**caris**®

caris®

HIPS and SIPS **7.1**
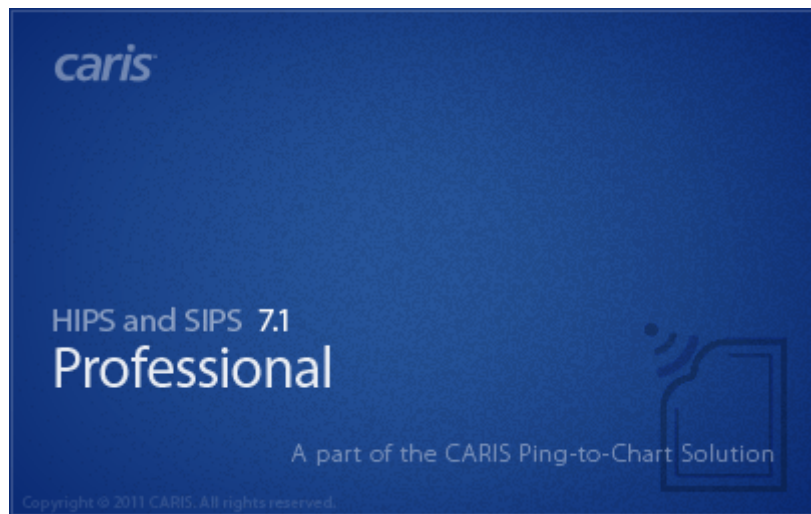Professional

A part of the CARIS Ping-to-Chart Solution

# HIPS and SIPS 7.1 Service Pack 2
# HIPS and SIPS Editors

Trademarks

CARIS, CARIS software and the CARIS logo are registered trademarks or trademarks of CARIS.

All other registered trademarks or trademarks are property of their respective owners.

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

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)

# Preface

This guide contains information about the following HIPS and SIPS 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 CARIS HIPS and SIPS Users Guide.

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

# Table of Contents

*1*

*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.

### In this chapter...

# Attitude Editor Interface

Attitude Editor is displayed in the HIPS and SIPS interface. To open the Editor:

1. Select a track line so it is highlighted.

2. Select the Attitude Editor command.

The Attitude Editor is displayed(as shown below).

| | |
|---|---|
| **Menu** | Tools > Attitude Editor |
| **Tool** | |
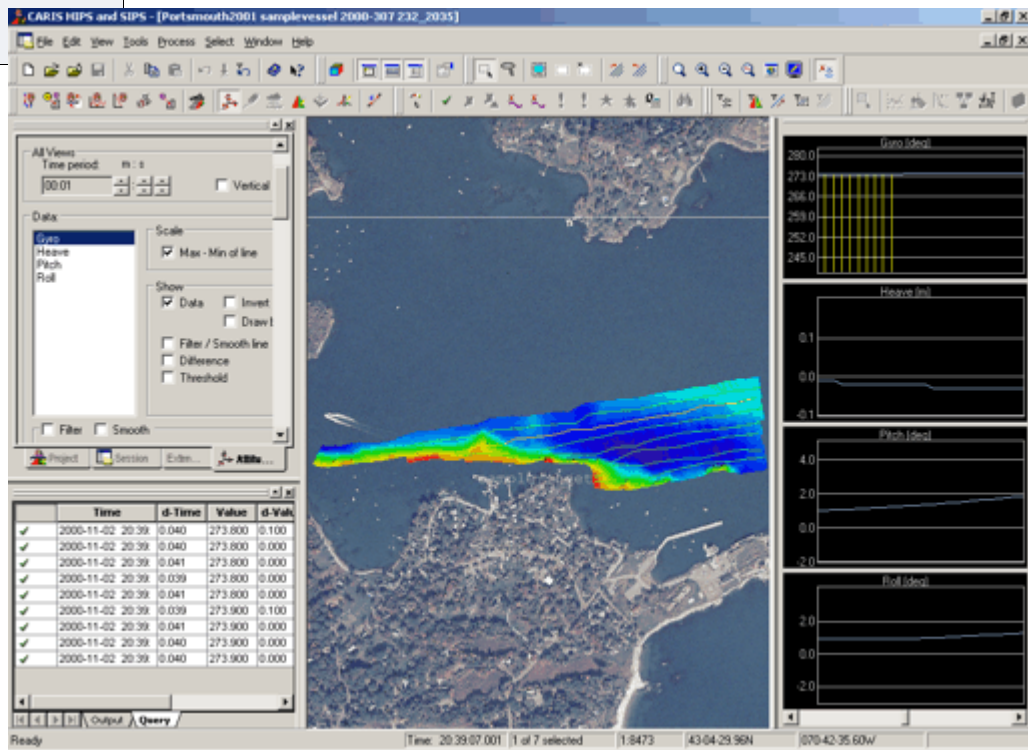
You can open the Attitude Editor while the Swath Editor is active and examine attitude and sounding data at the same time (see "SWATH EDITOR" ON PAGE 201).

The Project-Vessel-Day directory and file name of the selected track line is displayed on the title bar.

Data is graphed in the windows according to an X-Y axis. In the default layout, the X-axis represents time and the Y-axis represents data.

By default, the following data is displayed in the editor:

• **Gyro:** Gyro observations are displayed in degrees from positive 1 to 360.

• **Heave:** Heave observations are displayed according to the Units setting in the Options 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.

However, this is not the only data that can be displayed in the editor. All sensor data in HIPS/SIPS format can be displayed except for depths, navigation, and side scan data.

Besides the above sensors, the following sensors can be displayed (if available):

- loaded tide

- side scan gyro

- side scan cable out

- side scan depth

- side scan height

- delta draft

- GPS height

- GPS tide

- speed on water

Applanix's MS POV systems calculate TrueHeave values from real-time heave values. The computed TrueHeave data and real-time heave values are stored in a binary file that is not converted to HIPS/SIPS format when other data is imported. These files have to be separately loaded to the track line.

.

## Display Options

Show/hide sensor data

You can control which attitude sensors windows are displayed in Attitude Editor. By default Gyro, Heave, Pitch and Roll sensor windows are opened. However, if the survey data does not contain data from one of these sensors, that sensor will not be displayed. (It can be turned on with the Sensor Layout command.)

| Menu | Tools > Sensor Layout |
| --- | --- |

1. Select the Sensor Layout command.

The Sensor Layout dialog box is displayed.
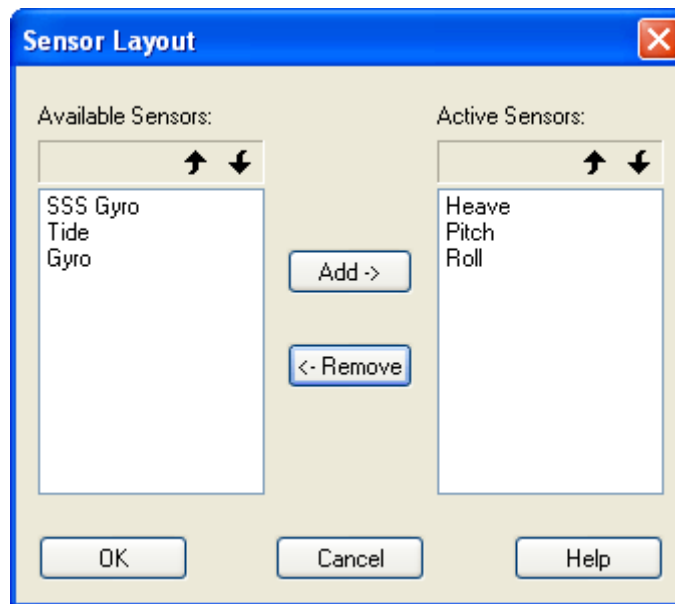


2. To remove a sensor from the *Active Sensors* list, select the sensor and click **Remove**.

3. To add a sensor to the *Active Sensors* list, select a sensor from the *Available Sensors* list and click **Add**.

4. Set the order that sensor windows are displayed by selecting a sensor and clicking the up or down arrow buttons, or by selecting the sensor and dragging it to a new position.

5. Click **OK** to apply settings.

The Active sensors are displayed in Attitude Editor according to the selected settings.

## Colour Display

The colour for displaying the various graph lines can be changed.

| Menu | Tools > Options |
| --- | --- |

1. Select the Options command.

The Options dialog box is displayed.

2. Click the Display tab and click the *Sensor Editors* category so it is highlighted.

3. Select a colour for the sensor filter line from the drop-down colour palette, or create a custom colour from the standard Windows colour picker.

4. Click **OK**.

See "DISPLAY" ON PAGE 80 for more information on colour options for the Attitude Editor.

# Attitude Editor tab

The Attitude Editor tab is visible in the Control Window when Attitude Editor is opened.
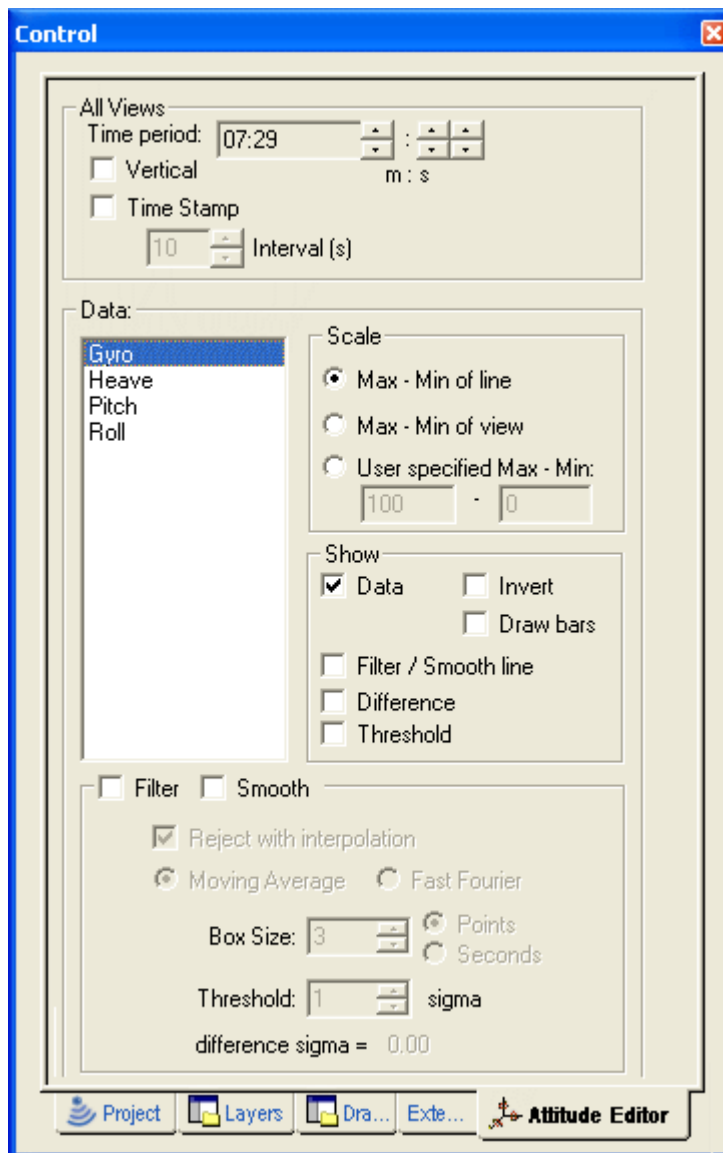


The tab contains settings for three types of functions:

## All Views

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

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

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.

6. To view the Attitude Editor in vertical mode, click the *Vertical* check box.

7. Select the *Time Stamp* check box to display the time stamp in the windows.

8. Click the *Time Interval* up or down arrow buttons to select an interval for time stamp display.

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

## Data

The *Data* fields control the graphs of the raw sensor data in the Attitude Editor window. Highlight the type of data: Gyro, Heave, Pitch or Roll, from the list and then apply a Scale and / or Show option.

## Scale

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

1. Select a sensor from the *Data* list.

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.

   • Select *User-defined Max-Min* to define maximum and minimum scale and type the values in the maximum and minimum fields.

## Show

The *Invert* command displays the data in a "mirror-image" view. This function is useful for viewing towfish depth in the water column.

1. Select an attitude data type in the *Data* list.

2. Click the *Data* check box to display the graph of that type of data in the Altitude Editor.

To invert the data in the graph,

3. Select the *Invert* command.

The *Draw Bars* function displays vertical bars for each sensor value in the graphs. This option makes it easier to see time gaps between sensor values.

The following three options graph othre lines for comaprison to the data.

• *Filter/Smooth Line* graphs the computed values derived from the adjacent data points and displays this line in the graph.

• *Difference* graph a line showing the difference between the original data values and the moving average or FFT value for each data point.

