP Z coordinate value.

The command SET DEFAULT RCP_Z is only available in REGISTRATION mode.

FORMAT: SET DEFAULT RCP_Z coordinate

Command parameters:

coordinate

The RCP Z coordinate value. A real (floating) point value is required.

DESCRIPTION:

The SET DEFAULT RCP_Z command allows a user to define the default RCP Z coordinate value.

The Z value will be automatically attached to a Registration Control Point when the RCP ADD is given, unless DISABLE DEFAULT RCP_Z has been specified. If the DISABLE DEFAULT RCP_Z command has been given, the Z value will be supplied as a default value that may be accepted by pressing carriage return in response to the prompt 'Enter Z value >'

The default RCP_Z value is 0

The current RCP_Z value may be examined using the SHOW SETTINGS command when in REGISTRATION mode.

Messages:

The following messages are specific to the SET, SET DEFAULT and SET DEFAULT RCP_Z commands:

*** ERROR *** Specifying command SET
Command qualifiers are COLOUR, CURSOR__COLOUR, DEFAULT, DELAY, FC,
FSN or LAYER

*** ERROR *** Specifying command SET DEFAULT
Command qualifiers are RCP_TYPE, RCP_Z, RCP_WZ, or RCP_WXY

*** ERROR *** Specifying command SET DEFAULT
The command is only valid in REGISTRATION mode

*** ERROR *** Specifying command SET DEFAULT RCP_WXY
Command requires a Z value

Examples:

Registration>**SET DEFAULT RCP_Z 100.0<CR>**
Registration>

SET DELAY

Sets the delay time interval between editing cursor updates

The command is only available in EDIT or REGISTRATION modes of operation

---

FORMAT: **SET DELAY delay_time**

Command parameters:

delay_time

The time delay in milliseconds. A positive integer is required. The default value is 100.

DESCRIPTION:

The SET DELAY command may be used to control the delay time between successive cursor updates on the screen. If the cursor seems to lag behind the movement of the mouse or trackerball, then the default value of 100 milliseconds should be increased.

A suitable delay setting may depend on the size of the window displayed and also on the amount of graphics activity taking place on the workstation.

Messages:

The following messages are specific to the SET and SET DELAY commands.

*** ERROR *** Specifying command SET
Command qualifiers are COLOUR, CURSOR_COLOUR, DEFAULT, DELAY, FC,
FSN, or LAYER

*** ERROR *** Specifying command SET DELAY
Command requires a positive integer

Examples:

Rover>**SET DELAY 200<CR>**
Rover>

SET FC

Sets IFF feature code for output.

Only available in the DIGITISE mode of operation.

FORMAT: **SET FC number**

Command parameters:

number

The feature code. An integer in the range 0 to 32767 is required.

DESCRIPTION:

This command enables the feature code to be set for the next feature to be constructed. This command is only available in the DIGITISE mode operation.

Messages:

The following messages are specific to the SET and SET FC commands.

*** ERROR *** Specifying command SET
Command qualifiers are COLOUR, CURSOR_COLOUR, DEFAULT, DELAY, FC,
FSN, or LAYER

*** ERROR *** Specifying command SET FC
Command only available in DIGITISE mode

*** ERROR *** Specifying command SET FC
Command requires integer in the range 0 to 32767

Examples:

Rover>**SET FC 5<CR>**
Rover>

SET FSN

Sets IFF feature serial number for output.

Only available in the DIGITISE mode of operation

FORMAT: SET FSN number

Command parameters:

number

The feature serial number. An integer in the range 0 to 65534 is required.

DESCRIPTION:

The command enables the feature serial number to be set for the next feature to be constructed. The command is only available in DIGITISE mode of operation.

Messages:

The following messages are specific to the SET and SET FSN commands.

*** ERROR *** Specifying command SET
Command qualifiers are COLOUR, CURSOR_COLOUR, DEFAULT, DELAY, FC,
FSN, or LAYER

*** ERROR *** Specifying command SET FSN
Command only available in DIGITISE mode

*** ERROR *** Specifying command SET FSN
Command requires integer in the range 0 to 65534

Examples:

Rover>**SET FSN 3<CR>**
Rover>

SET LAYER

Sets layer number for output.

Only available in the DIGITISE mode of operation.

FORMAT: **SET LAYER number**

Command parameters:

number

The layer number. An integer in the range 0 to 32767 is required.

DESCRIPTION:

The command enables the layer number to be set for the next feature to be constructed. It is only available in DIGITISE mode of operation.

Messages:

The following messages are specific to the SET and SET LAYER commands.

```
*** ERROR *** Specifying command SET
Command qualifiers are COLOUR, CURSOR_COLOUR, DEFAULT, DELAY, FC,
FSN, or LAYER
```

```
*** ERROR *** Specifying command SET LAYER
Command only available in DIGITISE mode
```

```
*** ERROR *** Specifying command SET LAYER
Command requires integer in the range 0 to 32767
```

Examples:

Rover>**SET LAYER 3<CR>**
Rover>

SETUP MAP

Allows a map to be registered to the currently selected DTI file.

FORMAT: **SETUP MAP**

Command parameters: None

DESCRIPTION:

The SETUP MAP command allows a source document to be registered to the currently selected DTI file.

In order to register the map and a DTI file , the file should have first been specified, and a source document should have been securely attached to the surface of a digitising table.

On giving the command you will be asked to digitise 4 rectangular registration points. The registration points represent 4 points on the map that correspond to the 4 corners of the DTI file. The points are digitised in the order top left (NW), bottom left (SW), bottom right (SE) and top right (NE) using any button on the table puck. A point should be digitised in response to a prompt on the terminal.
An error message is generated if any of the angles of the digitised rectangle are less than 88 degrees, or greater than 92 degrees (ie. if the corner points of the rectangle are more than 2 degrees off rectangular). In this case you will be asked to redigitise the 4 registration points.
Setup of the map may be aborted using <CTRL/Z> (pressing the Ctrl and Z keys together), or by pressing Button F on the table puck.

Following the registration of a map to the file , coordinate values required by the WINDOW and CENTRE commands, may be input using an appropriate button on the table puck.

Map registration also allows the DIGITISE mode of operation to be invoked with the ENABLE DIGITISE command.

If EDIT or REGISTRATION modes are enabled then cursor movement may be achieved using table puck coordinate input.

Messages:

The following messages are specific to the SETUP MAP commands

*** ERROR *** Specifying command SETUP
SETUP qualifier is MAP

*** ERROR *** Specifying command SETUP MAP
Command not valid without table

*** ERROR *** Specifying command SETUP MAP

No input file yet opened

*** ERROR *** digitising corners
More than 2 degrees off square - repeat

Examples:

Rover>**SETUP MAP<CR>**
Map NW Corner>
Map SW Corner>
Map SE Corner>
Map NE Corner>

SHOW ANNOTATION

Shows information about legend and user annotation information.

FORMAT: **SHOW ANNOTATION**

Command parameters: None

DESCRIPTION:

Details of labelling and legend positions for the currently selected DTI file are output to the terminal. The label size and colour index are also displayed.

Messages:

The following message is specific to the SHOW command.

*** ERROR *** Specifying command SHOW
Command qualifiers are ANNOTATION, DEVICE, ENABLE, FILEIN,
IFF, PARAMETERS, POSITION, RANGES, RCP_FILE, SELECTIONS or SETTINGS

Examples:

Rover>**SHOW ANNOTATION<CR>**

```
Current label  setting:
Position: X 400  Y 500   Size 3  Colour 2
Current legend setting:
Position: X 100 Y 100
Rover>
```

SHOW DEVICE

Outputs information about current screen-file allocation

FORMAT: **SHOW DEVICE**

Command parameters: None

DESCRIPTION:

Details of which DTI file is currently associated with which display quadrant are output to the terminal. The current viewing direction as specified by the ROTATE command is also output.

Messages:

The following message is specific to the SHOW command
*** ERROR *** Specifying command SHOW
Command qualifiers are ANNOTATION, DEVICE, ENABLE, FILEIN,
IFF, PARAMETERS, POSITION, RANGES, RCP_FILE, SELECTIONS or SETTINGS

Examples:

Rover>**SHOW DEVICE<CR>**

```
Current quadrant setting:
DTI File 1:     Quadrant 2
DTI File 2:     Quadrant 4
Current viewing direction is from the East

Rover>
```

SHOW ENABLE

Shows the current status of those options that may be enabled by means of the ENABLE command, or disabled using the DISABLE command.

FORMAT: **SHOW ENABLE**

Command parameters: None.

DESCRIPTION:

Displays the current status of all the ROVER options that may be enabled or disabled using the ENABLE and DISABLE commands.

The name of the option is shown, followed by either the word ON or OFF to indicate its current status.

If the command SHOW ENABLE is used before any ENABLE or DISABLE commands have been given, the default status of the options is displayed.
If the command is given in REGISTRATION mode, then the current status of various registration options are also displayed.

Messages:

The following message is specific to the SHOW command
*** ERROR *** Specifying command SHOW
Command qualifiers are ANNOTATION, DEVICE, ENABLE, FILEIN,
IFF, PARAMETERS, POSITION, RANGES, RCP_FILE, SELECTIONS or SETTINGS

Examples:

ROVER>**SHOW ENABLE<CR>**

Current status:

ABSOLUTE	On	CROSS_CURSOR	Off	COLOUR_MAP	Off
DIGITISE	Off	DIVIDE	Off	EDIT	On
FLOODING	Off	HATCH	Off	HEIGHT	On
INTERPOLATION	On	LEGEND	On	MASK	Off
OVERRIDE	Off	PATTERN	On	PICTURE 1	On
PICTURE 2	Off	POSITIONING	Off	REGISTRATION	Off
SAMPLE	Off	TABLE	Off	TEXT	Off

```
Rover>ENABLE REGISTRATION<CR>
Registration>SHOW ENABLE<CR>

Current status:

ABSOLUTE          On        CROSS_CURSOR      Off       COLOUR_MAP        Off

DIGITISE          Off       DIVIDE            Off       EDIT              On

FLOODING          Off       HATCH             Off       HEIGHT            On

INTERPOLATION     On        LEGEND            On        MASK              Off

OVERRIDE          Off       PATTERN           On        PICTURE 1         On

PICTURE 2         Off       POSITIONING       Off       REGISTRATION      On

SAMPLE            Off       TABLE             Off       TEXT              Off

DEFAULT RCP_ID    On        DEFAULT RCP_TYPE  On        DEFAULT RCP_WXY   On

DEFAULT RCP_WZ    On        DEFAULT RCP_Z     On

Registration>
```

SHOW FILEIN

Outputs information about the currently selected DTI file.

