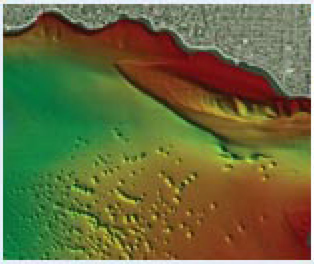
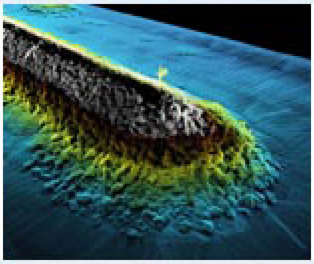
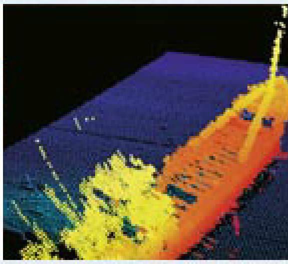
CARIS **HIPS and SIPS 9.1**

User Guide







THE MARINE GIS EXPERTS

Acknowledgements

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

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

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

Cover images (from left to right):

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

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

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

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

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

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

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

CARIS Asia Pacific
PO Box 1580
Milton QLD 4064
AUSTRALIA

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

June 2016

Trademarks owned by CARIS

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

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

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

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

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

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

Copyright owned by CARIS

Preface

This guide describes the CARIS[1] HIPS[2] and SIPS[3] workflow for processing data, using the multibeam to CUBE surface flow.

For single beam data workflow, please see “Process Single Beam Data” on page 795 in the Editors Guide.

For processing of LIDAR data, please see “Process LIDAR Data” on page 733 of the Editors Guide.

Please be advised that some 64-bit versions of third-party libraries remain unavailable at this time. Therefore, the following capabilities will not be available in the 64-bit version of HIPS and SIPS 9.1:

Navitronics format conversion

Hawkeye waveform viewer

If the above capabilities are required, you will need to use the 32-bit version of HIPS and SIPS.

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

Table of Contents

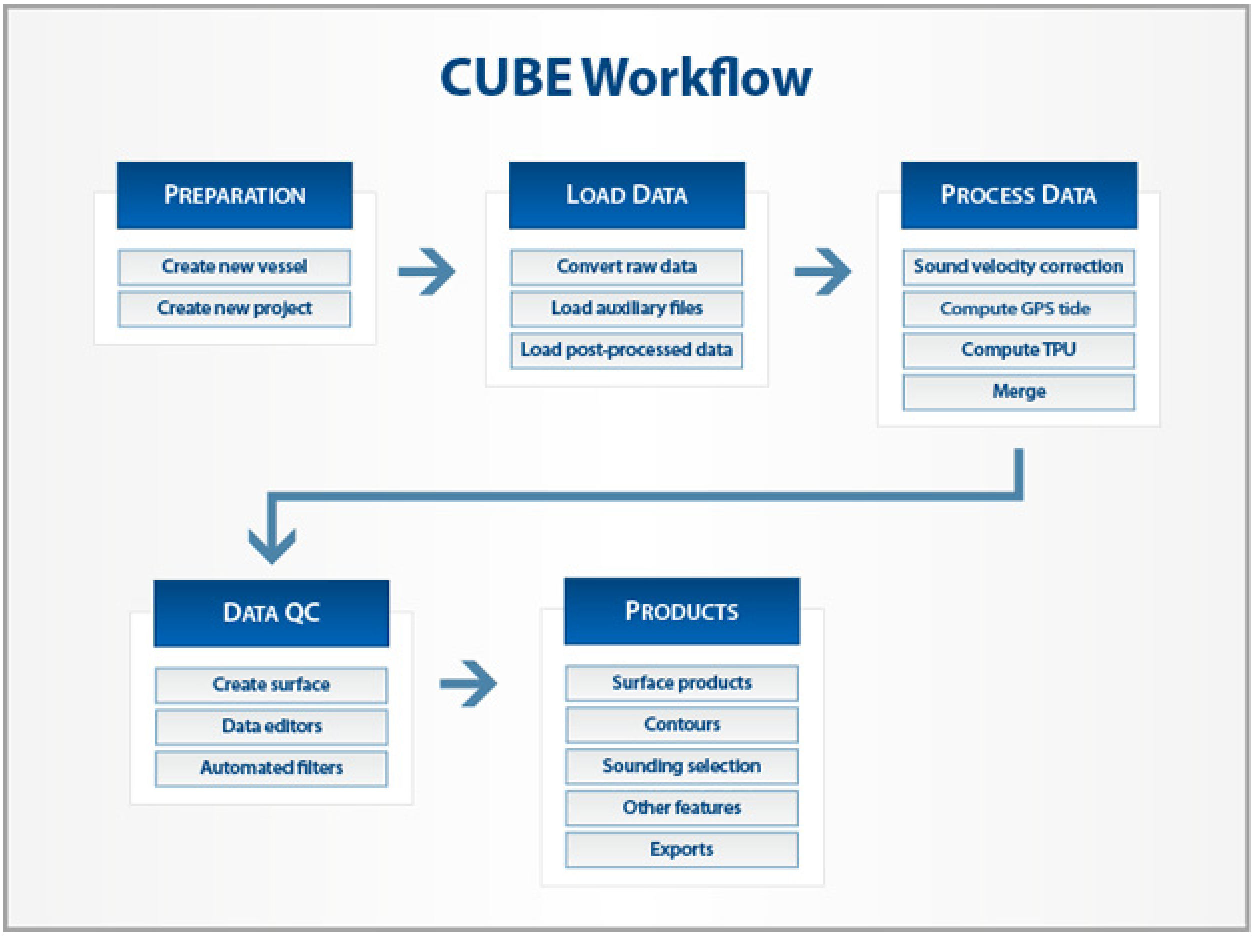
1

Workflow in HIPS and SIPS

The workflow described here uses multibeam data and CUBE to illustrate a way of processing data in HIPS and SIPS. This workflow will take you through the steps from raw data to the creation of a contoured product surface and the publishing or export of cleaned data.

In this chapter...

Workflow diagram: Multibeam



"Overview of workflow stages" on page 21

Other workflows are described in the HIPS and SIPS Editors guide. See:

- "Side Scan Editor workflow" on page 66
- "Process Single Beam Data" on page 795
- "LIDAR Data in HIPS" on page 734

In this user guide, see also:

- "Process Imagery Data" on page 409 (backscatter and side scan imagery workflow)
- "Processing Workflow for Water Column Data" on page 489

Since some data does not require all processing steps and some data can require repetition of one or more steps to produce a final product, this general workflow can be adapted to specific situations and data types.

The order in which certain functions are performed is important, however, many of the functions can be automated with the Batch Processor (see "Batch Processor" on page 87).

Overview of workflow stages

- Create a Vessel File: Create or edit vessel information on sensor locations and uncertainties. See "Create a New HVF" on page 31
- Create a new project: Set up the Project - Vessel - Day data structure. See "New Project" on page 63
- Convert raw data: Convert data into HIPS data format using automated process. See "Convert Data" on page 79
- Load auxiliary files and post-processed data.
- SVC: Load and edit sound velocity profiles and apply the correction. [Optional] See "Sound Velocity Correction" on page 160
- Compute GPS tide. See "Compute GPS Tide" on page 176
- Compute Total Propagated Uncertainty (TPU): Use uncertainty values entered in the HVF to compute the total propagated error of each individual sounding. See "Sources of Uncertainty Data" on page 183
- Merge: Combine vertical and horizontal information to produce geo-referenced data. If changes are later made to the navigation and/or motion data during the QC process, the Merge process is re-applied.
- Create a surface: Use merged data to produce various types of surface. See "Create a New Surface" on page 207
- Data editing: Examine and edit sensor data, such as navigation, gyro, heave, if problems have been identified in the surface.
- Filtering: Use automated or manual tools to filter soundings using swath geometry and/or according to IHO survey order accuracies. Filter water column data for editing in Subset Editor.
- Process Data in Subsets: Using hypothesis editing, validate the CUBE surface and edit geo-referenced soundings. Add bathymetry from edited water column data.

- Finalize surface: Update the validated surface after data has been edited, and ensure designated soundings are carried through to bathymetric products.
- Create Product surfaces: Produce a generalized product surface from a surface, and add:
    - Contours: Use automated functions with a surface to output contours.
    - Sounding Selection: Use a height source for selection of a representative sounding set.
- [Optional] Process Imagery Data: backscatter or side scan data can be processed in mosaics.
- Export Data: Selected soundings, surfaces and mosaics can be exported to various formats.

HIPS Vessel Files

All HIPS and SIPS projects must contain a HIPS Vessel File (HVF). The HVF describes the installation and calibration of equipment installed on the survey vessel. The information in the HVF is used in multiple processes including merging, mosaicking, and calculating sounding uncertainty.

The HVF can be created or modified through the HIPS Vessel Editor. The Vessel Editor is a separate application that is launched from the HIPS interface. See "Create a New HVF" on page 31.

> A vessel file must exist prior to creating a new project in HIPS and SIPS.

Working with Projects

Projects in HIPS and SIPS are organized into the following hierarchy:

`Project / Vessel / Day / Line`

The new project creation process generates a HIPS Project File (*.hips). Existing projects that were saved as *.hpf files, are converted to *.hips when you first open them in HIPS and SIPS.

Vessel and /Day layers can be added to an existing project. Day and Line layers can be renamed, taken out of the active project or deleted outright. See "PROJECTS IN HIPS AND SIPS" ON PAGE 62.

Projects can also be archived. See "ARCHIVE PROJECT" ON PAGE 77.