• *Thresholds* display 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
   ```

When you close the Attitude Editor, you are prompted to save any changes. This saves any smoothing changes to the SmoothedCoefficients file

## Filter and Smooth

You can reject data with residual values that fall outside user-defined threshold limits, and apply these changes to attitude data across entire track line(s), using the Filter andSmooth options.

(These options can also be applied from the Attitude filter. See .)

The Smooth Sensor function is used to smooth out localized variability. Any sensor data that can be viewed in the Attitude Editor can be smoothed.

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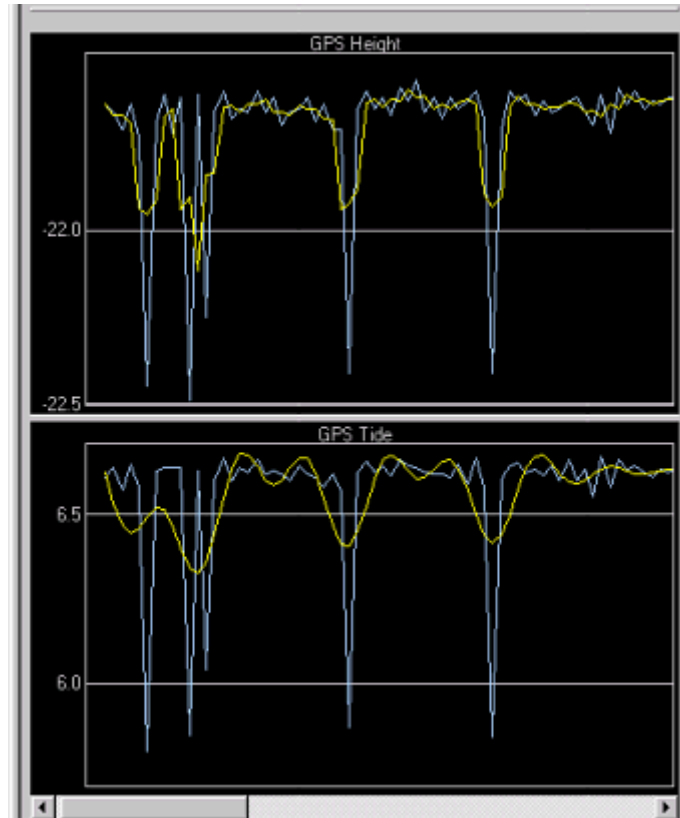


. The parameters for creating the smoothed data are saved to the SmoothedCoefficients file in the HCDS_Data\Project\Vessel\Day\Line folder. The file is in XML format. This file is applied to the track line during the merge process and SVP Correction.

Moving Average and Fast Fourier Transform

There are two methods available for smoothing: Moving Average and Fast Fourier Transform (or FFT).

- 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 Transformation* 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* (Step 6) 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).



FFT value or mean calculated for data point

Adjacent data points used for calculation of value

Box Size = 5
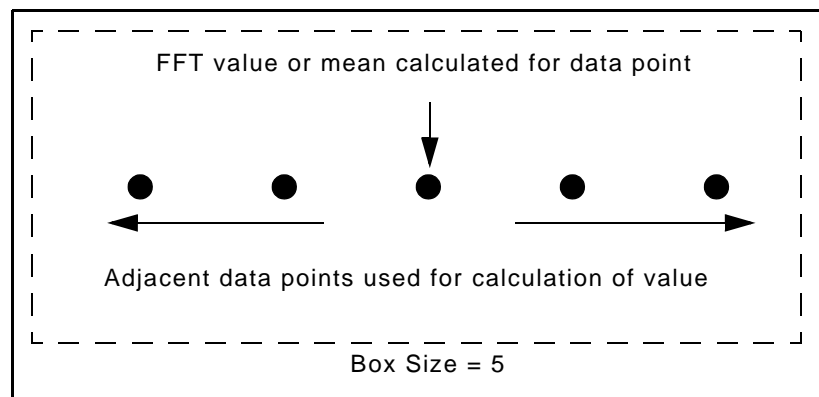
1. Make sure that the Control window is open and the Attitude Data tab is visible.

2. Select a sensor in the *Data* list box, or click inside a sensor window in the Attitude Editor.

3. Select the *Smooth* option.

4. Select either the *Moving Average* or *Fast Fourier* smoothing options.

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. Click **Yes** to save smoothed data.

# 2

## *Mosaic Editor*

Mosaic Editor is used to create and edit mosaics, and to analyze the sediment properties of backscatter data through the created imagery.

### *In this chapter...*

# Overview of Mosaic Editor

In Mosaic Editor you can correct multibeam backscatter and sidescan sonar imagery. You can also create mosaics from imagery data.

Overview of Mosaic Editor workflow
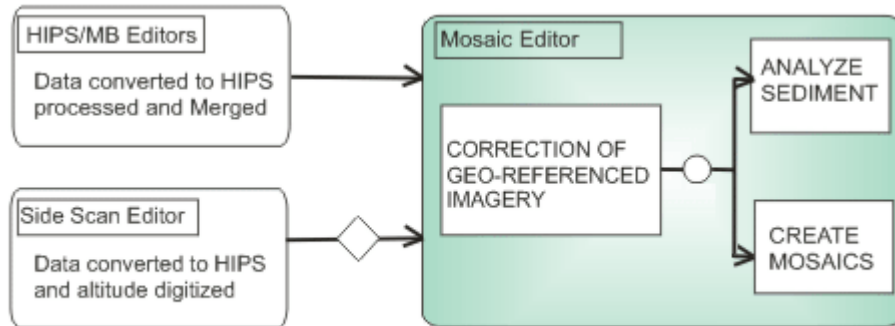


The full workflow for image processing in Mosaic Editor is described in PROCESS IMAGERY DATA in the Users Guide.

> Mosaics created as field sheet layers in HIPS and SIPS prior to version 7.0, can be opened in Mosaic Editor for comparison purposes, but they cannot be edited.

# Open Mosaic Editor

To open Mosaic Editor:

1. Select the Open Mosaic Editor command.

| | |
|---|---|
| **Menu** | Tools > Mosaic Editor > Open |
| **Tool** | |

The Mosaic Editor and Beam Pattern toolbars are displayed. The Control Window is refreshed to display the three control tabs of Mosaic Editor.

## Mosaic Editor toolbar
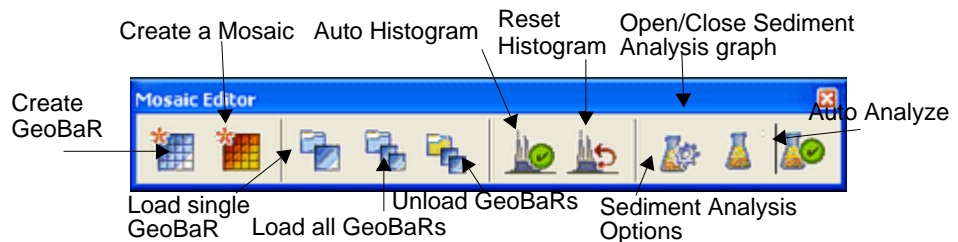


The Mosaic Editor toolbar contains the following controls:

- **Create a GeoBaR**: based on options selected in the Process tab.

- **Create a Mosaic**: from available GeoBaRs.

- **Load a single GeoBaR** for the selected trackline(s): you will be prompted to choose which GeoBaR(s) are loaded for each line.

- **Load all GeoBaRs** for the selected trackline(s)

- **Unload all GeoBaRs** for the selected trackline(s)

- **Sediment Analysis Options**: choose when to create a sediment analysis file for a line

- **Open/Close Sediment Analysis Graph:** display the graph or close an open graph.

- **Auto Analyze:** Apply sediment analysis to all selected lines for which GeoBaRs have been created.

## Beam Pattern toolbar

The Beam Pattern toolbar has the following controls:

- **Create Beam Pattern**: create a Geocoder beam pattern file.

- **Load Beam Pattern:** open an existing Beam Pattern File for use during Beam Pattern Correction.
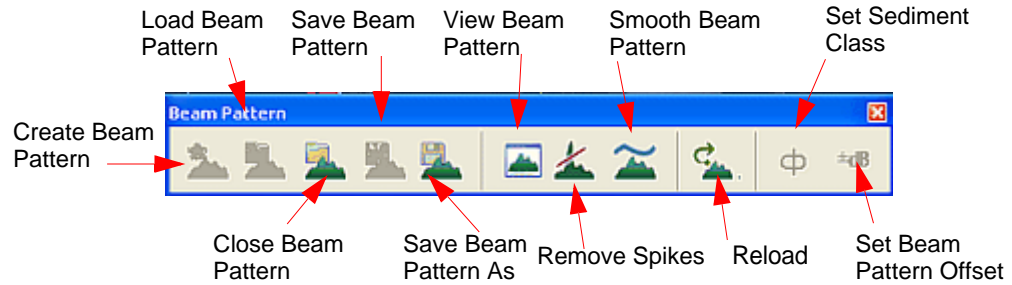
- **Close Beam Pattern:** close an open Beam Pattern file.

- **Save Beam Pattern:** save an open Beam Pattern file.

- **Save Beam Pattern As:** save an open Beam Pattern file under another name or location.

- **View Beam Pattern**: open the Beam Pattern graph window to display the average beam pattern intensity as a function of angle from nadir. With the Geocoder processing engine backscatter dB (decibel) is displayed, and with SIPS engine, intensity values are displayed.

- **Remove Spikes:** removes anomalies from the Beam Pattern Graph (not available with the Geocoder processing engine).

- **Smooth Beam Pattern:** uses a moving average filter to reduce noise in the beam pattern (not available with the Geocoder processing engine).

- **Reload**: restore the beam pattern file to the point just after you last saved it.

- **Set Sediment Class**: choose the sonar frequency (Hz) and mean grain size and set it for the selection (not available with the SIPS processing engine.)

- **Set Beam Pattern Offset** (dB) for the selection (not available with the SIPS processing engine).

Side Scan Editor

If you wish to see a waterfall display at the same time as your georeferenced imagery, you can open Side Scan Editor as well as Mosaic Editor.

1. Open Mosaic Editor.

2. Select a single trackline and open Side Scan Editor.

The SIPS Preview window will open and display the waterfall view.

# Mosaic Editor Control tabs

The tabs are

- Process - used to set the options for the creation of GeoBaRs

- Analysis - used to perform sediment analysis on backscatter

- Edit - used to edit the display of mosaics or GeoBaRs "on-the-fly" and change the draw order of the GeoBaRs

The image below displays the tabs in the Control window.
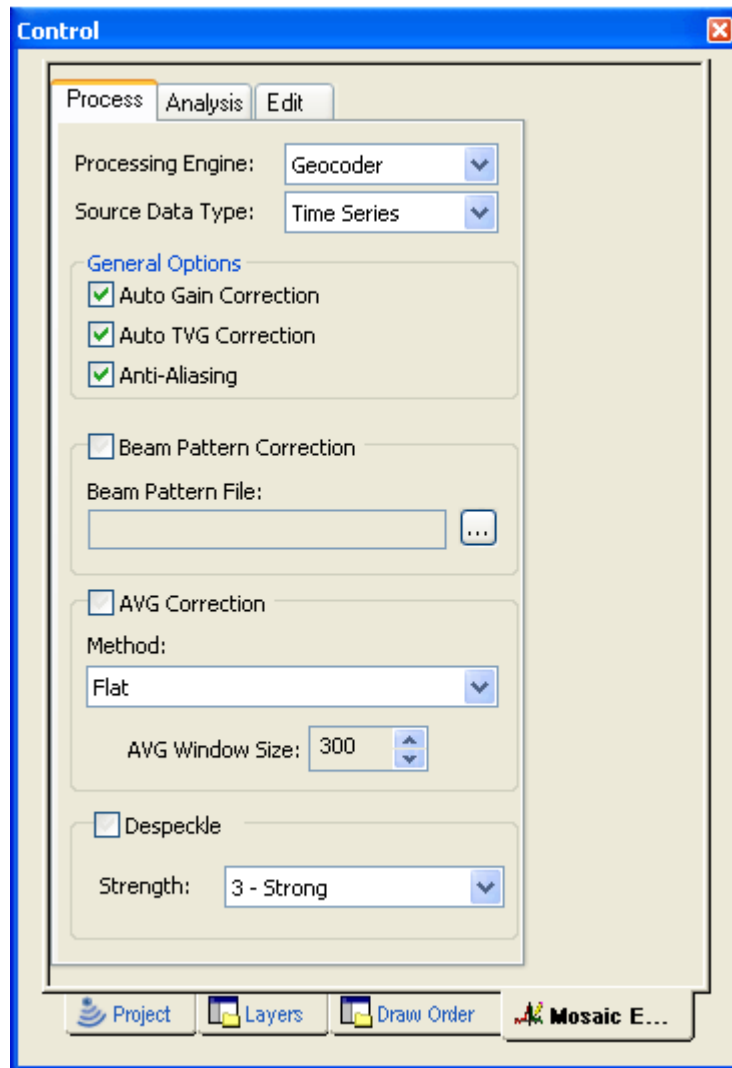
# Properties of Mosaics and GeoBaRs

Mosaics and GeoBaRs created in Mosaic Editor are listed in the Layers tab together with their attribute layers. When a layer is selected its properties are displayed in the Properties window. These properties can be viewed, and some edited, when a layer is selected.
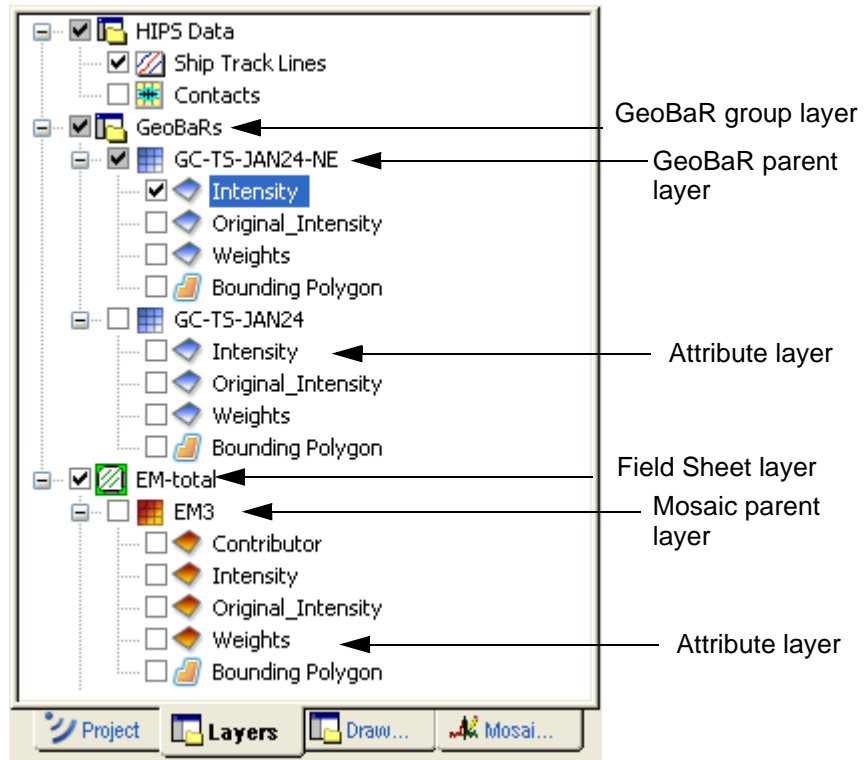


## Mosaic Layers

Most of the information in the Properties for a Mosaic parent layer is read-only, however, these fields can be edited:

| Properties | Function |
|---|---|
| **General** | |
| Vertical Coordinate System | Select the appropriate vertical coordinate system for the surface from a drop-down list. |
| **Creation**: | |
| Data Start Date | Date can be modified using the drop-down calendar |

| Properties | Function |
|---|---|
| Data End Date | Date can be modified using the drop-down calendar |
| Comments | Click in the field to type in comments. . |
| Platform Name | Click in the field to type in comments. |

The properties for the child layers of a mosiac are the same as for other surface attribute layers.

# GeoBaR layers

The Properties for the GeoBaR parent layer list the GeoBaR source files. It also contains the switch to apply consistent properties for all GeoBaRs.
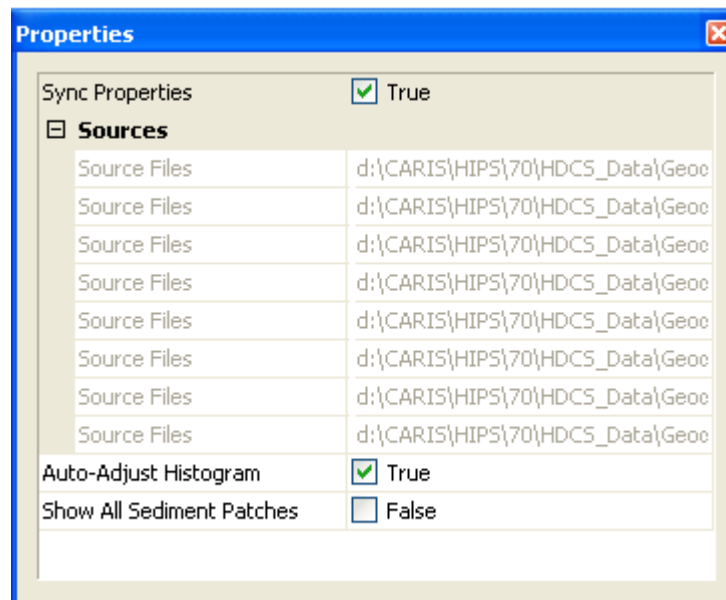


Sync Properties are set to True by default. This controls that when a property on a GeoBaR layer is set or changed, all other GeoBaR layers are changed to match it.

To have different properties apply individual GeoBaRs:

1. De-select the check box to set the value to False.

If there are GeoBaRs open in the Display, the *Sources* fields will display the source files for the open GeoBaRs.

The *Auto-Adjust Histograms* check box allows the same range of intensity values (min/max) to be set for all GeoBaRs created. By default this is set to True.

To disable this auto-adjust and have different intensity values for GeoBaRs:

2. De-select the *Auto-Adjust Histograms* check box to set the value to False.

The *Show All Sediment Analysis Patches* option is turned off (set to False) by default.

To display all patches within the selection area along a line, when the sediment analysis tool is activated:

3. Select the *Show All Sediment Analysis Patches* check box to set it to True.

GeoBaR settings in the Properties window are saved when a project is saved to a session file and will be displayed and applied when the session file is reopened.

# 3

## *Process LIDAR Data*

LIDARLIDAR data converted to HIPS format can be viewed and cleaned in Subset or Swath Editor. You can select, superselect 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 (lat/long 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.

For conversion of LIDAR data to HIPS format, see the Convert Data chapter of the HIPS and SIPS Users 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 "WAVEFORMS VIEW" ON PAGE 36).

# Workflow

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

## Lidar workflow
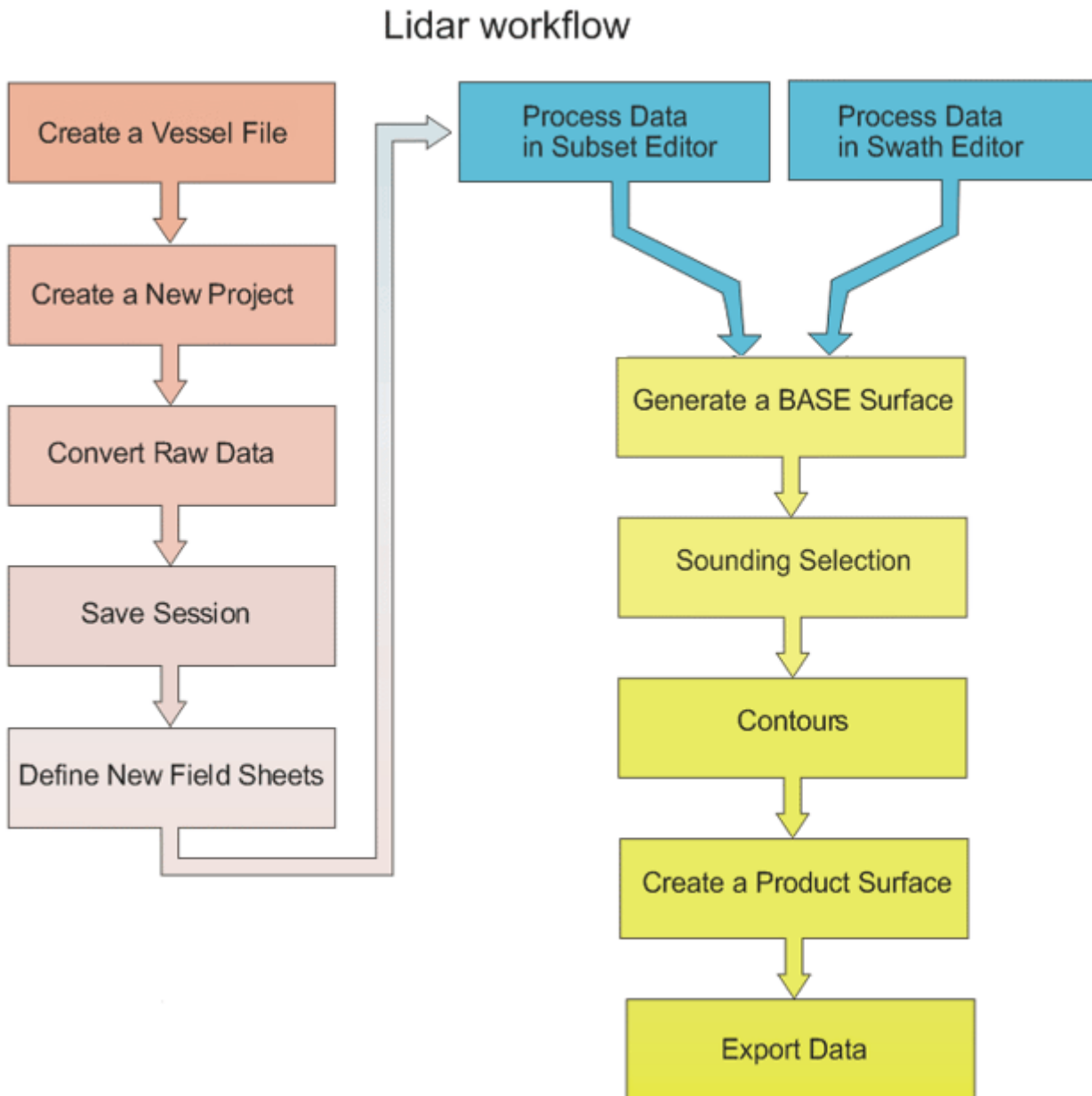


1. Create a new HIPS Vessel File (HVF) containing only lidar and navigation information.

2. Create a Project-Vessel-Day directory tree structure to store data.

3. Convert raw data files to HIPS format and store data in the project directory.

4. Save the currently open data as a session file.

5. Create a field sheet within a defined area and with a defined coordinate system.

6. Process data using Subset Editor or Swath Editor.

7. Process Data in Subset Editor
   - Create a Subset Tile layer within the field sheet.
   - 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.

8. Process Data 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.

9. Create a BASE surface.

10. Add contours and other layers.

11. Designate shoalest soundings .

12. Create a product surface.

13. 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 3D and 2D view and cleaning progress can be tracked using the Subset Tile function.

If Swath Editor is used the LIDAR tab and waveform view are also available.

The following image shows SHOALS lidar data displayed in Swath Editor.
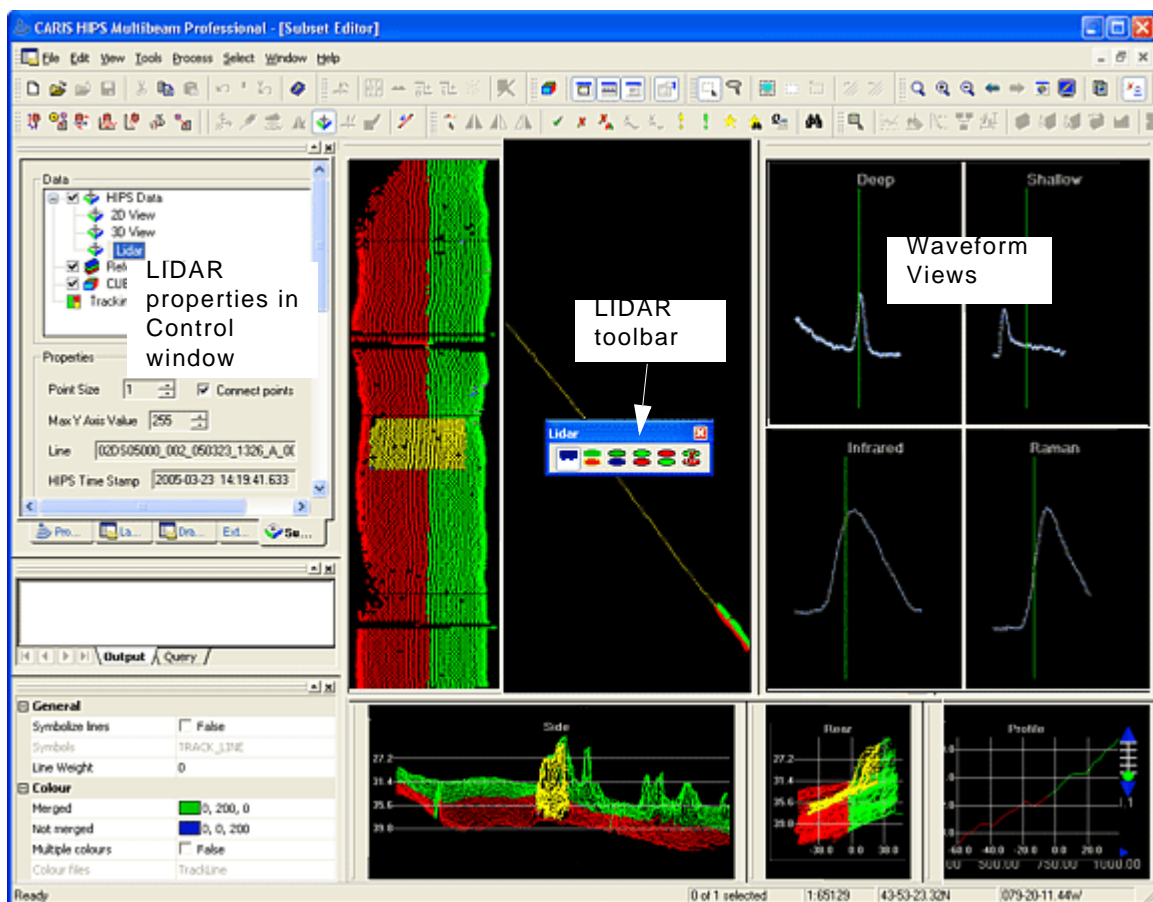
# LIDAR tab

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

- In Swath Editor, the LIDAR properties are displayed on a tabbed page in the Swath Editor tab

- In Subset Editor, these are displayed in the LIDAR layer in the Subset Editor tab.

The LIDAR tab in Swath Editor is illustrated below.
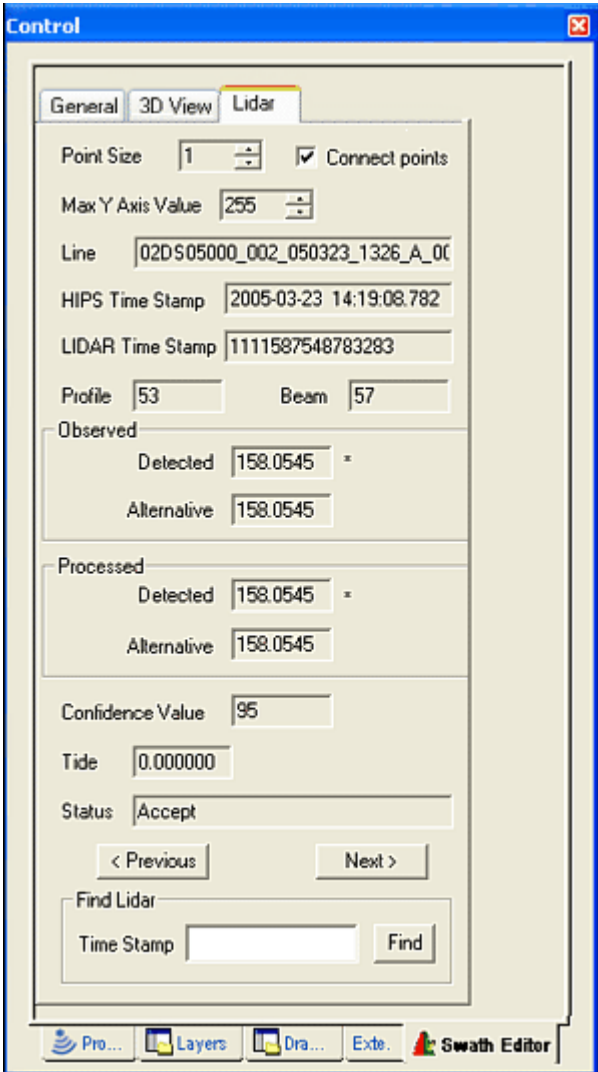


| | Menu | Edit > LIDAR > View Waveforms |
|---|---|---|
| | Tool |  |

To display the LIDAR controls in Swath Editor:

1. Select a line and open Swath Editor.

2. Select the View Waveforms command.

The LIDAR tabbed page appears on the Swath Editor tab in the Control 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.
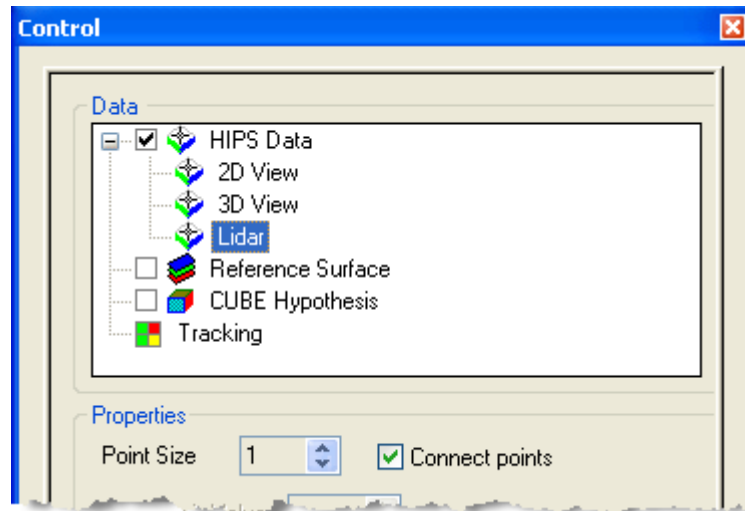


3. Select soundings in one of the Swath Editor view windows.

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 40.)

> 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.
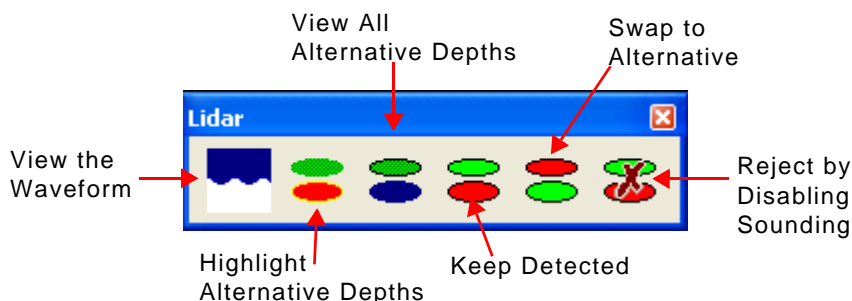


This is a floating toolbar that can be moved or docked anywhere in the HIPS main window. The tools on the toolbar become activated when Swath Editor is opened or data loaded to Subset Editor.

If the LIDAR toolbar is not visible in the interface,

1. Select the LIDAR toolbar command to display it.

| Menu | View > Toolbars > LIDAR |
|------|-------------------------|
| Pop-up | LIDAR |

## Waveforms View

The Waveforms view displays the waveform data for a super-selected sounding (see "SELECTING DATA" ON PAGE 38 for more information on superselection). The display in the Waveforms view is determined by the data format being viewed.

1. Select the View Waveforms command.

2. Select data in any of the Swath Editor or Subset Editor views.

| Menu | Edit > LIDAR > View Waveforms |
|------|-------------------------------|
| Tool | |

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.
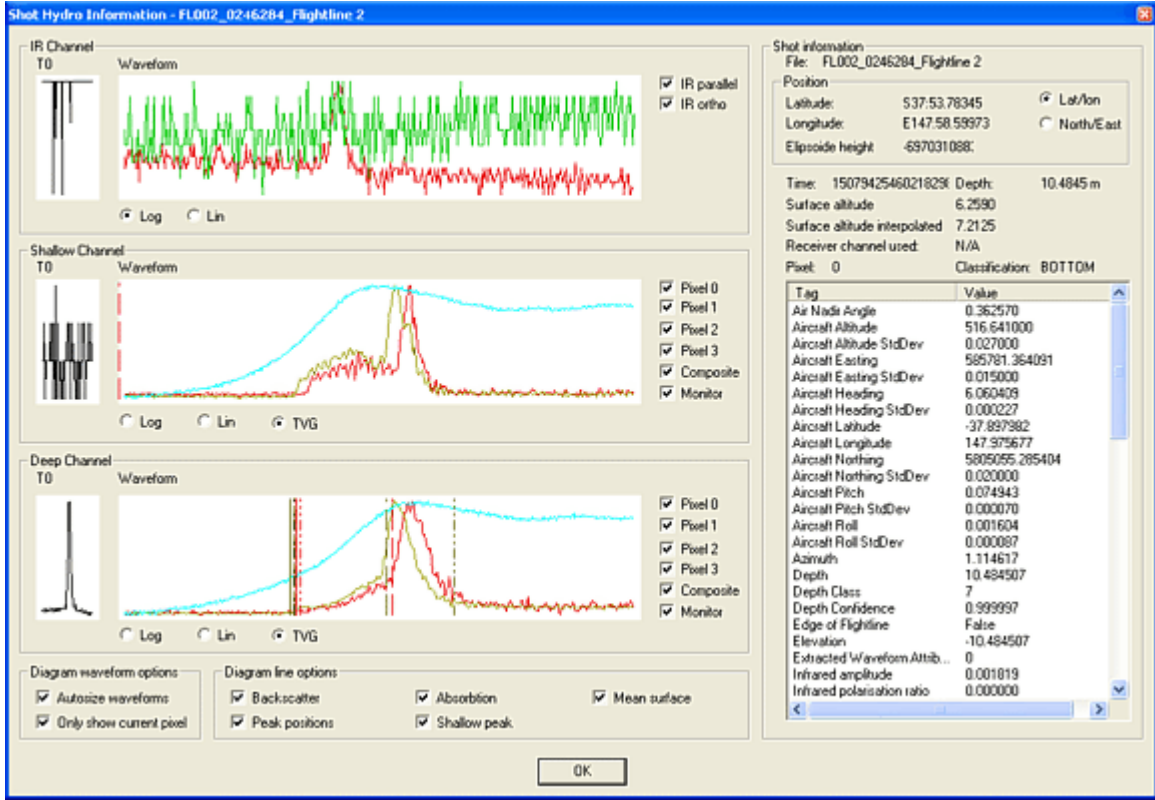


Note: Not all Hawkeye soundings contain waveforms that can be viewed. If a waveform cannot be found, an error message appears in the Output window.

# Selecting Data

Open data

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

1. Open either the Swath Editor or the Subset Editor.

If you are using Subset Editor,

2. Select an area and load the data.

| | |
|---|---|
| **Menu** | Edit > LIDAR > View Waveforms |
| **Tool** | |

3. Select the View Waveform command.

The Waveforms window (see "WAVEFORMS VIEW" ON PAGE 36) is displayed and the LIDAR tab is displayed in the 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.

4. Choose a selection tool from the Select menu. The *Edit By Range* option is selected by default.

5. 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.

6. 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 Worksheet window

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

Queried data can also be superselected.

1. Select data in any of the views.

2. Select the Query command.

The selected data is displayed in the Query tab of the Worksheet 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 40).

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

Highlight alternative soundings

| | |
|---|---|
| Menu | Edit > LIDAR > Highlight Alternative Depths |
| Tool |  |

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

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**

| Menu | Edit > LIDAR > View Alternative Depths |
|------|----------------------------------------|
| Tool |  |

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.

1. Select the View Alternative Depths command.

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

**Swap to alternative**

| Menu | Edit > LIDAR > Swap to Alternative |
|------|------------------------------------|
| Tool |  |

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.

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**

| Menu | Edit > LIDAR > Keep Detected |
|------|------------------------------|
| Tool |  |

This command is used to verify that the detected depth is preferred over the alternative depth (see "SOUNDING QUALITY AND STATUS FLAGS" ON PAGE 40).

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

| | |
|---|---|
| **Menu** | Edit > LIDAR > Reject by Disabling Sounding |
| **Tool** |  |

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

1. Select a sounding or group of soundings.

2. Select the Reject by Disabling Sounding command.

The soundings are now flagged as rejected.

| | |
|---|---|
| **Menu** | View > Show Rejected |
| **Tool** |  |
| **Key** | <F4> |

To see which soundings have been rejected:

1. Select the Show Rejected command so the disabled soundings are visible.

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> |

# 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

| | |
|---|---|
| **Menu** | Tools > Navigation Editor |
| **Tool** |  |

To open Navigation Editor:

1. Select a track line.

2. Select the Navigation Editor command.

The Navigation Editor components are displayed in the HIPS and SIPS interface.
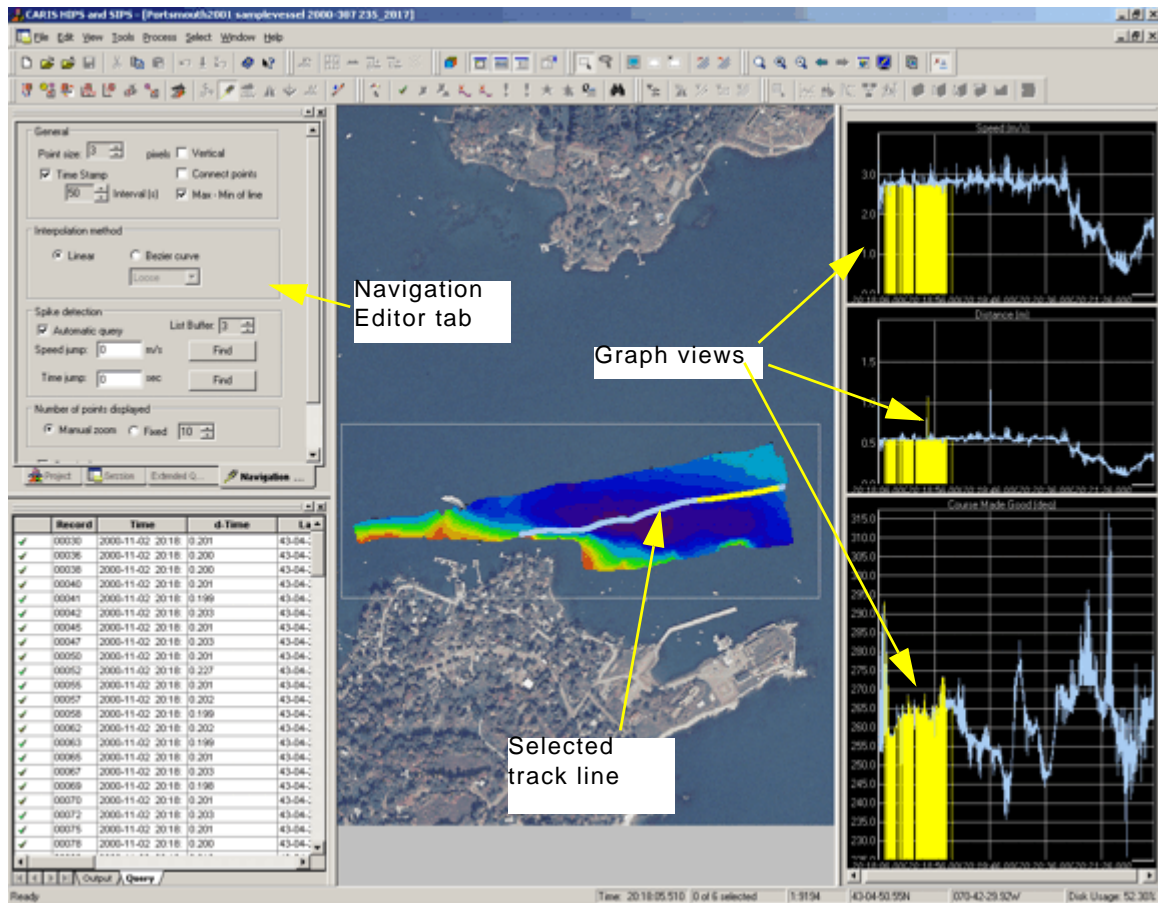


Navigation Editor is composed of three time-series graphs and the Navigation Editor tab in the Control window. The three graphs display information that is calculated between points in the recorded navigation data:

- **Speed:** Calculated speed.

- **Distance:** Calculated distance.

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

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.

