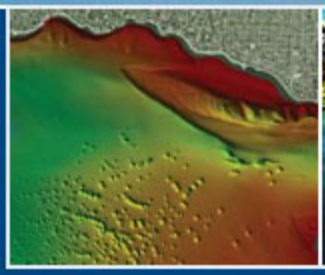
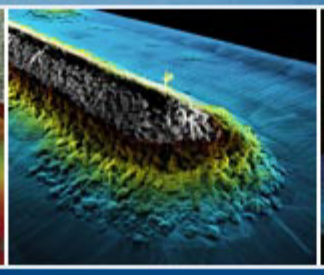
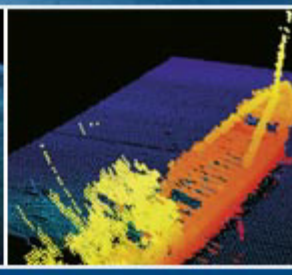
CARIS **HIPS and SIPS 9.1**

Editors







caris THE MARINE GIS EXPERTS

Acknowledgements

The development of HIPS and SIPS as a modern and powerful bathymetric and side scan sonar processing system was made possible through the cooperation of several organizations:

- The Canadian Hydrographic Service (CHS)
- The Ocean Mapping Group of the University of New Brunswick
- The National Oceanic and Atmospheric Administration (NOAA)
- University of New Hampshire

Data courtesy of the Shallow Survey Conference Common Data Set 2001 (CCOM - UNH).

Cover images (from left to right):

Burrard Inlet Pockmarks: image courtesy of Kalman Czotter - Canadian Hydrographic Service.
Burns Harbour North Breakwater, Indiana: image courtesy of Alexander Unrein, David Somers - Ocean Surveys, Inc.
Shipwreck of *GB Church*: EM3002 data provided by University of New Brunswick Ocean Mapping Group.

CARIS
(Global Headquarters)
115 Waggoners Lane
Fredericton, NB
Canada
E3B 2L4

Phone: 1 (506) 458-8533
(English/French/Spanish)
Fax: 1 (506) 459-3849
Email: info@caris.com
Assistance: support@caris.com
Web site: www.caris.com

CARIS USA
415 N. Alfred Street
Alexandria, VA
United States
22314

Phone: 1 (703) 299-9712
Fax: 1 (703) 299-9715
Email: info@caris.us

CARIS EMEA
Bremvallei 1
5237 LV 's-Hertogenbosch
The Netherlands

Phone: +31 (0)73 648 8888
Fax: +31 (0)73 648 8889
Email: sales@caris.nl
Assistance: support@caris.nl

CARIS Asia Pacific
PO Box 1580
Milton QLD 4064
AUSTRALIA

Phone: +61 (0) 7 3719 5132
Email: info@caris.com

April 2016

Trademarks owned by CARIS

This is a listing of USPTO-registered trademarks and trademarks owned by Universal Systems Ltd. doing business as CARIS ("CARIS") and might also be trademarks or registered trademarks in other countries. Please note that laws concerning use and marking of trademarks or product names vary by country. Consult a local attorney for additional guidance. CARIS permits the use of its trademarks and registered trademarks only where they are used in reference to CARIS and its products, the markings used are appropriate to the country or countries of publication, and CARIS is explicitly acknowledged as the owner of the mark. CARIS reserves the right to withdraw this permission at its sole discretion for any use it feels is inappropriate or adverse to its interests. CARIS otherwise prohibits the use of any of its registered symbols, insignia, or other identifying marks without express written approval. Violations are subject to liability for damages, injunctive relief, attorney's fees and other penalties.

Not all trademarks used by CARIS are listed in this document. Failure of a mark to appear on this page does not mean that CARIS does not use the mark nor does it mean that the product is not actively marketed or is not significant within its relevant market. The absence of a product or service name or logo from this list or the absence of a TM or TM Reg. USPTO notation against a product or phrase listed below does not constitute a waiver by CARIS of its trademark or other intellectual property rights concerning that name or logo.

The following are trademarks or USPTO-registered trademarks of CARIS:

- Article 76 Module
- Bathy DataBASE
- Bathy DataBASE Server
- BASE Editor
- BASE Manager
- BDB
- CARIS
- CARIS GIS
- CARIS Notebook
- CARIS Onboard
- ChartServer
- CPD
- Core Production Database
- Easy View
- EAM
- Engineering Analysis Module
- HIPS
- HPD
- HPD Server
- Hydrographic Production Database
- Limits and Boundaries Module
- LIN
- LOTS
- LOTS Browser
- LOTS Limits and Boundaries
- LOTS Article 76
- One Feature, One Time
- Paper Chart Composer
- Paper Chart Editor
- Ping-to-Chart
- Product Editor
- Publications Module
- S-57 Composer
- SIPS
- Source Editor
- Spatial Fusion
- Spatial Fusion Enterprise

Those trademarks followed by or footnoted as TM Reg. USPTO later in this document are registered trademarks of CARIS in the United States; those followed by or footnoted as TM Reg. CIPO are registered trademarks of CARIS in Canada; those followed by or footnoted as either TM Reg. USPTO and CIPO or TM Reg. USPTO, CIPO are registered trademarks of CARIS in both the United States and Canada; those followed by or footnoted as TM are trademarks or common law marks of CARIS in Canada and the United States, and in other countries.

The trademarks and names of other companies and products mentioned herein are the property of their respective owners.

Copyright owned by CARIS

Preface

This guide contains information about the following CARIS[1] HIPS[2] and SIPS[3] editors. It describes the parts components and function of each editor interface.

The role of each of these editors in the HIPS and SIPS workflow is described in the User Guide.

- "Attitude Editor" on page 27
- "Process LIDAR Data" on page 45
- "Navigation Editor" on page 61
- "Side Scan Editor" on page 65
- "Process Single Beam Data" on page 109
- "Subset Editor" on page 127
- "SVP Editor" on page 147
- "Swath Editor" on page 159
- "Tide Editor" on page 177
- "Vessel Editor" on page 187

The HIPS and SIPS workflow, interface elements and tools are described in these documents:

HIPS and SIPS User Guide

HIPS and SIPS Reference

HIPS and SIPS Tools

1. This term is a trademark of CARIS (Universal Systems Ltd.), Reg. USPTO & CIPO.
2. This term is a trademark of CARIS (Universal Systems Ltd.), Reg. USPTO & CIPO.
3. This term is a trademark of CARIS (Universal Systems Ltd.), Reg. USPTO & CIPO.

Table of Contents

1

ASCII Info File Editor

Use the ASCII Info File Editor to create and edit format information files that contain the position and attribute information for an ASCII dataset being imported. These files are used to map the text from an ASCII file to attribute values for a surface.

In this chapter...

Format Info Files

Format information files are XML files that contain the position and attribute information for an ASCII dataset being imported into HIPS. The file extension for format information files can be either XML or INFO.

These files are used to map the text in an ASCII file to attribute values for the surface being created. For example, the first three attributes defined in the format information file generally reference the latitude, longitude and Z values for each point in the dataset.

The information provided in a format information file varies greatly depending on the task at hand and the data available. Some files may simply provide the positional attributes, while others may define additional attributes available in the source dataset, such as Uncertainty or Temperature.

Sample files

Three sample format information files are provided with CARIS applications:

- asciinavigation++formatinformationfile.info
- asciinavigationformatinformationfile.info
- xyzformatinformationfile.info

These files can be found in the *Template* directory:
`...\CARIS\HIPS\<version>\Template`

The sample files can be used as a template that you customize to suit your data or you can create new format information files. Although a text editor could be used to review format information files, using the tools in the ASCII Info File Editor can simplify the process of editing and creating files.

Types of files

Two different types of format information files can be created:

- *Elevation*: This type should be used if opening or importing a file containing XYZ data.
- *Navigation*: This type should be used if creating a file for data containing location and time values. Navigation files are meant to be used in CARIS HIPS & SIPS only.

File formats

In addition to the file *type*, there is also an option for the file *format*. There are two different formats available:

- *Fixed*: A fixed format information file is used if the source data file has no delimiter between the attribute values. In this format, data is mapped according to the “Start” and “Width” values defined for each attribute column in the file.

- *Delimited*: A delimited format information file is used if the source data file uses a delimiter character to separate the attribute columns. With this method, the delimiter character identifies where attribute values start and stop.

The following is an example of an elevation type information file in the delimited format:

```
<?xml version="1.0" encoding="UTF-8"?>
<cif:FormatInformation xmlns:cif="http://www.caris.com/cif/2.1" version="2.1">
  <cif:SkipLines>2</cif:SkipLines>
  <cif:Type>Elevation</cif:Type>
  <cif:Delimited>
    <cif:Delimiter>,</cif:Delimiter>
    <cif:Column>
      <cif:AttributeName>X</cif:AttributeName>
      <cif:Multiplier>1.00</cif:Multiplier>
      <cif:CoordinateFormat>Ground</cif:CoordinateFormat>
      <cif:DataType>DOUBLE</cif:DataType>
      <cif:Index>0</cif:Index>
    </cif:Column>
    <cif:Column>
      <cif:AttributeName>Y</cif:AttributeName>
      <cif:Multiplier>1.00</cif:Multiplier>
      <cif:CoordinateFormat>Ground</cif:CoordinateFormat>
      <cif:DataType>DOUBLE</cif:DataType>
      <cif:Index>1</cif:Index>
    </cif:Column>
    <cif:Column>
      <cif:AttributeName>Z</cif:AttributeName>
      <cif:Multiplier>1.00</cif:Multiplier>
      <cif:Orientation>up</cif:Orientation>
      <cif:Unit>m</cif:Unit>
      <cif:DataType>DOUBLE</cif:DataType>
      <cif:Index>2</cif:Index>
    </cif:Column>
  </cif:Delimited>
</cif:FormatInformation>
```

Number of lines in header
Format Information File Type
Format Information File Format
Delimiter Value = Comma
Longitude (X) Column
Column Name
Column Index
Multiplier
Axis Direction
Data Type

Each of the fields highlighted in the example can be defined using the *Properties* fields in the ASCII Info File Editor.

See also "INFO FILE" ON PAGE 686 of the CARIS HIPS and SIPS Reference.

ASCII Info File Editor

Use the ASCII Info File Editor to create and edit format information files that contain the position and attribute information for an ASCII dataset being imported.

Menu	Tools > Editors > ASCII Info File

To open the ASCII Info File Editor:

1. Select the ASCII Info File Editor command.

The ASCII Info File Editor displays these windows:

- *File items:* displays a tree of the attributes in the data source. This is used to select a column to modify its properties and to add/remove columns in the file. See "File items" on page 15 for descriptions.
- *Properties:* fields which are used to view and modify the settings for the column selected in the *File items* list. See descriptions of these fields "General" on page 15
- *Show ASCII preview:* which displays lines of data from the source file to be imported, similar to viewing the file in a text editor. See "ASCII Preview" on page 17.

The editor also contains Tool buttons:

	New: Create a new information file.
	Open: Open an existing format information file.
	Save: Save any changes to the currently open file.
	Save As: Save the current settings to a new file:
	(Add column): Add an attribute column to the list in the File Items list.
	(Remove column): Remove the currently selected attribute column from the File Items list.

When the Editor is launched, all fields will be empty and the ASCII Preview window is hidden.

"Create a Format Information File" on page 19

"Edit a Format Information File" on page 25

"Format Info Files" on page 12

File items

Items in the file list represent the columns of the data in the info file.

Column name	Description
X	Latitude coordinates
Y	Longitude coordinates.
Z	Depth
DatagramTime	• Defines the format for time, as • DATE_YEAR • DATE_MONTH • DATE_DAY • TIME_HOUR • TIME_MINUTE • TIME_SECOND
STDLAT	Standard deviation for latitude of GPS position.
STDLON	Standard deviation for longitude of GPS position.
NUMSATS	Number of GPS satellites used to derive GPS position.
HDOP	Horizontal Dilution of Precision quality for GPS position.
VDOP	Vertical Dilution of Precision quality for GPS position.
PDOP	Position Dilution of Precision quality for GPS position.
GPSHEIGHT	Elevation/Z component of GPS position.

Properties

Depending on what File item you have selected, various Properties will be displayed in these areas: General, Subset and Ignored Values.

General

Properties of the attribute column currently selected in the File Items control panel. The following properties are available:

Field Name	Description
Skipped Lines	The number of lines at the beginning of an ASCII file that to be skipped. (Available when the file name is selected it the panel.)
Delimiter	Specify the character that will separate each column. (Available when the file name is selected it the panel.)

Column Name	The name to assign to the attribute column. This name will be assigned to the attribute values imported using this information file, as well as the attribute layer in the Layers window.
Column Index	The index number assigned to the selected attribute column in the source file. Using the index number specified in the format information file, an import tool assigns the values in the column to the appropriate attribute for each line. This property is only available for delimited format information files.
Column Multiplier	The multiplier value to apply during an import to scale the data. Any number is accepted for this value and a different scale can be specified for each column. If no scaling is required, enter 1.0.
Data Type	The type of data in the selected attribute column. This value cannot be changed for the default X, Y, and Z columns.
Coordinate Format	The format of the coordinates for the file.
Start	The number of the first column character in the attribute value being defined. The characters in a row are counted starting at 0.
Width	The number of column characters included in the values of the currently selected attribute.
Data Type	The type of data in the selected attribute column.
Axis Direction	The Z-axis convention of the data.
Unit Type	The type of measurement values in the data.
Unit	The unit of measure of the data.

Subset

Use these properties to identify subset values within a single attribute column.

Field Name	Description
Subset	Set to True to enable the Subset properties.
Subset Start	The start value of the subset value within the attribute value. Subset values always start counting at the first of the attribute value instead of the first of the row.
Subset Width:	The number of characters in the subset value.

Ignored Values

Use this property to indicate values that are to be ignored by an import tool if the data contains invalid values, for example, points without elevation values. When an import tool encounters

an ignored value, it will skip that point and move on to the next one.

Any number or string can be entered for this property.

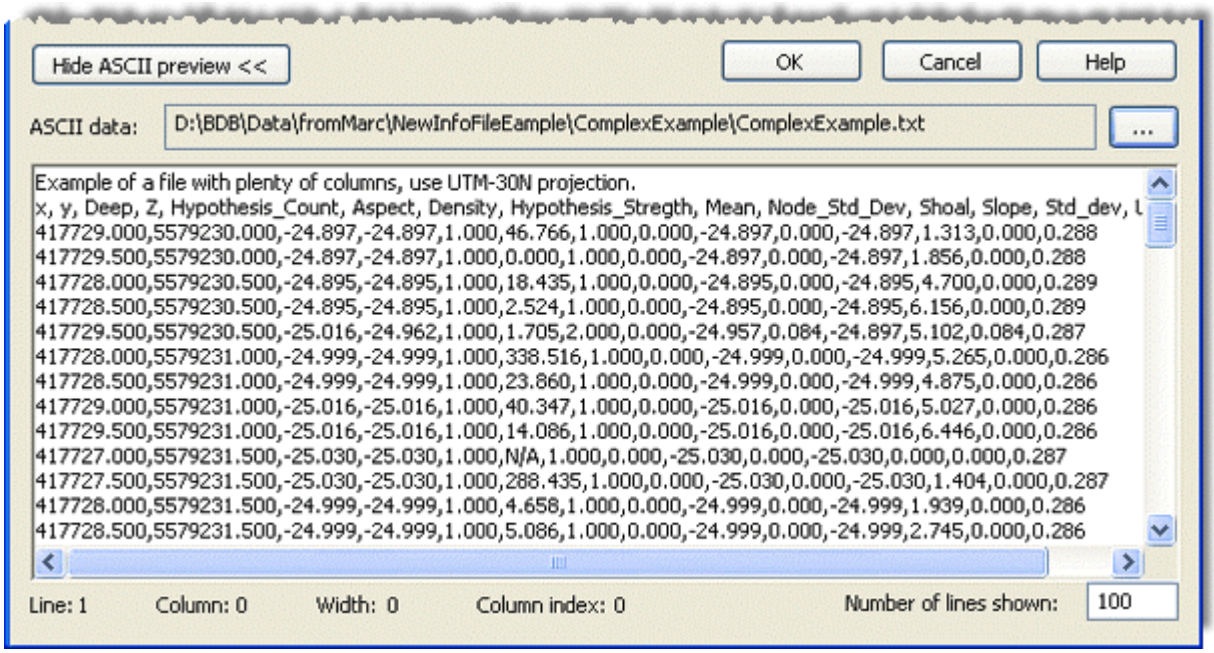
ASCII Preview

Because the format information file must match the contents of the ASCII file, it is recommended that you have the source data displayed in the preview when defining properties.

To view an ASCII source file:

1. Click **Show ASCII preview** to expand the ASCII preview
2. Click the **Browse** button (**...**) in the *ASCII data* field to navigate to the ASCII file.

The first 100 lines of the file will be displayed by default. Use the scroll bar to see all the lines in the preview.



To display a smaller set of lines from a large file:

3. Change the value in the *Number of lines shown* field.

Other fields at the bottom of the editor provide information about the data at the current location of the cursor within the data.

- *Line*: This field numerically identifies which row is selected in the file. Each line represents a point in the data. Line numbering starts at 1.
- *Column*: This field numerically identifies which character column of data is selected in the line. Every character in a line is considered a column and is numbered starting at 0.

- *Width*: This field identifies the number of characters currently selected in the preview. The width value is then used to identify the number of characters to include in an attribute value if no delimiter is used.
- *Column index*: This field identifies the column index number of the currently selected data. Each attribute in the source data is assigned to a column and indexed with a number starting at 0. This column contains all characters that make up the values for that attribute. This field is only available for delimited format information files.

Create a Format Information File

Two sample format information files are located in `C:\Program Files\CARIS\HIPS\<version>\Template`. For the structure of an info file, see "INFO FILE" ON PAGE 686.

Menu	File > ASCII Info File Editor
Tool	

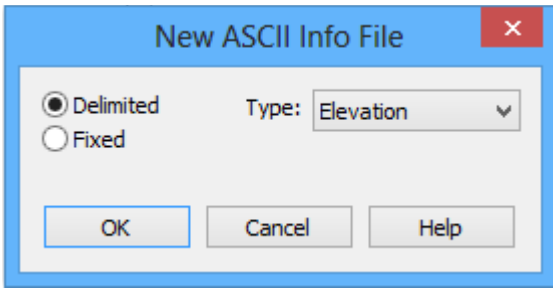
To create a new format information file in the Editor:

1. Open the ASCII Info File Editor.

In the ASCII Info File Editor:

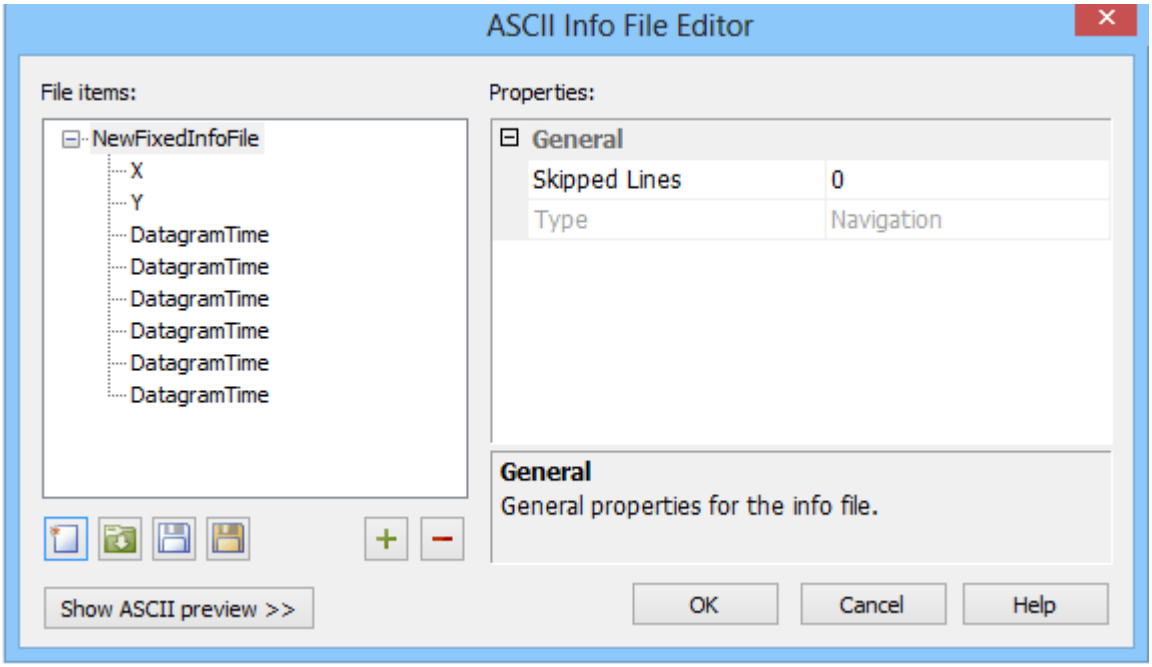
2. Click the **New format information file** button. 

The New ASCII Info File dialog box is displayed.



3. Select either Delimited or Fixed as the format for the file. See "FILE FORMATS" ON PAGE 12 for the characteristics of each.
4. Select the *Type* of file from the list. See "TYPES OF FILES" ON PAGE 12
5. Click **OK**.

The *File items* control will be populated with the default column entries for the selected file type.



- For an elevation file, this includes X, Y and Z. These columns cannot be removed or renamed.
- For a navigation file, this includes X, Y and six `DatagramTime` entries. The X and Y columns cannot be removed or renamed.

  The `DatagramTime` columns can be removed and/or renamed, as long as one time entry is left unchanged.

6. Define the properties of each column as needed. See
7. [Optional] Use the Remove button  to remove a selected column from the display.
8. [Optional] Add additional columns and attributes.

   "ADD ADDITIONAL COLUMNS" ON PAGE 20

   "ADDITIONAL COLUMN ATTRIBUTES" ON PAGE 21

9. Click **OK** to name and save the new file.

Add additional columns

For each additional column attribute included in the source data, an additional item must be added to the *File items* list. To add additional items:

10. Click the **Add** button  .

A new item, labelled `NewColumn`, is added to the tree structure and displayed in the *Column Name* field in the Properties list. The name can be changed to something more specific either:

- by selecting an option from the *Column Name* drop-down list, or,
- by manually entering a name in the field.

The drop-down list contains default column names for each of the available file types. For a navigation file, "`DatagramTime`" is provided. For an elevation file, you can choose between "DEPTH_TPU" and "POS_TPU", both of which are meant for uncertainty data.

If a name is selected from the drop-down list, the remainder of the Properties fields will either be populated with default values or be disabled, as they are defined automatically by the selection.

11. Repeat these steps for each column you want to add to the format information file.

Additional column attributes

Additional attributes are:

DatagramTime	• Defines the format for time, as • DATE_YEAR • DATE_MONTH • DATE_DAY • TIME_HOUR • TIME_MINUTE • TIME_SECOND
STDLAT	Standard deviation for latitude of GPS position.
STDLON	Standard deviation for longitude of GPS position.
NUMSATS	Number of GPS satellites used to derive a GPS position.
HDOP	Horizontal Dilution of Precision quality for GPS position.
VDOP	Vertical Dilution of Precision quality for GPS position.
PDOP	Position Dilution of Precision quality for GPS position.
GPSHEIGHT	Elevation/Z component of GPS position.

Next, define the properties for each attribute column represented in the file.

12. Select an item in the tree structure of the *File items* control.

Properties column

The *Properties* control will be populated with the properties for the selected column.

These properties and their values will differ based on the item selected. If the root of the file tree is selected, the properties for the file itself are displayed, e.g., the number of skipped lines, if any, the delimiter used, and the type.

If a column item is selected, the properties for the selected column are displayed. As you select each property field, a brief description is provided at the bottom of the control.

These properties are also described briefly in the following table.

Level	Property	Description
File Properties	Skipped Lines	This property is used to identify the number of lines in the header of the source data file. These lines will be skipped when reading the data to be imported. This value must be an integer.
	Delimiter	This property is used to identify the delimiter character that separates the attribute columns in the source data file. The delimiter value can be selected from the drop-down list, or a single character of the user's choice can be typed in the field (“,” , “.”, “|”, etc.). This property is only available for delimited format information files. If no delimiter is present, the “Start” and “Width” properties must be used to identify the boundaries of attribute columns.
	Type	This property identifies the type of the format information file (Elevation or Navigation). This property is for information purposes only and cannot be changed.
Column Item Properties	Column Name	This property assigns a name to the selected column. This name will function as the attribute name for the data in the column when imported to a surface. The surface layer containing the data from the column will also be named according to the value entered here. Column names can be defined as any text string, or selected from the drop-down list. Names in the drop-down list cannot be changed, nor can the default names for the X, Y, and Z columns.
	Column Index	This property is used to assign a column index number to the selected attribute column in the source file. Using the column number specified in the format information file, an import tool assigns the values in the column to the appropriate attribute for each line. This property is only available for delimited format information files.
	Column Multiplier	This property is used to scale the data during import. Any number is accepted for this value and a different scale can be specified for each column. If no scaling is required, enter *1.0*.

Level	Property	Description
Column Item Properties (cont.)	Data Type	This property identifies the type of data in the selected attribute column. This property cannot be changed for default X, Y or Z columns. The following data-types are available from the drop-down list: • FLOAT • DOUBLE • SHOR • USHORT • INT • UINT • UCHAR • TIME_HOUR • TIME_MINUTE • TIME_SECOND • DATE_YEAR • DATE_MONTH • DATE_DAY • STRING • COLOUR_RED • COLOUR_GREEN • COLOUR_BLUE • COLOUR_ALPHA
	Coordinate Format	The format of the coordinates for the file. This property is only available for the X and Y columns. The options include: • DecimalDegrees • DegreesDecimalMinutes • DegreesMinutesSeconds • Ground
	Start/Width	These properties are used to identify the data boundaries for each attribute column when a delimiter is not used. These properties are populated with the character column count for the characters in each line. The Start value would be the column number of the first character in the attribute value. The Width is the number of characters that make up the value. The first character in a line always has a Start value of 0. If the first attribute value contained 11 characters, then it would have a Width value of 11. The Start value of the next attribute would then be the next consecutive number in the line, which in this case would be 11 (if all characters in the value are to be included during import). This numbering is continued across all character columns in the line. These properties are only available for Fixed format information files. See “INFO FILE” ON PAGE 686 for more information on fixed data.
	Axis Direction	The convention for the positive Z-axis.
	Unit Type	The type of measurement values in the data. This field is used solely to filter the available entries in the *Unit* property; it is not actually used in the format information file.
	Unit	The unit of measure of the data. The options in this field can differ based on the selected *Unit Type*.

Level	Property	Description
Column Item Properties (cont.)	Subset	This is a set of properties used to define settings for an attribute in delimited data that contains sub-values, such as dates, times, colours, etc. When using subsets, each sub-value has a separate column index number and a different column name. The following properties are available for subsets:
		Subset: This property is used to indicate whether the selected attribute has sub-values or not. The property is defined as *True* if the entry represents a subset of a delimited field.
		Subset Start/Subset Width: These properties are used to define the Start and Width values of each sub-value in the selected column. The count for sub-values is similar to the count for a line in fixed data, except that the count starts with the first character in the attribute value, which is always 0. The Width value is the number of characters in the sub-value. These fields are only enabled if the *Subset* property is *True.* If working with delimited data, but the sub-values are not delimited, the same **Column Index** must be used for each sub-value. If the sub-values are delimited, there is no need to use a subset - each sub-value can be entered normally.
	Ignored Values	This property is used to indicate values that are to be ignored by an import tool if the data contains invalid values (e.g., points without elevation values). When an import tool encounters an ignored value, it will skip that point and move on to the next one. Any number or string can be entered for this property. Populating this field will add another empty “Ignore” field underneath it, allowing multiple values to be entered. If the source data has multiple representations of an invalid value (such as 0, 000 and 0.0), each representation must be reflected in the format information file, as the values are treated as strings.

Edit a Format Information File

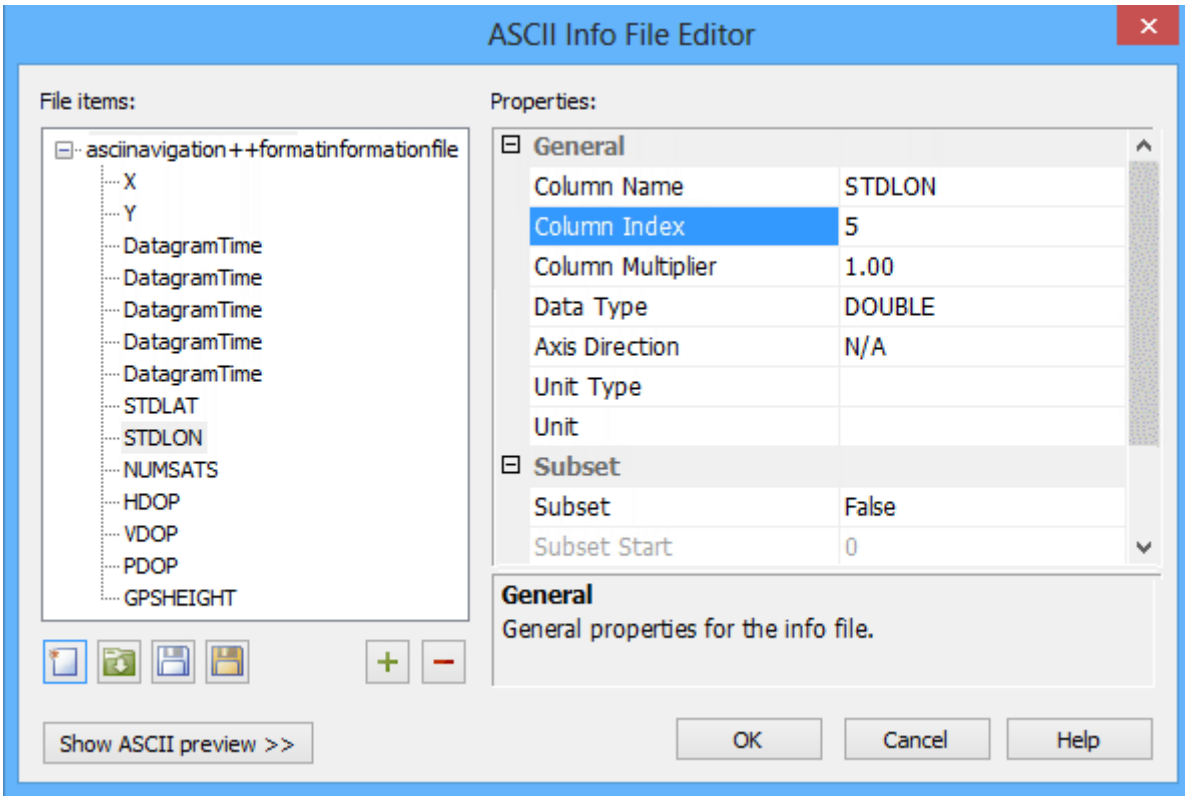
Menu	File > ASCII Info File Editor
Tool	

Two sample format information files are located in `C:\Program Files\CARIS\HIPS\<version>\Template`. For the structure of an info file, see "INFO FILE" ON PAGE 686.

To edit a format information file:

1. Launch the ASCII Info File Editor.
2. Click the **Open** button  and select a format information file to edit.

The Editor will display the columns and values for the selected file.



3. Select a column attribute from the *File items* list.
4. [Optional] Add or remove columns included in the *File items* list:
   - Use the **Add** button to add new items to the list.
   - Use the **Remove** button to remove selected items from the list.

The *Properties* control will be updated with values for the selected column.

5. Adjust the properties for the column.
6. Repeat steps 6 and 7 for each column as needed.

To see the contents of the ASCII file:

7. Click **Show ASCII preview**, and
8. Click the *ASCII data* Browse button (**...**) and navigate to open the source data for the format information file.

When all changes have been completed:

9. Click **Save** to save the changes to the open file.
   OR
   Click **Save As** and define a name and location to save the changes to a new file.

2

Attitude Editor

Attitude Editor displays sensor data related to the movement of a vessel or towed sensor. It also displays data from other auxiliary sensors that use a simple time/value data model. The data is plotted in chronological order and is time-stamped. You can edit data directly in the editor or set filter parameters to reject unwanted data.

Data for multiple lines can be viewed and edited. More than one type of sensor data can be displayed in the same graph window.

In this chapter...

Attitude Editor Interface

Attitude Editor consists of the Attitude Editor control window and multiple synchronized sensor graph windows.

Attitude Editor can be opened while Swath Editor is active so that attitude and sounding data can be examined at the same time.

Attitude Editor can be opened with more than one line selected. Lines can be selected from the Display window or from the list in the Project window (Project data tree).

Menu	Tools > Attitude Editor
Tool	

To open Attitude Editor:

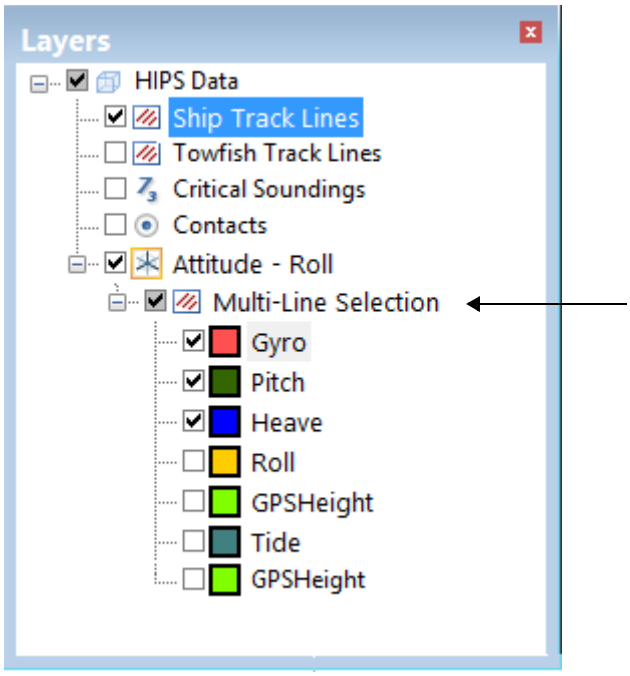
1. Select a track line or lines.
2. Select the Attitude Editor command.

The Attitude Editor control panel and graph windows for the default sensors are displayed.

"SENSOR GRAPH WINDOWS" ON PAGE 31

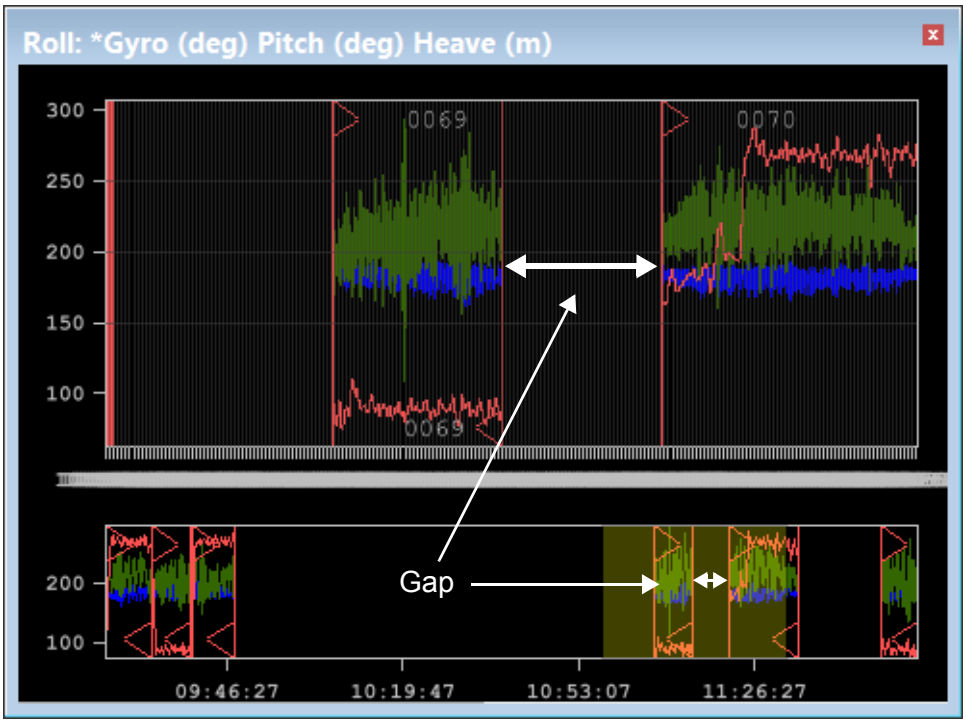
Display Data for Multiple Lines

The data from multiple lines can be edited in Attitude Editor. When multiple lines are selected this is indicated in the Layers window.



Overview of multiple lines

The line data is displayed in the graph, coloured using the settings for the sensors. The Overview graph shows a down-sampled graph of all of the lines as illustrated below.



Disparities in time are displayed as gaps in the graph lines.

Lines that are selected in the Display window when the Attitude Editor is active, will be coloured with the Selection colour set in Tools > Options > Display > Sensor Editors. See "SENSOR EDITORS" ON PAGE 655 to change this selection colour.

Move from line to line

In the Project window, any line that has a check mark is open in Attitude Editor. To move from one line to the next:

1. Use the Select > Track Line > Next (or Previous) command, or,
2. Click on a specific line in the Project window to display the data for that line.

Editing can be applied to any line open in the Editor, using the Accept, Reject With and Reject Break interpolation commands on the right-click menu.

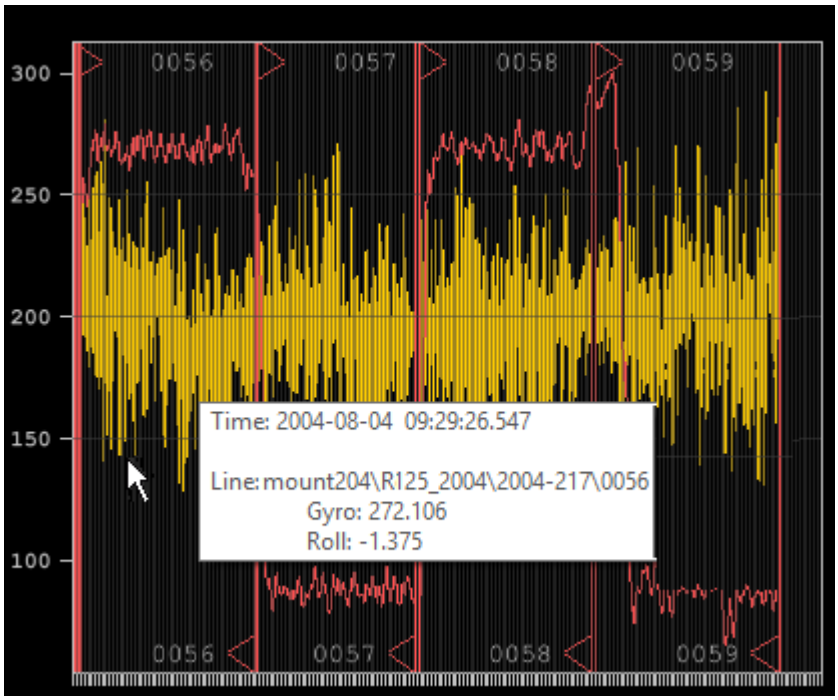
Filter and Smoothing options are disabled when in multi-line mode.

- Under Current View, The Filter / Smooth line, Difference and Threshold fileds are greyed out.

- All Data options are disabled.

Tooltips

When multiple lines are displayed in Attitude Editor, tooltips in a graph will display time of data, line identification in PVDL format as well as the active sensors, as in the example below.

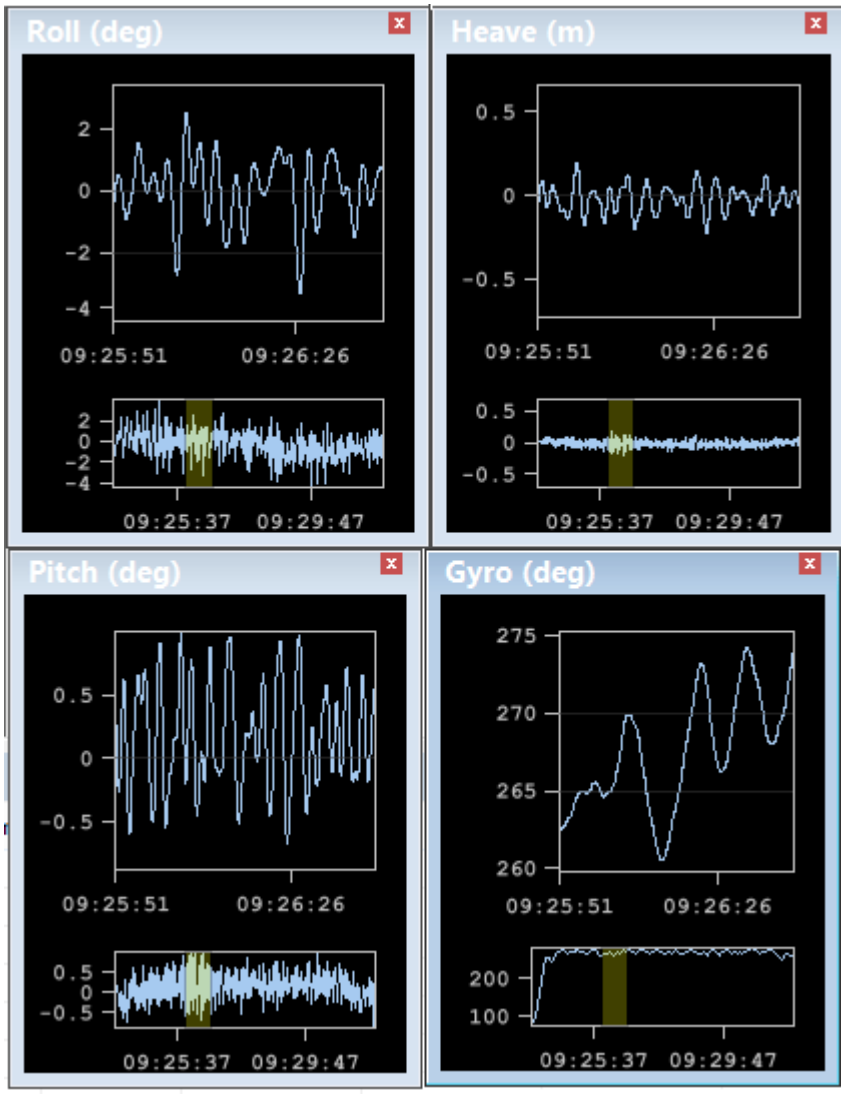


See "DISPLAY OPTIONS" ON PAGE 35 to turn on the tooltip option.

