SNAP UTILITIES

Version 2.1

Users Manual

Chris Crook
Land Information New Zealand
November 2003

Disclaimer

Download and use of these software applications is taken to be acceptance of the following conditions.

- Land Information New Zealand (LINZ) does not offer any support for this software.
- The software is provided "as is" and without warranty of any kind. In no event shall LINZ be liable for loss of any kind whatsoever with respect to the download, installation and use of the software.
- The software is designed to work with MS Windows. However, LINZ makes no warranty regarding the performance or non-performance of the software on any particular system or system configuration.

CONTENTS

Chapter One:Introduction

The SNAP adjustment program takes a number of data files of survey observations and a file of survey coordinates and adjusts the coordinates to best fit the data. Associated with SNAP are a number of utility programs to facilitate its use and to interpret the results of the adjustment.

This manual describes some of the utilities. The main utilities that are discussed are:

- SNAPPLOT — This is the most significant of the utilities. It provides a map type view of the survey network. Although it can display the input coordinate and data files before running SNAP, its main use is in displaying and analysing the results of the adjustment. It can also create Autocad DXF format files of the survey network which are suitable for input into CAD and other plotting packages.
- Translators — Several translators are available for converting data files generated by GPS processing software into input files of vectors for SNAP.
- Coordinate file utilities — The coordinate file represents the stations in the survey network. Before SNAP or SNAPPLOT can use data they require coordinate files describing the network. SITE2GPS is a utility for calculating station coordinates from supplied data and the coordinates of a few known stations. Other coordinate file utilities allow geoid height information to be added to a coordinate file, and change the coordinate system used in the file.
- Post processing utilities — SNAP produces a binary file which summarises the results of the adjustment of the data and a listing file which is a readable version of the results. Two programs have been written to obtain information from the binary file which is not listed conveniently in the binary file. These are LISTGPS, which tabulates GPS data in a form suitable for inclusion in survey plans, and BADLINES, which identifies lines in a GPS survey which are not within the required tolerance.

These utilities are described in the following chapters.

Chapter Two: SNAPPLOT

Introduction

What is SNAPPLOT

SNAPPLOT is an interactive graphical program for viewing the network and data you are using in an adjustment and the results of the adjustment. It can also create AutoCad DXF format files, which can be imported to most CAD and other graphics software packages.

System requirements

- To run SNAPPLOT you require a PC with VGA graphics capability and with a Microsoft compatible mouse or pointing device. This version requires a 386 or better computer(!). It used a DOS extender technology (predating MS windows), and may exhibit some incompatibilities with modern version of windows.

Running SNAPPLOT

Preparation

Before running SNAPPLOT you must prepare the data files that it requires. The files that SNAPPLOT may use are:

- a coordinate file listing the stations to be plotted
- one or more data files listing the observations to be plotted
- a SNAP command file which lists the coordinate file and the data file
- a binary file generated by SNAP which summarizes the result of an adjustment, including station and observational errors
- SNAPPLOT configuration files which define how the data are to be displayed

If you are running SNAPPLOT before adjusting the data with SNAP, then you require the coordinate file and the data files. You can also use a command file and configuration files. If you are running SNAPPLOT after running SNAP then you need only the SNAP command file and the binary file created by SNAP. The binary file contains the information about the coordinates and the observations.

The formats of the coordinate file and data files are described in the SNAP users manual. These files may be created using a text editor, or they may be generated by other programs (for example they may be generated from GPS data files).

The command file can be used by both SNAP and SNAPPLOT. It contains a list of commands which describe to SNAP what adjustment is to be done and what data are to be used. SNAPPLOT only uses a small number of the commands that may be in the file - the rest are ignored. It also uses one command (the plot command) that is ignored by SNAP. See "How SNAPPLOT uses a SNAP command file" on page 2-10 for more information about how command files are used.

The SNAP binary file is created by SNAP automatically when it does an adjustment. The file has the same name as the command file, but has extension ".BIN". It contains in machine readable form all the stations and observations used in the adjustment as well as the error ellipses of stations and the residual errors of observations.

SNAPPLOT may use one or more configuration files as it starts up. These files are used to set up users preferences. Configuration files can also be used while SNAPPLOT is running. As an example you may have several configuration files each defining a different colour scheme - one might be suitable for locating large residuals while another is set up to display station error ellipses. Configuration files are most easily created by SNAPPLOT itself. They are text files and can be edited using normal text editors. See "Using SNAPPLOT configuration files" on page 2-11 for more information.

Starting SNAPPLOT

SNAPPLOT can be run either by specifying the name of a SNAP command file from which the coordinate and data file names are read, or by explicitly specifying a coordinate and configuration file. If it is started using the name of a command file then it will look for the corresponding SNAP binary file. If it finds the binary file it gives the option of using it or ignoring it.

The command used to start SNAPPLOT using with command file is:

SNAPPLOT [*options*] *command_file* [*configuration_file ...*]

To start SNAPPLOT using just coordinate and (optional) data files, the command is

SNAPPLOT [*options*] -f *coordinate_file* [*data_file* ...]

The -f option in the command instructs SNAPPLOT that the coordinate and data files are being named explicitly. The brackets [] in these commands are not to be typed in - they denote optional components of the command line.

The options that may be used in the command are:

-b background_file	Defines the name of a file of background linework to be shown on the plot. This could include coastlines or road patterns, for example. See "Adding background linework to the map" on page 2-6 for more information on background files.
-c configuration_file	Specifies the name of a configuration file that will be used to control the initial display of information on the screen
-p projection	Defines the coordinate system that will be used to display the stations. projection is the code for a coordinate system defined in the COORDSYS.DEF file (see the SNAP users manual for more information about the COORDSYS.DEF file).
-s#	Defines the video mode (SNAPPLT3 only). # may be 1 for standard VGA resolution, 2 for 800x600 resolution, or 3 for 1024x768 resolution. This command is optional - the default is standard VGA resolution.

SNAPPLOT may load one or more configuration files automatically when it starts. See "Using SNAPPLOT configuration files" on page 2-12 for a description of these files.

If SNAP finds no errors reading the data files and configuration files it will then present a screen showing a map of the stations in the network and the observations between them (see figure 1).

To quit SNAPPLOT you can either choose "File | exit" from the menu, or just press Alt-X (that is press the X key while holding down the Alt key).

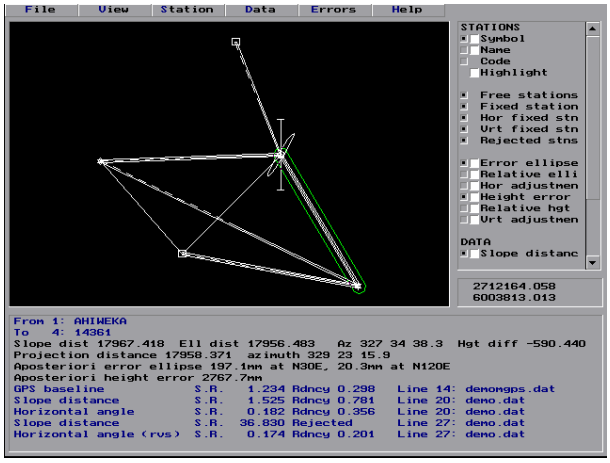

Figure 1: The SNAPPLOT screen

The SNAPPLOT screen

There are five main components in the SNAPPLOT screen. These are:

- the menu (across the top) from which commands are selected.
- the map window, in which the data are displayed. This can also be used to select stations or observations about which more information is required.
- the key (right hand side) which shows the colours of various components on the map and allows you to change the colours and choose which items are to be displayed.
- the coordinate window (immediately below the key) which shows the coordinates of the cursor when it is over the map window
- the text window (at the bottom) which displays text information about stations or observations. This window may also be used to select observations or stations to display in the map and to obtain more information about specific observations.