All projects must contain a vessel file. (See "CREATE A NEW HVF" ON PAGE 31). If your raw data files are set up in a Project/Vessel/Day/Line structure and a HIPS Vessel File already exists for that data, you can use the Conversion wizard to create the HIPS directory structure. (See "CONVERTING FILES TO HIPS FORMAT" ON PAGE 80 and "RENAME DAY AND LINE FOLDERS" ON PAGE 76 for more information.)

You can be connected to more than one project repository at one time, using the commands from the right-click menu in either the New Project or Open Project dialog box.

Session Files

Session files record the list of data layers open at the time the session was saved. The Session file also records the layer properties, drawing order, the on and off state of layers, and the last geographic extent of the data.

Session files enable you to re-open all data that was being processed the last time the project was open. The data types that are recorded in the session file include:

- lines
- background data
- surfaces
- last geographic view extent of the Display window

All session files have an .hsf file extension and are stored by default in …`\Hips\Session`. The previous format of the session file (.ses) can be opened and automatically upgraded to the new session file format.

Save a session

Menu	File > Save Session/ Save Session As

To save a session

1. Select the Save Session command, or select the Save Session As command if the session has not been saved before, or if you want to save an existing session under a new name.
   - If this is an already saved file, the program re-saves the existing .hsf file.
   - If this is a new file, then the Save As dialog box is displayed. Select a folder where you want to save the .hsf file and type a name in the File Name text box.

If a project contains a large number of track lines, you can make data processing more manageable and reduce the time it takes to load data into the application by selecting specific track lines for a session file.

1. Select the track lines you want to save in the session file.
2. Choose the Save Session As command and select the *Save Selected Lines Only* check box.
3. Type a name for the session file and click **Save**.

The Session file is saved.

Open a session

Menu	File > Open Session

Open one or more saved sessions.

1. Select the Open Session command.

The Open Session File dialog box is displayed.

2. Select a session file.
3. Click **Open**.

The data layers contained in the Session file are opened and listed in the Layers window.

Close a session

Menu	File > Close Session

1. Select the Close Session command.

If you have made changes to the session since the last save, you are prompted to save the current changes.

2. Click **OK**.

Convert Survey Data

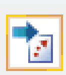
HIPS and SIPS files are created from survey data using the Conversion Wizard. The Conversion Wizard is a separate application that is launched from HIPS and SIPS .

Survey data converted to HIPS and SIPS format is in varying stages of completion, depending on the data format options. Data may or may not have been corrected for factors like heave/pitch/roll or sound velocity.

All HIPS/SIPS data is organized in a Project/Vessel/Day/Line directory structure. If you organize the survey data according to this structure, you can convert line data from an entire Project or multiple Vessel and Day folders contained in a common Project folder.

When entire project data has been converted, a Project/Vessel/Day/Line directory is created for the converted data. If there is already a Project folder that is identical to the preprocess folder, then the line data in the existing folder is over-written with the new data.

> Raw data files must be organized into the HIPS and SIPS Project/Vessel/Day structure in order to convert complete Project, Vessel and/or Day data files.

Menu	Import > Conversion Wizard
Tool	

1. Select a Conversion Wizard command.

For more information on how to use the Conversion Wizard, see “Converting Files to HIPS Format” on page 80.

Generic Data Parser

HIPS can convert almost any single beam ASCII file through the Generic Data Parser. The parser can also add or replace any sensor data in an existing project except swath and sweep bathymetry, and side scan imagery.

For example, if high-precision positions were obtained from a source other than the original raw data files converted into HIPS, this data could be loaded into an existing project to replace the original navigation data. The new position data must have a time stamp and be in ASCII format.

The Generic Data Parser is a separate program launched from HIPS and SIPS.

Menu	Import > Generic Data Parser
Tool	

1. Select the Generic Data Parser command.

The Generic Data Parser is displayed.

"Generic Data Parser" on page 93. For more information see Generic Data Parser, in the Tools guide.

2

Create Vessel File

The HIPS Vessel file (HVF) defines the offset configurations and associated error estimates for each of the sensors. These are necessary for creating final position and depth records for survey data.

The HIPS and SIPS workflow begins with creating an HVF, if necessary, editing the sensor configuration data.

In this chapter...

Vessel Files

The vessel file (HVF) in HIPS and SIPS describes the location and calibration of sensor equipment installed on the survey vessel. It defines the offsets and any associated error estimates for each of the sensors.

For a description of the coordinate system upon which the vessel configuration is based, see "Vessel Coordinate System" on page 58.

The HVF is created in the HIPS Vessel Editor, and saved to the folder …`\HDCS_Data\VesselConfig`.

If you create other data directories, the name of the `VesselConfig` folder must not be changed and it always has to be located inside the Raw Data directory on the same level as the project directories.

> In previous versions of HIPS and SIPS, a text file called the Vessel Configuration File (VCF) was used. VCF files can still be opened in the Vessel Editor and edited, but when the file is re-saved, it is converted to HVF format.

The HVF is divided into a number of distinct sections, each describing one type of sensor. The sections are time-tagged and multiple entries can be defined for different time periods. These entries can be edited in the Vessel Editor.

During the Merge process, corrected sensor data is combined with the observed data to create a final position/depth record.

Create a New HVF

A vessel file is created in HIPS Vessel Editor, a separate application launched from within HIPS and SIPS. For a description of the Vessel Editor and its properties, see “Vessel Editor Interface” on page 870.

The main steps to creating and configuring a HIPS Vessel file in Vessel Editor are:

- Open the HIPS and SIPS Vessel wizard to enter the parameters for the vessel file, based on the type of data being surveyed. (See “New file” on page 31.)
- Enter sensor position data using a 3D outline for the vessel. (See “Create Vessel Shape Outline” on page 39.)
- Enter additional information such as the ellipsoid used in the survey. (See “Default Ellipsoid” on page 60.)
- Enter and edit sensor configuration data. (See “Sensor Configuration” on page 42.)

New file

Menu	Tools > Editors > Vessel
Tool	

To open the Vessel Editor:

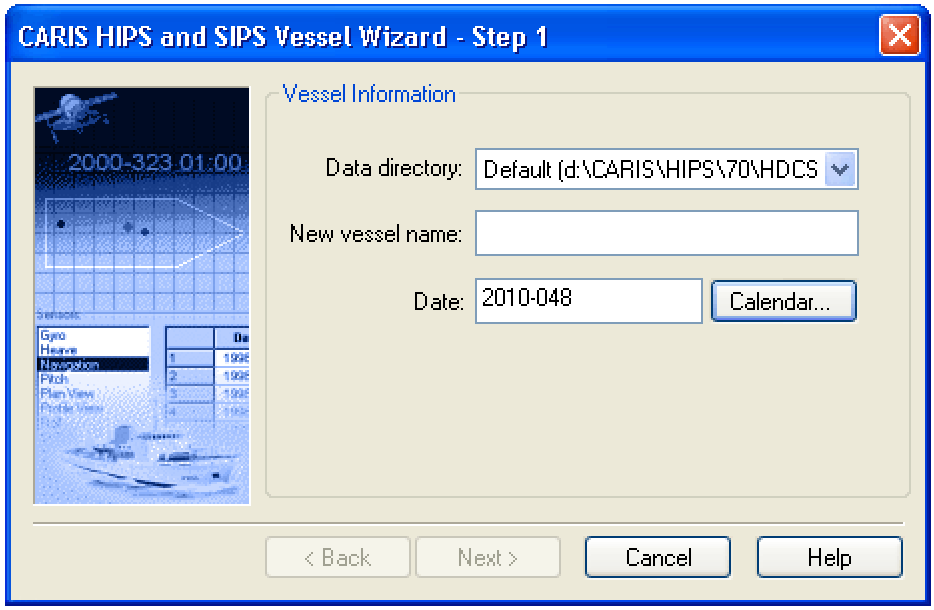
1. Click the Vessel Editor button on the Tools toolbar or select the command from the Tools menu.

In Vessel Editor:

Menu	File > New
Tool	

2. Select the New Vessel File command.

The Vessel Wizard - Step 1 (Vessel Information) dialog box is displayed.



The HVF will be saved to the `\VesselConfig` folder in your data directory.

1. Select the directory holding your data, from the drop-down list.
2. Type a name to identify the vessel used in the survey.

The name must not include any spaces.

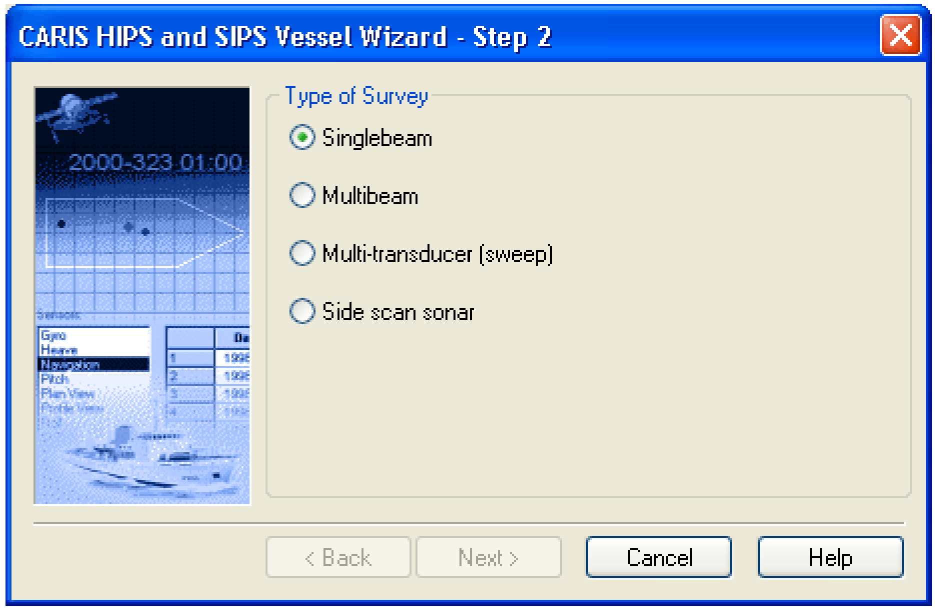
3. Click the **Calendar** button to insert the year and day the survey began.