# Navigation Editor tab

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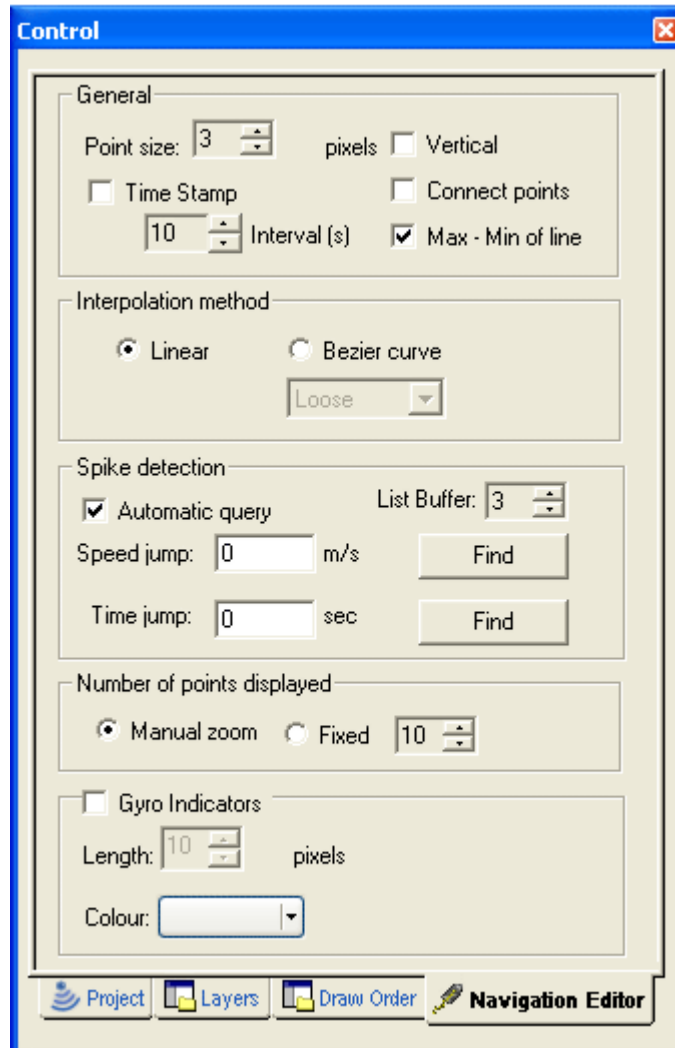


Resize data points

The point symbols can be resized to a maximum of 10 pixels.

1. Click the up or down arrow buttons beside the *Point Size* box to change the size of the symbols.

The view in the Display window is refreshed so the navigation data is shown in the new point size.

Time Stamps

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

1. Select the *Time Stamp* check box to display time stamp lines.

2. Click the up or down arrow buttons beside the *Interval* to select the gap (in seconds) between time stamps.

Set minimum/maximum of line

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.

## Number of points displayed

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.)

Use this option to determine the number of points visible in the Display window.

1.  Open the Control window.

2.  Select the *Navigation* tab.

3.  Select either the *Manual* or *Fixed* check boxes (Manual is selected by default).

If you selected the *Fixed* check box, enter the number of navigation points to be displayed by clicking the up or down arrow buttons.

This option determines the number of points visible in the Display window.

·   When *Manual* is selected, use the zoom tools to determine how many navigation points are visible in the Display window.

·   When *Fixed* is selected, a set number of points are displayed in the window.

1.  Select either the *Manual* or *Fixed* check boxes (Manual is selected by default).

2.  If you selected the *Fixed* check box, enter the number of navigation points to be displayed by clicking the up or down arrow buttons.

The amount of data displayed in the three graph views is determined by the number of navigation points visible in the Display window.

Connect points

Join the displayed navigation point symbols with a line representing the current interpolation method.

1.  Open the Control window.

2.  Select the *Navigation* tab.

3.  Select the *Connect Points* check box.

The navigation points are connected by line drawn according to the current interpolation method of the line.

Gyro indicator line    A gyro indicator line that shows vessel heading can also be displayed in the Navigation Editor. This feature is useful for detecting dramatic changes in heading that could effect data acquisition.

1. Select the *Gyro Indicators* check box to display the gyro line in the Display window.

2. Click the up or down arrow buttons to select the size (in pixels) of the gyro indicator line.

3. Select a colour for the line from the *Colour* picker, or create a custom colour using the standard Windows Colour Palette.

# 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

In Side Scan Editor you can

- use a waterfall display for editing
- apply advanced Time Varying Gain controls
- edit altitude data for side scan data
- digitize and edit contacts.

The following diagram outlines the process for processing side scan data in SIPS.



Workflow in Side Scan Editor
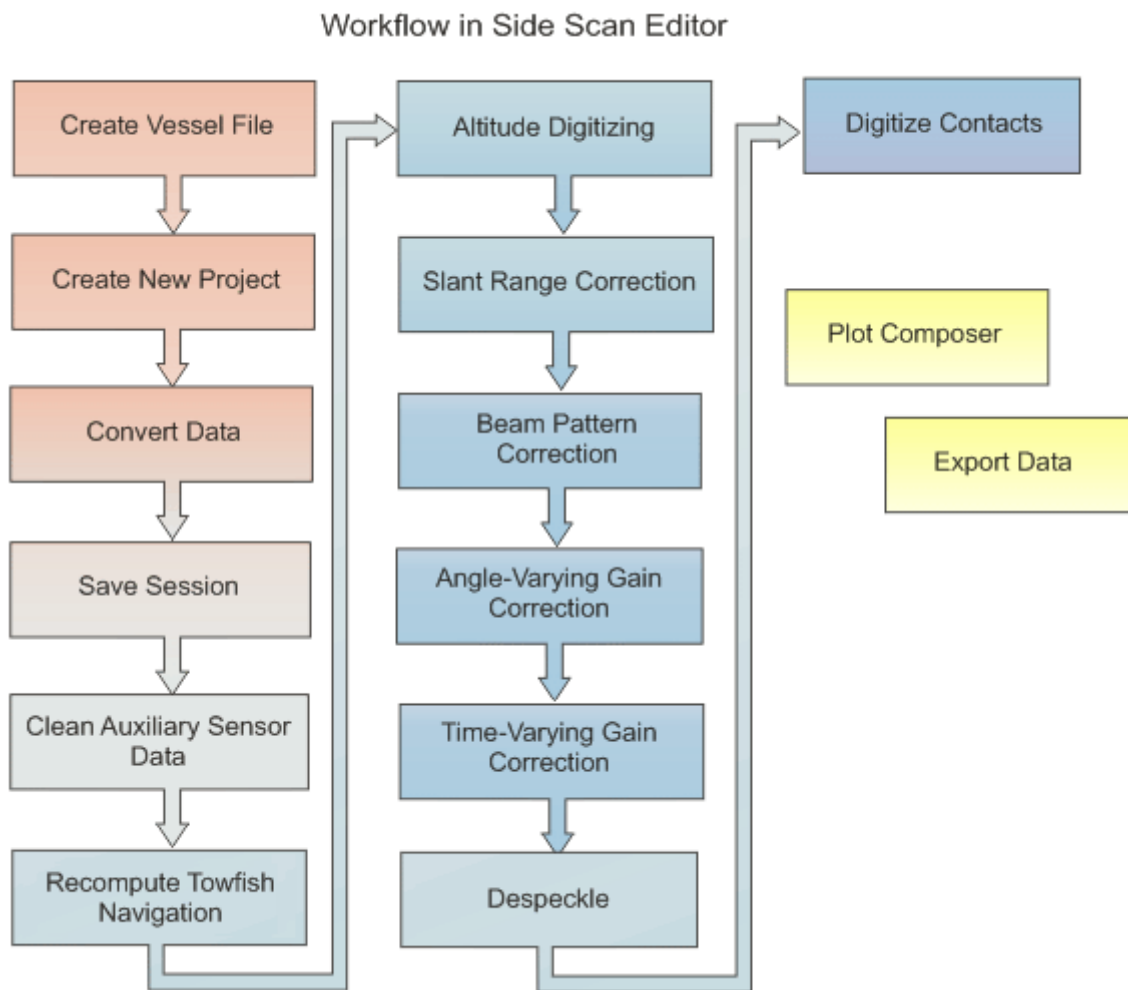
Altitude digitized side scan imagery can also be processed in Mosaic Editor. Mosaic Editor is also used to create mosaics from side scan data. See PROCESS IMAGERY DATA in the Users Guide.

Data in Side Scan Editor can be viewed in either raw or processed mode. Processing can be applied to a single ping, to a range of pings, or to one (or more) track lines.

1. If desired, the raw data may be edited by selecting, querying, rejecting or accepting pings. Rejecting pings removes them from consideration for any mosaics to be created from this data. (See "EDIT SIDE SCAN DATA" ON PAGE 71)

2. Once pings are edited, you may need to digitize the "first return" or altitude if the sonar did not detect and record this data during acquisition. The altitude is required to perform slant range correction. If digitized altitude data does exist, it may need to be edited first. (See "DIGITIZING SONAR ALTITUDE" ON PAGE 74)

3. With fish altitude and an average sound velocity, the raw data can be slant range corrected. Slant range correct data is shown in processed mode, where the across-track measurement is horizontal distance. (See "APPLY SLANT RANGE CORRECTION" ON PAGE 81)

    In this processed mode, you can edit and query pings, make height and distance measurements, and mark and edit contacts.

4. Use the correction tools to remove unwanted artifacts:

    • "BEAM PATTERN CORRECTION" ON PAGE 83)

    • "ANGLE-VARYING GAIN" ON PAGE 88

    • "TIME-VARYING GAIN" ON PAGE 89

    • "DESPECKLE" ON PAGE 92)

    The parameters for applying these correction tools can also be saved in a template file and loaded and applied again. (See "SIPS TEMPLATE WIZARD" ON PAGE 98).

5. Create and edit contacts

    See "CONTACTS" ON PAGE 93.

Side scan data can also be processed by creating GeoBaRs and mosaics in Mosaic Editor. (See "PROCESS IMAGERY DATA" ON PAGE 387).

# Side Scan Editor Interface

| | |
|---|---|
| **Menu** | Tools > Side Scan Editor > Open |
| **Tool** |  |
| **Pop-up** | Tools > Side Scan Editor |

The Side Scan Editor consists of a data display window, toolbars and the Side Scan Editor tabs in the Control window.

To open the Side Scan Editor:

1. Select a towfish track line in the HIPS and SIPS Display window.

2. Select a Side Scan Editor command.

When the Side Scan Editor is opened, the data for the selected track line is displayed in the Side Scan Editor preview window, which has a default location along the right side of the interface.

As well, a histogram of the sonar file is displayed in the Worksheet window, and the Side Scan Editor toolbar and the Beam Correction toolbar are activated.
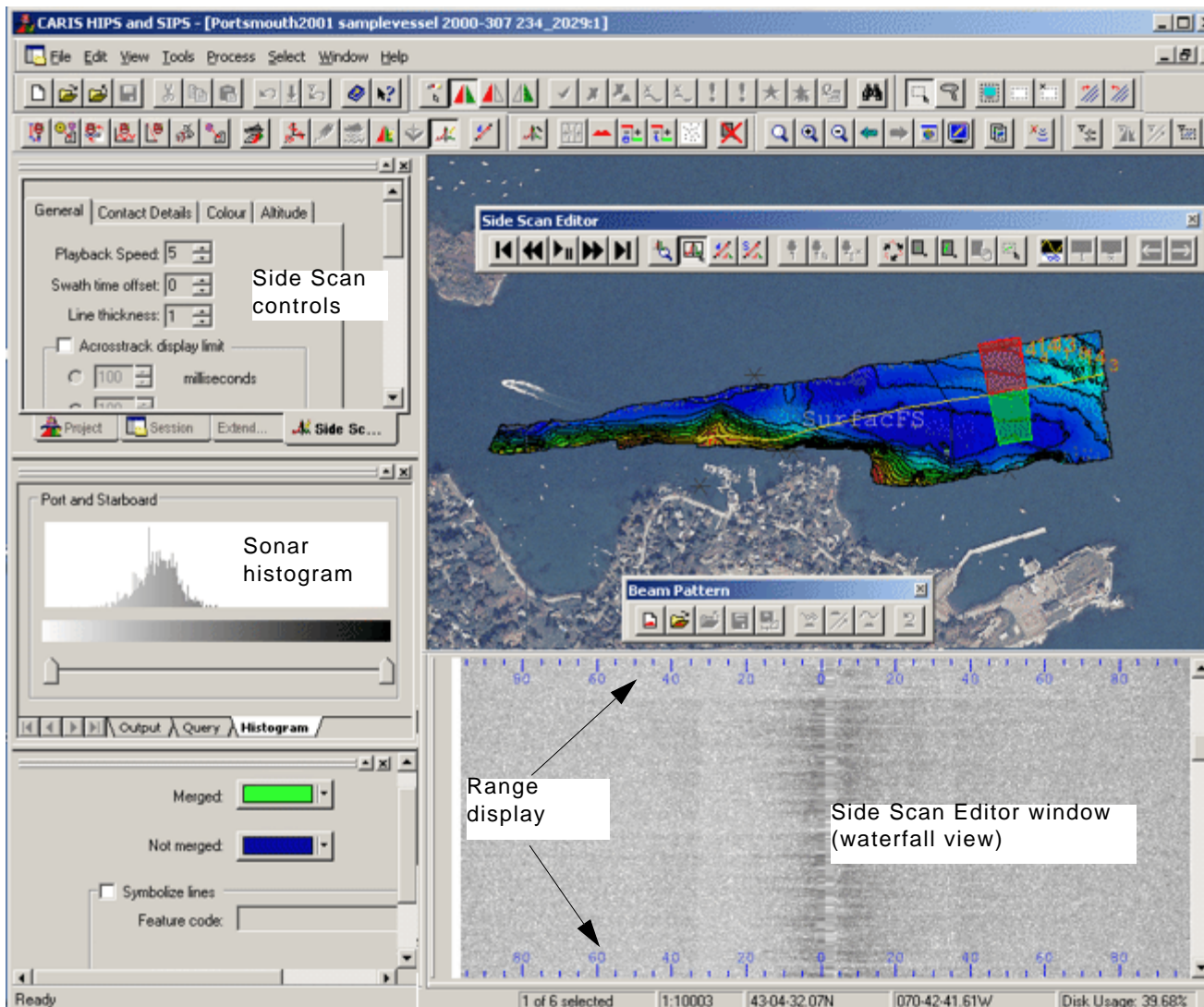
## Side Scan data display

The Side Scan Editor data display window (also called the "waterfall" view) can be undocked and moved to another position, for example, at the bottom of the interface.

There are two modes to display data in the SIPS Preview window:

| | |
|---|---|
| **Menu** | Tools > Side Scan Editor > Raw/Processed |
| **Pop-up** | Raw/Processed |

- **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 slant range correction process. The data displayed in this mode will reflect the last processing done to it.

1. 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.

> Note: If the data does not fill the entire height of the Editor window, 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 SIPS Preview window. The size of the coverage outline varies with the amount of side scan visible in the SIPS Preview window. The starboard and port areas of the coverage outline are appropriately colour coded, as illustrated below.
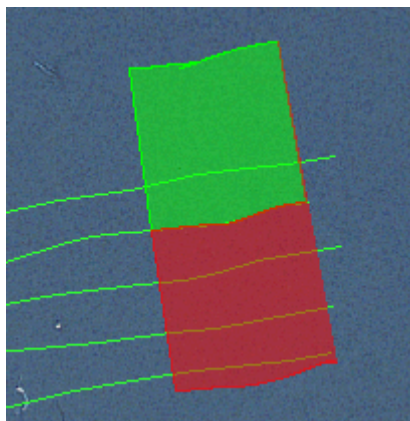
Colours for outlines and selection are controlled from the Tools >Options >Display dialog box .

Compression

The number of pixels used to display data in the Side Scan window is determined by the width of the display window and screen resolution. However, because the volume of data is often too great for the window to display all pixels, the imagery must be compressed.

There are three compression methods to choose from.

| Menu | Tools > Side Scan Editor > Compression >Skip/ Average/Maximum |

1. Select a Compression command.

- **Skip** selects pixels for display at regular intervals across the track.

- **Average** compiles average pixel intensity within a group of pixels spaced at regular across-track intervals.

- **Maximum** selects the brightest pixels from groups of pixels at regular across-track intervals.

The display in the Side Scan Editor is refreshed to show the data according to the selected compression option. The default display uses the Skip option.

# Side Scan Editor controls

The General tab of the Side Scan Editor contains controls for navigating and displaying data in the Editor window. This tab is visible in the Control Window when the Side Scan Editor is opened.

Across-track display

The width of the ping shown in the SIPS Preview window can be increased or decreased by time or distance, using the controls in the *Acrosstrack Display Limit* fields on the General tab.
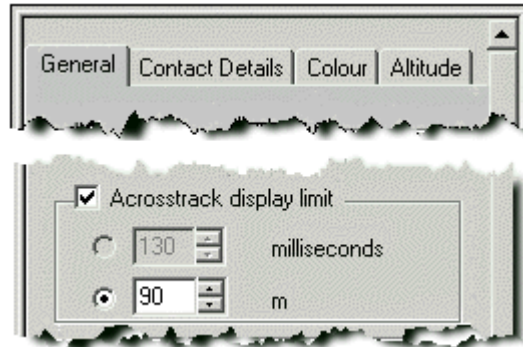


To set across-track display limits:

1. Make sure the Control window is open.

2. Select the General tab from the Side Scan Editor tab.

3. To have the entire width of the ping is shown in the Editor, leave the *Acrosstrack display limit* box unchecked.

4. To limit the width of ping displayed, check the *Acrosstrack display limit* box and select which value to use as limiter:

   • *Milliseconds*: Click the arrow button to change the time value. This option is automatically selected.

   • *Metres/Feet*: Click the arrow buttons to change the distance value.

As time or distance is incremented or decreased, the amount of the data is displayed in the window is changed.

The units of distance units can be changed using the *Display* tab in *Options* on the *Tools* menu.

Enable Range display

The range display runs across the top and bottom of the SIPS Preview window. In Raw mode the range displays time intervals in milliseconds. In Processed mode, distance is displayed in the units you have set in the Tools >Options >Display dialog box.

1. Select the Enable Range Display check box in the Side Scan Editor General tab.
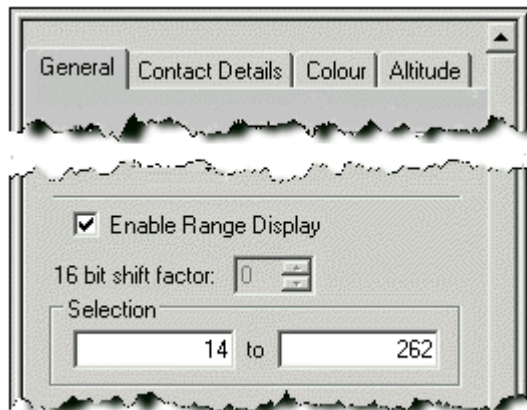
The colour of the range display is set to the same colour used for measure/offset tools in the Side Scan Display Options of the Tools menu.
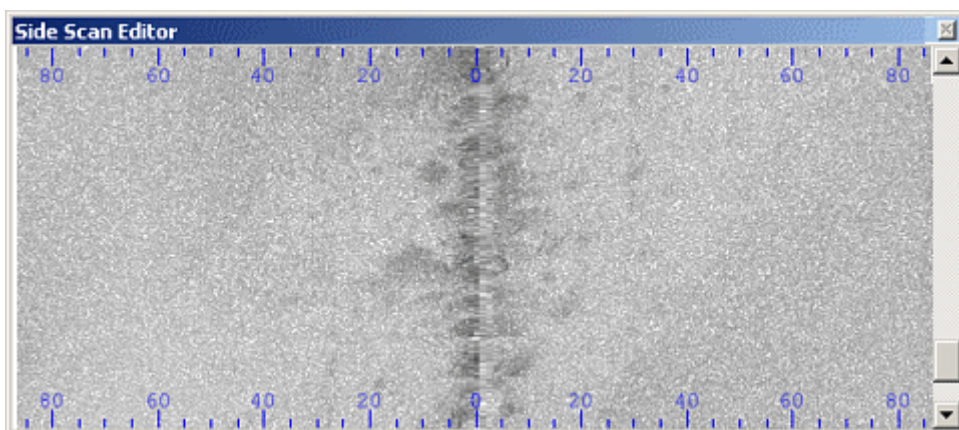
| 16-Bit Shift Factor | An 8-bit colour display in a 16-bit processed data set can be enhanced by bit-shifting to select which 8 bits are used to generate the display. This can create a more even distribution of pixels. |

1. Make sure that 16-bit processed data is displayed in the Side Scan Editor.

2. Click the up or down arrow keys beside the 16 Bit Shift Factor field to select a value from 0 to 8.

The display is refreshed.

Ping selection

The two Selection fields display the start and end ping numbers when you select part of the trackline displayed in the Editor window. You can use these fields to select specific start and end pings.

# Colour Mapping

Colour maps provide a clear visual representation of sonar intensity levels along the track line. Colour mapping is applied to the histograms displayed in the Worksheet window, as well as the port and starboard sections of ping in the SIPS Preview window.

You can select different colour maps for port and starboard sections, or select a single colour map for both sections. All colour maps are stored with a .cma extension in the directory …\Hips\System.

Change from GreyScale     To change the colour mapping from the default setting of GreyScale:

1.  Open the Control window and select the Colour tab from the Side Scan Editor tab.

2.  Select either *Port*, *Starboard* or *Both* to apply new colour to.

3.  Select a colour map from the drop-down menu (the default colour map is GreyScale.cma).

When the colour map is chosen, both the histogram and the ping window are refreshed, so that the port and starboard sections (or both) display the selected colour mapping.
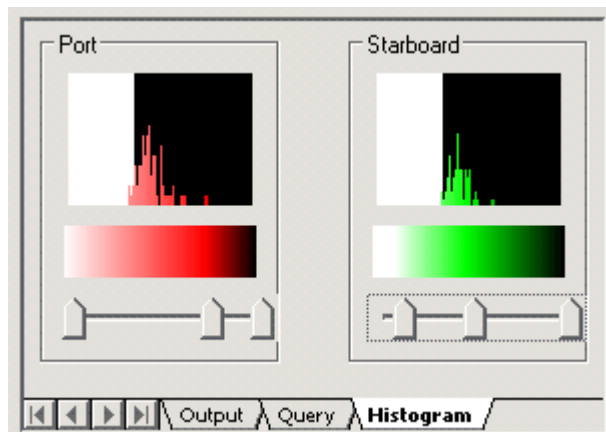


In the Histogram, the horizontal axis represents the intensity range of the pixels in the sonar file and the vertical axis represents the distribution of pixels in the editor.

Use the slider bar below the histogram in the Worksheet window to adjust colours. Each colour in the histogram has a slider control associated with it.

1.  Move the slider to adjust the pixel value of that colour in the histogram and in the ping.

The value shown in the Control window adjusts as you move the sliders.

Colour display can be adjusted also using the other options:

- **Blend** will blend the colour borders

- **Delete Colour** will remove it from the list, and from the areas that it colours

- **Clear** will remove all colours. To view ping clearly, restore GreyScale colour map.

Custom colour maps    You can create custom colour maps using either the Colour tab or the sliders below the histogram in the Worksheet window.

1. Make sure both the Worksheet and Control windows are open.

2. Click **Add Colour**, or click on a slider to open the colour mapping controls.
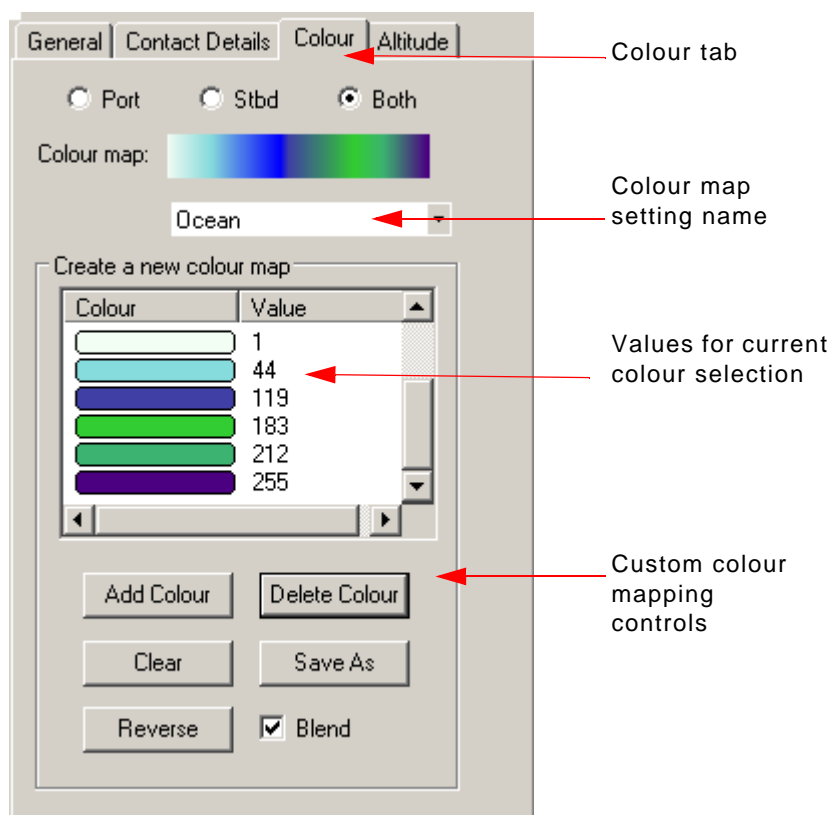


The *Value* field shows the pixel value of the displayed colour.

The pixel values range from 1 to 255 with the first colour always assigned to pixel value 1. Pixel value 0 is always assumed to be transparent.

3. Enter a pixel value for the colour, as needed.

4. Click on the button to open a colour palette

5. Select a colour from the colour palette or click **Create Custom Colour** to create a custom colour from the standard Windows colour picker.

6. Click **OK**.

The new colour and its assigned pixel value is added to the listing in the Control window, and applied to the histogram in the Worksheet window. The colour of the ping in the Side Scan Editor also changes to display the new colour in the appropriate areas.

7. Repeat Steps 2 to 6 to insert more colours, as needed.

As you select more colours, the colour values and the depths are displayed in the Control window.

Reverse    Sonar recorders have traditionally used dark colours to represent high signal returns. You may want to use the Reverse option to change the order in the colour map to reflect this practice.

To reverse the colour map order, click **Reverse**.

8. Click **Save As** to retain the settings as a colour map.

The Save As dialog box is displayed.

9. Type a file name for the colour map.

## Toolbar

The Side Scan Editor toolbar is displayed in the interface when the editor is started.

The toolbar can docked or undocked, or removed from the interface like other toolbars. Below is an example of the toolbar in the undocked position.
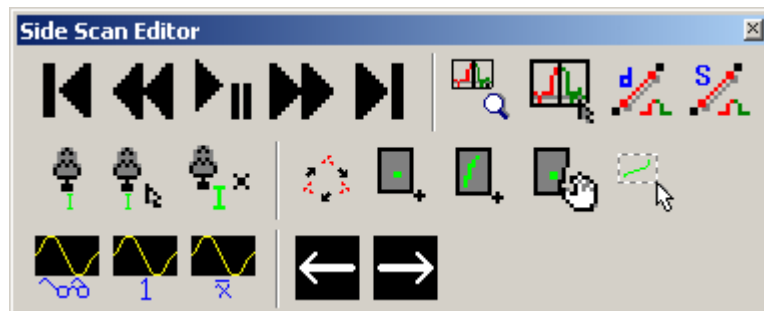


The toolbar contains controls to navigate through the track line and perform other functions.

| Toolbar Buttons | Purpose |
| --- | --- |
|  | Playback (see "NAVIGATE IN SIDE SCAN EDITOR" ON PAGE 60) |

| Toolbar Buttons | Purpose |
|---|---|
| | Zoom (see "ZOOM" ON PAGE 62) |
| | Select (see "SELECTION" ON PAGE 62) |
| | Measure distance and shadow (see "MEASURE DISTANCE" ON PAGE 64 and "MEASURE SHADOW" ON PAGE 65) |
| | Digitize (see "DIGITIZING SONAR ALTITUDE" ON PAGE 74) |
| | Contacts (see "CREATE CONTACTS" ON PAGE 93) |
| | View waveform (see "SIGNAL DISPLAY WINDOW" ON PAGE 68) |
| | Revert and reapply (see "REVERT/REAPPLY" ON PAGE 79 |

## Navigate in Side Scan Editor

You can navigate up or down the track line with:

- the scroll bar on the SIPS Preview window
- the mouse wheel
- the keyboard
- the Playback controls.

Scroll bar
- Clicking the scroll arrow buttons moves the view in the editor forward or backward by a single ping.
- Clicking the scroll bar, or dragging the scroll box forward or backward increments your view by a set 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

| Menu | Tools > Side Scan Editor > Playback Controls |
|------|---------------------------------------------|

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



Go to Start of trackline → | Page Reverse / Page Forward / Play Reverse / Stop / Play Forward | ← Go to End of trackline
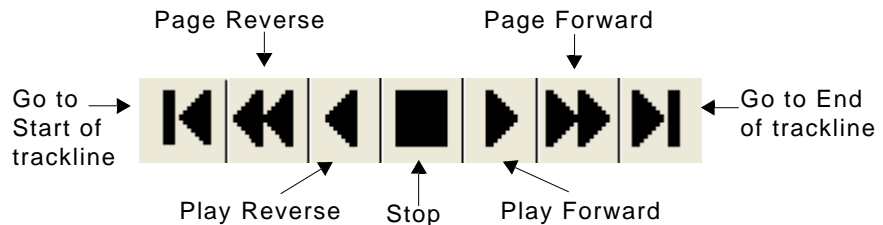