The size of the text window can be changed with the menu command "View | Text area size". This can shrink the window down to just four lines of text, or enlarge it to fill half the screen.

Most of the interaction with the screen is done using the mouse. In this documentation the following terms will be used to describe mouse actions:

- clicking an object. This means positioning the mouse pointer over an object and pressing and releasing one of the mouse keys. This can be either "left-clicking" or "right-clicking" depending upon the key that is used. Sometimes you will hold down one of the keyboard keys at the same time as clicking an object. For example Shift+left-click means to hold down a Shift key while left-clicking an object.
- dragging. This means positioning the cursor, pressing a button, and holding the button down while moving the cursor to a new location, then releasing the button. Again the left or right buttons can be used, and one or more keyboard keys may be pressed while dragging. For example Shift+left-drag means holding down the shift key while dragging with the left mouse button.

Menu selections are described by the name of the menu item. Where a menu selection involves using a submenu this is indicated by a vertical bar "|". For example the menu command "View | Text area size" referenced above means "click on the view menu item, then click on the Text area size item of the submenu that appears".

Using the map window

The map window is the main window for displaying the data. It can show the stations, error ellipses, height errors and the observations. It can also show a backdrop of other

linework such as coastlines or roads to help recognize the location of stations (see Adding background linework to the map, page 2-6).

The colours used for the various items and which items are displayed is controlled by the key (described below).

Stations on the map are shown by a symbol which depends upon whether they have been adjusted or not. The symbols are

circle	an adjusted (or floating) station
triangle pointing up	fixed horizontally (may be adjusted vertically)
triangle pointing down	fixed vertically (may be adjusted horizontally)
two triangles superimposed	fixed both horizontally and vertically
square	not used in the adjustment

Navigating around the map

When SNAPPLOT starts it shows a map scaled to include all the stations in the network. However you can zoom in or out to display different portions of the network in more detail. The following list describes how to do this:

- To zoom in on a region — left-drag a rectangle covering the area of interest. While you are dragging the rectangle you can press the shift key down to move it to a new location.
- To pan the view — shift+left-drag a point on the map to the position you want it to appear in.
- To zoom out — shift+right-click any point on the map, or select "View | Zoom out" from the menu.
- To revert to a previous view — right-click any point on the map or select "View | Previous view" from the menu
- To view the entire network — select "View | All" from the menu

You can also centre the map over a station or observation that is highlighted in the text window by shift+left-clicking on the item in the text window.

Options for displaying stations

You can control the size of symbols that SNAPPLOT uses for stations, which stations are plotted (to a limited degree), and the annotations of station code and name. These parameters are modified by using the key (see "Using the key" page 2-7), and by the station options dialog box.

To see the station options dialog box select "Station | Options" from the menu. In this dialog box you can specify the size of the symbol used to mark stations and the size of text used for station codes and names. The sizes are specified in metres at true scale. You can also select to automatically resize the symbol or text as the map scale changes - these options are controlled by the "autoscale" boxes in the dialog box.

In the dialog box you can select one of two different fonts to show station names and codes - either a scaleable font, which reflects the requested text size, or a fixed size font. The fixed size font is generally more legible. However if you choose to use the fixed size font then the display on screen will not generally reflect the contents of any DXF files created from the display. DXF files are always created with a variable sized font.

The dialog box also allows you to choose whether to display stations at their offset locations. For more information about this see "Showing stations at offset locations" on page 2-6.

By using the key you can select which stations to show. For example you can choose not to show any rejected stations. You can also choose whether to display the station symbol, or the station text, and what colour to use if you do display them.

Changing the scale of station errors and displacements

Error ellipses and height errors are displayed at an exaggerated scale on the map. For example if the axis of an error ellipse is 0.1m long it could be exaggerated 1000 times so that it would appear to be 100m long on the map. The amount of exaggeration is set by the Error Options dialog box, accessed by selecting "Error" from the menu.

A different exaggeration can be selected for the horizontal and vertical components of the error. The exaggerations are used for showing station displacements (the change to a station's coordinates during the adjustment) as well as for errors.

The dialog box allows you to "autoscale" the exaggeration. If you select this option then the exaggeration will change as the scale of the map changes in order to keep the apparent size of the errors constant. When autoscaling is selected SNAPPLOT calculates an default exaggeration based on which items are being plotted (that is which of horizontal and vertical station errors, horizontal and vertical relative errors, and horizontal and vertical station displacements).

The Error Options dialog box also allows you to select how the errors are represented. They can be apriori or aposteriori errors, and can show standard errors or percentage confidence limits.

Options for displaying data

In a survey most lines are observed more than once, often with different types of measurements. SNAPPLOT has a facility for offsetting the lines representing an observation so that they do not overlap on the screen. This option is accessed with the Observation Options dialog box, accessed by selecting "Data | Options" from the menu.

There are three options for offsetting observations. These are:

- Draw just one line. All observations are merged and just a single line is drawn.
- Merge observations of the same type. Each observation has a corresponding entry in the key. Observations for the same entry are merged into a single line, but observations for different entries are offset. So if the key shows observations coloured by data type then all horizontal angles will be merged and all slope distances will be merged, but horizontal angles and slope distances will be shown by separate offset lines. They will be offset even if they have the same colour in the key.
- Show all observations of each line. Every observation is drawn with a separate offset line.

The offset between the lines is specified in metres at true scale. You can also choose to "autoscale" the offset, so that the displayed offset appears constant on the screen as the map scale changes.

SNAPPLOT also has an option for showing the direction of observations by using a solid line at the instrument station end of the line and a dashed line at the target station end. This only applies for data types such as horizontal angles for which the direction is important. Data types for which instrument and target station are interchangeable are always shown as solid lines.

Finding out about a stations and observations

You can find out about any station or observation on the map by left-clicking the mouse over the item of interest. More information about the item will be displayed in the text

window. Also a line will be drawn around the item to show what is referenced by the text window.

When a station is selected you can left-click it again to lock the selection. This is indicated by a double circle around the station. You can then click on other stations in the network to measure their distance and azimuth from the locked station. To unlock a station you can either lock another station (by clicking it twice), or left-click on the background away from any station or observation, or left-click on the locked station.

Showing stations at offset locations

Sometimes it is convenient to display stations shifted from their true location. For example if three stations are on line it is difficult to distinguish observations to or from the middle station from those between the end stations. The network is more understandable if one of the stations is shifted to one side. Similarly, it may be convenient to show remote stations much closer to the body of the network than their true location. In SNAPPLOT you can shift a station away from its correct position to clarify the map.

To move a station to a new location, ctrl+shift+left-drag the station (that is, hold down the control and shift keys, and left-drag the station). You can restore a station to its correct position by using ctrl+shift+right-click over the station.

When stations are displaying away from their correct location the coordinate window displays “Stations not in true positions”.