FORMAT: **SHOW FILEIN**

Command parameters: None

DESCRIPTION:

Details of the currently selected DTI file are displayed on the terminal.

The header values are shown in the current units of measurement. This is dependent on the header type of the input file, or may be set explicitly using the UNITS command. By default or if the ENABLE ABSOLUTE command has been given then metre or projection values are expressed in absolute values. Otherwise they are displayed as offsets from the SW corner of the matrix.

If the ENABLE SAMPLE command has been given then the X and Y sample values for the currently selected DTI file are also output.

The DTI file number as required by the SELECT FILEIN command is also displayed.

Messages:

The following message is specific to the SHOW command.

```
*** ERROR *** Specifying command SHOW
Command qualifiers are ANNOTATION, DEVICE, ENABLE, FILEIN,
IFF, PARAMETERS, POSITION, RANGES, RCP_FILE, SELECTIONS or SETTINGS
```

Examples:

```
Rover>SHOW FILEIN<CR>
File    :  LSL$DTI:TEST.DTI
Header  :   LSLA Data:  WORD

Units are DTI Matrix Value

Matrix Coverage   SW:     1      1  NE:   301   251
Matrix Window     SW:     1      1  NE:   200   200
Matrix Interval    E:     1           N:     1
Value Range         :     0    to    851

DTI File  1

Rover>
```

SHOW IFF

Outputs information about the currently opened IFF file.

FORMAT: **SHOW IFF**

Command parameters: None

DESCRIPTION:

The name of the IFF file, and the IFF coordinate range and window settings are shown. By default or if the ENABLE ABSOLUTE command has been given, then the coordinates are expressed as absolute values. Otherwise they are displayed as offset values from the SW corner of the file area.

Messages:

The following message is specific to the SHOW command.

```
*** ERROR *** Specifying command SHOW
Command qualifiers are ANNOTATION, DEVICE, ENABLE, FILEIN,
IFF, PARAMETERS, POSITION, RANGES, RCP_FILE, SELECTIONS or SETTINGS
```

Examples:

Rover>**SHOW IFF<CR>**

```
IFF file:       LSL$IF:TEST.IFF
IFF file range:
SW: 270000.00 180000.00   NE: 310000.00  220000.00
IFF file window :
SW: 280000.00 190000.00   NE: 320000.00  190000.00

Rover>
```

SHOW PARAMETERS

Outputs information about the current range,step and colour indices.

FORMAT: SHOW PARAMETERS

Command parameters: None

DESCRIPTION:

Details of the current range and step settings,step upper limits and associated colour indices are output for the currently selected DTI file. The current overlay colour and in EDIT mode editing cursor colour are also shown.

Messages:

The following message is specific to the SHOW command.

```
*** ERROR *** Specifying command SHOW
Command qualifiers are ANNOTATION, DEVICE, ENABLE, FILEIN,
IFF, PARAMETERS, POSITION, RANGES, RCP_FILE, SELECTIONS or SETTINGS
```

Examples:

Rover>**SHOW PARAMETERS<CR>**

```
Range          : 200 to 300
Number of steps   : 10
Step upper limits : 210 220 230 240 250 260 270 280 290 300
Colour indices    :  3   6   9  12  15  18  21  24  27  30
Number of Colours : 31
First Colour      : 2
Planes Setting    : 5

Rover>
```

SHOW POSITION

Outputs information about the position of the raster editing cursor or vector digitising cursor.

Only available in DIGITISE or EDIT modes of operation.

FORMAT: **SHOW POSITION**

Command parameters: None

DESCRIPTION:

In EDIT mode SHOW POSITION outputs to the terminal the current editing cursor position. The values are shown in the current units of measurement. By default or if the ENABLE ABSOLUTE command has been given then metre or projection values are expressed in absolute values. Otherwise they are displayed as offsets from the SW corner of the matrix. The DTI value at this position is also shown.

In DIGITISE mode the command outputs the current position of the digitising cursor in IFF coordinates.

SHOW POSITION is only valid in DIGITISE or EDIT modes of operation.

Messages:

The following messages are specific to the SHOW and SHOW POSITION commands.

```
*** ERROR *** Specifying command SHOW
Command qualifiers are ANNOTATION, DEVICE, ENABLE, FILEIN,
IFF, PARAMETERS, POSITION, RANGES, RCP_FILE, SELECTIONS or SETTINGS
```

```
*** ERROR *** Specifying command SHOW POSITION
Command only valid in EDIT or DIGITISE modes
```

Examples:

Edit>**SHOW POSITION<CR>**

```
Editing Cursor Position:
X:    35       Y:    234
DTI Value: 345
```

Digitise>**SHOW POSITION<CR>**

```
Digitising Cursor Position
X:   3457.30   Y:   6789.88
```

SHOW RANGES

Outputs information about the value ranges for the currently selected DTI file.

FORMAT: **SHOW RANGES**

Command parameters: None

DESCRIPTION:

SHOW RANGES outputs information about value ranges for the currently selected DTI file to the terminal. The file range stored in the file header, the range of values in the current WINDOW, and the current range as specified with the RANGE command are shown. The WINDOW range may require to be calculated by this command. This may take an appreciable time for large windows.

Messages:

The following message is specific to the SHOW command.
*** ERROR *** Specifying command SHOW
Command qualifiers are ANNOTATION, DEVICE, ENABLE, FILEIN,
IFF, PARAMETERS, POSITION, RANGES, RCP_FILE, SELECTIONS or SETTINGS

Examples:

Edit>**SHOW RANGES<CR>**

```
File Range          : 0  to  1344
Window Range        : 567 to 789
Current Range       : 400 to 700

Rover>
```

SHOW RCP_FILE

Displays the contents of the currently selected Registration Control Point (RCP) file.

The command SHOW RCP_FILE is only available in registration mode.

FORMAT: **SHOW RCP_FILE**

Command parameters: None

DESCRIPTION:

The SHOW RCP_FILE command is used to display the contents of the currently selected Registration Control Point (RCP) file.

An RCP file may be made current using the SELECT RCP_FILE command. By default the file most recently opened or created using the OPEN RCP_FILE and CREATE RCP_FILES commands, is selected as the current RCP file.

Messages:

The following messages are specific to the SHOW and SHOW RCP_FILE commands:

```
*** ERROR *** Specifying command SHOW
Command qualifiers are ANNOTATION, DEVICE, ENABLE, FILEIN,
IFF, PARAMETERS, POSITION, RANGES, RCP_FILE SELECTIONS or SETTINGS

*** ERROR *** Specifying command SHOW RCP_FILE
Command only valid in REGISTRATION mode

*** ERROR *** Specifying command SHOW RCP_FILE
No RCP files are currently open

*** ERROR *** Specifying command SHOW RCP_FILE
Currently selected RCP file is not open
```

Examples:

Registration>**SHOW RCP_FILE<CR>**

Registration>

SHOW SELECTIONS

Outputs information about current selections made on the IFF file.

FORMAT: **SHOW SELECTIONS**

Command parameters: None

DESCRIPTION:

The currently active selections made with the SELECT and DESELECT commands are listed.

Messages:

The following message is specific to the SHOW command.

```
*** ERROR *** Specifying command SHOW
Command qualifiers are ANNOTATION, DEVICE, ENABLE, FILEIN,
IFF, PARAMETERS, POSITION, RANGES, RCP_FILE, SELECTIONS or SETTINGS
```

Examples:

```
Rover>SHOW SELECTIONS<CR>
All Layers Selected
All Feature Codes Selected
All FSNs selected
Currently selected colour 6

Currently Selected DTI file: 2
File: LSL$DTI:TEST.DTI
Rover>
```

SHOW SETTINGS

Shows the current value of those parameters that may be set using the SET command.

Command parameters: None

FORMAT: **SHOW SETTINGS**

DESCRIPTION:

Displays the name of all ROVER parameters that may be set using the SET command, and their current values.

If the SHOW SETTINGS command is given before using a SET command, the values shown are the default values that are allocated to the parameters by ROVER.
If the command is given in REGISTRATION mode, then the current RCP attribute values are also displayed.

Messages:

The following message is specific to the SHOW command.

*** ERROR *** Specifying command SHOW
Command qualifiers are ANNOTATION, DEVICE, ENABLE, FILEIN,
IFF, PARAMETERS, POSITION, RANGES, RCP_FILE, SELECTIONS or SETTINGS

Examples:

Rover>**SHOW SETTINGS<CR>**
Current Settings:

FC	1	LAYER	2	FSN	6
COLOUR	2	CURSOR_COLOUR	7	DELAY	100

Rover>**ENABLE REGISTRATION<CR>**
Registration>**SHOW SETTINGS<CR>**
Current Settings:

FC	1	LAYER	2	FSN	6
COLOUR	2	CURSOR_COLOUR	7	DELAY	100
RCP_TYPE	XX	RCP_WXY	1.0	RCP_WZ	0.0
RCP_Z	0.0				

Registration>

SMOOTH_EDIT

Starts a two line smoothing edit operation

Only available in EDIT mode of operation.

FORMAT: **SMOOTH_EDIT**

Command parameters: None

DESCRIPTION:

SMOOTH_EDIT command specifies that the next editing sequence of commands will specify two lines between which DTI values will be interpolated.

A sequence of EDIT and MOVE commands are used to define the first line. On issue of the END command the user is prompted for a DTI value. If no value is given (carriage return is pressed) the existing values of that line are assumed. The second line should then be specified in a similar way. On completion of the second line, values between the two lines are linearly interpolated. Appropriate messages guide the user through the sequence of commands required.

The SMOOTH_EDIT command is only available in EDIT mode of operation.