The Vessel Date must be before the start of your data collection.

4. Click **Next.**

Type of Survey

The Step 2 dialog box prompts you to enter the sonar type used in the survey.



1. Select a sonar type by clicking the appropriate check box.

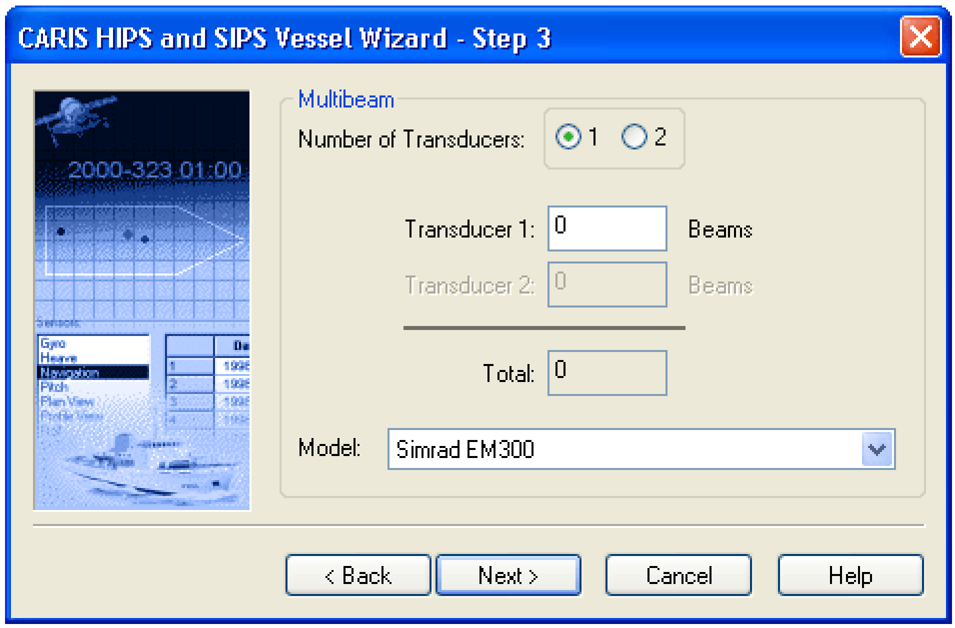
The next step will depend on your choice of survey type

If you selected *Side Scan Sonar*, click **Finish**.

2. If you selected another survey type, click **Next**.
   - If you selected Singlebeam, go to: "Motion Sensors" on page 36
   - If you selected Multibeam, go to: "Multibeam" on page 34
   - If you selected Multi-transducer, go to: "Multi-Transducer" on page 35

Multibeam

If you selected Multibeam in the Step 2 dialog box, Step 3 will ask you for information on the sonar model and number of transducers.

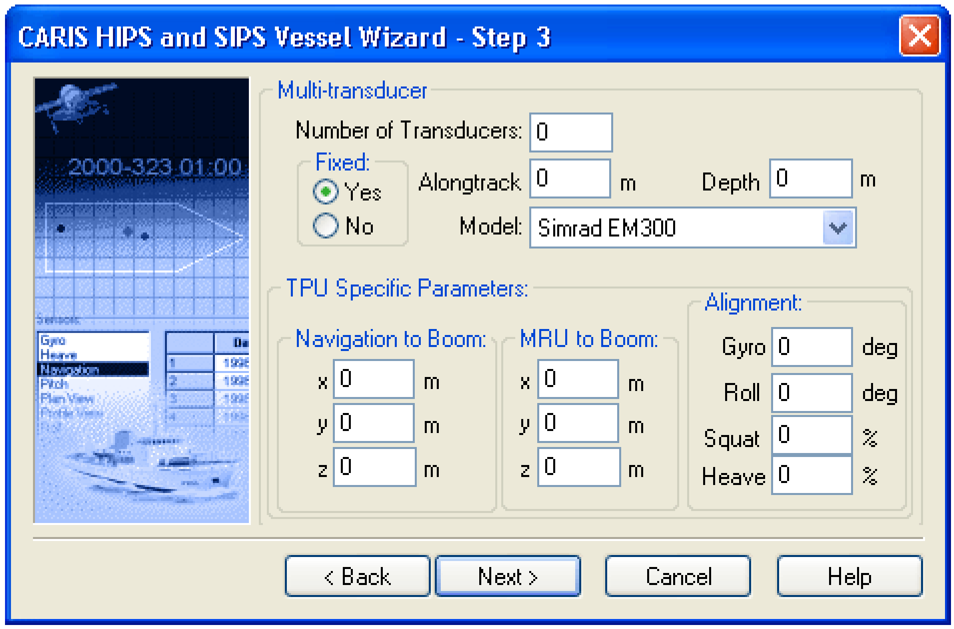


1. Select the appropriate check box to indicate that one or two transducers were used in the survey.
2. Enter the number of beams in each transducer.
3. Select the sonar model from the drop-down list.
4. Click **Next.**

Continue with “Motion Sensors” on page 36.

Multi-Transducer

If you selected Multi-transducer in the Step 2 dialog box, you now enter information on the number and types of transducers and for TPU specific parameters.



1. Enter the *Number of Transducers* on a boom.
2. Select *Yes* if the transducers are mounted to a fixed boom or to the hull of the vessel, or select *No* if the transducers are on a movable (decoupled) boom.
3. Type the default *Alongtrack* value of all the transducers relative to the ship's reference point.
4. Type the default *Depth* of all the transducers relative to the ship's Reference Point.
5. Select the sonar model from the *Model* drop-down list.

Parameters for navigation to boom and motion recording unit (MRU) to boom are necessary to calculate Total Propagated Uncertainty.

6. Type the distances from the positioning system to the centre of the boom in the *Navigation to Boom* X-Y-Z fields.
7. Type the distance from the motion recording unit to the centre of the boom in the *MRU to Boom* X-Y-Z fields.

The *Roll* and *Gyro* misalignment values are needed for fixed-boom system. However, for a decoupled boom, only a *Gyro* misalignment value is needed.

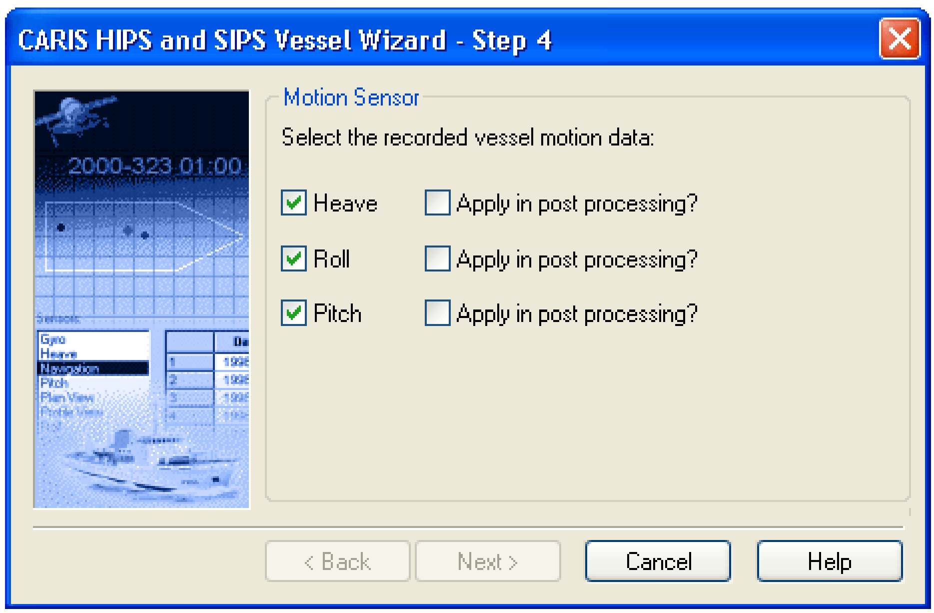
The *Squat* and *Heave* percentage values are transducer responses to overall heave and squat. For fixed-boom transducers this should be 100% and less than 100% for decoupled-boom transducers.

8. Type a degree offset in the *Roll* and *Gyro* fields.
9. Type a percentage value in the *Squat* and *Heave* fields.
10. Click **Next.**

Continue with "Configuration Options" on page 37.

Motion Sensors

This dialog box determines which attitude sensors are displayed in the HVF.