All stations can be reverted back to their correct positions by selecting “Station | Options” from the menu. This will present a dialog box with an option to “Ignore station offsets”. Select this option and click the “OK” button to remove the dialog box. This option doesn’t remove the station offsets, it just ignores them. By unselecting the option from the dialog box you can show the offset positions again. When you shift a station to an offset position this option is automatically unselected.

Station offsets can be specified in a configuration file or in the command file. See the sections “How SNAPPLOT uses a SNAP command file” (page 2-10) and “Using SNAPPLOT configuration files” (page 2-11).

Adding background linework to the map

SNAPPLOT can draw a backdrop of roads, lines, and other features under a survey network. To do this you need to create one or more files defining the linework. A linework file is in a simple ASCII format described below. Each linework file must be associated with the job by a “plot background” command in the command file (see page 2-11) or by the -b option on the SNAPPLOT command line (see page 2-2).

Each line in the linework file defines a pen code and the coordinates of a point. Linear features on the map start with a non-zero pen code. Subsequent points in the feature have a pen code of zero.

The coordinate system for the linework file need not be the same as that used to plot the map - SNAPPLOT will convert the coordinates if necessary. It can be defined either in the linework file or in the “plot background” command. A definition in the linework file overrides that in the command. The coordinate system must be a coordinate system defined in the coordinate system definition file (COORDSYS.DEF) and is defined by the code used in that file. For example, New Zealand Map Grid is defined by the code NZMG. See the SNAP users manual for more information about the coordinate system definition file.

To define the coordinate system in the linework file include a line of the form

#coordsys *code*

where *code* is a one of the codes in the coordinate system definition file (for example “NZMG” for New Zealand Map Grid). The coordinate system will apply to all subsequent points in the linework file unless it is overridden by another #coordsys line.

If the coordinate system of the linework is not defined or cannot be converted to the plot projection then the data in the file are ignored.

The coordinates of the features are represented by an easting and a northing in metres for a projection coordinate system, or a longitude and latitude in decimal degrees for a geodetic coordinate system.

Each feature in the linework file has an associated layer, such as “road” or “coastline”, that is listed in the key. The layer can be defined in the linework file or in the “plot background” command. Definitions in the file override that in the command. The layer is defined in the linework file by a line of the form

#layer *layer_name*

If no layer name is defined for a feature then SNAPPLOT will use the pen number of the first point of the feature as the layer name.

The following example shows part of a linework file.

```
#coordsys NZMG
#layer coastline
1  2561003.0 6206834.0
0  2569998.0 6206835.0
0  2569994.0 6206836.0
.....
```

If this file was called coast.dat then it would be plotted by adding the following line to the SNAP command file:

plot background coast.dat

Creating a DXF file of the map

SNAPPLOT can create a Autocad DXF format file which reflects the map as displayed in the map window. This file can be used by most CAD and drawing packages to load a copy of the map, possibly edit it, and then send it to a plotter or printer.

The steps to create a DXF file are:

- Zoom in to the area of interest
- Set up the key to show the items to be included in the DXF file
- Select “File | Create DXF file” from the menu
- Enter the name of the DXF file to be created. The default is to create a file with the same name as the command file, but with extension “.DXF”.

The DXF file should exactly reflect the map. The only significant difference is that the map may be using a fixed sized font for station names and codes, whereas the DXF file always uses a scaleable font. To ensure that the map is drawn with a scaleable font select “Station | Options” from the menu to get the “Station options” dialog box, and make sure that the “Used fixed size font on screen” option is unselected.

Using the key

The key shows a list of the components that are or may be displayed on the map. Each item may include a selection box, used to indicate whether or not the item is displayed, and a colour block, which shows the colour used.

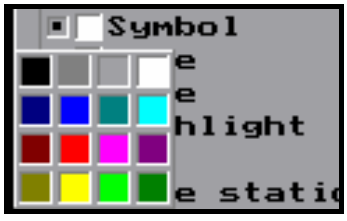

Figure 2: A portion of the key showing the colour selection palette.

Figure 2 shows as section of the key. The entry for “Symbol” has both a selection box (currently selected, as indicated by the black square), and a colour block showing the colour that station symbols will be plotted, currently white.

To select or unselect an item on the key simply left-click

the mouse over the selection box. To change the colour of an item, position the mouse over the colour box and press and hold down the left button. This will display a colour selection palette over the colour block (figure 2). Move the mouse over the desired colour and release the button.

Changes to the key are not immediately implemented in the map. After all the desired changes are made left+click the map to apply them.

Note that some items may be referenced by several items in the key. For example the key includes options to select data of each data type, and also to select used or rejected observations. A rejected horizontal angle is controlled by the horizontal angle selection box and by the rejected observation selection box. SNAPPLOT will only plot the observation if both boxes are selected.

The simplest way to understand the effect of the selection boxes is to think of them as turning off groups of data or stations.

Often observations of one colour are hidden as other observations are drawn near them or over the top of them. You can force SNAPPLOT to redraw observations of a specific colour by left-clicking the text area of the key item. Note that this only applies for observations (the data section of the key), not for stations or error information. You can use this method to change the colour of one category of observations without redrawing the map. Change the colour for the item as described above, then click on the text area of the key.

The colour of an item in the data section of the key can be changed to the highlight colour (that is the colour of the highlight item) by shift+left-clicking on the item in the key. This automatically changes the colour of the item and redraws the corresponding observations. To restore the data back to its original colour shift+left-click the text again, or shift+left-click on the text of another data item.

Changing the colour scheme for data

By default SNAPPLOT colours observations according to the type of observation (angle, distance, or whatever). However you can choose other colour schemes with the “Data | Colour by...” menu item. The options available (which appear on a submenu) are:

- data type — Observations are coloured according to the data type
- data file — Observations are coloured to indicate the data file from which they were read
- standardised residuals — Observations are coloured according to their standardised residual. SNAPPLOT prompts for the maximum residual to show and the number of categories. For example if you choose a maximum residual of 3.0 and 3 categories, then SNAPPLOT will create four entries in the key for standardised residuals between 0.0 and 1.0, between 1.0 and 2.0, between 2.0 and 3.0, and greater than 3.0.
- redundancy — Observations are coloured according to the redundancy of the observation. SNAPPLOT prompts for the number of categories to show. It will create the specified number of categories between 0.0 and 1.0.
- user defined classifications — Any user defined classifications in the data may be used as a key for the data (See the SNAP user manual for more information about classifying data).

Highlighting groups of observations

In addition to using the key to colour observations, groups of observations or stations can be highlighted. That is, they are drawn with the colour shown as “Highlight” in the key,

rather than in the colour otherwise defined by the key. For example you can highlight all observations with a standardised residual greater than 2.5, or all rejected stations and observations to them.

Highlighting is done using the "Stations | Highlight" and "Data | Highlight" menu items. Each option displays a dialog box which is used to select what to highlight.

The "Highlight stations" dialog box (Figure 3) presents a list of stations within which you can choose which stations to highlight.

To add or remove a station from the list of highlighted stations simply left-click the station in the list. The list can display either by station name or by station codes. Click the "Show station names" box to toggle between showing names and showing codes.