To indicate that smooth editing is taking place the program prompt becomes 'Smooth>' during the editing operation.

ABANDON or <CTRL/Z> (pressing the Ctrl and Z keys together) will abort the smooth editing operation operation.

Puck button D is defined to be SMOOTH_EDIT.

Messages:

The following messages are specific to the SMOOTH_EDIT command.

*** ERROR *** specifying command SMOOTH_EDIT
Command only available in EDIT mode

*** ERROR *** specifying command SMOOTH_EDIT
Selected DTI file is opened read-only

*** ERROR *** specifying command SMOOTH_EDIT
Command not available during edit operation

Examples:

Edit>**SMOOTH_EDIT<CR>**
Please define first line
Smooth>**MOVE 50 50<CR>**
Smooth>**EDIT<CR>**
Smooth>**MOVE 50 60<CR>**
Smooth>**EDIT<CR>**
Smooth>**END<CR>**
Smooth> DTI Value> **500<CR>**
Please define second line
Smooth>**MOVE 55 50<CR>**
Smooth>**EDIT<CR>**
Smooth>**MOVE 55 60<CR>**
Smooth>**EDIT<CR>**
Smooth>**END<CR>**
Smooth> DTI Value> **<CR>**
Edit>

In this example two lines are defined. The first is assigned a value of 500. The second line consists of existing DTI values because only <CR> was specified after the prompt.

SPAWN

Allows a DCL sub-command to be spawned within the ROVER program.

FORMAT: **SPAWN command**

Command parameters:

command

The DCL command which is to be spawned. A Valid DCL command is required.

DESCRIPTION:

SPAWN allows a DCL command to be obeyed from within the ROVER program. This is to allow file management operations such as DIRECTORY, DELETE and RENAME to be carried out during program execution.

Messages:

The following message is specific to the SPAWN command.

*** ERROR *** Specifying command SPAWN
SPAWN requires a valid DCL command

Examples:

Rover>**SPAWN DIRECTORY LSL$DTI<CR>**
Directory DUA0:[DTI]

AFRICA.DTI;2 LS41.DTI;1 WALES.DTI;3 TEST.DTI;13

Total of 3 files

Rover>**SPAWN RENAME AFRICA.DTI SOUTHAMERICA.DTI<CR>**
Rover>

SRI

Selects the symbol representation file

FORMAT: **SRI file-spec**

Command parameters:

file-spec

The SRI file specification. Any part of the file specification not supplied will be taken from the default 'LSL$FRT:.SRI'

DESCRIPTION:

The specified SRI file is selected.

The SRI determines the graphical representation of point symbols, and symbol strings.

If this command is omitted before the OVERLAY command then an SRI with the same name as the supplied FRT will be opened. If such a file does not exist then this command is required before OVERLAY is issued.

Messages:

The following message is specific to the SRI command.

*** ERROR *** Specifying command SRI
"System error message"

Examples:

Rover>**SRI OS.SRI<CR>**
Rover>

START

Starts the construction of an IFF feature using the program's digitising option.

Only available in DIGITISE mode of operation

FORMAT: START

Command parameters: None

DESCRIPTION:

The construction of an IFF feature is started using the program's digitising option. A crosshair cursor will appear on the screen, and the prompt 'Digitise Point>' will be output on the terminal, to show that a feature is being constructed.

This command is only available in DIGITISE mode of operation.

Puck button 4 is defined as START.

Messages:

The following message is specific to the START command.

*** ERROR *** Specifying command START
Command only available in DIGITISE mode

Examples:

Digitise>**START<CR>**
Digitise Point>

STEP

Defines a series of DTI value step intervals for the currently selected DTI file.

FORMAT: **STEP values[...]**

Command parameters:

values

One or more step interval values. Up to 7 integer values may be specified. At least one value is required.

DESCRIPTION:

This command divides the values in the currently selected DTI file into a series of steps which are displayed in different colours.

One or more DTI step intervals may be defined using this command for the currently selected DTI file. A single value denotes a constant step interval, while a variable step interval may be defined by entering up to 7 values.

A constant step interval, derived by dividing the minimum and maximum values in the DTI file by the number of display colours, is set up by default on opening a DTI file. The COLOURS and FIRST_COLOUR commands also reset the STEP in this way.

If a single step value of 1 is given, then each step is associated with a single colour on display. The legend reflects this situation as each value is placed alongside the centre of each box.

If step intervals are set such that there are too few colours to display all the steps, then the upper steps will be displayed in white.

The colour index that will be associated with each step may by examined by means of the command SHOW PARAMETERS.

In EDIT mode this command causes automatic redisplay of the currently selected DTI file.

Messages:

The following messages are specific to the STEP command.

*** ERROR *** Specifying command STEP
Command requires at least one integer argument

*** ERROR *** Specifying command STEP
Step values should greater than 1

*** WARNING *** range exceeded
step "stepnumber" is truncated to "DTI-value"

*** WARNING *** Too many steps for available colours
Upper steps shown in white

Examples:

Rover>**STEP 100<CR>**
Rover>

In this example a constant step interval of 100 has been specified.

Rover>**STEP 5 5 10 10 50 50 100<CR>**
Rover>

Here the step intervals are variable. The step intervals above 230 will be set to a constant value of 100.

TABLE RCP_FILE

Selects the RCP file that will receive RCP input using the table puck and digitising table.

The command TABLE RCP_FILE is only available in registration mode.

FORMAT: TABLE RCP_FILE number

Command parameters:

number

The number of the RCP file. An integer value in the range 1 to 4 is required.

DESCRIPTION:

The TABLE RCP_FILE command is used to select the Registration Control Point (RCP) file into which RCPs recorded using the table puck and digitising table will be entered.

By default RCPs entered from the table will be added to the current RCP file selected using the SELECT RCP_FILE command, however the TABLE RCP_FILE command may be used to allocate a particular RCP file to the table. This option, when utilised with the SETUP MAP command given in REGISTRATION mode, allows a user to enter map control points from the table, and image control points from the terminal or workstation mouse, without the need to redefine which RCP file is current.

Messages:

The following messages are specific to the TABLE RCP_FILE command

```
*** ERROR *** Specifying command TABLE
Command qualifier is RCP_FILE

*** ERROR *** Specifying command TABLE
Command only valid in REGISTRATION mode

*** ERROR *** Specifying command TABLE
Command requires an integer argument in the range 1 to 4
```

Examples:

Registration>**TABLE RCP_FILE 2<CR>**
Registration>

TOLERANCE

Sets the point density for curves.

FORMAT: **TOLERANCE [a b c]**

Command parameters:

a,b,c

The coefficients which control the spacing of the interpolated points. The defaults are a=250.0, b=0.0, c=0.0. Absence of trailing parameters will not affect the corresponding tolerances if these have been previously set. If no tolerance values are specified, or if the sum of the three tolerances is zero after the command has been issued, the default tolerances are set.

DESCRIPTION:

This command affects how interpolated lines are output with the OVERLAY command.

The three coefficients (a,b,c) control the spacing of interpolated points. The approximate separation of points (d) is given by:

d = a + 2*SQRT(2br) + cr (r is radius of curvature)

which means (if other coefficients were zero) that

a gives a constant separation of A IFF units
b gives a constant 'arc to chord' distance of b IFF units
c gives a constant angular deviation of c radians

Examples:

Rover>**TOLERANCE 250 100 1.5<CR>**
Rover>

TRI

Selects the text representation IFF file

FORMAT: **TRI file-spec**

Command parameters:

file-spec

The TRI file specification. Any part of the specification not supplied is taken from the default 'LSL$FRT:.TRI'.

DESCRIPTION:

The specified TRI file is selected.

The TRI file is used to determine the font characteristics of IFF text features drawn in Picture 2 using the OVERLAY command.

If this command is omitted before the OVERLAY command then a TRI with the same name as the supplied FRT will be opened. If such a file does not exist then this command is required before OVERLAY is issued.

Messages:

The following message is specific to the TRI command.

*** ERROR *** Specifying command TRI
"System error message"

Examples:

Rover>**TRI OS.TRI<CR>**
Rover>

UNITS

Specifies the units of measurement that will be used when defining an area of interest in the input DTI file.

FORMAT: UNITS units

Command parameters:

units

A keyword defining the measurement units, chosen from:

MATRIX	Matrix grid interval units, i.e rows and columns
METRES	Metre offsets from the south west corner of the DTI file
SECONDS	Latitude and Longitude in seconds of arc
LATLONG	Latitude and Longitude in degrees, minutes and seconds
PROJECTION	Projection Record Units (eg. mms on the source document)

DESCRIPTION:

The UNITS command defines the units of measurement that will be used when defining an area of interest in the currently selected DTI file by means of the WINDOW command, and for specifying the editing cursor position with the MOVE command
The command also controls in what format details from the header of the DTI file are displayed, when the SHOW FILEIN command is given.

The command should be given after defining the an DTI file since an appropriate default units of measurement is set up when the file is opened.

Messages:

The following messages are specific to the UNITS command.

*** ERROR *** Specifying command UNITS
Command should be followed by MATRIX, METRES, LATLONG, SECONDS or PROJECTION
Current setting is "units"

*** ERROR *** Specifying command UNITS
Command qualifier is invalid for the input file

Examples:

Rover> **UNITS MATRIX<CR>**
Rover>

UP

Moves the raster editing cursor up by one pixel

Only available in EDIT or REGISTRATION modes of operation.

FORMAT: **UP**

Command parameters: None

DESCRIPTION:

UP moves the raster editing cursor up by one DTI pixel.

The command is only available if ENABLE EDIT has been specified.

Puck button 9 is defined as UP.

Messages:

The following message is specific to the UP command.

*** ERROR *** Specifying command UP
Command only available in EDIT or REGISTRATION modes

Examples:

Edit>**UP<CR>**
Edit>

UPDATE_FILEIN

Opens a DTI file for write access

FORMAT: **UPDATE_FILEIN file-spec**

Command parameters:

file-spec

The file specification for the input DTI file. Any part of the file specification not supplied is taken from the default 'LSL$DTI:DTI.DTI'

DESCRIPTION:

This command opens and maps into memory a DTI file for write access. The file is opened on the lowest available DTI file number. The DTI file number is displayed on the terminal, and is the means by which the file is selected when a number of DTI files are opened.