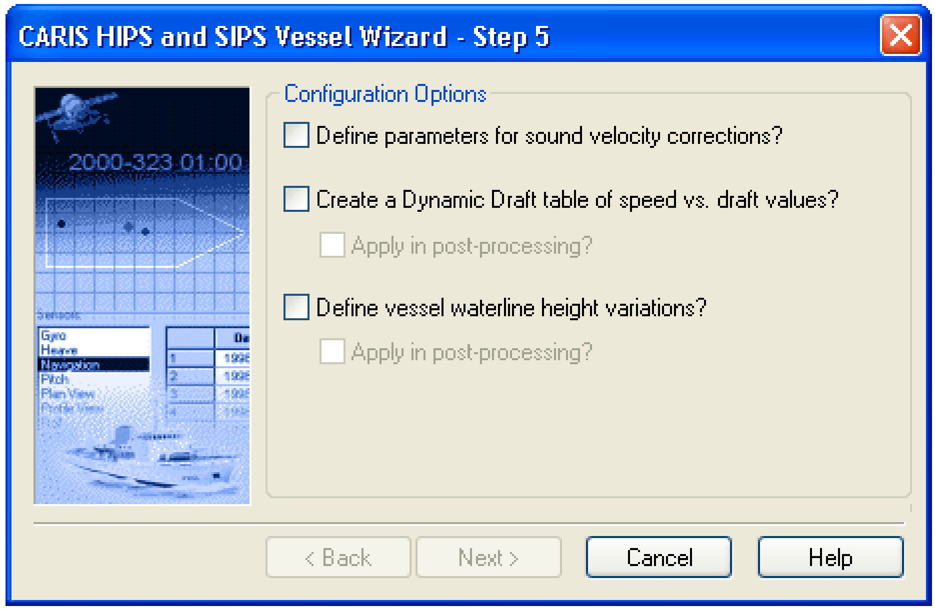


1. Create entries for any (or all) of the following sensors by clicking the appropriate check box:
   - *heave*
   - *pitch*
   - *roll*
2. Select the *Apply in Post Processing* option to apply the motion data during the Merge process or during Sound Velocity Correction.
3. Click **Next.**

Continue with "Configuration Options" on page 37.

Configuration Options

The Configuration Options dialog box is displayed



Use this dialog box to set configuration options:

- SVP (Sound Velocity Profile) pole parameters
- dynamic draft values
- waterline height variation

To include sound velocity corrections:

4. Enable the *Define Parameters for Sound Velocity Corrections* option.

If you select this option, an extra step is added to the wizard and the **Next** button becomes active.

To apply dynamic draft values to vessel speeds:

5. Enable the *Apply Dynamic Draft* option.

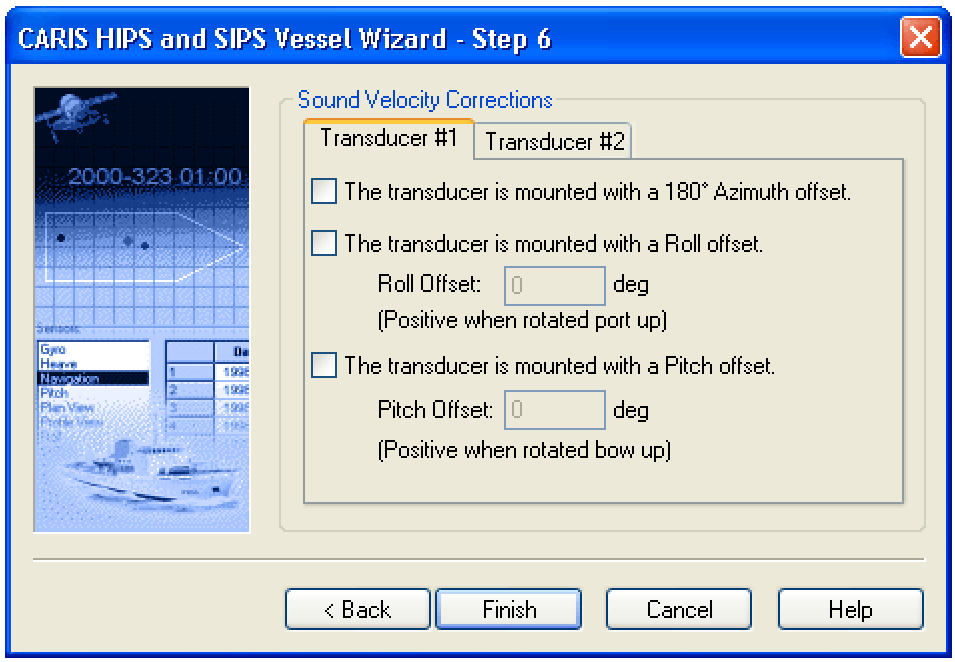
A draft-versus-speed table is created in the Vessel Editor. You can enter values in this table once the vessel file is created.

To apply long period variations in the waterline height in processing data:

6. Enable the *Define Vessel Waterline Height Variation* option.
7. Enable the appropriate *Apply in Post Processing* options, as needed.
8. Click **Next** if the button is active. Otherwise, click **Finish**.

Sound Velocity Corrections

The Sound Velocity Corrections dialog box is displayed only if you selected the *Define parameters for sound velocity corrections* check box in the Configuration Options dialog box.



In this dialog box you enter any transducer pole offsets.

Transducer head offsets must be taken into account when the SVP is applied to the data.

1. Select the 180-degree Azimuth box if the transducer is reverse mounted.
2. If the transducer is mounted with a large roll offset for use under wharves or along banks, then select the *Roll Offset* check box
3. Type the amount of offset (in degrees) in the *Roll Offset* box.
4. If the transducer is mounted with a large pitch offset such that it is pointed forward or backwards, then select the *Pitch Offset* check box.
5. Type the amount of offset (in degrees) in the *Pitch Offset* box.
6. Repeat the above steps for the second transducer if there are two poles.
7. Click **Finish**.

A new HIPS Vessel File is created. You can now create an outline for the vessel (see "Create Vessel Shape Outline" on page 39).

Create Vessel Shape Outline

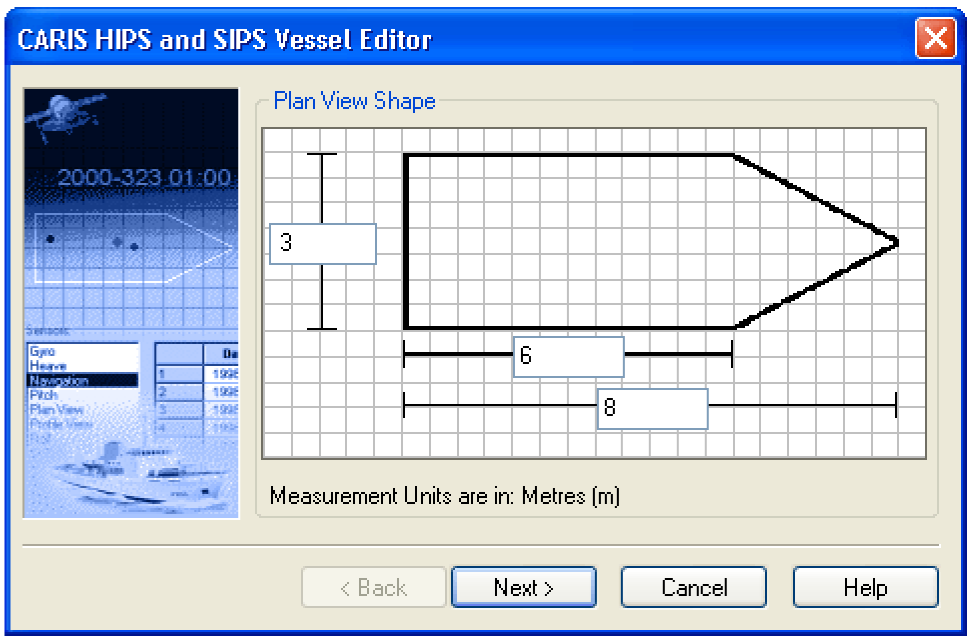
Create a 3-D outline of the vessel, by defining its length, width and height, as well as the position of the reference point in the vessel.

You can also revise an existing vessel outline.

Menu	Edit > Vessel Shape
Tool	

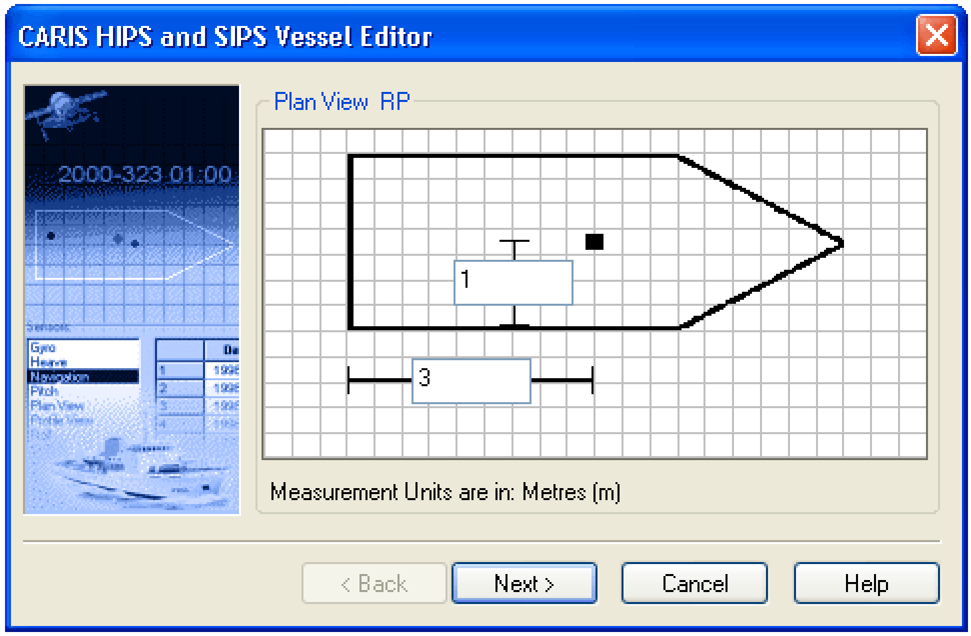
1. Open the vessel file if it is not already open.
2. Select the Vessel Shape command.