Sensor graph windows

Data is graphed in the sensor windows on X-Y axes. In the default layout, the X-axis represents time and the Y-axis represents data.

By default, data for Gyro, Heave, Pitch and Roll sensors are displayed.



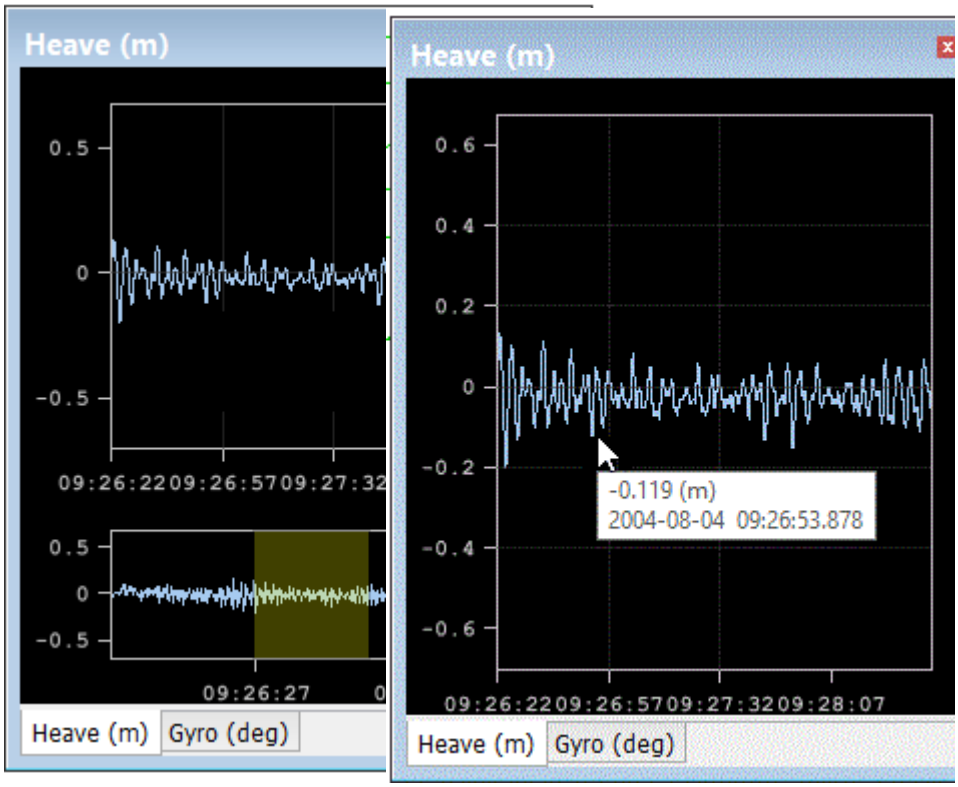
Each sensor window contains two graphs. The graph in the top of the window shows time series data for the sensor. The graph at the bottom of the window is a down-sampled view of the time series for the entire selected line.

This overview graph can be useful for locating data outliers and gaps. It can be displayed or hidden for each individual graph.

To hide the overview graph:

1. Right-click on the graph window, and select Hide Overview from the pop-up menu.

The overview is hidden, as in the example below.



Overview Slice

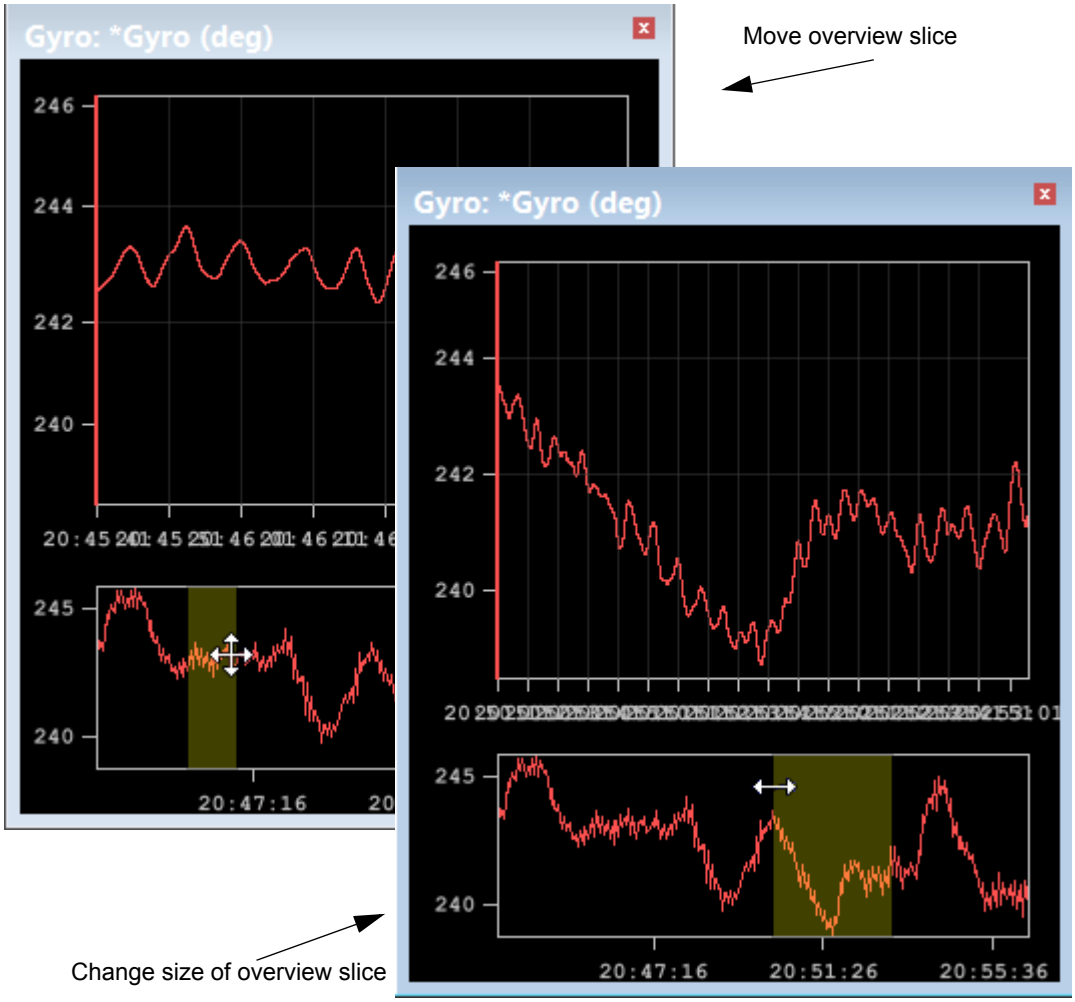
The overview graph contains a display slice that can be used to control the amount of data displayed and its position along the graph line.

To move along the graph line:

1. Click in the overview slice and drag it right or left to the new position.

As the slice moves, so does the related data. All the open sensor graphs are synchronized so moving the slice in one moves along and highlights the data in all the other graphs.

Resizing the slice control changes how much of the data is shown in the graph window. The wider the slice is, the denser the data is in the display. Making the slice narrower spreads the data out, as in the example below.



To resize the overview slice:

1. Hover over the left or right edge of the slice, until the cursor changes to an arrow.
2. Drag the slice to make it larger or smaller.
3. Alternatively, increase or decrease the value in the Time Period box on the Attitude Editor control panel.

The graph display and the size of the slice will be changed in all the synchronized graph windows. As well the Time Period value will be adjusted.

Display sensor data

All sensor data in HIPS/SIPS format can be displayed in the Attitude Editor graphs except for depths, navigation, and side scan data.

The default graph windows display:

- **Gyro:** Gyro observations are displayed in degrees from positive 0 to 360.
- **Heave:** Heave observations are displayed according to the Elevation > Depth Units set in the Options > Display Units dialog box and are positive as the vessel moves upward.
- **Pitch:** Pitch observations are displayed in degrees and are positive when the bow is down.
- **Roll**: Roll observations are displayed in degrees and are positive when the starboard side is up.

The following sensors can also be displayed if data is available for them:

- surface sound speed profile (SSP)
- tide (loaded or converted)
- GPS tide
- GPS height
- delta draft
- speed on water (SOW)
- side scan gyro
- side scan depth
- side scan height
- side scan cable out
- delayed heave
- RMS data, loaded using Load Auxiliary Data

However, if the survey data does not contain any data from a selected sensor, the graph window will report that there is no data to display.

Menu	Tools > Editors > Attitude > (list of sensors)
Pop-up	Window (Layers window)

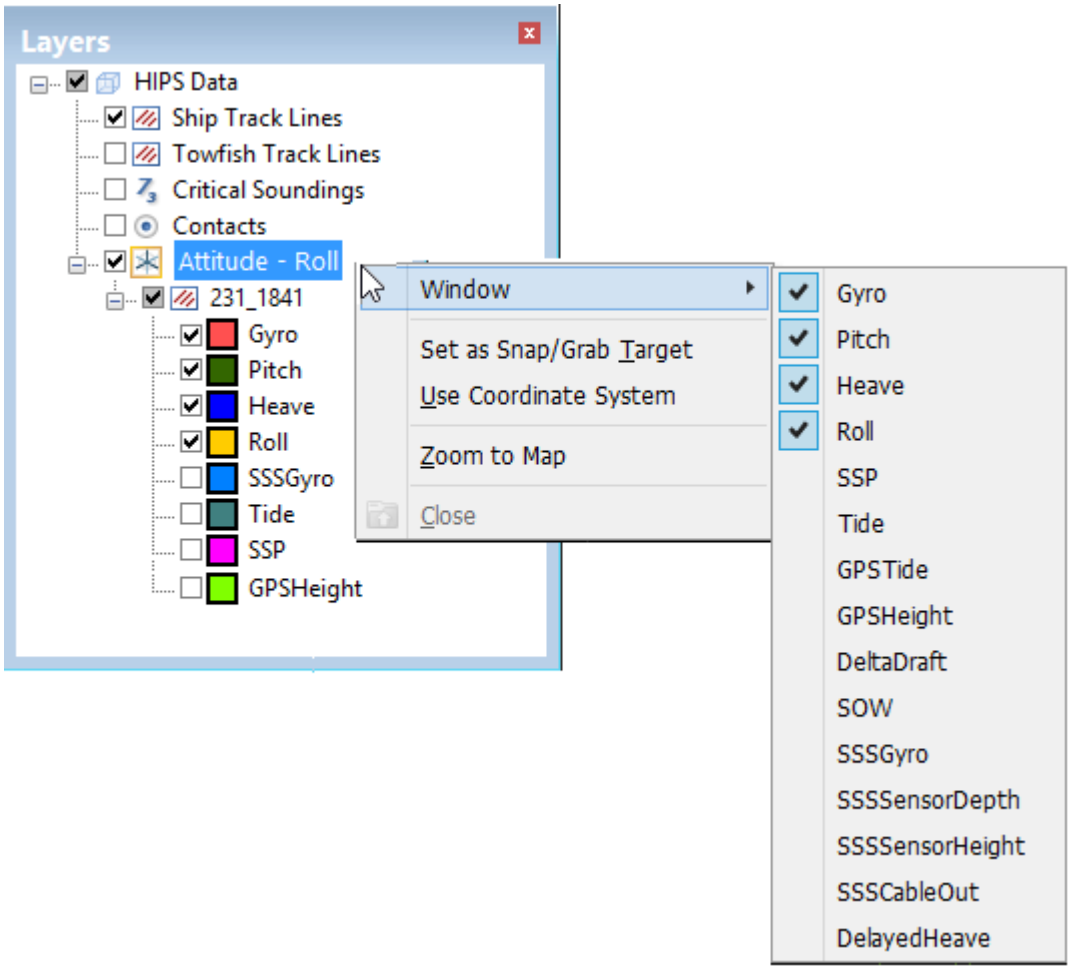
To display a graph window for a sensor other than the default sensors:

1. Select the sensor by name from the Attitude Editor list.

Sensor windows can also be turned on from the Layers window.

2. Right-click on the top Attitude layer, and select Window from the pop-up menu.

3. Select the name of the sensor window to display.



The data for each selected sensor will be graphed in its own window.

Display Options

The options for display, for example, of labels on scale lines and tooltips showing data values, can be set for the graph windows using the *Sensor Editors* settings in Tools > Options > Display. See also "SENSOR EDITORS" ON PAGE 655.

Menu	Tools > Options > Display > Sensor Editors

1. Select the Options command to display the Options dialog box.
2. Click the Display tab and select the *Sensor Editors* category so it is highlighted.
3. Select the *Labels* check box to display horizontal and vertical scale lines and annotations. Select a colour for the labels and scale lines.
4. Set colours for background, selected and superselected data, port and starboard data, etc., or create custom colours from the Windows colour picker.

5. Select the *Show sensor tool tips* check box to display data values under your cursor.
6. Select the *Automatic recentre* check box to automatically redraw the Display window so that the highlighted portion of the line is always displayed in the centre when scrolling through data in the graph windows.
7. Click **Apply** to make the change, or click **OK** to apply the change and exit the Options dialog box.

Display Multiple Data Types

More than one type of data can be displayed in the same graph window. Each data type is coloured individually, to identify it in the Layers window and in the sensor graph. These colours can be changed in the Layers properties.

When multiple data types are displayed in a graph window, the tooltip box will display the values for all the data types in the window. See "Display Options" on page 35.

The settings for each graph window are remembered. Data can be displayed using either a overplotted or fixed vertical scale.

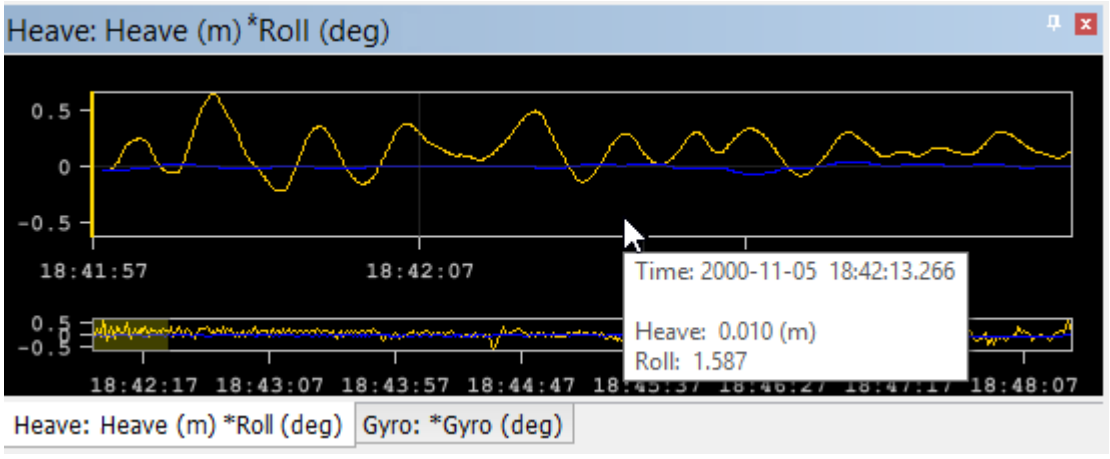
Active Data Type

To change display settings or apply filtering to one data type in a multiple display, the data type selected in the Layers window becomes the active data type.

With potentially many data types graphed in one sensor window, the active data type determines:

- the vertical scale for the Overplotted setting
- the colour of the vertical axis line
- which data will be queried or edited from all the data types in the graph window
- which data type the Smooth/Filter, Difference and Threshold graph lines will show for in the sensor window.

An asterisk next to the name of a data type, whether in the title bar or the window tab, identifies the active data type. In the example below, Roll is the active data in the Heave window.



See "Attitude Editor controls" on page 38 for the controls which are applied to the active data layer.

Attitude Editor controls

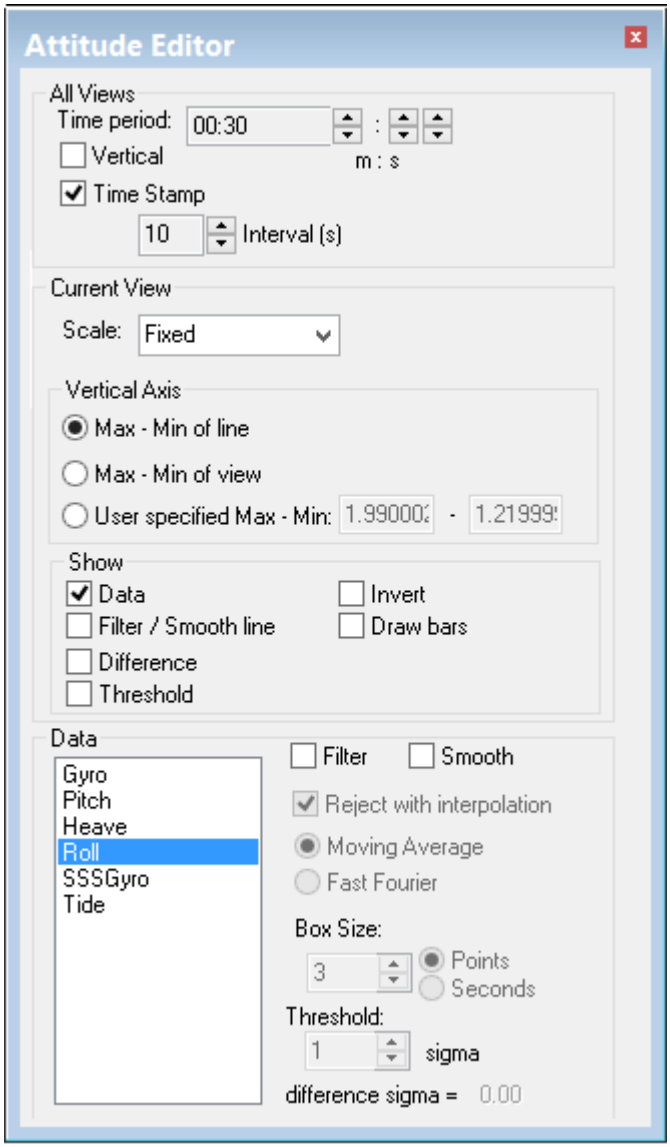
The Attitude Editor controls contain functions such as the scale of graph line and time intervals.

Controls are applied to:

"ALL VIEWS" ON PAGE 39

"CURRENT VIEW" ON PAGE 39

"DATA OPTIONS" ON PAGE 42



All Views

All Views settings apply to all windows open in Attitude Editor.

Time is displayed on the X-axis, by default. As the time scale is increased, more data is displayed in the graphs. As the time is decreased, less data is displayed.

1. Click the up or down arrows beside the *Time Period* box to set time in minutes and seconds.

You can also adjust the size of the slice in the Overview window for the same effect.

The *Vertical* option switches the data graph windows from a horizontal layout to a vertical layout. In this mode, the X-axis represents the data values and the Y-axis represents time.

2. To view the Attitude Editor in vertical mode, click the *Vertical* check box.
3. Select the *Time Stamp* check box to display the time stamp in the graph windows.
4. Click the *Time Interval* up or down arrow buttons to set how often you want the time stamps displayed.

The display in all graphs is adjusted to the new time setting.

Current View

C*urrent View* settings include vertical axis options and controls for how the data is displayed in a graph.

These options apply to the currently active sensor window, or to the active data type.

Scale

Scale can be set to Fixed or Overplotted. Overplotted is the default setting.

- Fixed: the scale uses the Max-Min values based on values of data from the sensors displayed in the graph window
- Overplotted: the scale is based on the Max-Min values of the active data type.

To change the scale setting to Fixed:

1. Click in the sensor window containing the data to be examined.
2. Select Fixed from the Scale list.

Vertical Axis

By default, the data scale is shown on the Y-axis (vertical axis). The scale of the axis can be based on the minimum and maximum values of data in the entire line, or the minimum and maximum values of the data currently in view, or a user-defined axis.

To set the scale for the Y-axis:

1. Select a sensor window.
2. Select from one of three scaling options:
   - Select *Max-Min of Line* to scale data according to the minimum and maximum data values of the entire line.
   - Select *Max-Min of View* to scale data according the minimum and maximum values currently in view in the sensor window.
   - Select *User-defined Max-Min* to define maximum and minimum scale and type the values in the maximum and minimum fields.

Show Data

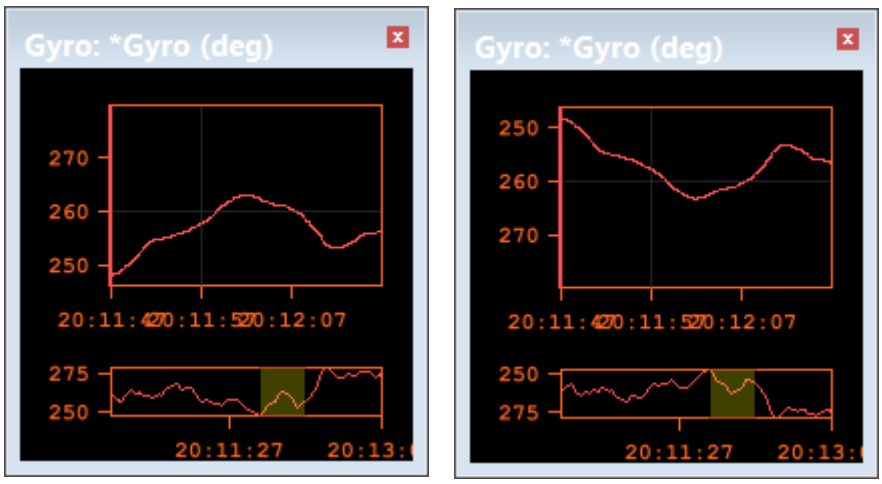
Enabling the *Data* check box activates the *Invert* and *Draw* bars options.

The *Invert* option displays the data in a “mirror-image” view in the active sensor window. This function is useful for viewing towfish depth in the water column,

To invert the data in the graph:

3. Select the *Invert* option.

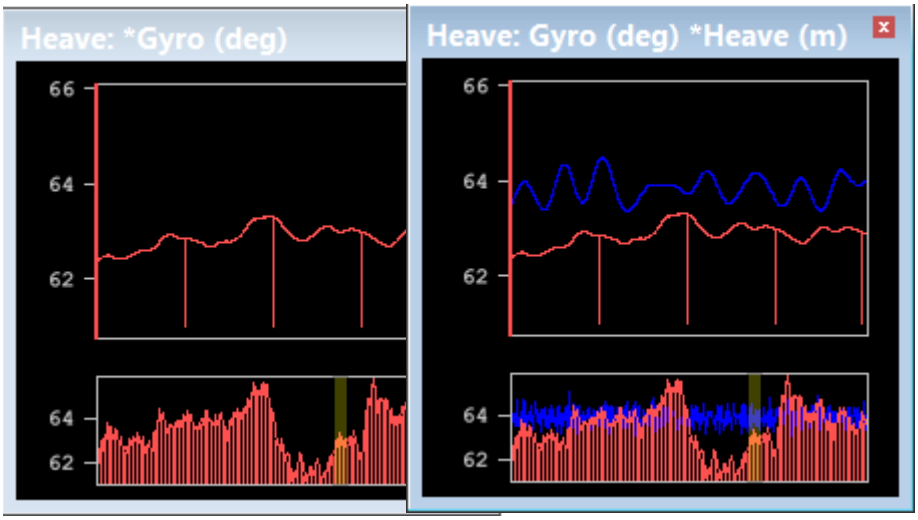
The vertical scale is inverted and the graph line redrawn.



The *Draw Bars* function displays vertical bars under the graph line. This option makes it easier to see time gaps between sensor values. If multiple data types are displayed in one graph, the vertical bars are drawn only for the active data type.

To view the graph with vertical bars:

4. Select the Draw bars check box.



The following options graph other lines for comparison to the data graph.

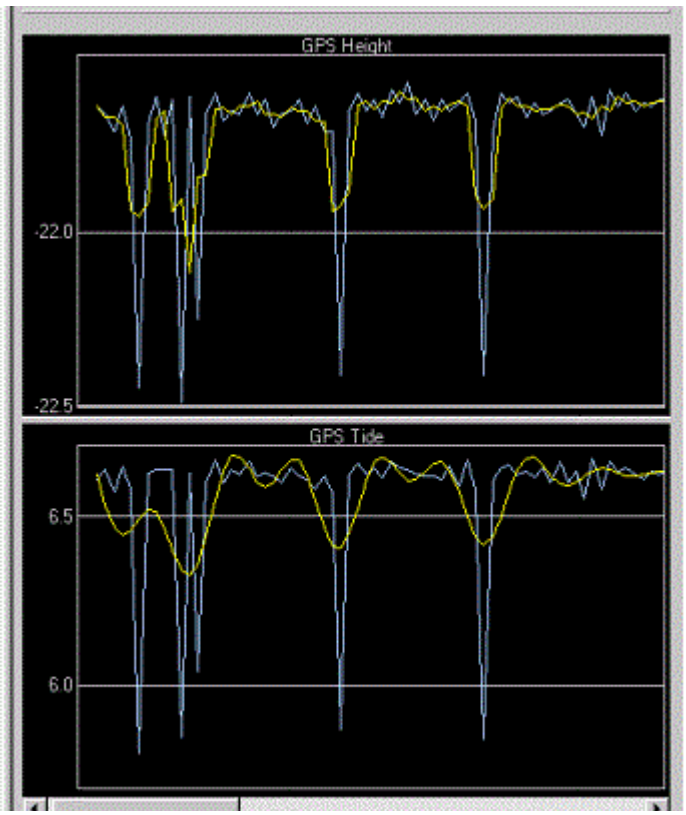
- *Filter/Smooth Line* graphs the computed values derived from the adjacent data points and displays this line in the graph. This option applies the parameters set in the Data options.
- *Difference* graphs a line showing the difference between the original data values and the moving average or FFT value for each data point.
- *Threshold* option displays horizontal lines in the graph indicating the difference values that are clipped during smoothing. Threshold is a multiple of the standard deviation (sigma).
  - The upper threshold signal is determined by the following formula:

    ```
    original mean value + (threshold value X the
    standard deviation)
    ```

  - The lower threshold signal is determined by the following formula:

    ```
    original mean value - (threshold value X the
    standard deviation
    ```

In this example, GPS height and tide, are graphed. The blue line represents the original data and the yellow line represents the smoothed data.



Data Options

Use the Data options to reject data with residual values that fall outside threshold limits, and apply these changes to attitude data across the entire track line, using the Filter and Smooth options.

The *Filter* function is used to reject attitude that falls outside of defined boundaries. Soundings with the same time stamp as the rejected data are also flagged as rejected during merge. You can choose to use interpolation when rejecting data.

The *Smooth* function is used to smooth out localized variability. Any sensor data that can be viewed in the Attitude Editor can be smoothed. The *Smooth* option activates all the options except *Reject with interpolation*.

Moving Average and Fast Fourier Transform

There are two methods available for smoothing: Moving Average and Fast Fourier transform.

- The *Moving Average* function calculates an average sensor value over a window of data using the Box Size parameter. The window is defined as a number of data points or seconds

centred on a point. The average value for all of the values in the window is calculated and will be given to the central point.

- The *Fast Fourier* option performs a low-pass filter on the selected sensor. You select a Box Size in either points or seconds. This value is converted into seconds and then inverted to become the cut-off frequency. Increasing the Box Size results in greater smoothing.

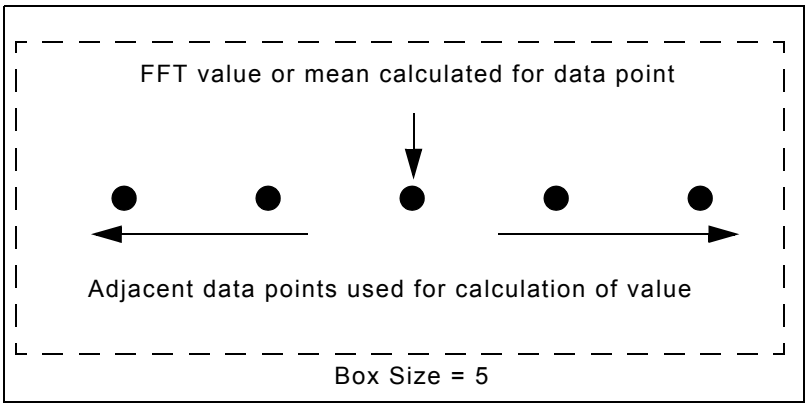
Box size

The *Box Size* determines how many neighbouring data points are used in the smoothing calculation. The number of neighbouring data points can determined using time or a set number of points.

The Moving Average and the Fast Fourier algorithm use the Box Size parameter in different ways.

- The Moving Average function uses the Box Size to determine the window that is used to calculate the average for the new "smoothed" centre point.
- The FFT algorithm uses the Box Size to define the cut-off frequency that is then applied to all data.

In the following example, the box size is set to five data points. A FFT value or mean for any data point is calculated from itself and 'the two data points on either side of it (if available).



To apply filtering and smoothing:

1. Select a data type from the Data list or click inside a sensor window.
2. Select the *Filter* check box to activate the *Reject with interpolation* option.
3. Select the *Smooth* check box.
4. Select either the *Moving Average* or *Fast Fourier* smoothing options. "MOVING AVERAGE AND FAST FOURIER TRANSFORM" ON PAGE 42
5. Select either the *Points* or *Seconds* check boxes.
6. Click the *Box Size* up or down arrow buttons to select the number of adjacent data points or a time range.
7. Set a Threshold value in a multiple of standard deviation (sigma).

(These options can also be applied from the Attitude filter. See "FILTER ATTITUDE DATA" ON PAGE 327 of the User Guide.)

When you close the Attitude Editor, you are prompted to save any changes. The parameters for creating the smoothed data are saved to the `SmoothedCoefficients` file in the `HCDS_Data\Project\Vessel\Day\Line` folder. The file is in XML format. This file is applied to the track line during processing, e.g., merge, SVP Correction and Compute Towfish Navigation.

3

Process LIDAR Data

LIDAR data converted to HIPS format can be viewed and cleaned in Subset or Swath Editor. You can select and flag the status of soundings. Data can also be rejected or accepted.

If waveform data is available it can be viewed in a waveform viewer.

In this chapter...

LIDAR Data in HIPS

Light detection and ranging (LIDAR) uses laser technology to measure bathymetry. In such surveys, laser wavelengths are transmitted to the water from a transceiver mounted on a plane or helicopter. Sounding data is returned to the transceiver in a continuous stream that forms a swath-like pattern.

HIPS currently supports these systems:

- Laser Airborne Depth Sounder (LADS)
- Scanning Hydrographic Operational Airborne LIDAR Survey (SHOALS) system
- Hawkeye from Airborne Hydrography AB (AHAB).

LIDAR Projects

LIDAR projects are organized in the same Project-Vessel-Day-Line hierarchy as sonar projects. Each project contains the following files:

- HIPS Vessel File
- an Observed Depths file (along-track and across-track sounding positions before merge)
- a Processed Depths file (latitude/longitude sounding positions after merge)
- a bathymetry file
- gyro data
- navigation data
- tide data

As well, LADS, SHOALS and Hawkeye projects can contain waveform data. LAS format data can also be opened in HIPS, but this format does not contain waveform data.

To convert LIDAR data to HIPS format, see:

"HAWKEYE" ON PAGE 111

"LADS" ON PAGE 120

"SHOALS" ON PAGE 128 of the User Guide.

Data format

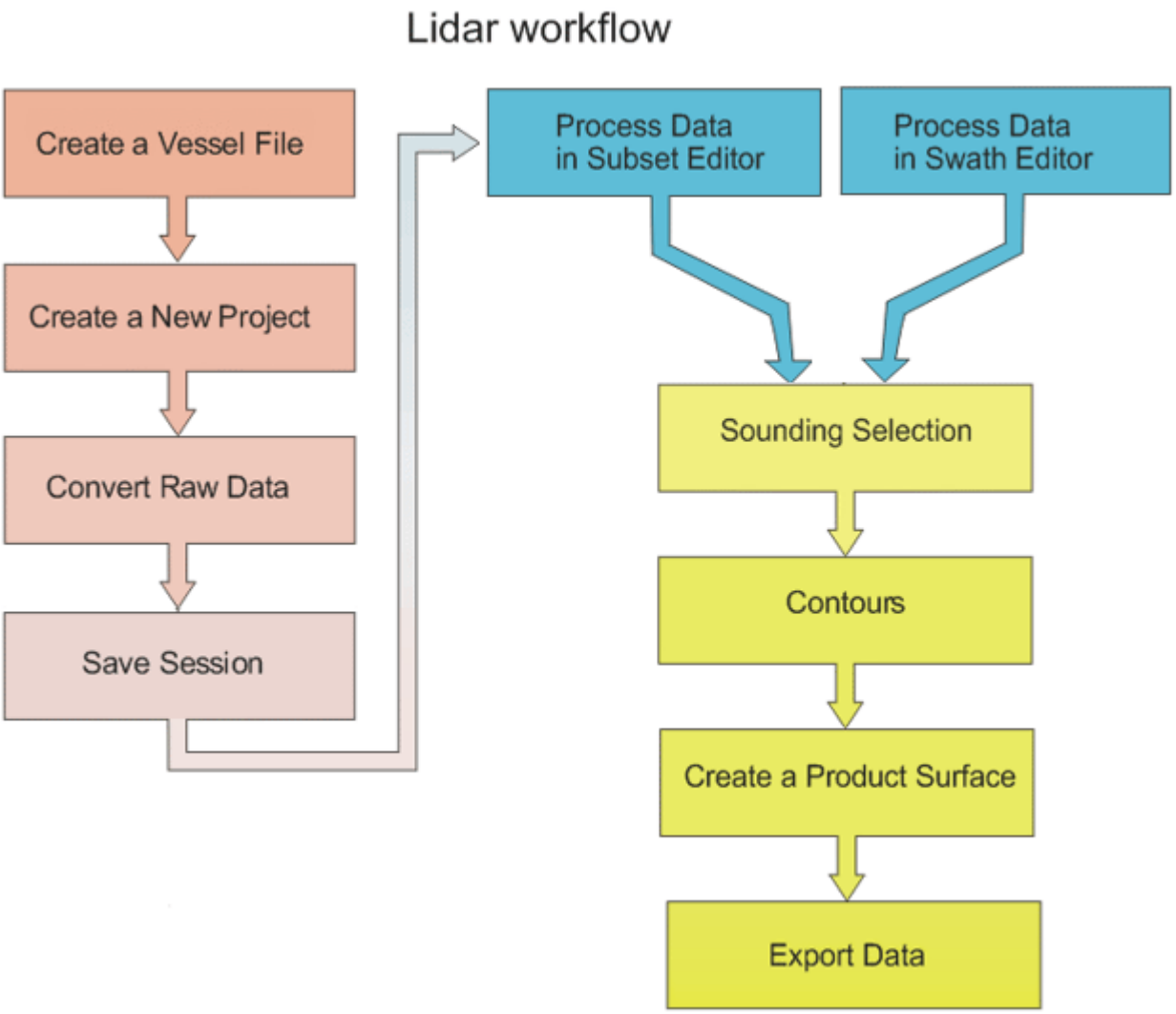
Soundings are recorded in a bathymetry file. Each sounding has:

- a latitude and longitude position
- a depth value
- a time stamp
- an across-track and along-track position
- a quality flag
- a status flag.

Waveform data can also be included for each sounding. If available, this data is stored in a waveform file (see "WAVEFORM VIEWER" ON PAGE 54).

Workflow

LIDAR projects follow a similar workflow to other processing in HIPS. The following chart shows the most important steps for cleaning LIDAR data



1. Create a new HIPS Vessel File (HVF) vessel containing sensor locations and uncertainties related to the LIDAR and navigation information. See "Create a New HVF" on page 31 in the User Guide.)
2. Create a new project with Project-Vessel-Day directory structure to store the data. (See "New Project" on page 63 in the User Guide.)
3. Convert the raw data files to HIPS format and assign to the new project directory. See "Convert Data" on page 79 in the User Guide.
4. Save the currently open data as a session file. See "Session Files" on page 25.
5. Either process data using Subset Editor:
   - Create a Subset Tile layer.

- Open the Subset Editor and use the subset bounding box to select an area in the Subset Tile layer.
- View and clean data.
- Change the quality flag of soundings, if needed.
- Create designated soundings using the Designate Sounding and Find and Designate commands.
- Track the progress of cleaning by flagging tiles as Complete, Partially Complete or Reset.

6. Or, process in Swath Editor:
   - Open the Swath Editor to examine and clean sounding data.
   - Select soundings to view waveform data and other information.
   - Change the quality flag of soundings, if needed.
   - Reject data or change the status of a rejected sounding back to
   - accepted.
   - Create designated soundings using the Designated Sounding and
   - Find Designated Sounding commands.
7. Create a surface. See “Create a New Surface” on page 207 of the User Guide.
8. Add contours and other layers.
9. Designate shoalest soundings.
10. Create a product surface.
11. Export finished soundings to a CARIS map or other format.

.

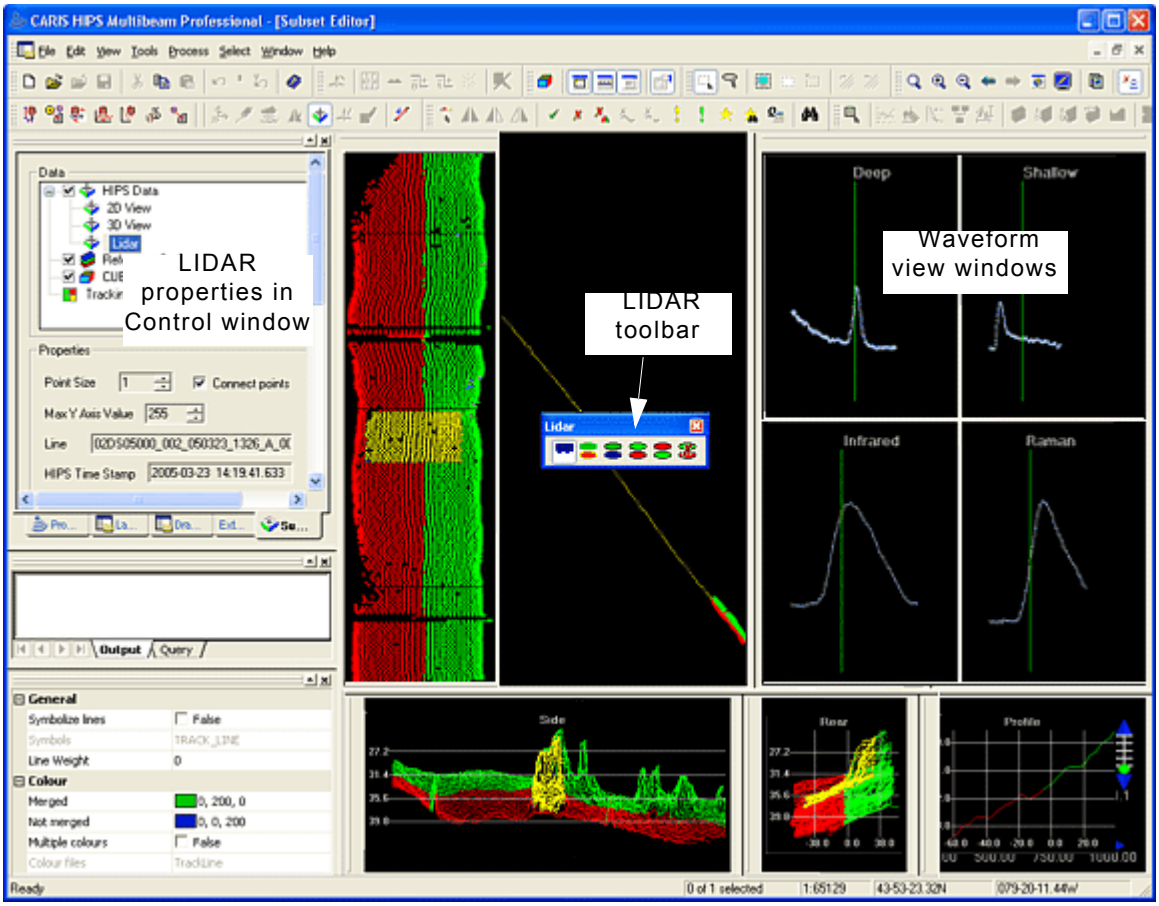
Display LIDAR data

LIDAR data can be processed in either Swath Editor or Subset Editor, depending on how you want to view the data.

If you use Subset Editor, the data can be viewed with the 2D and 3D view, and the waveform viewer, and the cleaning progress can be tracked using the Subset Tile function.

If editing in Swath Editor a LIDAR tab and waveform view are available.

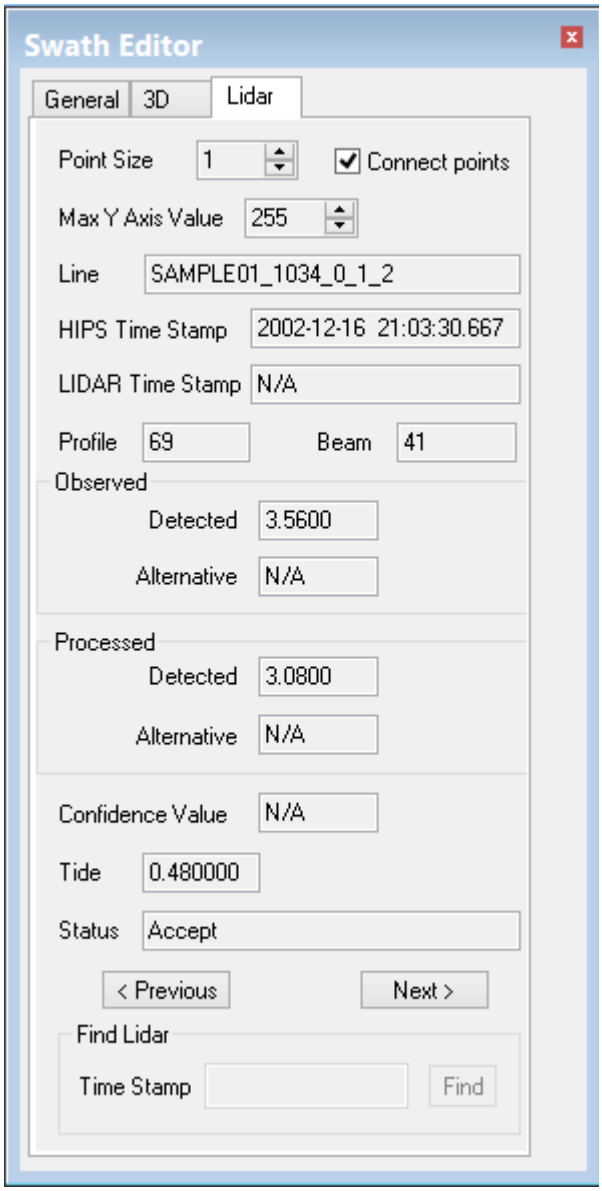
The following image shows SHOALS LIDAR data displayed in Subset Editor.