Up to 4 DTI files may be opened at one time. By default the DTI file opened with the last FILEIN or UPDATE_FILEIN command is the current DTI file. If the logical name LSL$MATRIX_DEFAULT_WINDOW is defined with a value of "1", then a default area of interest is set for the file based on the maximum window which may be displayed in the default display area on the graphics device at four times magnification. Otherwise if ENABLE SAMPLE has not been specified a window corresponding to the entire file extents is set unless this is too large to be displayed in which case truncation takes place. The area of interest may be altered at any time by use of the WINDOW command.

If the rotation value (DTI_ORDER_CORNER) held in the header of the DTI file is not the default, (south west) then a warning message is output when the file is opened. Rover however, sets up the viewing direction taking into account the rotation value. Thus, provided the rotation value in the header is correct, Rover will display the file in the correct orientation. The ROTATE command be used to change the default viewing direction.

Information on the mapped DTI file may be obtained at any time by typing 'SHOW FILEIN'

This command is required if editing operations are to be carried out on the file. (cf FILEIN)

In EDIT mode issue of the UPDATE_FILEIN command results in automatic display of the file.

Messages:

The following messages are specific to the UPDATE_FILEIN command.

*** ERROR *** Specifying command UPDATE_FILEIN

```
Four DTI are currently opened
CLOSE command should be used

*** ERROR *** Specifying command UPDATE_FILEIN
Filename is missing
```

Example:

```
Rover>UPDATE_FILEIN TEST<CR>
*** File LSL$DTI:TEST.DTI opened as DTI file 1

File    :  LSL$DTI:TEST.DTI
Header  :   LSLA Data:  WORD

Units are DTI Matrix Value

Matrix Coverage   SW:     1      1  NE:   301   251
Matrix Interval    E:     1          N:     1
Value Range         :     0    to    851

Rover>
```

WAIT

Suspends the program for a specified number of seconds.

FORMAT: WAIT interval

Command parameters:

interval

The time interval in seconds. A real value is required.

DESCRIPTION:

This command suspends program execution for the specified time interval. It is useful for demonstration purposes if ROVER is being run from a command file.

Messages:

The following messages are specific to the WAIT command.

```
*** ERROR *** Specifying command WAIT
Command requires a real in the range 1 to 1000
```

Examples:

Rover>**WAIT 5.0<CR>**
Rover>

WINDOW

Specifies an area of interest in the currently selected DTI file.

FORMAT: **WINDOW xmin ymin xmax ymax**

Command parameters:

xmin ymin
: The coordinates of the bottom left hand corner of the defining rectangle.

xmax ymax
: The coordinates of top right hand corner of the defining rectangle.

The format of the command parameters is controlled by the UNITS command.

UNITS MATRIX - Requires 4 integer values defining the rectangle in terms of column and row numbers

UNITS METRES - Requires 4 real (floating point) values defining the rectangle as metre offsets from the SW corner of the DTI file. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates (eg. U.K. National Grid coordinates); if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

UNITS SECONDS - Requires 4 real (floating point) values defining the absolute position of the rectangle in seconds of arc. The values are supplied in the order latitude followed by longitude for the SW corner, and latitude followed by longitude for the NE corner. A negative latitude value indicates a position in the Southern Hemisphere, and a negative longitude value a point to the west of Greenwich.

UNITS LATLONG - Requires 4 values defining the absolute latitude and longitude position of the rectangle in the format DDD MM SS H where DDD is the number of degrees, MM is the number of minutes, SS is the number of seconds, and H is the hemisphere (N,S,E,W). The values are supplied in the order latitude followed by longitude for the SW corner, and latitude followed by longitude for the NE corner.

UNITS PROJECTION Requires 4 real (floating point) values defining the rectangle in projection record units. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates; if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

Note that in all cases, the input values are adjusted to the nearest column and row values.

DESCRIPTION:

The WINDOW command is used to limit display to a particular rectangular geographical area. Only nodes in the DTI file that lie within this area are drawn when the DISPLAY command is given.
The area of interest should lie within the geographical bounds of the DTI file.

If the logical name LSL$MATRIX_DEFAULT_WINDOW is defined with a value of "1", then a default area of interest is set for the file based on the maximum window which may be displayed in the default display area on the graphics device at four times magnification. Otherwise a window corresponding to the entire file extents is set unless this is too large to be displayed or ENABLE SAMPLE has not been specified in which case truncation takes place.

If a digitising table is available, and a map has been registered to the DTI file, the coordinates of the window may be defined using any button on the table puck.

If an input IFF file is open then the WINDOW command also causes the IFFWINDOW to be reset in order to maintain matrix/vector registration.

In EDIT mode issue of this command causes immediate redisplay of the new window.

Messages:

The following messages are specific to the WINDOW command:

*** ERROR *** Specifying command WINDOW
Command not available during editing operation

*** ERROR *** Specifying command WINDOW
Command requires 4 coordinate values

*** ERROR *** Specifying command WINDOW
DTI file "dtinum" has not yet been opened

*** ERROR *** Specifying command WINDOW
NE coordinates should exceed SW coordinates

*** ERROR *** Specifying command WINDOW
Unable to use supplied LAT LONG values

*** WARNING *** Specified window exceeds matrix limits
Window will be truncated to fit

Examples:

ROVER>**WINDOW 1 1 200 200<CR>**
ROVER>**UNITS LATLONG<CR>**
ROVER>**WINDOW 42 00 00N 3 00 00E 42 30 00N 2 58 40E**
ROVER>

ZOOM

Resets the current DTI and IFF windows depending on the zoom factor and redisplays.

FORMAT: **ZOOM [factor]**

Command parameters: None

factor

The zoom factor. A positive real number may be specified. If no parameter is given then the default zoom factor is 2.

DESCRIPTION:

ZOOM resets the current raster and IFF windows depending on the zoom factor. The default zoom factor is two. This halves the size of the window, and therefore increases the scale of the displayed image. A scale factor of between 0 and 1 increases the size of the window. In EDIT mode the new window is centred around the current DTI cursor coordinates.

If many ZOOM commands will be used during a Rover session then it is recommended that the colour mapping option is invoked with the ENABLE COLOUR_MAP command. This improves the efficiency of raster display.

Messages:

The following messages are specific to the ZOOM command:

*** ERROR *** Specifying command ZOOM
Command takes 1 positive real argument

*** ERROR *** Specifying command ZOOM
Command not available during editing operation

Examples:

Rover>**ZOOM 4<CR>**
Rover>

MESSAGES (WARNING) None

MESSAGES (ERROR) None

MESSAGES (FATAL) None

MESSAGES (OTHER)

In addition to messages which are generated by the program itself, other messages may be produced by Laser-Scan libraries. In particular, messages may be generated the DTI library, and by the Laser-Scan I/O library, LSLLIB.

DTI library messages are introduced by '%DTI', and are documented in the DTILIB Reference Manual. In all cases the messages indicate a fatal error, that will cause processing to halt.

LSLLIB messages are introduced by '%LSLLIB' and are generally self-explanatory. Such messages rarely indicate a fatal error, and are generated most frequently by entering a command in an invalid format in response to the ROVER prompt.

APPENDIX A

```
; system constants file for ROVER
; 5 planes picture 1  3 planes picture 2
; COLOUR DEFINITIONS
;       Red     Green   Blue    Blink   Comment
256                                     ; number of colours
         0       0       0       0      ; 1 for background
        FF      FF      FF       0      ; 2 white
        80      80      FF       0      ; 3 blue
        50      80      20       0      ; 4
        50      90      30       0      ; 5
        70      90      30       0      ; 6
        78      A0      20       0      ; 7
        71      B1       0       0      ; 8
        71      C1       0       0      ; 9
        61      E1      10       0      ; 10
        10      EE      10       0      ; 11
        41      EE      10       0      ; 12
        61      EE      10       0      ; 13
        81      EE      10       0      ; 14
        91      EE      10       0      ; 15
        A0      EE      10       0      ; 16
        B0      EE      10       0      ; 17
        CA      EE      10       0      ; 18
        DB      EE      10       0      ; 19
        EE      EE      10       0      ; 20
        EE      E0      10       0      ; 21
        EE      D0      10       0      ; 22
        EE      BA      10       0      ; 23
        EE      A9      10       0      ; 24
        EE      B0      10       0      ; 25
        EE      A0      10       0      ; 26
        EE      98      10       0      ; 27
        EE      8A      10       0      ; 28
        EE      7E      10       0      ; 29
        EE      77      10       0      ; 30
        EE      6E      10       0      ; 31
        EE      61      10       0      ; 32
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
```

```
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF      FF      FF       0      ; white
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
        FF       0       0       0      ; red
         0      FF       0       0      ; green
         0      FF       0       0      ; green
         0      FF       0       0      ; green
         0      FF       0       0      ; green
         0      FF       0       0      ; green
         0      FF       0       0      ; green
         0      FF       0       0      ; green
```

```
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0      FF       0       0      ; green
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
        0       0      FF       0      ; blue
```

```
         0       0      FF       0      ; blue
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        80      80       0       0      ; brown
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
```

```
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      80       0       0      ; orange
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
        FF      FF       0       0      ; yellow
;
; BLINK DEFINITIONS
;       Blink   On      Off     Comment
        1       6       6       ; generator 1
        2       6       6       ; generator 2
        3       6       6       ; generator 3
        4       6       6       ; generator 4
```

CHAPTER 9

MODULE VECTORISE

MODULE VECTORISE

FUNCTION

VECTORISE is a program to extract the boundaries of areas in a raster image, and store the boundary lines in a junction structured vector IFF file. The raster or grid data is supplied to the program in a Laser Scan Digital Terrain Image file.

FORMAT

$ VECTORISE

PROMPTS

VECTORISE is an interactive, command driven program. Command input is expected when the following prompt is issued:

VECTORISE>

Commands are issued at the terminal in response to the prompt.