The Plan View Shape dialog box is used for entering width and length of the vessel.



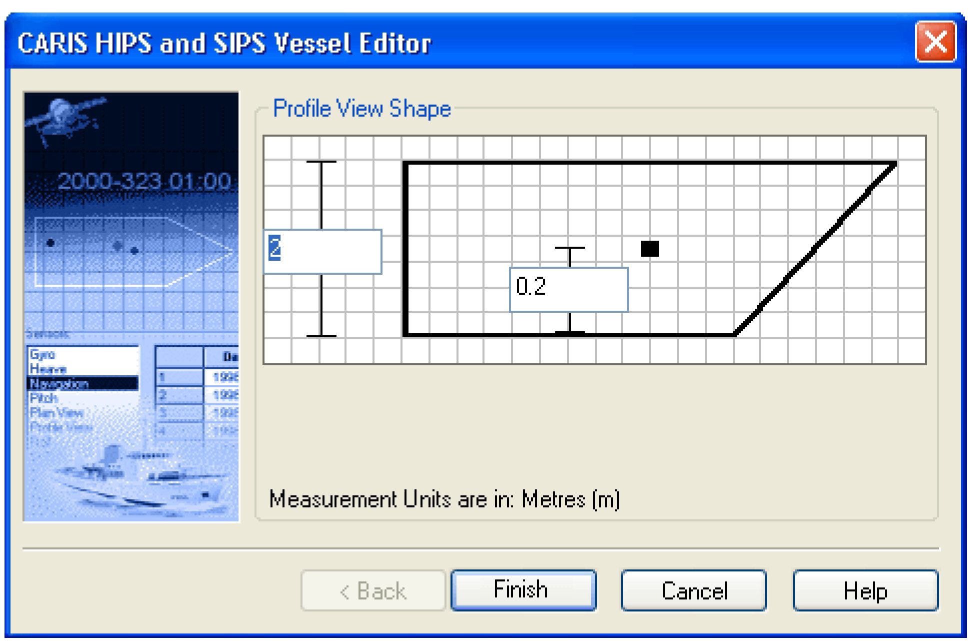
1. Enter a width for the vessel in metres, measured from port to starboard.
2. Enter vessel length for the vessel from stern to the base of the bow.
3. Enter the length from the stern to the tip of the bow.
4. Click **Next**.

The Plan View PR dialog box is displayed. It is used to set the position of the Reference Point (RP).



1. Type the distance from the stern to RP.
2. Type the distance from the starboard side to the RP.
3. Click **Next**.

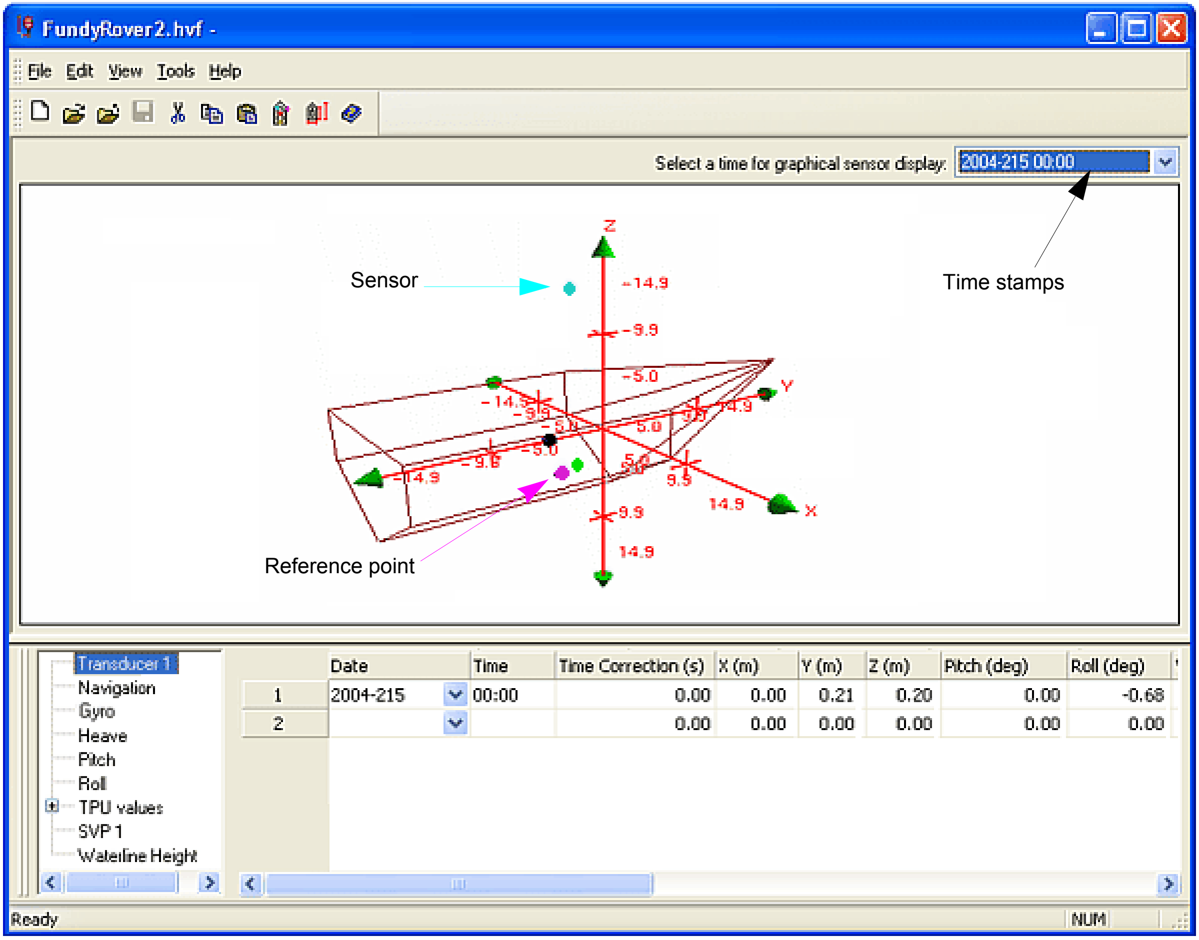
The Profile View Shape dialog box is displayed.



Set the height measurements for the vessel:

1. Type the height of the vessel.
2. Type the height of the vessel RP from the keel.
3. Click **Finish**.

The outline of the vessel is displayed in Vessel Editor.



Negative values are used for these measurements:

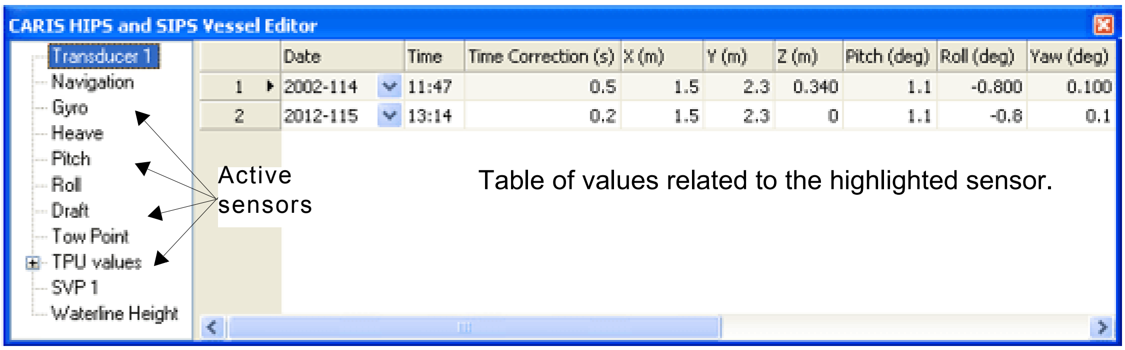
- from the origin (crosspoint of the axes) to port on the X axis,
- from the origin aft towards the stern on the Y axis,
- from the origin point up, on the Z axis.

Negative values on the axes are displayed with a minus sign (-).

Sensor Configuration

The active sensors listed in the Vessel Editor are determined by the sensor parameters you selected when creating the vessel file.

If you want to add data for a sensor not already included in the vessel file, you can add a sensor to the list and enter its data. You can also remove a sensor and its values from the vessel file.



In general, if sensor offsets or calibration values have already been applied to logged data during data acquisition, then do not enter the same offsets and calibration parameters in the Vessel Editor.

Also, if compensation for heave, pitch, and roll has already been made to the recorded sounding data during data acquisition, for example as in Kongsberg data, then it must not be applied again.

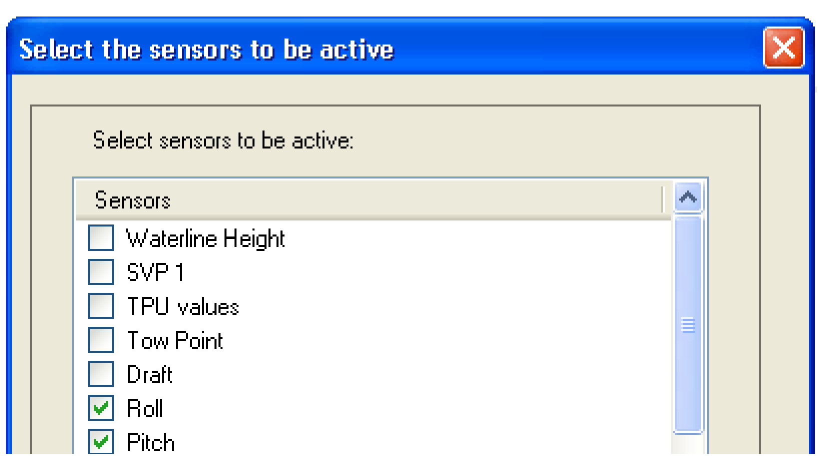
Active sensors

Menu	Edit > Active Sensors
Tool	

To add or remove sensors from the list of active sensors:

1. Select the Active Sensors command.

The Active Sensors dialog box is displayed.