The view in the Side Scan Editor will scroll at the speed set on the Side Scan tab of the Control window

The ping coverage outline in the Display window moves with the display in the Editor.

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

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

The Side Scan Editor moves up the track line, 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 trackline in larger increments.

Playback speed

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

1. Make sure the Control window is open and Side Scan Editor tab is displayed.

2. Select speed value from the *Playback Speed* field in the General tab..

View contacts while scrolling

If you want your 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

| | |
|---|---|
| **Menu** | Tools > Side Scan Editor > Zoom |
| **Tool** | |

You can magnify an area in the Side Scan window with the Side Scan Editor Zoom tool.

1. Select the Side Scan Zoom tool.

2. Press and hold the mouse button and drag the cursor over the area you want to enlarge.

A new window displaying the enlarged area will open.

You can display the image so that the entire zoomed area fits the zoom window or you can scale the image according to a selected percentage.

3. Right-click the mouse in the zoom display window and choose any of the following options from the pop-up menu:

   • Select the *Fit to Window* option so that the entire area of the zoomed image is displayed in the window.

   • Select the *Aspect Ratio* to keep the same height and width ratio for the zoomed area. When this option is cleared, the image is resized so that it fills the entire zoom window.

   • Select the *Scale* option to resize the zoomed area to a scale. The scale is selected from the drop-down list. When a scale is selected, the image is displayed according to the selected percentage.

   • Select *Close* to close the window.

4. You can use the Measure Distance and Measure Shadow tools in the Zoom window (see "MEASURE DISTANCE" ON PAGE 64 and "MEASURE SHADOW" ON PAGE 65). When you release the mouse button, the results are shown in the Contacts tab in the Control Window.

5. Deactivate the Zoom tool when finished.

## Selection

| | |
|---|---|
| **Menu** | Tools > Side Scan Editor > Select |
| **Tool** | |

Data can be selected in the Side Scan Editor using the Side Scan selection tool. The selection tool can be used to select a range or a single ping within that range. Selected data can be viewed, edited, or corrected using a variety of tools.

To range-select data:

1. Choose the Select Side Scan button on the Side Scan Editor toolbar.

The cursor will become a cross-hair when positioned in the SIPS Preview window.

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

When you release the mouse button, the selected range is shown bounded with a rectangular yellow box:

The range remains selected until you click in the Editor window again.

To select a single ping:

3.  Select a range of side scan (see above).

4.  While the range is selected, press the <Up Arrow> key, which produces a third horizontal line representing the selected ping.
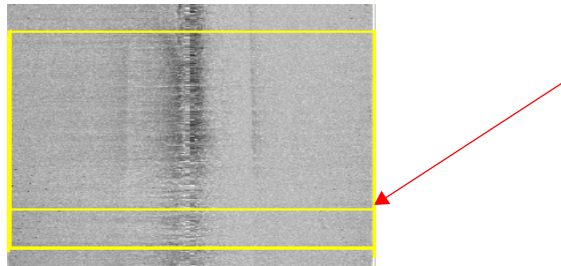


5.  Continue pressing the <Up Arrow> key to move the selection up, or press the <Down Arrow> key to move the selection down.

Select by ping number

You can also use the Selection fields on the General tab of the Side Scan Editor Control window to select a range in the Editor window.

When you make a selection in the Editor window the start and end ping numbers of that selection are displayed in the General tab. To refine your selection, enter the specific ping numbers fields to specific start and end pings.
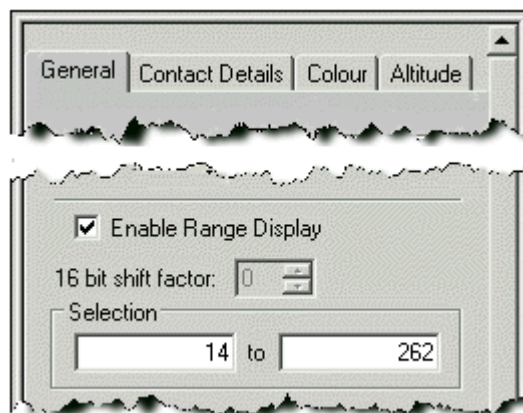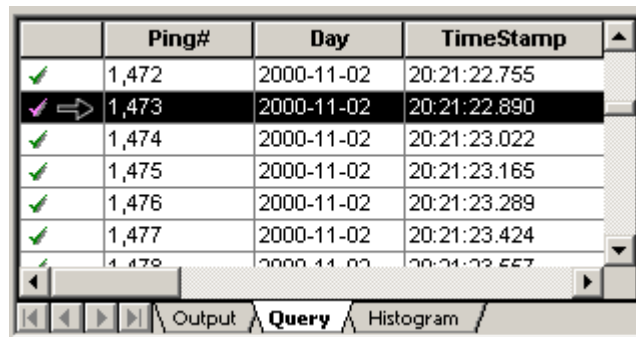
| | | |
|---|---|---|
| **Menu** | Edit > Query | |
| **Tool** |  | |
| **Pop-up** | Query | |
| **Key** | <Q> | |

**Superselection**

Superselection is selecting data within an already selected range. You can superselect pings from the Worksheet window and display the superselected data in the Side Scan Editor.

1. Select a range of side scan data.

2. Select the Query command.

A table listing the pings in the selection is displayed in the Query tab of the Worksheet window.

| | Ping# | Day | TimeStamp | |
|---|---|---|---|---|
| ✓ | 1,472 | 2000-11-02 | 20:21:22.755 | |
| ✓ ⇒ | 1,473 | 2000-11-02 | 20:21:22.890 | |
| ✓ | 1,474 | 2000-11-02 | 20:21:23.022 | |
| ✓ | 1,475 | 2000-11-02 | 20:21:23.165 | |
| ✓ | 1,476 | 2000-11-02 | 20:21:23.289 | |
| ✓ | 1,477 | 2000-11-02 | 20:21:23.424 | |
| ✓ | 1,478 | 2000-11-02 | 20:21:23.557 | |

Output \ **Query** \ Histogram

3. To superselect a single ping, select the row in the Selection tab.

The superselected ping will be highlighted within the selection in the Side Scan Editor.

3. To superselect a range of pings, click in the first column on the first row of the range and drag the mouse through to the last row of the range you want to select.

A rectangle is displayed containing the superselected range of pings within the selection.



Range superselection within a regular selection
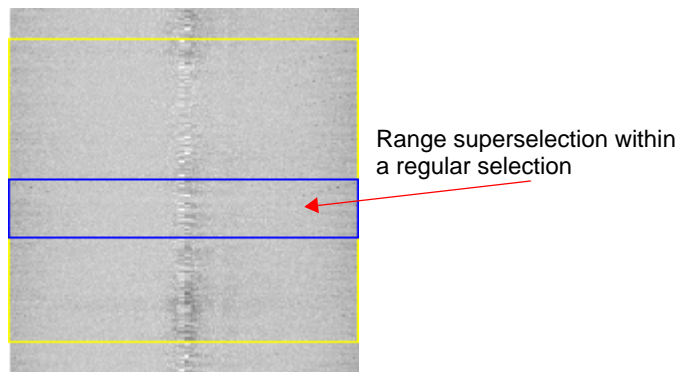
## Measure Distance

**Measure distance**

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

| Menu | Tools > Side Scan Editor > 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, and drag the cursor across the Side Scan Editor.

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

## Measure Shadow

| Menu | Tools > Side Scan Editor > Measure Shadow |
|------|-------------------------------------------|
| Tool |  |

This tool enables you to determine the height of an object by measuring its shadow. This feature is only available in processed mode, and can be measured in either the Side Scan Editor or in the Side Scan Zoom window.

1.  Make sure the side scan display is in processed mode.

2.  Select the Measure Shadow tool.

3.  Press and hold the mouse button, and drag the cursor *horizontally* across the Side Scan Editor.

The height measurement is shown in the side scan window.

# Side Scan Editor and Swath Editor

The Swath Editor can also be open at the same time as the Side Scan Editor.

Side Scan Editor                    Coverage outline for multibeam data
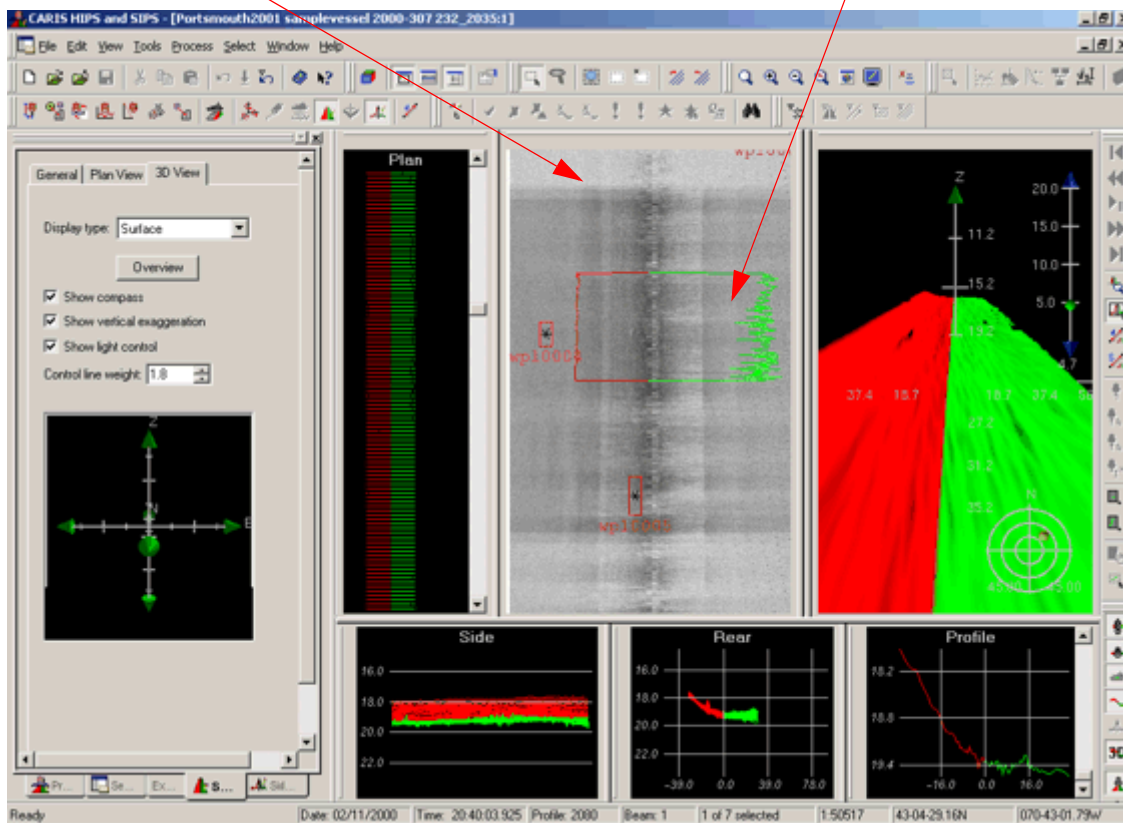


When the Side Scan is open in this mode, the playback functions used to scroll the track line, are unavailable. You can, however, move up and down the track line by scrolling the Plan View in the Swath Editor.

All other Side Scan functions and tools are available.

When both the Side Scan and Swath Editors are open, a coverage outline of the multibeam data is displayed in the Side Scan Editor. This feature is useful when the across-track coverage of the multibeam is less than that of the side scan.

Swath Time Offset
If the multibeam data was recorded from a vessel and the side scan from a tow fish, then the coverage outline will not be in the correct along-track location.

The *Swath Time Offset* adjusts the along-track locations of the swath outline in the Side Scan Editor by applying a time correction value. The time correction value can be either positive or negative. Negative values are used for track lines where the side scan sonar is positioned before the multibeam echo sounder.

To adjust the swath time offset:

1. Make sure the Side Scan Editor is open. (The Side Scan Editor tab in the Control window will be visible.)

2. Click the <Up> or <Down> arrows for the Swath Time Offset field on the Side Scan editor tab until the correct delay (in seconds) between the vessel and towfish is displayed.



Swath Time Offset field
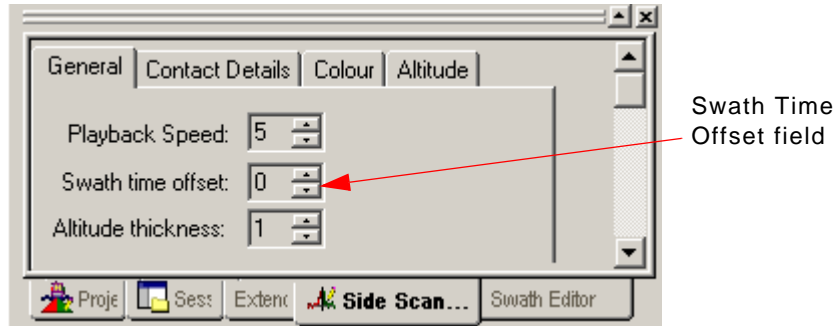
# Signal Display Window

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 pre-and post-processing state of any correction can be compared using the Revert and Reapply functions

The Signal Display Window can display a mean of all sonar intensities in a selection or the sonar intensity values for a single ping. The display can be switched back and forth between mean and single profile modes by use of the buttons on the toolbar.

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

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.

## Open Signal Display Window

| | |
|---|---|
| **Menu** | Tools > Side Scan Editor > Signal Display Window > Open/Close |
| **Tool** | |

1. Select a range in the Side Scan window using the Side Scan Selection tool (see "SELECTION" ON PAGE 62).

2. Select the Signal Display Window tool.

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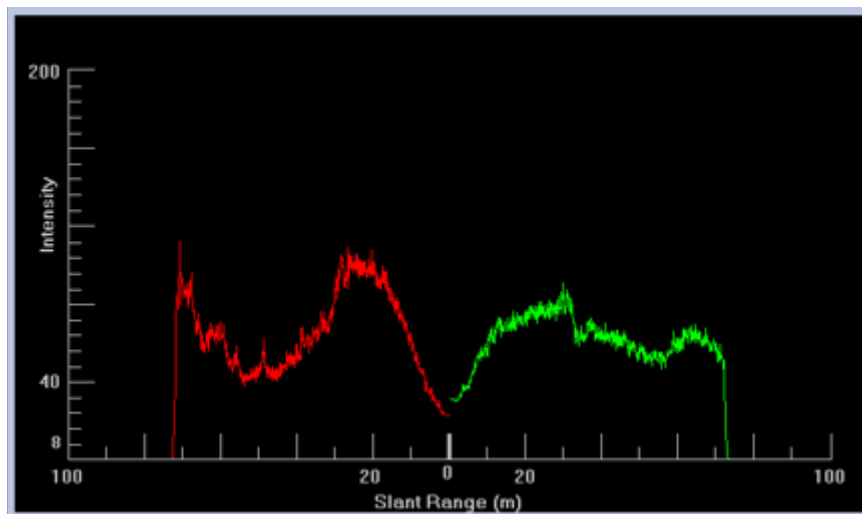
The vertical axis on the graph represents intensity level of the signal(s) being examined. The horizontal axis represents either time or slant-range distance depending on the display mode of the Side Scan Editor.

| | |
|---|---|
| **Menu** | Tools > Side Scan Editor > Signal Display Window > View Selection Mean |
| **Tool** |  |

## View Selection Mean

The mean of the intensity values in the selected range can be displayed.

1.  Select the View Selection Mean tool.

## View Single Ping

You can also view signal intensity for a single ping.

1.  Select the View Single Ping tool.

| | |
|---|---|
| **Menu** | Tools > Side Scan Editor > Signal Display Window > View Single Ping |
| **Tool** |  |

The Signal Display Window is refreshed to show the sonar signal intensities for the single ping selected in the Side Scan window.
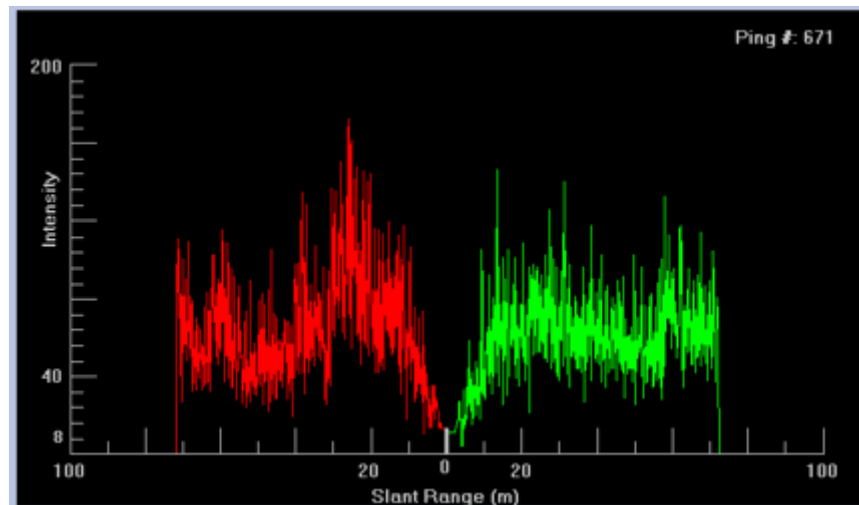


Pressing the Up Arrow key while an area of the side scan data is range selected will select a single ping

2.  Use the Up and Down arrow key to view the sonar intensities for other single pings.

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.
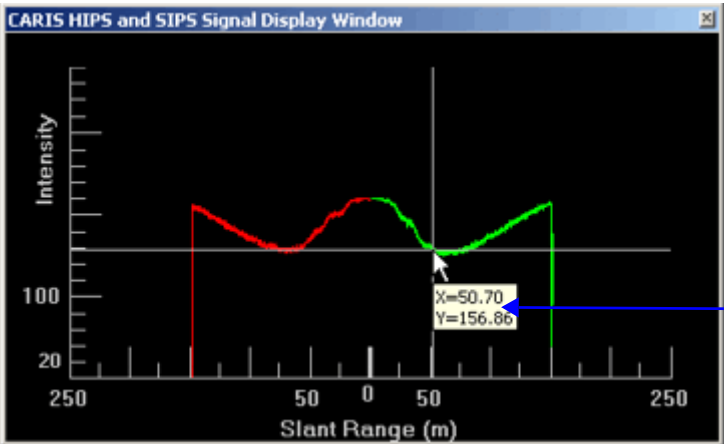
Tool tip

# Edit Side Scan Data

In Side Scan Editor you can reject or accept lines as you do in other HIPS and SIPS editors, except that you select a ping rather than an individual sounding or beam. (This line editing does not need to be done within the Side Scan Editor, it can be done in the HIPS Display window.)

If navigation sensor data has been edited with in Navigation Editor it may be necessary to recompute towfish navigation. (See .)

If you intend to apply a slant-range correction, the altitude of the sonar must be known. If this is not already logged with your data, the altitude must be digitized. (See .)

## Accept/Reject/Query

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

1. From the Side Scan toolbar, use the Side Scan Selection tool

   to select the range of pings from the SIPS Preview window ("waterfall view") to be accepted or rejected (see ).

2. [Optional] From the HIPS Edit Toolbar, select Edit Port Side Only or Edit Starboard Side Only.

3. Select the desired status from the menu or toolbar: Accept or Reject.

When a ping is rejected in the Side Scan Editor, it is given a status flag that excludes it from further processing in SIPS.

| | |
|---|---|
| **Menu** | Edit > Status Flag > Reject |
| **Tool** | ✗ |
| **Pop-up** | Reject |

4. Select the Reject command.

The status of the ping or range of pings is set to Rejected.

However, Rejected data is not deleted and can be changed to Accepted status again with the Accept command.

Accept

To make rejected data available for further processing in SIPS:

| | |
|---|---|
| **Menu** | Edit > Status Flag > Accept |
| **Tool** | ✓ |
| **Pop-up** | Accept |

1. From the Side Scan toolbar, use the Side Scan Selection tool to select the rejected pings (see ).

2. Select the Accept command.

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

Query

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

Use the Query function to view information about selected pings.

1. From the Side Scan toolbar, choose the Side Scan Selection tool and select a range of pings to be queried (see "SELECTION" ON PAGE 62).

2. Select the Query command.

Information about the selected data is shown in the Selection tab of the Worksheet window.

# 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 next ship's 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 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.

If you want SIPS to compute the towfish navigation data:

| **Menu** | Process > Recompute Towfish Navigation |
|---|---|

1. Select one or more towfish track lines.

2. Select the Recompute Towfish Navigation command.

The Recompute Towfish Navigation dialog box is displayed.
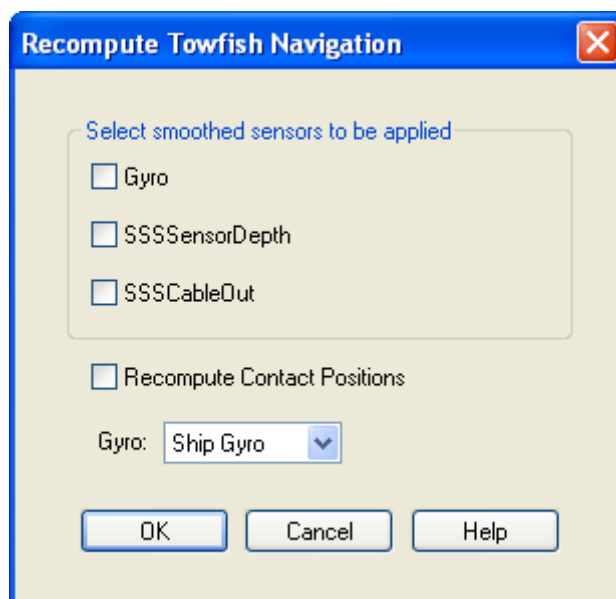
Towfish sensor data can be smoothed in the Attitude Editor (see "ATTITUDE EDITOR" ON PAGE 572) and this data can be applied when the towfish navigation is computed.

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 97).

## Digitizing Sonar 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. The 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.

Digitized seabed

Offset from nadir
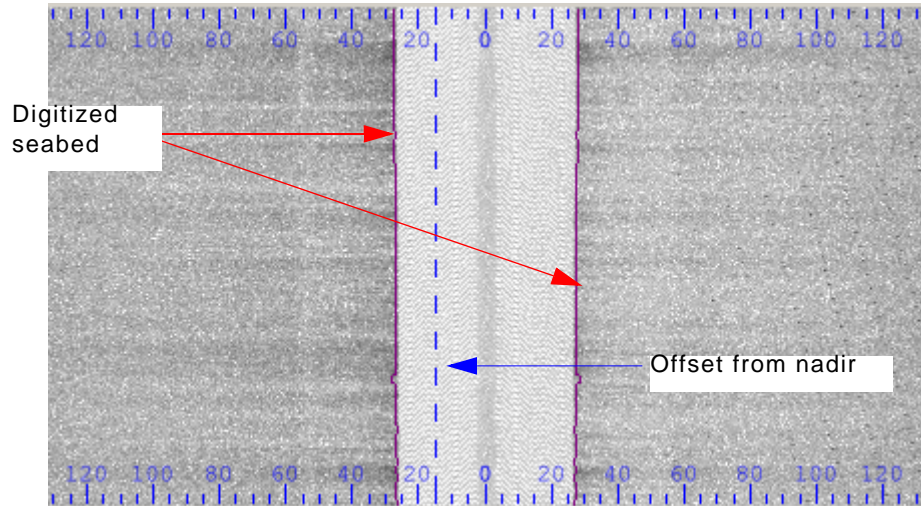
**Digitization and colour maps**

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. Also see "COLOUR MAPPING" ON PAGE 57.

1. Make sure that only two colours are displayed in the editor (for example, 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.

4. To remove the digitized lines, select the Clear command.

| Menu | Tools > Side Scan Editor > Altitude > Clear |
|------|---------------------------------------------|
| Tool |  |

## 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.

Settings for automatically digitizing the side scan return are in the Altitude tab of the Control window.
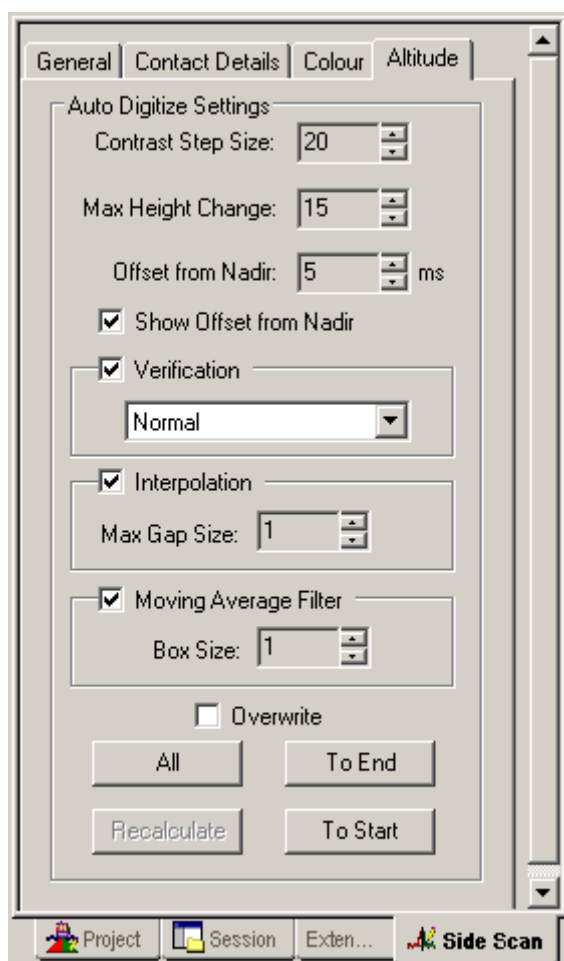
The Altitude tab options are described below in the order that they appear in the tab. While the *Contrast Step Size* and *Max Height Changes* are compulsory for digitizing, the other options can be applied depending on the specific conditions of your data.

Contrast Step Size

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

The *Contrast Step Size* is the minimum difference in contrast expected between the intensity values of the water column and the seafloor. The Signal Display Window may be helpful in determining a good 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.

1. Use the <Up> or <Down> arrow buttons to select the percentage for the *Contrast Step Size* and *Max Height Change* values.

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

2. Set the value in milliseconds for an *Offset from Nadir* by clicking the up or down arrow buttons.

3. Select *Show Offset from Nadir* to display the offset visually as a dashed line.

   • The line colour is set to the *Measure / Offset tools* colour in Tools > Options > Side Scan Editor.

   • The thickness of the line is set in the *Line thickness* field on the General tab in the Side Scan Editor Control tab.

| | |
|---|---|
| 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 seafloor. |

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.

4. Select the Verification check box.

5. Select a desired strength-level for the verification from the drop-down list. Choices range from "Very Weak" to "Very Strong". The default is "Normal".

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

6. Select the *Interpolation* option.

7. Select the size of the gap to be filled by clicking the up or down buttons beside the *Max Gap Size* field.