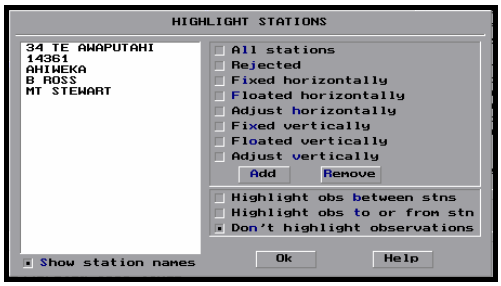

Figure 3: The highlight stations dialog box

The dialog box also allows groups of stations to be added or removed. To do this select on or more of the check boxes specifying groups of stations, then click the "Add" or "Remove" button. For example to highlight all rejected stations, click the "Rejected" check box, then click "Add".

The highlight stations dialog box also provides options for highlighting observations. You can select to highlight observations to or from any of the highlighted stations, or only observations between the highlighted stations, or to not highlight observations.

Selecting "Data | Highlight" from the menu displays the "Highlight observations" dialog box. This allows you to select a category of observations to highlight. The categories available are:

- Observations with a standardised residual greater than a threshold value which is defined in the dialog box
- Observations with an aposteriori standardised residual greater than the threshold
- Observations with a redundancy less than the threshold
- Rejected observations
- Unused observations (observations between rejected stations which are not explicitly rejected themselves)

This dialog box also allows highlighting observations to and from or between highlighted stations.

Using the text window

The text window provides interactive access to much of the information that is presented in a SNAP listing file. The window can display any of the following items:

- General information — The name of the job and statistics of the adjustment. This information is displayed when SNAPPLOT starts up. It can be redisplayed at any time by left-clicking on an empty region of the map
- List of stations — To obtain a list of all stations select "Stations | List" from the menu. The stations can be listed either in order of station code or in order of name. This list is not generally useful other than as a means of getting more information about individual stations. To get information about an individual station left-click the mouse over the line referring to the station. This will present information about the station in the text window and draw a circle

around it in the map window. Pressing shift+left-click on the station will bring it into the centre of the map.

- List of observations — To obtain a list of individual observations select "Data | List" from the menu. You can select to list all observations, rejected observations, unused observations (either rejected observations or observations to rejected stations), or used observations. The observations are listed in order of standardised residual from the largest to the smallest. To get more information about an observation left-click on it. To bring an observation into the centre of the map shift+left-click it.
- Station information — To get information about a specific station either left-click on the station in the map window, or left-click on the station in the list of stations in the text window. SNAPPLOT will list information about the station and a list of stations that it is connected to.
- Line information — To get information about a observed line between two stations you can left-click on the observation in the map window. To get information about the line between any two stations left-click twice on one station to "lock" it, then left-click on the other station. The line information includes the distance and bearing between the stations and a list of observations between them. To get more information about one of the observations left-click on the observation in the text window.
- Observation information — Detailed information about individual observations can be obtained from the "Line information" or "List of observations" views in the text window.

The text window remembers what it has displayed. You can revert to previous views by right-clicking on the text window. The last 20 views can be accessed in this way.

How SNAPPLOT uses a SNAP command file

SNAPPLOT uses the same command file as SNAP to determine what information to plot (see the SNAP users manual for detailed information about command files). However it ignores most of the commands in the command file. The only commands it reads from the file are the title, coordinate_file, data_file, plot, and include commands. If it is using a binary file created by SNAP SNAPPLOT also ignores the title, coordinate_file, and data file commands.

The syntax of these commands is summarized briefly here. For more information refer to the SNAP users manual.

SNAP commands used by SNAPPLOT

title ***text***

text is a description of the survey

coordinate_file ***file_name*** **[*format*]**

specifies the coordinate file used by SNAPPLOT. The format is optional and can be either "SNAP" or "GB". The "GB" format is used by DOSLI Geodetic Branch.

data_file *file_name* [*format*]

specifies a data file to be used by SNAPPLOT. The format is optional and can be either “SNAP” or “GB”. The “GB” format is used by DOSLI Geodetic Branch.

include *command_file*

specifies the name of a command file from which to read additional commands.

The plot command

The plot command is used only by SNAPPLOT - SNAP ignores it. The command is entered as

plot *option data*

where *option* defines the purpose of the command and *data* is additional information relating to it. This command is used to provide information which is specific to the adjustment. In contrast plot configuration files are generally for information which can apply to a number of different adjustments.

The options available in the plot command are as follows:

plot configuration *config_file_name*

specifies the name of a SNAPPLOT configuration file to load before displaying the data.

plot projection *projection_id*

specifies the coordinate system that will be used to plot the network (the coordinate system must defined in the COORDSYS.DEF file, see the SNAP users manual). This must be a projection coordinate system. If the command file does not include a “plot projection” command then SNAPPLOT will use the coordinate system in the coordinate file. For geodetic coordinate systems (latitude and longitude) it will use an arbitrary transverse mercator projection to actually display the data. An example of the plot projection command is

```
plot projection NZMG
```

plot *background background_file* [*coordsys_id* [*feature_name*]]

This command specifies the name of a file of linework that will be used as a backdrop for plotting the network (see “Adding background linework to the map”, page 2-6). In this command *background_file* is the name of the file of linework, *coordsys_id* is the code for the coordinate system used in the file (corresponding to a code in the COORDSYS.DEF file), and *feature_name* is an optional name for the type of feature in the file (for example coastline).

plot offset_station *station_code east_offset north_offset*

Specifies stations that are to be plotted out of position on the map. In this command *station_code* is the code of the station to be offset, and *east_offset* and *north_offset* are the offset distances in metres. The offset is always expressed in metres east and north, regardless of the coordinate system of the data file. A separate command is required for each offset station.

Using SNAPPLOT configuration files

SNAPPLOT configuration files are used to save and recall settings such as the colours selected on the key, the scaling of errors, the offsets at which stations are plotted, and the

size of the text window. Configuration files are used as SNAPPLOT starts up to initiallize the program. They can also be loaded as SNAPPLOT is running to change the configuration.

You can have different configuration files set up to highlight different aspects of the data. For example one might be set up to highlight observations with bad residuals, while another may emphasize station error ellipses.

Creating SNAPPLOT configuration files

The easiest way to create a configuration file is to run SNAPPLOT, set up the configuration that you want and then select "File | Save configuration" from the menu. You will be prompted for the name of the configuration file. The default extension for the file is ".SPC" (for SnapPlot Configuration). If you do not specify a file name then the configuration file will have the same name as the command file (but extension .SPC), and will be loaded automatically each time the job is viewed with SNAPPLOT.

The configuration file is a text file that can be edited with any normal text editor. Any of the commands can be removed from the configuration file - the corresponding parameters will be left unchanged when the file is loaded. One use for this facility is to create configuration files which just hold information about the key. You can set up the key as you want it, save the configuration file, and then edit the file to leave just the commands defining the key. You can then load this file back at any time to change the key, yet leave other parameters unchanged. You can create several such files to reveal different aspects of the data. The configuration files can be incorporated into a "Config" menu so that different configurations can be easily selected.

Using SNAPPLOT configuration files

Configuration files can be used in the following ways:

- automatically loaded by SNAPPLOT as it starts up
- specified in the command file with the plot configuration command, and loaded as the program starts up
- specified on the SNAPPLOT command line and loaded as the program starts
- loaded while SNAP is running by selecting File | Restore config from the menu
- loaded while SNAP is running by selecting from the Config menu. This is described in more detail in the following section.