LIDAR tab

Properties and controls for LIDAR data are located on a tab in the Control window.

- In Subset Editor, the LIDAR properties are displayed in the LIDAR layer of the Subset Editor tab.
- In Swath Editor, these are displayed on a tabbed page of the Swath Editor window, as illustrated below.



Menu	Edit > LIDAR > View Waveforms
Tool	

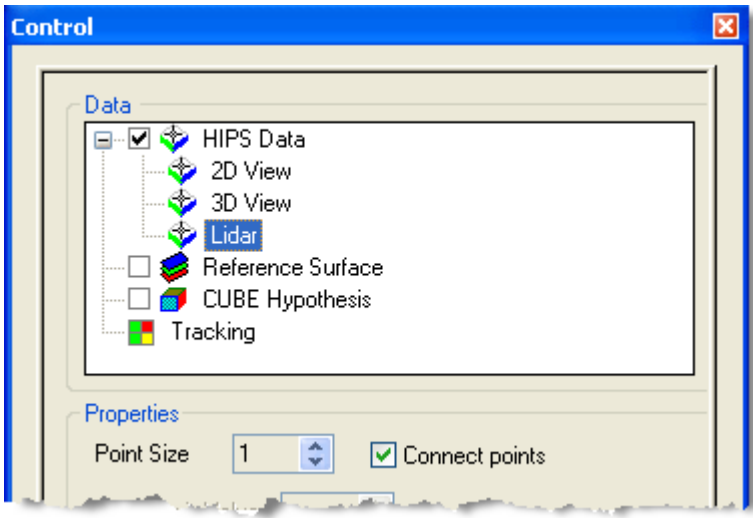
To display the LIDAR controls in Swath Editor:

1. Right-click on the data in a View window to see a pop-up menu.
2. Select View Waveform.

The LIDAR tabbed page appears on the Swath Editor window.

To display the LIDAR tab in Subset Editor:

1. Open Subset Editor and make a subset of the LIDAR data.
2. Highlight the LIDAR layer in the data tree on the Subset Editor tab.



3. Select soundings in one of the Swath Editor view windows.

Properties

The following properties for the selected sounding are displayed in the LIDAR tab in both editors:

- *Point Size*: Change the size of the points that describe the waveform graphs.
- *Connect Points*: Select this option to draw a line through each of the points in the graph.
- *Max Y Axis Value*: Change the scale of the Y axis to display less or more of the waveform.
- *Line*: Identifies the superselected line.
- *HIPS Time Stamp*: Year-Month-Day-Hour-Minute-Seconds-Milliseconds.
- *LIDAR Time Stamp*:
  - Half milliseconds since midnight (SHOALS only).
  - Microseconds (Hawkeye data).
- *Profile (number)*: The along-track position of the sounding
- *Beam (number)*: The across-track position of the sounding.

Observed fields display the sounding values when converted to HIPS format. If any processing (such as tide) is applied to this data the new value will be displayed in the *Processed* fields.

- *Detected*: The value designated by the acquisition system as being the sounding depth. If a Detected depth is selected, an asterisk is displayed beside the field.
- *Alternative*: If the data includes an alternative depth value from the return, it will be displayed here. If you select an alternative depth, an asterisk is displayed beside the field. (See"SELECTING DETECTED AND ALTERNATIVE DATA" ON PAGE 58.)

Note: Not all Hawkeye data contain alternate depths that can be highlighted or swapped with the primary depths.

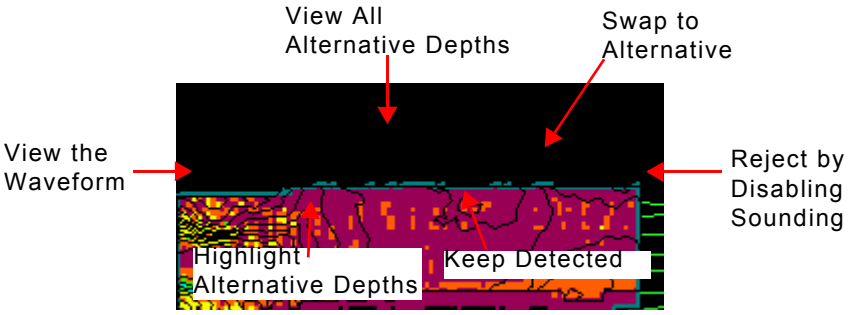
- *Confidence value*: The confidence value for the sounding (SHOALS only).
- *Tide*: Tide data for the sounding, if there is any in the file. (Not all formats contain tide data, in which case the field shows a value of zero.) If tide data is added, data must be merged before tide value can be displayed.

Note: Hawkeye data does not contain tide data.

- *Status*: Current status flag for the selected sounding.
- *Find LIDAR Time Stamp*: Select a sounding by its time stamp. Time stamp for Hawkeye data must be entered in microseconds.

Toolbar

The LIDAR toolbar controls the display and status options for soundings and associated waveform data in the Swath and Subset Editors.



This is a floating toolbar that can be moved or docked anywhere in the HIPS main window. The tools on the toolbar become activated when Swath Editor is opened or data loaded to Subset Editor.

Menu	View > Toolbars > LIDAR
Pop-up	LIDAR

If the LIDAR toolbar is not visible in the interface,

1. Select the LIDAR toolbar command to display it.

Waveform Viewer

The Waveform view displays the waveform data for a super-selected sounding (see "SELECTING DATA" ON PAGE 56 for more information on superselection). The display in the Waveform view is determined by the data format being viewed.

Menu	Edit > LIDAR > View Waveform
Pop-up	View Waveform
Tool	

1. Select data in any of the Swath Editor or Subset Editor views.
2. Select a View Waveform command.

The first sounding in the selection (i.e. with the lowest beam and profile number) is superselected when the Waveforms window is opened.

LADS

The LADS waveform window contains one panel that shows a graph of specific waveform data for the superselected sounding. One line represents the detected depth sounding and the other line represents the alternative depth sounding (if applicable)

SHOALS

There are four panels in the waveform view for SHOALS data. Each panel shows a graph of specific waveform data for the superselected sounding. The X axis represents time and Y axis represents amplitude.

To view the complete waveform for SHOALS data, the Y-axis scale may have to be increased. The default value is 255.

1. Click the *Max Y-Axis Value* up or down arrow buttons until the entire wave form is within view.

Point display

The points that describe the waveform graphs can be increased or decreased.

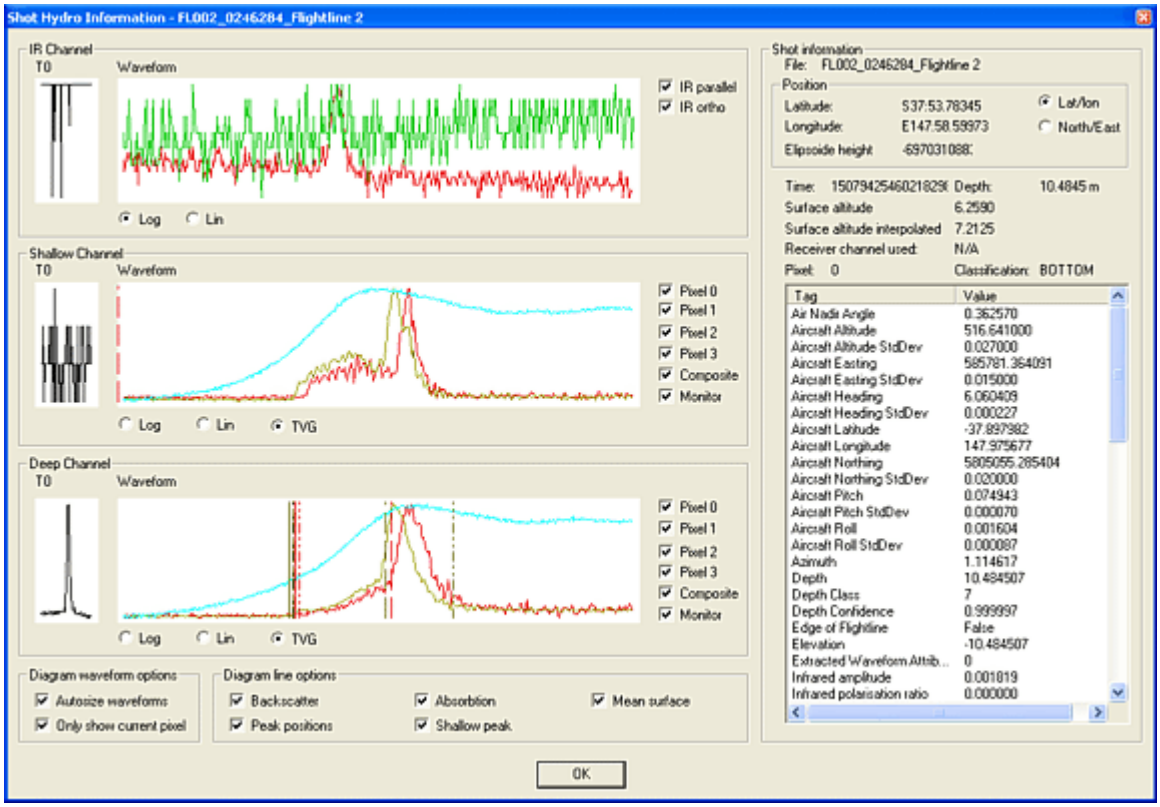
1. Select the LIDAR tab in the Control window if it is not already displayed.
2. Click the *Point Size* up and down arrow buttons.

As the value changes, the displays in the four views are redrawn to show the points at the selected size.

3. To draw a line through the points in the view, select the *Connect Points* check box.

Hawkeye

The waveform viewer for Hawkeye data was developed by AHAB. This viewer is activated when you select Hawkeye data in any of the Swath Editor or Subset Editor views. The Hawkeye viewer displays information for the currently superselected sounding, if it has waveform data.



Not all Hawkeye soundings contain waveforms that can be viewed. If a waveform cannot be found, an error message appears in the Output window.

Selecting Data

Open data

LIDAR data can be viewed and cleaned in the Swath and Subset Editors.

Menu	Edit > LIDAR > View Waveform
Pop-up	View Waveform
Tool	

In Swath Editor:

1. Select data in a Swath Editor view.
2. Select the View Waveform command.

In Subset Editor:

1. Select an area and Load the data.
2. Select the View Waveform command.

The Lidar Waveform window (see "WAVEFORM VIEWER" ON PAGE 54) is displayed and the Lidar tab is displayed in the Swath Editor control window. (Nothing is visible in the Waveforms window until data is selected.)

Selection/ Superselection

There are two levels of selection in LIDAR mode: regular selection and superselection.

- Selection refers to all highlighted soundings in any of the editor views (e.g., Profile View, 2D View, etc.).
- A superselected sounding is a subset of the selected soundings and is highlighted in a different colour from the regular selection. When a group of soundings are selected, the first in the selection (i.e. the lowest beam and profile number) is automatically super-selected.

All selected soundings can be queried or rejected, but you can only view waveform data and sounding attributes for one superselected sounding at a time. The superselection option is only active when the Waveforms window is open.

3. Choose a selection tool from the Select menu. The *Edit By Range* option is selected by default.
4. Select data in any of the views.

The selected data is highlighted in the selection colour determined in the Tools > Options > Display > LIDAR dialog box. The superselected sounding is displayed in the Plan View (in Swath Editor) and the 2-D and 3-D Views (in Subset Editor).

Waveform data for the superselected sounding is displayed in the Waveforms view.

Move to next sounding

You can use the LIDAR tab to superselect a neighbouring sounding in the current selection.

5. Click the **Next** or **Previous** button in the LIDAR tab of the Control window. This moves you to the next or previous selected sounding by ping and beam number.

The graph in the Waveforms window is refreshed to show the new data and the attributes for the newly superselected sounding are shown in the data fields on the LIDAR tab.

Find sounding

You can also superselect a sounding by locating it by its time stamp (SHOALS data only).

1. Type the value of the LIDAR time stamp in the *Time Stamp* field.
2. Click **Find**.

The sounding with that time stamp is highlighted as superselected and the sounding data is displayed in the LIDAR tab fields.

Superselect data in the Selection window

Queried data can also be superselected.

1. Select data in any of the views.
2. Select the Query command.

Menu	Edit > Query
Tool	
Pop-up	Query
Key	<Q>

The selected data is displayed in the Selection window.

3. To superselect a sounding, click a row so it is highlighted.

The beam is highlighted as superselected in the Plan View.

Sounding Quality and Status Flags

When a LIDAR sounding is converted into HIPS, it is given a quality flag that can be changed during processing. This table shows the quality statuses that can be attached to a sounding.

Quality Flag	Description
0	The sounding has a detected depth only. The alternative depth is invalid.
1	The sounding has both a detected and a valid alternative depth. A quality control/assurance decision has not yet been made.
2	The sounding has a detected depth and a valid alternative depth, and the user has decided to keep the detected depth.
3	The sounding has both a detected and alternative value, but the alternative value has been swapped with the detected value).

Status flags

Status flags are assigned to soundings during conversion, these status are determined from information provided by the specific acquisition format.

Selecting Detected and Alternative Data

In some instances, two depth values (detected and alternative) are returned. You can decide which depth value is most relevant and change the sounding's quality flag accordingly (see "Sounding Quality and Status Flags" on page 58).

> Depth values can be selected and changed in both the Swath Editor and the Subset Editor.

Highlight alternative soundings

This command highlights soundings that contain valid alternative depths. It does not select or superselect soundings.

Menu	Edit > LIDAR > Highlight Alternative Depths
Tool	

1. Select the Highlight Alternative Depths command. (This command only highlights data in the Plan View.)

Soundings with a Quality flag of 1, 2, or 3 are highlighted in colour that is set in the Tools > Options > Display > LIDAR dialog box.

You can select from three options:

- View all alternative depths
- Swap to alternative
- Keep detected.

View all alternative depths

This command refreshes the display so that the alternative depths for soundings with a Quality 1 flag are shown. This is a

useful tool for viewing the position of alternative depths in relation to neighbouring soundings.

Menu	Edit > LIDAR > View Alternative Depths
Tool	

1. Select the View Alternative Depths command.

The data in the Views refresh to display the alternative soundings.

Swap to alternative

This command applies an alternative depth in a superselected sounding to the detected depth. With SHOALS data, the depth is swapped, and with LADS data, the depth and position are swapped.

Menu	Edit > LIDAR > Swap to Alternative
Tool	

1. Select soundings and superselect a sounding with a Quality 1 flag.
2. Select the Swap to Alternative command.

An asterisk beside the depth value indicates whether the detected or alternative depth is currently in the HIPS depth file.

Keep detected

This command is used to verify that the detected depth is preferred over the alternative depth (see "Sounding Quality and Status Flags" on page 58).

Menu	Edit > LIDAR > Keep Detected
Tool	

1. Select soundings and superselect a sounding with a Quality 1 flag.
2. Select the Keep Detected command.

The Quality flag is changed to 2.

Cleaning Sounding Data

Soundings can be assigned a “Rejected by Disabling Sounding” flag. These soundings will not be included in the merge or any post-processing operations. Data can rejected in the Swath and Subset Editors.

Menu	Edit > LIDAR > Reject by Disabling Sounding
Tool	

To reject soundings:

1. Select a sounding or group of soundings.
2. Select the Reject by Disabling Sounding command.

The soundings are now flagged as rejected.

To return rejected soundings to Accepted status:

1. Select the soundings.
2. Select the Accept command.

Menu	Edit > Status Flag > Accept
Tool	
Pop-up	Accept
Key	<A>

Use the Display Filter function to see rejected soundings. See “VIEW SOUNDINGS STATUS” ON PAGE 332

4

Navigation Editor

Navigation Editor enables you to examine and clean position data for both the ship and towfish.

In this chapter...

Navigation Editor Interface

Navigation data can be viewed and queried in the Display window. Multiple navigation sources can be viewed and edited individually when set as the active source.

Three time-series graphs: Speed, Distance and Course Made Good are displayed in dockable windows. The view in these graphs are synchronized: data selected in one window will select the corresponding data in the other two graphs.

In the Display window, Navigation Editor shows the track lines as a series of continuous symbols, with each point representing a navigation fix. These symbols become visible as you zoom in on the selected line.

You can change the CARIS feature code used to represent these navigation symbols using the Options in the Tools > Options General dialog box.

Menu	Tools > Editors > Navigation Editor
Tool	

To open Navigation Editor:

1. Select a track line.
2. Select the Navigation Editor command.

The Navigation Editor and graphs are displayed. Three graphs display:

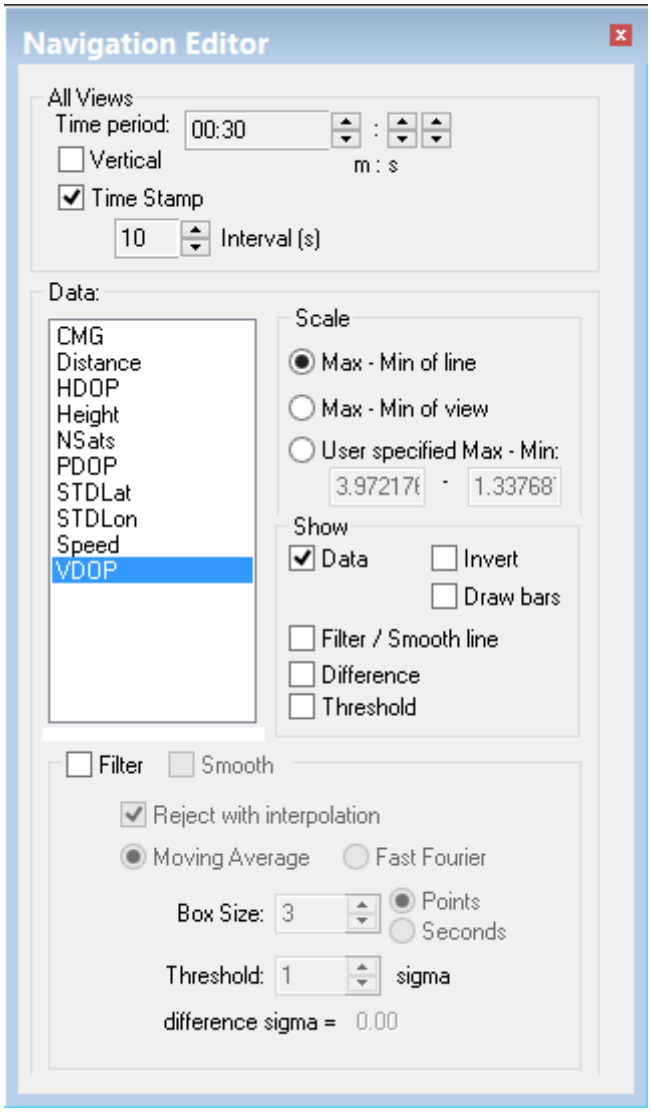
- **Speed:** Calculated speed.
- **Distance:** Calculated distance.

Course Made Good: Calculated direction (in degrees) from one navigation fix to the next.

When lines with multiple navigation sources are open in Navigation Editor, the data sources are displayed simultaneously in the Navigation graphs. To differentiate between sources, set different colours for each using the Properties.

Navigation Editor controls

Display of various aspects of the data in Navigation Editor is controlled by the options on the Navigation Editor.



Vertical

By default the data in the graphs is displayed horizontally. To change the display:

3. Select the Vertical check box.

Time Stamps

Display lines to represent time stamps in Navigation Editor windows. The time stamp interval is displayed in seconds and can be changed.

4. Select the *Time Stamp* check box to display time stamp lines.
5. Click the up or down arrow buttons beside the *Interval* to select the gap (in seconds) between time stamps.

Scale

When this option is selected the vertical scaling of the graphs fits the maximum and minimum range of values for the entire line. Otherwise the vertical scaling of the graphs fits the max-min range of the data currently in view.

1. Open the Control window.
2. Select the *Navigation* tab.
3. Select the *Max-Min of Line* check box.

Filter /Smooth line

1. Select the *Filter / Smooth line* check box to display the line in the Display window.

5

Side Scan Editor

Side Scan Editor provides tools for viewing and editing side scan sonar data, and for digitizing and editing contacts.

In this chapter...

Side Scan Editor workflow

Use Side Scan Editor to:

- edit data from a waterfall display
- visualize and apply imagery corrections
- edit side scan altitude data
- create mosaics
- digitize and edit contacts.

Side scan data must have digitized altitude before creating mosaics.

Streamlined workflow

1. Open converted data in HIPS and SIPS.
2. Open one line in Side Scan Editor for investigation and to determine appropriate correction settings.
3. Correct altitude if necessary. Automatically digitize altitude.
4. Apply an appropriate TVG and Gain correction using the controls in the Properties window. (Display will update dynamically as values are changed.)
5. Create a beam pattern file and load it using the controls in the Properties window.
6. When satisfied with the quality of the image, close Side Scan Editor and save the settings.
7. Create a mosaic from selected lines (using the Tools > Mosaics > New command.

   Advanced settings can be drawn from the settings configured in Side Scan Editor properties.
8. Create and edit contacts.

Side Scan Log

Side Scan Editor can automatically create a log file for each line viewed, which will track and record when a Side Scan Editor session begins and ends.

The log will also record when an area of data is viewed longer than a specific number of seconds.

To activate automatic waterfall view logging:

1. Select *Automatic Side Scan Waterfall Logging* option in the Tools > Options > Editing dialog box.

Enabling the logging function will also enable the *Log entry when idle for more than* option. This will set the amount of elapsed time before a log entry is made.

2. Set the number of seconds that data is viewed before the time is logged.
3. Click **Apply**.

The session log records each change of pings viewed on a selected line, and the time frame in which they are viewed.

The log is saved as an xml file, such as the example below.

```
<?xml version="1.0" encoding="UTF-8"?>
<SessionLog>
 - <Session>                                   A                    B
      <Line end_ping="2,905" start_ping="1" line_name="231_1841"/>
   C  <Reviewer name="SPhillips"/>
    - <View>                  D
         <PingsOnScreen max_ping="2,642" min_ping="1,646"/>
      E  <PortBeamsOnScreen max_beam="1,025" min_beam="0"/>
         <StarboardBeamsOnScreen max_beam="1,025" min_beam="0"/>
      F  <StartTime time="18:09:31" date="03-16-2016"/>
         <EndTime time="18:29:38" date="03-16-2016"/>
   G  </View>
    - <View>
         <PingsOnScreen max_ping="997" min_ping="1"/>
         <PortBeamsOnScreen max_beam="1,025" min_beam="0"/>
         <StarboardBeamsOnScreen max_beam="1,025" min_beam="0"/>
         <StartTime time="18:29:41" date="03-16-2016"/>
         <EndTime time="18:30:21" date="03-16-2016"/>
      </View>
   H  <StartTime time="18:09:30" date="03-16-2016"/>
      <EndTime time="18:30:21" date="03-16-2016"/>
 I </Session>
</SessionLog>
```

Contents of the Side Scan Log for a line include:

A: End and start ping number for the entire line

B: Line identifier

C: Reviewer name, as set in the Windows user name

D: Range of the pings selected in the Side Scan (waterfall) View

E: The minimum / maximum port and starboard beam range in the view. (Because the scan range is up to 1000m and screen resolution is limited, it is possible that only one channel will be viewed at a time.)

F: The start and end time during which the waterfall area was viewed.

G: Indicates the end of one viewing period log, and the start of the next.

H: The elapsed time of the Side Scan session for the selected line.as indicated by the session start and end time.

For options related to the display of side scan data, see "SIDE SCAN EDITOR" ON PAGE 657 of the User Guide.

Side Scan Editor Interface

Side Scan Editor consists of the Side Scan View window (waterfall view), a Side Scan Editor toolbar and various controls in the Properties window.

As well, when contacts are selected, the Image Preview and Attributes windows can be opened to view aspects of the contacts.

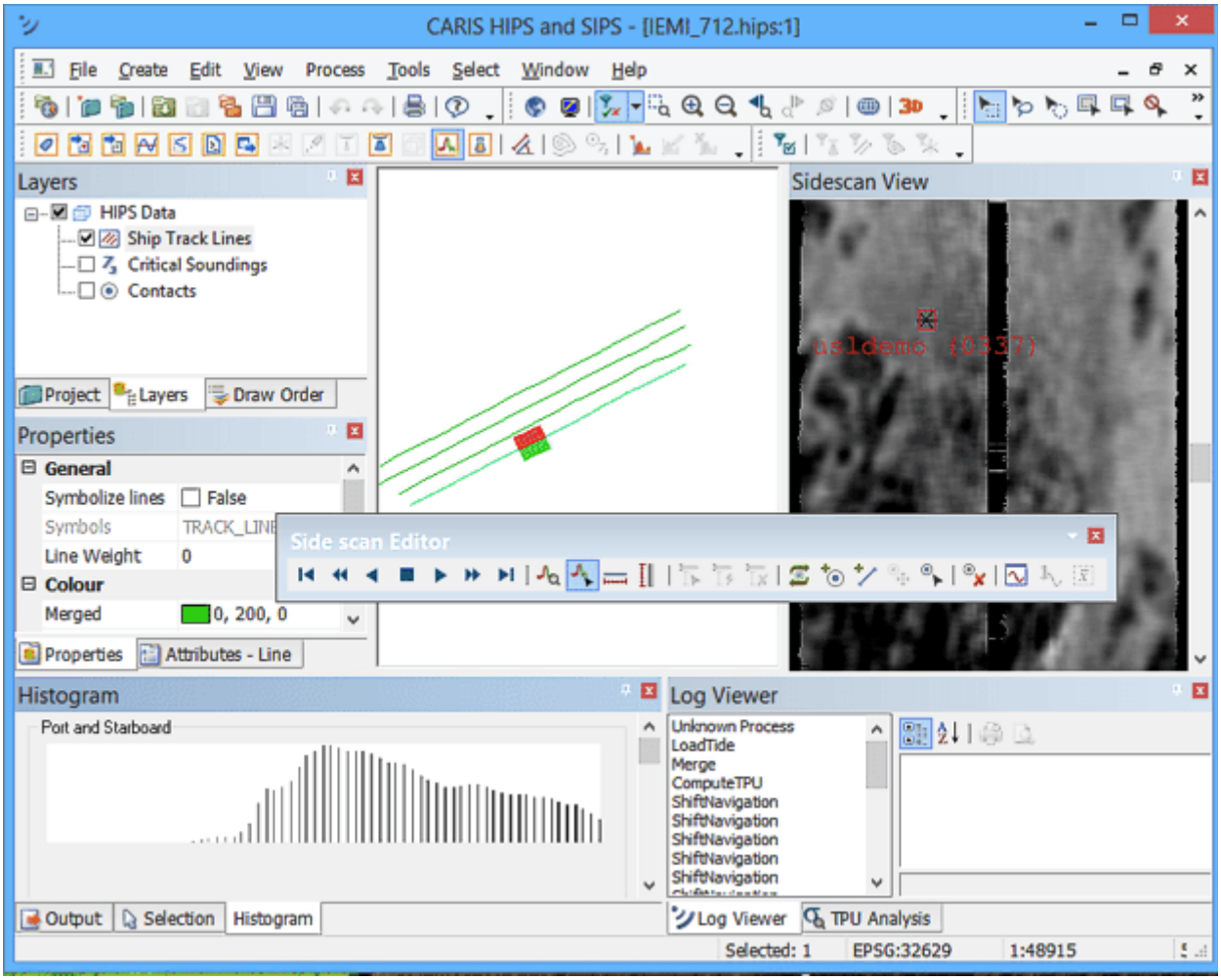
To open Side Scan Editor:

Menu	Tools > Editors > Side Scan > Open
Tool	

1. Select a towfish track line in the HIPS and SIPS Display window.
2. Select a Side Scan Editor command.

When Side Scan Editor is opened, the data for the selected track line is displayed in the Side Scan View window.

As well, a histogram of the sonar file is displayed in the Histogram window, and the Side Scan toolbar is activated.



Side Scan data display

Side scan data is displayed in the Side Scan View window (also called the “waterfall” view). This window can be undocked and moved to another position.

There are two modes to display data in the Side Scan View window:

- **Raw**: (the unprocessed sonar data). In raw mode the across-track axis represents time, thus the data is shown in the sequence in which it was received. The central portion of the image still displays the water column. This mode will always show the raw data for the selected track line.
- **Processed**: (the processed sonar data). In processed mode the across-track scale represents distance. The water column is no longer visible as it is removed during the automatic slant range correction. (The data displayed in this mode will reflect the configuration of corrections in the Properties window.)

Menu	Tools > Editors > Side Scan > Raw / Processed
Pop-up	Raw Processed

1. Right-click in the waterfall view to select a display mode.

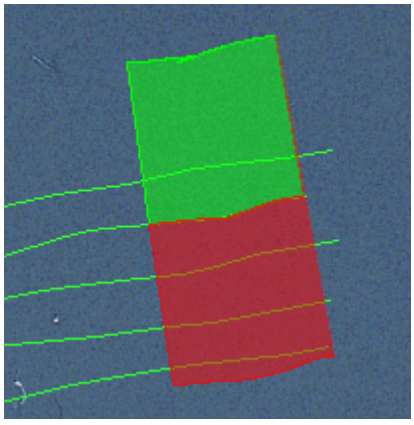
The display is refreshed to show the data in the selected mode. The selected mode is indicated with a check mark in the menu.

If the data does not fill the entire height of the Side Scan View, it will be displayed in the bottom of the window and can be processed normally. The empty area above the data will indicate “No data to display”.

Coverage outline

A coverage outline in the Display window automatically highlights the location of the side scan data that is visible in the View window. Changing the location of the outline changes the data that is displayed in the Side Scan View, and vice versa.

The starboard and port areas of the coverage outline are appropriately colour-coded, as illustrated below.



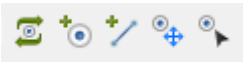
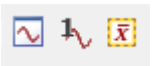
Colours for outlines and selection are controlled from the Tools >Options >Display > Side Scan Editor dialog box.

You can use the Side Scan Measure Distance and Measure Shadow tools in the View window (see "MEASURE DISTANCE IN SIDE SCAN VIEW" ON PAGE 79 and "MEASURE SHADOW" ON PAGE 79). When you release the mouse button, the results are shown in the Coordinates window.

Side Scan Toolbar

The Side Scan Editor toolbar is displayed when the editor is opened.

The toolbar contains controls to navigate through the track line and perform other functions.

Toolbar Buttons	Purpose
	Playback (see "NAVIGATE IN SIDE SCAN EDITOR" ON PAGE 73)
	Zoom (see "ZOOM IN SIDE SCAN VIEW." ON PAGE 75)
	Select (see "SELECTION" ON PAGE 77)
	Measure distance and shadow (see "MEASURE DISTANCE IN SIDE SCAN VIEW" ON PAGE 79 and "MEASURE SHADOW" ON PAGE 79)
	Digitize (see "DIGITIZING ALTITUDE" ON PAGE 86)
	Contacts (see "CREATE CONTACTS" ON PAGE 98)
	View waveform (see "SIGNAL DISPLAY WINDOW" ON PAGE 80)

Navigate in Side Scan Editor

You can navigate up or down the track line with:

- the scroll bar on the Side Scan View window
- the mouse wheel
- the keyboard
- the Playback (scrolling) controls on the Side Scan toolbar.

Scroll bar

Each click of a scroll arrow button (at the top and bottom of the scroll bar) moves the display in the View window forward or backward by a single ping.

- Clicking in the scroll bar, or dragging the scroll box forward or backward moves your view by a range of pings.

Mouse wheel

Alternately, you can move up or down the track line by turning the mouse wheel (if available).

Keyboard

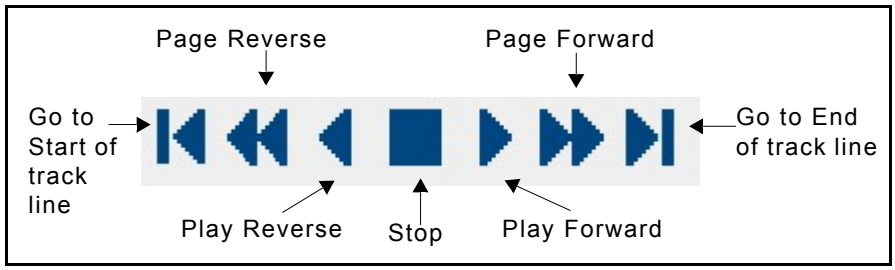
You can also navigate up or down the track line using the keyboard.

- Press <**Page Up**> or the space bar to move the preview forward by one "page" (the height (# of pings) visible in your SIPS Preview window is the increment).
- Press <**Page Down**> or hold the <**Ctrl**> key while pressing the space bar to move the preview back by one "page".
- Press <**Home**> to go to the start of the track line, and press <**End**> to go to the end of the track line.

Playback controls

Menu	Tools > Editors > Side Scan > Playback Controls

Side Scan Editor can automatically scroll data from the start of the track line to the end by using the *Playback* controls.



The ping coverage outline in the Display window moves with the display in the Side Scan View.

1.Click **Start** to go to the start of the track line.

2.Click **Play Forward** to start scrolling the data.

Data in the View window is displayed, ping by ping, until it reaches the end of the surveyed data.

3. Click **Play Reverse** to scroll back down the track line.
4. Click **Start** to reset the view to the first ping.
5. Click **Stop** to pause the scrolling
6. Use **Page Forward** and **Page Reverse** to move along the track line in larger increments.

If you extend the Side Scan View window, the side of the coverage outline in the Display window will also enlarge.

Playback speed

You can adjust the scrolling speed for automatic scrolling in the *Playback Speed* field in the Display properties list.

View contacts while scrolling

To set contacts to remain visible in the waterfall display while scrolling or using the Playback functions:

1. Select Side Scan Editor from the Tools > Options > Display tab.
2. Select the *Display contacts during playback* check box, and click **Apply**.

Zoom in Side Scan View.

You can magnify an area in the waterfall display using the Zoom tools:

- Zoom to an Area
- Zoom In
- Zoom Out

To zoom to an area:

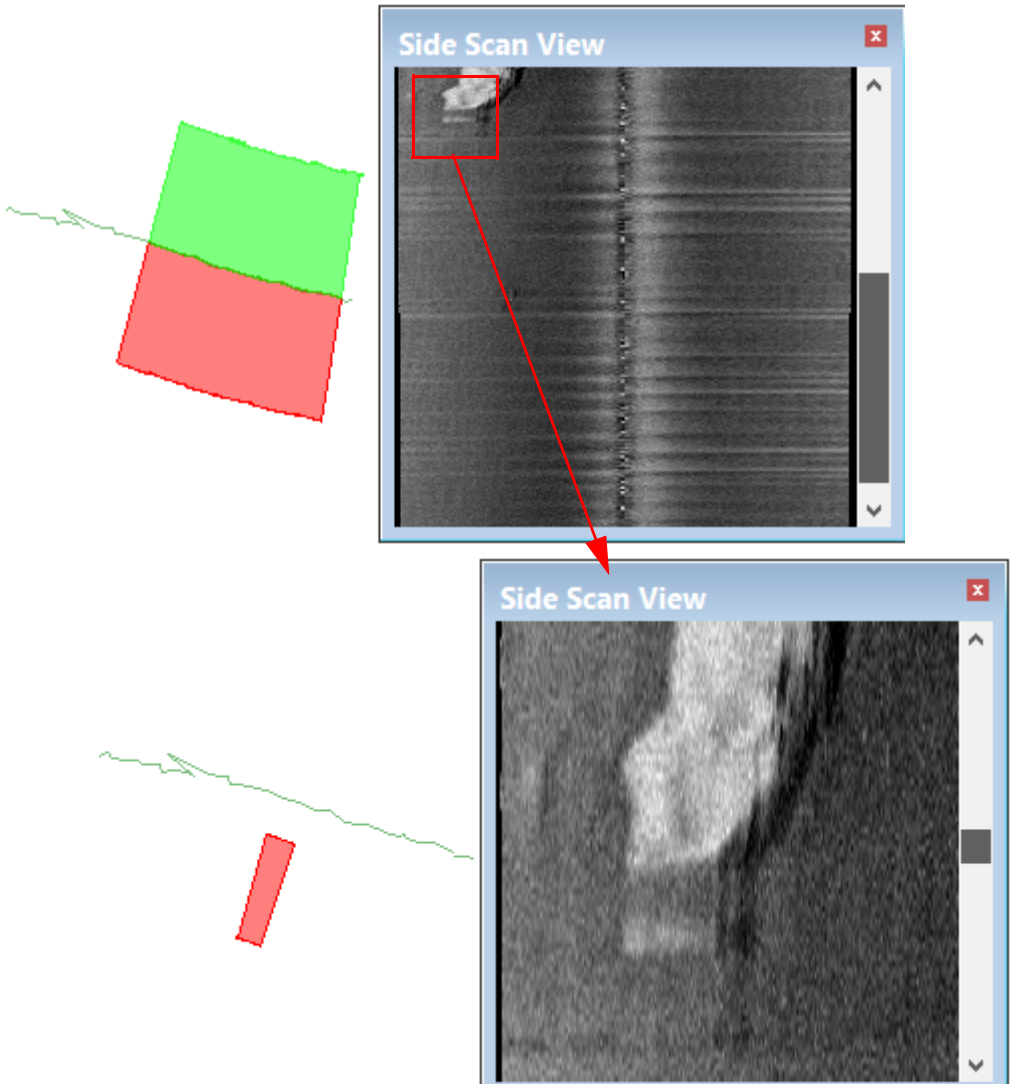
1. Select a Zoom Area command.

Menu	View > Zoom > Area
Pop-up	Area
Tool	
Key	F10

The cursor changes shape.

2. Drag the cursor over the area you want to enlarge.

The data in the zoomed area is magnified and the coverage in the Display window becomes smaller and indicates the location of the enlarged data.


Side Scan View
Side Scan View

Selection

Use the Side Scan selection tool to select a range of data. The profiles for the selected data can then be viewed and edited.

A selection can be further defined using superselection, or by specifying profile or ping numbers.

Menu	Tools > Editors > Side Scan > Select
Tool	

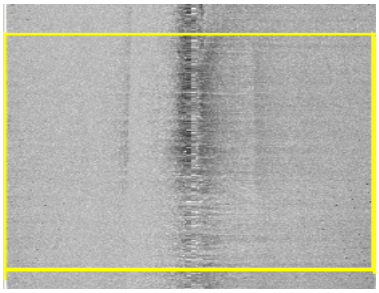
To range-select data:

1. Click the Select Side Scan button on the Side Scan Editor toolbar, or the command from the menu.

The cursor will become a cross-hair when positioned in the View (waterfall) window.

2. Press the mouse button and drag the cursor across the area that you want to select.

The selected range is shown bounded with a rectangular yellow box:



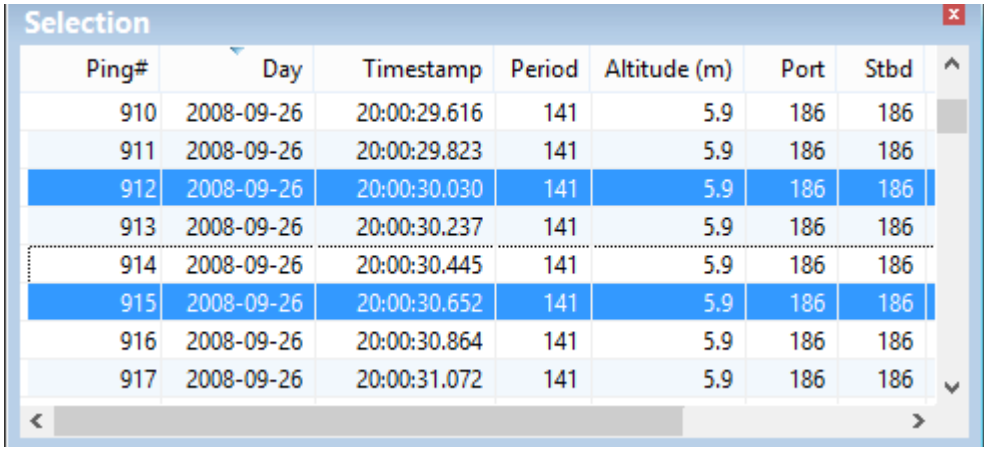
This range will remain selected until you click outside the range.

Menu	Edit > Query
Tool	
Pop-up	Query

Details of data selected in the View can be seen using the Query command.

1. Select data in the waterfall view.
2. Right-click the selected data.
3. Select the Query command from the pop-up menu.

The pings in the selection and their profile data are listed in the Selection window.



Ping#	Day	Timestamp	Period	Altitude (m)	Port	Stbd
910	2008-09-26	20:00:29.616	141	5.9	186	186
911	2008-09-26	20:00:29.823	141	5.9	186	186
912	2008-09-26	20:00:30.030	141	5.9	186	186
913	2008-09-26	20:00:30.237	141	5.9	186	186
914	2008-09-26	20:00:30.445	141	5.9	186	186
915	2008-09-26	20:00:30.652	141	5.9	186	186
916	2008-09-26	20:00:30.864	141	5.9	186	186
917	2008-09-26	20:00:31.072	141	5.9	186	186

Superselection

Superselection selects data within an already selected range of data. You can superselect a single ping or a range of pings in the Selection window and they will be highlighted in the View window.