| | |
|---|---|
| Moving Average Filter | The *Filter* option smooths the digitized line. |

8. Click the Moving Average Filter check box.

9. Click the <Up> or <Down> arrow buttons to select the number of pings used for averaging in the Filter Width field.

| | |
|---|---|
| **Menu** Tools > Side Scan Editor > Altitude > Auto Digitize | You can use the Auto Digitize tool, or use the buttons at the bottom of the Altitude tab to complete digitizing. If you are using the buttons, continue with Step 10, otherwise, go to Step 11. |

10. Choose any of the following digitization options:

   • Click **To Start** to extend the digitized region to the back of the entire line.

   • Click **To End** to extend the digitized region to the end of the line.

   • Click **All** to digitize the entire selection or the entire line.

11. If you change a value in the tab, click **Recalculate** to update the Digitization.

12. Select *Overwrite* to enable re-digitization of selected areas without clearing. With Overwrite disabled, only zero-altitude/cleared fish heights will be available for digitization.

| | |
|---|---|
| **Menu** | Tools > Side Scan Editor > Altitude > Clear |
| **Tool** | |

13. Click anywhere on the seabed to select the start and end pings for digitization and also to indicate the side (port or starboard) on which to perform the calculations.

14. [Optional] Select the Clear command to remove the digitization.

## Manual digitizing

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

1. Select the Manual Digitize command.

2. Press and hold the mouse button and drag the cursor along the bottom contact.

| | |
|---|---|
| **Menu** | Tools > Side Scan Editor > Altitude > Manual Digitize |
| **Tool** | |

> 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 62)

2. Select the Interpolate Selection command.

| | |
|---|---|
| **Menu** | Tools > Side Scan Editor > Altitude > Interpolate Selection |

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

# Image Processing

Side scan imagery has been traditionally processed on a line-by-line basis, using a "waterfall" display, and a number of processes. These processes are available from the Process menu and the Process Side Scan toolbar:
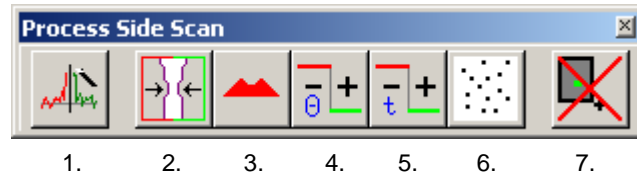


1. "SLANT RANGE CORRECTION" ON PAGE 80

2. "APPLY SLANT RANGE CORRECTION" ON PAGE 81

3. "BEAM PATTERN CORRECTION" ON PAGE 83

4. "ANGLE-VARYING GAIN" ON PAGE 88

5. "TIME-VARYING GAIN" ON PAGE 89

6. "DESPECKLE" ON PAGE 92

7. "PURGE REJECTED CONTACTS" ON PAGE 96

Imagery corrections can also be applied using the SIPS Template Editor (see "SIPS TEMPLATE WIZARD" ON PAGE 98).

## Revert/Reapply

| | |
|---|---|
| **Menu** | Tools > Side Scan Editor > Revert |
| **Tool** | ← |

| | |
|---|---|
| **Menu** | Tools > Side Scan Editor > Reapply |
| **Tool** | → |

The most recent correction made in *processed* mode can be undone or redone using the Revert and Reapply tools.

1. Select the Revert tool to reverse the last correction.

2. Select the Reapply tool to reapply the last correction.

This ability to go back a step without losing your work enables you to "toggle" between the before and after of a process. For example, with the Signal Display Window open, use Revert/Reapply to see the effect of slant range correction.

# Slant Range Correction

Side scan sonar is initially logged as a series of time-indexed intensity values for each ping. Slant range correction re-samples this raw side scan data to produce profiles of image pixels where the across-track scale is related to distance from nadir instead of time.

**Across-track positioning using height sources**

The standard slant range correction operation applies a sound velocity estimate to the two-way travel time for each intensity value. This produces an estimated ray length that when combined with the known sonar height (from the digitized or logged bottom) is used to produce the across-track distance to the pixel using simple trigonometry. This method presupposes that the bottom is flat across the ping and can result in the across-track misplacement of objects over varied or steeply sloping terrain.

SIPS can use a height source (such as a BASE Surface, grid, or tile) in the slant range correction operation to supply depth values to improve the across-track positioning of pixels.

In the following figure, raw side scan data is represented as a series of time (t) and pixel intensity values (i).

## Original time signal

| $t_1$, i | $t_2$, i | $t_3$, i |
|---|---|---|

*Two-way travel time (msec)* ⟶
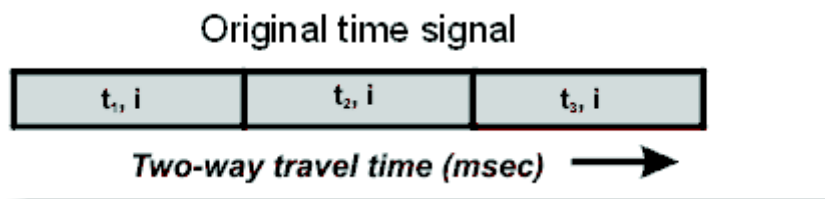
When a height source is used during slant range correction, a depth profile is first derived from the source along the path of the side scan ping. Then, the two-way travel time to each pixel is converted to a distance from nadir using the estimated velocity of sound in water. The distance is then used to calculate the position of the image pixel along the ping drawn from the height source.

Interpolation of across-track position from profile
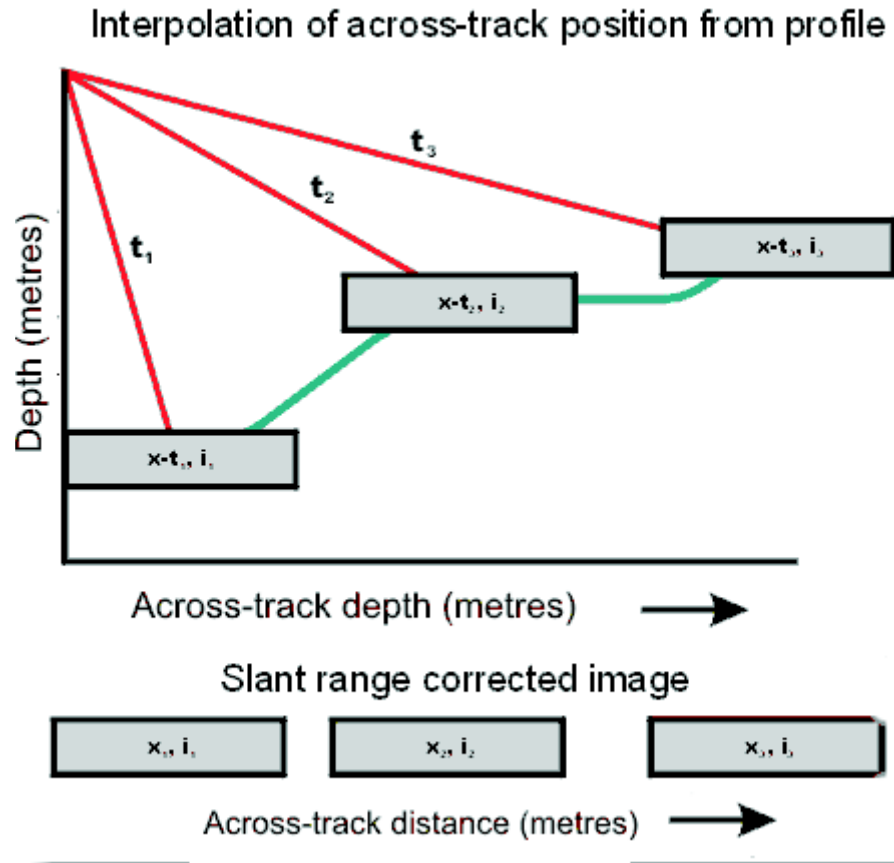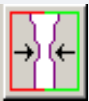


The resulting output is an intensity value indexed to across-track distance (x) rather than time.

## Apply Slant Range Correction

In the Side Scan Editor, Slant Range Correction is applied to data in raw mode. Other corrections in Side Scan Editor are made to data that has been slant range corrected. Once you slant-range correct data, you can return to raw mode to start again, however, any correction previously made to processed data will be lost.

| | |
|---|---|
| **Menu** | Process > Slant Range Correct |
| **Tool** |  |

1. Select the track line(s) in the Display window.

2. Select the Slant Range Correct tool.

The Slant Range Correction dialog box is displayed.
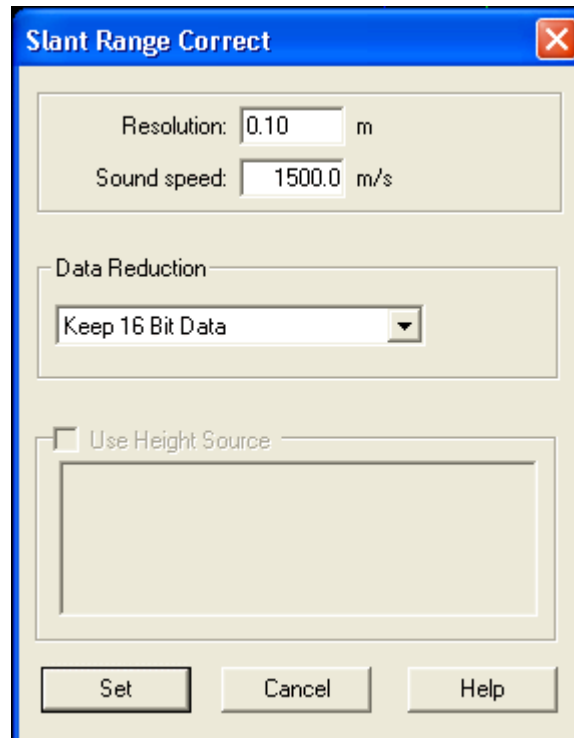
1.  Type the resolution (in metres or feet) for the slant range corrected file. The resolution value defaults to the minimum value appropriate to the sonar system. If this is not available then it defaults to 0.10 metres or 0.33 feet. A finer resolution generates a larger file.

2.  Enter the average speed of sound.

3.  Select the setting for Data Reduction from the drop-down list.

    •   The default is to use the 16-bit factor that is stored with the line.

    •   Select *Use 8-Bit Shift* if you want to create a slant range file using only eight bits of storage per pixel. This option is important if the data that you are slant range correcting was originally 16-bit and all 16 bits were maintained at conversion.

    •   Select *Use Shift Factor Per Line*

4.  Select the *Use Height Source* check box if you want to apply a height source, then select the height source from the list. (This is only available if you have a BASE surface or other height source open.)

5.  Click **Set**. The slant range corrected data is displayed in the Side Scan Editor.

# Beam Pattern Correction

Beam pattern correction uniformly removes angular artifacts from the sonar systems. This effect is produced by each transducer uniquely.

Beam pattern correction relies on a user-generated beam pattern file to identify and remove this effect. Tools are provided to create, view and process beam pattern files. The process uses the beam pattern file to uniformly correct the selected line(s), producing a more consistent appearance in mosaics.

Once the beam pattern file is created it can be applied to one or more lines in the Display window, or to one or more pings in the Side Scan Editor.

If no Beam Pattern Correction file exists you will need to create one in order to apply Beam Pattern Correction parameters. All the tools to create a beam pattern file are on the Beam Pattern Toolbar, accessible while the Side Scan Editor is open.

## Create Beam Pattern Correction File

To create a beam pattern file you must select an area that is as flat, featureless and homogenous as possible. It is recommended that a "patch test" be done during a survey over an area known to have such properties. In order to create a more robust and accurate beam pattern correction file, it is also important to vary the fish height over the range expected to be used during the survey.

Beam pattern files can be created from a range of pings or from the entire line.

| Menu | Tools > Beam Pattern Correction > Create |
| --- | --- |
| Tool |  |
| Pop-up | Create Beam Pattern… |

1. Use the Select Side Scan tool to select a homogeneous area in the Side Scan Editor window.

2. Select the Create Beam Pattern tool.

The Create Beam Pattern dialog box opens and displays the Start and End ping numbers for your selected area.
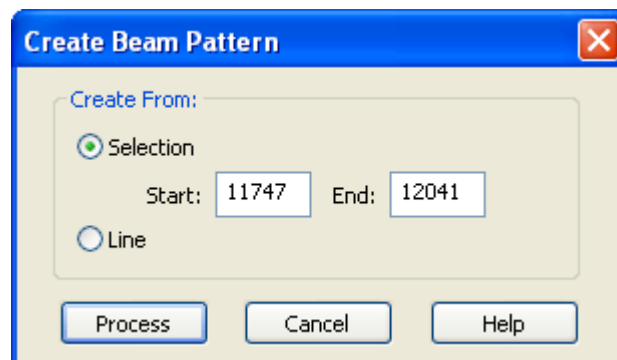
You can create a beam pattern file from a range of pings larger than the range visible in Side Scan Editor window, by entering the start and end ping numbers. However, the pings should be from a homogeneous area.

3. Select **Process** to create the Beam Pattern file.

The Windows Save As dialog box is displayed.

4. Type a name for the Beam Pattern file and select **Save**.

The file is saved as a SIPS Beam Pattern File with extension *.spb.

You can also create the file from a whole line, but the same criteria should be applied.

1. Select a track line.

2. Select the Create Beam Pattern tool.

The Create Beam Pattern dialog box is displayed with the Create From Line option selected.

3. Select **Process** to create the Beam Pattern file.

4. Name and save the file.

## Adjust Beam Pattern

The **Save Beam Pattern**, **Save Beam Pattern As**, **View**, **Remove Spikes**, **Smooth,** and **Reload** tools are now active in the menu and on the Beam Pattern Correction toolbar (below).
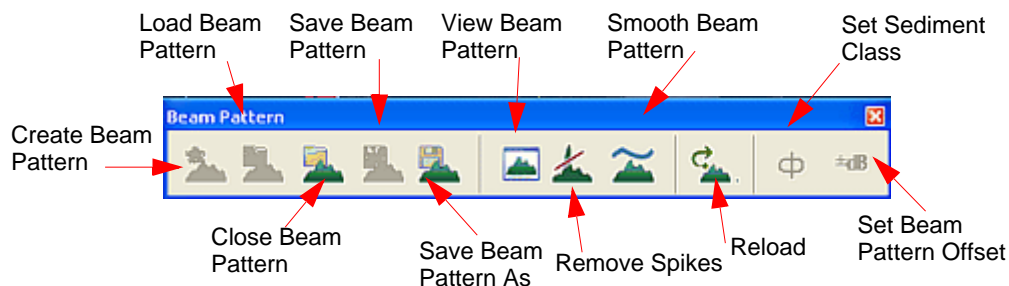


View Beam Pattern

To see a graph of the beam pattern

1. Select the View Beam Pattern command.

| Menu | Tools > Beam Pattern Correction > View |
|------|----------------------------------------|
| Tool | |

The Beam Pattern Graph window is displayed.
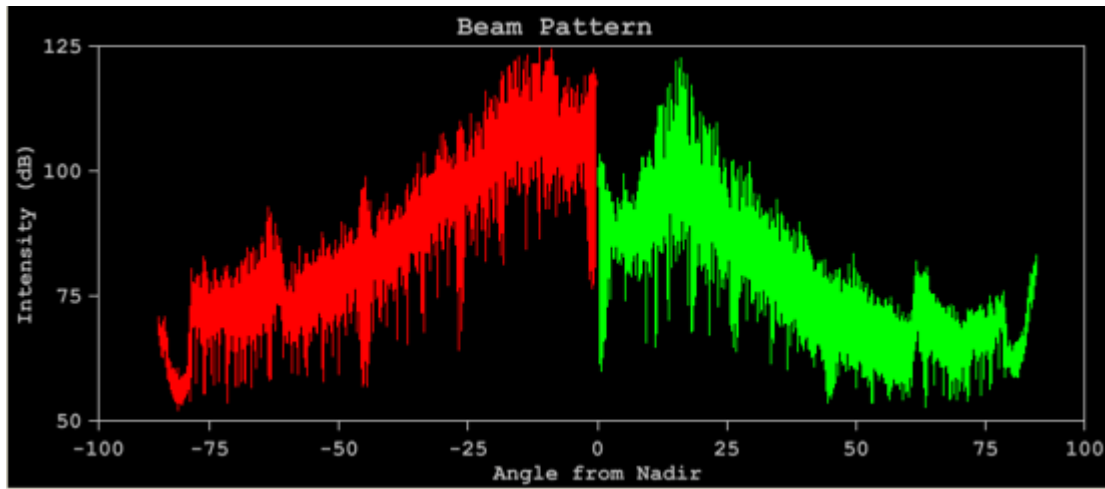


**Remove Spikes/Smooth**

| | |
|---|---|
| Menu | Tools > Beam Pattern Correction > Remove Spikes |
| Tool |  |

The Remove Spikes tool removes anomalies from the Beam Pattern Graph.

2.  Select the Remove Spikes command.

The next step is to apply the Smooth Beam Pattern tool. The Smooth Beam Pattern tool uses a moving average filter to reduce noise in the beam pattern.

An example of the graph before and after using the Smooth tool is shown below. The left-hand window shows the unsmoothed graph and right-hand window shows the graph after smoothing.



After smoothing

Before smoothing
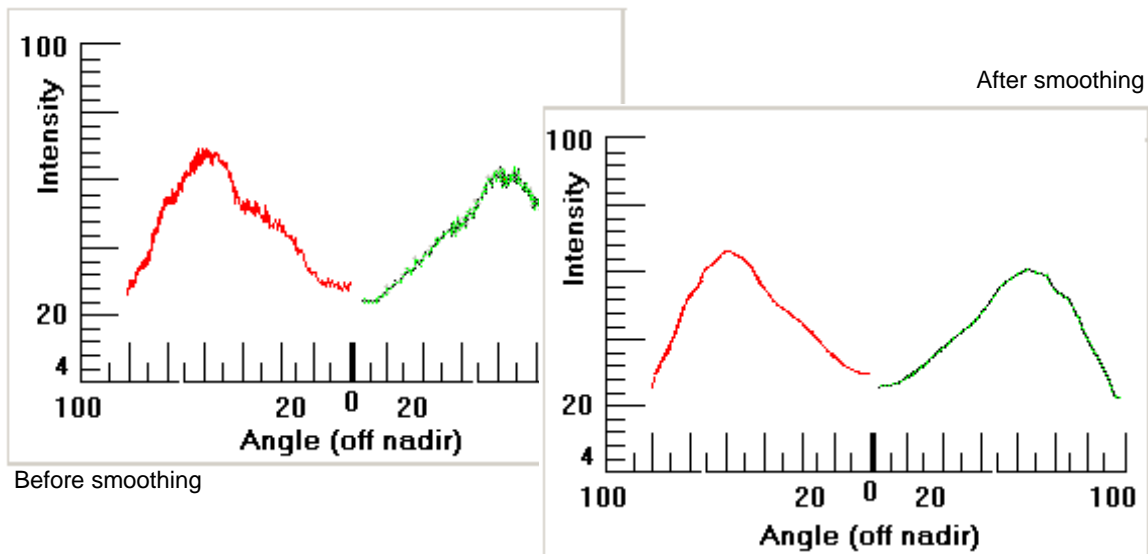
| | |
|---|---|
| Menu | Tools > Beam Pattern Correction > Smooth Beam Pattern |
| Tool |  |

3.  Select the Smooth Beam Pattern tool.

The Smooth Beam Pattern tool can be used repeatedly until the preferred degree of smoothing is reached.

Reload

The **Reload** tool reloads saved changes from the active beam pattern file. Any changes to the beam pattern after the last save are lost.

For example, if you remove spikes and then Save, then apply Smooth Beam a few times, the Reload tool will restore the graph to the point just after you saved it.

| | |
|---|---|
| **Menu** | Tools > Beam Pattern Correction > Reload |
| **Tool** | |

4. Select the Reload tool.

When you have completed smoothing the data, you can save these settings using the Save Beam Pattern or Save Beam Pattern As… tools.

5. Select Save Beam Pattern to save the filtering and smoothing changes to the Beam Pattern File.

6. Select the Save Beam Pattern As command to save the file as a new file.

## Apply Beam Pattern Correction

The Beam Pattern Correction file can be applied to a selection in the Side Scan Editor or to track line(s) in the Display window.

| | |
|---|---|
| **Menu** | Process > Beam Pattern Correction |
| **Tool** | |

1. Make a selection in the Editor, or,
   select a track line or group of track lines in the Display window.

2. Select the Beam Pattern Correction command.

The Beam Pattern Correction dialog box is displayed.
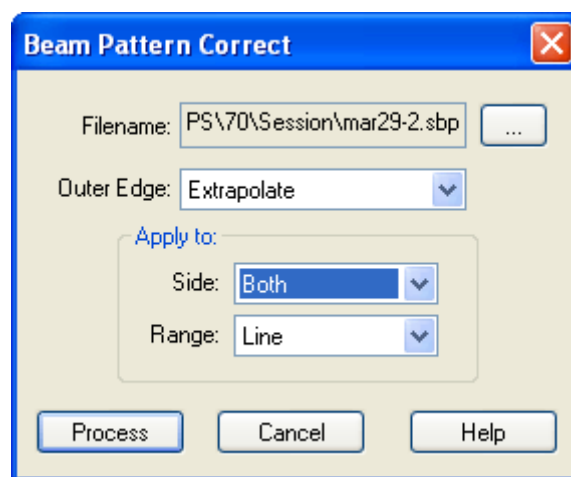


The Beam Pattern file will default to the last file that was created, loaded, saved or executed. You can use the Browse button to locate a different file for processing. The default location for Beam Pattern files is <Data Dir>/Sidescan.

The outer edges of some pings may have been recorded at angles beyond the extent of the beam pattern file being used. The Outer Edge option must be specified for handling these cases.

3. Select a method for handling sonar data collected outside the bound of the beam pattern file:"

- Extrapolate: Extrapolate the beam pattern to correct angles beyond the outer edge of the generated beam pattern file.

- Trim: Remove sonar data that is beyond the extent of the beam pattern file. Trimmed data will appear as 0 intensity white space in the Side Scan Editor.

- Do Nothing: Does not correct sonar data beyond the extent of the beam pattern file. Uncorrected data will appear the same before and after Beam Pattern Correction in the Side Scan Editor.

4. Apply the Beam Pattern Correction to the both sides of the side scan or to either the starboard or port sides.

5. You can apply the Bean Pattern Correction to your entire survey by selecting all the track lines, or you can open the Side Scan Editor and select a portion of the data:

- *Ping*: the correction will apply to the single ping selected in the SIPS Preview window.

- *Selection*: the correction will apply to the range of pings you selected in the SIPS Preview window.

- *Line*: the correction will apply to the entire trackline you have selected.

6. Click **Process** to apply the correction.

# Angle-Varying Gain

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

AVG correction can be applied to a selection or a single ping in the Side Scan Editor or to one or more track lines in the Display window.

| Menu | Process > AVG Correct |
|---|---|
| Tool |  |

1. Make a selection in Side Scan Editor. (See "SELECTION" ON PAGE 62 for selecting areas and pings).

2. Select the AVG Correct tool.

The AVG Correct dialog box is displayed.
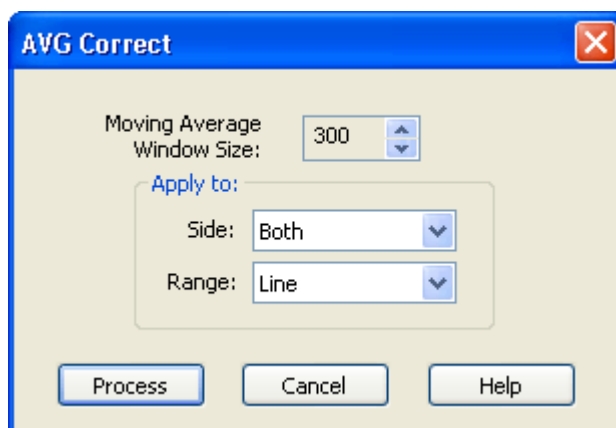


3. Select the number of adjacent pings to include in the moving average by clicking *Moving Average Window Size* up or down arrow buttons.

For example, an average based on a window size of 100 will include 50 pings before and after the ping being corrected.

4. Apply angle-varying gain to the port, starboard, or both sides of the side scan.

5. Apply the angle-varying gain correction to either a *Ping*, *Selection*, or *Line* by selecting the appropriate option.

6. Click **Process**.

# Time-Varying Gain

Time-Varying Gain (TVG) Correction allows you to adjust the signal intensity by applying a non-uniform, time-dependent gain.

During acquisition, this 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 maybe necessary to augment the TVG correction applied during acquisition.

The TVG Correction command also allows you to apply a uniform gain correction without applying the any time-dependent gains by using only the Gain control.

TVG Correction can be applied to a single ping, selection, or the currently selected line in the Side Scan Editor, or to one or more track lines in the Display window.