When SNAPPLOT starts up it searches for a number of configuration files automatically and loads them if it finds them. The files that it searches (in the order that it uses them) are:

- SNAPPLOT.SPC in the same directory as the SNAPPLOT program.
- SNAPPLOT.SPC in the user's directory. This directory is defined using the SNAPDIR environment variable.
- SNAPPLOT.SPC in the same directory as the command file
- job_name.SPC, where job_name is the name of the command file.

It will then load any configuration files referenced in the command file with the plot configuration command, and finally the configuration file specified on the command line used to start up SNAPPLOT.

SNAPPLOT will search for configuration files in three directories:

- the directory containing the command file
- the users directory. This directory is defined by the SNAPDIR environment variable.
- the program directory in which the SNAPPLOT program exists.

SNAPPLOT will use the first configuration file it finds by searching the directories in this order.

Creating a configuration file menu

You can set up a list of "favourite" configuration files in a SNAPPLOT menu, so that they can be readily loaded with a few mouse clicks. To create a config menu, carry out the following steps:

1. Create the configuration files to be included in the menu (see "Creating SNAPPLOT configuration files", page 2-12). Put these files into a general purpose directory (rather than a directory for a specific job).
2. If you do not have a default configuration file then create one. This is SNAPPLOT.SPC in either the program directory or in a users directory as defined by the SNAPDIR environment variable.
3. Add config_menu commands to the default configuration file (see "config_menu", page 2-13) for each configuration file. Each config_menu command should include the full file name (including drive and directory) for the configuration file, and the corresponding text to appear in the menu.

After these steps have been completed, SNAPPLOT will display a menu item "Config", from which each configuration file can be selected.

Configuration file commands

Screen layout

display_mode *mode*

This command defines the screen resolution. This can only be used in a start up configuration files - it is ignored in configuration files loaded from the menu. *mode* is a number between 1 and 3 inclusive. The value 1 corresponds to normal VGA resolution (640x480), 2 corresponds to 800x600 resolution, and 3 corresponds to 1024x768 resolution. (This is only used by SNAPPLT3 at present).

text_rows *rows*

Defines the size of the text window at the bottom of the screen. *rows* is the number of lines in the window. The minimum value is 4 lines. The maximum value corresponds to half the screen, and depends upon the display mode.

config_menu *cfg_file_name menu_text*

Adds a configuration file to the "Config" menu. This is useful for frequently used configuration files. This command is only used in start up configuration files - it is ignored in configuration files loaded from the menu. This is most sensibly placed in one of the automatically loaded configuration files to reference other configuration files in the default directories (either the program or users directory). In this command *cfg_file_name* is the name of the configuration file, and *menu_text* is text that will appear in the menu. The menu text can include an "&" before a character to define it as the hot-key character (i.e. the character that can be used to select the menu item). It can also include spaces.

Station plotting options

station_size [type] size [times_default]

Specifies the size that station symbols and text (names and codes) will be plotted. In this command *type* is one of "symbol" or "text" to indicate what the size

applies to. If neither are specified then the size applies for both. *size* is the size that will be used and is either a size in metres at true scale, or it is expressed a multiple of the default size that SNAPPLOT will use (e.g. 2.0 times_default). If the size is expressed as a multiple of the default then the actual size will vary as the scale of the plot changes so that the plotted size is approximately constant.

use_fixed_size_font yes|no

This command provides the option of using a fixed size font for plotting station names and codes. The fixed size font is generally more legible than the variable sized font. However it is not used when DXF files are plotted, so that while this option is selected the screen will not exactly reflect the contents of a DXF file generated from it.

offset_station *code east_offset north_offset*

Specifies stations that are to be plotted out of position on the map. In this command *station_code* is the code of the station to be offset, and *east_offset* and *north_offset* are the offset distances in metres. The offset is always expressed in metres east and north, regardless of the coordinate system of the data file. A separate command is required for each offset station.

ignore_station_offsets yes|no

Specifies whether stations are to be plotted in their offset locations.

Observation plotting options

observation_colours by_type
observation_colours by_file
observation_colours by_std_residual [apriori|aposteriori] *max_sres ngroups*
observation_colours by_redundancy *ngroups*
observation_colours by_classification *class_name*

Defines the colour scheme that will be used to plot observations. The basic options are to colour observations by_type (e.g. slope distance, horizontal angle), by_file (i.e. according to the data file from which the observations are read), by_std_residual, by_redundancy, or by_classification.

To colour by standardised residual you must specify whether apriori or aposteriori standardised residuals are to be used, what is the maximum standardised residual to be in the groupings that will be generated, and how many groups to form.

To colour by redundancy you must specify how many groups (ranges of redundancy values) to form.

To colour by classification you must specify which classification to use. This should correspond to one of the classifications defined in one of the data files.

observation_options *options...*

Specifies some options related to plotting observations. In this command *options* can include any of the following:

show_obs_direction	Observations for which the direction of observation is important (for example horizontal angle observations) are shown with a dotted line at the target station and a solid line at the instrument station.
no_show_obs_direction	All observations are shown as a solid line
merge_all_obs	All observations between two stations are represented with a single line between the stations (that is, observations are not offset).

merge_similar_obs	Observations of a line corresponding to different entries in the key are offset from one another.
no_merge_obs	All observations of a line are offset from one another.

observation_spacing *offset* [times_default]

This command specifies the offset between observations if the merge_similar_obs or no_merge_obs observation options are in force. The offset is expressed as either a distance in metres at true scale, or as a multiple of a default offset calculated by SNAPPLOT (e.g. 2 times_default)

highlight_observations *option* [*threshold*]

Specifies which observations are to be highlighted. *option* can be one of

none	No observations are highlighted
rejected	Rejected observations are highlighted
unused	Observations between rejected stations that are not explicitly rejected are highlighted.
std_residual	Observations with apriori standardised residual greater than *threshold* are highlighted
apost_std_residual	Observations with aposteriori standardised residual greater than *threshold* are highlighted
redundancy	Observations with redundancy less than *threshold* are highlighted
to_stations	Observations to or from highlighted stations are highlighted
between_stations	Observations between highlighted stations are highlighted

Error plotting options

error_type [apriori|aposteriori] standard_error
error_type [apriori|aposteriori] *conf* confidence

Defines how errors and standardised residuals will be represented by SNAPPLOT. The error can be apriori or aposteriori (the default is to leave unchanged), and standard errors or a percentage confidence limit. The percentage confidence limit applies only to error ellipses and height errors - it does not apply to the errors of observations or residuals written in the text window.

error_scale [horizontal|vertical] *exaggeration* [times_default]

Defines the scaling used for plotting error ellipses and height errors and the station adjustments. These are plotted at an exaggerated scale on the plot (e.g. 1000 times true size at true scale). The command can apply to just the horizontal or just the vertical errors. If neither horizontal nor vertical is specified it applies to both. The exaggeration can be expressed as a fixed value (e.g. 1000), or as a multiple of the default value determined by SNAPPLOT (e.g. 2.0 times_default).

Setting colours

key "*key item name*" [on|off] [*colour*]

This command specifies the settings for an item on the key. *key item name* is the name of the item on the key, and is enclosed in quotes. "*on*" or "*off*" specify that the item is selected or unselected, and *colour* specifies the colour for the item. *colour* can be one of black, white, blue, green, cyan, red, purple, yellow,

dark_blue, dark_green, dark_cyan, dark_red, dark_purple, brown, grey, or dark_grey.