DESCRIPTION

VECTORISE is designed to extract the boundaries of areas in a raster image, and store the boundary lines in a junction structured IFF file.

Vectorisation may be carried out on any grid data held in a Laser Scan Digital Terrain Image (DTI) file. The process may be applied to a wide variety of data ranging from elevation data held as a DTM (Digital Terrain Model), to remotely sensed radiometric data. Typically however, the input raster image will be a classified image such as a slope or visibility map, or classified remotely sensed satellite data.

It should be appreciated that VECTORISE is designed only to perform boundary extraction, and is not designed to extract linear features from an image. It is therefore not strictly appropriate to the vectorisation of scanned map data which is likely to contain a high proportion of linework, text and point symbols. If 'linear' features exist in the input raster image, VECTORISE will produce an 'areal' feature formed from the left and righthand edges of the original feature.

The areas formed from the raster image may be defined by a single value (eg. all pixels with a value of 1), or by a range of values (eg. the area may be composed of pixels with values in the range 1 to 5). This is controlled using the RANGES command. A single range or series of ranges may be specified.

VECTORISE outputs the area edges to a junction structured IFF file. An area may be composed of one or more links, with each link being stored as a separate feature with distinct lefthand and righthand codes. A node is recorded in the file where three or more links meet, or at the start and end of a single link (loop) feature. Any redundant nodes that may be created by the vectorisation process, are removed before output to the IFF file.

If the user requires an IFF file in which the complete boundary of an area is held as a single feature (ie. as a closed loop), the file produced by VECTORISE may be processed through the POLYGONS module IPOLYGON. IPOLYGON will create an IFF file of polygon features formed from the lefthand and righthand attribute coded links, and will in addition identify those polygons nested within other polygons to enable their correct display.

By default the edges of areas as they are represented in the raster image are extracted and stored in the IFF file. The resulting links therefore reflect the original grid nature of the data. To correct the jagged or 'staircase' appearance of the vector data, a line smoothing algorithm which is especially applicable to these data is provided within VECTORISE. Smoothing may be selected by giving the ENABLE SMOOTH command prior to vectorisation. Smoothing is performed on each link in turn as part of the IFF output operation. Note that similar smoothing can be performed using the BEZIER option within the IMP program IFILTER.

FEATURE CODING AND AC LABELS

VECTORISE makes a distinction between links that form part of the image border, as defined by the current area of interest, and links that form part of the boundary of an area which is internal to the image. Border links and interior links are given by default different feature codes in the junction structured file. This enables the subsequent easy removal of border links from the file should this be required for a particular application.

Border links are given a feature code of 1 by default. The default may be changed using the command SET BORDER_FC.

Interior links are given a feature code of 2 by default. The default may be changed using the command SET INTERIOR_FC.

Each link is given a lefthand and righthand attribute label. The lefthand label is held in the text field of an AC 4 entry, and the righthand label is held in the text field of an AC 5 entry.

The AC 4 and AC 5 labels assigned to a link are dependent on the area value to the left and right of the line. For example, if the area to the right falls within the second range as defined using the RANGE command, it will be given a righthand label of "2".

In these cases the label will be numeric, however two special non-numeric labels are also used by VECTORISE. A label of "Surrounding void" is associated with all border links and refers to the area external to the image. A label of "Outside range" is associated with all areas in the image that are formed using a null data value. Null areas may be present in an image as a result of an earlier classification operation.

REPRESENTATION OF THE DATA IN THE IFF FILE

VECTORISE outputs the boundary of an area as one or more links to an IFF junction structured file (extension IFJ). A new link is created whenever the lefthand or righthand code of a line changes. A node is created where links meet. For details of how the nodes and links are held within the IFF file, the user is referred to the IFF User Guide.

VECTORISE COMMANDS

COMMAND LIST

The following commands are defined :

```
@               !               DISABLE ABSOLUTE
DISABLE DIAGNOSTICS             DISABLE LOG_FILE
DISABLE SMOOTH  ENABLE ABSOLUTE ENABLE DIAGNOSTICS
ENABLE LOG_FILE ENABLE SMOOTH   EXIT            FILEIN
GO              HELP            IFF             RANGES
RETURN          SET BORDER_FC   SET INTERIOR_FC SET LAYER
SHOW ENABLE     SHOW FILEIN     SHOW RANGES     SHOW SETTINGS
SPAWN           UNITS           WAIT            WINDOW
```

@

Take command input from the specified file.

FORMAT: @ file-spec

Command parameters:

file-spec

The file to be opened and used for command input.

Any parts of the file-spec not supplied will be taken from the default specification 'SYS$DISK:[].COM;0'.

DESCRIPTION:

VECTORISE offers the facility of command input from an indirect command file. The '@' character preceding a file-spec will cause VECTORISE to open and read commands from the specified file until:

1. a RETURN command is detected and command input is returned to SYS$COMMAND.

2. the end of file is detected. This provokes an error message and command input is returned to SYS$COMMAND.

Nested command files are not supported (i.e. a command file containing an '@' command), although sequential '@' commands are supported when read from SYS$COMMAND.

As an aid to batch log interpretation VECTORISE will echo all commands read from an indirect command file.

Messages:

The following messages are specific to the @ command:

*** ERROR *** Specifying command @
Command file specification is missing

*** ERROR *** Specifying command @
Unable to open indirect command file 'file-spec'

*** ERROR *** Specifying command @
Nested command files not supported

Examples:

```
VECTORISE> @DEFAULTS<CR>
VECTORISE> ENABLE LOG_FILE
VECTORISE> ENABLE SMOOTH
VECTORISE> SET BORDER_FC 10
VECTORISE> RETURN
Returning to terminal input
VECTORISE>
```

!

Treat all text to the right of the '!' as a comment.

FORMAT: ! [comment text]

Command parameters:

comment text

text that is to be treated as a comment and which will be excluded from command interpretation.

DESCRIPTION:

An exclamation mark is the standard TVES package comment delimiter. All text (and numbers) which lie to the right of a '!' character are excluded from command interpretation. Comments are useful for annotating command procedures used in batch processing etc.

Messages: None.

Examples:

VECTORISE> **FILEIN TEST !open the file<CR>**
VECTORISE> **!Define a window<CR>**
VECTORISE> **WINDOW 1 1 10 10<CR>**
VECTORISE>

DISABLE ABSOLUTE

Cancels a previous ENABLE ABSOLUTE command.

FORMAT: DISABLE ABSOLUTE

Command parameters: None.

DESCRIPTION:

DISABLE ABSOLUTE cancels a previous ENABLE ABSOLUTE command. If DISABLE ABSOLUTE is given, then coordinate values required by the WINDOW command, supplied in metre or projection units, must be specified as an offset from the SW corner of the matrix.
By default window values should be specified as absolute coordinates.

Messages:

The following error message is specific to the DISABLE command:

*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, DIAGNOSTICS, LOG_FILE or SMOOTH

Examples:

VECTORISE> **DISABLE ABSOLUTE<CR>**
VECTORISE>

DISABLE DIAGNOSTICS

Cancels a previous ENABLE DIAGNOSTICS command.

FORMAT: DISABLE DIAGNOSTICS

Command parameters: None.

DESCRIPTION:

DISABLE DIAGNOSTICS allows the user to cancel a previous ENABLE DIAGNOSTICS command.
By default diagnostic printout, which is sent to SYS$OUTPUT, is enabled.

Messages:

The following error message is specific to the DISABLE command:

*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, DIAGNOSTICS, LOG_FILE or SMOOTH

Examples:

VECTORISE> **DISABLE DIAGNOSTICS<CR>**
VECTORISE>

DISABLE LOG_FILE

Disables the output of file and vectorisation process statistics to a log file.

FORMAT: **DISABLE LOG_FILE**

Command parameters: None.

DESCRIPTION:

Disables the output of file and vectorisation process statistics to a log file. By default no log file is created.

Messages:

The following error message is specific to the DISABLE command:

*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, DIAGNOSTICS, LOG_FILE or SMOOTH

Examples:

VECTORISE>**DISABLE LOG_FILE<CR>**
VECTORISE>

DISABLE SMOOTH

Disables the option to smooth all interior links prior to output to the IFF file.

FORMAT: **DISABLE SMOOTH**

Command parameters: None.

DESCRIPTION:

Disables the option to smooth all interior links prior to output to the IFF file.
The vectorisation process involves extracting the boundaries of areas as they are represented in the input DTI file. The resulting links therefore reflect the grid nature of the input data, and if unsmoothed, the links have a jagged and characteristic 'staircase' appearance. For many mapping applications a smoothed or interpolated version of the links is often required. This may be achieved by enabling smoothing using the ENABLE SMOOTH command. For applications where an exact correspondence between the vector and raster data is required, or where the 'staircase' appearance of the data is of no concern, smoothing may be disabled using the DISABLE SMOOTH command.
By default no smoothing is applied to the links.

Messages:

The following error message is specific to the DISABLE command:

```
*** ERROR *** Specifying command DISABLE
Command qualifiers are ABSOLUTE, DIAGNOSTICS, LOG_FILE or SMOOTH
```

Examples:

```
VECTORISE>DISABLE SMOOTH<CR>
VECTORISE>
```

ENABLE ABSOLUTE

Selects the use of absolute coordinates values.

FORMAT: ENABLE ABSOLUTE

Command parameters: None.

DESCRIPTION:

If ENABLE ABSOLUTE is given, then coordinate values required by the WINDOW command, supplied in metre or projection units, must be specified as absolute (rather than relative) coordinate values.
For example if the projection indicates U.K. National Grid, then the WINDOW values may be specified as 6 figure National Grid coordinates.

By default window values should be specified as absolute coordinates. This option can be disabled using the DISABLE ABSOLUTE command.

Messages:

The following error message is specific to the ENABLE command:

```
*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, DIAGNOSTICS, LOG_FILE or SMOOTH
```

Examples:

VECTORISE> **ENABLE ABSOLUTE<CR>**
VECTORISE>

ENABLE DIAGNOSTICS

ENABLE DIAGNOSTICS allows the user to enable diagnostic printout.

FORMAT: ENABLE DIAGNOSTICS

Command parameters: None.