To select a single ping from a queried selection

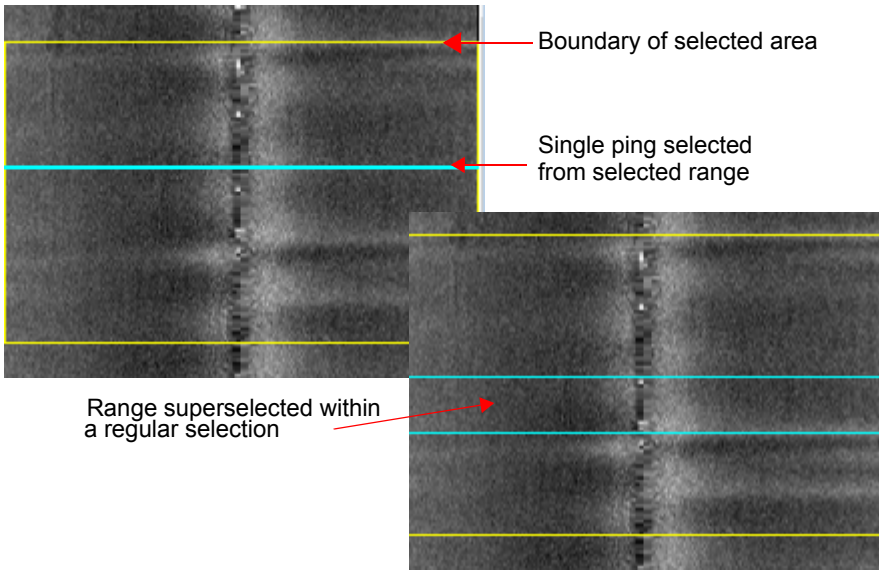
1. Click on the row in the Selection window.

The superselected ping will then be highlighted in blue within the already selected selection in the View, as illustrated below.

To select a sequential range of pings:

2. Press and hold the Shift key, and in the Selection window, click in the first row of the range and then in the last row of the range you want to select. Release the Shift key.

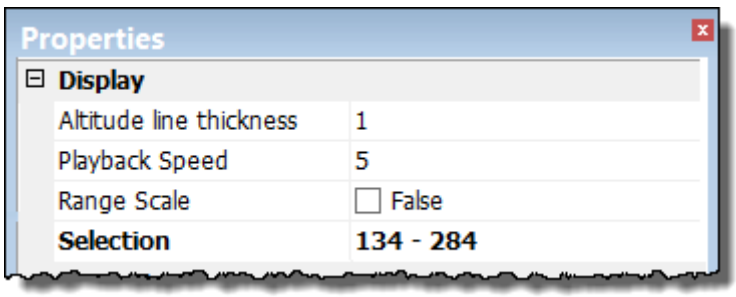
The superselected range of pings is highlighted, and in the waterfall view, a rectangle is displayed containing the range of pings.



Holding down the Ctrl key while selecting rows in the Selection window will display the individual pings as blue lines in the View window.

Select by ping number

When you make a selection in the Side Scan View, the start and end ping numbers of that selection are displayed in the Selection field of the Properties window.



To refine your selection:

1. Type the ping number, or range of numbers you want to select.

Measure Distance in Side Scan View

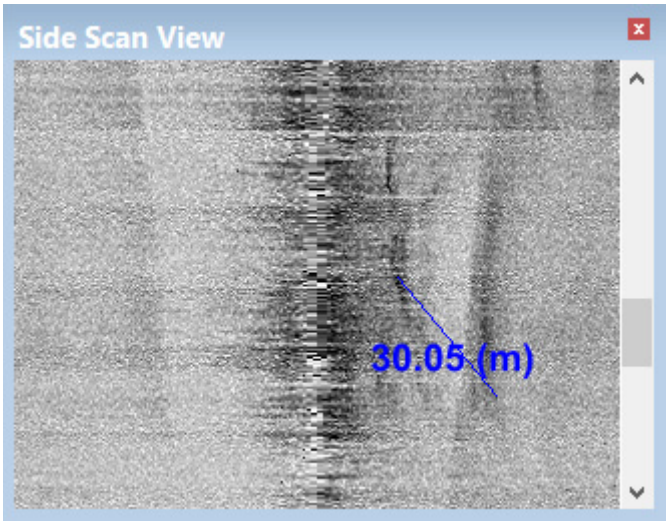
Measure distance

This tool enables you to measure distance between two points with the mouse. This feature is used in processed mode in the Side Scan View window. Distance units are determined by the settings in the Display Units tab of the Tools >Options dialog box.

Menu	Tools > Editors > Side Scan > Measure Distance
Tool	

1. Make sure the side scan display is in processed mode.
1. From the Side Scan Editor toolbar, select the Measure Distance tool.
2. Press and hold the mouse button.
3. The cursor will change to a ruler icon. 
4. Drag the cursor across the waterfall display.

As you drag the cursor across the data, the distance is displayed.



Measure Shadow

This tool enables you to determine the height of an object by measuring its shadow in the waterfall view. This feature is only available in processed mode.

Menu	Tools > Editors > Side Scan > Measure Shadow
Tool	

1. Select the Measure Shadow tool.
2. Press and hold the mouse button.
3. The cursor will change to a ruler icon. 
4. Drag the cursor *horizontally* across the display in the waterfall view.

The height measurement is shown in the side scan window.

Signal Display Window

The Side Scan Editor includes a Signal Display Window for viewing sonar intensity levels. This window provides a plot of signal intensity against across-track distance (or time, in raw mode).

The Signal Display Window can display either a mean of all sonar intensities in a selection, or else the sonar intensity values for a single ping. You can switch back and forth between mean and single profile modes using the buttons on the toolbar.

The Signal Display Window can be used in both the raw and processed modes.

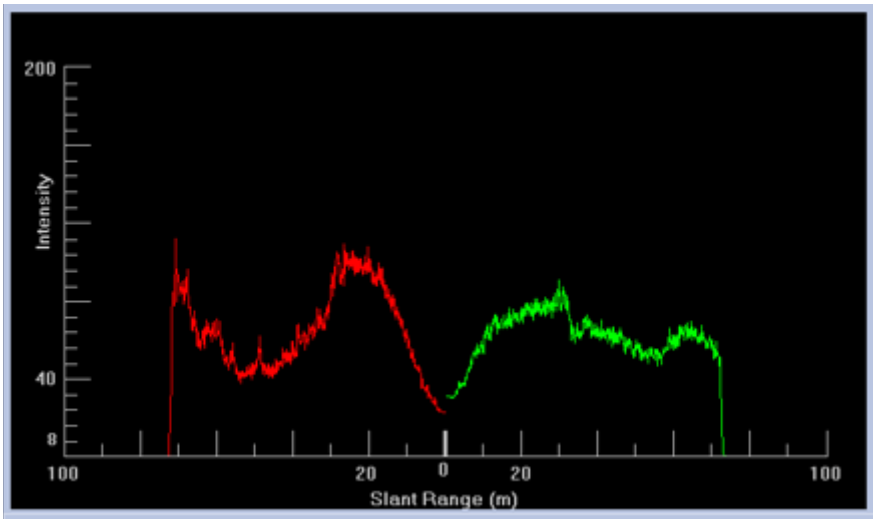
If a beam pattern has been loaded to the line in Side Scan Editor, it will be displayed as a yellow line in the Signal Display window.

Open the Signal Display Window

1. Select a range in the Side Scan View using the Side Scan Selection tool (see "SELECTION" ON PAGE 77).
2. Select the Signal Display Window tool.

Menu	Tools > Editors > Side Scan > Signal Display Window > Open/Close
Tool	

The Side Scan Signal Display window is displayed.



The vertical axis on the graph represents intensity level of the signal(s) being examined. The horizontal axis represents either time or slant-range distance depending on the display mode of the Side Scan Editor.

By default the window will display the graph of the mean values for the selected range.

When the Signal Display Window is reopened, it will remember its previous setting and display either the mean intensity values for the entire selection or the intensity values of a single profile.

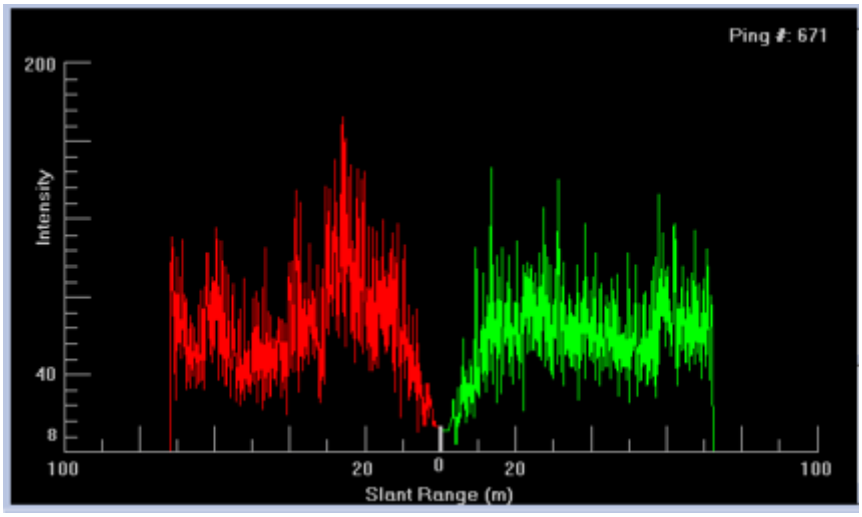
View Single Ping

Menu	Tools > Editors > Side Scan > Signal Display Window > View Single Ping
Tool	

To view the signal intensity for a single ping, instead of the mean for a range of pings selected in the Side Scan View:

1. Select View Single Ping.

The Signal Display window is refreshed to show the sonar signal intensities for the first ping in the selection in the Side Scan window. (The ping closest to the bottom of the Side Scan View.)

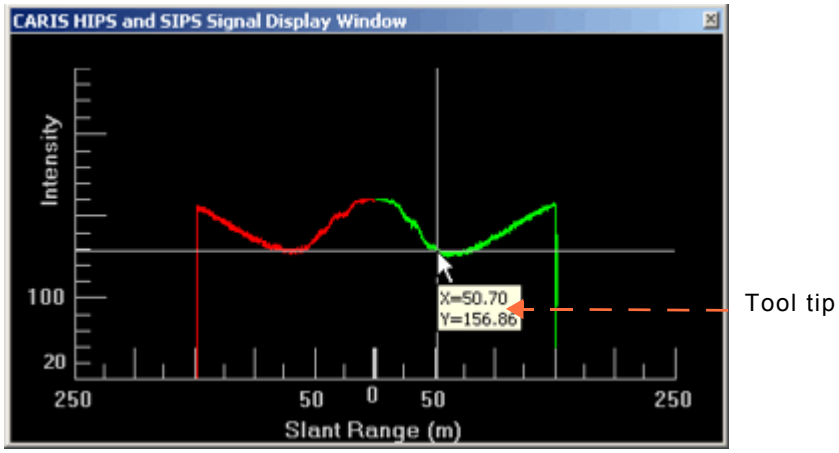


2. Use the Up arrow key to scroll up through the selected range. (Use Down to reverse direction.)

As you scroll through the selected range, the Signal Display Window will refresh to show the intensity values for each ping. The ping number is displayed in the top-right corner of the Signal Display Window in this mode. The location of the ping is also indicated by a new line within the selection. The single ping selection always defaults to the first ping in the selection.

You can view the position and specific intensity values of points by clicking on any point in the graph in the Signal Display

Window. The intensity and time/distance values for that point are shown in a tool tip.



View Selection Mean

Menu	Tools > Editors > Side Scan > Signal Display Window > View Selection Mean
Tool	

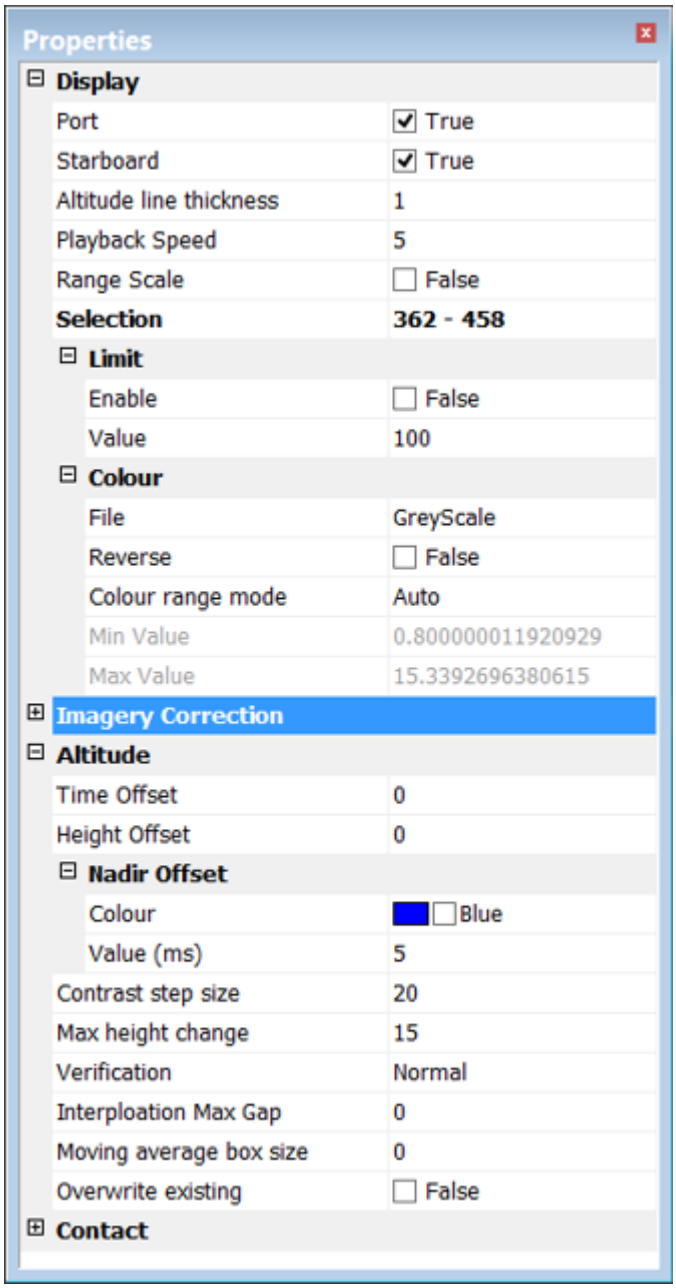
To switch the display in the Signal Display Window from the values for a single ping to the mean of the intensity values in the entire selected range:

1. Select the View Selection Mean tool.

Side Scan Editor Properties

The Properties window contains controls for display, imagery correction and altitude adjustment, as well as the properties of contacts.

The image of the Properties window below shows the properties fields which control the display of data in the Side Scan View.



"DISPLAY PROPERTIES" ON PAGE 84

"IMAGERY CORRECTIONS" ON PAGE 90

"DIGITIZING ALTITUDE" ON PAGE 86

"CONTACTS" ON PAGE 98

Display properties

These properties control the display of side scan data in the Side Scan View (waterfall view).

Property	Description
Port	If set to True, port side data is visible in the View window. If False, only starboard data is visible. Either Port or Starboard or both must be enabled.
Starboard	If set to True, starboard side data is visible in the View window. If False, only port data is visible. Either Port or Starboard or both must be enabled.
Altitude line thickness	Set the thickness of the line used to digitize the altitude of Raw data in the waterfall view. Set a value from 1-8.
Playback speed	Set the speed the Playback tools use when scrolling through pings in the Side Scan View (3 is slower, 20 is faster). See "PLAYBACK CONTROLS" ON PAGE 73.
Range scale	The range scale runs across the top and bottom of the View window. In Raw mode the range displays time intervals in milliseconds. In Processed mode, distance is displayed in the horizontal units set in the Tools >Options >Display >Units dialog box. To display scale: • Enable to check box to set to "True". (The colour of the range scale is controlled by the Measure / Offset Tools setting in the Tools > Options> Display > Side Scan Editor dialog box.)
Selection	Displays the start and end numbers of the range of pings selected in the Side Scan View. To select a specific range: • type the numbers of the start and end pings, e.g., 255-275
Limit	Set across-track display limits by adjusting the width of the ping in the Side Scan Editor View. • By default *Enable* is set to "False" so the entire width of the ping is shown in the window. • To expand or narrow the view, set to "True" and type a value. The smaller the number, the wider the ping is stretched.
Colour	• *File*: Select the display colour for the waterfall view from the drop-down list. (The default is GreyScale.) • *Reverse*: Sonar recorders have traditionally used light colours to represent high signal returns. Select the *Reverse* option to change the order in the colour map to reflect this practice. • Colour Range mode: can be set to Auto or Manual. If set to Manual, then Min and Max Values in dB need to be set. • Any values below the minimum value are assigned colour 0. Any above the maximum value are assigned colour 255.

Using the Port and Starboard Display Controls

The Port and Starboard controls in the Properties window determine which data is *visible* in the View window.

There are similar controls to apply *edits* to port, starboard or to both. These are available from Edit > Status Flags > Apply Edit To menu.

To edit the data, one side or the other must be enabled in the Properties window.

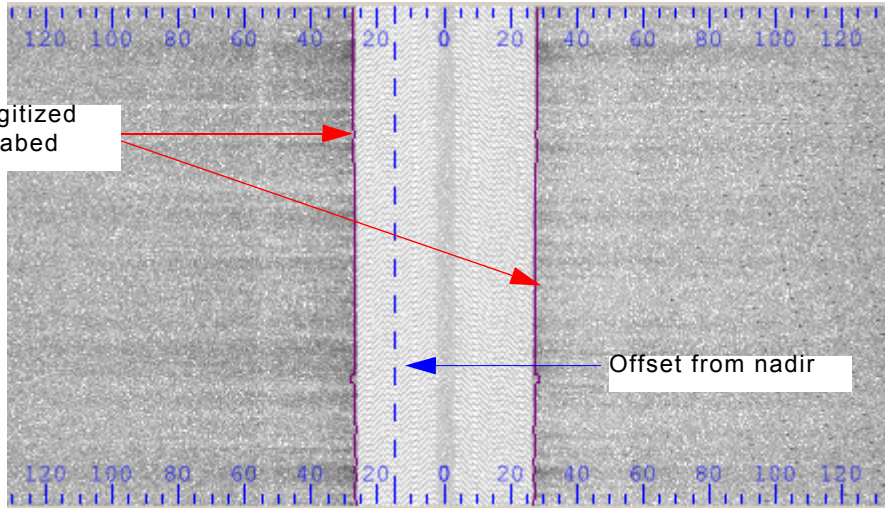
When one side of data is visible and edits are applied to the other, the following is the protocol that is followed in Side Scan Editor.

Display set to	Edit applied to	Outcome
Port Only	Starboard	Edits are *not* applied.
	Port	Edits applied to Port data
	Both sides	Edits applied to Port data
Starboard Only	Port	Edits are *not* applied.
	Starboard	Edits applied to Starboard data
	Both sides	Edits applied to Starboard data
Both enabled	Both sides	Edits applied to all data
	One side	Edits applied to the active side.

Digitizing Altitude

Before side scan data can be processed, the altitude of the sonar must be known. With some sonars, altitude is logged in the raw data files and is converted by SIPS. However, in other cases, altitude is not logged or is logged and needs to be edited. Side Scan Editor provides automated and manual tools to digitize the bottom directly from the side scan trace.

A raw side scan sonar display with a digitized seabed is shown below.



Digitization and colour maps

Sometimes there is no clear distinction between the water column and the seabed. You can compensate for this by adjusting the display of intensity values in the editor so that the seabed and water column are clearly outlined.

1. From the Properties, select a colour file that only displays two colours in the Side Scan View. An example is the default colour GreyScale.cma.
2. Use the sliders in the colour histogram to adjust the display in the editor.
3. Once the there is clear distinction between the water column and seabed use the digitizing tools to trace the first bottom return.

"AUTOMATIC DIGITIZING" ON PAGE 86

"MANUAL DIGITIZING" ON PAGE 88

Automatic digitizing

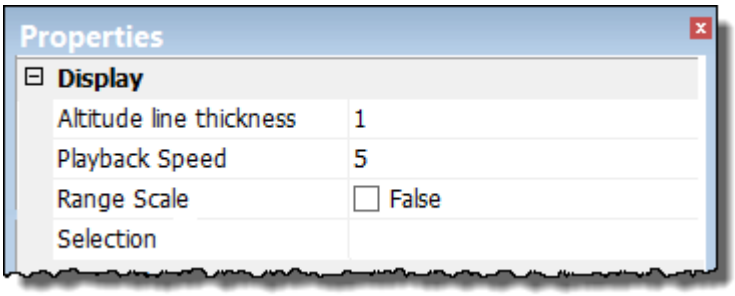
The automatic digitizing method traces the first bottom return using a sophisticated multi-pass algorithm. Automatic digitizing is most effective when there is a clear distinction between the water column and the bottom. This method can be applied to a range of pings or to the entire line.

Menu	Tools > Editors > Side Scan > Altitude > Auto Digitize
Tool	

To automatically digitize altitude:

1. Select the Auto Digitize command.
2. [Optional] Select the Clear command to remove the digitization.

Settings for automatically digitizing the side scan return are in the Properties window, as illustrated below.



Altitude line thickness

Set the thickness of the line used to digitize the altitude of Raw data in the waterfall view. (Value from 1 to 8.)

Time offset

Type a value in seconds to apply a time shift to the altitude data.

Height offset

Type a value in metres for a fish height offset.

Nadir Offset

Strong solid returns in the water column, such as surface reflections, can be compensated for by setting an offset from nadir in raw data mode.

The nadir line colour is set to Blue. To change colour:

1. Click in the Colour field and select a new colour from the colour picker.
2. Type a value in milliseconds for the Nadir Offset *Value*.

The *Contrast Step Size* and *Max Height Change* options must be set to correctly auto digitize the side scan.

Contrast Step Size

The *Contrast Step Size* is the minimum difference in contrast expected between the intensity values of the water column and

the sea floor. The Signal Display Window may be helpful in determining a good step size.

3. Set a value for the step size.

Max Height Change

The *Max Height Change* is the maximum allowable ping-to-ping change in detected bottom position (in pixels). It is a restriction on the variation anticipated in detected bottom position. It could be visualized as error bars guiding the algorithm along the bottom.

4. Set a percentage for the *Max Height Change* value. The default is 15.

Verification

The verification option tests the detected point to see if its intensity is consistent with nearby values further across-track that should also represent sea floor.

Its purpose is to detect and ignore anomalous points in the water column (for example, noise from equipment, reflections off the surface, marine life), by checking to see if there is water rather than sea floor on the other side of a detected point.

5. Click in the Verification field and select a desired strength-level for the verification from the drop-down list. Choices range from “None” to “Very Weak” to “Very Strong”. The default is “Normal”.

Interpolation Max Gap

The *Interpolation Max Gap* option fills along-track gaps in the detected bottoms.

6. Select the *Interpolation Max Gap* field and type a value for the size of the gap to be filled.

Moving Average box size

This filtering option smooths the digitized line.

7. Click in the Moving Average box size field and type the number of pings to be used for averaging during digitizing.

Overwrite existing

8. Set to “True” to enable re-digitization of selected areas without clearing. If set to “False”, only zero-altitude/cleared fish heights will be available for digitization.

Manual digitizing

Add altitude lines by selecting areas along the first bottom return.

Menu	Tools > Editors > Side Scan > Altitude > Manual Digitize
Tool	

1. Select the Manual Digitize command.
2. Press and hold the mouse button and drag the cursor along the bottom contact.

You can also use the space bar to move along the track line while in digitizing mode.

As you drag the cursor, a digitizing line is drawn in the window. When you digitize one side, the other side is automatically mirrored.

3. Alternatively, create a point-to-point line by clicking carefully along the first return.
4. Cancel the Manual Digitize tool when finished.

At any time during the digitizing process, you can select the Clear tool to remove the digitization.

Interpolate Selection

Interpolation digitizes a straight line between the fish height values found at the start and end of the selection. Any altitude data within the selection is overwritten during this operation.

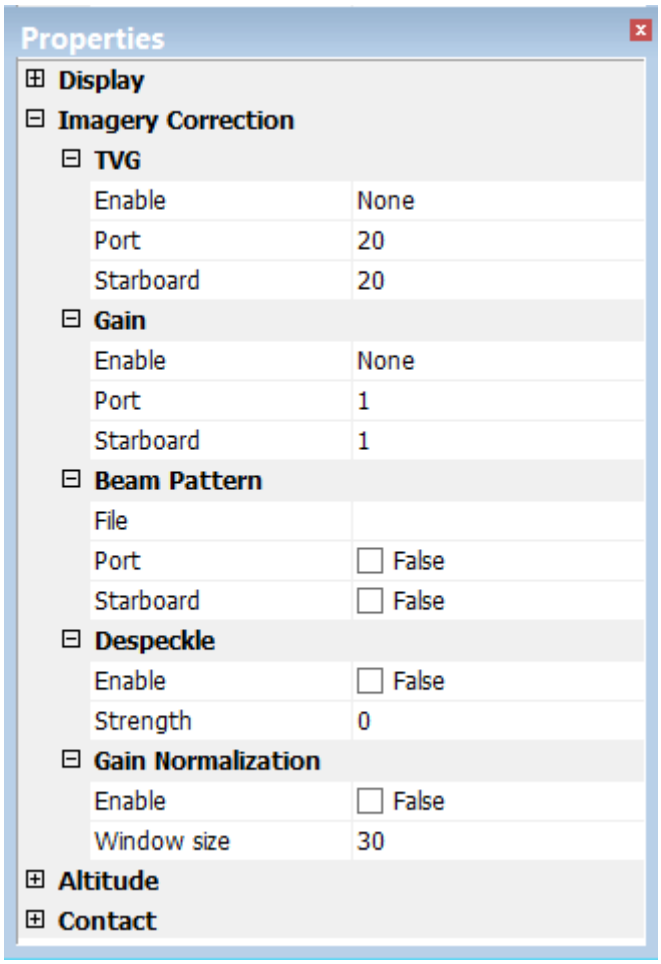
1. Use the Side Scan Selection tool to select an area of raw mode data that has been digitized. (See "SELECTION" ON PAGE 77)
2. Select the Interpolate Selection command.

Menu	Tools > Editors > Side Scan > Altitude > Interpolate Selection

The pixels between the digitizing points are interpolated and the digitizing lines are smoothed.

Imagery Corrections

Imagery processing controls are found in the Properties window. Correction settings can also be configured in the Create Mosaic dialog box under Imagery Corrections and Advanced Options.



See "Time-Varying Gain" on page 90

See "Gain" on page 91

See "Beam Pattern Correction" on page 91

See "Despeckle" on page 92

See "Gain Normalization" on page 93

Time-Varying Gain

Time-Varying Gain (TVG) Correction is necessary because of the attenuation that occurs due to absorption and spreading as the sonar beam travels back to the receiver. Since these returns are received over a predictable and constant time period, a time-varying curve can used to increase gain in order to compensate for the decayed sonar return.

In post-production, it may be necessary to augment the TVG correction applied during acquisition. The process entails applying gain varied by time values so that inner-most samples have the least gain and the outer-most samples have the highest gain correction.

TVG is calculated as:

```
10 ^ (range * TVG / 20 / 100)
```

where TVG is this value, and range is the distance from nadir.

TVG Correction is applied to the lines in the Display window.

1. In the Properties window, click in the Enable field and select a value from the list: None, Linked, Both, Port Only or Stbd Only.
2. In the Port and Starboard fields, set the percentage of TVG to apply.

Gain

Apply a uniform gain correction without any time-dependent adjustments using the Gain controls.

The calculation for gain correction is:

```
corrected = original * 10 ^ (gain / 20)
```

where gain is the value from the property field.

1. In the Properties window, click in the Enable field and select a value from the list: None, Linked, Both, Port Only or Stbd Only.
2. In the Port and Starboard fields, set the percentage of gain to apply.

Beam Pattern Correction

Beam pattern correction uniformly removes angular artifacts from sonar systems. Correction is applied using a beam pattern file.

In Side Scan Editor beam pattern files can be created from a range of pings selected in the waterfall view or from a selected line. Beam pattern files can also be created from side scan data without having the editor open, from a line or lines selected in the Display window.

Once the beam pattern file is created it can be applied on the fly using the settings in the Properties window.

Create Beam Pattern File

Menu	Process > Beam Pattern > Create > SIPS Side Scan

To create a beam pattern file in Side Scan Editor:

1. Select the Create Beam Pattern command.
2. In the *Source* field, select Selection.
3. Click Browse and set a name and path for the file.
4. Click **OK.**

To create a beam pattern file from selected pings in the Side Scan view:

Menu	Process > Beam Pattern > Create > SIPS Side Scan

1. Select a range of pings in the waterfall view.
2. Select the Create Beam Pattern command.
3. In the *Source* field, select Selection.
4. Click Browse and set a name and path for the file.
5. Click **OK.**

The file is saved with extension `*.bp`.

Side Scan beam pattern files can also be created from more than one line. See "CREATE SIPS SIDE SCAN BEAM PATTERN FILE" ON PAGE 411.

Apply Beam Pattern Correction

Once a beam pattern file is created, it can be applied to data during mosaic creation (see "NEW MOSAIC" ON PAGE 417) or in Side Scan Editor using fields in the Properties window.

To apply Beam Pattern Correction in Side Scan Editor:

1. Select a line in the Display window.
2. In the Beam Pattern section of the Properties window, click the Browse button in the *File* field and select the appropriate *.bp file.
3. Enable the check boxes (set to "True") to apply the correction to Port or Starboard or both.

A loaded beam pattern can be seen in the Signal Display window, where it will appear as a yellow line. See "SIGNAL DISPLAY WINDOW" ON PAGE 80.

Despeckle

Despeckling uses a calculated value based on neighbouring intensity levels to replace the current pixel's intensity if it is outside the specified range. The value calculated from the neighbouring intensities can be derived by one of two methods mean or median.

Despeckling imagery is a visual process, consisting of adjusting the despeckle values and viewing the display until the desired results are obtained.

The following steps are used for despeckling images:

1. Select a track line.
2. Select the check box (set it to "True") to apply Despeckle.
3. Set a Strength value.

The line is filtered to remove noise from the sonar image.

Angle-Varying Gain

Angle-Varying Gain (AVG) correction removes the angular response of sediment from the imagery, normalizing the mean angular intensities with a moving average filter.

1. Select the number of across track samples to include in the moving average by setting a value in the *Window Size* field.

Gain Normalization

Gain Normalization removes the angular response of sediment from the imagery, normalizing the mean angular intensities, ping-by-ping, with a moving average filter. The number of pings set determines the size of the "window" used to calculate the moving average. For example, an average based on a window size of 100 includes 50 pings before and after the selected ping.

Gain Normalization is applied to lines in the Display window.

1. In the Properties window, select the Enable check box to set the value to "True".
2. In the *Window size* field, type the number of adjacent pings to include in the moving average window.

Edit Side Scan Data

Data can be accepted and rejected in Side Scan Editor as in other HIPS and SIPS editors, except that you select a ping rather than an individual sounding or beam.

If navigation sensor data has been edited with Navigation Editor it may be necessary to recompute towfish navigation. (See "RECOMPUTE TOWFISH NAVIGATION" ON PAGE 96.)

If the altitude of the sonar is not already logged with your data, it must be digitized. (See "DIGITIZING ALTITUDE" ON PAGE 86.)

Accept/Reject/Query

You can change the status of an entire ping, or only the status of the starboard or the port side.

> Edits are only applied to data that is visible. If Port or Starboard data is turned off in the Properties, it will not be edited.

Menu	Tools > Editors > Side Scan > Select
Tool	

1. Use the Side Scan Selection tool to select the range of pings from waterfall view to be edited (see "SELECTION" ON PAGE 77).

Menu	Edit > Status Flag > Apply Edit to
Tool	On Status Edit toolbar 

2. Right-click and select Query from the pop-up menu to view the selected data in the Selection window.
3. From the Edit >Status Flag menu or Status Edit toolbar, select either Port, Starboard or Both to apply the edits to.

Menu	Edit > Status Flag > Reject/
Tool	On Status Edit toolbar 

4. Select the Reject command to reject all the selected data. This command is also available from the pop-up menus in the View and Selection windows

The status of the ping or range of pings is set to Rejected and are excluded from further processing in SIPS.

To view Rejected data use the Display Filter function. See "VIEW SOUNDINGS STATUS" ON PAGE 332 for information.

Accept

Rejected data is not deleted and can be reverted using the Accept command. To make rejected data available for further processing:

Menu	Edit > Status Flag > Accept
Tool	
Pop-up	Accept

1. Select the rejected pings (see "SELECTION" ON PAGE 77).
2. Apply the Accept command.

Previously rejected data is now flagged as accepted and is visible again in the Side Scan View.

Recompute Towfish Navigation

To compute towfish navigation from the ship's navigation requires a horizontal distance and direction from the ship's towpoint location to the towfish. The towpoint is defined in the vessel file.

- **Horizontal distance**. Recorded horizontal layback data or actual cable length both share the same data structure in SIPS called SSS Cable Out. SIPS can distinguish between the two by the towfish sensor depth data. If the sensor depth data is zero, the horizontal layback is used. If the sensor depth data is non-zero it will be combined with the cable out data using trigonometry to produce horizontal distances. All distances to the towfish are applied from the towpoint position on the ship as defined in the vessel configuration file.
- **Direction**. The direction from the ship's towpoint location to the towfish is derived using a "follow-the-dog" method. The average of the first 30 seconds of the ship's "course-made-good" is used for the initial direction. Thereafter, each successive position computed for the towfish occurs along the direction from the ship's next position to the previous towfish position.

If horizontal layback or tow cable length data is not available, or if the Recompute Towfish Navigation step is not executed, then the recorded towfish navigation data converted into HIPS and SIPS is used for generating the positions of contacts and for mosaics.

If towfish navigation is not available at all then the position of the side scan imagery data is assumed to be the same as the ship navigation data.

Navigation data for towed sensors can come from the following sources:

- If computed during data logging and stored in the raw side scan format, it can be converted directly into the SIPS towfish navigation data structure. The Recompute Towfish Navigation process is therefore not required.
- It can be computed by SIPS if specific sensor data is available. SIPS requires ship navigation data with cable out and towfish depth to compute towfish navigation, or ship navigation with horizontal layback.

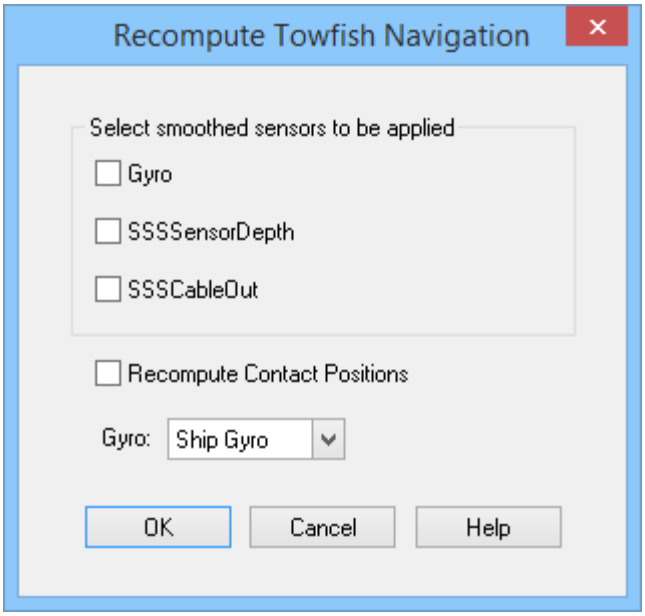
Side Scan Editor does not have to be open to use the Recompute Towfish navigation function.

To recompute towfish navigation:

1. Select one or more towfish track lines.
2. Select the Recompute Towfish Navigation command.

Menu	Process > Compute > Towfish Navigation

The Recompute Towfish Navigation dialog box is displayed.



Towfish sensor data can also be smoothed in the Attitude Editor (see "DATA OPTIONS" ON PAGE 42) and this data can be applied when the towfish navigation is computed.

3. Check the box for the sensor data to apply.
4. Select the *Recompute Contact Positions* check box to update the contact positions based on the new navigation data.
5. Select either *Ship Gyro* or *Ship Course Made Good* to be applied.
6. Click **OK**.

If contacts are contained in the track lines where towfish navigation has been updated, you must update the contact positions to match the new navigation (see "RECOMPUTE CONTACT POSITIONS" ON PAGE 108).

Contacts

A contact is the representation of an object or feature on the sea bottom (for example, a wreck, or a pipeline) that is visible in the sonar data. Contacts are created using the Add point contact or Add line contact commands.

In HIPS and SIPS, you can place individual contacts directly in the waterfall view. A contact record is created for each contact, and each contact has a unique identifying number. This information is stored in the *.hips file. This and other related information can be viewed and edited in the Attributes window.

Contact images are automatically created and maintained in the *.hips file, and can be viewed in the Image Preview window.

Contacts can be queried and edited. Contact information can be exported. (This is described in Export "Contacts" on page 506 of the User Guide.)

"Create Contacts" on page 98

"Edit Contacts" on page 101

"Contact Images" on page 104

"Recompute Contact Positions" on page 108

Create Contacts

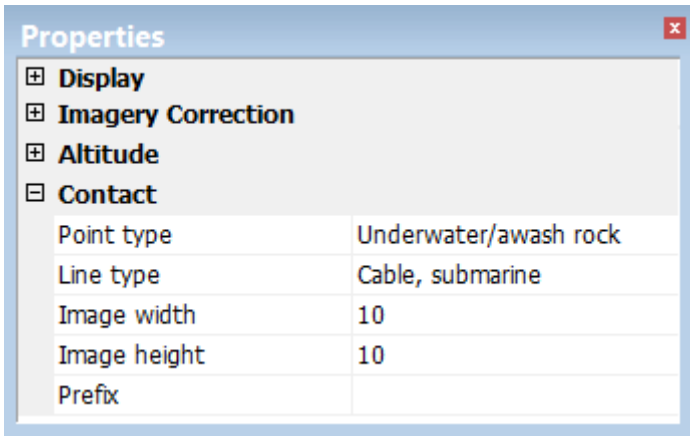
Point type contacts can be created for objects such as rocks, underwater posts or piles, as well as wrecks.

Line contacts can be created for continuous objects such as pipelines or underwater cables.

Once created, the contact type cannot be changed.

For example, a point contact created as an underwater rock cannot be changed to a pile. Instead, it must be removed and recreated as the correct type.

Characteristics of each contact are set in the Properties window.



1. Set default contact properties in the appropriate fields in the Properties window.
   - *Point Type*: Click in the field and select a type, for example, wreck or pile, from the drop-down list. These types are defined in the Catalogue Editor, and stored in the HIPS and SIPS Features catalogue.
   - *Line Type*: Click in the field and select a type, for example, a cable or pipe line, from the drop-down list. These types are also defined in the Catalogue Editor, and stored in the HIPS and SIPS Features catalogue.
   - *Image Width*: Type a value for the width of the contact image in pixels. This determines a dimension of the image in the Image Preview and well as an exported image of the contact.
   - I*mage Height*: Type the height for the contact image in pixels. This determines a dimension of the image in the Image Preview and well as an exported image of the contact
   - *Prefix*: An optional prefix, taken from the line number, can be automatically be added the contact name on export.

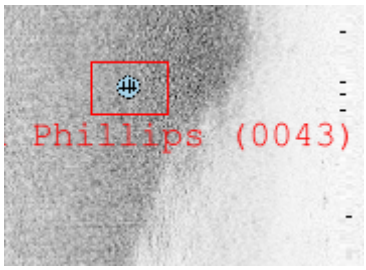
Point Contact

Menu	Tools > Editors > Side Scan > Contacts > Add Point Contact
Tool	

To add a point contact:

1. Set the Side Scan View / waterfall view to Processed.
2. Select the Add Point Contact tool.
3. Click in the waterfall view where you want to place the contact.

A contact marker is placed in the Side Scan Editor at the point where you clicked. The type of contact is indicated by the symbol displayed. The contact is identified with the name of the Current User and a four-digit number (series starts at 0001).

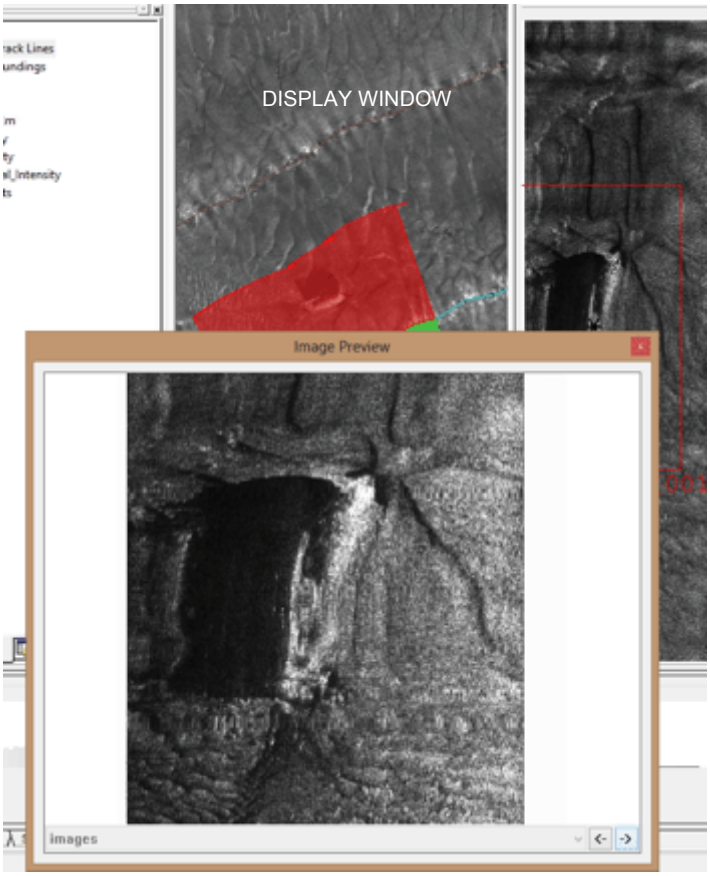


A bounding box is displayed around the contact. This box always snaps to the contact so that the contact point is in the middle of the box. The bounding box represents the image area of the contact.

Menu	File > Save
Tool	

4. Save data.
5. Open the Image Preview window.
6. Select the new point contact.

The image of the contact is displayed in the Image Preview window, as in the screen capture below. In the background of this screen-shot, the contact is visible in the waterfall view and the selected part of the line can be seen in the Display window.