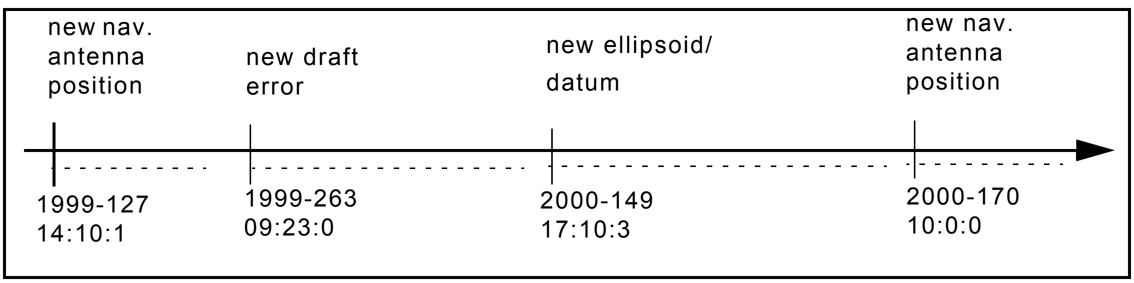
If a box is checked, the sensor is included in the vessel file and displayed in the Editor table.

2. Check or clear the sensor check boxes as required.
3. Click **OK**.

The list is refreshed to show the selected active sensors.

Time Stamp

A vessel's configuration changes over time. Different draft settings may be used in a survey, or the position of the transducers on a boom may change, or a different ellipsoid may be used. A time stamp is used to keep track of these changes. The time stamp records the time from which a configuration is valid and is recorded with each entry to the HVF. Below is an example of time stamps showing changes in vessel configuration.



Time in HIPS is based on the recorded reference time. No distinction is made for UTC.

Time Correction

In the HVF, most sensors have a *Time Correction* field that contains the time difference between the sensor clock and the reference clock. Sensors are time tagged and all clocks are compared to the reference clock so sensor readings can be synchronized.

The *Time Correction* field is specified in seconds and is positive if ahead of the reference clock:

```
Time Correction = Recorded Time - Reference Time
```

Sensors

Different sensors can require specific edits applied to the parameter data. The following sections explain the individual sensor data fields.

Transducer

The swath or multibeam sonar typically has beams that form a fan shape, radiating from the centre of the transducer.

1. Click *Transducer1* in the Sensors list box to highlight the selection and display the swath data fields.
2. Type data as needed in the following fields:
   - *Date*: The year and Julian day of current swath time stamp.
   - *Time*: The hour and minute of the current swath time stamp.
   - *Time Correction*: The time correction value.

The X-Y-Z fields set the location of the transducer from the Reference Point (0).

- *X*: The athwart-ship distance of the transducer, positive to starboard.
- *Y*: The along-ship distance of the transducer, positive to the bow.
- *Z*: The vertical distance of the transducer, positive into the water.

The *Pitch*, *Roll* and *Yaw* fields refer to misalignment of the transducer, during mounting, from the vessel coordinate system.

- *Pitch*: The offset is positive when rotating the transducer towards the bow (bow up).
- *Roll*: The offset is positive when rotating the transducer away from starboard (starboard down).
- *Yaw*: Indicates the horizontal rotational offset and is positive for a clockwise rotation.
- *Manufacturer*: The maker of the transducer.
- *Model*: The particular make of transducer. The model information is necessary for the calculation of Total Propagated Uncertainty.
- *Serial Number*: The serial number of the transducer.

Since the Kongsberg data read into HIPS is already corrected, the Transducer values in the HVF must be set as follows:

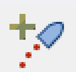
- The *X/Y/Z* offsets are zero because the Kongsberg data acquisition has already applied static draft and shifted the swath profile to the vessel reference point.
- The *Roll /Pitch /Yaw t*ransducer mounting rotations are typically zero because the Kongsberg data acquisition has already applied patch test calibration results.

Navigation

The navigation section of the HVF describes the location of the navigation source (for example, antenna or motion sensor). This section is used as a link between the fixed positioning coordinate system and the instantaneous vessel coordinate system.

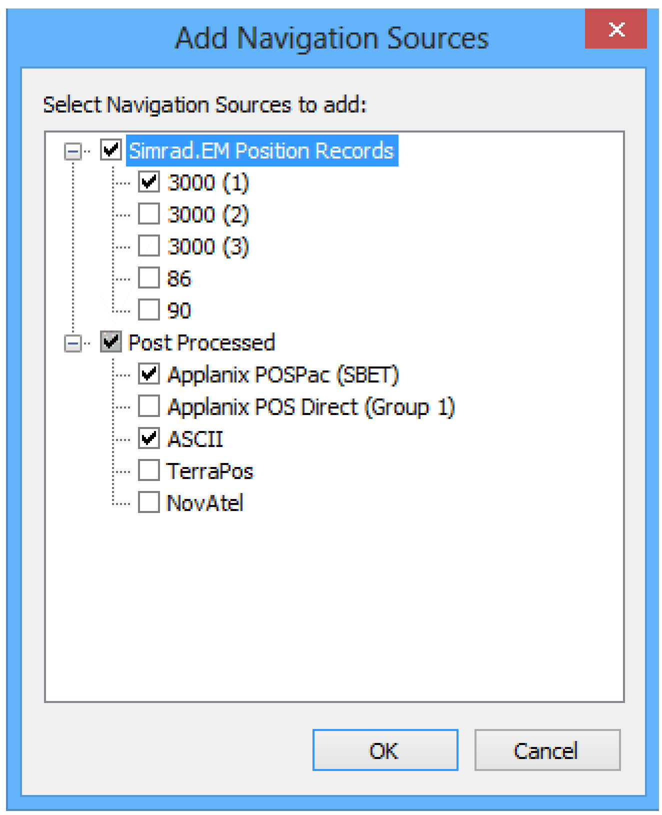
Navigation can loaded from multiple sources for each line, and each of these sources can have their own configuration in Vessel Editor. If a configuration is not specified for a specific source type, the default Navigation sensor configuration is used.

To configure navigation sources:

Menu	Edit > Add Navigation Sources
Tool	

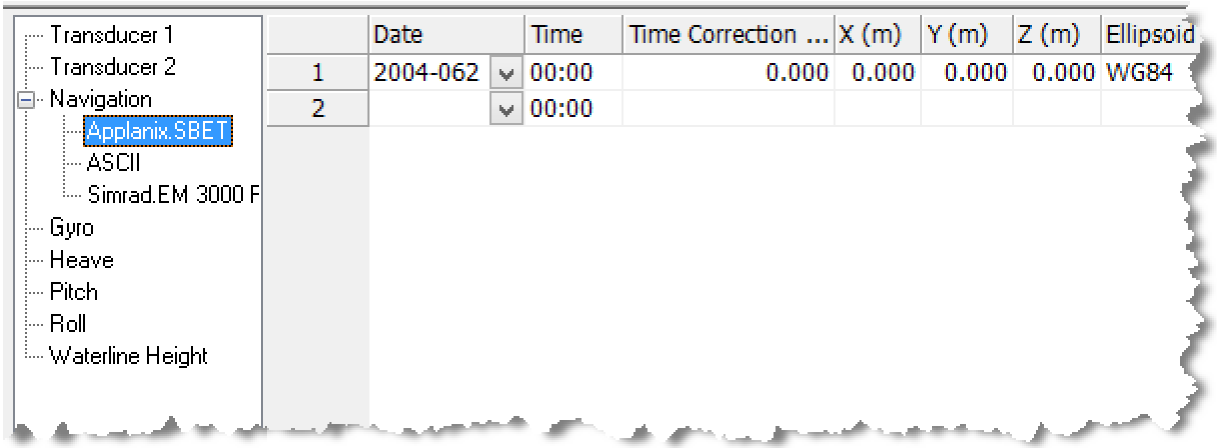
1. Click *Navigation* in the Sensors list box to highlight the selection and enable the navigation data fields.
2. Select the Add Navigation Sources command.

The Add Navigation Sources dialog box is displayed.



3. Select the sources for the project from the list of available formats.
4. Click **OK**.

Each of the selected sources will be listed in the sensors table.



5. Type data as needed in the following fields:
   - *Date*: The year and Julian day of the current navigation time stamp.
   - *Time*: The hour and minute of the current navigation time stamp.
   - *Time Correction*: The time correction value.

The *X-Y-Z* fields set the location of the navigation source from the Reference Point (0).

- *X*: The athwart-ship distance of the source, positive to starboard.
- *Y*: The along-ship distance of the source, positive towards the bow.
- *Z*: The vertical distance of the source, positive into the water.
- *Ellipsoid*: From the drop-down list, select the ellipsoid on which the navigation is based. The ellipsoid should be the same as the one used in the survey.

If you will be applying GPS tide data it must use the ellipsoid selected here.

- *Manufacturer*: The maker of the navigation equipment.
- *Model*: The particular make of navigation equipment.
- *Serial Number*: The serial number of the navigation equipment.