DESCRIPTION:

ENABLE DIAGNOSTICS allows the user to enable diagnostic printout.
The diagnostic printout consists of a series of messages indicating which operation is currently being preformed by VECTORISE, and the percentage progress.
The messages are sent to SYS$OUTPUT. Note that if you are using a hardcopy terminal no percentage progress figures are generated.
By default diagnostic printout is selected, and may be disabled using the command DISABLE DIAGNOSTICS

Messages:

The following error message is specific to the DISABLE command:

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, DIAGNOSTICS, LOG_FILE or SMOOTH

Examples:

VECTORISE> **ENABLE DIAGNOSTICS<CR>**
VECTORISE>

ENABLE LOG_FILE

Enables the output of file and vectorisation process statistics to a log file. The contents of the log file may be examined using a screen editor or sent to a line-printer.

FORMAT: **ENABLE LOG_FILE**

Command parameters: None.

DESCRIPTION:

Enables the output of file and vectorisation process statistics to a log file. The contents of the log file may be examined using a screen editor or sent to a line-printer.
The log file is given the extension LOG, and a name partly constructed from the output IFF filename, thus 'LSL$IF:IFF_filename.LOG'
An example log file is included in the documentation in the section VECTORISE LOG FILE.
By default no log file is created.

Messages:

The following error message is specific to the ENABLE command:

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, DIAGNOSTICS, LOG_FILE or SMOOTH

Examples:

VECTORISE>**ENABLE LOG_FILE<CR>**
VECTORISE>

ENABLE SMOOTH

Enables the option to smooth all interior links prior to output to the IFF file.

FORMAT: **ENABLE SMOOTH**

Command parameters: None.

DESCRIPTION:

Enables the option to smooth all interior links prior to output to the IFF file. The vectorisation process involves extracting the boundaries of areas as they are represented in the input DTI file. The resulting links therefore reflect the grid nature of the input data, and if unsmoothed the links have a jagged and characteristic 'staircase' appearance. For applications where an exact correspondence between the vector and raster data is required, the links should be held in this form, however for mapping applications a smoothed or interpolated version of the links is often required.
If smoothing is enabled, a Bezier interpolation algorithm using parameters that have been carefully chosen as most applicable for vectorised data, is applied to the links before output to the IFF file. Interpolation is performed on each link in turn. No interpolation is applied to links forming part of the image border, or to links that are defined by less than 4 coordinate points.
By default no smoothing is applied to the links.

Messages:

The following error message is specific to the ENABLE command:

*** ERROR *** Specifying command ENABLE
Command qualifiers are ABSOLUTE, DIAGNOSTICS, LOG_FILE or SMOOTH

Examples:

VECTORISE>**ENABLE SMOOTH<CR>**
VECTORISE>

EXIT

Terminates a run of VECTORISE

FORMAT: **EXIT**

Command parameters: None.

DESCRIPTION:

The EXIT command is used to exit from VECTORISE.
<CTRL/Z> (pressing the Ctrl and Z keys together) may also be used to exit from the program.

Messages: None.

Examples:

VECTORISE>**EXIT<CR>**
$

FILEIN

Selects and opens the DTI file to be vectorised.

FORMAT: FILEIN file-spec

Command parameters:

file-spec

The file specification for the input DTI file. Any part of the file specification not supplied is taken from the default 'LSL$DTI:DTI.DTI', although if no file-spec is supplied, you will be asked to supply one in response to the prompt **Input DTI filename>**

DESCRIPTION:

This command opens and maps into memory a DTI file, containing the raster data to be processed. Details derived from the header of the file are displayed on the terminal to confirm that the file has been successfully opened.

If the logical name LSL$MATRIX_DEFAULT_WINDOW is defined with a value of "1", then a default area of interest defined in matrix units of bottom left hand corner 1,1 and top right hand corner 200,200 is set when the file is opened. If the logical name is absent or has any other value, or if the input DTI file has fewer than 200 columns or 200 rows, then a default window equivalent to the total matrix is set. The default window may be redefined using the WINDOW command.

Messages:

The following error message is specific to the FILEIN command:

```
*** ERROR *** Specifying command FILEIN
Input DTI filename is missing
```

Example:

```
VECTORISE>FILEIN TEST<CR>

LSL$DTI:TEST.DTI
Header: LSLA     Data: WORD

Units are DTI Matrix Values

Matrix Coverage    SW:    1     1    NE:   301   251
Matrix Window      SW:    1     1    NE:   301   251
Matrix Interval     E:    1           N:     1
Value Range          :      0   to    851
```

VECTORISE>

GO

Initiates vectorisation.

FORMAT: **GO**

Command parameters: None.

DESCRIPTION:

The GO command is used to initiate the vectorising process.

The IFF, FILEIN and RANGES commands must have been previously given to define the output IFF file, the the image to be processed, and to control how vectorisation should proceed.

The VECTORISE > prompt is displayed when vectorisation has been completed. The output IFF file is closed, however the input DTI file remains open until the EXIT command is given, or a new input file is selected using the FILEIN command. The current window and range values are preserved unless a new input DTI file is selected; the status of options set using ENABLE or DISABLE commands, or the values of various variables defined using the SET command remain unchanged.

Messages:

The following error messages are specific to the GO command

```
*** ERROR *** Specifying command GO
The output IFF file must be specified before GO

*** ERROR *** Specifying command GO
The input DTI file is undefined

*** ERROR *** Specifying command GO
No ranges have been defined
```

Examples:

```
VECTORISE>GO<CR>
VECTORISE>
```

HELP

Invokes on-line help

FORMAT: **HELP [command]**

Command parameters:

command

the command on which help is required

DESCRIPTION:

A brief description is given of the function and format of the specified command. If no parameter is specified then a list of all commands available is given.

Messages: None.

Examples:

VECTORISE>**HELP ENABLE LOGFILE**

ENABLE LOG_FILE

Enables the output of file and vectorisation process statistics
to a log file.
The contents of the log file may be examined using a screen
editor or sent to a line-printer.

VECTORISE>

IFF

Specifies the name of the output IFF file.

FORMAT: **IFF file-spec**

Command parameters:

file-spec

The file specification for the output IFF file. Any part of the file specification not supplied is taken from a default partly constructed from the main part of the input DTI filename 'LSL$IF:dti_filename.IFJ'

DESCRIPTION:

IFF specifies the name of the output IFF file. This is the file which will receive the vectorised data.

An IFF file must be specified before the command GO is issued.

If the input DTM contains a DTI Projection Record, details from the record are transferred to the Map Descriptor (MD) entry of the IFF file.

Messages:

The following message is specific to the IFF command:

*** ERROR *** Specifying command IFF
Output IFF filename is missing

Examples:

VECTORISE>**IFF TEST<CR>**
VECTORISE>

RANGES

Specifies the data ranges which are used to classify the DTI file data values into a series of areas.

FORMAT: **RANGES lower_value [:upper_value] [,...]**

Command parameters:

lower_value,upper_value

One or more upper and lower integer or real values.
If only one value is specified then the entire DTI value range is divided into a series of ranges with an interval of lower_value.
A colon is used to separate a lower and upper value, and a comma or space to separate a series of ranges.
For a DTI file containing REAL data values, the RANGE values will be interpreted as real numbers. Owing to the intrinsic differences between real and integer numbers, the ranges calculated will be different, in particular for the range calculation with only a lower_value given.

DESCRIPTION:

This command defines the ranges within the DTI file which are used to partition the DTI file into separate areas for vectorisation.
The DTI value range may be divided into a series of equal value ranges by supplying one parameter. Alternatively the image may be segmented into a series of irregular value ranges by specifying a series of lower and upper values.
RANGE automatically invokes a SHOW RANGES command, to show how the DTI image has been divided into range values for vectorisation.

Messages:

The following error messages are specific to the RANGES command:

*** ERROR *** Specifying command RANGES
The input DTI file must be specified before RANGES

*** ERROR *** Specifying command RANGES
Command requires at least 1 integer argument

*** ERROR *** Specifying command RANGES
Values are not in required format

*** ERROR *** Specifying command RANGES
Invalid range <lower_value> to <upper_value>

*** ERROR *** Specifying command RANGES
Range is outside DTI data range

```
*** ERROR *** Specifying command RANGES
<lower_value> is an invalid interval value

*** ERROR *** Specifying command RANGES
Maximum number of ranges will be exceeded if an interval value of <lower_value>
is used
```

Examples:
In the following examples the input DTI file has a data value range of 0 to 100.

VECTORISE>**RANGE 1:10,11:20,21:100<CR>**

```
No. of Ranges    :     4
Range 1          :     0
Range 2          :     1     :    10
Range 3          :    11     :    20
Range 4          :    21     :   100
```

VECTORISE>**RANGE 1:70,10<CR>**

```
No. of Ranges    :     5
Range 1          :     0
Range 2          :     1     :    70
Range 3          :    71     :    80
Range 4          :    81     :    90
Range 5          :    91     :   100
```

VECTORISE>**RANGE 20<CR>**

```
No. of Ranges    :     6
Range 1          :     0
Range 2          :     1     :    20
Range 3          :    21     :    40
Range 4          :    41     :    60
Range 5          :    61     :    80
Range 6          :    81     :   100
```

VECTORISE>

RETURN

Restores command input from an indirect command file to SYS$COMMAND.

FORMAT: RETURN

Command parameters: None.

DESCRIPTION:

Restores command input from an indirect command file to SYS$COMMAND.

A typical application is to use an indirect command file to set up a number of run time defaults for a flowline, and then return to input from the terminal for the run specific commands. To do this RETURN must be the last command in the indirect command file.

A RETURN command is only valid if present in a command file.

Messages:

The following messages are specific to the RETURN command:

*** ERROR *** Specifying command RETURN
The RETURN command is only valid in a command file

Returning to terminal input

Examples:

```
VECTORISE> @DEFAULTS<CR>
VECTORISE> ENABLE LOG_FILE
VECTORISE> ENABLE SMOOTH
VECTORISE> SET BORDER_FC 10
VECTORISE> RETURN
Returning to terminal input
VECTORISE>
```

SET BORDER_FC

Defines the feature code that is attached to all links that form part of the image border.