SNAPPLOT tasks

Reviewing bad observations

One of the main uses of SNAPPLOT is to review the results of adjustments and identify possible problems in the data. Some facilities that you can use to help locate bad observations are:

- Listing all observations in the text window by selecting "Data | List" from the menu. The observations are listed in order from the largest to the smallest standardised residual so that the worst residuals are easy to locate. You can left-click or shift+left-click an observation to display detailed information in the text window and highlight the observation in the map window. Right-click on the text window to return to viewing the list.
- Colouring observations by their standardised residuals by selecting "Data | Colour by | Std residual" from the menu. This shows the geographic distribution of observations with large residuals. It is useful for identifying bad fixed stations coordinates and understanding how errors in one bad observation are distributed around neighbouring observations.
- Highlighting observations with significant residuals by selecting "Data | Highlight" from the menu. This is similar in effect to colouring by standardised residuals.

Understanding adjustments which don't converge

When a SNAP adjustment fails to converge it is often useful to look at how stations are being adjusted. To do this stop the adjustment after one or two iterations (see the SNAP user manual for information about how to do this), and review the results with SNAPPLOT. You can use the key to show the horizontal and vertical adjustments of stations. If a group of stations has a particularly large adjustment this may indicate a weakness in the adjustment configuration which can be resolved by fixing or floating one or more stations. The steps to do this are:

- add the command "maximum_iterations 1" to the SNAP command file
- rerun SNAP
- view the resulting binary file with SNAPPLOT
- use the key to turn on "Hor adjustment" and "Vrt adjustment", and turn off error ellipses and height errors. This will show the displacements to each station after one adjustment.

Assessing network configuration

SNAPPLOT provides some assistance for network design and assessing network strength. The main interests in network design are

- ensuring that the errors or relative errors of station coordinates meet the design criteria. SNAPPLOT allows error ellipses and height errors to be displayed, which helps identify areas where the criteria are not being met
- identifying observations with insufficient redundancy where the network needs to be strengthened. Where there is low redundancy the survey may be suffer from undetected gross errors. SNAPPLOT allows observations to be coloured according to their redundancy by selecting "Data | Colour by | Redundancy" from the menu. You can also select "Data | Highlight" to highlight observations with redundancy less than a specified amount.

- identifying observations with high redundancy. These observations may be removed from the observation scheme without significantly weakening the network. These observations may be identified by colouring data according to the redundancy.

Assisting designing a GPS survey network

SNAPPLOT can be used to visualize proposed GPS network designs by using it with data created by the SITE2GPS program (see page 3-6). It allows the various observing sessions to be easily highlighted and allows network configuration to be assessed. The steps to do this are:

- Create a coordinate file defining the approximate locations of the stations to be surveyed.
- Create an input file for SITE2GPS in which each line is a list of station codes for a GPS observing session
- Use SITE2GPS to create a SNAP data file of the proposed observations. In this data file the GPS observations are classified by "SESSION"
- Load the coordinate and data file into SNAPPLOT
- Select "Data | Colour by | 1 SESSION" from the menu
- To highlight specific observing sessions, shift+left-click on the key at the text for that session

Chapter Three: Coordinate file utilities

SNAP comes with several utilities for processing coordinate files. These are:

- SNGEOID — adds or removes geoid undulations from a coordinate file. The undulations are calculated from a binary data file which defines the undulations on a regular grid.
- SNAPCONV — changes the coordinate system of the coordinate file.
- DAT2SITE — calculates approximate coordinates of points using data files for input. This can either add stations to an existing coordinate file or create a new file.
- SITE2GPS — generates a GPS data file from a coordinate file. The data are calculated from the coordinates and so exactly fit them. The main use for the data file is in network design.

SNGEOID - add geoid undulations

Introduction

SNAP coordinate files can optionally contain information about the gravity field that SNAP can use in an adjustment. This information can include the geoid height (also called the separation or undulation) and the deflection of the vertical. (See the SNAP users manual for more information about the coordinate file format).

The height information for a station in a coordinate file can be one of

- orthometric height
- ellipsoidal height
- orthometric height and geoid height.

SNGEOID is a program to convert between these options. It calculates geoid heights from a gridded model stored in a binary file.

The geoid data file is called GEOID.BIN and is stored in the same directory as the SNGEOID program. If another file is to be used it can be defined by setting the environment variable GEOIDBIN using a command like

```
C:> SET GEOIDBIN=C:\DATA\MYGEOID.BIN
```

The format of the binary file is described below.

The binary file that is supplied with SNGEOID contains geoid undulations which are calculated from the Ohio State University global gravity model OSU91A. This is a spherical harmonic model to degree and order 360. The geoid undulations have been calculated from this on grid with 0.25 degree spacing in latitude and longitude. The error of this model is probably of the order of 1 metre.

Running SNGEOID

SNGEOID is a simple command line utility. The syntax for the command is

SNGEOID [-r] [-re] *station_file* [*new_file*]

In this command

station_file	is the name of the station coordinate file to be modified
new_file	is the name of the new file that will be created. If this is not specified then the new file will have the same name as the original, but with extension .NEW.

There are two switches that can be used to change the action of SNGEOID.

-r	Causes SNGEOID to remove the geoid separations and leave a file containing orthometric heights
-re	Causes SNGEOID to remove the geoid separations and leave a file containing ellipsoidal heights.

If neither of these switches is present then SNGEOID carries out its default actions, which is to calculate and add geoid separations to the coordinate file.

The geoid binary file format

The binary file contains a header, a parameter block, and a data block for each latitude in the grid. The header defines the location of the parameter block, and the parameter block defines the location of each data block.

The data in the file are stored in DOS binary formats for long integers (4 bytes), short integers (2 bytes), and double precision floating point numbers (8 bytes).

Geoid heights are stored as short integers. To convert these to heights in metres they are multiplied by a resolution defined in the file header. All the heights for a row in the grid (constant latitude) are stored in consecutively.

Text strings are stored as short integer defining the length of the string, followed by the string itself. The last character of the string should be a null byte (ASCII 0).

The format of the header block is

File id string	25 bytes	“SNAP geoid binary file<13><10><26>”, where <x> is a character with ASCII code x
Parameter block location	long	The offset into the file of the parameter block

The format of the parameter block is

First latitude	double	Latitude of first row in the grid
Last latitude	double	Latitude of last row in the grid
First longitude	double	Longitude of first element in a row
Last longitude	double	Longitude of last element in a row
Resolution	double	Conversion from grid values to geoid heights in metres
Row count	short	Number of rows in the grid
Column count	short	Number of columns in the grid
Description 1	string	First line of description

Description 2	string	Second line of description
Description 3	string	Third line of description
Coordsys	string	Coordinate system in which the undulations are calculated. This should match a coordinate system in the COORDSYS.DEF file. For the supplied binary file it is "WGS84".
Row locations	long	For each row in the grid defines the offset in the file of the start of the data block for the row. The row locations are stored as a consecutively in the parameter block.

SNAPCONV - change the coordinate system

SNAPCONV converts files from one coordinate system to another. The syntax for SNAPCONV is:

SNAPCONV *input_file new_code output_file*

where

input_file	is the name of the coordinate file to be converted
new_code	is the code for the coordinate system of the output file. This is one of the codes defined in the coordinate system definition file (see the SNAP users manual for more information)
output_file	is the name of the output file to be created

SNAPCONV will convert the coordinates and also the geoid information in the file (deflections and geoid height). Orthometric heights are not changed by SNAPCONV.