Gyro

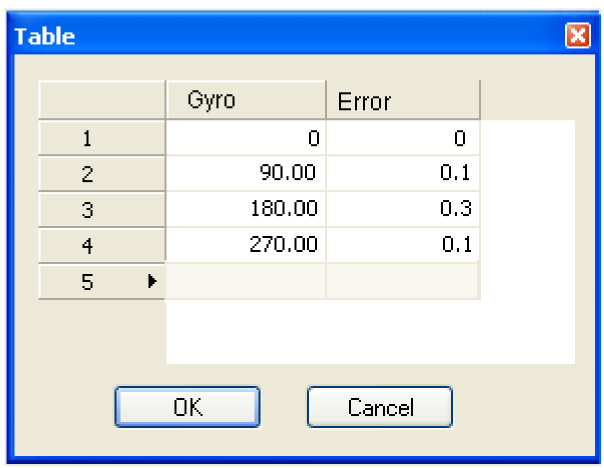
The gyro sensor refers to any device providing a heading orientation of the vessel.

1. Click *Gyro* in the Sensors list box to highlight the selection and display the gyro data fields.
2. Type data as needed in the following fields:
   - *Date*: The year and Julian day of the current gyro sensor time stamp.
   - *Time*: The hour and minute of the gyro sensor time stamp.
   - *Time Correction*: The time correction value.

The Gyro Error is measured in degrees and is the difference between the recorded sensor value and the applied heading of the vessel:

```
Applied Heading = Recorded value - Gyro Error.
```

3. Click an **Edit** button in the *Error* column to open the Gyro Error Table.



4. Type new values in the *Gyro* and *Error* fields and click **OK**.

The Error Table is closed.

5. Type data as needed in the following fields:
   - *Manufacturer*: The maker of the sensor equipment.
   - *Model*: The particular make of sensor equipment.
   - *Serial Number*: The serial number of the sensor equipment.

Heave

The heave sensor records the vertical motion of the vessel. Although the heave sensor has its own section in the Vessel Editor, it is part of the same sensor package as Pitch and Roll in most survey configurations.

1. Click *Heave* in the Sensors list to highlight the selection and display the data fields.
2. Type data (as needed) in the following fields:
   - *Date*: The year and Julian day of the current heave sensor time stamp.
   - *Time*: The hour and minute of the current heave sensor time stamp.
   - *Time Correction*: The time correction value.

The *X-Y-Z* fields set the location of the heave sensor from the Reference Position (0).

The X-Y-Z location of the heave sensor must be defined ONLY when you intend to apply heave data during Merge AND when remote heave compensation is necessary.

- *X*: The athwart-ship distance of the sensor, positive to starboard.
- *Y*: The along-ship distance of the sensor, positive to the bow.
- *Z*: The vertical distance of the sensor, positive into the water.
- Error: The applied instantaneous values for the heave sensor are computed by subtracting the appropriate errors from the recorded values:

  `Applied Heave = Recorded Heave - Heave Error.`
- *Apply*: Select Yes to apply heave data in the merge process.
- *Manufacturer*: The maker of the sensor equipment.
- *Model*: The particular make of sensor equipment.
- *Serial Number*: The serial number of the sensor equipment.

The location of the heave sensor is shown in the Vessel Editor by a coloured dot.

Kongsberg systems apply dynamic Heave, Pitch, and Roll values to the swath data during survey. Therefore, the HVF must be set up with the *Apply* switches for Heave, Pitch, and Roll set to '”No” so these values are not applied twice.

Pitch

Pitch refers to the rotational motion of the vessel around the X-axis (port/starboard). Although the pitch sensor has its own section in the Vessel Editor, it is part of the same sensor package as roll and heave in most survey configurations.

1. Click *Pitch* in the Sensors list box so the selection is highlighted and the data fields are displayed.
2. Type data (as needed) in the following fields:
   - *Date*: The year and Julian day of the current pitch time stamp.

- *Time*: The hour and minute of the current pitch time stamp.
- *Time Correction*: The time correction value.
- *Error*: The applied instantaneous values for the pitch sensor are computed by subtracting the appropriate errors from the recorded values:

  `Applied Pitch = Recorded Pitch - Pitch Error.`
- *Apply*: Select Yes to apply the pitch data in the merge process.
- *Manufacturer*: The maker of the sensor equipment.
- *Model*: The particular make of sensor equipment.
- *Serial Number*: The serial number of the sensor equipment.

The location of the pitch sensor is indicated in the Vessel Editor by a coloured dot.

Kongsberg systems apply dynamic Heave, Pitch, and Roll values to the swath data during survey. Therefore, the HVF must be set up with the *Apply* switches for Heave, Pitch, and Roll set to '"No" so these values are not applied twice.

Roll

Roll refers to the rotational motion of the vessel around the Y-axis (fore/aft). Although the roll sensor has its own section in the Vessel Editor, it is part of the same sensor package as pitch and heave in most survey configurations.

1. Click *Roll* in the Sensors list box to highlight selection and display the roll data fields.
2. Type data (as needed) in the following fields:
   - *Date*: The year and Julian day of the current roll time stamp.
   - *Time*: The hour and minute of the current roll time stamp.
   - *Time Correction*: The time correction value.
   - *Error*: The applied instantaneous values for the roll sensor are computed by subtracting the appropriate errors from the recorded values:

     `Applied Roll = Recorded Roll - Roll Error.`
   - *Apply*: Select Yes to apply the roll data in the merge process.
   - *Manufacturer*: The maker of the sensor equipment.
   - *Model*: The particular make of the sensor equipment.
   - *Serial Number*: The serial number of the sensor equipment.

The location of the roll sensor is indicated in the Vessel Editor by a coloured dot.

Kongsberg systems apply dynamic Heave, Pitch, and Roll values to the swath data during survey. Therefore, the HVF must be set up with the *Apply* switches for Heave, Pitch, and Roll set to '"No" so these values are not applied twice.

Dynamic Draft

The squat and lift of a vessel changes as the speed changes. For some vessels, if the squat/lift is not accounted for, significant errors are introduced into the soundings. In this section, you can specify up to 10 speed-draft value pairs. During merge, the difference between the instantaneous draft and the static draft is computed and the final depth compensated.

Draft is measured in metres/feet, and speed is measured in knots. All draft values should be relative to the same reference, but the actual reference is not important. The first Speed-Draft pair must correspond to a state where no draft correction is necessary. The delta draft values, relative to the initial draft, are computed and used to correct observed soundings.

This formula is used to correct soundings for dynamic draft:

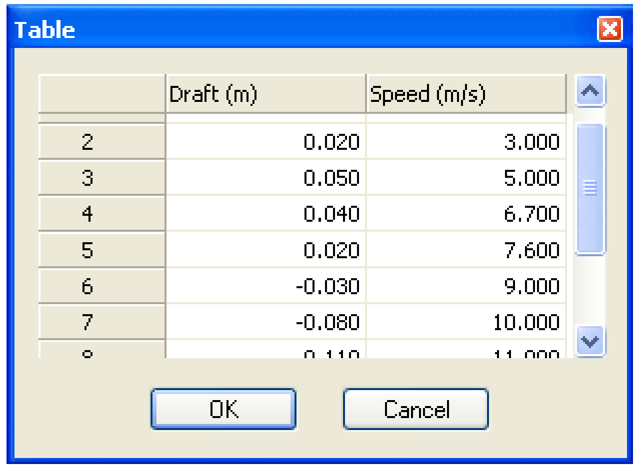
```
Depth = observed depth – waterline + delta draft.
```

Delta draft is computed from the present vessel speed, as derived from the navigation system.

You can also create your own dynamic draft model and load Delta Draft information directly which will override the Dynamic Draft table in the HVF.

1. Click *Dynamic Draft* in the sensors list box to highlight the selection and display the dynamic draft data fields.
2. Type data as needed in the following fields:
   - *Date*: The year and Julian day of the current time stamp for the draft table.
   - *Time*: The hour and minute of the current time stamp for the draft table.
   - *Apply*: Select Yes to apply the table during merge (or No to not apply the table).
3. Click **Edit** in the *Error* field.

A dialog box containing a table with up to 10 speed-draft pairs is displayed.



4. Click inside the *Speed* cell and type a speed value.
5. Click inside the *Draft* cell and type a corresponding draft value.
6. Continue entering speed /draft pairs as needed.
7. Click **OK**.

The speed-draft table is saved in the HVF.

SVP

A sound velocity profile (SVP) records the speed of sound at various depths in the water column. HIPS lets you apply SVP data in sound velocity corrections, but not for all types of sonars because some sonar data logging systems compensate for sound velocity during data acquisition.

To ensure the SVP is accurately applied in HIPS, the transducer X-Y-Z offset values must be entered.

1. Click *SVP 1* in the Sensors list box.

The selection is highlighted and the SVP transducer data fields are visible. If there is a second transducer then click *SVP 2* and complete the same procedures as listed below.

2. Type data as needed in the following fields:
   - *Date*: The year and Julian day of the current SVP pole time stamp.
   - *Time*: The hour and minute of the current SVP pole time stamp.
3. Select Yes if there are dual transducers or No if there is only a single transducer.

The following three fields are for entering the X-Y-Z coordinates of the pole. All coordinates are measured from the Reference Point.

4. Type the transducer’s X-Y-Z offsets in the following fields:

- *X*: The athwart-ship distance of the transducer, positive to starboard.
- *Y*: The along-ship distance of the transducer, positive to the bow.
- *Z*: The vertical distance of the transducer, positive into the water.

> The Reference Point for Kongsberg data is the centre of rotation. It should be possible to retrieve offset values from the Kongsberg Installation Datagram.

The next fields refer to the alignment of the transducer. The values you entered in the wizard when creating the HVF are displayed in these fields. The fields are only to be used for large transducer mounting offsets.