FORMAT: **SET BORDER_FC feature_code**

Command parameters:

feature_code

The feature code value to be used to identify border links in the IFF file.
The feature code is an integer value in the range 0 to 32767.

DESCRIPTION:

Defines the feature code that is attached to all links that form part of the image border. The image border is defined by the means of the WINDOW command. If the SET BORDER_FC command is not supplied, links that form part of the border will be assigned a value of 1 by default.

Messages:

The following error messages are specific to the SET and SET BORDER_FC commands:

*** ERROR *** Specifying command SET
Command qualifiers are BORDER_FC, INTERIOR_FC or LAYER

*** ERROR *** Specifying command SET BORDER_FC
Command requires an integer argument

*** ERROR *** Specifying command SET BORDER_FC
Command requires a feature code value in the range 0 to 32767

Examples:

VECTORISE>**SET BORDER_FC 10<CR>**
VECTORISE>

SET INTERIOR_FC

Defines the feature code that is attached to all links that form part of an interior area boundary.

FORMAT: **SET INTERIOR_FC feature_code**

Command parameters:

feature_code

The feature code value to be used to identify interior links in the IFF file.
The feature code is an integer value in the range 0 to 32767.

DESCRIPTION:

Defines the feature code that is attached to all links that form part of the boundary of an interior area.
If the SET INTERIOR_FC command is not supplied, interior links will be assigned a value of 2 by default.

Messages:

The following error messages are specific to the SET and SET INTERIOR_FC commands:

*** ERROR *** Specifying command SET
Command qualifiers are BORDER_FC, INTERIOR_FC or LAYER

*** ERROR *** Specifying command SET INTERIOR_FC
Command requires an integer argument

*** ERROR *** Specifying command SET INTERIOR_FC
Command requires a feature code value in the range 0 to 32767

Examples:

VECTORISE>**SET INTERIOR_FC 20<CR>**
VECTORISE>

SET LAYER

Defines the layer number that is created in the output IFF file.

FORMAT: SET LAYER layer_code

Command parameters:

layer_code

The number of the layer.
The layer number is an integer value in the range 1 to 32767.

DESCRIPTION:

Defines the number of the layer that is created to contain the vectorised data in the output IFF file.
If the command SET LAYER is not supplied, the vectorised data is written to layer 1 in the output IFF file.

Messages:

The following error messages are specific to the SET and SET LAYER commands:

```
*** ERROR *** Specifying command SET
Command qualifiers are BORDER_FC, INTERIOR_FC or LAYER

*** ERROR *** Specifying command SET LAYER
Command requires an integer argument

*** ERROR *** Specifying command SET LAYER
Command requires a layer number in the range 1 to 32767
```

Examples:

VECTORISE>**SET LAYER 10<CR>**
VECTORISE>

SET SCALE

Sets the scale in the output IFF file MD (Map Descriptor) entry.

FORMAT: SET SCALE scale

Command parameters:

scale

The scale to be set in the output IFF file MD (Map Descriptor) entry. By default a scale value of 50000, for 1:50000 is used.

DESCRIPTION:

The scale field in the MD is by default set to 50000, as this information is unavailable in the DTI file header. The user may override this default scale by use of the SET SCALE command. The argument to the SET SCALE command should be specified as an real value to represent the denominator of the scale fraction, thus a desired scale of 1:250000 must be specified as 250000.

Messages:

*** ERROR *** Specifying command scale.
Command requires a positive real number

Examples:

VECTOTISE>**SET SCALE 100000<CR>**
VECTORISE>

SHOW ENABLE

Shows the current status of those options that are enabled by means of the ENABLE command, or disabled using the DISABLE command.

FORMAT: **SHOW ENABLE**

Command parameters: None.

DESCRIPTION:

Displays the current status of all the VECTORISE options that may be enabled or disabled using the ENABLE and DISABLE commands.
The name of the option is shown followed by either the word ON or OFF to indicate its current status.
If the command SHOW ENABLE is used before any ENABLE or DISABLE commands have been given, the default status for the options is displayed.

Messages:

The following message is specific to the command SHOW:

*** ERROR *** Specifying command SHOW
Command qualifiers are ENABLE, FILEIN, RANGES or SETTINGS

Examples:

VECTORISE>**SHOW ENABLE<CR>**

```
Current status:

ABSOLUTE        On      DIAGNOSTICS     On        LOG_FILE       Off

SMOOTH         Off

VECTORISE>
```

SHOW FILEIN

Shows information on the currently selected DTI file.

FORMAT: **SHOW FILEIN**

Command parameters: None.

DESCRIPTION:

Details extracted from the header of the currently opened DTI file are displayed on the terminal. Details of the current window or area of interest defined by the WINDOW command are also displayed.
The header values are shown in the current units of measurement. This is dependent on the header type of the input DTI file, or may be set explicitly by the user with the UNITS command.

Messages:

The following messages are specific to the commands SHOW and SHOW FILEIN:

```
*** ERROR *** Specifying command SHOW
Command qualifiers are ENABLE, FILEIN, RANGES or SETTINGS

*** WARNING *** The input DTI file is undefined
```

Examples:

```
VECTORISE>SHOW FILEIN<CR>

LSL$DTI:TEST.DTI
Header: LSLA     Data: WORD

Units are DTI Matrix Values

Matrix Coverage    SW:    1     1    NE:   301   251
Matrix Window      SW:    1     1    NE:   301   251
Matrix Interval     E:    1           N:     1
Value Range          :      0   to    851

VECTORISE>
```

SHOW RANGES

Shows information on the currently defined ranges.

FORMAT: **SHOW RANGES**

Command parameters: None.

DESCRIPTION:

Details of the current defined value ranges are displayed on the terminal. The number of ranges, and the range of values associated with each area are shown.

Messages:

The following messages are specific to the commands SHOW and SHOW RANGES:

```
*** ERROR *** Specifying command SHOW
Command qualifiers are ENABLE, FILEIN, RANGES or SETTINGS

*** WARNING *** No ranges have been defined
```

Examples:

VECTORISE>**SHOW RANGES<CR>**

```
No. of Ranges    :    4
Range 1          :    0
Range 2          :    1     :    10
Range 3          :   11     :    20
Range 4          :   21     :   100

VECTORISE>
```

SHOW SETTINGS

Shows the current value of those attributes that may be set using the SET command.

FORMAT: **SHOW SETTINGS**

Command parameters: None.

DESCRIPTION:

Displays the name of all VECTORISE variables that may be set using the SET command, and their current values.
If the SHOW SETTINGS command is given before using a SET command, the values shown are the default values that are allocated to the variables by VECTORISE.

Messages:

The following message is specific to the command SHOW:

*** ERROR *** Specifying command SHOW
Command qualifiers are ENABLE, FILEIN, RANGES or SETTINGS

Examples:

VECTORISE>**SHOW SETTINGS<CR>**

Current values:

```
BORDER_FC        1        INTERIOR_FC      2        LAYER            1

SCALE       500000.00

VECTORISE>
```

SPAWN

The SPAWN command enables you to create a subprocess while within VECTORISE.

FORMAT: SPAWN command-line

Command parameters:

command-line

Specifies a DCL command string to be executed as if typed in response to a '$' prompt. When the command completes, the subprocess terminates and control is returned to VECTORISE. The command line cannot exceed 80 characters.

DESCRIPTION:

The SPAWN command enables you to create a subprocess while within VECTORISE. When the subprocess terminates control is returned to VECTORISE.

Messages:

The following error messages are specific to the SPAWN command:

```
*** ERROR *** Specifying command SPAWN
Command requires a valid DCL command line
```

```
*** ERROR *** Unable to spawn command, returning to VECTORISE
```

Examples:

VECTORISE> **SPAWN DIRECTORY LSL$DTI:*.DTI<CR>**

```
Directory DUA3:[LSL.DTI]

TEST1.DTI;1             8/8       18-AUG-1987 07:56  [LSL,DAVEC]
TEST2.DTI;1             7/8       18-AUG-1987 17:17  [LSL,DAVEC]
TEST2.DTI;2             7/8       18-AUG-1987 17:34  [LSL,DAVEC]

Total of 3 files, 22/24 blocks.

VECTORISE>
```

UNITS

Specifies the units of measurement that will be used when defining an area of interest in the input DTI file, or when displaying header details from the DTI file. The command also controls the units of measurement used in the output IFF file.

FORMAT: UNITS units

Command parameters:

units

A keyword defining the measurement units, chosen from:

MATRIX	Matrix grid interval units, i.e rows and columns
METRES	Metres on the ground
LATLONG	Latitude and Longitude (in degrees, minutes and seconds)
SECONDS	Seconds of arc
PROJECTION	Projection Record Units (eg. mms on the source document)

DESCRIPTION:

The UNITS command enables the user to specify in what units of measurement he wishes to define an area of interest in the input DTI file using the WINDOW command, or in what units of measurement details from the header of the DTI file are displayed using the SHOW FILEIN command.
The command should be given after defining the input DTI file, since an appropriate default units of measurement is set up whenever an input DTI file is opened.

The command may also be used to define the units of measurement in the output IFF file. If matrix units are currently selected, the coordinates in the IFF file are recorded as real (floating point) column and row values; if metres or projection units are selected before giving the GO command, the IFF data is recorded as metre or projection unit coordinate values. If the selected units of measurement is seconds or latlong when the GO command is given, the coordinates in the IFF file are recorded as column and row values.

Messages:

The following error messages are specific to the UNITS command:

*** ERROR *** Specifying command UNITS
Command qualifiers are MATRIX, METRES, PROJECTION, SECONDS or LATLONG

*** ERROR *** Specifying command UNITS
Command qualifier is invalid for the input file

Examples:

VECTORISE> **UNITS MATRIX<CR>**
VECTORISE>

WAIT

Suspend processing for the specified number of seconds.

FORMAT: WAIT seconds

Command parameters:

seconds

The number (floating point) of seconds for which VECTORISE processing is to be suspended.

DESCRIPTION:

The WAIT command causes processing to be suspended for a specified number of seconds. It is designed for use in software demonstration situations and is of no value in a production flowline.