| | |
|---|---|
| **Menu** | Process > TVG Correct |
| **Tool** |  |

1. Make a selection in the Side Scan Editor. (See "SELECTION" ON PAGE 62 for selecting areas and pings).

2. Select the TVG Correct command.

The TVG Correct dialog box is displayed.
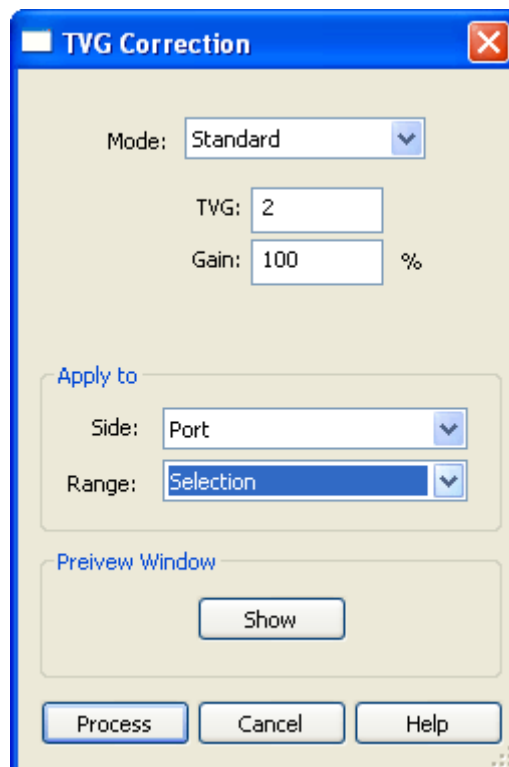
There are two modes for TVG correction: Standard and Advanced.

3. Select either the Standard or Advanced mode from the *Mode* drop-down list. (The options available depend on the mode selected.)
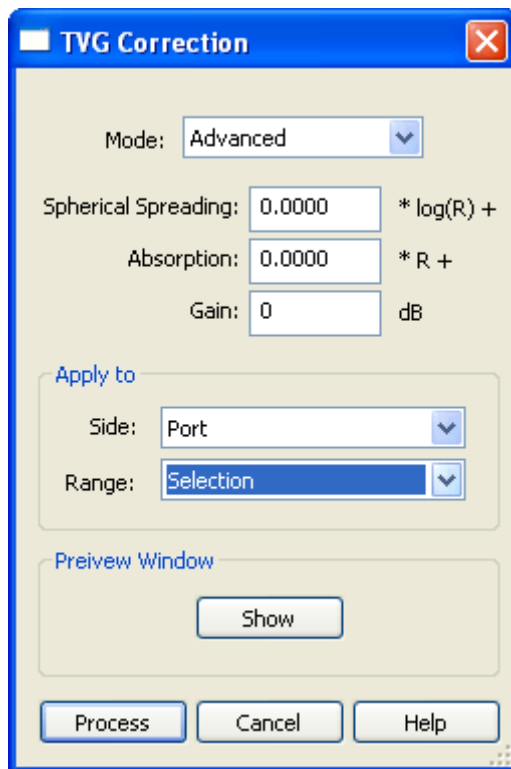
The TVG dialog box can expand into a Preview window for data selected or superselected in the Side Scan Editor window. Preview is not available when a track line is selected (the **Show** button is greyed out.)

4. Click **Show** to view the Preview window. To close the window, click **Hide**.

The Preview window is divided into three panes:

- Gain (shown as a percentage)

- Intensity values before TVG is applied.

- Intensity value after TVG is applied.

Changes are instantaneously shown in the Preview window.

5. The TVG correction can be made to port, starboard, or both sides by selecting the appropriate from the *Side* drop-down list.

6. The correction can be made to a single ping, a selection, or the entire line by selecting any of these options from the Range drop-down list. (The options depend on the selection made in the Side Scan Editor or in the Display window).

Standard

The Standard mode contains two options for correcting TVG.

7.  Increase or decrease the percentage *Gain* by clicking the up or down arrow buttons.

8.  Increase or decrease the *TVG* value by clicking the up or down arrow buttons. This changes the TVG curve.

Advanced      The Advanced mode contains more precise options for correcting TVG.

9.  *Spherical Spreading* compensates for the loss of energy caused by signal displacement along the sea floor. Increase or reduce the signal intensity by clicking the Spherical Spreading up or down buttons.

10. The *Absorption* value compensates for attenuation due to energy loss in water. The scale factor is in dB/unit distances. Click the up or down arrow buttons to increase or reduce this value.

11. Click **Process** to apply the TVG correction.

# 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.

| | |
|---|---|
| **Menu** | Process > Despeckle |
| **Tool** | |

The following steps are used for despeckling images:

1. Select a track line.

2. Select the Despeckle tool.

The Despeckle dialog box is displayed.
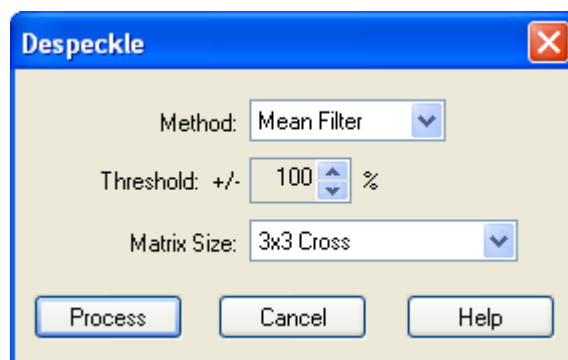
3. Select the Mean or Median Filter from the *Method* drop-down list.

4. Use the up and down arrows to select a *Threshold* for defining an acceptable range in which the pixel value will not be replaced.

For example, if the mean value is 50 and the threshold is +/- 20% then the acceptable range is 40 to 60. Thus a pixel value of 38 would be replaced by the mean value of 50 as it is outside the range, but a pixel value of 42 would not be replaced.

5. Select a pattern of neighbouring pixels from the *Matrix Size* drop down list be used in calculating the mean/median.

6. Click **Process**

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

# 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 can be georeferenced, have positions calculated, and have attributes attached to them in the Side Scan Editor using the Add Point or Add Line Contacts tools.

In SIPS, you can place individual contacts directly in the waterfall view. A contact record is created for each contact, and includes sufficient information to be subsequently exported to a CARIS map or to a database application. (

A contact file is created for each survey line and is stored in the line folder within the project directory structure. Each contact file is a text file that can be viewed and edited using a text editor. Each file consists of zero or more contact records.

Contact information can be exported. (This is described in Export "CONTACTS" ON PAGE 491 of the User Guide.)

## Create Contacts

Point type contacts can be created for objects such as rocks, underwater posts or pile as well as wrecks. Contacts can also be created for continuous objects such as pipelines or underwater cables.

Contact information is entered on the General tab of the Control window and updated on the Contact Details tab.

1. Make sure the Control window is open and the General tab in the Side Scan Section is displayed.

2. Enter information in the appropriate fields on the General tab.

   • Contact Type: Select a point or line contact type from the appropriate drop-down list. Each contact type is assigned a CARIS feature code, and there is an identifying symbol for each type. The contact types shown in the lists and the corresponding CARIS feature codes are stored in …\Hips\system\ContactFeature.txt.

   • *User Initials*: Type a maximum of three characters to identify the creator of the contact. The initials are used in the editor as part of a contact identifier.

   • *Prefix*: The prefix is usually the line number. It is combined with the contact number to generate a 12-character key for database applications.

   • *Image Width*: (optional) Enter a width for any image file created from the contact.

   • *Image Length*: (optional) Enter a length for any image file created from the contact.

- *Display with single colour.* Select this to have all contact markers and their identifiers appear in the same default colour. (The default colour can be set in Side Scan Editor Display Options on the Tools menu.) Leave blank to have a one colour apply to the contact markers and a different colour applied to the bounding box and contact labels.

### Point Contact

| | |
|---|---|
| **Menu** | Tools > Side Scan Editor > Contacts > Add Point Contact |
| **Tool** |  |

3. To insert a point contact, select the Add Point Contact tool and click 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 by the initials entered in the General tab, and a four-digit number (starting at 0001). The information entered in the General tab is assigned to the contact.
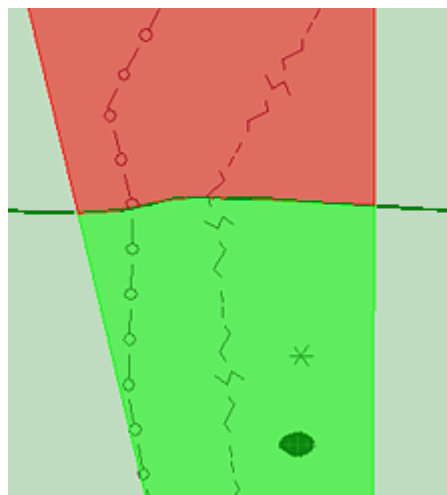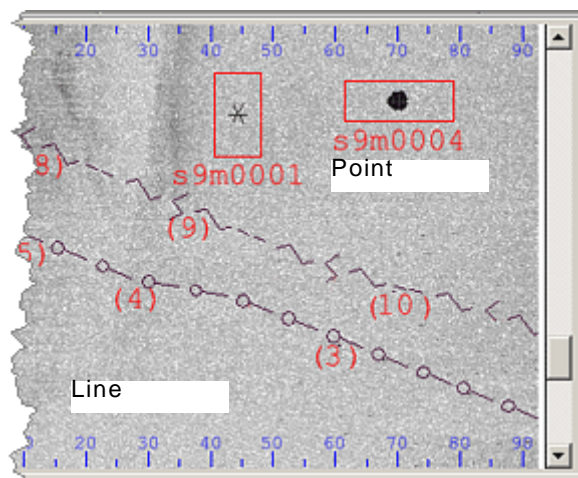


The image bounding box always snaps to the contact so that the contact point is in the middle of the box. The bounding box represents the image area of the contact. You can adjust the size of the image using the *Image Width* and *Image Length* fields on the Contract Details tab in the Control window.

### Line Contact

| | |
|---|---|
| **Menu** | Tools > Side Scan Editor > Contacts > Add Line Contact |
| **Tool** |  |

To insert a contact record for a line:

1. Select the Add Line Contact tool and click the point where you want to insert the contact.

2. Click another section of the Side Scan Editor.

The two points are joined by a line.

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

4. Select **Edit Line**> **Remove last** to delete the line segment just created.

5.  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 Editor. Each line is identified with a number, and each type of line is drawn in a different style. The information entered in the General tab is applied to the contact. The contact is identified with the initials entered in the General tab, along with a four-digit number (starting at 0001).

**Repeat New Feature**

| Menu | Tools > Side Scan Editor > Contacts > Repeat New Feature |
|------|------|
| Tool | |

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:

1.  Select the Repeat New Feature command.

2.  Enter contact information in the appropriate fields on the General tab in the Control window.

3.  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.

## Editing Contacts

You can reposition both line and point contacts and update the data associated with a contact.

| Menu | Tools > Side Scan Editor > Contacts > Select Contact |
|------|------|
| Tool | |

The Select Contact tool is used to select point or line contact(s) for editing.

1.  To select a *single* contact, click on or draw a box around the contact so that it becomes highlighted. Clicking on any part of a line will select the entire contact.

2.  To select a number of contacts, press the mouse button and drag the cursor across an area so that the contacts within it are highlighted.

**Move Contact**

| Menu | Tools > Side Scan Editor > 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.

Changing Contact Information

You can change the information that is associated with the contact when it was first created, either by typing it in or with measurement tools.

1. Choose the Select Contact tool.

2. Select a single contact.

The contact is highlighted and the contact information is displayed in Contact Details tab in the Control window.

3. Type new information in the fields and/or select a new object type from the drop-down list.

4. Click **Update**.

Use the Measure Distance and Measure Shadow tools on the Side Scan Editor toolbar to amend the contact information.

Use Measure tools

5. 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.

If measuring in the SIPS Preview window, the value in the Contact Width field will reflect the measurement as you are making it. If measuring in the Zoom tool, the value will be updated when the mouse button is released.

| Menu | Tools > Side Scan Editor > Measure Distance |
|------|---------------------------------------------|
| Tool | |

6. To retain the updated information, release the mouse button and then the <Ctrl> key.

7. To update the Contact Length information, choose the Measure Distance tool and press <Alt> while dragging the cursor across the contact area.

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

| Menu | Tools > Side Scan Editor > Measure Shadow |
|------|-------------------------------------------|
| Tool | |

9. Click **Update**. If you do not click Update, the values will be lost when the contact is de-selected.

The contact information is updated.

Purge rejected contacts

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.

The rejected contact is deleted.

| Menu | Tools > Side Scan Editor > Contacts > Purge Rejected Contacts |
|------|--------------------------------------------------------------|
| Tool | |

# Recompute Contact Positions

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

1. Select the ship or towfish track lines layer in the Layers tab of the Control window.

| **Menu** | Process > Recompute Contact Position |
| --- | --- |

2. Select one or more track lines.

3. Select the Recompute Contact Position command.

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.

# SIPS Template Wizard

In the SIPS Template Wizard you can set options for imagery correction into a single flow and saved the settings in a template file to be consistently applied to data.

The Wizard can be run from Side Scan Editor to apply any or all of:

- Slant Range Correction
- Beam Pattern Correction
- AVG Correction
- TVG Correction
- Despeckle

as well as creating GeoBars to be used in mosaics.

The SIPS Template Wizard and its processes are described in"USING SIPS TEMPLATE WIZARD" ON PAGE 441 in the PROCESS IMAGERY DATA section of the Users Guide.

# *6*

# *Process Single Beam Data*

The 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 Workflow

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



Single Beam Workflow
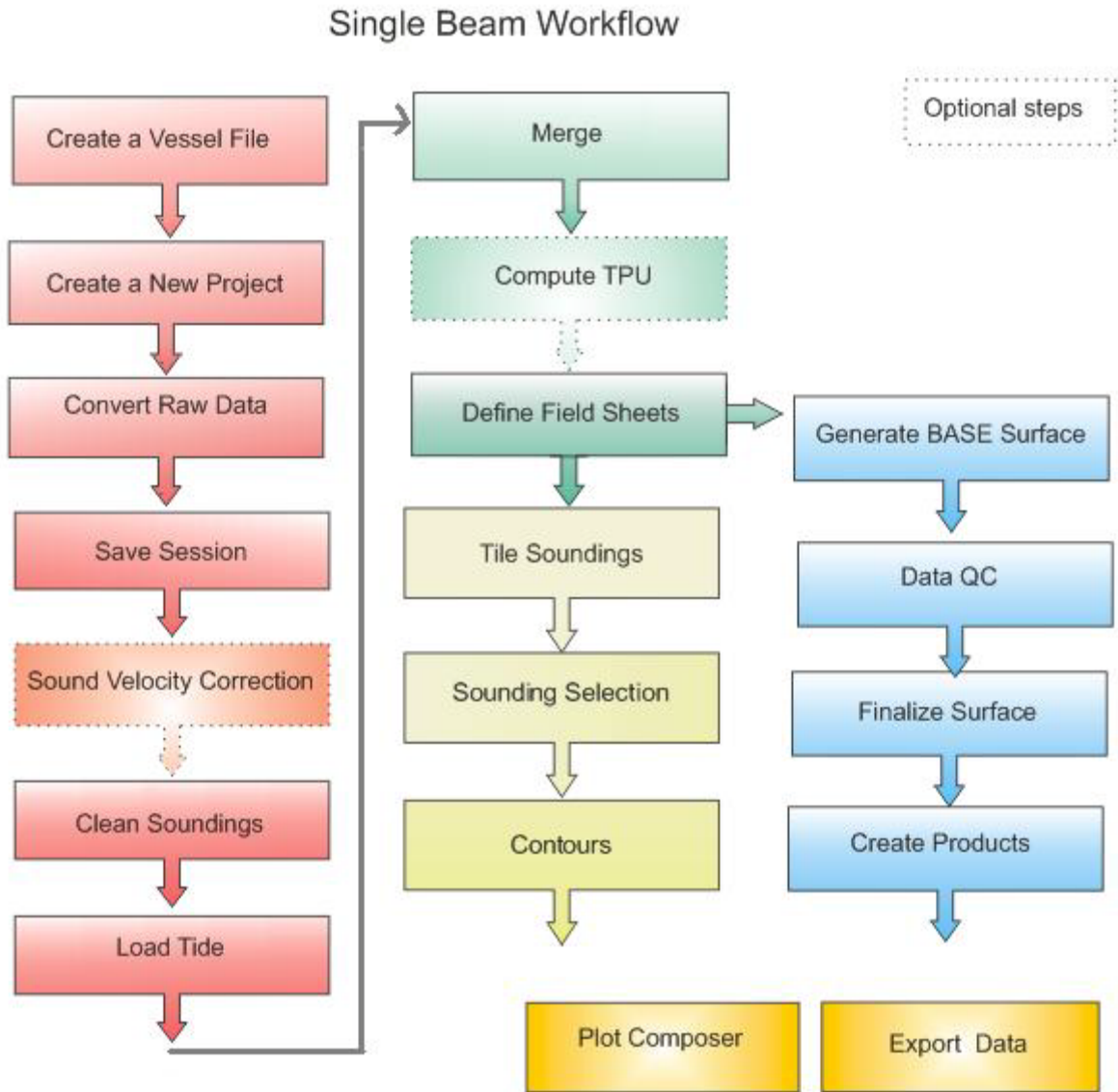
# Single Beam Editor Interface

| | |
|---|---|
| **Menu** | Tools > Single Beam Editor |
| **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.
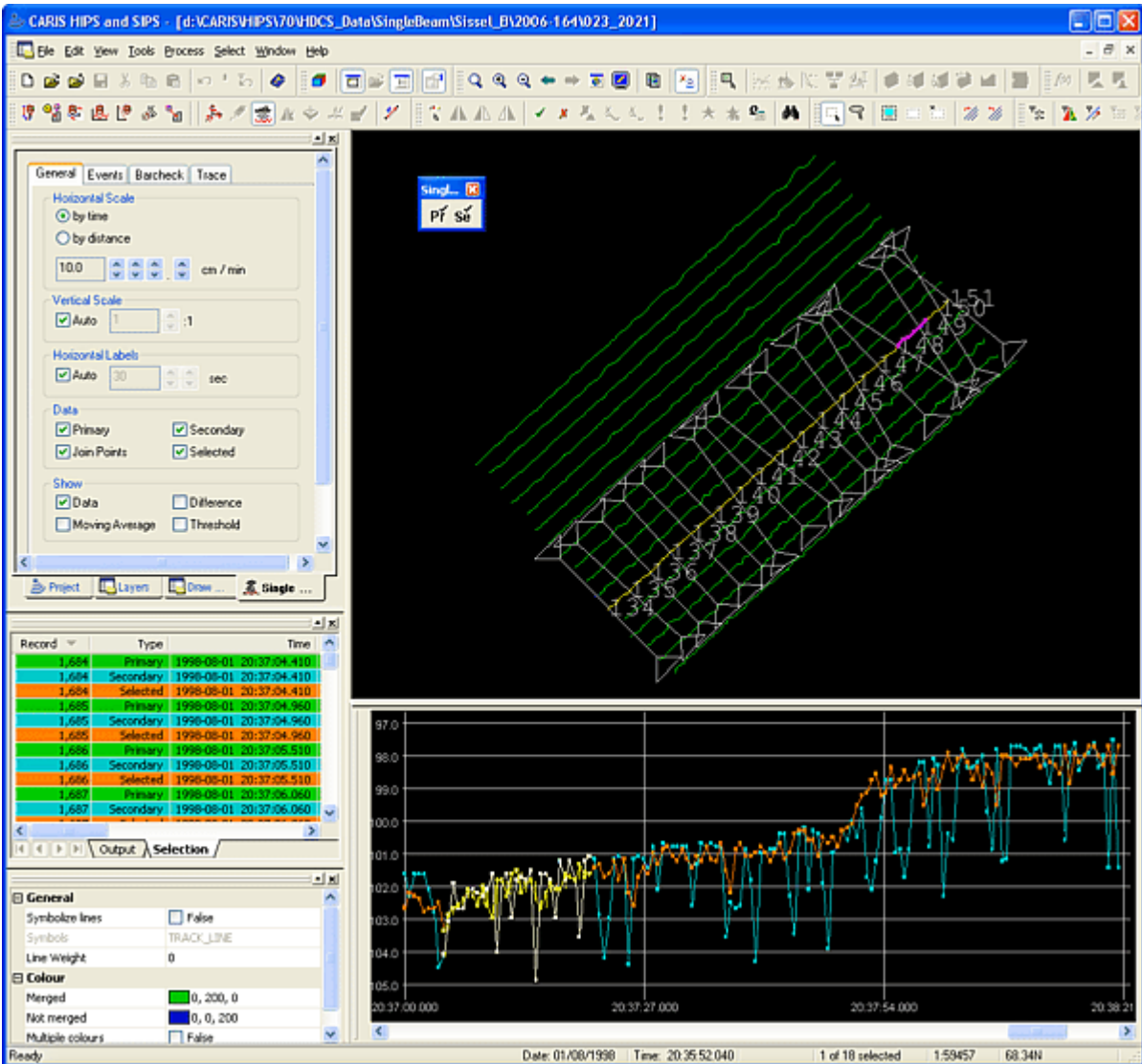


The Single Beam Editor window is located (by default) below the Display window. The editor can be un-docked and moved to another location like other windows in HIPS.

The display and events options are controlled from the Single Beam Editor tab in the Control 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 timestamps 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 109). The toolbar can moved and docked in the interface like other toolbars (see "DISPLAY TOOLBARS" ON PAGE 31).

# Single Beam Editor tab

The Single Beam Editor tab in the Control window contains four tabbed pages:

- "GENERAL TAB" ON PAGE 104

- "EVENTS TAB" ON PAGE 105

- "BARCHECK TAB" ON PAGE 107

- "TRACE TAB" ON PAGE 108
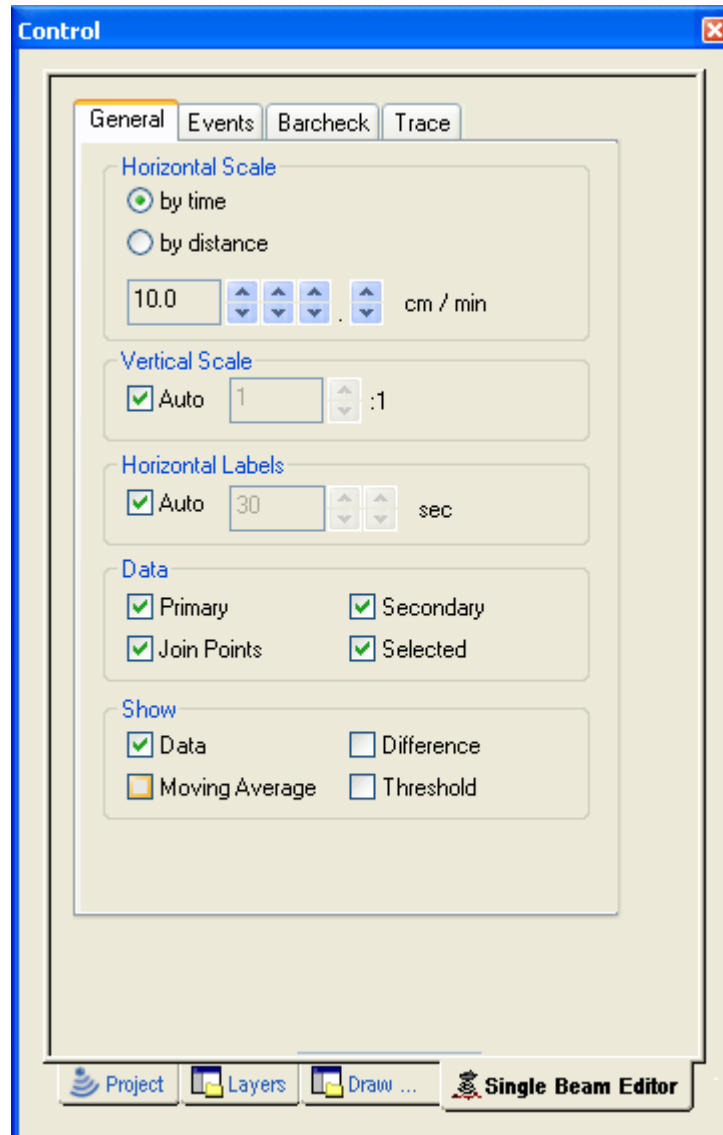
# General tab

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 tab in the Control window. For more information on applying Moving Average filtering to single beam data, see "SINGLE BEAM CLEANING" ON PAGE 258 in the Users Guide.

- *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.
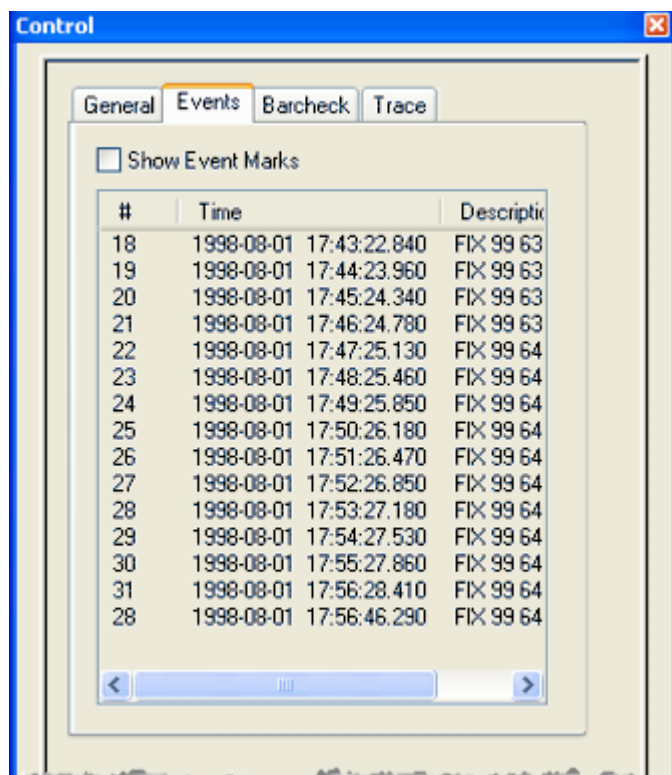
To display the event marks in the Single Beam Editor window:

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

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



Event mark

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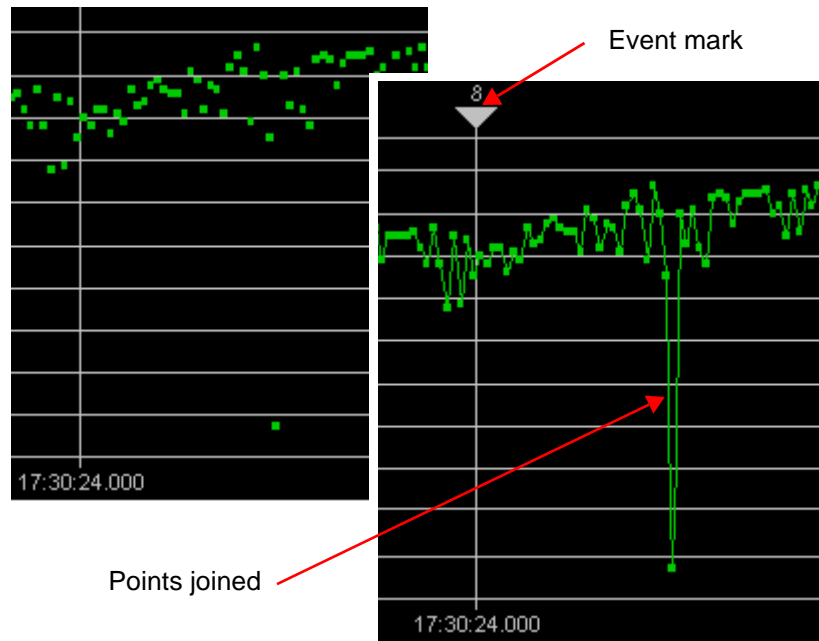
To centre the data in the Single Beam window on a specific event mark:

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.
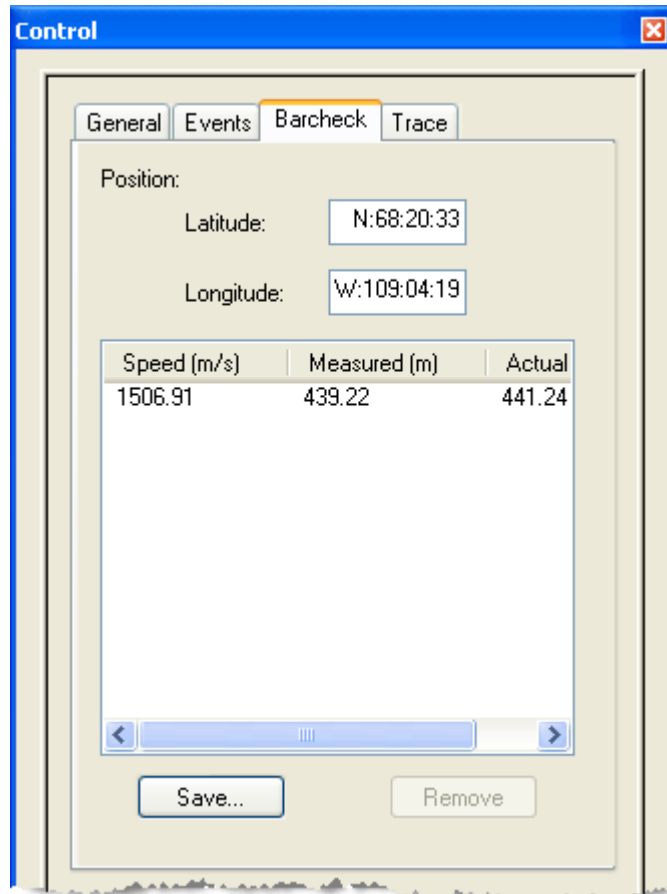


| | |
|---|---|
| **Menu** | Tools > Single Beam Editor> Add to Barcheck |
| **Pop-up** | Add to Barcheck |

To add barcheck values:

1. Select soundings in the Single Beam Editor window.

2. Select Add to Barcheck command.

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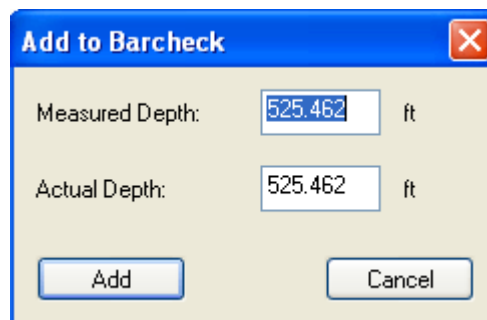
The *Measured Depth* field displays the depth that was recorded by the sounder. If more than one sounding is selected, then a mean is calculated from the selected soundings.

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.
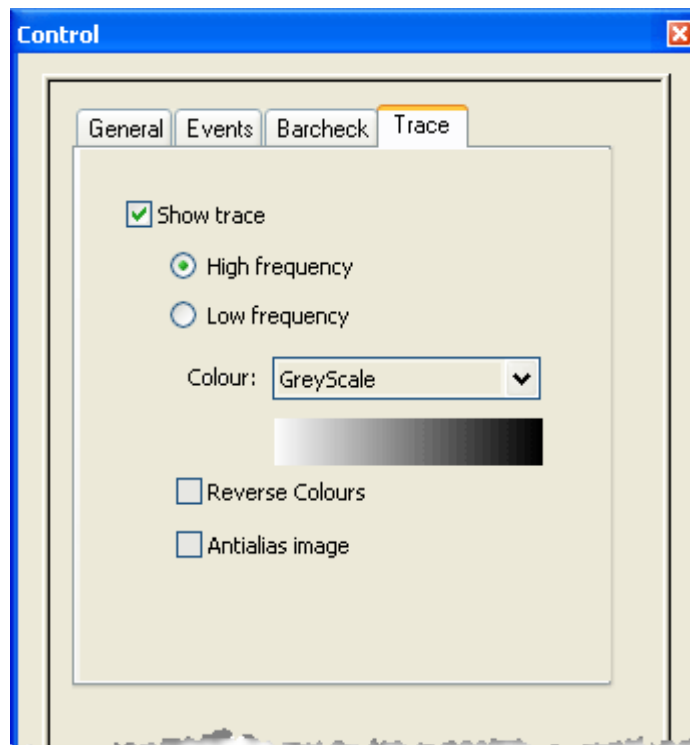
# Primary and Secondary Data

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. Make sure the Single Beam Editor tab in the Control window is displayed.

2. Select or de-select 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.)
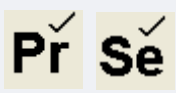
1. In the General tab, select the Primary and Secondary check boxes so that all the frequency data is visible in the editor.

| Menu | Edit > Single Beam > Primary/Secondary |
|---|---|
| Tool | Pr✓ Se✓ |
| Pop-up | Primary/Secondary |
| Key | <1>/<2> |

2. In the editor window, press and hold the mouse button and drag the cursor to highlight both the Primary and Secondary data.

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 lets 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 lets you set the echo sounder's depth values to the analog sounding depth values.

## Change Depths

1. Set the *Selected* check box in the Data fields of the SingleBeam Editor General tab, if needed.

2. Select a sounding(s) in the Single Beam Editor window.

3. Select the Change Depths command.

The Change Depths dialog box is displayed.

| | |
|---|---|
| **Menu** | Tools > Single Beam Editor> Change Depths |
| **Pop-up** | Change Depths |
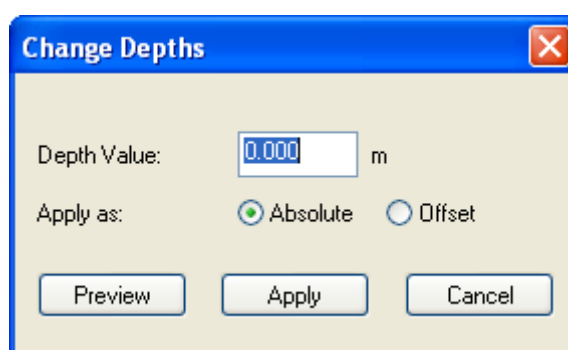
To move the sounding to a specific depth:

4. type the new depth in the *Depth Value* field, and

5. select the *Absolute* option.

To move the sounding by a set amount:

6. Type the amount by which to move the sounding.