DAT2SITE - adds new stations to a coordinate file

Introduction

DAT2SITE is a program to add new stations to a coordinate file. The new stations are defined by observations in data files. The program is useful for calculating initial coordinates for an adjustment where they are not know in advance. For example it can be used to determine the coordinates of an eccentric station from the observations that define it.

The program takes as input an optional file of known stations coordinates and one or more data files. It then identifies all the station codes referenced in the data files which are not listed in the coordinate file. It progressively calculates coordinates for the unknown stations using observations to the known stations. It can locate stations using the following methods:

- calculating an offset position using GPS baseline vector
- calculating a position using a distance and azimuth from a known station (the azimuth may be defined by an angle observation relative to another known station)
- traversing by a series of distances and angles between two (or more) known stations
- calculating using azimuths from two or more known stations

- calculating using distances from three or more known stations (distances from just two station are ambiguous)
- calculating from an angle and distance to a known station and an angle to at least one other unknown station
- calculating by resection using angles to three or more known stations
- calculating the elevation using the height difference to a known station. The height difference may be calculated from a zenith distance if the distance (horizontal or slope) between the known and unknown station is known or can be inferred.

GPS data are used to calculate horizontal and vertical coordinates directly. When DAT2SITE uses terrestrial data it calculates either horizontal or vertical coordinates, but not both.

Vertical coordinates are calculated using height differences to vertically fixed stations. The height difference may be measured directly using levelling or may be derived from a zenith distance for two stations between which the distance is known (either measured or inferred from already fixed horizontal coordinates).

Where distance measurements are used to calculate horizontal coordinates the program uses horizontal or ellipsoidal distances in preference to slope distances. It will reduce slope distances to horizontal if it has information to do so. However if this information is not available it will treat a slope distance as if it were horizontal. Obviously this is not correct, but it may be adequate to calculate approximate coordinates for stations.

The main weakness of the program is that it does very little checking of the data. If the data files contain gross errors or incorrectly named stations then DAT2SITE will generate very inaccurate station coordinates. Also it ignores the errors assigned to observations and ignores instrument and target heights.

The coordinates generated by DAT2SITE are in no way correct. The program uses very simplistic algorithms for many calculations. For example some calculations are done using a projection without correcting the data to the projection. The reason for doing this is simply that the program is intended only to calculate approximate coordinates for input to a least squares adjustment. The coordinates generated should not be considered to have any value beyond that.

Running DAT2SITE

The program can be run in interactive mode or in batch mode. In interactive mode it can create a new coordinate file, whereas in batch mode it can only be used to add stations to an existing coordinate file.

Before running the program you must create the input files. These are the data files and the (optional) input coordinate file. If the program is running in batch mode then you can also use a SNAP command file to specify the names of the coordinate and data files. The formats of these files are described in the SNAP users manual. The only commands that DAT2SITE reads from a command file are the #coordinate_file and #data_file commands.

To run the program in batch mode use the command

DAT2SITE command_file_name

or

DAT2SITE -f coord_file_name data_file data_file ...

The program will read the data files, form a list of stations which are missing from the coordinate file, and attempt to calculate the locations of the stations. Those stations which can be located are added to the output coordinate file.

To run the program interactively, just enter the command

DAT2SITE

You will then be prompted for the input coordinate file and the names of the data files. If the coordinate file does not exist you are given the option of creating a new file. The program will then read the data files and build a list of stations that are missing from the coordinate file. It will locate as many as it can, then ask for the coordinates of the remaining stations. After you have entered the coordinates of one or more of these stations it will attempt to locate further stations based upon that input.

When the program has finished it program will generate a new version of the coordinate file with the same name as the input file, but with extension ".NEW".

It also creates a listing file. This has the same name as the command file, or the coordinate file if there is no command file, but with extension ".LIS". It contains a list showing each station that has been fixed and showing how it has been located. It also lists any outstanding stations which have not been located.

The following example shows the output generated while running the program.

```
C:\DATA\TEST> dat2site

DAT2SITE: Calculates trial coordinates using observational data


DAT2SITE requires a SNAP coordinate file and one or more SNAP data files


============================================================

Enter input coordinate file name: test.crd
File test.crd does not exist
Do you want to create a new coordinate file? Y/N: y

Enter the coordinate system code or ?: NZMG

============================================================

Enter the names of the data files

After the last file enter a blank line to start calculating coordinates

Enter the SNAP data file name: test.dat

Enter the SNAP data file name:

============================================================

The following stations have not been fixed
  GLTHL       R18         RB27        RB48        RF

Enter coordinates of one or more of these stations
DAT2SITE will then attempt to calculate the rest

     Enter station id (blank to quit): r18
     Enter station easting: 2494827.0
     Enter station northing: 5938210.0
     Enter station elevation: 103.24
     Is this all correct? Y/N: y

     Enter station id (blank to quit):

Fixing RF using GPS
Fixing RB48 using GPS
Fixing GLTHL using GPS
Fixing RB27 using GPS

============================================================

Updated coordinates written to test.crd

Log file written to test.LIS
```

SITE2GPS - create a GPS data file from a coordinate file

Introduction

SITE2GPS is a program to generate an artificial GPS data file from a coordinate file and an optional ASCII file defining the vectors to be included. The main uses to create input files to assist in network analysis and design, and for use with the BADLINES program (page **Error! Bookmark not defined.**) to check that a GPS survey has met the desired accuracy standards.

Input files

The SITE2GPS program requires two input files. One is a SNAP coordinate file defining the codes and coordinates of the survey stations (obviously these do not need to be accurate for network design). The other is an optional file listing all the vectors that are to be included in the output file listing all the vectors that are to be included in the output file.

The format for the vector list file is very simple. Each line in the file contains a list of the station codes that are to be used for an observing session. The codes should be separated by one or more spaces. SITE2GPS can either create a multistation GPS data file in which the stations from the session form a single multistation set, or it can create a set of baselines for each possible pair of stations in the session.

The vector list file can also contain data definition commands that are copied directly to the output data file. The most likely command to include is to define the errors of the GPS data. The syntax of this command is

#gps_enu_error ### ### ### mm ### ### ### ppm

where the ### are replaced with the east, north, and up components of the error in millimetres and part per million.

The following example shows a vector list file for SITE2GPS.

```
#gps_enu_error 20 20 40 mm 3 3 5 ppm
S1 S2 S3 S4 S5
S1 S2 S6 S7 S8
S1 S6 S9 S10
```

Running SITE2GPS

To run the program use the command

SITE2GPS [*options*] *crd_file vec_file dat_file*

In this command

crd_file	is the name of the input coordinate file
vec_file	is the name of the input vector list file
dat_file	is the name of the SNAP data file that will be created

The command can include the following options:

-a	Automatically connect every station to every other station. If this option is selected then the vector list file name (vec_file) should be omitted from the command. SITE2GPS will act as if the input file consists of just one line listing all the stations in the coordinate file.
-b	The output file will contain individual baselines instead of multistation GPS data.
-c#	Lines less # km will be omitted from the output file (applies only if baselines are chosen)

-f#	Lines greater then # km will be omitted from the output file (applies only if baselines are chosen)
-n#	Limits the number of stations that will be connected to a reference station. Only the nearest # stations will be used.