Messages:

The following error message is specific to the WAIT command:

```
*** ERROR *** Specifying command WAIT
Command requires a real argument
```

Examples:

```
VECTORISE> WAIT 4.0<CR>
VECTORISE>
```

WINDOW

Specifies an area of interest in the DTI file.

FORMAT: WINDOW xmin ymin xmax ymax

Command parameters:

xmin ymin
: The coordinates of the bottom left hand corner of the defining rectangle.

xmax ymax
: The coordinates of top right hand corner of the defining rectangle.

The format of the command parameters is controlled by the UNITS command.

UNITS MATRIX Requires 4 integer values defining the rectangle in terms of column and row numbers

UNITS METRES Requires 4 real (floating point) values defining the rectangle in metre values. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates (eg. U.K. National Grid coordinates); if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

UNITS SECONDS Requires 4 real (floating point) values defining the absolute position of the rectangle in seconds of arc. The values are supplied in the order latitude followed by longitude for the SW corner, and latitude followed by longitude for the NE corner. A negative latitude value indicates a position in the Southern Hemisphere, and a negative longitude value a point to the east of Greenwich.

UNITS LATLONG Requires 4 values defining the absolute latitude and longitude position of the rectangle in the format DDD MM SS H where DDD is the number of degrees, MM is the number of minutes, SS is the number of seconds, and H is the hemisphere (N,S,E,W). The values are supplied in the order latitude followed by longitude for the SW corner, and latitude followed by longitude for the NE corner.

UNITS PROJECTION Requires 4 real (floating point) values defining the rectangle in projection record units. By default, or if the ENABLE ABSOLUTE command has been given, the values are absolute coordinates; if the DISABLE ABSOLUTE command has been given, then the coordinates are offsets from the SW corner of the matrix.

Note that in all cases, the input values are adjusted to the nearest column and row values.

DESCRIPTION:

The WINDOW command is used to limit the vectorising process to a specified area of interest in the DTI file.
The values must be specified in the order bottom left hand (or south west) corner then top right hand (or north east) corner.
The area of interest should lie within the bounds of the DTI file.
If the logical name LSL$MATRIX_DEFAULT_WINDOW is defined with a value of "1", then a default area of interest defined in matrix units of bottom left hand corner 1,1 and top right hand corner 200,200 is set when the file is opened. If the logical name is absent or has any other value, or if the input DTI file has fewer than 200 columns or 200 rows, then a default window equivalent to the total matrix is set.

Messages:

The following messages are specific to the WINDOW command:

```
*** ERROR *** Specifying command WINDOW
The input DTI file must be specified before WINDOW

*** ERROR *** Specifying command WINDOW
Command requires 2 x y coordinate pairs

*** ERROR *** Specifying command WINDOW
NE corner values must exceed SW corner values

*** ERROR *** Specifying command WINDOW
Supplied values exceed matrix extents

*** ERROR *** Specifying command WINDOW
Latitude and longitude values in wrong format
```

Examples:

VECTORISE>**WINDOW 1 1 200 200<CR>**
VECTORISE>**UNITS LATLONG<CR>**
VECTORISE>**WINDOW 42 00 00N 3 00 00E 42 30 00N 2 58 40E**
VECTORISE>

EXAMPLE VECTORISE SESSION

```
$ VECTORISE

VECTORISE> filein test

LSL$DTI:TEST.DTI
Header: LSLA     Data: WORD

Units are DTI Matrix Values

Matrix Coverage    SW:    1     1    NE:   301   251
Matrix Window      SW:    1     1    NE:   200   200
Matrix Interval     E:    1           N:     1
Value Range          :      0   to    851

VECTORISE> window 50 50 150 100
VECTORISE> ranges 200

No. of Ranges   :      6
Range   1       :      0
Range   2       :      1   to    200
Range   3       :    201   to    400
Range   4       :    401   to    600
Range   5       :    601   to    800
Range   6       :    801   to    851

VECTORISE> enable smooth
VECTORISE> enable log_file
VECTORISE> iff test
VECTORISE> go


+--------------------------------------------------------------+
|                                                              |
|                Carrying out Line Following                   |
|                    Patch Number    1                         |
|                                                              |
+--------------------------------------------------------------+

+--------------------------------------------------------------+
|                                                              |
|                Writing segments to IFF file                  |
|                                                              |
+--------------------------------------------------------------+

Total number of segments output ...............  31


+--------------------------------------------------------------+
|                                                              |
|                   Writing node to IFF file                   |
|                                                              |
+--------------------------------------------------------------+
```

```
Total number of nodes output ..................  22


VECTORISE> exit
```

In this example, file LSL$DTI:TEST.DTI has been selected as the input file. An area of interest has been defined in matrix units using the WINDOW command. Only the data inside the specified rectangle will be processed. A constant range value of 200 has been used, and this has resulted in the data being classified into 6 area ranges. The options to interpolate the links before output, and to write processing and statistical information to a log file have been enabled. The vector data resulting from the vectorisation process is written to the file LSL$IF:TEST.IFJ Since no SET LAYER command has been given, the data is written to layer 1 in the file, and since no SET BORDER_FC or SET INTERIOR_FC commands have been given, border links are given a feature code of 1, while all other links are given a feature code of 2.

EXAMPLE VECTORISE SESSION

If the ENABLE LOG_FILE command is given prior to initiating the vectorising process, a LOG file containing details of the run is generated.

```
--------------------------------------------------------------------------
VECTORISE LOG file Created on                : 15-SEP-87   10:24:27
--------------------------------------------------------------------------
Input from file     :  LSL$DTI:SWDTED.DTI

Matrix Coverage   SW:     1     1  NE:   301   251
Matrix Window     SW:   100   100  NE:   150   150
Matrix Interval    E:     1         N:     1
Value Range         :     0   to    851

Output to file      :  DUA0:[LSL.IFF]TEST.IFJ;2
--------------------------------------------------------------------------
Range Values        :

Range    1          :      0
Range    2          :      1   to    200
Range    3          :    201   to    400
Range    4          :    401   to    600
Range    5          :    601   to    800
Range    6          :    801   to    851

--------------------------------------------------------------------------

Output is to layer 1
Border links are feature code 1
Interior links are feature code 2
Option to smooth links is NOT selected

--------------------------------------------------------------------------
Vectorisation summary             :

Total number of segments output   :     33
Border segments                   :     18
Interior segments                 :     15

Total number of nodes output      :     22
Border nodes                      :     18
Junction nodes                    :      2
Loop nodes                        :      2

Link point storage used           :   0.15%
Link ends storage used            :   0.08%
Node point storage used           :   0.14%
Node arm storage used             :   0.08%
--------------------------------------------------------------------------
```

VECTORISE messages

MESSAGES (SUCCESS)

These messages are used to indicate that the program has succeeded in performing some action, and do not require any user action.

NORMAL, VECTORISE normal successful completion

Explanation: VECTORISE has terminated successfully, without encountering any errors.

User action: None

MESSAGES (WARNING)

These messages are output when an error has occurred that can be corrected immediately by the user or that the program will attempt to overcome.

ERRSMOOTH, Error when smoothing FSN 'integer'

> **Explanation:** The program has been unable to smooth the link. VECTORISE will write the link to the output file unsmoothed
>
> **User action:** Please notify Laser-Scan of the problem

MESSAGES (FATAL)

These messages indicate a severe error in processing, or some form of system failure, which has caused the program to terminate.

FOLLOWERR, Error during line following

> **Explanation:** An internal consistency error has occured during the line following process.
>
> **User action:** Please submit a SPR to Laser-Scan, along with any log file and the input DTI file

LOSTNODE, Lost node during line following

> **Explanation:** An internal consistency error has occured during the line following process.
>
> **User action:** Please submit a SPR to Laser-Scan, along with any log file and the input DTI file

TOOMNYARMS, Maximum number of arms ('integer') exceeded

> **Explanation:** VECTORISE is currently dimensioned to handle a maximum number of arms. This limit was exceeded during the run of the program.
>
> **User action:** Please submit a SPR to Laser-Scan who may be able to redimension the program.

TOOMNYCOLS, Number of columns ('integer') is too large

> **Explanation:** VECTORISE is currently dimensioned to vectorise an area defined by a maximum number of columns. This limit was exceeded during the run of the program.
>
> **User action:** Please submit a SPR to Laser-Scan who may be able to redimension the program.

TOOMNYLNKS, Maximum number of links ('integer') exceeded

> **Explanation:** VECTORISE is currently dimensioned to handle a maximum number of links. This limit was exceeded during the run of the program.
>
> **User action:** Please submit a SPR to Laser-Scan who may be able to redimension the program.

TOOMNYNODES, Maximum number of nodes ('integer') exceeded

> **Explanation:** VECTORISE is currently dimensioned to handle a maximum number of nodes. This limit was exceeded during the run of the program.
>
> **User action:** Please submit a SPR to Laser-Scan who may be able to redimension the program.

TOOMNYPTS, Maximum number of points ('integer') exceeded

Explanation: VECTORISE is currently dimensioned to handle a maximum number of coordinate points. This limit was exceeded during the run of the program.

User action: Please submit a SPR to Laser-Scan who may be able to redimension the program.

TOOMNYROWS, Number of rows ('integer') is too large

Explanation: VECTORISE is currently dimensioned to vectorise an area defined by a maximum number of rows. This limit was exceeded during the run of the program.

User action: Please submit a SPR to Laser-Scan who may be able to redimension the program.

MESSAGES (OTHER)

In addition to the above messages which are generated by the program itself, other messages may be produced by the command line interpreter (CLI) and by Laser-Scan libraries. In particular, messages may be generated by the DTILIB library and by the Laser-Scan I/O library, LSLLIB. DTILIB library messages are introduced by '%DTILIB' and are documented in the MATRIX package reference manual. In most cases DTI errors will be due to a corrupt input file, and this should be the first area of investigation. If the cause of the error cannot be traced by the user, and Laser-Scan are consulted, then the output file should be preserved to facilitate diagnosis. LSLLIB messages are introduced by '%LSLLIB' and are generally self-explanatory. They are used to explain the details of program generated errors.

INDEX