You can adjust the size of the image using the *Image Width* and *Image Length* attributes.

Line Contact

Menu	Tools > Editors > Side Scan > Contacts > Add Line Contact
Tool	

To insert a line contact record:

1. Select Add Line Contact.
2. Click the point in the waterfall view where you want to insert the contact.
3. Click another section of the view.

The two points are joined by a line.

4. Continue adding line segments as needed to represent the contact.

5. Select **Edit Line**> **Remove last** to delete the line segment just created.
6. To escape the Line Contact tool, right-click the mouse and select **Edit Line**> **End** from the pop-up menu.

The contact lines are drawn in the Side Scan View. Each line is identified with a number, and each type of line is drawn in a different style.

The contact is identified with the name of the currently logged in user, plus a four-digit number (starting at 0001).

Repeat New Feature

When the Repeat New Feature tool is activated, you can add any number of contacts of the same type and attributes, without having to set each one separately. To add multiple similar contacts:

Menu	Tools > Editors > Side Scan > Contacts > Repeat New Feature
Tool	

1. Select the Repeat New Feature command.
2. Use the Add Point Contact tool to place the contacts, one after the other, into the Side Scan Editor.

The Repeat New Feature tool button will remain depressed while you add contacts. To change to a different type of contact or to change contact data, deactivate the Repeat New Feature.

Edit Contacts

You can reposition both line and point contacts and update the data associated with a contact. As well, contacts can be queried and edited using the tools described in "Edit Side Scan Data" on page 94.

You cannot change the type of an existing contact. For example, a point contact created as an underwater rock cannot be changed to a pile. Instead, remove the contact and create a new one with the correct type set in the Properties.

"Select Contacts" on page 101

"Change Contact Attributes" on page 105

"Use Measure tools" on page 106

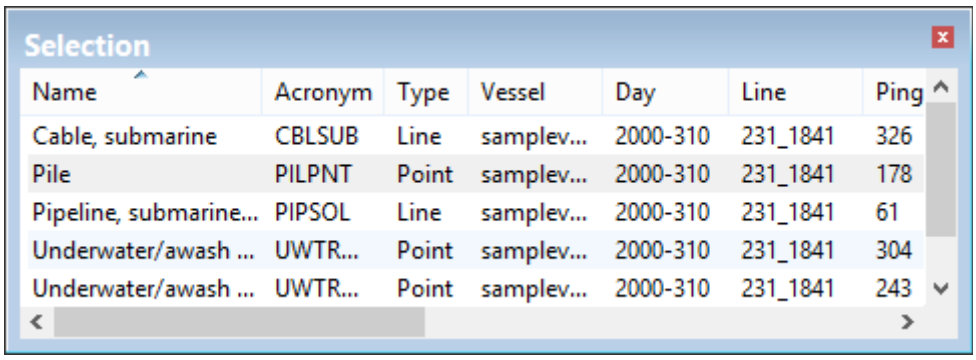
"Reposition Contacts" on page 107

Select Contacts

The Select Contact tool is used to select point or line contact(s) in the waterfall view.

Menu	Tools > Editors > Side Scan > Contacts > Select Contact
Tool	

1. To select a *single* contact, click on or drag a selection around the contact in the waterfall view so that it becomes highlighted. Clicking on any part of a line will select the entire contact.
2. To select a *number* of contacts, drag the selection across a large enough area so that the contacts within it are highlighted.



When a number of contacts are listed in the Selection window, the superselected contact will be highlighted in the waterfall view.

As well, the coordinates of the superselected contact are displayed in the Coordinates window.

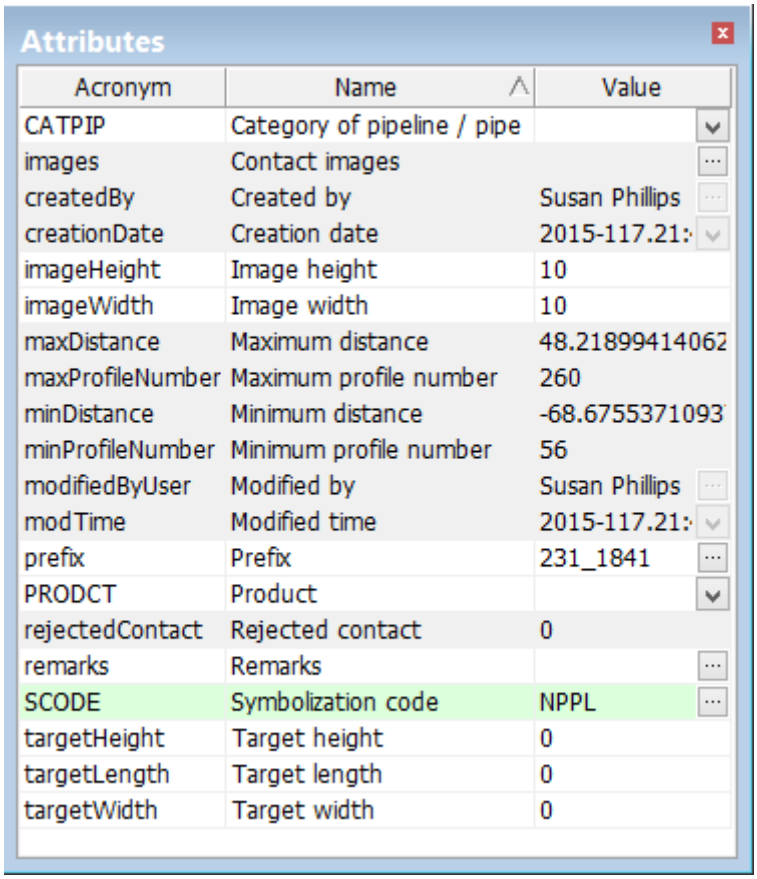
Contact Attributes

1. Enable the Select Contact tool.
2. Select a contact in the waterfall display.
3. Open the Attributes window.
4. Select a contact in the Selection window.

The attributes of the contact are displayed in the Attributes window. Fields that uniquely identify the contact are not editable and are highlighted in grey.

5. Type new information in the editable fields, or click **Browse** to select new values.

The following image shows a sample of attributes for a point contact:



Different types of contacts have various attributes, such as the following:

Attribute	Description
Creation date	Date the contact was created.
Image Height and Width	Size of the area around the contact image (set in the Properties (in pixels)
Contact images	Lists any embedded images, and any external raster images linked to the contact. External images are located in the folder set for the External Files variable in the Environment tab of the Tools > Options dialog box.
Maximum/ minimum distance	For a line contact, shows the distance of the first and last point of the line in from nadir. If a point contact both values will be the same.
Minimum profile number / Maximum profile number	For a line contact, profile numbers show the starting and ending position of the line in the waterfall view. If a point contact both values will be the same.
Modified time	Date the contact was modified.

Attribute	Description
Modified by	The name of the Current User logged into to the computer (as seen in Tools > Options).
Prefix	The prefix is usually the line number. It is combined with the contact number to generate a 12-character key for database applications.
Rejected contact	Status of contact as defined in Catalogue Editor. 0 = not rejected, 1 = rejected.
Remarks	Editable text entered by user
Target Height and width	Size of the contact in the waterfall view. Use the Measure Distance and Measure Shadow tools to determine these dimensions.
Symbolization code	A CARIS Feature code used for symbology of contacts when they are exported to CARIS Map.
Created by	The name of the Current User logged into to the computer (as seen in Tools > Options).

Contact Images

When a contact is created, SIPS automatically captures the contact image from the waterfall view and stores it as a raster attribute of the contact feature object in the *.hips file.

Having this image embedded in the *.hips file means that any user opening the project will be able to view the contact image in the Image Preview window.

Each raster image is saved with a unique name in the following format.
`<linename>_<YYMMDD><HHMMSS>_<contact#>_<type>`

This naming format is also followed when the image is exported, and the image name is displayed in the Attributes window for the related contact.

(Other contact information can be viewed and edited in the Attributes window. See "EDIT CONTACTS" ON PAGE 101.)

To view a saved contact image:

1. Open the project containing the contact.
2. Open the Image Preview window.
3. Select the contact in the Display window.

Its image will be displayed in the Image Preview.

Link External Images

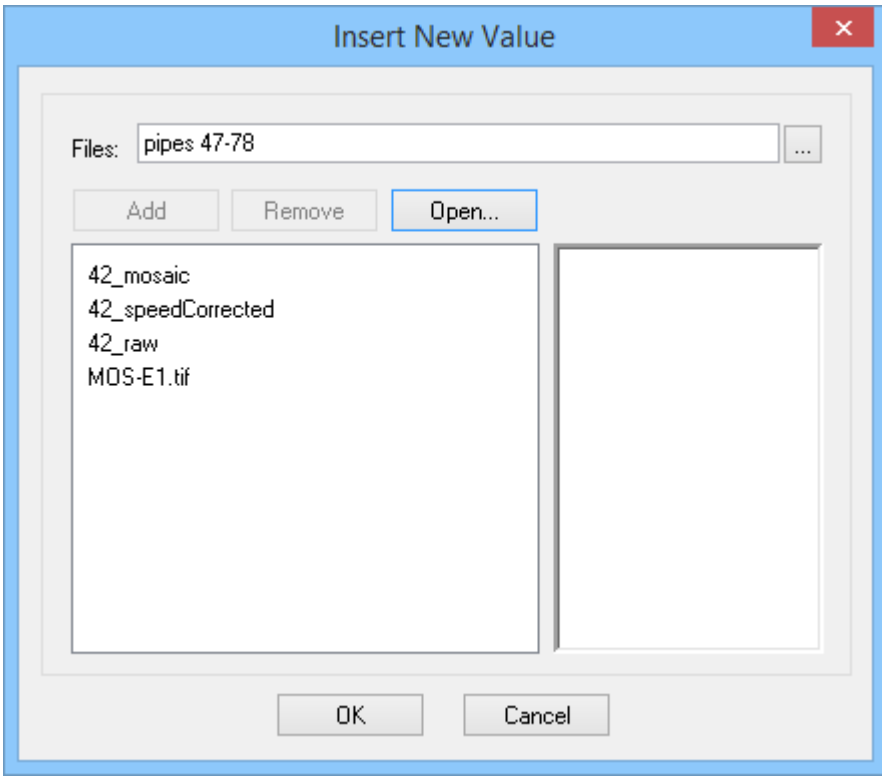
External image files can also be linked to contacts, and can be viewed in the Image Preview.

To add an external image to a contact:

1. Select the contact.

2. Open the Attributes window.
3. Click **Browse** in the Contact Images field.

The Insert New Value dialog box is displayed.



4. Click **Browse** to locate the desired external file.

The location of external images is set by the External Files variable in the Environment tab of the Tools > Options dialog box.

5. Click **Add** to add the file to the list. Click **OK**.

The Contact images field in the Attribute window will list the new file, which can be opened and viewed in the Image Preview when the contact is selected again.

Change Contact Attributes

You can change attribute information that is currently associated with a selected contact.

The contact type cannot be changed. For example, a Point contact created as an underwater rock cannot be changed to a pile simply by changing the Point type property. Instead, remove the incorrect contact and recreate it as the correct type.

1. Open the Attributes window.
2. Select a contact in the Selection window.

The attributes of the contact are displayed in the Attributes window. Fields that uniquely identify the contact are not editable and are highlighted in grey.

3. Type new information in the editable fields, or click **Browse** to select new values.

Different types of contacts have various attributes, such as the following:

Attribute	Description
Line	Line identification number (used for point contacts)
Created by	The name of the Current User logged into to the computer (as seen in Tools > Options).
Creation date	Date the contact was created.
Modified time	Date the contact was modified.
Modified by	The name of the Current User logged into to the computer (as seen in Tools > Options).
Profile distance	Location in relation to nadir.
Rejected contact	Status of contact as defined in Catalogue Editor. 0 = not rejected, 1 = rejected.
Remarks	Editable text entered by user
Image Height and Width	Size of the area around the contact image (set in the Properties (in pixels)
Target Height and width	Size of the contact in the waterfall view. Use the Measure Distance and Measure Shadow tools to determine these dimensions.
Minimum profile number / Maximum profile number	If it's a line contact, profile numbers show the starting and ending position of the line in the waterfall view. If a point contact both values will be the same.
Value of sounding	An S-57 attribute for the specific type of contact. Contains the value of the measurement of a sounding relative to the chart datum.
Contact images	Lists any embedded images, and any external raster images linked to the contact. External images are located in the folder set for the External Files variable in the Environment tab of the Tools > Options dialog box.
Symbolization code	A CARIS Feature code used for symbology of contacts when they are exported to CARIS Map.

Use Measure tools

4. To update the contact *width* information, choose the Measure Distance tool and press and hold <Ctrl> while dragging the cursor across the area of the contact to be measured.

Menu	Tools > Editors > Side Scan > Measure Distance
Tool	

If you are measuring in the SIPS waterfall view, the value in the Image Width field will reflect the measurement as you are making it. If measuring in the Zoom view, the value will be updated when the mouse button is released.

5. To update the length information for the contact, select the Measure Distance tool and press <Alt> while dragging the cursor across the contact area.

Menu	Tools > Editors > Side Scan > Measure Shadow
Tool	

6. To update the Contact Height field, select the Measure Shadow tool and press <Ctrl> while dragging the cursor across the contact area.

The contact information is updated.

Reposition Contacts

Use the Move Contact tools to reposition point or line contacts within the Side Scan View.

Menu	Tools > Editors > Side Scan > Contacts > Move Contact
Tool	

1. To move a selected contact, select the Move Contact tool.
2. Position the cursor on top of the selected contact. Press and hold the mouse button. This will turn the highlighted contact into a red box with direction arrows.
3. Drag the contact to a new position.
4. Deactivate the Move Contact tool to finalize the new position of this contact. You are returned to the Select Contact tool.

You can use the Move Contact tool to move an entire line contact, or to adjust or realign part of the line.

5. To move a selected line contact, position the cursor on any part of the line, press and hold the mouse button, and drag the whole line to a new position.
6. To realign *part* of a selected line, click on the line. This will highlight nodes which will appear as squares on the line.
7. Click on the node to be adjusted, which will turn the node square red. Position the cursor on the node. Press and hold the mouse button, and drag it to a new position. This will pull the attached lines with it.

Use the Measure Distance and Measure Shadow tools on the Side Scan Editor toolbar to update the contact information. See "USE MEASURE TOOLS" ON PAGE 106

Purge rejected contacts

Contacts that have been rejected during editing can be restored using the Accept command, as for other data. If you want to permanently remove the contact from the project, use the Purge command.

To delete a rejected contact from the data file.

1. Use the Select Contact tool to select the rejected contact.
2. Select a Purge Rejected Contacts command.

Menu	Tools > Editors > Side Scan > Contacts > Purge Rejected
Tool	 Side Scan Editor toolbar

The rejected contact is deleted.

See "EDIT SIDE SCAN DATA" ON PAGE 94 for more information.

Recompute Contact Positions

Update the geographic positions of contacts after towfish navigation has been recomputed for a line or group of lines.

1. Select the ship or towfish track lines layer in the Layers window.
2. Select one or more track lines.
3. Select the Recompute Contact Positions command.

Menu	Process > Compute > Contact Positions

Position information for contacts along the selected track lines are updated to match the new towfish navigation.

The start time, end time and elapsed time for the recompute process is reported in the Output window. Any error in the processing will be reported there also.

View contacts while scrolling

If you want your contacts to remain visible in the waterfall display while scrolling along a track line or using the Playback functions.

1. Select Side Scan Editor from the Tools > Options > Display tab.
2. Select the *Display contacts during playback* check box, and click **Apply**.

6

Process Single Beam Data

Processing of Single Beam data follows a similar workflow to that of multibeam, from creation of projects to processing and merging of data to creating a surface to select soundings or add contours.

Single Beam Editor contains data cleaning tools for data converted to HIPS from single beam echo sounding surveys. The Data can be cleaned both interactively and with filters, and the display of data can be scaled horizontally and vertically, and sounding depths can be adjusted.

In this chapter...

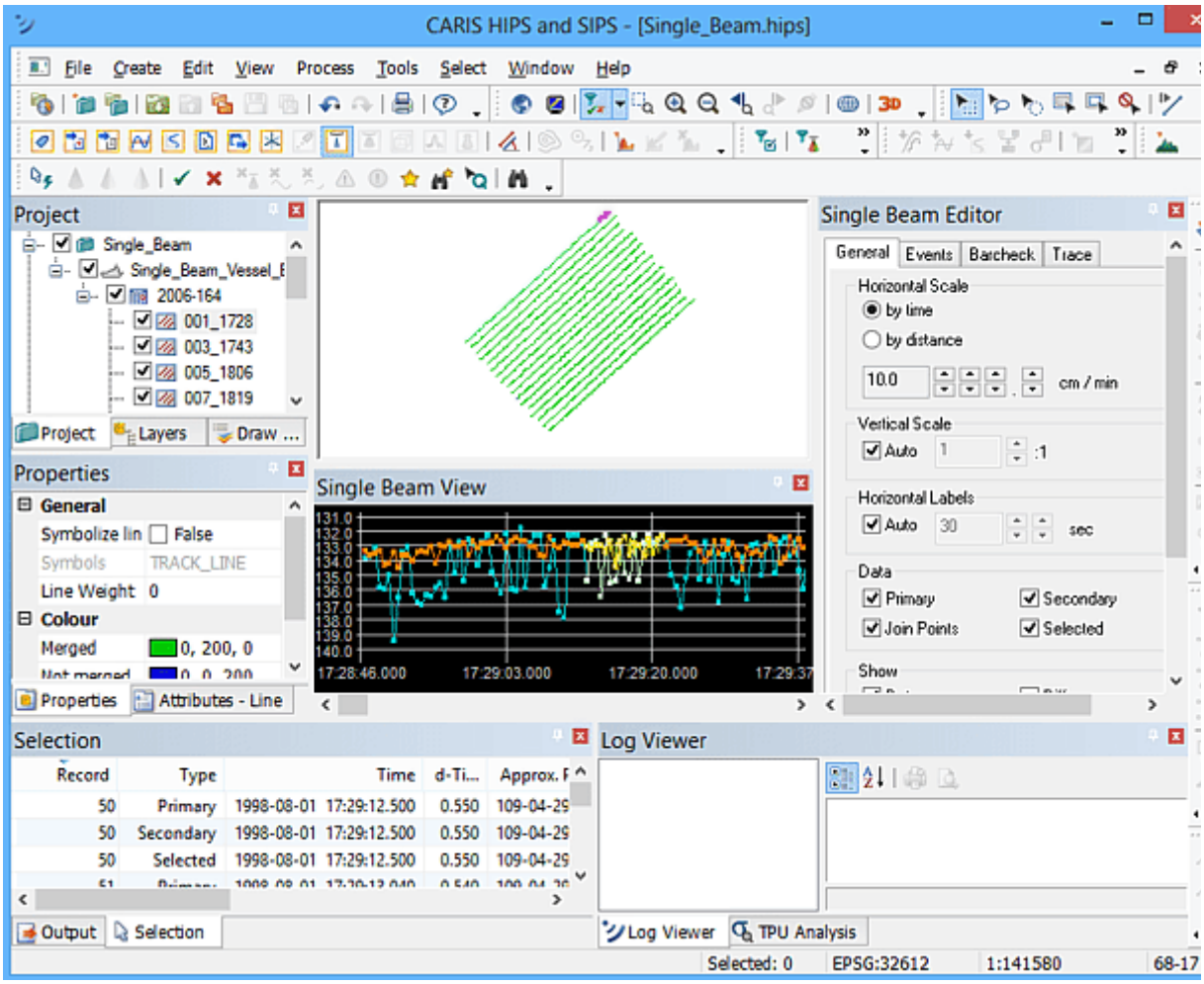
Single Beam Editor Interface

Menu	Tools > Editors > Single Beam
Tool	

To open the Single Beam Editor:

1. Select a survey line in the Display window.
2. Select the Single Beam Editor command.

The Single Beam Editor is opened in the HIPS interface.



The Single Beam Editor window is located (by default) below the Display window. The editor can be un-docked and moved to another location like other windows in HIPS.

The display and events options are controlled from the Single Beam Editor window.

The soundings are shown in the editor as dots connected by lines. You can set the horizontal scale to represent distance or time. Distances are derived from the navigation data. The first sounding with an associated position is considered the start of the line. All subsequent distances are computed from this point using sounding positions interpolated from navigation data.

When the horizontal scale is by time, the sounding time stamps are used.

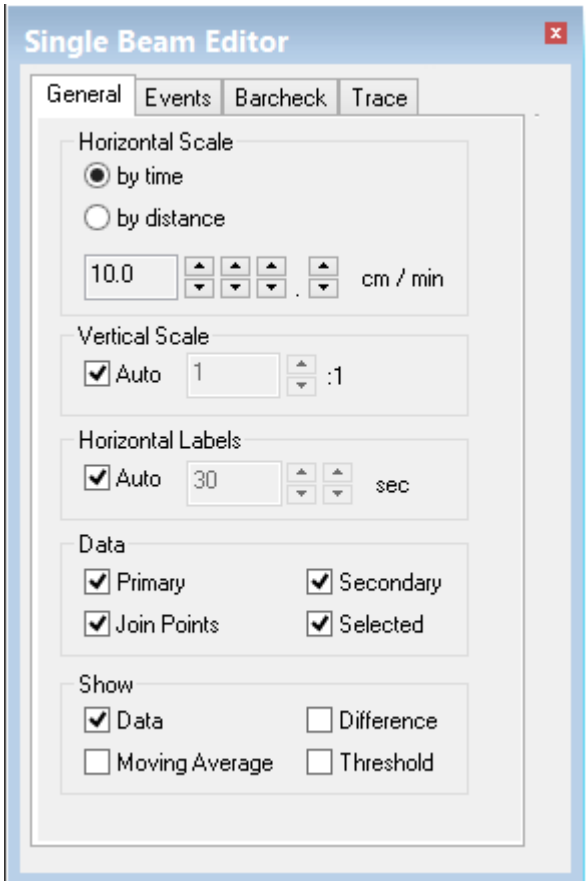
You can move up or down the track line by clicking the scroll bar or turning the mouse wheel (if available).

Toolbar

The Single Beam toolbar is automatically displayed when the editor is opened. This toolbar is used for selecting primary or secondary data as Selected data (see "PRIMARY AND SECONDARY DATA" ON PAGE 118). The toolbar can moved and docked in the interface like other toolbars (see "DISPLAY TOOLBARS" ON PAGE 536).

Single Beam Editor controls

The Single Beam Editor window contains four tabbed control pages, as follows:



- "GENERAL TAB" ON PAGE 112
- "EVENTS TAB" ON PAGE 114
- "BARCHECK TAB" ON PAGE 116
- "TRACE TAB" ON PAGE 117

General tab

Change horizontal scale

The range of data that is visible along the horizontal scale can controlled by distance or time. As you increase these values more data is displayed in the editor.

This provides a scaling method that simulates the echo sounder paper speed controls so that the digital profiles can resemble the analogue records.

1. In the General tab, select either the *By Time* or *By Distance* option.

2. Select a range of data using the *Horizontal Scale* arrow buttons.

Adjust vertical scale

The vertical scale displays the distance values. The *Auto* setting uses the maximum and minimum soundings in the current view to set the depth values for that view. As you move along the track line, the depth values change to accommodate the visible sounding data.

You can set a scale as a ratio of the vertical scale to the horizontal scale. As the ratio is increased, the vertical display of data becomes more exaggerated.

To set a vertical scale:

1. Clear the *Auto* check box.
2. Click the *Vertical Scale* arrow buttons to select an appropriate scale.

The display in the Single Beam Editor View is refreshed to accommodate the selected scale.

Adjust horizontal labels

The horizontal labels show the depth or time values at regular intervals along the survey line. The *Auto* selection spaces the labels far enough apart so you can see them clearly. You can determine the spacing you want for the labels.

1. Clear the *Auto* check box in the *Horizontal Labels* section.
2. Click the arrow buttons to select a new scale for the labels. If you select *By Distance* for the horizontal scale, then the labels are displayed as metres/feet. If you select *By Time*, the labels are displayed hours/minutes/seconds/fractions of a second.

The display in the Single Beam Editor is refreshed to show the labels according to the selected settings

Display Data

The Data options set which data is displayed. The default is to show all the data types.

1. Use the check boxes to toggle the display of data on and off.
   - Primary: displays primary data
   - Secondary: displays secondary data
   - Join Points: connects the sounding points with a line.
   - Selected: displays the Selected frequency when dealing with Dual frequency data.

The display colours of the Primary, Secondary and Selected data are set in Tools > Options > Display > Single Beam.

View moving average data

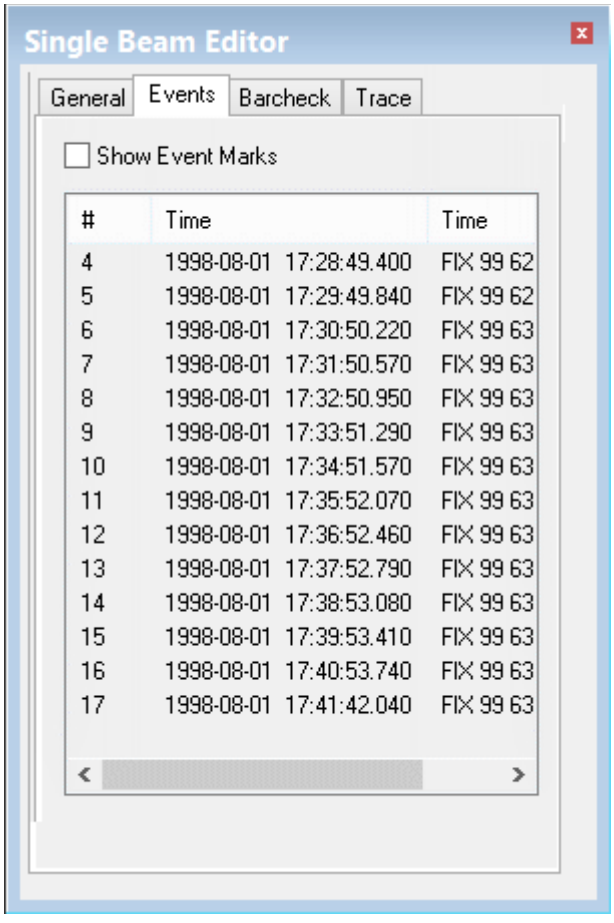
You can see the moving average data in the Single Beam Editor by selecting any of the following check boxes in the Single Beam Editor.

- *Data* graphs the raw sounding data. This box is checked by default.
- *Moving Average* graphs the computed moving average from the adjacent data points as described above.

- *Difference* graphs the difference between the original data values and the moving average for each data point.
- *Thresholds* displays horizontal lines in the graph indicating the difference values that are rejected during filtering. Threshold is a multiple of the standard deviation (sigma).

Events tab

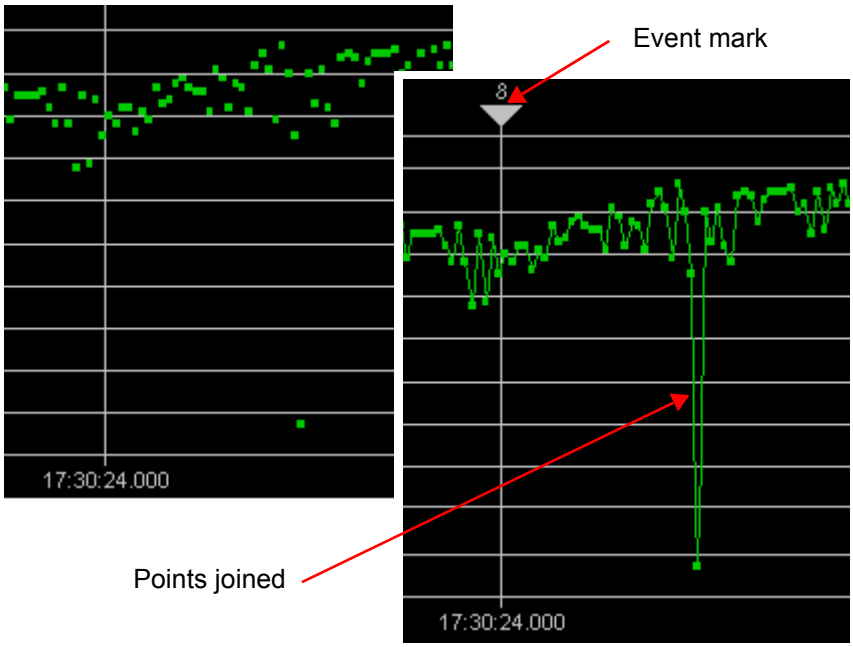
Event marks are commonly generated by the logging system at periodic intervals. Each event mark contains a unique identifier and time stamp. These event marks are displayed in table format on the Events page.



To display the event marks in the Single Beam Editor window:

1. Select the *Show Event Marks* check box.

The event marks are displayed at the top of the window.



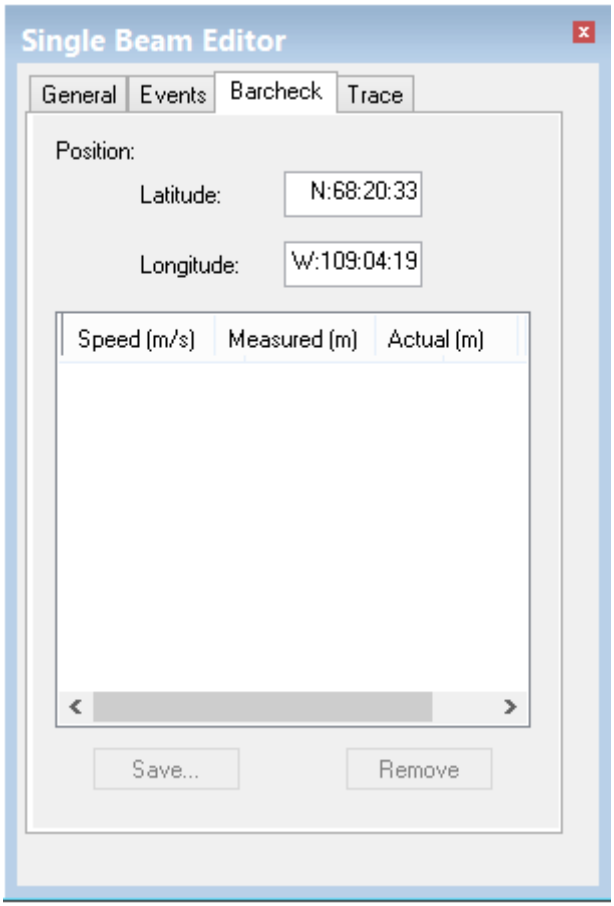
To centre the data in the Single Beam window on a specific event mark:

2. Double click on the event mark in the table.

The display will shift so that the data display in the editor window is centred on the selected event mark.

Barcheck tab

If there is barcheck data available, it can be used to generate a Sound Velocity Profile (SVP) file based on a list of measured depths against a list of actual depths. The Barcheck tab displays barcheck values and is used to add barcheck values.

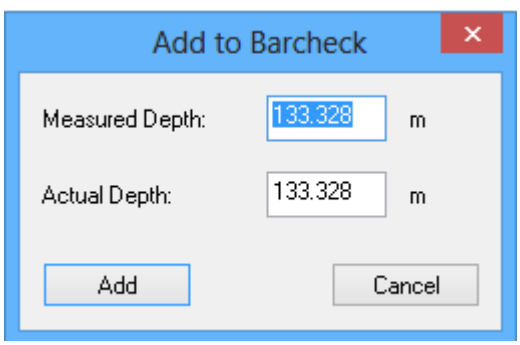


To add barcheck values:

1. Select soundings in the Single Beam Editor window.
2. Select Add to Barcheck command.

Menu	Tools > Editors > Single Beam > Add to Barcheck
Pop-up	Add to Barcheck

The Add to Barcheck dialog box is displayed.



The *Measured Depth* field displays the depth that was recorded by the sounder. If more than one sounding is selected, then a mean is calculated from the selected soundings.

3. Enter a new depth values from the barcheck data in the *Actual Depth* field.
4. Click **Add**.

The depth values and the sound velocity for the coordinates are now displayed in the Barcheck tab.

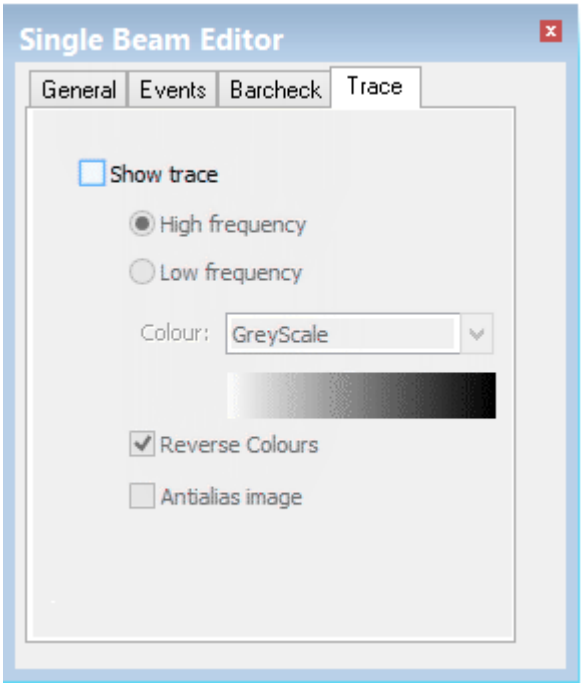
5. Repeat these steps as needed.
6. To delete a set of values from the table, highlight it and click **Remove**.
7. Click **Save**.

The Save dialog box is displayed. The default location for the file is `..\HIPS\SVP`.

8. Type a name for the file and click **Save**.

Trace tab

HYPACK data can contain a bin file that contains an image of the single beam trace. If this data is available the *Show Trace* option is activated. Setting this option will display the trace image as background in Single Beam Editor.



Primary and Secondary Data

Dual frequency data can be imported through the Generic Data Parser, the Hypack converter and the Winfrog converter.

These frequencies (and the associated data) are designated as Primary or Secondary. By default, all Primary frequency data is designated as the Selected Frequency, and processed further in HIPS and other CARIS products.

Show/Hide data

1. Open Single Beam Editor.
2. Select or clear the following check boxes to show or hide data. When the box is checked, the sounding data associated with that frequency is visible in the editor.
   - *Primary*
   - *Secondary*
   - *Selected*

The Single Beam Editor is refreshed to show the selected frequencies.

Select Primary/ Secondary data

Although Primary data is automatically designated as the Selected frequency, you can set the Secondary depths as Selected, and toggle between Primary and Secondary to view data. (The two frequencies cannot be averaged.)

1. In the General tab, select the Primary and Secondary check boxes so that all the frequency data is visible in the editor.
2. In the editor window, press and hold the mouse button and drag the cursor to highlight both the Primary and Secondary data.

Menu	Tools > Editors > Single Beam > Primary/Secondary
Tool	
Pop-up	Primary/Secondary
Key	<1>/<2>

The **Select Primary** and **Select Secondary** buttons are activated.

3. Select a Primary or Secondary command.

The highlighted Primary or Secondary data is now marked as Selected.

To see which frequency has been set as Selected, toggle the check boxes in the Data section of the General tab on and off.

Change Sounding Depths

The Single Beam Editor enables you modify the digital depths created by the echo sounder's digitizer. Sometimes when fish or other obstructions in the water column trigger the digitizer, the sea bottom may still be visibly detectable on the analog sounding graph. This function enables you set the echo sounder's depth values to the analog sounding depth values.

"CHANGE DEPTHS" ON PAGE 119

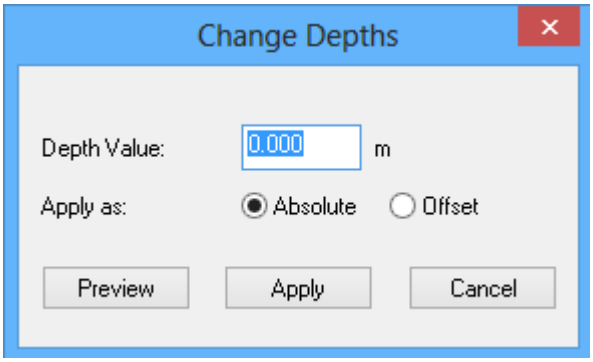
"ADD DEPTH" ON PAGE 120

Change Depths

1. Set the *Selected* check box in the Data fields of the Single Beam Editor General tab, if needed.
2. Select a sounding(s) in the Single Beam Editor window.
3. Select the Change Depths command.

Menu	Tools > Editors > Single Beam > Change Depths
Pop-up	Change Depths (in Single Beam View window

The Change Depths dialog box is displayed.



To move the sounding to a specific depth:

4. Select the *Absolute* option.
5. Type the new depth in the *Depth Value* field.
6. Click **Preview** to view the sounding at the new level.
7. Click **Apply** to set the sounding at the new depth and close the dialog box.

You can move a sounding to a deeper depth level, by applying a positive value (for example, 5.5). To move the sounding to a shallower depth level, use a negative value (for example, -5.5).

To move the sounding by a set amount:

8. Type the amount by which to move the sounding.
9. Select the *Offset* option.
10. Click **Preview** to view the sounding at the new level.

11. Click **Apply** to set the sounding at the new depth and close the dialog box.

You can also change depth values directly in the editor. However, this option omits the preview function of the dialog box.)

1. Select soundings.
2. Position your cursor over the selected data.

The cursor becomes a two-headed arrow with a Z symbol beside it.

3. Click and drag the cursor either up or down.

The sounding depths are visible as you move the cursor.

4. Release the mouse button when the soundings are at the new depth.
5. The
6. To cancel the effect of moving these soundings to the new depths, select **Undo** from the Edit menu.

Add Depth

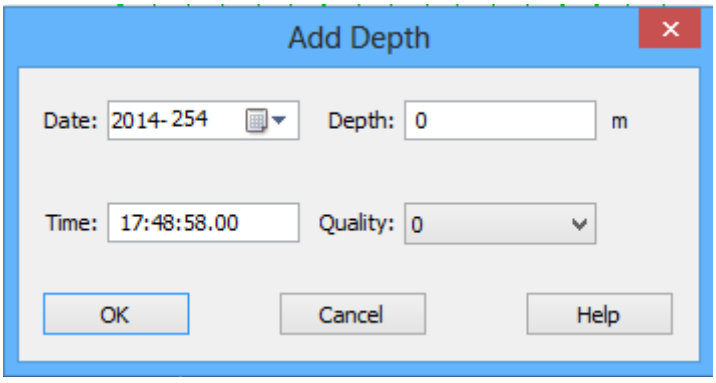
Add a new sounding by supplying time and depth. The position of the new sounding is interpolated from the existing navigation data (using the supplied time). A quality code can also be entered. If you enter a new depth, a new observed depths file and a new slant range (travel time) file is generated.

You can add depths by selecting an existing sounding and inserting a new depth or by creating a new time stamp without selecting an existing sounding.

1. Select the Add Depths command.

Menu	Tools > > Editors > Single Beam > Add Depth
Pop-up	Add Depths

The Add Depth dialog box is displayed.



2. Select the *Time* field and type a new time stamp value.
3. Select the *Depth* field and type a new depth value.
4. Select a new quality flag, if needed.
5. Press **OK** to add the new depth value to the file.

Single Beam Cleaning

Sounding data can rejected interactively or by applying the single beam filters (see "Single Beam Filtering" on page 124).

This section covers how to interactively reject, accept and query data in the Single Beam Editor. These processes are similar to the Swath Editor.

"Reject Data" on page 121

"Accept Data" on page 121

"Query Data" on page 122

"Auto-Cursor Mode" on page 123

Reject Data

Each sounding carries several status flag switches that are turned on and off as needed. Each sounding begins with Accepted status. If rejected for any reason during processing it can be returned to Accepted status without loss of data.

Menu	Edit > Status Flag > Reject
Tool	
Pop-up	Reject
Key	<R>

To reject a sounding:

1. Select the data to be rejected so it is highlighted.
2. Select the Reject command.

The Rejected by Swath Editor status flag is now on and the sounding may or may not be displayed depending if the rejected data is set to Show Rejected. (the same status flag is used for both swath and single beam data).

Accept Data

Change Rejected soundings back to Accepted status.

Menu	Edit > Status Flags > Accept
Tool	
Pop-up	Accept
Key	<A>

1. Select the rejected data so it is highlighted.
2. Select the Accept command.

The soundings are now flagged as Accepted.

Query Data

Menu	Edit > Query
Tool	
Pop-up	Query
Key	<Q>

View data on selected soundings.

1. Select the data to be queried.
2. Select the Query command.

The following data is displayed in the Selection window. The data is colour-coded depending on whether it is Primary, Secondary or Selected frequency data.

- Record
- Type (primary, secondary, or selected)
- Time stamp
- d-Time
- Approximate Position
- Distance
- d-Distance
- Depth
- d-Depth
- Status

Change sounding status

You can change the status of the queried sounding.

1. Select the sounding data.
2. Select Accept or Reject.

The changed status will be displayed in the Selection window.

See also "VIEW SOUNDINGS STATUS" ON PAGE 332

Auto-Cursor Mode

Menu	Edit > Status Flag > Auto Cursor
Tool	

The auto cursor combines the select and Accept/Reject and Query functions into a single procedure. The function is available in the HIPS and SIPS Editors.

1. Select the Auto Cursor command.
2. Select an Accept or Reject command, or select Query.

The cursor changes shape to reflect the command you have chosen: a "✓" for Accept, an "X" for Reject and a "Q" for Query.

3. Press and hold the mouse button and drag the cursor over data in the Editor window so that the data is highlighted.
4. Release the mouse button.

The highlighted data will be Accepted or Rejected. If you chose the Query command, the highlighted data will be displayed in the Selection window.

Single Beam Filtering

Using the Single Beam Editor you can filter single beam data converted from single and dual frequency echo sounders. This tool inherits some of the functions of the Swath Editor such as vertical exaggeration and filtering functions.