Chapter Four:
Post processing utilities

After doing an adjustment SNAP creates a binary file which summarises the results of the adjustment. This file is used by SNAPPLOT to display error information about the stations and observations. Two other programs have been written to extract additional information from binary file. These are

- LISTGPS Creates a table of information about GPS observations to facilitate generating survey plans
- BADLINES Identifies lines in a GPS survey which have exceeded a specification

LISTGPS - listing GPS observations

Introduction

The LISTGPS program is used to generate information required for submitting GPS data on survey plans. These plans may require information such as projection bearings and sea level distances. LISTGPS derives these quantities from the GPS observations and creates an ASCII data file. This file can then be used as input for other software (such as CAD or word processing programs). LISTGPS may also be used for listing other aspects of data from adjustments.

The format of the listing file generated is defined by a table format definition file. This is an ASCII file which defines the columns that will be printed in the output listing file. LISTGPS takes data from the binary file generated by SNAP and formats it into an output file using the definitions in the format definition file.

Running LISTGPS

Before running LISTGPS you must create a SNAP binary file (by doing a network adjustment) and a table format definition file. You then run the program using the command:

LISTGPS *binary_file* [*table_definition_file*] *output_file*

In this command *binary_file* is the name of the SNAP binary file, *table_definition_file* is the name of the file defining the output table format, and *output_file* is the name of the file that will be created. The square brackets [] denote that the table definition file name is optional - they are not part of the command line.

If the table definition file name is not given then LISTGPS assumes that the name is "LISTGPS". It also assumes the default extension ".TBF" for the file.

LISTGPS will look in the following places for the table definition file:

- the directory containing the binary file

- the users home directory (defined by the SNAPDIR environment variable)
- the directory in which the LISTGPS program is stored.

The table format definition file

The table definition file defines the format of the output file that LISTGPS will create. Essentially this file contains one or more table definitions, each of which defines several columns of data.

The format definition file is structured similarly to a SNAP command file. Each line in the file is a command consisting of one or more items separated by blank characters. The first item on the line is the name of the command. This may be followed by one or more parameters. For example the command may be "delimiter", which defines the text to be used between columns in the generated table, and the value may be "tab", meaning use a tab character. In the format definition file this is entered simply as

```
delimiter  tab
```

The format definition file can contain comment lines which are ignored by LISTGPS. These are lines starting with the exclamation mark "!".

Each output table is represented by a set of commands between a "table" and "end_table" command. The main commands used to define the table are the "column" command, which adds a column of output to the table and the "data" command which specifies the data that will be extracted into the table.

The overall structure of the file is thus

```
general commands
table
  table format commands
  column ...
  column ...
  ...
end_table
```

The following sections describe the formats of these commands

General commands

The following commands can occur outside a table definition:

text
text to be copied to the output file
end_text

The text .. end_text commands define text that is to be copied directly to the output file without modification. Any amount of text can be included between these commands. The end_text command must start in the first column of the command definition file.

angle_format *format_definition*

Defines the format used for angles (such as bearings) in the output file. Angles are output as degrees, minutes, and seconds. The *format_definition* defines the text that will occur after each component of the angle. It is a string formatted as

```
#ddd#mmm#sss
```

where *ddd* is the string printed after the degrees, *mmm* is the string printed after the minutes, and *sss* is the string printed after the seconds. The character # is any character that does not occur in the strings and is used to delimit the three strings.

Special characters can be included in the strings using the string format described below.

table
table definition commands
end_table

The table command prints a table in the output file using the format defined by the table definition commands.

Table definition commands

The following commands are used within a table definition to specify the layout of the table. The table definition must include a "data", "delimiter", and "column" command.

data ***data_source***

Specifies the data that will be printed in the table. Currently *data_source* must be "gps" (for gps observations) or "stations" (for the stations).

delimiter ***delimiter_string***

Defines the text that will be printed between each column of data. The delimiter string can be "tab", "comma", "blank", "none" or text expressed using the string format described below.

quote ***quote_string***

Defines the text that will be used on either side of quoted columns. (To specify that a column is quoted see the definition of the column command.) The quote string is defined in the same way as the delimiter_string.

coordsys ***code***

Defines the coordinate system used to represent coordinates in tables of station data.

column ***name options***

Adds a column to the table. *name* defines the data item that will be listed in the column, and the *options* provide additional formatting information. For GPS data *name* can be one of the following:

Name	Data item
from	First station code
to	Second station code
obs_ell_dist	"Observed" ellipsoidal distance
calc_ell_dist	Calculated ellipsoidal distance
ell_dist_err	Error in ellipsoidal distance
ppm_ell_dist_err	Error in ellipsoidal distance as part per million
rf_ell_dist_err	Error in ellipsoidal distance as a representative fraction
hor_vec_err	Length of the error in horizontal component of vector
ppm_hor_vec_err	Error in horizontal component of vector as ppm of horizontal length of vector
rf_hor_vec_err	Error in the horizontal component of the vector as a representative fraction of the length of the horizontal

Name	Data item
	vector.
obs_prj_brng	"Observed" projection bearing
calc_prj_brng	Calculated projection bearing
prj_brng_err	Error in projection bearing (in seconds)
ppm_prj_brng_err	Error in the projection bearing represented as part per million (or microradians)
rf_prj_brng_err	Error in the projection bearing as a representative fraction

For station data the name can be one of

code	The station code
name	The station name
northing	The station northing
easting	The station easting
height	The height of the station
h_max_error	The length of the major axis of the error ellipse
h_min_error	The length of the minor axis of the error ellipse
h_max_brng	The orientation of the major axis
change_east	The east component of the coordinate change
change_north	The east component of the coordinate change
change_up	The east component of the coordinate change

The column command can include options which affect the formatting of the column. An option is generally formatted as *option=value.* There should be no spaces in or between the option and the value. The valid options are:

width=###	Specifies the number of characters used to print the column. If the data exceeds this width then the column is expanded to accommodate it.
align=#	Defines the alignment of the data in the within the column. # can be one of "left", "centre", or "right"
ndp=#	Defines the number of decimal places that will be used for numeric columns
quote	Specifies that the field will be enclosed between quote characters (defined by the quote command).
prefix=###	Defines a string that will be printed in front of the field (outside the quote marks). The format of the string is defined below
suffix=###	Defines a string that will be printed after the field (outside the quote marks). The format of the string is defined below
header=###	Defines the text that will be written as a label at the top of the column. The header can be written on more than one line by including new line characters "\n" in the string. See below for more information about formatting strings.

Text strings in commands

Many of the commands and options define text strings that will be printed in the output file. These can include non-printing or special characters as well as normal text. When LISTGPS reads the string it interprets the backslash (\) and underscore (_) characters specially.

The underscore character is replaced by a blank character. This is used when a space character cannot be included in the string explicitly (for example in the prefix=### option of the column command).

The backslash character introduces an “escape sequence”. This is series of characters which represent another character which cannot be explicitly put into the command file. The escape sequences that LISTGPS recognises are:

\b prints a blank character (equivalent to underscore)

\n print a new line

\t print a tab character

\x## print the character with hexadecimal value ##. For example to include carriage return (0D hex) the string would be \x0D.

\\ print the backslash character

_ print the underscore character