   To move the sounding to a deeper depth level, enter a positive value (for example, 5.5). To move the sounding to a shallower depth level, enter a negative value (for example, -5.5).

7. Select the Offset option.

8. Click **Preview** to view the sounding at the new level.

9. 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.

When the mouse is positioned over the selected data, the cursor becomes a two-headed arrow with a Z symbol beside it.

2. Press and hold the mouse button, and drag the cursor either up or down.

The highlighted soundings move with the cursor.

3. Release the mouse button when the soundings are at the new depth.

4. To cancel the effect of moving these soundings to the new depths, select **Undo** from the Edit menu.

## Add Depths

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.

| Menu | Tools > Single Beam Editor > Add Depths |
|------|------------------------------------------|
| Pop-up | Add Depths |

1. Select the Add Depths command.

The Add Depths dialog box is displayed.



2. Select the *Time* field and type a new timestamp 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 114).

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

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

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.

A sounding rejected within the Single Beam Editor has the Rejected by Swath Editor flag turned on (the same status flag is used for both swath and single beam data).

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 visible.

| | |
|---|---|
| **Menu** | View > Show Rejected |
| **Tool** | ✗ |
| **Key** | <F4> |

You can show or hide rejected data. When the rejected data is turned off, these soundings are no longer visible in the Single Beam Editor.

1. Select the Show Rejected command.

## Accept Data

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

Change Rejected soundings back to Accepted status.

1. Select the rejected data so it is highlighted.

2. Select the Accept command.

The soundings are now flagged as Accepted.

## Query Data

View data on selected soundings.

1. Select the data to be queried.

2. Select the Query command.

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

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

· Record

· Type (primary, secondary, or selected)

· Timestamp

· d-Time

· Approximate Position

· Distance

· d-Distance

· Depth

· d-Depth

· Status

Change sounding status

Once data information is displayed, you can change the status of the sounding.

1. Select the sounding data in the Worksheet window.

2. Select Accept or Reject.

The changed status will be displayed in the Selection tab.

## 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 tab.

# 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.

1. Select a track line.

2. Select the Set Filters command.

| Menu | Tools > Set Filters |
|------|---------------------|
| Tool | |

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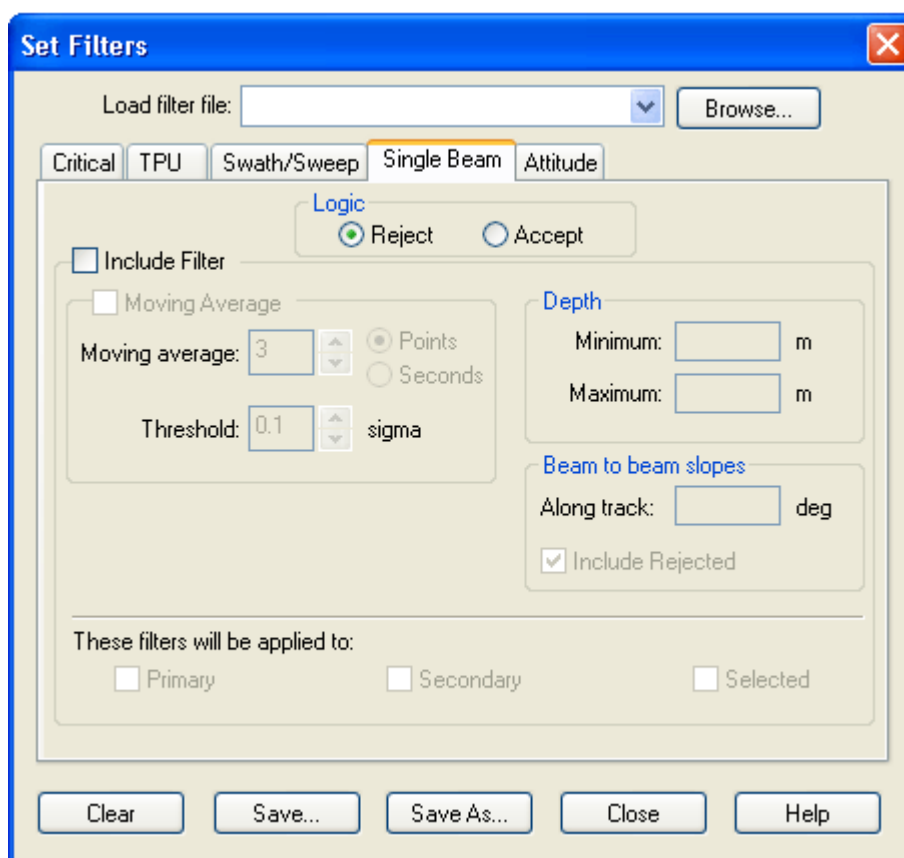
3. Make sure the Single Beam tab is displayed.

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 115)

   • *Depth* (see "DEPTH" ON PAGE 116)

   • *Beam-to-beam slopes* (see "BEAM-TO-BEAM SLOPES" ON PAGE 116)

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 104.

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.

| | |
|---|---|
| **Menu** | Tool > Apply Filters > 1 Screen/To End of Line/Selected Lines/ All Lines |
| **Tool** |  |

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

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.

**Selected Lines**. Apply the filters to all of the currently selected track lines. This function does not require that the editor be open.

**All Lines**. Apply the filters to all track lines currently open in the Display window. This function does not require that the editor be open.

# 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 BASE Surface areas 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 water column imagery (see "WCI DATA IN SUBSET EDITOR" ON PAGE 460.)

in the HIPS and SIPS Users Guide.

# Subset Editor Interface

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

Subset bounding box

| | |
|---|---|
| **Menu** | Tools > Subset Editor > Load Subset |
| **Tool** |  |

## Open 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.



3. Select the Load Subset command.

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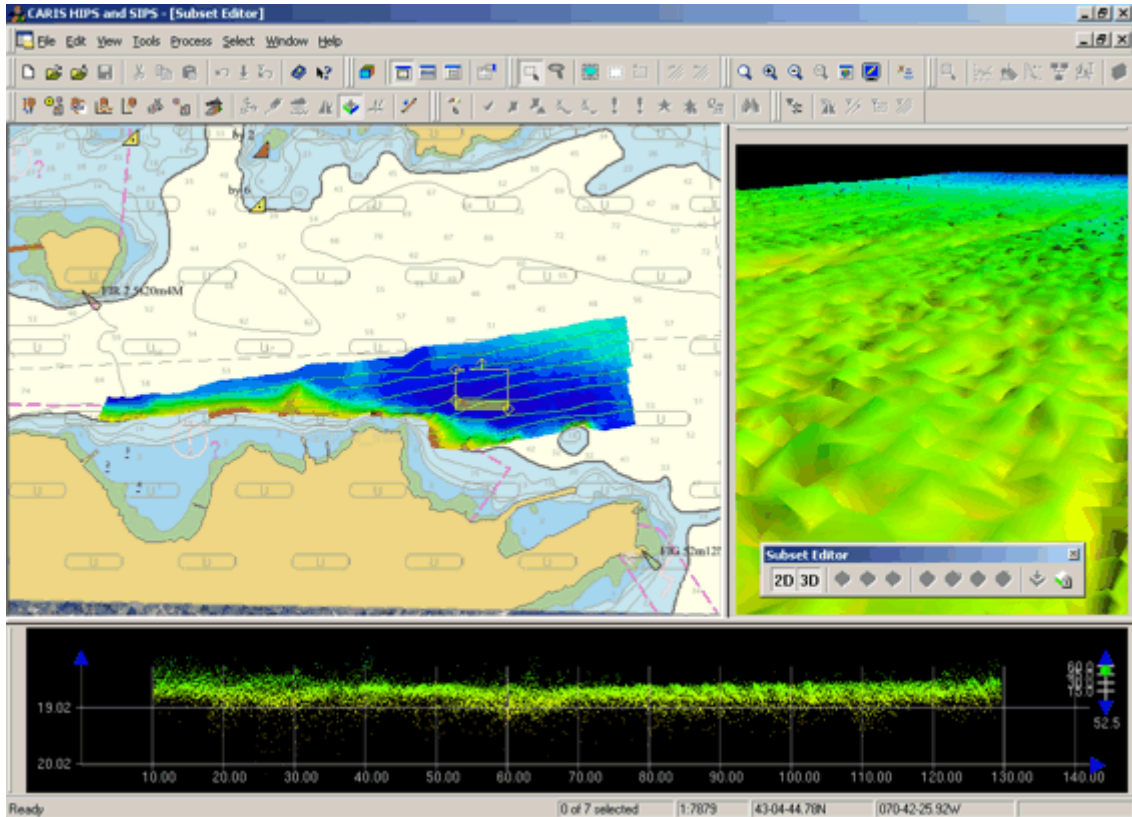
The Subset Editor has the following components:

- **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.

- **The 3D View**: The 3D (three-dimensional) View shows all soundings in the subset area. The view in the window can be rotated or changed.

- **The 2D View**: The 2D (two-dimensional) View displays soundings from the subset area that are within the 2D slicing box.

## 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 .
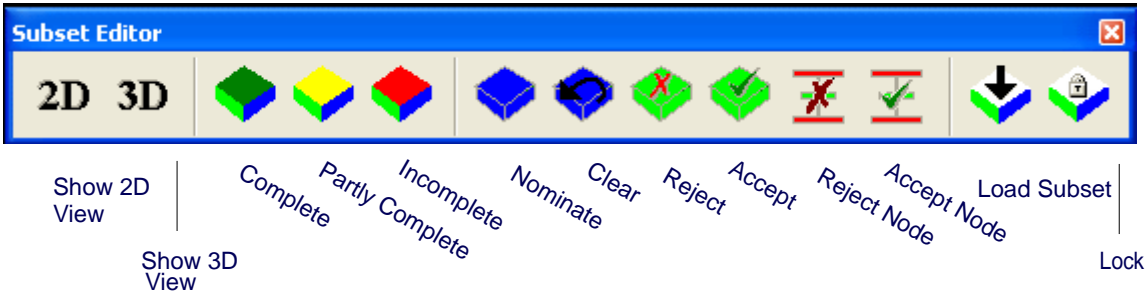


## Size and Resize Subset

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.

Below is an example of the subset bounding box.



The arrow helps maintain a sense of direction when the subset is rotated.

The 3D bounding box determines the subset area. The area inside this box is displayed in the 3D View.

The 2D slice box determines the area that is displayed in the 2D Subset View.
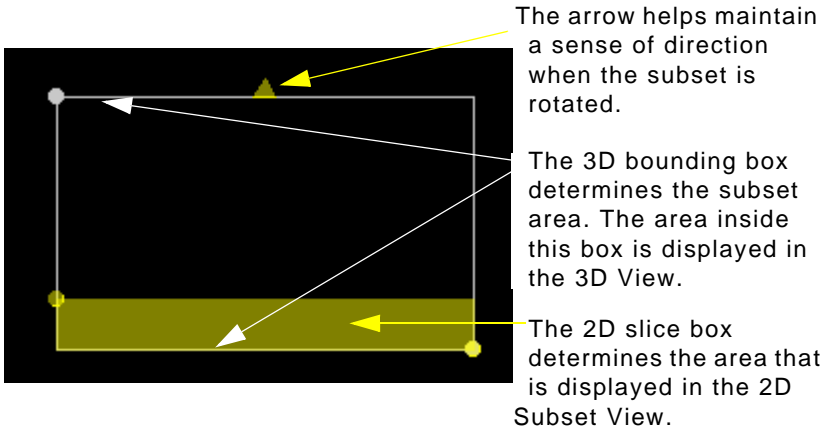
To resize the subset areas:

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 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 corner balls of the box.

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

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 Editor tab, select the *Auto Load Subset* check box to turn this option on.

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.

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

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

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

# Subset Editor Tab

Subset Editor tab is opened in the Control window when the Editor is started. The tab controls the data properties for the data layers in the Editor. The image below shows the properties for the HIPS Data layer.
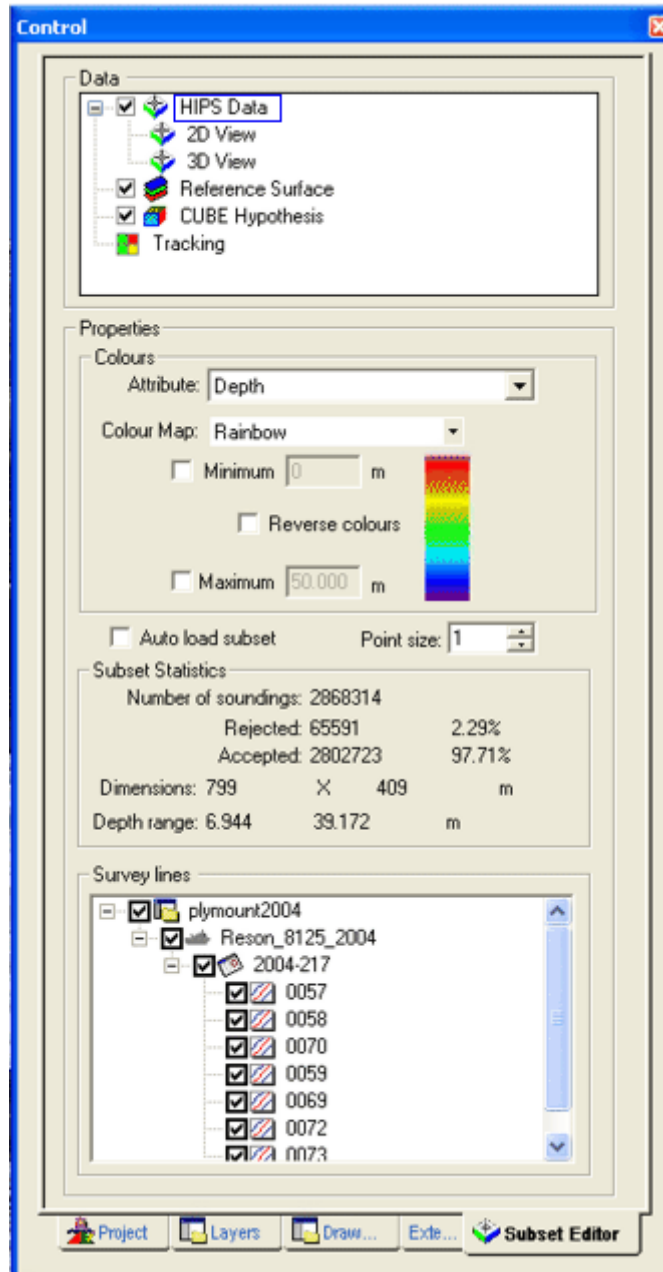


The top part of the tab displays the different data layers that can be viewed in Subset Editor. When you select one of these layers, the bottom part of the tab is refreshed to display the controls for the data properties for that layer.

The check boxes beside the layers control the display of data in the 3D and 2D Views. When the check boxes are selected and the layers are highlighted, data associated with that layer are shown in the 3D Display. When the check boxes are cleared, the data is no longer displayed.

The following layers are listed in the Subset Editor tab:

- **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 information on the controls related to these layers, see the following sections:

  - "VIEW SUBSET INFORMATION" ON PAGE 126.
  - "COLOUR OPTIONS" ON PAGE 127
  - "SUBSET 2D VIEW" ON PAGE 128
  - "SUBSET 3D VIEW" ON PAGE 132

- **Reference Surface**: This layer controls the properties related to the display of BASE Surfaces in the 3D View of Subset Editor. For more information related to this layer, see the following section:

  - "SUBSETS AND BASE SURFACES" ON PAGE 326 in the HIPS and SIPS 7.1 Service Pack 2.

- **CUBE Hypothesis**: This layer controls the display and editing properties related to CUBE data. For more information related to this layer, see the following section:

  - HYPOTHESIS EDITING ON PAGE 241 OF THE HIPS AND SIPS USER GUIDE.

- **Tracking**: This layer controls the properties for subset tiles. For more information on the controls related to this layer, see the following section:

  - TRACKING CLEANING STATUS ON PAGE 325 OF THE HIPS AND SIPS USER GUIDE

## View Subset Information

Each time a subset is generated, the following information is shown in the middle section of the Subset Editor tab:
- total soundings in the current subset
- total memory used by subset
- total accepted soundings
- total rejected soundings
- width
- height
- minimum depth level
- maximum depth level

# Colour Options

You can control how soundings are displayed in the subset.

1. Make sure the Control window is open.

2. In the Properties > Colours section of the Subset Editor tab, open the drop-down list in the *Attribute* field.

3. Select one of the following attributes from the drop-down list to colour-code soundings:

   • **Depth**: The recorded depth level of the soundings.

   • **Amplitude**: Backscatter intensity values.

   • **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**: The sonar's wave pattern.

   • **Detection Method**: The phase detection method.

Colour Map

You can represent a range of values using a colour map. The colour ranges can be applied to other subsets so you have a standard 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.

4. 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.

5. [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.

# 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 123). As the slice box is re-sized or moved to another position in the subset, the display in the 2DView changes accordingly.

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

| | |
|---|---|
| **Menu** | Tools > Subset Editor > 2D View |
| **Tool** | **2D** |

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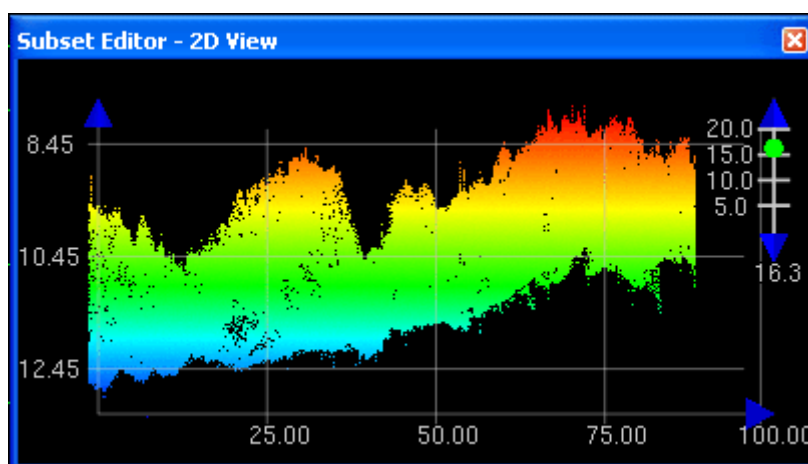


All controls for the 2D View are in Subset Editor tab.

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

The Properties for the 2D View are shown.

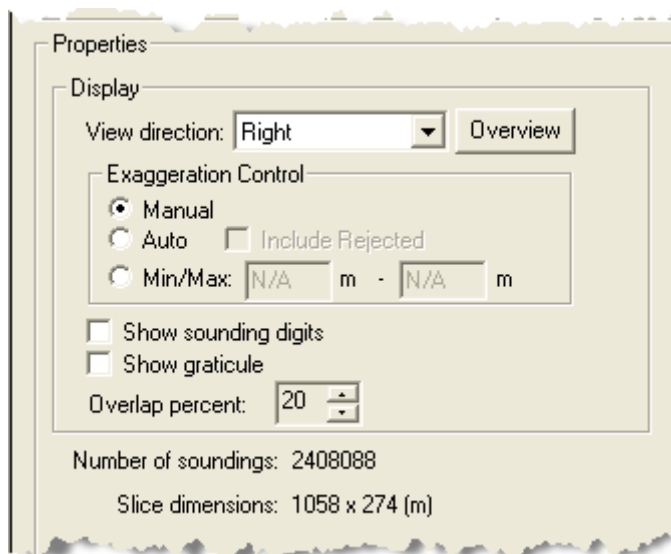| Change view perspective | When the 2D View is first displayed, the view perspective is determined by the setting in the *View direction* drop-down list. The setting options are: |
|---|---|

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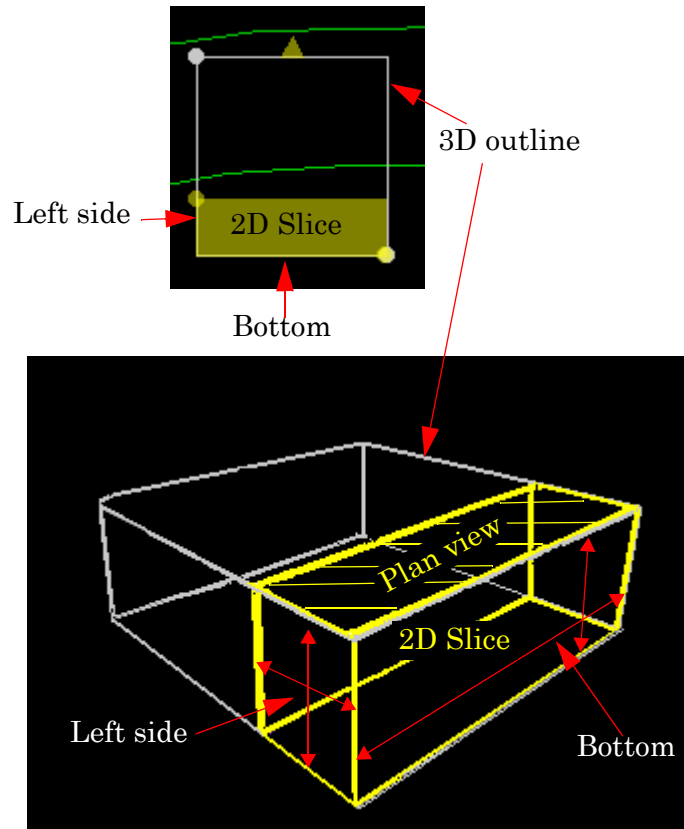
| | |
|---|---|
| 2D vertical exaggeration | The vertical exaggeration is the ratio of the vertical scale versus the horizontal scale. You can set the vertical exaggeration with the vertical exaggeration scale in the 2D View. |

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.

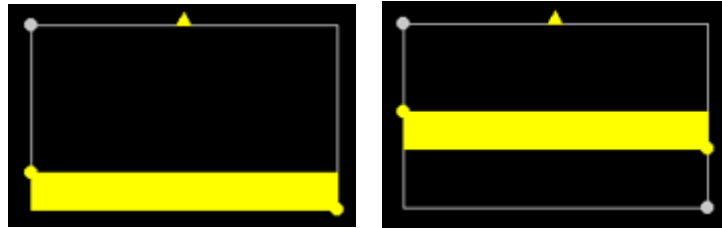| | |
|---|---|
| Properties | At the bottom of the list of options the following properties of the 2D slice is displayed: |

- 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 tab 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 166 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**.

| Menu | Tools > Subset Editor > 3D View |
|------|-------------------------------|
| Tool | **3D** |

# Subset 3D View

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 Editor tab.

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

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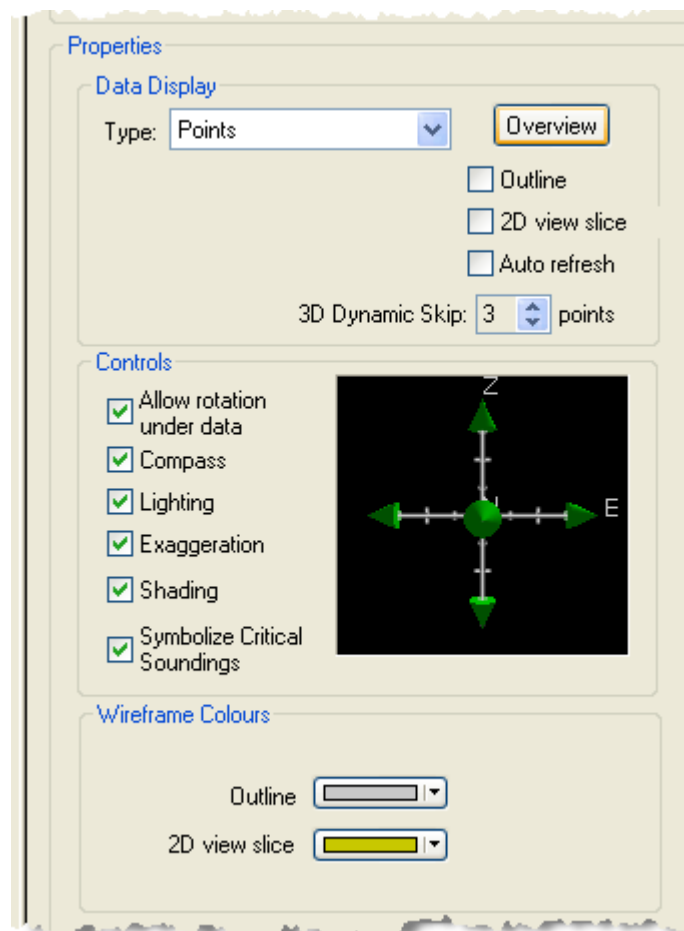


## 3-D Subset Display

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 3-D View is refreshed to show the soundings according to the selected display type. The options for each display type is explained in the following table.

| 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. |
| Surface Wireframe | The soundings are shown as a Triangulated Irregular Network (TIN) that connects 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.<br>2. If you selected the *Shoal Bin Size* option, type the size of the area for selecting the shoalest sounding.<br>3. Click **Regenerate**. |
| 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. |
| Spheres | Soundings are shown as unconnected spheres. | 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.<br>2. Click **Regenerate**. |
| 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.
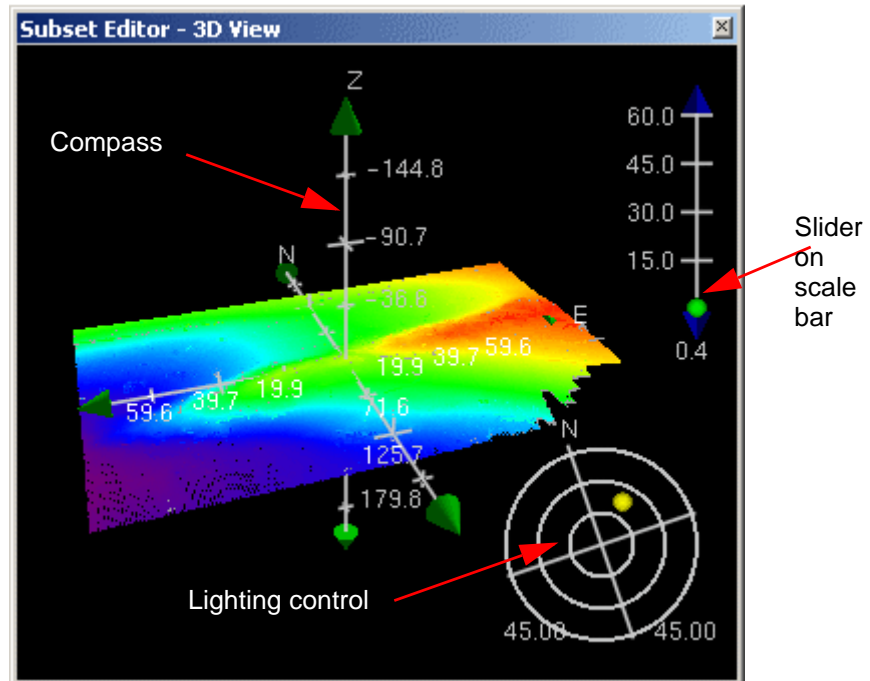
2. Select or clear the *Compass*, *Lighting* and *Exaggeration* check boxes to show or hide these controls in the 3D View.

| | |
|---|---|
| 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 129 for illustration.

| | |
|---|---|
| Allow rotation under data | This option rotates the 3D display by 180º to view soundings underneath the subset. When this option is not selected, the subset can only be rotated by 90º. |

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

| | |
|---|---|
| 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 166 for more information.

## Measure Distance

You can use the Measure Distance tool in the 2-D and 3-D Views to evaluate data problems, such as timing offsets, heave problems, etc.

1. Select the Measure Distance command.

A ruler icon is displayed next to the cursor.

| Menu | Tools > Measure Distance |
|------|--------------------------|
| Tool |  |

2. Press and hold the mouse button and drag the cursor along the 2-D View or 3-D View. To maintain a straight horizontal line, press and hold the <Shift> key while drawing the line. To maintain a straight vertical line, press and hold the <Ctrl> key while drawing the line.

As the cursor is dragged across the View, a line is drawn from the start point to the point where the mouse button is released. The distance between the two points is shown along the line.

# 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

SVP Editor helps you to edit existing SVP data, or to create new data.

### *In this chapter...*

# SVP Editor Interface
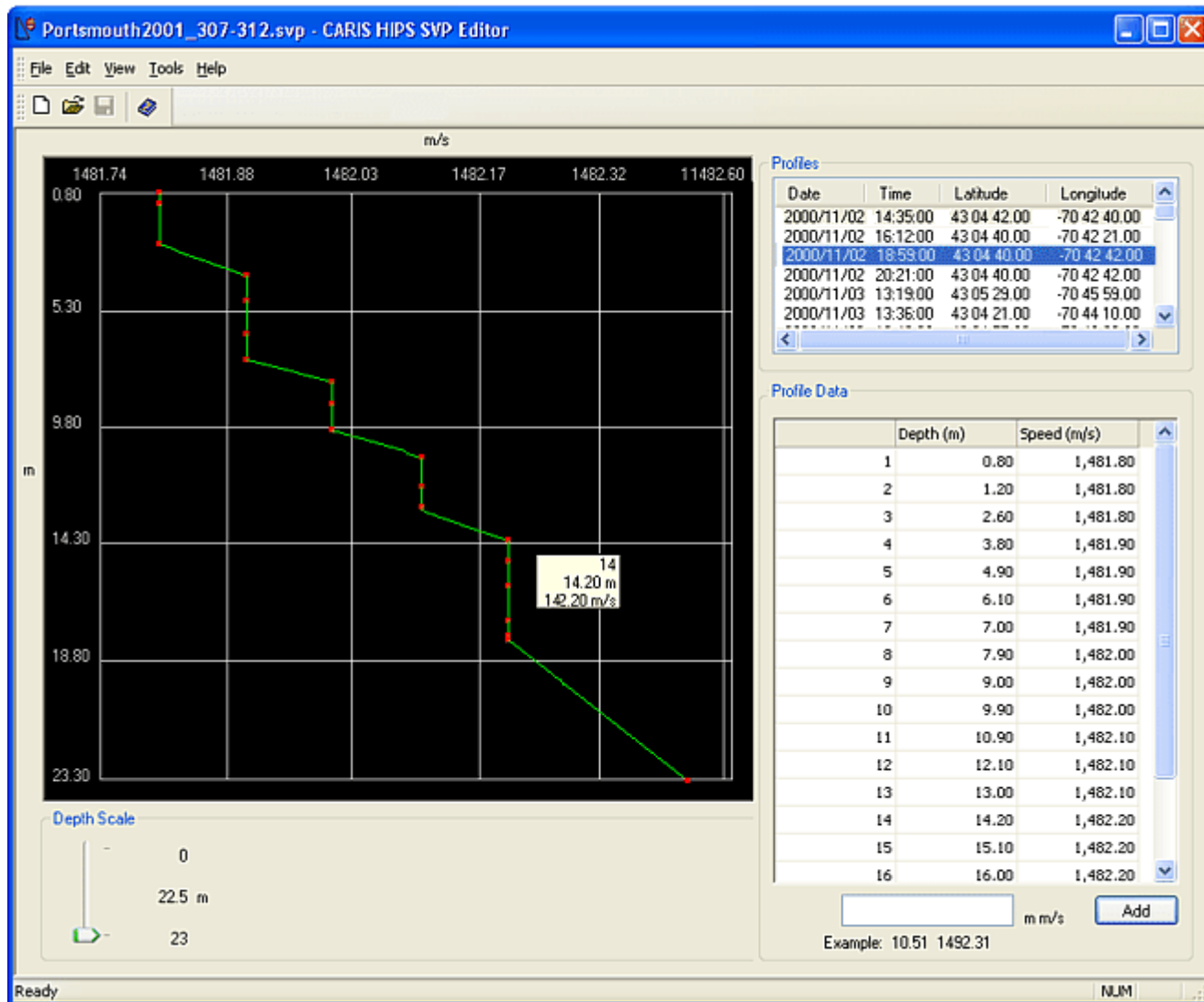
| | |
|---|---|
| **Menu** | Edit > Sound Velocity Profiles |
| **Tool** |  |

SVP Editor is a separate program that is launched from HIPS and SIPS.

To open SVP Editor:

1.  Select an SVP Editor command.

HIPS SVP Editor is displayed.



The *Profiles* area lists profiles contained in the SVP file. The profiles include a latitude and longitude position for applying the *Nearest in Distance* option. Below the *Profiles* section is a table showing the depth/speed values for the selected profile. The left-hand side of the editor contains a graphical representation of the depth/speed data for the selected profile.

The *Depth Scale* slider determines the view for the depth/speed graphed area by setting a maximum depth value. You can change the vertical display of data by dragging the slider.

| | |
|---|---|
| **Menu** | File > Open |
| **Tool** |  |
| **Key** | <Ctrl+O> |

# Open an Existing File

1. Select the Open File command.

The Open dialog box is displayed. All SVP files are located, by default, in ..\Hips\Svp.

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.
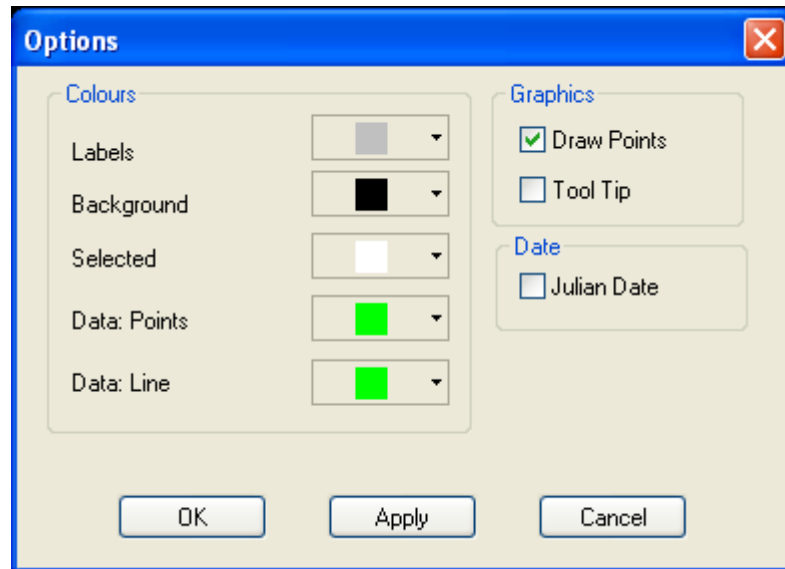
# Options
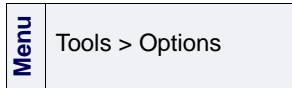
You can select the colours used for the graphical display.

1. Select the Options command.

<table>
<tr><td>**Menu**</td><td>Tools > Options</td></tr>
</table>

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

Sound Velocity Profile files are in text format and can contain more than one section. The first line of the file specifies the version of the SVP file (line "a" in the example below).

The second line displays the path and name of the 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")
- timestamp when the profile was recorded ("d")
- the latitude and longitude of the profile location ("e").

The SVP data is displayed in two columns, one containing the depth, and the other the speed value for that depth.

The following is an extract of the beginning of a sample SVP file:.