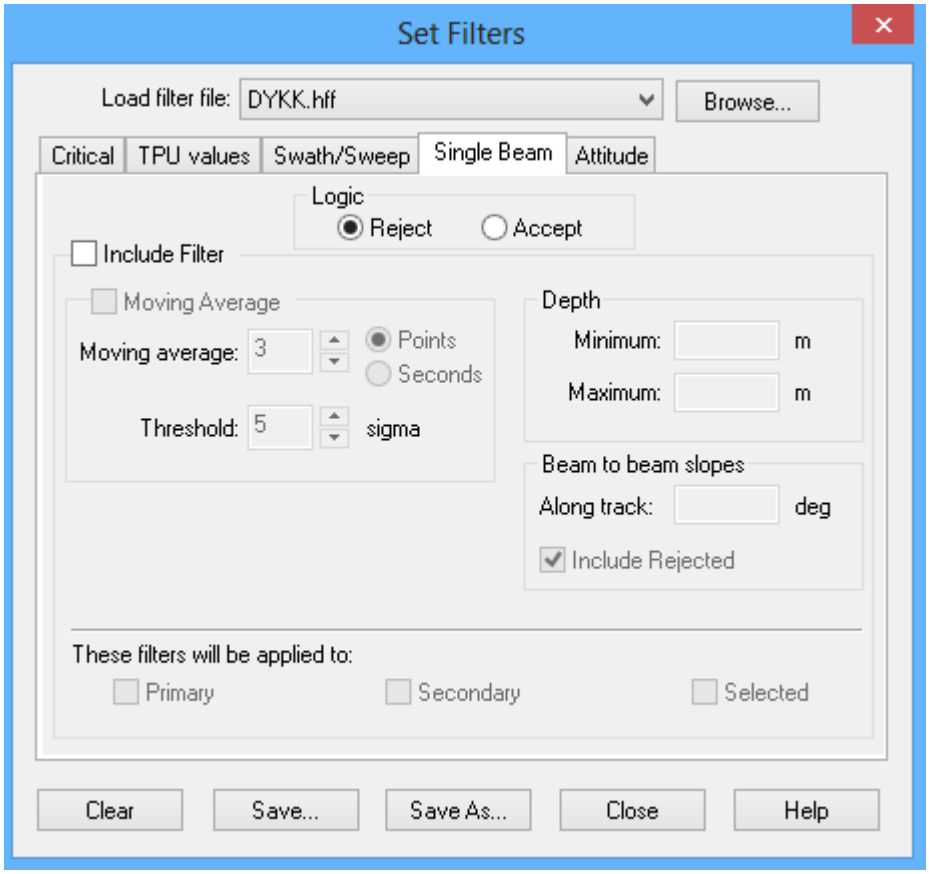
When dealing with bathymetric datasets, it is advantageous to reject outliers automatically. To do this, the Single Beam Editor has a set of filters.

The parameters for these filters are set through the Set Filters dialog box, and saved in a HIPS Filter File (.hff) that can be reopened for use in another track line.

Menu	Tools > HIPS Data Filters > Settings
Tool	

1. Select a track line.
2. Select the Set Filters command.

The Set Filters — Single Beam tab of the Set Filters dialog box is displayed.



3. Make sure the Single Beam tab is active.
4. [Optional] Select an existing HIPS Filter File from the list or click **Browse** to select a file.

If you chose to load a filter file, all values in that file are shown in the fields. You can go to Step 7 or change any of the values.

5. Select either a *Reject* or *Accept* logic command to apply to the filtered data.
6. Select the *Include Filter* check box to activate the filtering options.
7. Select from three filtering options:
   - *Moving Average* (see "MOVING AVERAGE FILTER" ON PAGE 125)
   - *Depth* (see "DEPTH" ON PAGE 126)
   - *Beam-to-beam slopes* (see "BEAM-TO-BEAM SLOPES" ON PAGE 126)

You can define and apply more than one filter test. The filters are applied in sequence. Any data rejected by one filter is not considered in the next filter test.

8. Select the frequency to apply to the filters by making sure the appropriate box is checked.
9. Click **Close**, or click **Clear** to remove data from the fields.

Click **Save As** to save the settings to a filter file.

Moving Average Filter

The Moving Average Filter rejects (or accepts) data with difference values that fall outside of defined threshold limits. To view the Moving Average filtering options in the Single Beam Editor, see "GENERAL TAB" ON PAGE 112.

A moving average is calculated for each sounding (data point) by using the adjacent soundings. The amount of adjacent data is specified either as the number of points or as a time period.

1. Select the *Moving Average* check box to enable the associated fields.

Set the moving average using seconds

This option uses a time range (in seconds) to calculate a mean average for each data point.

For example, if you select 10 seconds as the moving average, the filter calculates a mean average for any data point using itself and any data points in the five-second range either side of it.

Setting the moving average using points

This option uses a range of adjacent data points to calculate a mean average for each data point.

For example, if the moving average is set to five data points. A mean for any data point is calculated using itself and the two data points on either side of it (if available).

2. Select how you want the moving average to be calculated by selecting either *Points* or *Seconds*.
3. Use the arrow buttons to select the number of seconds or data points.

A difference value for each sounding is calculated for each sounding from the adjacent soundings.

Thresholds

The thresholds indicate what difference values are rejected (or accepted) during filtering.

The upper threshold is determined by the following formula:

```
original mean value + (threshold value X the
standard deviation)
```

The lower threshold is determined by the following formula:

```
original mean value – (threshold value X the
standard deviation)
```

Select a threshold value by clicking the arrow buttons.

Depth

1. Enter the depth values for filtering soundings. There are two options:
   - *Minimum Depth*: Reject (or accept) any soundings that are shallower than the depth specified (e.g., 10 m).
   - *Maximum Depth*: Reject (or accept) any soundings that are deeper than the depth specified (e.g., 50 m).

Beam-to-Beam Slopes

For each sounding, calculate the slopes, in degrees, to the previous and next soundings. If both slopes exceed the defined value and are of opposite sign, then reject the sounding.

1. Enter the degree value for the slope.
2. Select the I*nclude Rejected* option to include previously rejected soundings when recalculating the slopes during multiple runs of the filter.

Apply Filters

The last step is applying the automatic depth filter to the data.

1. Select a Filter command to apply the criteria set by the Set Filters dialog box.

Menu	Tools > HIPS Data Filters >Apply > 1Screen / End of Line
Tool	

You have four options for applying the filters.

Filter 1 Screen. Apply the filters only to the soundings that are currently visible in the Single Beam Editor.

To End of Line. Apply the filters to the track line currently open in the Single Beam Editor, from the first sounding currently visible in the editor to the end of the line.

7

Subset Editor

Subset Editor is a tool for viewing and editing geo-referenced sounding data.

In this chapter...

Overview

Subset Editor differs from Swath Editor in that soundings no longer have only an across-track and along-track relationship, but after correction for navigation, vessel heading and other auxiliary sensors, each sounding has a latitude and longitude value attached to it.

With Subset Editor you can:

- view data in a three-dimensional mode so features such as depressions and rises are visible.
- rotate data to view it from different angles.
- view surfaces in the 3-D window.
- view and edit CUBE surfaces, using subset tiles to track the areas that have been completed.
- flag soundings as needed to status of:
  - rejected
  - accepted
  - outstanding
  - examined
  - designated.
- view and manipulate water column imagery (see "WCI DATA IN SUBSET EDITOR" in the HIPS and SIPS User Guide.)
- apply a datum shift

Subset Editor Interface

Open Subset Editor and Load Data

Menu	Tools > Subset Editor
Tool	
Pop-up	Tools > Subset Editor

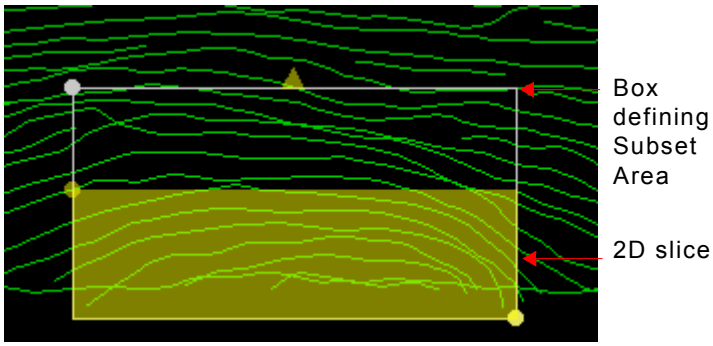
Track lines should be Merged before Subset Editor is opened.

1. Select the Subset Editor command.

The cursor changes shape.

2. Press and hold the mouse button, and drag the cursor across the area of the Display window where you want to create the subset.

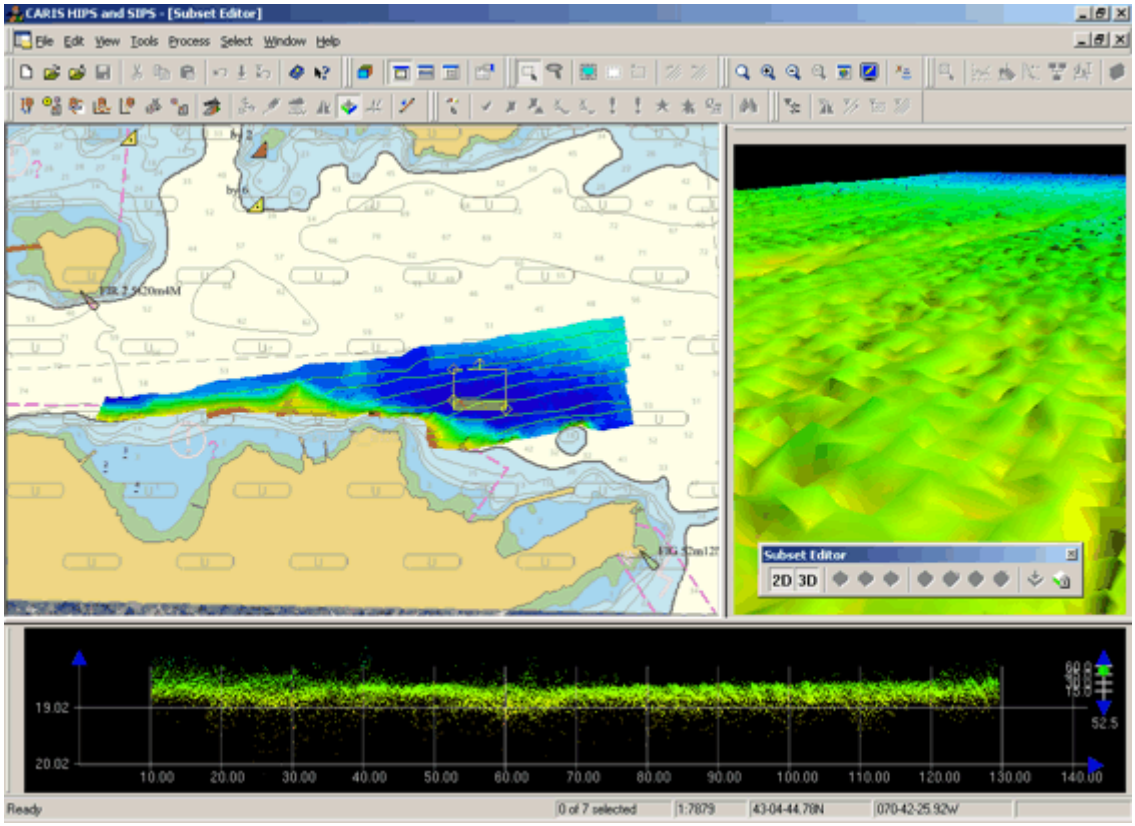
As the cursor is dragged across the Display window, a bounding box is drawn to show the subset area.



Menu	Tools > Subset Editor > Load Subset
Tool	

3. Select the Load Subset command.

The subset is generated and displayed in the Subset Editor windows. The following illustrates an example of HIPS and SIPS with the Subset Editor open.



Subset Editor has the following components:

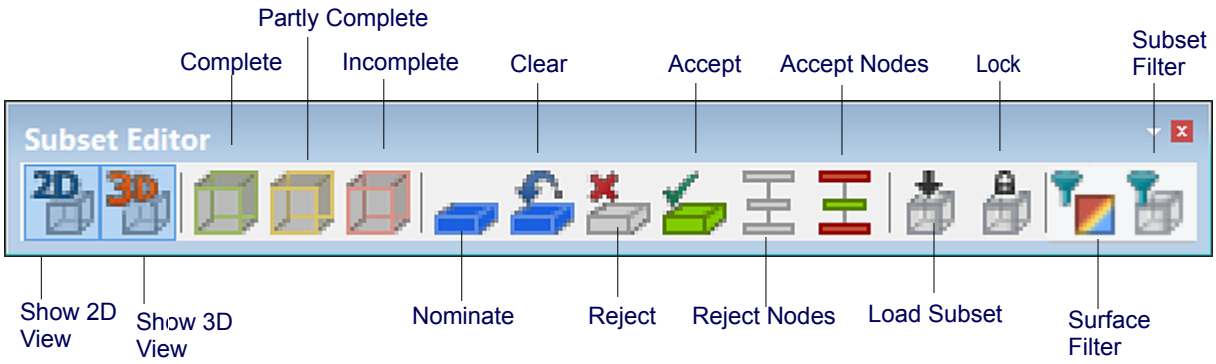
- **Subset bounding box**: The bounding box defines the area where the subset is generated. Each time the box is moved or resized, a new subset is generated. The yellow 2D slice box defines the area to appear in the 2D window.
- **3D View window**: The 3D (three-dimensional) View shows all soundings in the subset area. The view in the window can be rotated or changed.
- **2D View window**: The 2D (two-dimensional) View displays soundings from the subset area that are within the 2D slicing box.
- **Subset Editor Control window**: pages of options for display and properties of data that is displayed on Subset Editor. "SUBSET CONTROL WINDOW" ON PAGE 133

Toolbar

Subset Editor commands can be activated from the toolbar. This floating toolbar is automatically opened when the editor is started. (If it is not visible when Subset Editor is opened, the View > Toolbars > Subset Editor command can restore it.)

The toolbar can moved and docked like other toolbars

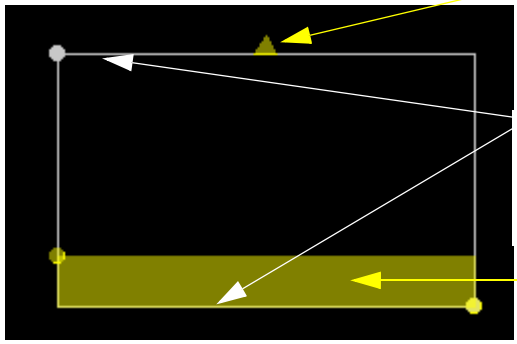
.



Size and Resize Subset

The size of the subset is determined by the bounding box. As the bounding box area is changed, a new subset is generated and the displays in the 2-D and 3-D Views are refreshed to show the new area.

The illustration below is an example of the subset bounding box.



To resize the subset areas:

1. Position the cursor over any of the balls at the corners of the boxes.
2. Press and hold the mouse button and drag the ball.
3. As the ball moves with the cursor, the box is resized to a new area.

You can also move the 2-D Subset slice box up or down in the bounding box by pressing the arrow keys. The arrow at the top of the box shows the direction of the 2-D slice when the up arrow key is pressed.

When the box is resized, a new subset is generated.

To rotate the subset:

1. Press and hold the <Ctrl> key.
2. Position the cursor over any of the balls at the corners of the subset bounding box.

Press and hold the mouse button to rotate the bounding box to the desired angle.

Lock subset

The Lock option locks the subset area so you do not accidentally resize or move the subset while moving it in the Display window. A new subset cannot be created while the lock is active.

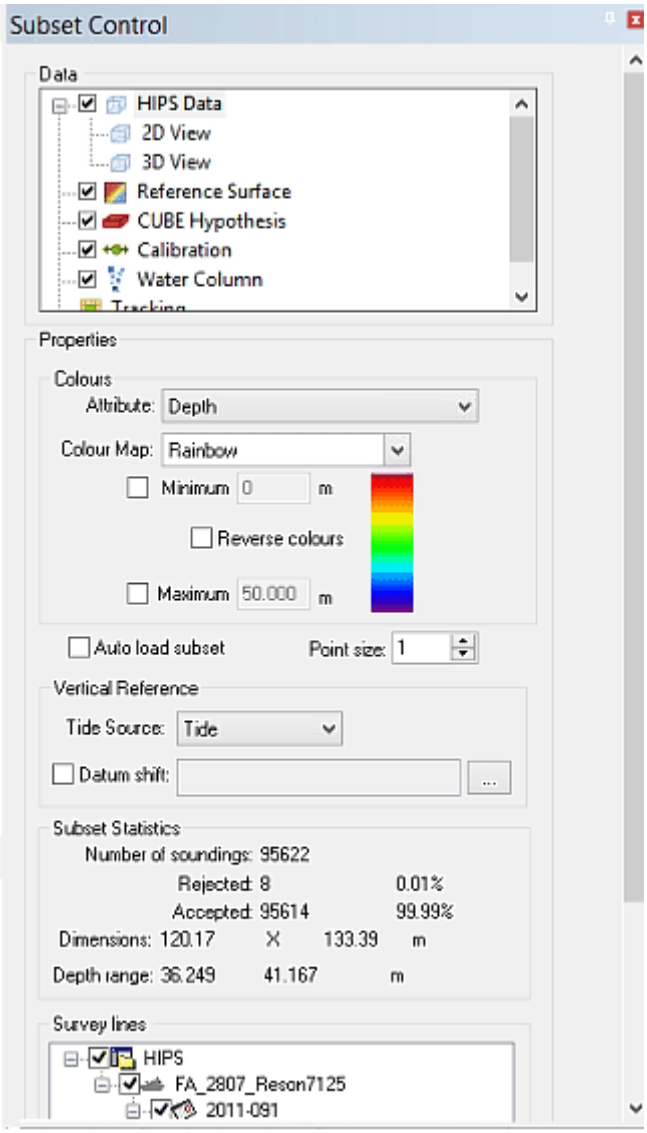
1. Select the Lock command after creating a new subset or resizing a existing one.

Menu	Tools > Subset Editor > Lock
Tool	
Key	<L>

The subset area is now locked into place and cannot be adjusted until the lock is removed.

Subset Control Window

The Subset Control window is displayed when the Editor is opened.



This window displays controls for properties of the various data layers that can be viewed in Subset Editor. When you select one of these layers, the body of the window will display the controls for that layer. The following image shows the properties for the HIPS Data layer, which is described below.

These layers are:

- **HIPS Data**: This layer controls the properties for sounding selection and display. The **3D and 2D** sub-layers control the display in their respective views.

  For properties of these layers, see the following sections:

- “HIPS Data Layer Properties” on page 135
- “Subset 2D View” on page 138
- “Subset 3D View” on page 142

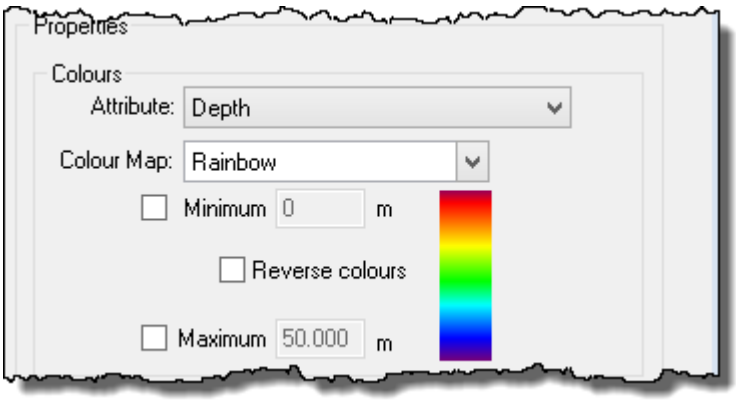
- **Reference Surface**: This layer controls the properties related to the display of surfaces in the 3D View of Subset Editor. For more information related to this layer, see:
  - "Subsets and Surfaces" in the CARIS HIPS and SIPS User Guide.
- **CUBE Hypothesis**: This layer controls the display and editing properties related to CUBE data. For more information related to this layer, see:
  - "Hypothesis Editing" in the HIPS and SIPS User Guide.
- **Calibration**: This layer controls the display of sensor data and the tools to enable you to resolve navigation latency and transducer mounting offsets and to update the project vessel file.
  - See "Calibration in Subset Editor" in the Tools Guide for more information.
- **Water Column:** This layer contains the controls for loading and unloading WCI data, and filtering by depth, slant range and intensity.
  - See "WCI Data in Subset Editor" in the HIPS and SIPS User Guide for more information.
- **Tracking**: This layer controls the properties for subset tiles. For more information on the controls related to this layer, see:
  - "Tracking Cleaning Status" in the HIPS and SIPS User Guide for more information.

HIPS Data Layer Properties

Properties of the HIPS Data layer include:

Colours

You can control how soundings are displayed in the subset, in the Properties > Colours section of the window.



1. Select an attribute from drop-down list in the *Attribute* field, to colour-code that specific data:
   - **Depth**: The recorded depth level of the soundings.
   - **Depth Uncertainty**: The vertical uncertainty values accorded to soundings.
   - **Position Uncertainty**: The horizontal uncertainty values accorded to soundings.
   - **IHO S-44**: The S-44 Survey Order ranking applied to soundings.
   - **IHO S-57**: The S-57 Zones of Confidence (CATZOC) attributes ranking applied to soundings.
   - **Project**: The project where the data was saved.
   - **Vessel**: The vessel(s) used on the survey.
   - **Day**: The day the sounding data was recorded.
   - **Line**: The track line in the project.
   - **Amplitude**: Backscatter intensity values.
   - **Detection Method**: The phase detection method.

Colour Map

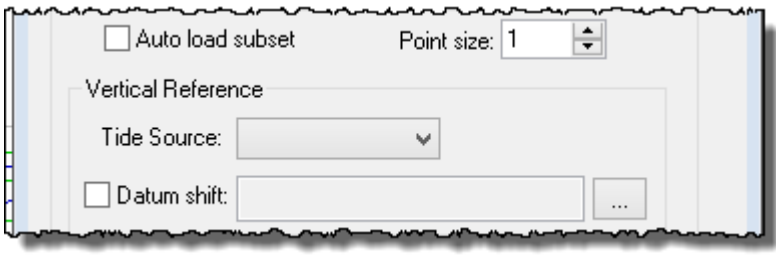
You can represent a range of values using a colour map. A colour map can be applied to other subsets so you have a consistent colour code for displaying data.

- The colour at the top end of the selected map represents the minimum value (and all values less than the minimum value).
- The colour at the bottom of the map represents maximum value (and all values greater than the maximum value).
- The colours in between the minimum and maximum colours represent the depths and uncertainties between the defined range.

1. To set a minimum and maximum values, select these two check boxes and type the values in the respective fields. If these values are not set, then they are automatically read from the subset.
2. [Optional] Select the *Reverse Colours* check box to reverse the order in the colour map.

The display in the Views is refreshed to show the subset in the selected colours.

Auto load subset



The *Auto Load Subset* option automatically loads subset data in the 3-D and 2-D Views when a subset area is created or changed.

1. In the Subset Control window, select the *Auto Load Subset* check box to turn this option on.
2. Select a value between 1 and 10 in the Point size field, to set the displayed size of the points in the 2D and 3D views.

Vertical Reference

Data can be shifted vertically using either a tide or a currently open surface to visually change the data in the display. (No changes are made to the stored data.)

Tide data generates final depths relative to a tide datum by subtracting the tide values from the sounding depth.

1. Select either Tide or GPS Tide from the *Tide Source* drop-down list. (Default is Tide.)

An open surface or an XYZ file opened as background data, can be used to shift using a grid of datum heights.

2. Select the Datum Shift check box and click Browse.

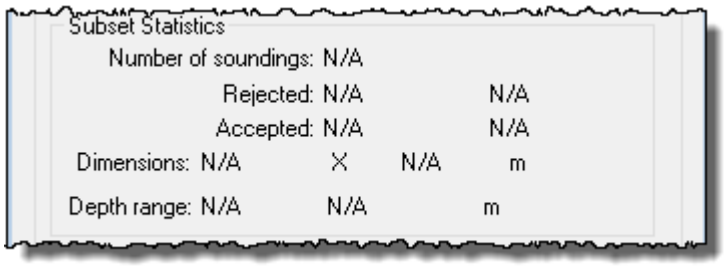
The Select Surface dialog box will display open surfaces.

3. Select a surface to use.
4. Click **OK.**

Subset Statistics

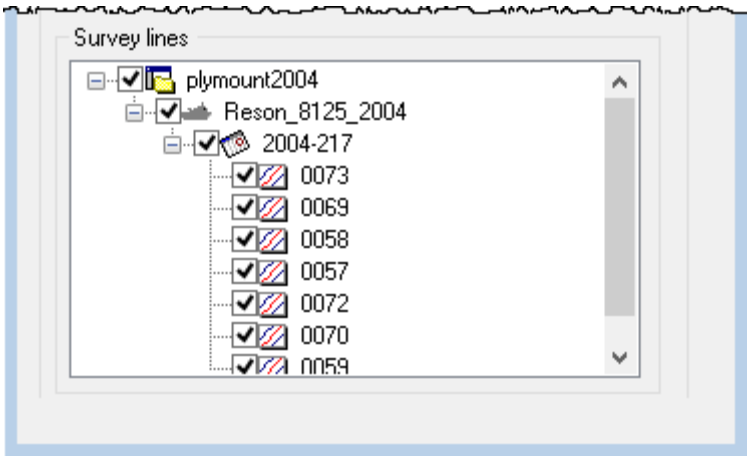
Each time a subset is generated, the following information is shown in the middle section of the Subset Control window:

- total soundings in the current subset
- number and percentage of soundings with Accepted and Rejected status
- dimensions of the subset
- minimum and maximum depth levels within the range of the subset.



Survey Lines

The Survey lines section displays the currently open data in a data tree.



Subset 2D View

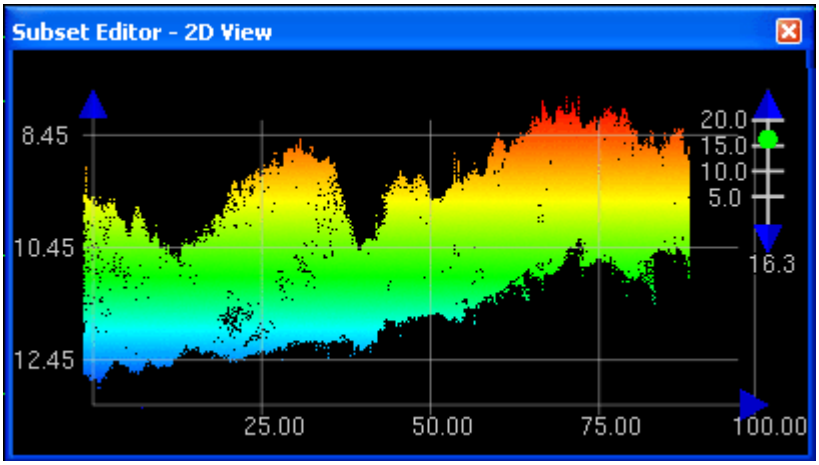
The 2D View displays a profile of soundings within the 2D slicing box. You can use this view to examine and clean data.

The location and size of this cutaway view is determined by the size of the 2D slice box (see "SIZE AND RESIZE SUBSET" ON PAGE 131). As the slice box is re-sized or moved to another position in the subset, the display in the 2DView changes accordingly.

Menu	Tools > Subset Editor > 2D View
Tool	

1. Select the 2D View command to display data in the 2D View window.

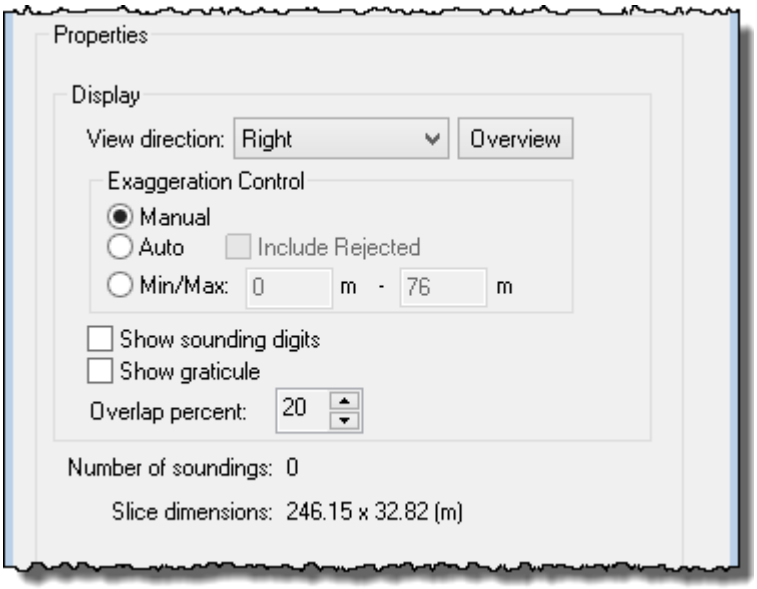
The 2D View is displayed. The example below displays depth data in the 2D View with the graticule (grid) option turned on.



All controls for the 2D View are in Subset Control window.

2. In the Subset Control window, expand the HIPS Data layer and select the 2D View sub-layer.

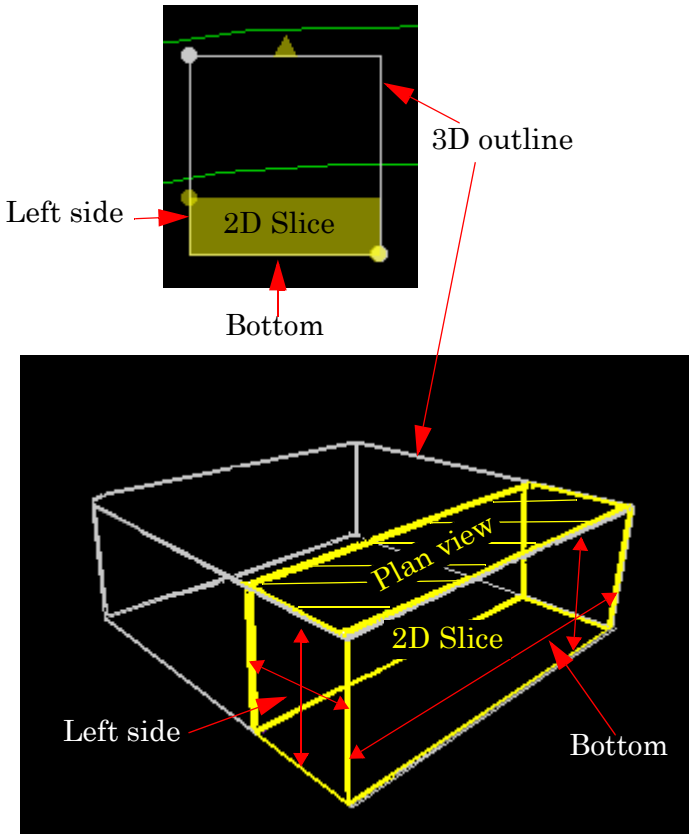
The Properties for the 2D View are shown.



Change view perspective

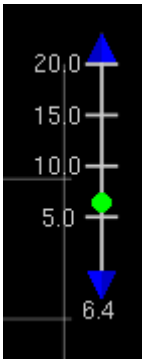
When the 2D View is first displayed, the view perspective is determined by the setting in the *View direction* drop-down list. The setting options are:

- Right: view data from the right side of the slice box
- Top: view data from the upper side of the slice box
- Left: view data from the left side of the slice box
- Bottom: view data from the lower edge of the slice box
- Plan: view data looking down into the box from above
- Automatic: the perspective is determined by the dimensions of the 2D Slice Box.
  - If the height of the slice box is greater than the width, the view is automatically shown from the Left side.
  - If the width of the slice box is greater than the height, the view is automatically shown from the Bottom.



2D vertical exaggeration

The vertical exaggeration is the ratio of the vertical scale versus the horizontal scale. You can set the vertical exaggeration with the vertical exaggeration scale in the 2D View.



1. Position the cursor over the ball on the vertical exaggeration scale.
2. Press and hold the mouse button, and move the ball up or down the scale.
3. To change scale values, click the arrows at the ends of the scale to increase or decrease the scale by a value of 10.

As you move the ball, the data in the 2D View is rescaled.

The *Auto Exaggeration* option determines the scale automatically by using all visible depths.

When *Auto Exaggeration* mode is on, the *Include Rejected* option controls whether or not the rejected data is used to determine the minimum and maximum sounding depth to be displayed in the 2D View.

4. Select the *Auto Exaggeration* check box.
5. Select the *Include Rejected* option to resize the display in the view to include rejected data.

The 2D View is adjusted to display data according to the maximum and minimum settings.

Display options

The display can also be modified to draw sounding values in the View and display a grid with depth and distance values.

1. Select the *Show Sounding Digits* check box to draw sounding values in the View.

Values representing distance and depth are displayed along the axes. The vertical scale represents depth and the horizontal scale represents distance from the lower left corner of the slice.

To show a grid of these values:

2. Select the *Show Graticule* check box.

Values shown in the 2D View are determined by the View Direction option:

- If the Right/Top/Left/Bottom options are selected, both the depth and distance are shown.
- If the Plan option is selected, distance in both directions is displayed.

Properties

At the bottom of the list of options the following properties of the 2D slice is displayed:

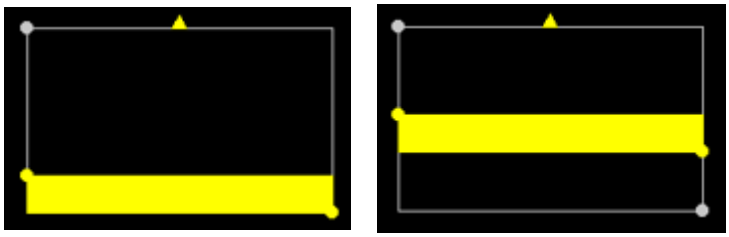
- number of soundings in the 2D slice

- dimensions of the slice (height x width)

When the 2D slice bounding box is resized, the information in the window is refreshed with new data.

Move 2-D Display

You can move the 2-D Slice box within the subset with the arrow keys.



The 2-D Slice box is moved up as the Up arrow key is pressed.

1. Press the arrow keys to move the 2-D Slice Box within the subset.
2. Select an *Overlap Percent* to determine how much of the 2-D Slice box overlaps the previous area in the 2-D View when the box is moved. (The higher the percentage is, the larger the amount of overlap and the greater the number of slice views within the subset.)

Pan and zoom options

The pan and zoom functions are the same as in the 3D View in the Swath Editor. See "PAN AND ZOOM" ON PAGE 176 for more information.

You can also pan in the Views by clicking on a sounding with the middle-mouse button. This will shift the subset so that the sounding is centred in the View.

Overview button

To return the display in the View to original setting before zoom and pan operations, click **Overview**.

Subset 3D View

Menu	Tools > Subset Editor > 3D View
Tool	

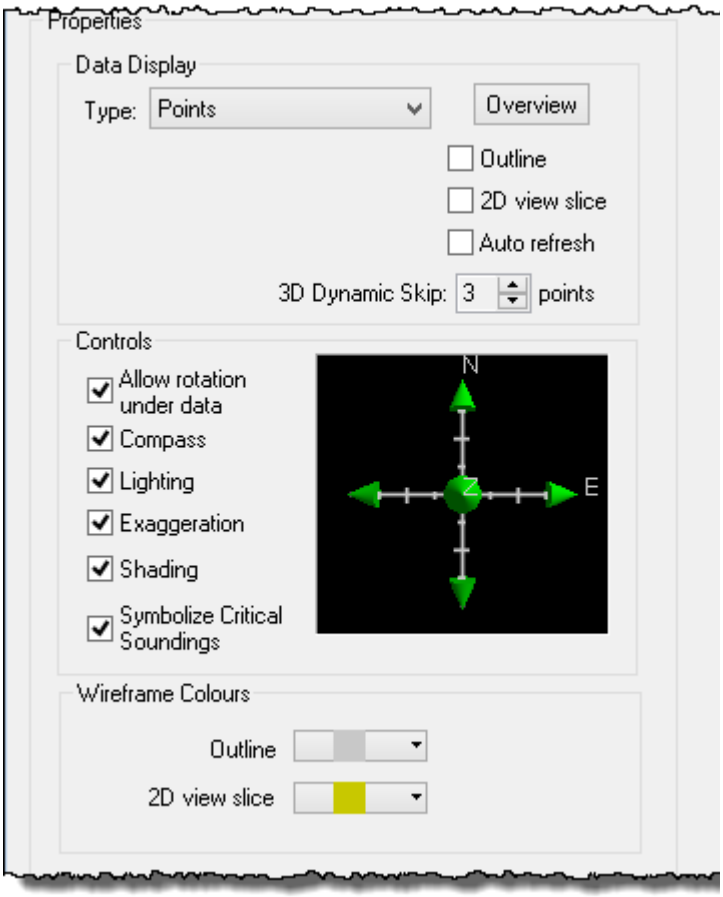
The 3D View displays the entire subset. You can use this view to examine and clean data.

1. Select the 3D View command to open the 3D view window.

All controls for the 3D View are contained in the 3D View sub-layer of the Subset Control window.

1. Click the Expand (+) icon beside the HIPS Data layer and select the 3D View sub-layer in the Subset Control window.

The properties for the 3D View are displayed in the Data Display area below.



3-D Subset Display

The *Data Display Type* selected will determine the soundings that will be shown in the 3-D View:

1. Select the display type from the list.
   - Points
   - Spheres
   - Cylinders

- Surface
- Surface Wireframe

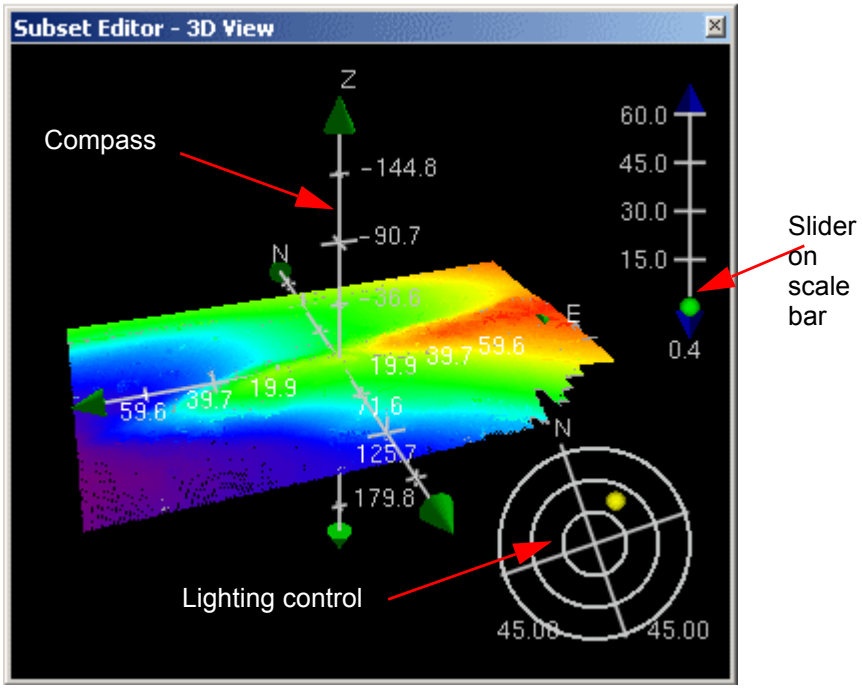
The Subset 3-D View is refreshed to show the soundings according to the selected display type. The following options can be set for each display type.

Display Type	Definition	Options
Surface	The soundings are shown as a smooth surface.	Soundings are drawn to the 3-D View either by selecting the shoalest sounding in a user-defined area (or bin), or by selecting all soundings. 1. Select the *Shoal Bin Size* to select the shoalest sounding in an area (bin), or select *Use All Points* to display every sounding. 2. If you selected the *Shoal Bin Size* option, type the size of the area for selecting the shoalest sounding. 3. Click **Regenerate**.
Surface Wireframe	The soundings are shown as a Triangulated Irregular Network (TIN) that connects soundings.	
Points	Soundings are shown as unconnected points.	3D Dynamic Skip option determines how many sounding profiles and beams are displayed. For example, the value 6 means that only every 6th profile and 6th beam are shown in the view. 1. Select a skip value by clicking the up or down arrow buttons, or directly typing a value in the *3-D Dynamic Skip* field. 2. Click **Regenerate**.
Spheres	Soundings are shown as unconnected spheres.	
Cylinders	Soundings are shown as cylinders. If the soundings contain horizontal and vertical uncertainty values, then the cylinders are sized according to those values. For example, soundings on the outer edge of a swath have a larger radius than soundings on the inner edge because of the greater horizontal uncertainty. Soundings without TPU values are drawn to a set size.	

Controls

Rotation, illumination and exaggeration options

The compass (or control axes) rotates the view, the lighting control adjusts illumination, and the slider on the scale bar adjusts vertical exaggeration.



2. Select or clear the *Compass*, *Lighting* and *Exaggeration* check boxes to show or hide these controls in the 3D View.

Shading

Darkens the image in the 3-D View.

3. Select the *Shading* check box.

Symbolize Critical Soundings

Display soundings with Outstanding, Critical and Designated flags.

4. Select the Symbolize Critical Soundings check box.

Overview

Reset the View so the entire subset area is visible (default).

5. Click **Overview**.

Show subset outline

Display an outline of the subset bounding box.

6. Select the *Show Subset Outline* option. When this option is checked, the outline is displayed in the 3D View.

Show 2D slice

Display an outline of the 2D slice area.

7. Select the *Show 2D* slice option. When this option is checked, the 2D slice outline is displayed in the 3D View.

See "CHANGE VIEW PERSPECTIVE" ON PAGE 139 for illustration.

Allow rotation under data

This option rotates the 3D display by 180° to view soundings underneath the subset. When this option is not selected, the subset can only be rotated by 90°.

8. Select the *Allow Rotation Under Data* option to rotate the subset display by 180°.

Wireframe colours

Select a colour to display 3-D and 2-D outline in the 3-D View.

9. Select a colour from the picker, or click **Other** and create a custom colour from the palette.

Pan and zoom

The pan, zoom and rotation functions are the same as in the 3-D View in the Swath Editor. See "PAN AND ZOOM" ON PAGE 176 for more information.

Measure Distance in Subset Editor

The Measure Distance tool in the Subset Editor 2-D and 3-D Views can be used to evaluate data problems, such as timing offsets, heave problems, etc. In 3D View the tool will measure along the X and Y planes, but not Z.

Menu	Tools > Measure Distance
Tool	

1. Select the Measure Distance command.

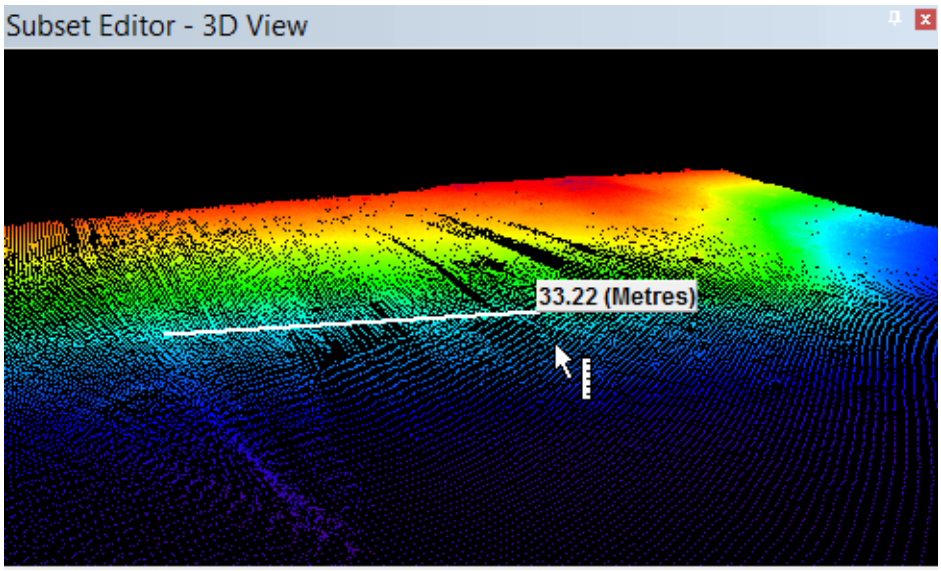
A ruler icon is displayed next to the cursor.

2. Press and hold the mouse button at one point of the distance you are measuring.
3. Drag the cursor along the image toward the end point of the distance.

As you drag the cursor across the view, a line is drawn, and the distance between the two points is shown in a tool tip. This is updated as the line is drawn.

To stop measuring:

4. Release the mouse button.



Close Subset Editor

Menu	Tools > Subset Editor
Tool	
Pop-up	Tools > Subset Editor

1. Select the Subset Editor command.
2. If changes were made to the data, you are prompted to save them.

When data in Subset Editor is changed and saved, the `ProcessedDepths` and `ObservedDepths` files are updated to reflect the changes.

3. Click **OK** to save changes.

Subset Editor is closed and no longer visible in the HIPS interface.

8

SVP Editor

Use SVP Editor to edit existing SVP data, or to create new data.

In this chapter...

SVP Editor Interface

SVP Editor is a separate program that is launched from HIPS and SIPS that can be used to edit an existing SVP file before it is loaded and applied during sound velocity correction.

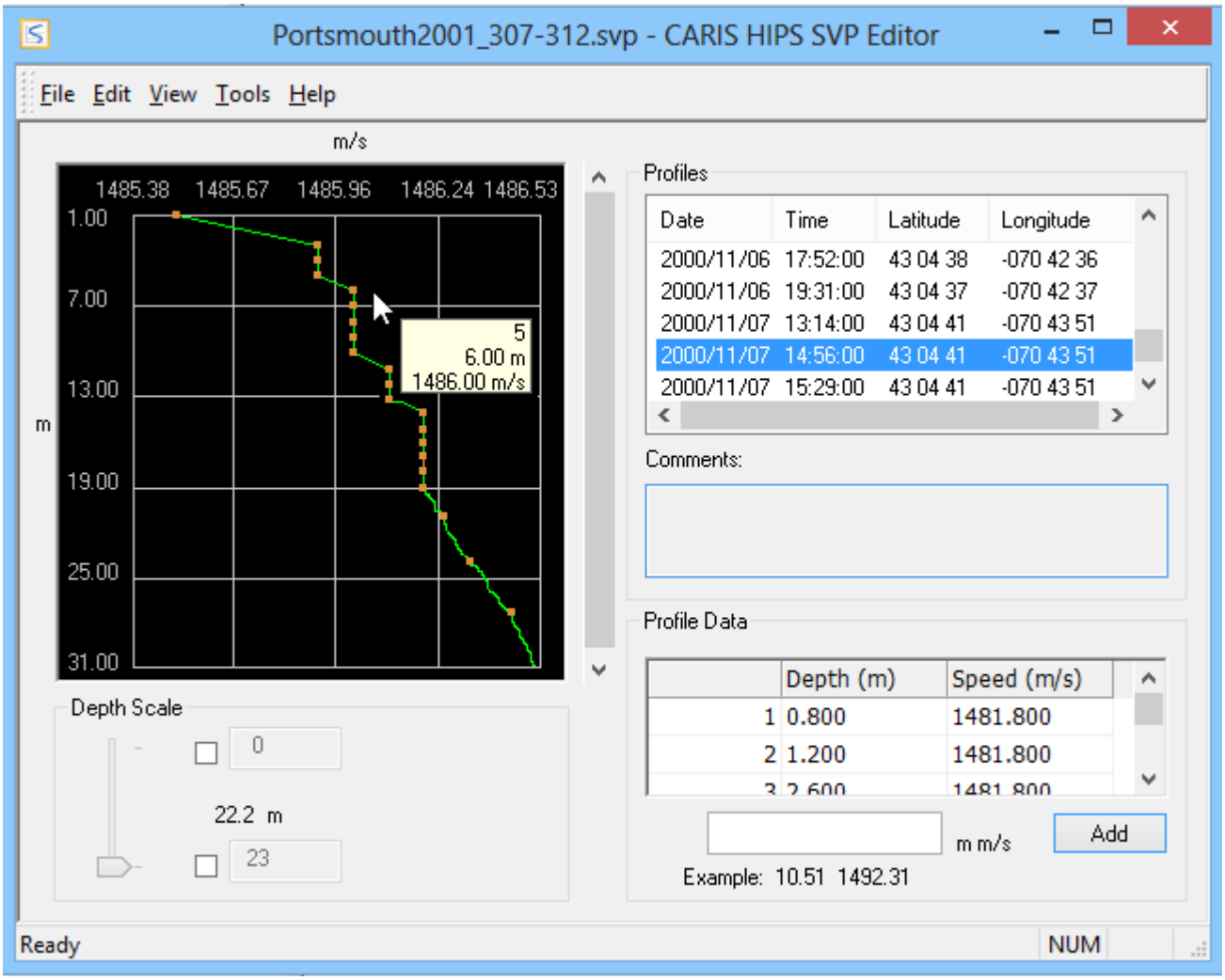
Menu	Edit > Sound Velocity Profiles
Tool	

To apply SVC, see SOUND VELOCITY PROCESSING ON PAGE 158.

To open SVP Editor:

1. Select an SVP Editor command.

HIPS SVP Editor is displayed.



The left-hand side of the editor contains a graphical representation of all the depth/speed data for the selected profile. In the example above, the points and tool tip options have been turned on. See OPTIONS ON PAGE 151 for details.

2. Select a point on the graph to highlight the values in the Profile Data table.

Range Slider

Since some data types include a bottom end buffer that distorts the graphing of the profile, a *Depth Scale* slider is available to limit the range of data.

To set a range:

3. Enable the check boxes.

This activates the value entry fields.

4. Enter the minimum and maximum values.

As the values are adjusted, the graph display is updated.

Profiles

The *Profiles* box lists profiles by date and time, showing the latitude and longitude for each. This data is used when applying the *Nearest in Distance* option for profile selection during sound velocity correction.

Profile Data

The *Profile Data* box shows the depth and speed values for the profile selected in the *Profiles* table.

The Comments box displays comments added or edited for a selected sound velocity profile. See

Open an Existing File

Menu	File > Open
Tool	
Key	<Ctrl+O>

1. Select the Open File command.

The Open dialog box is displayed. The location of the SVP files can be set in Tools > Options > File Locations.

2. Select an SVP file so it is highlighted, or type the file path in the *File Name* text box.
3. Click **OK**.

A list of profiles is displayed. Data in the first profile is displayed in the Profile Data table and the speed/depth graphed area.

4. To navigate between profiles, click the time stamps in the *Profiles* list box.

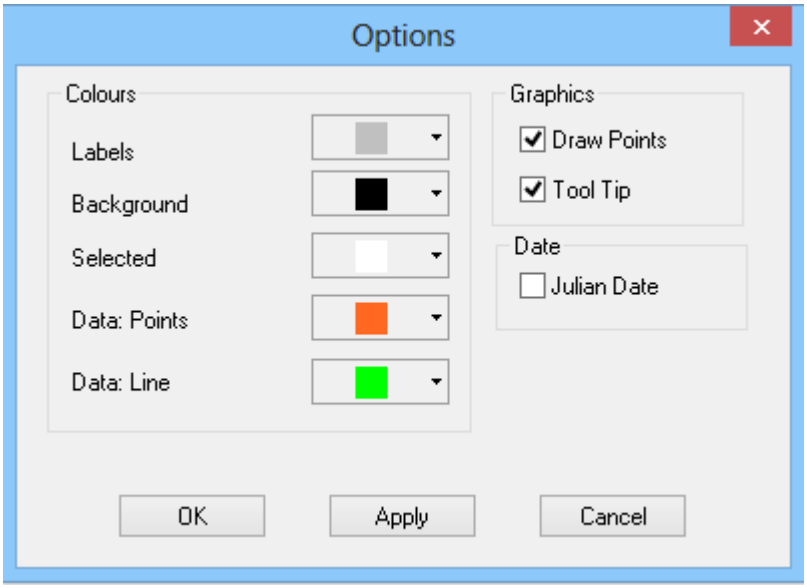
Options

You can select the colours used for the graphical display.

Menu	Tools > Options

1. Select the Options command.

The Options dialog box is displayed.



Colour Options

2. Select a colour for the Display from the drop-down palette or create a custom colour from the Windows colour picker.
3. Click **OK**.

The display is refreshed to show the selected colours of background, selected points or lines.

Draw points

Show or hide the depth/speed data points in the graphical display.

4. Select the *Draw Points* check box.

If the box is checked, the data points are visible in the graphical display.

5. Click **Apply** to apply and save changes to the current file.
6. Click **OK**.

Show/hide tool tips

A tool tip is a label containing the depth/time position for a data point in the graphical display. The label becomes visible when the mouse cursor is moved in the graphical display.

7. Select the *Tool Tips* check box to show or hide the tool tips.

If the box is checked, the depth/time labels are visible.

8. Click **Apply** to apply and save changes to the current file.
9. Click **OK**.

Show Julian date

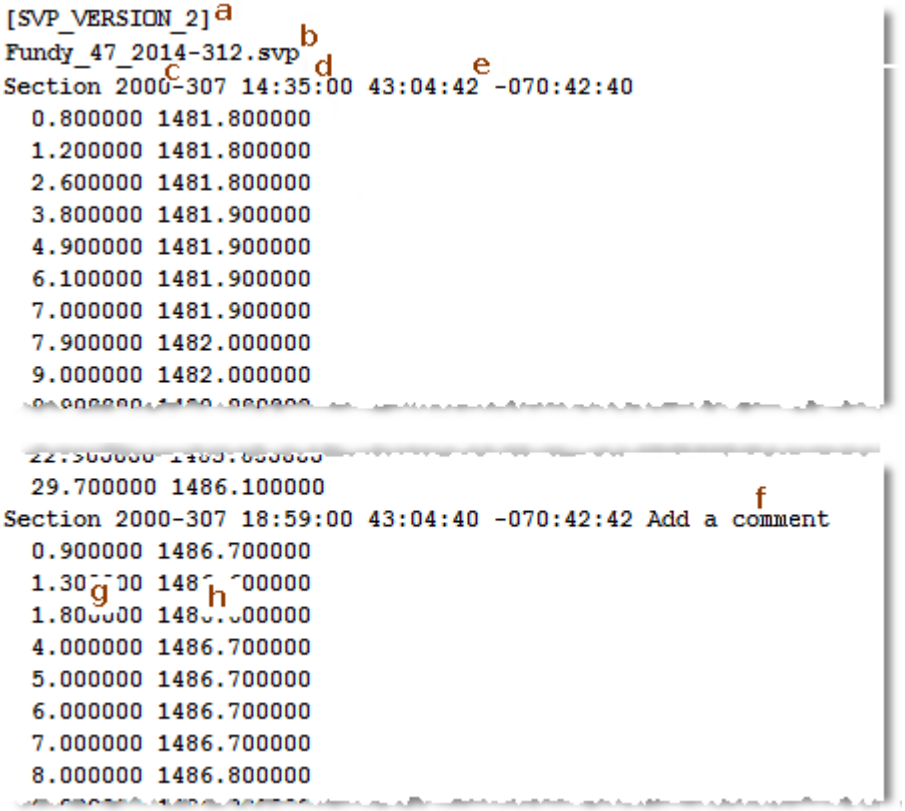
Switch the date fields in the SVP editor between the regular calendar format and Julian calendar format.

10. Make sure the *Julian Date* box is checked if you want to select this feature.
11. Click **Apply** to apply and save changes to the current file.
12. Click **OK** to close the dialog box.

SVP Files

Since Sound Velocity Profile files are in text format they can be opened in a text editor as well as in SVP Editor. These files can contain more than one section.

The following is an extract from a sample SVP file:



The first line of the file specifies the version of the SVP file (line "**a**" in the example above).

The second line displays the name of the SVP file ("**b**").

The next line is a section heading for each profile, containing:

- the year and day (Julian date) that the profile was recorded ("**c**")
- time stamp when the profile was recorded ("**d**")
- the latitude and longitude of the profile location ("**e**").
- the text of a comment added to the profile using the Edit Profile function ("**f**").
- the SVP data displayed in two columns, one column ("**g**") containing the depth, and the other("**h**"), the speed value for that depth.

Create a New SVP File

Menu	File > New
Tool	
Key	<Ctrl+N>

To create a new SVP file:

1. Select the New File command.
2. Add new SVP profiles (see ADD A NEW PROFILE ON PAGE 154.)
3. Create new depth/speed values, as needed.
4. Select the Save As command from the File menu.
5. Type a name for the SVP file in the file name box.
6. Click **Save**.

The file is saved the directory you selected.

Add a New Profile

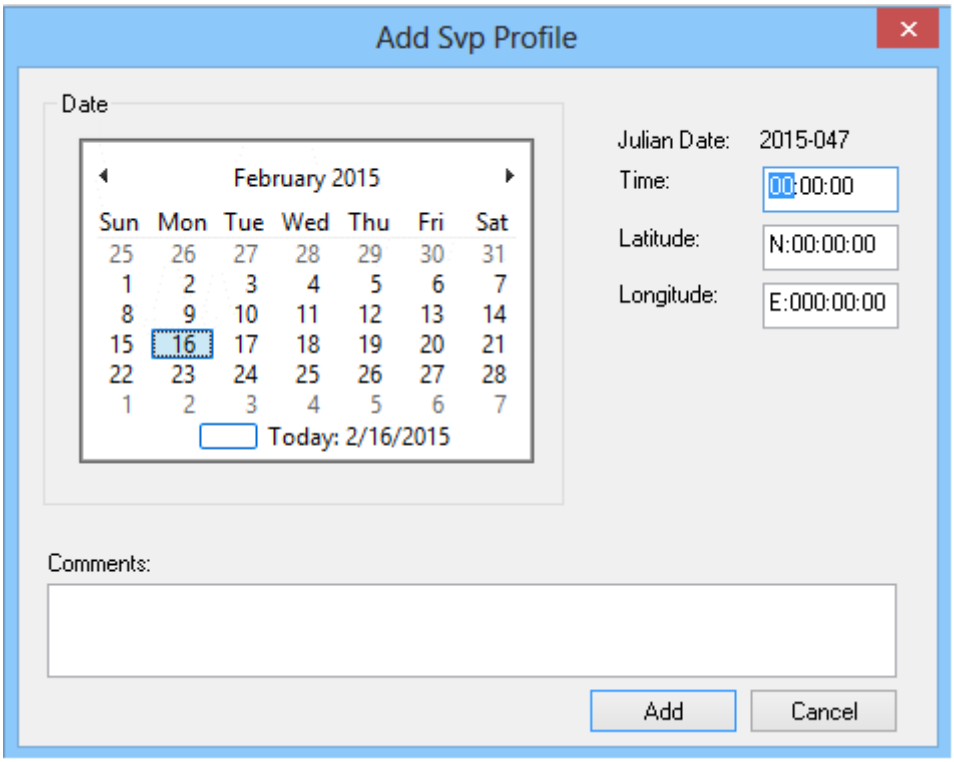
New SVP profiles can be added to an existing file or when creating a new SVP file.

To add a profile to an existing file:

Menu	Edit > Add Profile
Key	<Ctrl+Insert>

1. Open the SVP file.
2. Select the Add Profile command.

The Add SVP Profile dialog box is displayed.



All profiles must be time-stamped.

3. Click the arrow buttons at the top of the calendar to select a month and year. (All profiles must be time stamped.)
4. Click on a date in the calendar to select the day for the profile.

The Julian Date field is updated to display the in the

5. Enter the new hour/minute values (use the 24-hour clock) by clicking each section in the *Time* box and typing new values.
6. Click in the Latitude coordinates field and type coordinates.
7. Click in the Longitude field and type coordinates.
8. Add text to the Comments field.

This will be displayed in the Comments box in the main interface when a profile is selected. As well, comments are added to the section headers in the SVP file.

9. Click **Add** to apply settings.

The profile will be added to the list in the Profiles table.

Add Speed and Depth data

To enter new depth/speed data for the profile:

10. Select the new profile so it is highlighted.

The *Profile Data* table and the graph will be empty.

11. Click the **Add** button below the *Profile Data* table to activate the values text box.

New speed and depth data must be entered together. Depth values must be grater than 0m. Speed values must be between 1400.0 and 2000.00. There are two options for entering data:

- separate the depth and speed values with a space, for example: `15 1200`
- separate the data with a comma: `15,1200`

12. Type new depth *and* speed values in the text box beside the **Add** button.

The depth/speed values will be added to the new profile, and displayed in the profile graph.

13. Repeat Steps until all profiles for the new SVP file are entered.

Edit SVP Data

You can use the SVP Editor to modify existing speed/depth values, append new data to an existing file, delete data or delete a complete profile.

Edit speed or depth values

Individual values in the depth/speed table can be changed. To change depth/speed values:

1. Select a profile in the Profiles table to display the data in the SVP Editor.

The Profile Data fields will be updated to show the data for the selected profile.

2. Click on a *Depth* or *Speed* cell in the *Profile Data* table to select the value.
3. Type a new value.
4. Click outside the cell.

The cell displays the new value.

to add depth and speed together.

Add new values

To add depth and speed values together:

1. Select an existing profile in the Profiles table.

There are two options for typing values when entering speed and depth data together:

- separate the depth and speed values with a space, for example: `15 1200`
- separate the data with a comma: `15,1200`

Depth values must be greater than 0m. Speed values must be between 1400.0 and 2000.00.

2. Type new depth *and* speed values in the text box beside the **Add** button.
3. Click **Add** or press **Enter**.

Edit positions

Enter new latitude and longitude positions for SVP data. This information is important if you want to use the *Nearest in Distance* option for sound velocity correction ((see SOUND VELOCITY PROCESSING ON PAGE 158 of the User Guide).

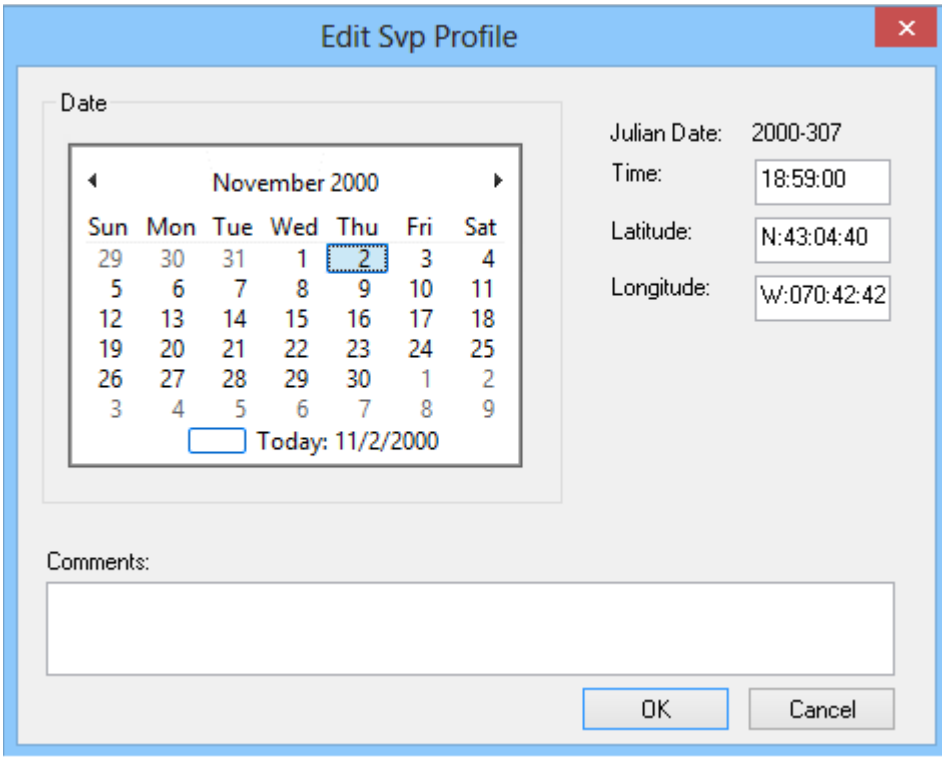
To edit geographic coordinates:

1. Select a profile by a time stamp.

Menu	Edit > Edit Profile

2. Select the Edit Profile command.

The Edit Profile dialog box is displayed, and the calendar box shows the day month and year of the selected profile.



3. Click in the Latitude coordinates field and type new coordinates.
4. Click in the Longitude field and type new coordinate.
5. Click **OK**.

The new geographic coordinates are displayed in the *Latitude* and *Longitude* sections of the Profiles area.

Delete data

To delete speed/depth data:

1. Dr dragging the cursor over the data points in the graphical display, or
2. click a row number in the table.

The selected data is highlighted.

Menu	Edit > Delete
Key	<Delete>

3. Select the Delete command.

The Speed/Depth data points are removed from the graphical display and the values are no longer displayed in the table.

Remove Profile

To remove a currently open profile from the SVP file:

1. Select a profile by clicking the time stamp in the *Profiles* list.

The depth/speed values are displayed in the editor.

Menu	Edit > Remove Profile
Key	<Ctrl+Delete>

2. Select the Remove Profile command.

The profile is removed.

Exit SVP editor

Close the SVP Editor.

Menu	File > Exit

1. Select the Exit command.
2. If you have not saved any changes you are prompted to do so.

The SVP Editor is closed.

9

Swath Editor

Swath Editor provides data-cleaning tools for multibeam, multi-transducer and LIDAR data.

You can also use Swath Editor to designate shoalest soundings.

In this chapter...

Swath Editor Interface

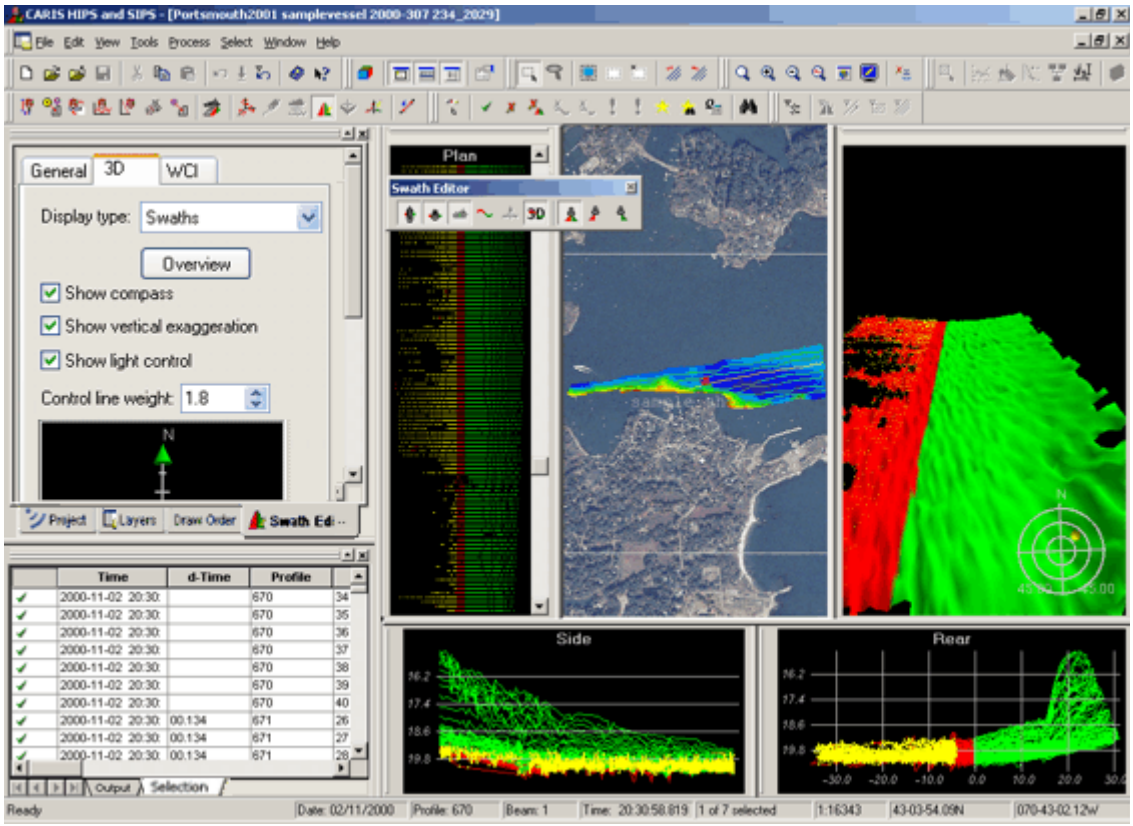
Menu	Tools > Swath Editor > Open
Tool	

A track line must be selected before Swath Editor editor can be opened.

1. Select the Open Swath Editor command.

Swath Editor opens and displays data in six different view windows (see "VIEWING DATA" ON PAGE 162).

The directory path and file name of the selected track line is displayed on the title bar. The image below shows the default layout for the Swath Editor windows.



The main elements of the Swath Editor are:

- the Swath Editor window containing a General tab and a 3D tab.
- Swath Editor data view windows
- Swath Editor toolbar. (See

Swath Editor Controls

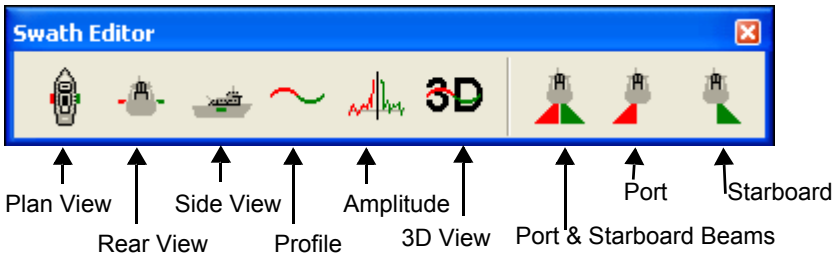
The Swath Editor tab has three tabbed pages:

- The General tab controls the data display in the Swath Editor windows (see "Swath Editor Display Options" on page 165).
- The 3D tab contains the Swath 3D view controls (see "3D View Controls" on page 172).
- The WCI tab controls display options for the water column imagery windows (see "WCI Controls in Swath Editor" on page 490).

If the dataset also contains side scan data, you can open Side Scan Editor while Swath Editor is open.

Swath Editor Toolbar

You can display or hide these views is controlled from the Swath Editor toolbar.



This is a floating toolbar that is automatically displayed when the editor is opened. The toolbar can be docked or floated over any area of the interface. Its appearance is controlled from the Toolbar dialog box (see "Display Toolbars" on page 536).

Viewing Data

Swath Editor consists of view windows, which open by default when the Swath Editor is opened.

- Plan View
- Rear View
- Side View
- Profile View
- Amplitude View
- 3D View
- "Plan View" on page 162
- "Rear View" on page 163
- "Side View" on page 163
- "Profile View" on page 163
- "Amplitude View" on page 163

"3D View" on page 164

As well, water column imagery can be viewed in Across-track and Along-track windows. These windows will display automatically if water column data is available when Swath Editor is opened. (See "Water Column Data in Swath Editor" on page 486

View windows can be displayed, hidden, resized or repositioned anywhere on the interface. The layout of the view windows is remembered by the application when you exit the editor.

Plan View

The Plan View is an overhead view of the swath profiles and beams. All profiles are spaced equally in the along-track direction. Across-track beam spacing is to correct scale. Each beam is represented by a square.

1. Select the Plan View command.

Menu	Tools >Swath Editor >Plan
Tool	

The number of swath profiles displayed in the Plan View is determined by three factors:

- height of the view
- point size of the beams
- distance (in points) between swath profiles (see "Data Display in Views" on page 164)

Soundings can be differentiated by applying options from the General page of the Swath Editor tab in the Control window to:

- colour by depth interval (see "Colour By Depth" on page 168)
- sun illumination (see "Light Control" on page 174).

Use the Display options in the Tools > Options dialog box to:

- colour soundings by their location in the swath (port or starboard beams) or
- colour soundings by the detection method used by the sonar (not supported by all sonars)

Rear View

Menu	Tools > Swath Editor > Rear
Tool	

The Rear View presents the survey as seen from behind the vessel. Each line represents one swath.

1. Select the Rear View command.

Side View

Menu	Tools > Swath Editor > Side
Tool	

The Side View presents the survey as seen from the side of the vessel with the vessel travelling from left to right. Each line represents soundings from the same beam.

1. Select the Side View command.

You can change the display (see "Data Display in Views" on page 164). and depth label settings for this view (see "View soundings as points or lines" on page 166).

Profile View

The Profile View displays a single swath.

1. Select the Profile View command.

Menu	Tools > Swath Editor > Profile
Tool	

The time stamp and profile number of the swath are shown in the status bar.

Amplitude View

The Amplitude View displays the beam-averaged amplitude data that is recorded with the swath bathymetry data.

1. Select the Amplitude View command.

Menu	Tools > Swath Editor > Amplitude
Tool	

Beam-averaged amplitude data are intensity values for each sounding depth created by a multibeam sonar system.

3D View

The three-dimensional (3D) View shows all soundings currently visible in the Plan View in three-dimensional format. The data in this window can be rotated and moved around (see "COMPASS" ON PAGE 173).

Menu	Tools > Swath Editor > 3D View
Tool	

1. Select the 3D View command.

Data Display in Views

You can show or hide the display of data in the Rear, Side and Profile Views:

- show or hide port beams
- show or hide starboard beams
- show both port and starboard beams (default setting)

Menu	Tools > Swath Editor > Port Beams/ Starboard Beams
Tool	

1. Choose the Port or Starboard Beams command.

The display is refreshed to show only the Port or Starboard data.

2. To view both starboard and port beam data, select the Port and Starboard Beams command.

Both Port and Starboard beams are displayed in the Rear, Side and Profile Views.

Swath Editor Display Options

Display of data in Swath Editor can be manipulated through the controls in the General, 3D View and Water Column tabs in the Control window.

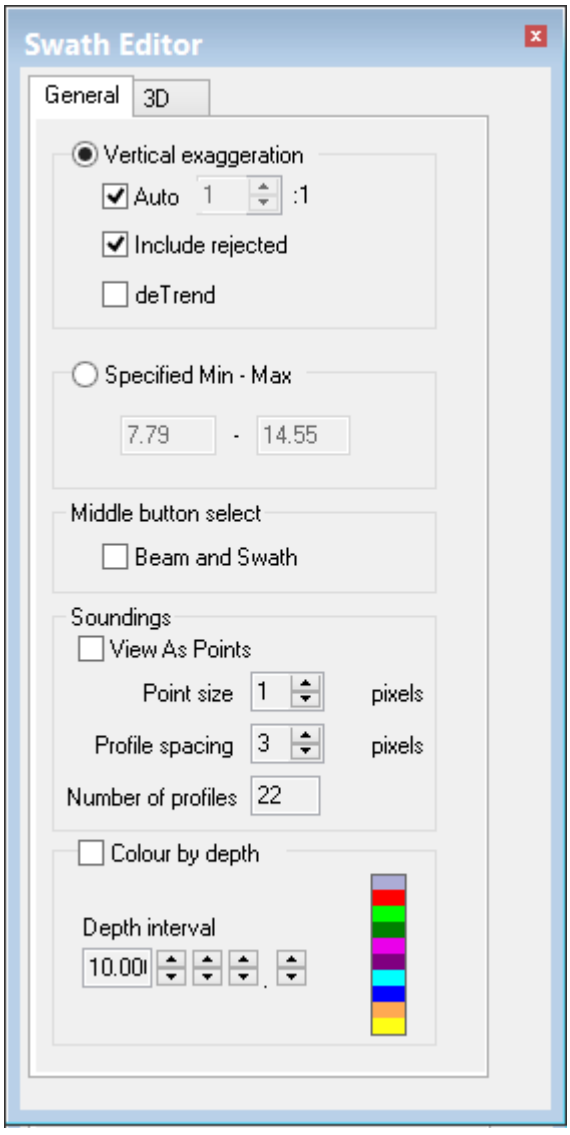
"GENERAL TAB" ON PAGE 165

"3D VIEW CONTROLS" ON PAGE 172

"WCI CONTROLS IN SWATH EDITOR" ON PAGE 490

General tab

The display options for the Plan, Rear, Side, Profile and Amplitude View windows are controlled through the General tab.



The following sections describe the options in this tab.

Vertical Exaggeration

The vertical exaggeration is the ratio of the vertical scale versus the horizontal scale used in the Rear and Side Views.

The vertical exaggeration can be determined automatically by including all visible depths or it can be manually set.

When *Auto* mode is on, the *Include Rejected* option controls whether or not the rejected data is used to determine the minimum and maximum sounding depth in the Rear and Side Views.

1. Deselect the *Auto* mode by clearing the check box.
2. Click the up or down arrow buttons beside the *Vertical Exaggeration* box to set the vertical exaggeration.

You can set the vertical and horizontal scale of the view to control the amount of rejected data in the view

3. Select the *Include Rejected* option to resize the display in the view to include rejected data.

Specified Min-Max

Use a constant scale as an alternative to vertical exaggeration by setting a minimum and maximum range for displaying data.

1. Select the *Specified Min-Max* check box to implement this option.
2. Type the minimum and maximum ranges for the scale.

The views are automatically refreshed to show data according to the scale.

Middle mouse button

Use the Middle Button Select option to select an individual swath or beam profile and see its location in the Display window.

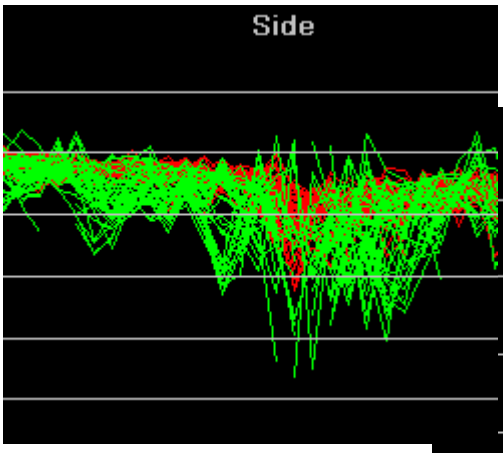
See "SELECT SWATH AND BEAM" ON PAGE 170 for examples on using this tool.

View soundings as points or lines

By default, soundings in the Side, Rear, and Profile views are connected by lines. Sometimes they can be more clearly distinguished if displayed as points.

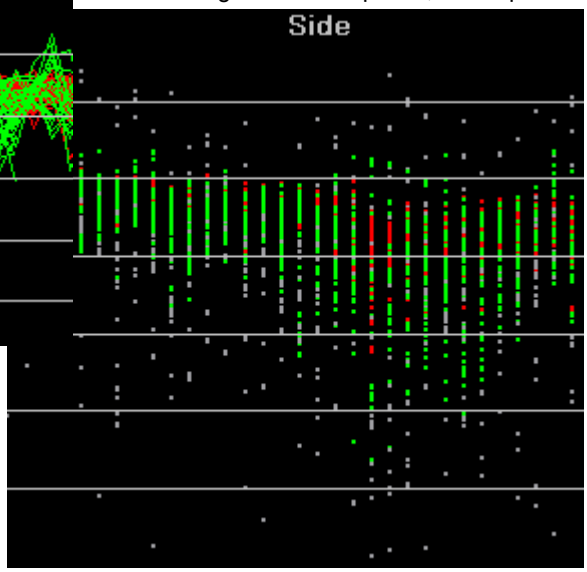
To change lines to points:

1. Select the General page of the Swath Editor tab in the Control window.
2. Select the *View as Points* check box.



Soundings connected by lines

Soundings viewed as points, size 2 pixels



3. Click the up or down buttons beside the *Point Size* box to set a size value between one and five pixels.

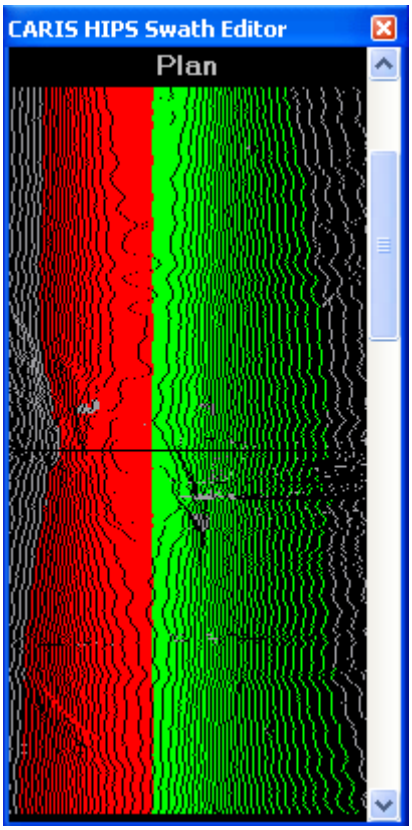
Changing the *Point Size* will also change the size of the points in the Plan view.

Profile spacing

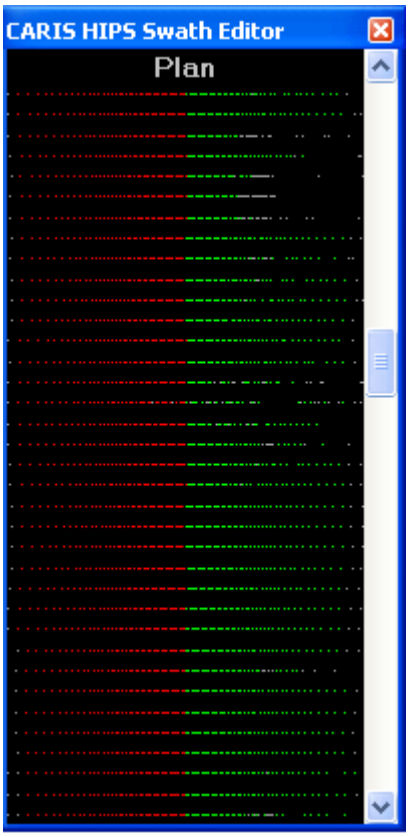
The *Profile spacing* control adjusts the displayed distance between the beams in the Rear, Side, Plan and 3D views. You can adjust the spacing between beams from 1 pixel to 10 pixels apart.

The *Number of Profiles* box displays the maximum number of profiles visible in the Plan view at its current size and profile spacing. This value is incremented up or down as you change the profile spacing.

The difference between minimum and maximum spacing of the same data within the same size window is illustrated below.



Profile spacing set to 1 pixel
256 profiles visible in this view



Profile spacing set to 10 pixels
38 profiles visible

Colour By Depth

Display soundings as colour coded by depth interval.

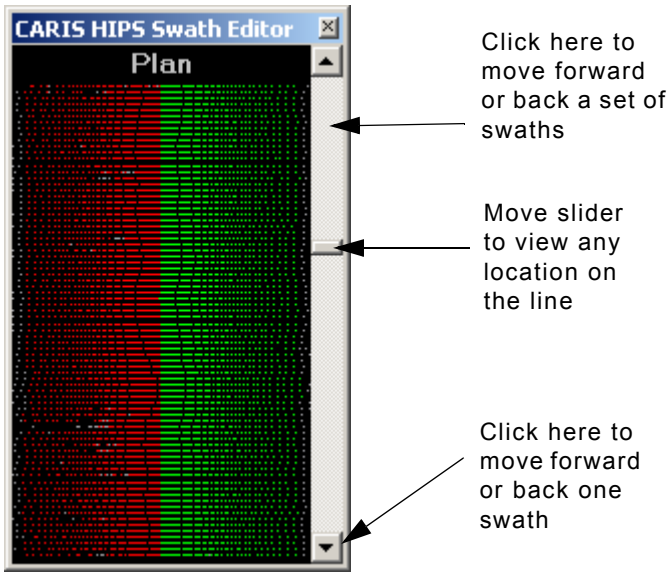
1. Select the General page of the Swath Editor tab in the Control window.
2. Select the *Colour by Depth* box so it is checked.
3. Click the up or down buttons beside the *Depth Interval* box to select a depth interval for the colour band.

The soundings are now displayed by depth level.

Examining Swath Data

The scroll bar in the Plan View represents the time period of the entire line. You can use the slider to move alone the track line or use the scroll arrows for more precision.

Each click on an arrow button moves the screen forward or backward one swath. Clicking the grey area of the scroll bar moves the screen forward or backward for a set of swaths. As you move along the track line, all the views refresh to display the same data as in the Plan View window.



As you move up or down the track line, all the views refresh to display the data from the swaths that are visible in the Plan view.

The geographic location of the swaths currently in view is shown by the sounding coverage outline in the Display window (see "Swath Coverage Outline" on page 171).