```
[SVP_VERSION_2] a
D:\HIPS\SVP\Fundy47_2006_307-309.svp b
Section 2006-307 14:35:00 43:04:42 -070:42:40
              c              d         e
 0.80 1481.80 f
 1.20 1481.80
 2.60 1481.80
 3.80 1481.90
 4.90 1481.90
 6.10 1481.90
 7.00 1481.90
 7.90 1482.00
 9.00 1482.00
```

```
Section 2006-308 13:19:00 43:05:29 -070:44:59
 0.90 1484.00
 2.90 1484.20
 3.90 1484.30
 4.90 1484.30
 6.00 1484.30
 7.00 1484.30
 8.00 1484.30
 9.00 1484.40
```

```
Section 2006-309 13:18:00 43:05:30 -070:46:55
 1.10 1485.20
```

# Create a New SVP File

| | |
|---|---|
| **Menu** | File > New |
| **Tool** |  |
| **Key** | <Ctrl+N> |

| | |
|---|---|
| **Menu** | File > Save As |

To create a new SVP file:

1. Select the New File command.

2. Add new SVP profiles (see "CREATE NEW PROFILE" ON PAGE 144.)

3. Create new depth/speed values, as needed.

4. Select the Save As command.

5. Type a name for the SVP file in the file name box.

6. Click **Save**.

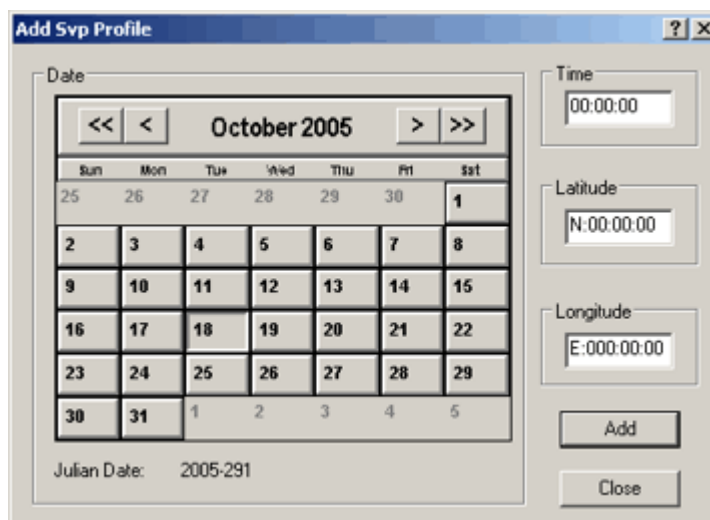The file is saved the directory you selected.

## Create New Profile

| | |
|---|---|
| **Menu** | Edit > Add Profile |
| **Key** | <Ctrl+Insert> |

New SVP profiles can be created through SVP Editor.

1. Select the Add Profile command.

The Add SVP Profile dialog box is displayed.



2. Click the arrow buttons at the top of the calendar to select a month and year (all profiles must be time stamped).

3. Click a day button on the calendar to select a day for the file.

4. Enter the new hour/minute values (use the 24-hour clock) by clicking each section in the *Time* box and typing new values.

5. Enter latitude and longitude coordinates in the appropriate boxes, if needed.

6. Click **Add** to apply settings.

The new file information is displayed in the *Profiles* box.

If you are enter new speed/depth data, you must enter this data in pairs. There are two options for entering data:

- separate the depth/speed data with a space: `15 1200`

- separate the data with a comma: `15, 1200`

## Enter new depth/speed data

7. Click a profile to display the data in the SVP editor.

8. Type new depth *and* speed values in the text box beside the **Add** button.

9. Click **Add**.

10. Repeat Steps 1 through 3 until all necessary values 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.

**Change depth/speed values**

Individual values in the depth/speed table can be changed.

1. Position the cursor over any *Depth* or *Speed* cell in the table.
2. Click inside the cell.
3. Type a new value.
4. Click outside the cell.

The table is changed to reflect the new data.

**Edit geographic coordinates**

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 241).

| Menu | Edit > Edit Position |
|------|----------------------|

1. Click a time stamp beside a profile.
2. Select the Edit Position command.

The Edit Position dialog box is displayed.
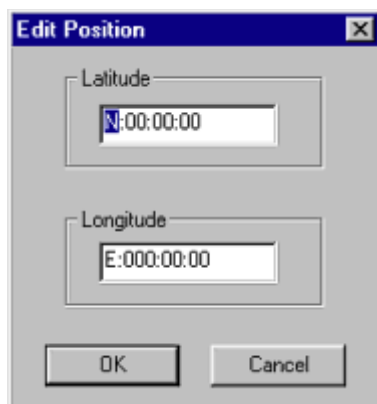


3. Click the latitude coordinates and type new ones.
4. Click the longitude coordinates and type new ones.
5. Click **OK**.

The new geographic coordinates are displayed in the *Latitude* and *Longitude* sections of the Profiles area.

**Delete speed/depth data**

1. Press and hold the mouse button, while dragging the cursor over the data points in the graphical display, or click a row number in the table.

The selected data is highlighted.

| Menu | Edit > Delete |
|------|---------------|
| Key | <Delete> |

2. 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 Profiles

| | |
|---|---|
| **Menu** | Edit > Remove Profile |
| **Key** | <Ctrl+Delete> |

Remove a currently open profile from the SVP file.

1.  Select a profile by clicking the time stamp in the *Profiles* list box.

The depth/speed values are displayed in the editor.

2.  Select the Remove Profile command.

The profile is removed.

## Exit SVP editor

Close the SVP Editor.

1.  Select the Exit command.

2.  If you have not saved any changes you are prompted to do so.

The SVP Editor is closed.

| | |
|---|---|
| **Menu** | File > Exit |

*9*

*Swath Editor*

Swath Editor provides data-cleaning tools for multibeam data, multi-transducer data 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 152).

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.
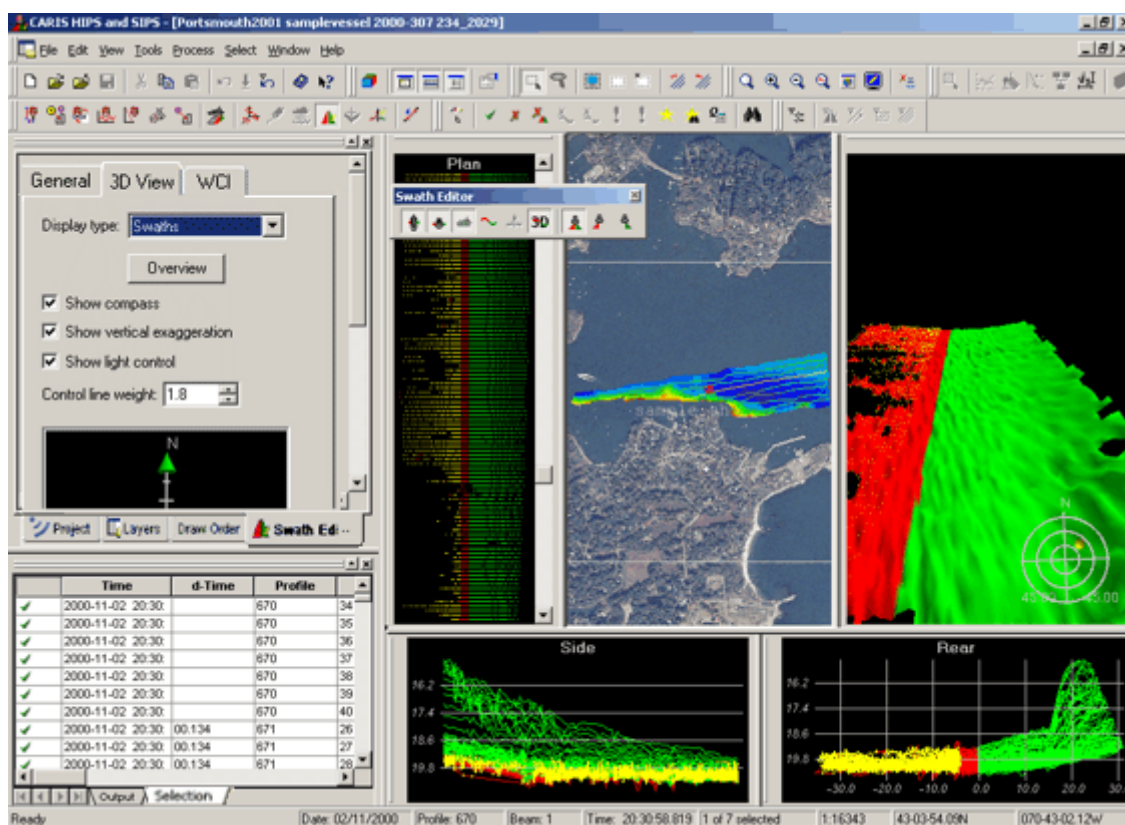


The main elements of the Swath Editor are:

• the Swath Editor tab in the Control window

• Swath Editor data view windows

• Swath Editor toolbar

## Swath Editor Controls

The Swath Editor tab has three tabbed pages:

• The General tab controls the data display in the the Swath Editor windows (see "SWATH EDITOR DISPLAY OPTIONS" ON PAGE 155).

- The 3D tab contains the Swath 3D view controls (see "3D VIEW CONTROLS" ON PAGE 162).

- The WCI tab controls display options for the water column imagery windows(see "WCI CONTROLS" ON PAGE 459).

> 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.
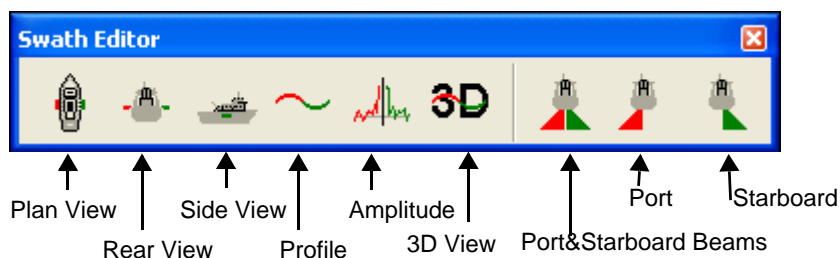


This is a floating toolbar that is automatically displayed when the editor is opened. The toolbar can be docked or floated over any area of the interface. Its appearance is controlled from the Toolbar dialog box (see "DISPLAY TOOLBARS" ON PAGE 33).

# Viewing Data

Swath Editor consists of six view windows. These windows open by default when the Swath Editor is opened:

- Plan View
- Rear View
- Side View
- Profile View
- Amplitude View
- 3D View
- 

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 "WCI IN SWATH EDITOR" ON PAGE 455

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.

## deTrend

If you are viewing data that lies up or down a slope you can use the deTrend command to "flatten" the view so that you can more easily see anomalies in the data. This is especially useful in the Rear and Side Views.

The deTrend function works by removing the average slope in the alongtrack direction.

To apply the deTrend command,

| Menu | Tools > Swath Editor > deTrend |
| --- | --- |

1. Select a track line.
2. Select the deTrend command

The windows which contain data which has been flattened this way are now labelled " deTrend".

## Plan View

The Plan View presents 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 154)

To help distinguish soundings by colour you can apply a number of options. Using the options on the General page of the Swath Editor tab in the Control window, you can:

- colour by depth interval (see "COLOUR BY DEPTH" ON PAGE 158)
- sun illumination (see "LIGHT CONTROL" ON PAGE 164).

Using using the Display options in the Tools > Options dialog box, you can

- 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)

| | |
|---|---|
| **Menu** | Tools > Swath Editor > Rear |
| **Tool** | |

## Rear View

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

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.

| | |
|---|---|
| **Menu** | Tools > Swath Editor > Side |
| **Tool** | |

1. Select the Side View command.

You can change the display (see "DATA DISPLAY IN VIEWS" ON PAGE 154). and depth label settings for this view (see "VIEW SOUNDINGS AS POINTS OR LINES" ON PAGE 157).

## 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

| Menu | Tools > Swath Editor > Amplitude |
|------|----------------------------------|
| Tool |  |

The Amplitude View displays the beam-averaged amplitude data that is recorded with the swath bathymetry data.

1. Select the Amplitude View command.

Beam-averaged amplitude data are intensity values for each sounding depth created by a multibeam sonar system.

## 3D View

| Menu | Tools > Swath Editor > 3D View |
|------|-------------------------------|
| Tool |  |

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 163).

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.
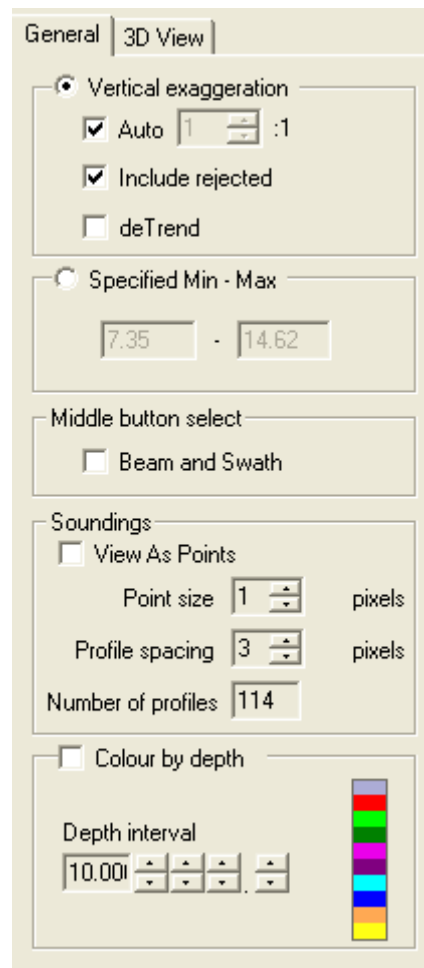
## General tab

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.

This deTrend option is used when the general slopes of the sea floor topography makes it difficult to see individual depth spikes. DeTrend removes the general slope of the sea floor from the Rear and Side views and redraws the swath and beam profiles making it easier to locate depth spikes.

4. Select the *deTrend* check box to view data. Select the command by selecting from Tools > Swath Editor > deTrend.

## 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 160 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 viewed as points, size 2 pixels

Soundings connected by lines
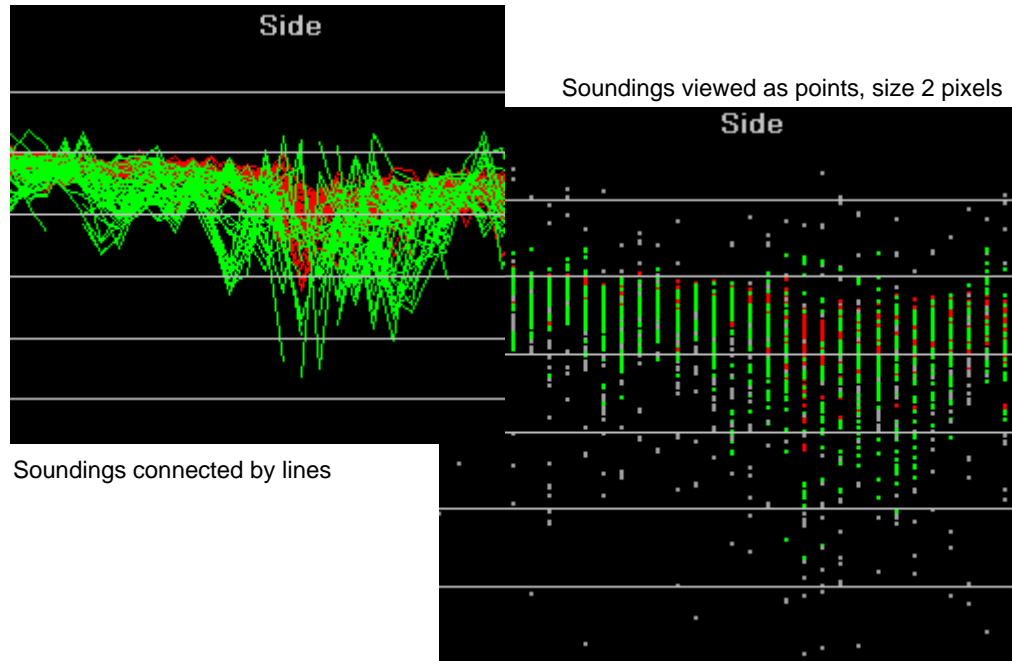
3.  Click the up or down buttons beside the *Point Size* box to set a size value between one and five pixels.

Changing the *Point Size* will also change the size of the points in the Plan view.
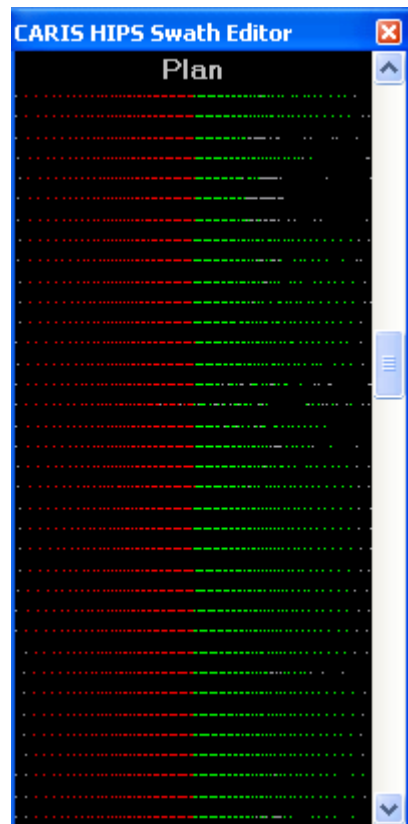
## Profile spacing

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.
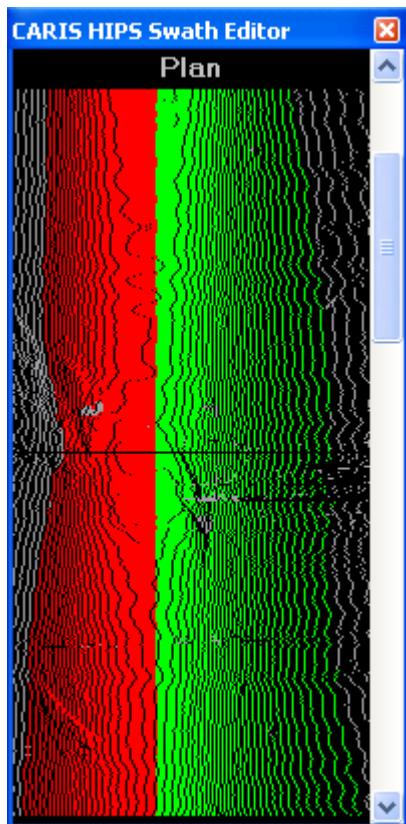
Profile spacing set to 1 pixel
256 profiles visible in this view

Profile spacing set to 10 pixels
38 profiles visible

## Colour By Depth

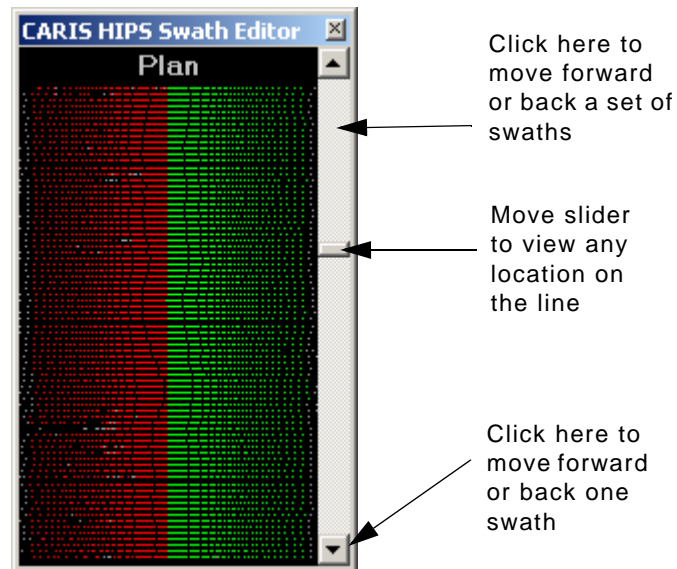Display soundings as colour coded by depth interval.

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 trackline 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.



Click here to move forward or back a set of swaths

Move slider to view any location on the line

Click here to move forward or back one swath

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 161).

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 lets 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).



MIddle mouse click here...

shows swath and beam in Display
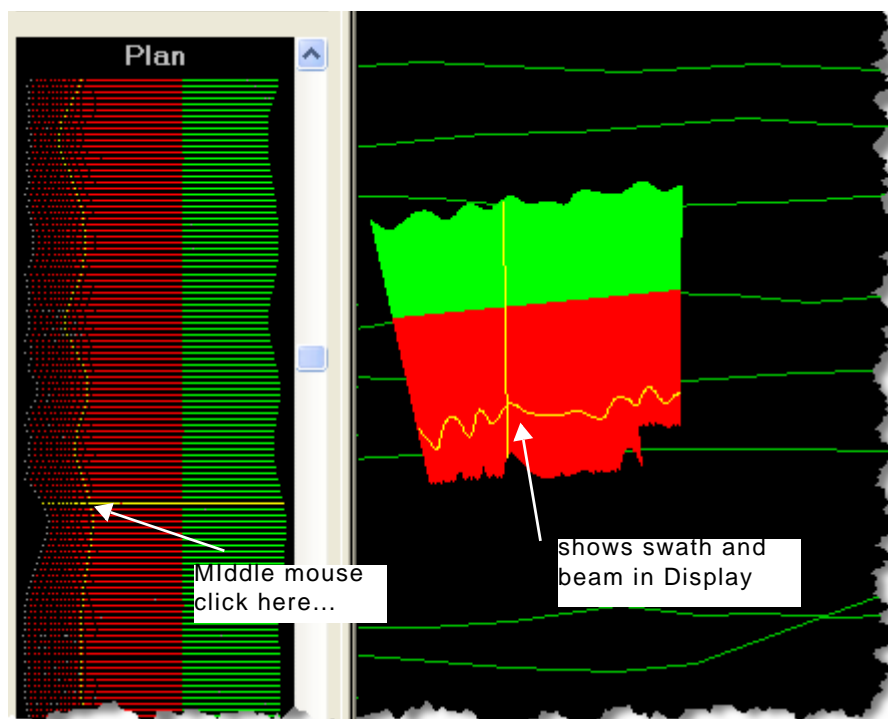
To view both beam and swath profiles

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 185).

# 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 90).
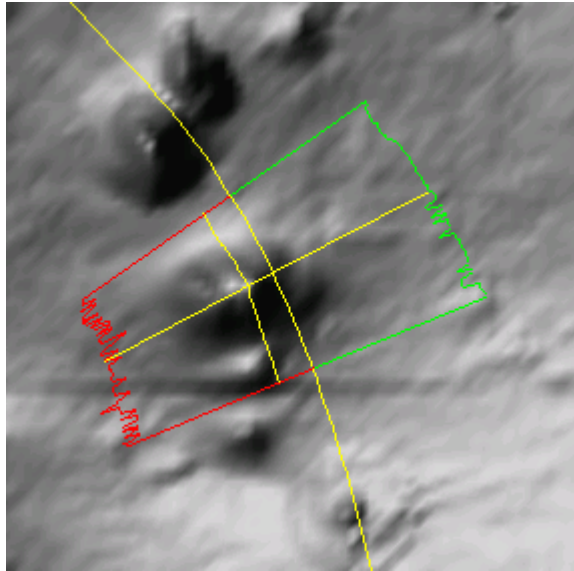


**Automatic Recentre**     The automatic recentre option ensures that when the coverage outline reaches the edge of the Display window (when scanning along the track line), the display redraws with the coverage outline in the centre. This option is available in the General Options dialog box. (see "GENERAL" ON PAGE 77).

# 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:

| Menu | Select > Next Line |
| --- | --- |
| Tool |  |

| Menu | Select > Previous Line |
| --- | --- |
| Tool |  |

- 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.

# 3D View Controls

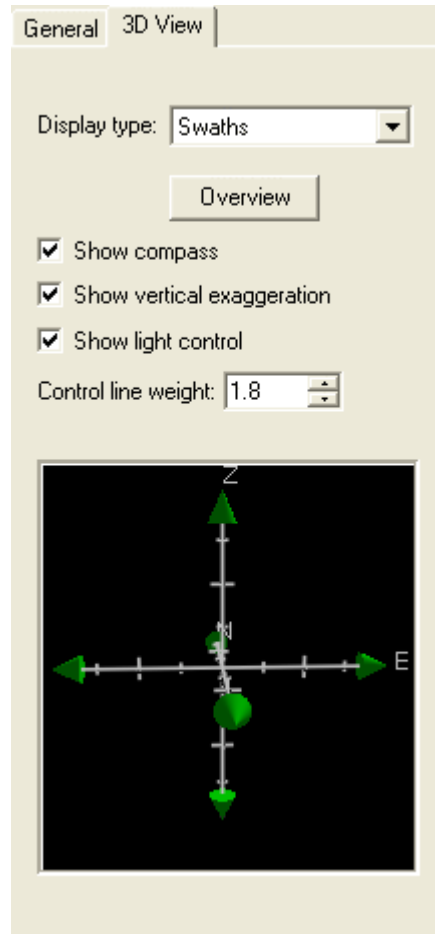| | |
|---|---|
| **Menu** | Tools > Swath Editor > 3D View |
| **Tool** | 3D |

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.

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).
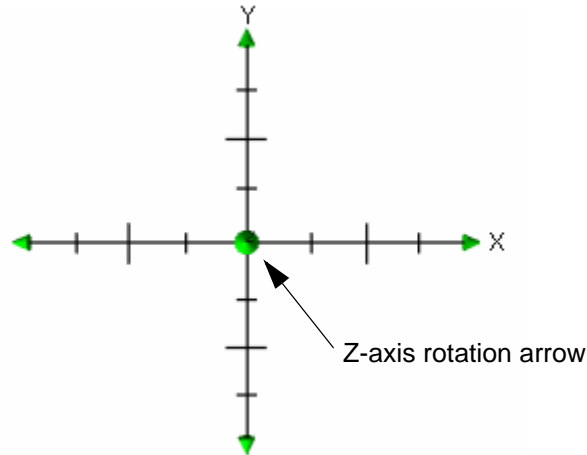
These controls are:

- 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.



Z-axis rotation arrow

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.
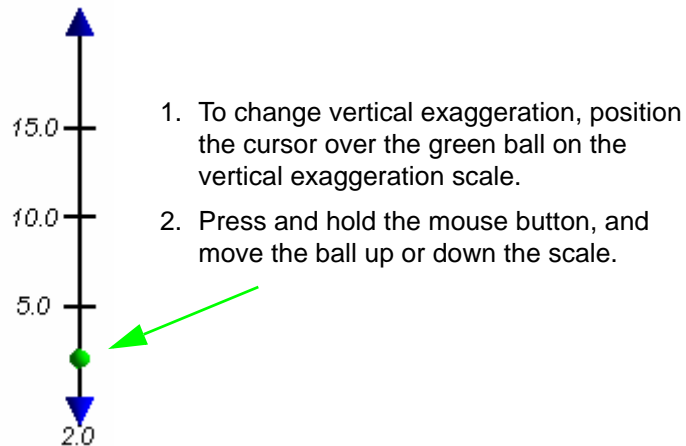
You can also change the vertical exaggeration using a dialog box.

3.  Double-click the ball in the vertical exaggeration slider.

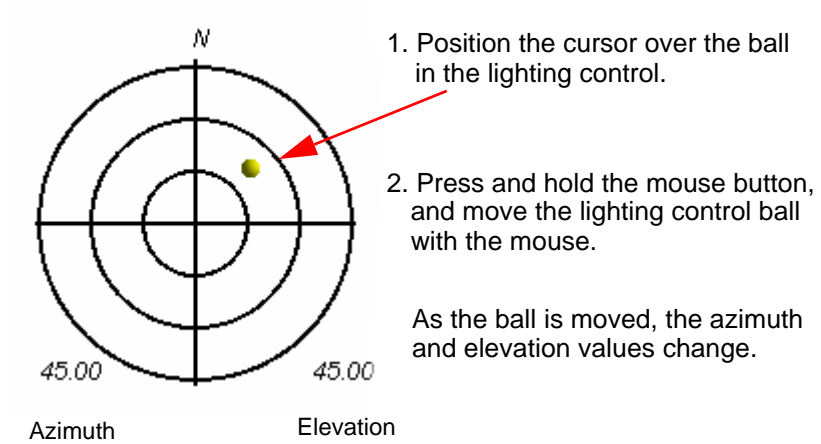The Vertical Exaggeration dialog box is displayed.

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*

Use Tide Editor to create or edit tide files. Tide data is used to generate final depths relative to the tide datum by subtracting the tide from the sounding depth.

### *In this chapter...*

# Open Tide Editor

Tide Editor presents tide file information in both graphical and tabular formats. 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.
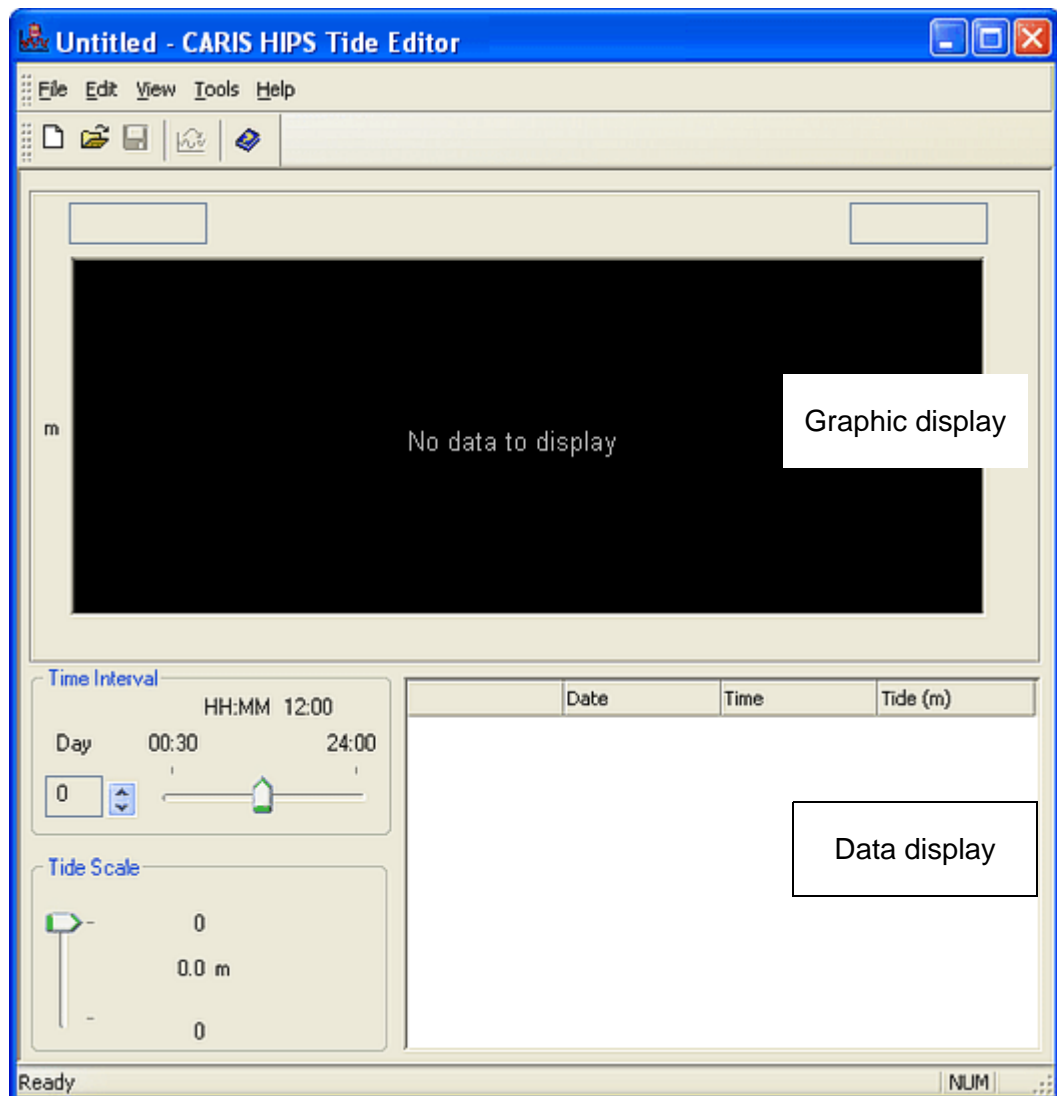
| Menu | Edit > Tide |
|------|-------------|
| Tool |  |

Tide files are located by default in …\Hips\*version*\Tide and have a .tid file extension.

To open Tide Editor:

1. Select the Tide Editor command.

The Tide Editor interface is displayed.

| | |
|---|---|
| **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**, or type the file name and path in the field.

The selected 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 scales can be adjusted using slider controls.
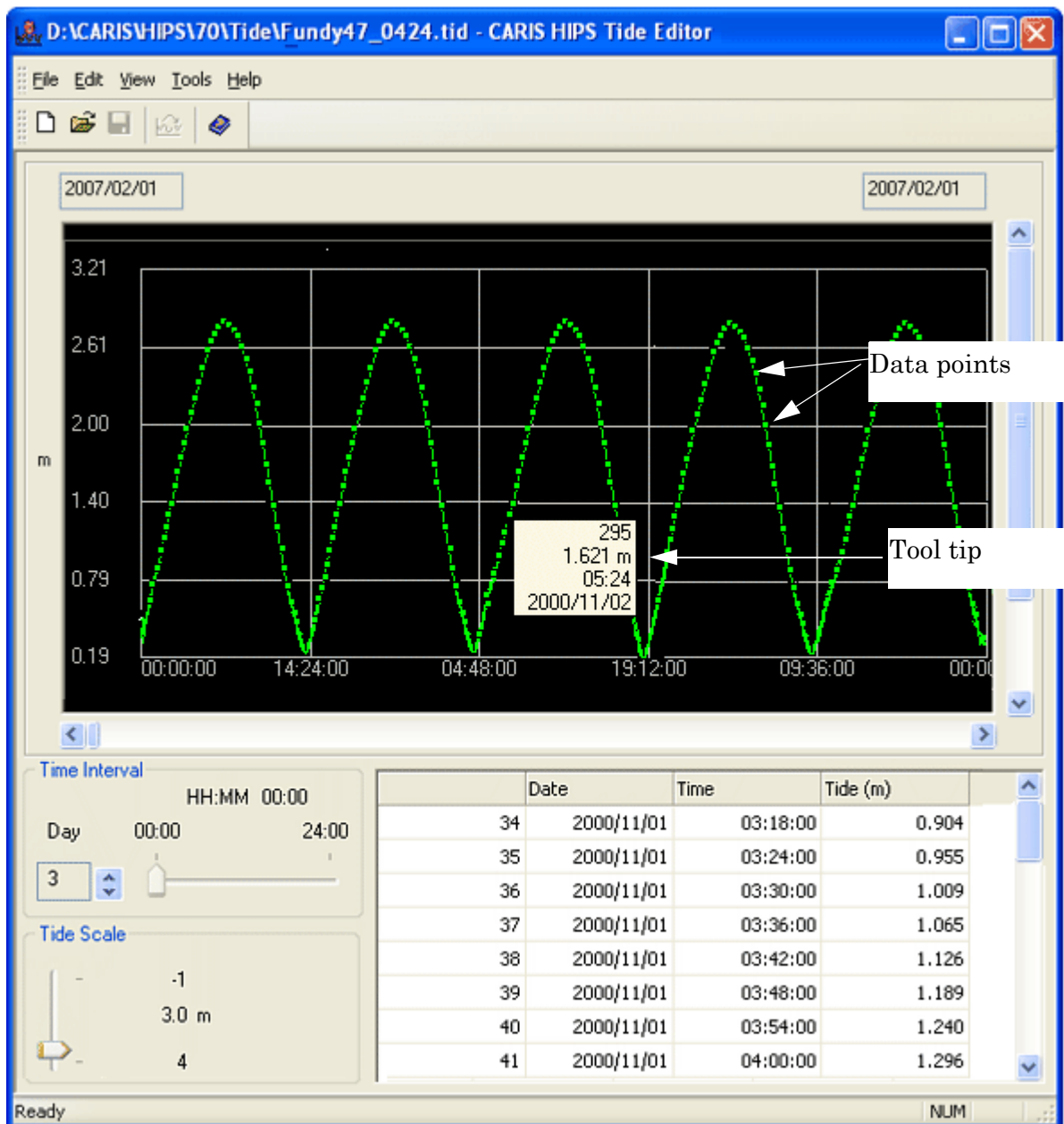
The tide data is displayed in a table below the graphical display.

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 lets you view data by 24-hour increments.

- The hour/minute option lets you 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.

For display options, see "DISPLAY OPTIONS" ON PAGE 174.

## Exit Tide Editor

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

Edit table data

Edit data in an existing tide file.

1. Select a Date/Time/Tide cell in the tide table.

The cursor is displayed in the cell.

2. Remove the existing data and type a new entry.

The graphical display refreshes to show the new data.

Delete a tide entry

1. Press and hold the mouse button while dragging the cursor over data in the graph, or click the appropriate number cell in the tide table.

The data is highlighted.

2. Select the Delete command.

| | |
|---|---|
| **Menu** | Edit > Delete |
| **Key** | <Delete> |

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 lets you 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. Select the General tab if it is not already displayed.

3. Enter a value (in minutes) in the *Time Gaps* box.

4. Click **OK**.

5. Select the Next Time Gap command.

| | |
|---|---|
| **Menu** | Tools > Next Time Gap |
| **Tool** | |
| **Key** | <Ctrl+G> |

When the interval (in minutes) between adjacent data points is the same as, or greater than, the *Time Gaps* value, the adjacent data points are highlighted.

6. Repeat the process, if necessary, by selecting the Next Time Gap command for each tide data point on the graphed line.

You can enter new data between two values.

Enter new data

Enter data for a new or existing tide file.

1. Select the Add Tide command.

The Add Tide dialog box is displayed.

| | |
|---|---|
| **Menu** | Edit > Add Tide |
| **Key** | <Insert> |

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 lets you 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> |

| | |
|---|---|
| **Menu** | File > Save/Save As |
| **Tool** | |
| **Key** | <Ctrl+S> |

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 171).

The data is displayed in graph format in the editor.

3.  Repeat until all values are entered.
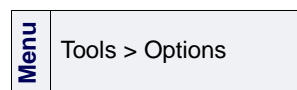
4. When ready, select the Save or Save As command.

The Save As dialog box is displayed.
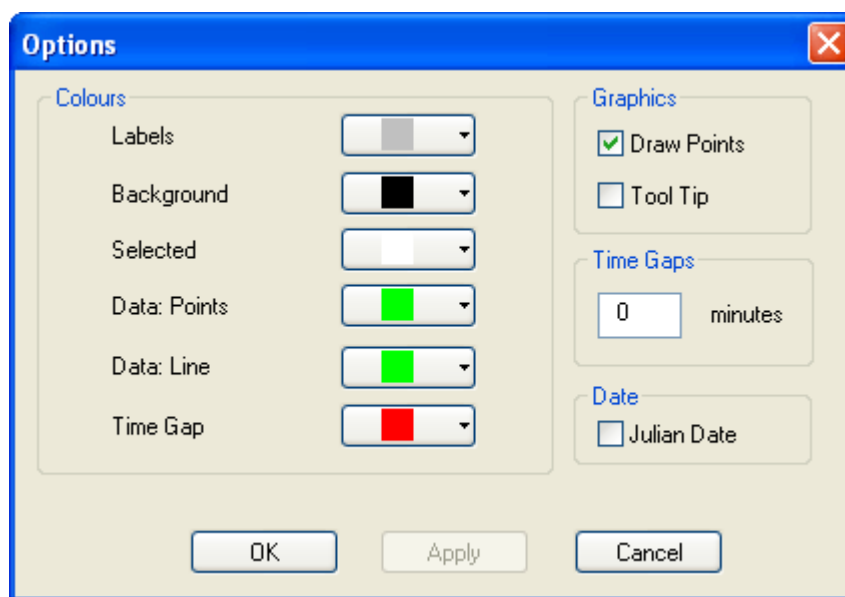
5. Type a name for the file and click **OK**.

# Display Options

You can adjust the display of graphical data: colours, display of data points and tools tips.

<table>
<tr><td>**Menu**</td><td>Tools > Options</td></tr>
</table>

1. Select the Options command.

The Options dialog box is displayed.



Colours | Set colours for display elements. Select a colour from the drop-down colour palettes, or create a custom colour from the Windows colour picker.

Click **OK** to to show the new colours.

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:

2. De-select 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 label on or off.

3. Select the *Tool Tip*s check box. If this option is checked then it is enabled.

Set date display | The date in tide file table is displayed by default in regular calendar format. To display in Julian Day format:

4. 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...*
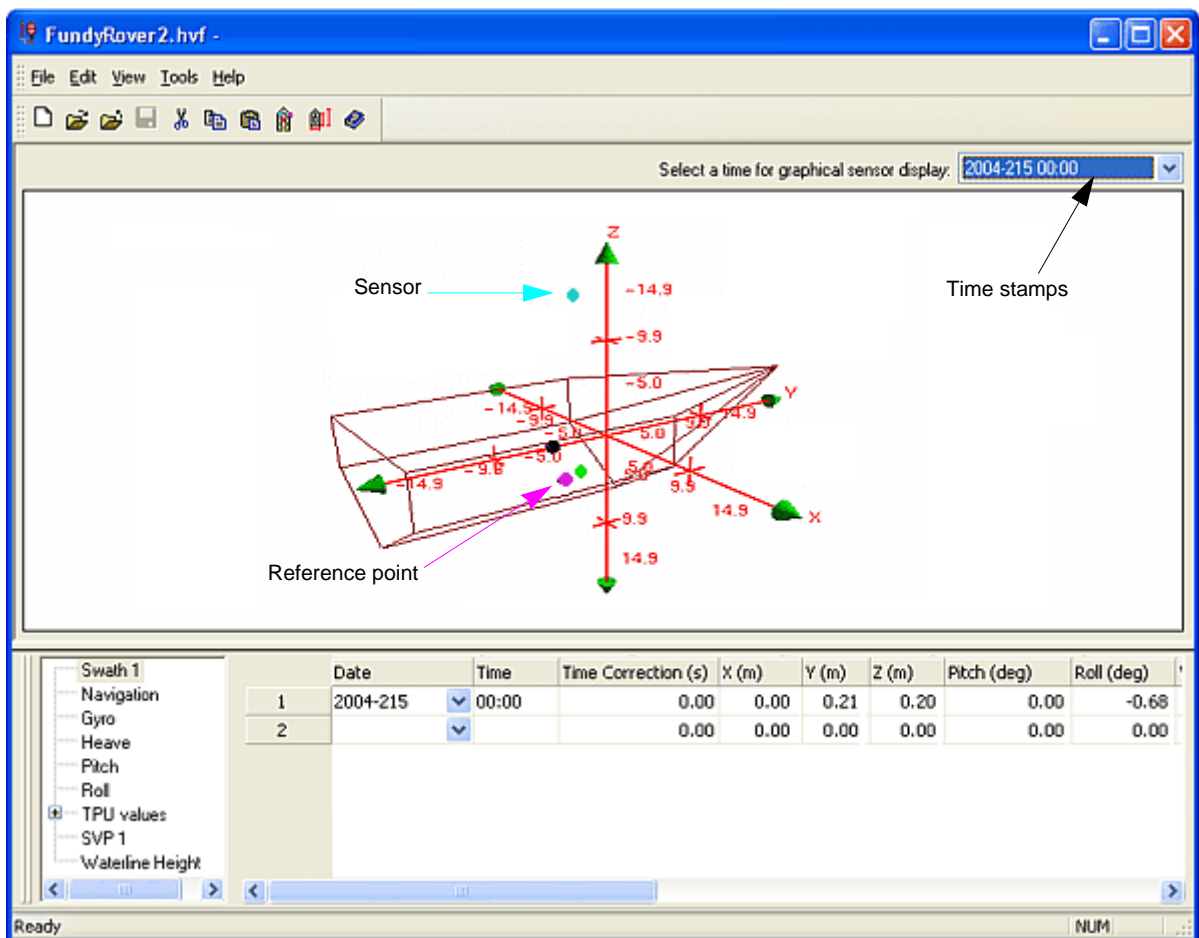
# Vessel Editor Interface

## Open Vessel Editor

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 180. The location of a sensor is represented by a coloured dot or line. See "COLOUR OPTIONS FOR SENSOR DISPLAY" ON PAGE 182 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)

| | |
|---|---|
| **Menu** | File > New |
| **Tool** |  |

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 27.

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 35.

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 38

## 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
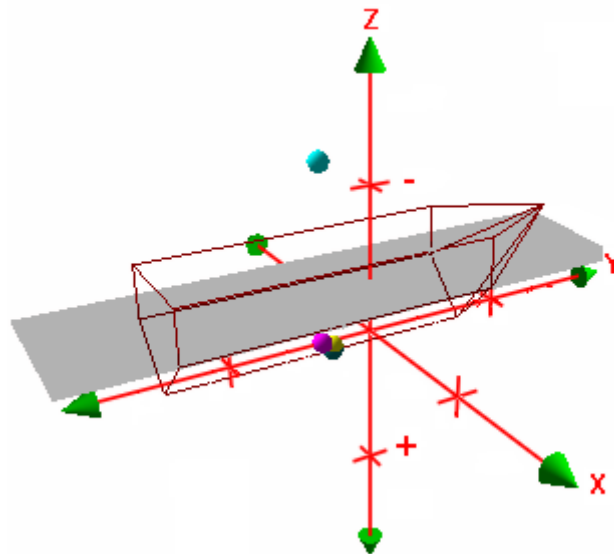
Close Vessel Editor and all files.

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 along the Z-axis, click the middle arrow head and drag the cursor upward so that the image is rotated. The compass can be rotated 90º.

2. To rotate the image 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. 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 <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 undocked 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.

1. Select the Customize command.

The Customize dialog box is displayed.

| **Menu** | View > Customize |

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.
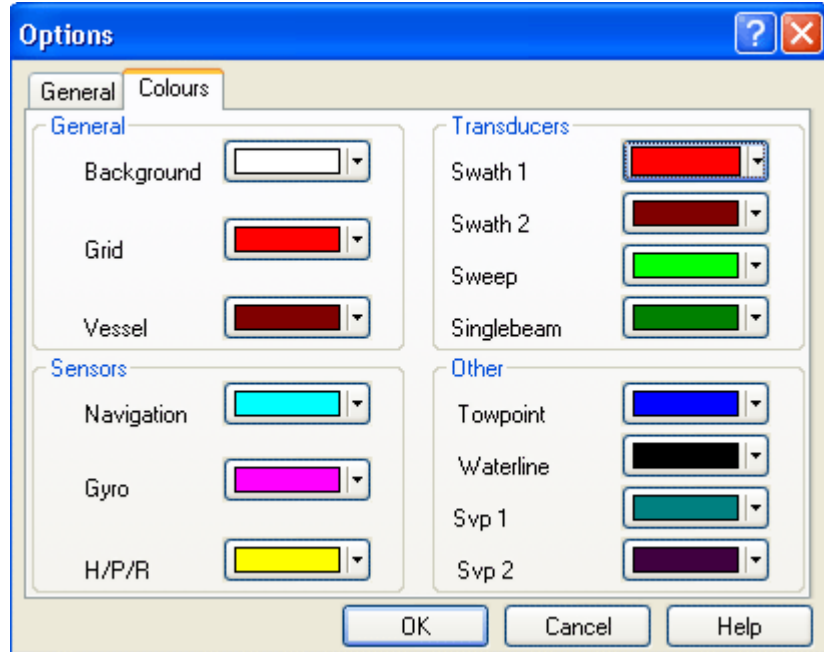
3. Click **OK**.
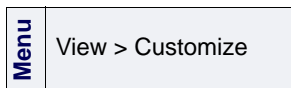
# Colour options for sensor display
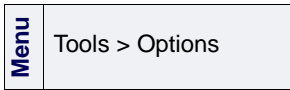
The locations of sensors and other equipment in the Vessel Shape 3D display are shown by colour markers. (You can set other colours for sensor markers.)

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.