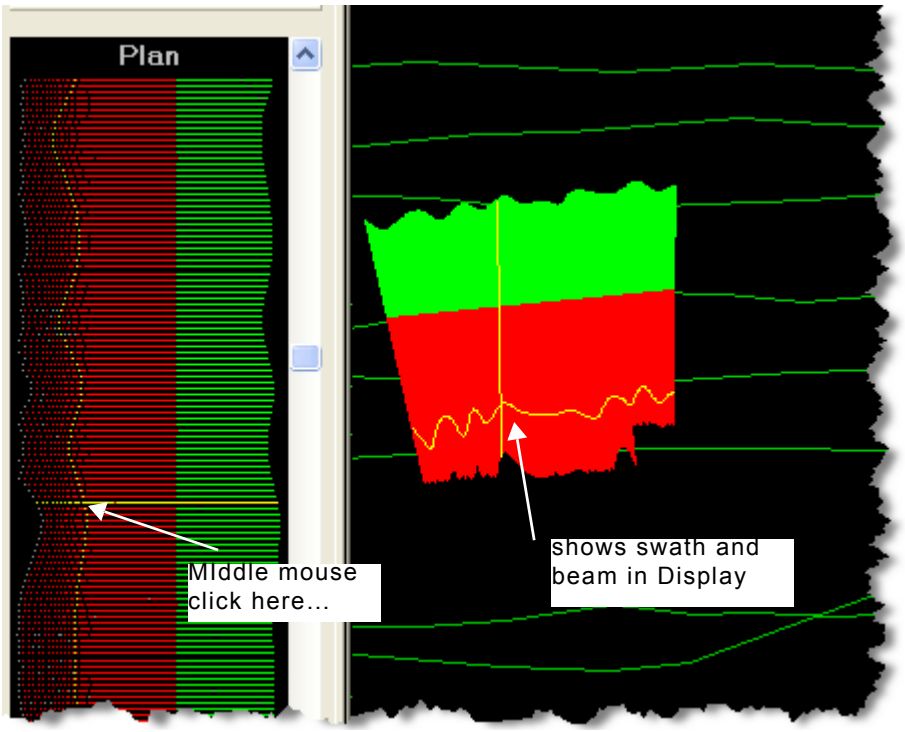
As an alternative, use the following methods to scroll through a track line:

- Press the space bar to move up the track line.
- Use the mouse wheel (if available) to move up or down the track line.
- Press <Ctrl + spacebar> to move down the track line.
- Use the mouse wheel (if available) to move up or down a track line.
- Press the <Home> or <End> buttons to move to the start or end of the track line.

Select Swath and Beam

The Middle Button Select option enables you select an individual swath or beam profile and see its location in the Display window.

By moving the cursor in the Plan, Rear, 3-D or Side View while holding down the middle mouse button, individual swaths and beams will be highlighted in the Display (see example below).



To view both beam and swath profiles

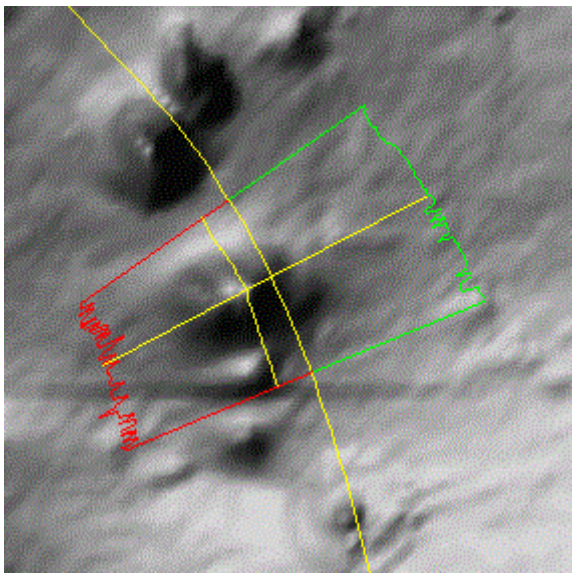
1. Select the *Beam and Swath* check box in the *Middle Mouse Select* section of the General tab in the Control window.
2. Place the cursor in either the Plan View, Rear View or Side View and click the middle mouse button.

The selected swath is highlighted.The other views are updated to reflect the same beam and swath location. The positions of the swaths and beams are also displayed in the sounding coverage outline.

If the Attitude Editor is open, you can view the sounding's position in relation to attitude data by the yellow line in any of the sensor windows (see "ATTITUDE EDITOR" ON PAGE 27).

Swath Coverage Outline

)The swath coverage outline is a marker in the Display window that shows the location of data currently visible in Swath Editor (as in the example below. The display of the outline is controlled from the Options dialog box (see "SENSOR EDITORS" ON PAGE 655 in the Reference Guide).



Automatic Recentre

The automatic recentre option ensures that when the coverage outline reaches the edge of the Display window (when scanning along the track line), the display redraws with the coverage outline in the centre. This option is available in the General Options dialog box.

Changing Track Lines

After examining a track line, you may want examine another line.

There are four options for navigating between track lines in Swath Editor:

- Select the Next Line command
- Select the Previous Line command
- Click a line in the Display window so that the line is opened in Swath Editor at that location
- Select a track line file name Project data tree in the Control window.

Menu	Select > Next Line / Previous Line
Tool	

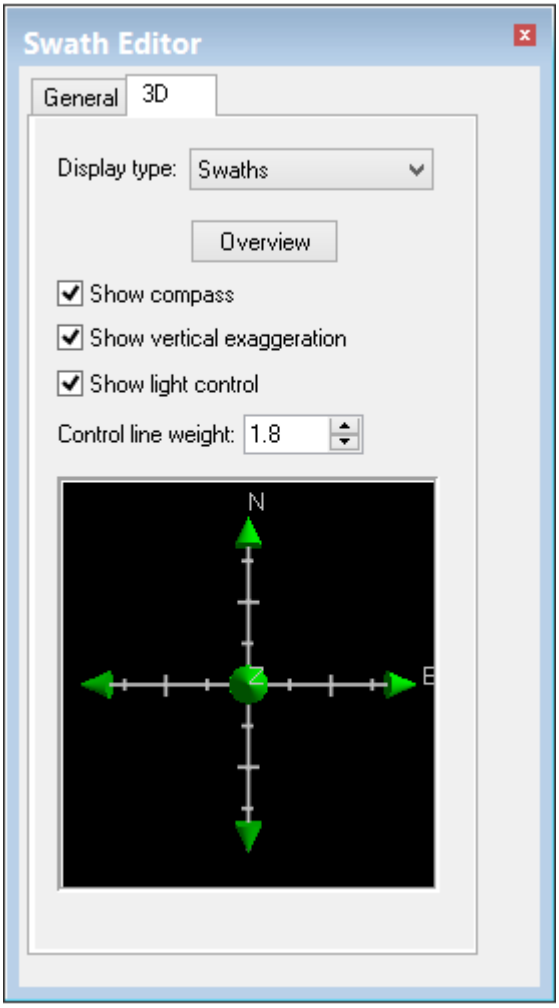
3D View Controls

Swath data can be displayed in 3D format. The 3D feature provides greater flexibility for examining and cleaning data.

1. Select the 3D View command.

Menu	Tools > Swath Editor > 3D View
Tool	

The 3D View window is displayed in Swath Editor. This 3D display is manipulated by three controls on the 3D View tab of the Swath Editor tab in the Control window (displayed below).



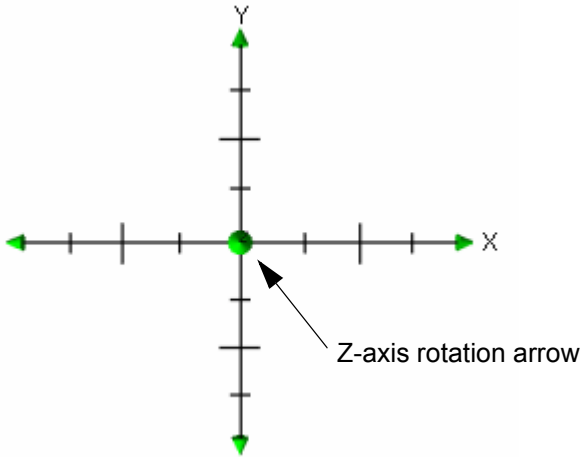
These controls are:

- the compass
- the vertical exaggeration slider
- the light control.

The line thickness of these controls can be altered by changing the *Control line weight* value in the 3-D View tab.

Compass

In the 3D View window, rotation along the X-Y-Z axis is handled by moving the arrow heads at the ends of the compass.



1. To rotate the image along the Z-axis, position the cursor on the middle arrow head, press and hold the mouse button, and drag the cursor so that the image is rotated. The compass can be rotated 90º.
2. To rotate the image along the X-Y axis, position the cursor on any of the other arrow heads, press the mouse button and drag the arrow head in a left-right or up-down direction. The compass can be rotated 360º.
3. Alternatively, the image can be rotated in the horizontal plane by grabbing the Z-arrow, then pressing the left-mouse button and moving the cursor up and down. Rotation around the Z-axis can be performed by pressing the <Shift> key and moving the cursor left and right in these windows.

The image is rotated in the same direction as the arrow heads.

4. To return the image to its original position, click **Overview** in the 3D View tab.

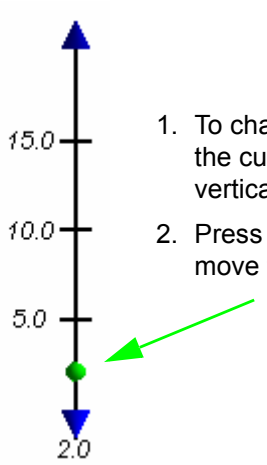
The image is restored to its original position.

Vertical Exaggeration Slider

By changing the vertical exaggeration, you can display seabed features more clearly.

Vertical exaggeration is controlled by the scale slider in the 3D View. The slider can be displayed or hidden by selecting (or

clearing) the *Show Vertical Exaggeration* check box in the 3D Controls tab.



1. To change vertical exaggeration, position the cursor over the green ball on the vertical exaggeration scale.
2. Press and hold the mouse button, and move the ball up or down the scale.

You can also change the vertical exaggeration using a dialog box.

3. Double-click the ball in the vertical exaggeration slider.

The Vertical Exaggeration dialog box is displayed.

4. Type a value in the Vertical Exaggeration field and click **OK**.

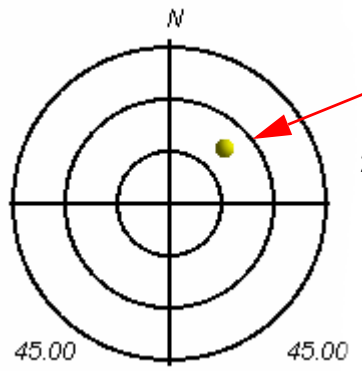
The display is refreshed to show the image according to the selected exaggeration scale. The scale value is displayed below the slider.

Light Control

The lighting control handles shading effects by controlling the position and angle of the light.

The elevation value (on the bottom right of the compass) varies from 90º to 0º starting from the middle of sundial. The azimuth value (on the bottom left of the compass) varies from 0º to 360º in a clockwise direction. The lighting control can be displayed or

hidden by selecting (or clearing) the *Show Light Control* check box in the 3D View tab.



1. Position the cursor over the ball in the lighting control.
2. Press and hold the mouse button, and move the lighting control ball with the mouse.

As the ball is moved, the azimuth and elevation values change.

You can also change the shading effect using a dialog box.

1. Double-click the ball in the lighting control.

The Set Sun Position dialog box is displayed.

2. Type new values in the *Sun Elevation* and *Sun Azimuth* fields and click **OK**.

The image is refreshed to the new lighting angle.

Pan and Zoom

There are three pan options for the 3D View:

- To pan along the X-Y axis: place the cursor in the 3D View, press the middle mouse button and drag the display to a new location.
- To pan along the Z-axis: place the cursor in the 3D View, press and hold the <Shift> key and middle mouse button and move the cursor in the window.
- To centre the 3D View window on a selected point: position the cursor over the point and click the middle mouse button.

There are three zoom options:

- Press and hold the <Ctrl> key and right-mouse button and drag the cursor up in the display to zoom into display.
- Press and hold the <Ctrl> key and right-mouse button and drag the cursor down in the display to out of display
- Rotate the mouse wheel in a counter-clockwise direction to zoom into the display, or rotate the mouse wheel in clockwise direction to zoom out of the display.

Close Swath Editor

To close Swath Editor

1. Click the Swath Editor tool button again. 

All Swath Editor view windows are closed and you are prompted to save any changes.

Click **OK** to save changes.

10

Tide Editor

Soundings are corrected for tide when Merge is applied. A tide file containing time-stamped tide observations is saved for each line. The tide file must span the entire time frame of the survey line, with observations taken at regular intervals.

This data may need to be edited or interpolated using Tide Editor.

In this chapter...

Open Tide Editor

Tide file information can be viewed in both graphical and tabular formats in Tide Editor. You can use the editor to edit an existing file or to create a new tide file to apply to survey lines before the Merge process.

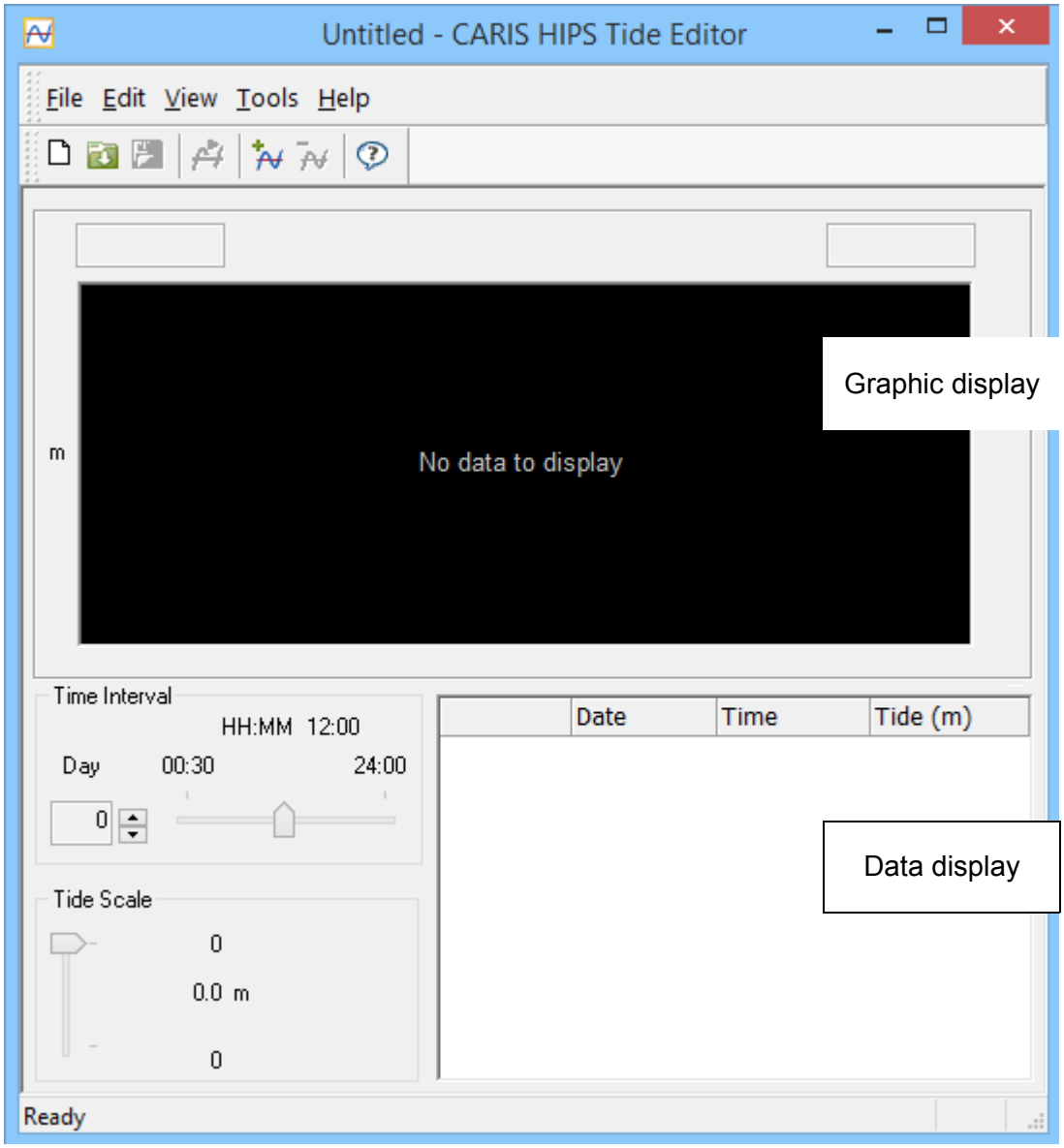
Tide files are located by default in …\Hips\version\Tide and have a .tid file extension.

Menu	Edit > Tide
Tool	

To open Tide Editor:

1. Select the Tide Editor command.

The Tide Editor interface is displayed. (For display options, see "DISPLAY OPTIONS" ON PAGE 184.)



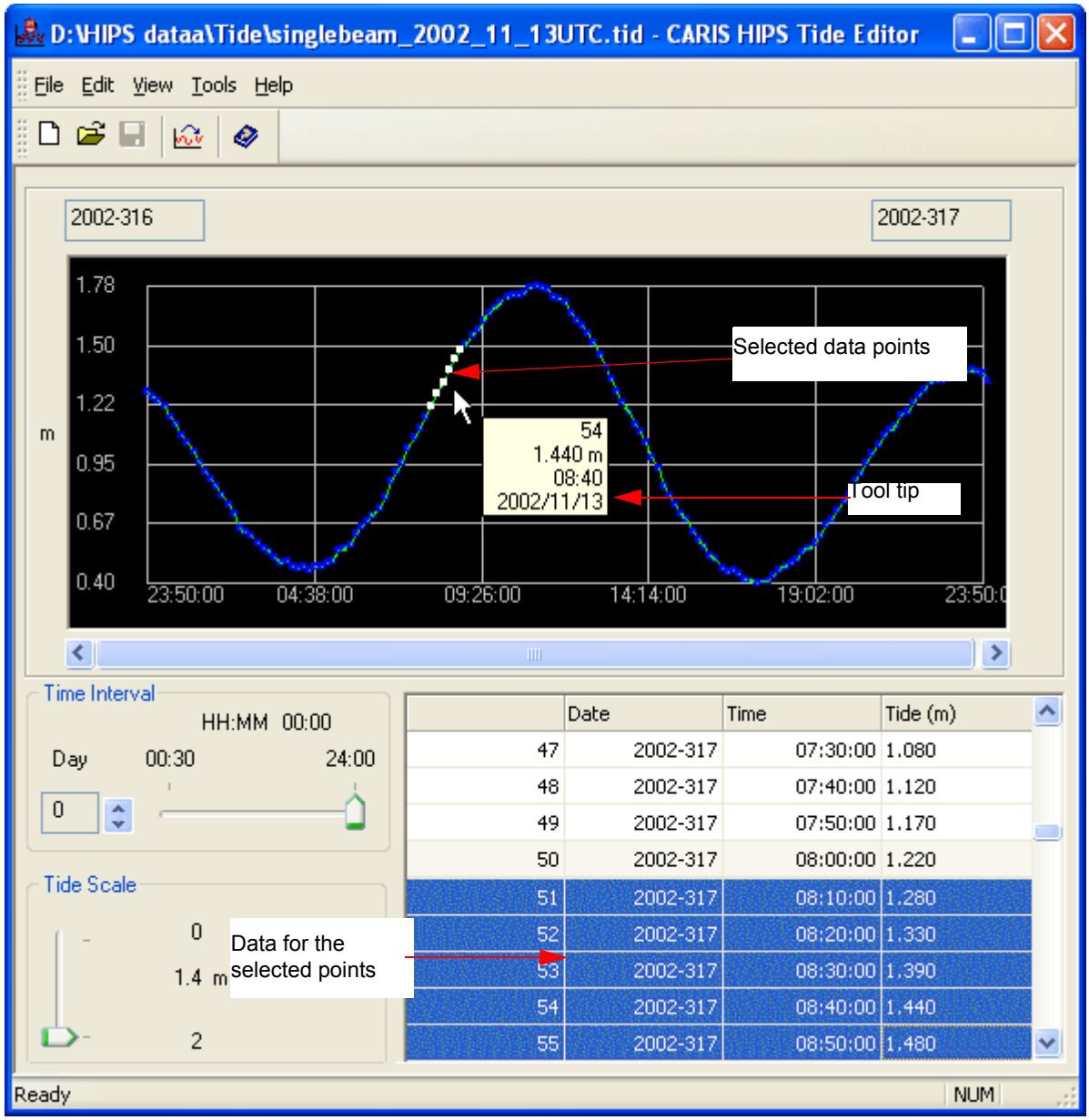
Menu	File > Open
Tool	
Key	<Ctrl+O>

To open tide data in the editor:

2. Select an Open command to display the Open dialog box.
3. Select a tide file and click **Open**.

The tide file is displayed in the editor.The top part of the editor shows a tide versus time graph of the data. Tide-level values are represented on the vertical axis. Time is represented on the horizontal axis. These can be adjusted using slider controls.

In the example below, the data points in the graph show as a dark blue on the lighter blue line. Selected points are white.



When data is selected points in the graph, the table below it is refreshed to display the rows containing the data for the selected points.

If you click in the table, those rows will be highlighted. You can move to from graph to table and back, and the corresponding data will be highlighted in both views.

When you change the time interval or time scale, you may need to click in the table again to see the highlighting for any selected data points.

Adjust tide scale

To control the vertical scale of the graphical display:

- Move the *Tide Scale* slide bar to select a range for the height of the vertical tide axis.

Adjust time interval

Change the horizontal scale by adjusting the *Time Interval* setting.

- The day option enables you to view data by 24-hour increments.
- The hour/minute option enables you to view data in increments of less than 24 hours.

1. Click the *Day* arrow buttons to select the range of days for displaying data.
2. If the *Day* value is set to zero then move the slide bar to select the minute/hour range for the display of data.

The display of tide data is adjusted for the new time interval along the horizontal axis.

Save a tide file

Save changes to an existing tide file.

1. Select the Save command.

The file is saved with the changes.

Exit Tide Editor

To close Tide Editor:

1. Select the Exit command.

Menu	File > Exit

You are prompted to save any unsaved changes.

2. Click **Yes** to save the changed file.

Tide Editor is closed.

Editing Tide Data

To edit data in an existing tide file.

1. Open the tide file.
2. Select a Date/Time/Tide cell in the tide table.
3. Remove the existing data and type a new entry.

The graphical display refreshes to show the new data.

To delete a tide entry:

1. Select the data points in the graph, or click the appropriate number cell in the tide table.

The data is highlighted.

Menu	Edit > Delete
Key	<Delete>

2. Select the Delete command.

The tide data is removed from the graphed line and the table.

Search time gaps

Tide gauges, recording tidal height observations, may be subject to occasional outages resulting in gaps in the tidal time series. The time gaps option enables you to search for these gaps. You can enter new data, if needed.

Menu	Tools > Options

1. Select the Options command.

The Options dialog box is displayed.

2. Enter a value (in minutes) in the *Time Gaps* box.
3. Click **OK**.

Menu	Tools > Next Time Gap
Tool	
Key	<Ctrl+G>

4. Select the Next Time Gap command.

When the interval (in minutes) between adjacent data points is the same as, or greater than, the *Time Gaps* value, the adjacent data points to the gap are highlighted in the colour set in Options.

5. Repeat the process, selecting the Next Time Gap command and moving through each data point on the graphed line.
6. Where necessary, enter new data between two values.

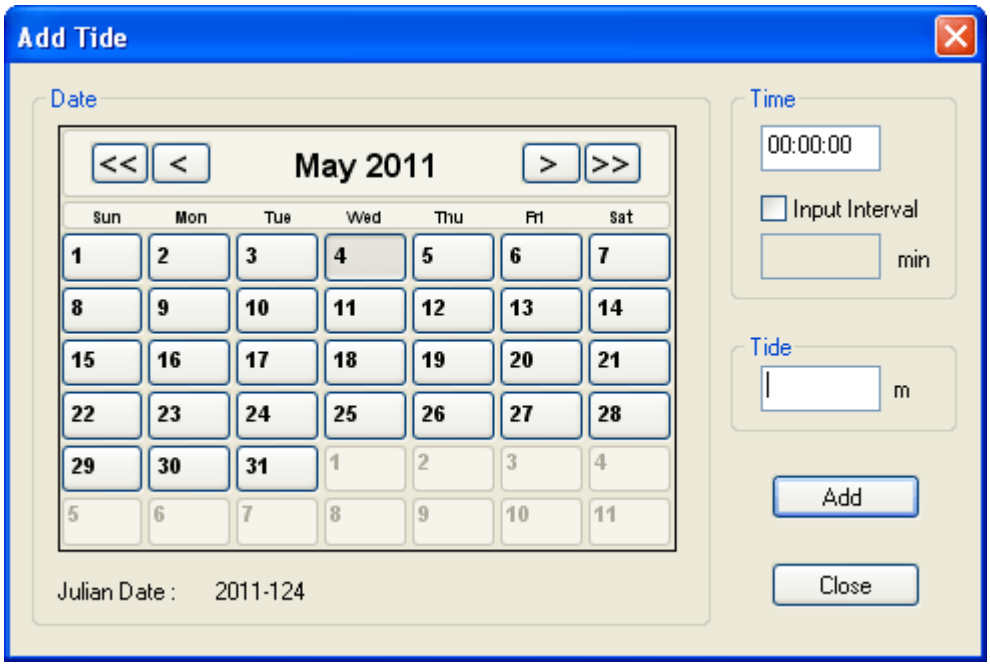
Enter new data

Enter data for a new or existing tide file.

Menu	Edit > Add Tide
Key	<Insert>

1. Select the Add Tide command.

The Add Tide dialog box is displayed.



2. Change the date (if necessary) by clicking the arrow buttons at the top of the calendar until you have selected the desired month and year.
3. Click the appropriate day button on the calendar.

The selected date (year and day) is displayed below the calendar.

The default time is the time of the currently selected tide record.

4. Click any of the three sections (00:00:00) in the *Time* box so they are highlighted and type the desired time (use the 24-hour clock).

The *Input Interval* option enables you to quickly enter a series of tide values at a regular time interval, using the <Enter> key. You do not need the mouse to move the cursor between fields.

5. If you want to regularly increment the times for tide data, click the *Input Interval* check box.
6. Enter the minutes that the time value is to be incremented.
7. Enter a tide value.
8. Click **OK**.

The new tide data is displayed in the editor. If you selected the Input Interval option, the time is incremented by the value set in the *Input Interval* box.

Create a New Tide File

Menu	File > New
Tool	
Key	<Ctrl+N>

You can create new tide file.

1. Select the New command.

Any tide file that is currently open in the editor is closed.

2. Select the Add Tide command and type a new tide entry (see "ENTER NEW DATA" ON PAGE 182).

The data is displayed in graph format in the editor.

3. Repeat until all values are entered.
4. When ready, select the Save or Save As command.

Menu	File > Save/Save As
Tool	
Key	<Ctrl+S>

The Save As dialog box is displayed.

5. Type a name for the file and click **OK**.

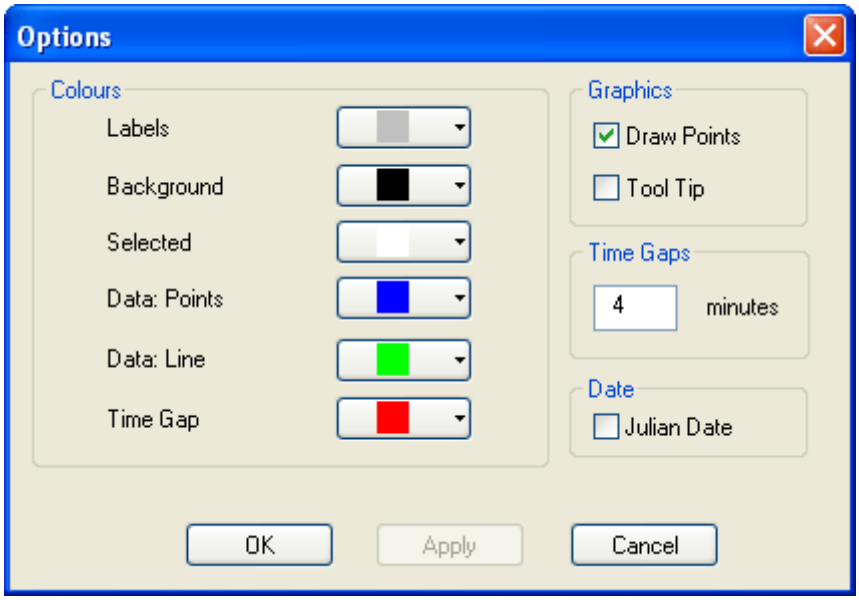
Display Options

You can adjust the display of graphical data: colours, display of data points and tools tips.

1. Select the Options command.

Menu	Tools > Options

The Options dialog box is displayed.



Colours

Set colours to identify display elements, for example to differentiate data points from the data line.

2. Select a colour from the drop-down colour palettes, or create a custom colour from the Windows colour picker.

Show/hide tide points

The editor displays the tide data as a line graph. You can display the points making up this line, or turn them off. This option is set on by default.

To turn off the display of tide points:

3. Clear the *Draw Points* check box.

Show/Hide tool tips

A tool tip is a label that displays a specific tide level on the graph, when the cursor is positioned over it. You can turn these tool tips on or off.

4. Select the *Tool Tip*s check box. If this option is checked then it is enabled.

Set Time Gaps

The time gaps option enables you to search for gaps in the tidal time series caused by occasional outages when recording tidal heights.

When the interval (in minutes) between adjacent data points is the same as, or greater than, the set *Time Gaps* value, the data points adjacent to the gap are highlighted.

Use the Next Time Gap command to search for these gaps. See "SEARCH TIME GAPS" ON PAGE 181.

5. Type a value in minutes in the *Time Gaps* box.

Set date display

The date in tide file table is displayed by default in regular calendar format. To display in Julian Day format:

6. Click the *Julian Date* box. (If the box is checked, this option is selected.)

11

Vessel Editor

Use Vessel Editor to create HIPS vessel files, and to edit sensor offsets and uncertainty estimates in those files.

In this chapter...

Vessel Editor Interface

Open Vessel Editor

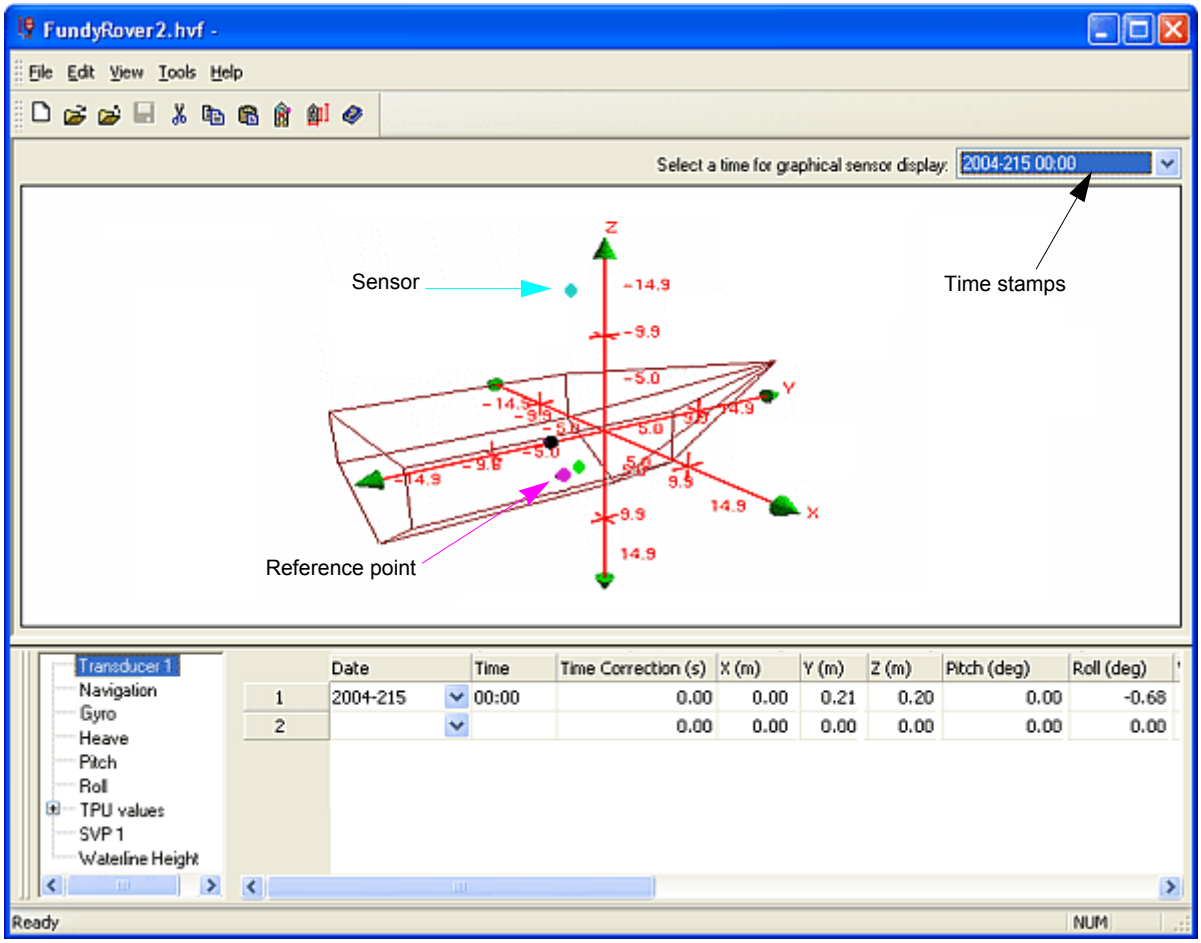
Menu	Tools > Editors > Vessel
Tool	

Vessel Editor is launched from the HIPS and SIPS interface.

1. Select the Vessel Editor command from the Edit menu or from the Tools toolbar.

Vessel Editor is displayed.

The image below shows an example of Vessel Editor displaying a vessel file and a vessel outline.



Vessel Editor displays a 3D wire-frame diagram of the vessel. The display can be manipulated in three dimensions using the 3D Display controls. See "3D Display Controls" on page 192. The location of a sensor is represented by a coloured dot or line. See "Colour options for sensor display" on page 194 to change the colours of the sensor display.

In the upper right-hand corner of the 3D display is a drop-down list containing all the unique time stamps in the vessel file. You

can select a time stamp and view the outline configuration at that single point in time.

Sensor data is displayed in a table below the 3D vessel outline. This data can be edited.

HIPS Vessel File (HVF)

The HIPS and SIPS Vessel wizard is used to set the parameters needed to create a new vessel file. The wizard takes you through the steps to enter vessel information and sensor configuration information based on the type of survey data.

The Vessel Wizard is launched from the New command. Step 1 of the Wizard asks for vessel information, such as the name of the vessel and survey date. The number of dialog boxes that follow Step 1 depends on the type of sonar selected in the step 2 dialog box.

To create an HVF using the Vessel Wizard see "CREATE A NEW HVF" ON PAGE 31.

Once the vessel file is created you can enter the sensor position data using a 3D outline for the vessel. See "CREATE VESSEL SHAPE OUTLINE" ON PAGE 39.

Then you can enter or edit sensor information for the active sensors as determined by the sensor parameters selected when the vessel file was created. You can also designated other sensors as active. See "SENSOR CONFIGURATION" ON PAGE 42

Open and Close Files

To open a saved file in Vessel Editor:

1. Select the Open command from the file menu.

The Open File dialog box is displayed.

2. Select the file from the list and click **OK**.

The file opens in Vessel Editor with the vessel's outline displayed in the editor, and a list of active sensors displayed in the Sensors list box.

Vessel Configuration Files that were created in previous versions of HIPS and SIPS can be opened and edited in Vessel Editor. The files are automatically re-saved as HIPS Vessel Files.

To close an existing file:

1. Select the Close command from the File menu.

You are prompted to save changes made in Vessel Editor.

2. Select a Save command.
3. Click **OK**.

To create a text file listing the parameters entered in Vessel Editor:

1. Open the HVF in the Vessel Editor.

2. Select the Vessel Report command. from the Tools menu.

The Save As dialog box is displayed. The default directory is `Hips\version number\HDCS_Data\VesselConfig`.

Exit the Vessel Editor

Close Vessel Editor and all files.

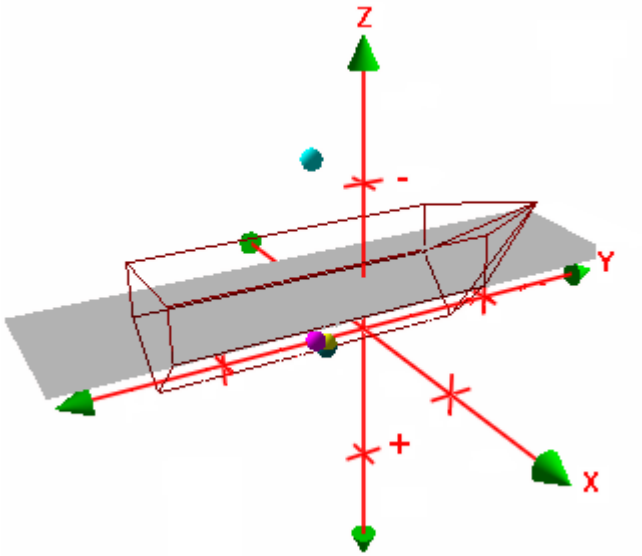
1. Select Exit from the File menu.

3D Display Controls

The outline displayed in the editor can be rotated, zoomed and panned using the compass (control axes), keyboard, and mouse.

Compass

Rotation along the X-Y-Z axis is handled by moving the arrow heads at the ends of the compass.



1. To rotate the image up to 90º along the Z-axis, click the middle arrow head and drag the cursor upward so that the image is rotated.
2. To rotate the image up to 360º along the X-Y axis, click any of the other arrow heads and drag the arrow head in a left-right or up-down direction.
3. Alternatively, the image can be rotated in the horizontal plane by grabbing the Z-arrow, then pressing the <Ctrl> key and the left-mouse button and moving the cursor up and down in the view. Rotation around the Z-axis can be performed by pressing the <Shift> key and moving the cursor left and right in these windows.

Pan

There are three pan options:

- To pan along the X-Y axis: place the cursor in the 3-D area, press the middle mouse button and drag the display to a new location.
- To pan along the Z-axis: place the cursor in the 3-D area, press and hold both the <Shift> key and middle mouse button and move the cursor in the window.
- To centre the 3-D Display window on a selected point: position the cursor over the point and click the middle mouse button.

Zoom

The zoom option is similar to pan.

1. Press and hold both the <Ctrl> key and right-mouse button while dragging the mouse up or down to zoom in or zoom out.

Modify Toolbar

The Vessel Editor toolbar is visible below the menu bar when the application is opened. The toolbar can be un-docked from this location and positioned within the interface, or on your desktop.

You can also hide the default toolbar by removing the check mark next to Toolbar on the View menu.

To make the toolbar visible again,

1. Select Toolbar from the View menu so it is checked.

Move toolbars

Toolbars can be moved to any location on the desktop:

1. Position the cursor over any area of the toolbar not covered by a button,
2. Press and hold the mouse button while dragging the toolbar to a new location.
3. Release the mouse button to position the toolbar.

Toolbars will automatically dock when they are close to certain areas of the interface. To stop the toolbar from automatically docking, hold down the <Ctrl> key while moving the toolbar.

Modify appearance

You can alter the appearance and size of toolbar buttons.

Menu	View > Customize

1. Select the Customize command.

The Customize dialog box is displayed.

2. Select any of the following three options by checking a box.
   - *Tool Tips*: Display the name of a command button when the cursor is placed over it.
   - *Cool Look*: Remove the border outline from the buttons.
   - *Large Buttons*: Display buttons with 32 x 32 pixel icons instead of 16 x 16 pixel icons.
3. Click **OK**.

Colour options for sensor display

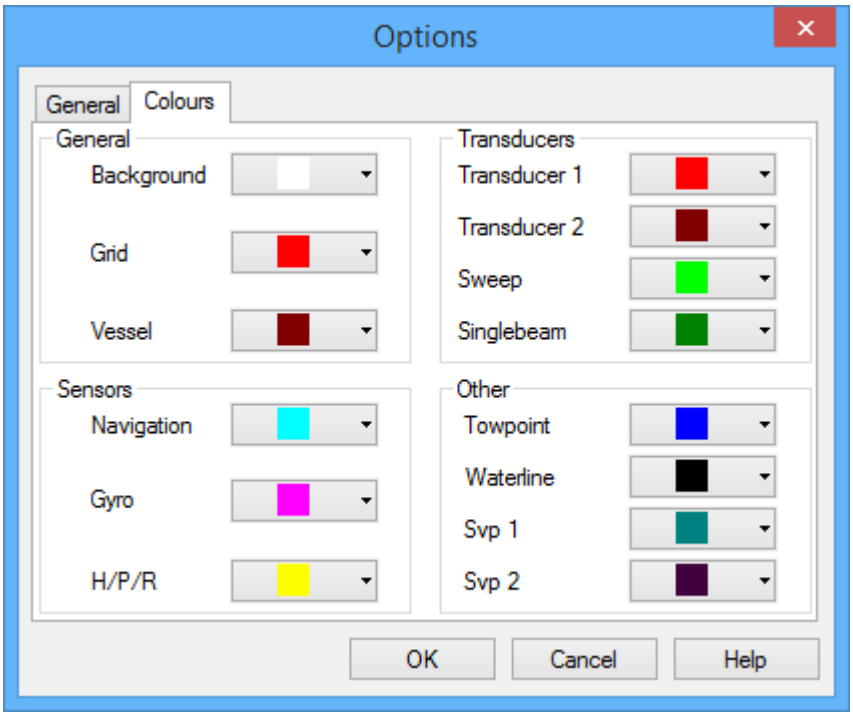
The locations of sensors and other equipment in the Vessel Shape 3D display are shown by colour markers. (You can set other colours for sensor markers.)

Menu	Tools > Options

1. Select the Options command.

The Options dialog box is displayed.

2. Select the Options - Colours tab.



3. Select a colour from the pull-down palette, or create a custom colour from the standard Windows colour picker.
4. Click **OK** to activate the changes and close the Options dialog.

Index

Numerics

A

B

C

D

E

F

G

H

I

L

M

N

O

P

R

S

T

U

V

W

Z