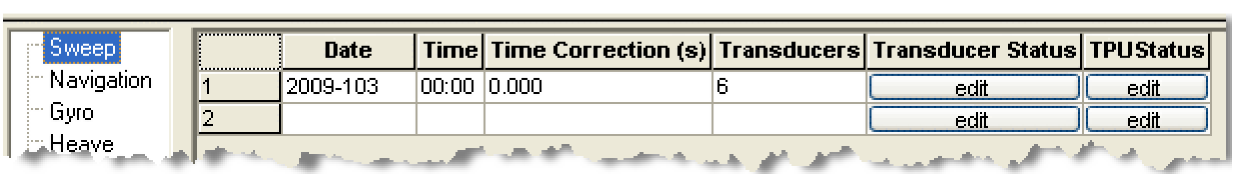
- *Pitch*: Pitch offsets of the transducer.
- *Roll*: Roll offsets of the transducer.
- *Yaw*: Rotation of the transducer (either 0 or 180 degrees).

For information on setting the sound velocity corrections, see "Sound Velocity Corrections" on page 38.

Sweep

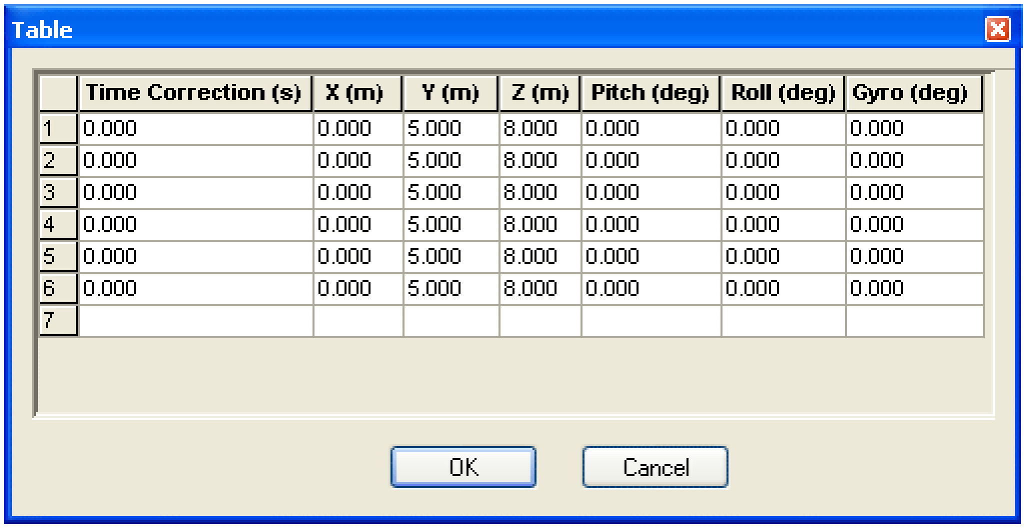
Sweep systems typically contain multiple vertical beam transducers mounted on a boom and pointed straight down. The reference point for a sweep system can be anywhere on the X-Y plane, but must be on the water surface.

1. Click *Sweep* in the Sensors list box to highlight the selection and display the data fields.



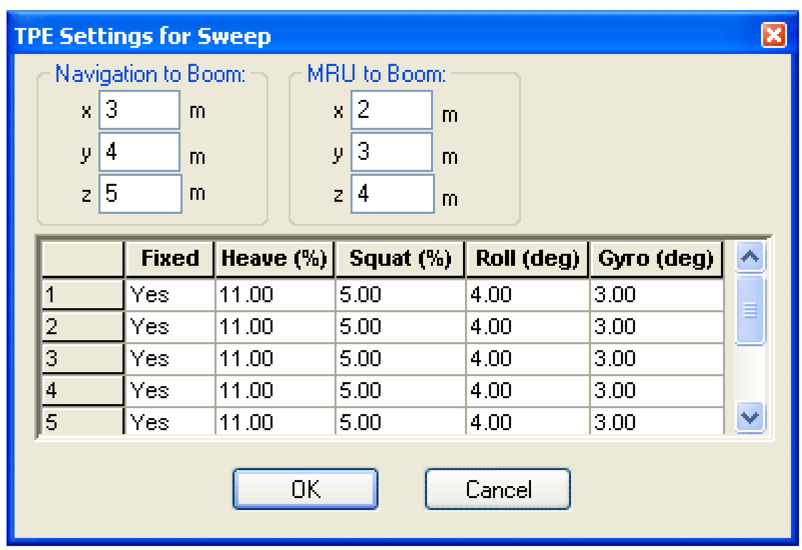
2. Type data as needed in the following fields:
   - *Date*: The year and Julian day of the current sweep time stamp.
   - *Time*: The hour and minute of the current sweep time stamp.
   - *Time Correction*: The time correction value.
   - *Transducers*: Number of beams mounted on the boom.
3. Click the **Edit** button in the *Transducer Status* field.

The table of values for individual beams on the boom is displayed.



4. Type values in the fields and click **OK**.
5. Click **Edit** in the *TPU Status* field.

The TPU Settings dialog box is displayed. Enter TPU values for each beam on the boom and edit values entered during creation of the new vessel file in the wizard.



6. Type values as needed and click **OK.**

Towed Sensors

If a towed sensor such as a side scan sonar is used on a survey, then HIPS and SIPS can compute the sensor's position relative to the ship's position by calculating the horizontal layback and a direction to the sensor.

1. Click *Towed* in the Sensors list box to highlight the selection and display the Towed data fields.

2. Type data as needed in the following fields:
   - *Date*: The year and Julian day of the current towed sensor time stamp.
   - *Time*: The hour and minute of the current towed sensor time stamp.
   - *Time Correction*: The time correction value.
   - *Layback Error*: The error in the computed or recorded horizontal layback. This can be used, for example, to compensate the recorded layback data or tow cable length for the distance between the origin of the measurements and the defined tow point location in the tow cable length. This value is subtracted from the computed horizontal layback.
   - *X*: Offset of the tow point from the vessel's reference position.
   - *Y*: Offset of the tow point from the vessel's reference position.
   - *Z*: Height of the tow point (negative upwards) in relation to the datum that is referenced to the sensor depth (in most cases, this is the waterline height).
   - *Manufacturer*: The maker of the towed sensor.
   - *Model*: The make of the towed sensor.
   - *Serial Number*: The serial number of the towed sensor.

Waterline Height

This section of the vessel file defines long-period changes in the vessel's draft due to fuel burn or other loading changes.

Specifically, it defines the height of the waterline below the Reference Point.

If the Waterline Height is not defined here, HIPS uses a default of zero as the height.

If you want waterline height to be applied during sound velocity correction, there must be a value in the Waterline Height section of the HVF.

No interpolation of waterline height is done during sound velocity correction.

1. Click *Waterline Height* in the Sensors list box to highlight the selection and display the waterline data fields.
2. Type data (as needed) in the following fields:

   Date: The year and Julian day of the current waterline time stamp.

   Time: The hour and minute of the current waterline time stamp.

   Waterline: The distance from the RP, positive when below the RP.

   Apply: Select Yes to apply the waterline data in the Merge process.

   Comments: A text field for your use.

> For Kongsberg data, the Waterline value must be set to the same value recorded as **WLZ** in the Kongsberg Installation Datagram.
>
> This Waterline value will only be used during Sound Velocity Correction.
>
> Set the Apply switch to **“No”**. If it is set to “Yes”, it will be applied twice, once in SVC and again in Merge.

TPU -Total Propagated Uncertainty

The values entered here will be used in the calculation of Total Propagated Uncertainty (TPU). TPU is derived from a combination of estimates of the accuracy of each individual sensor, estimates such as.

- nav/gyro/heave/pitch/roll/tide errors
- latency error estimate
- sensor offset error estimates

These uncertainty estimates are combined with individual sonar model characteristics in DeviceModels.xml to calculate horizontal and vertical uncertainty values for every sounding along a track line when TPU is applied

Sensor accuracy values for various sonar types can be viewed on the TPU Computation Resource page of the CARIS web site www.caris.com/tpu.

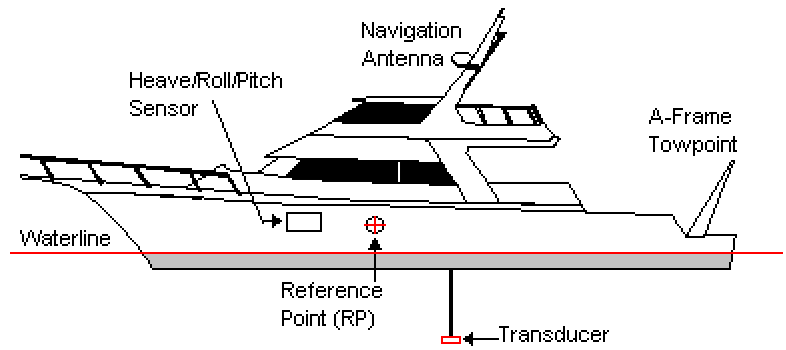
Sensor accuracy values for TPU must be entered as 1-sigma.

1. Expand the TPU section of the HVF by clicking the + icon.
2. Type data as needed in the Offsets section:
    - *MRU to Transducer*: The physical offset in three dimensions from the motion recording unit to transducer 1 on the vessel.
    - *MRU to Transducer2*: The physical offset in three dimensions from the motion recording unit to transducer 2 on the vessel.
    - *Navigation To Transducer*: The physical offset in three dimensions from the navigation antenna to transducer 1 on the vessel.
    - *Navigation To Transducer2*: The physical offset in three dimensions from the navigation antenna to transducer 2 on the vessel.
    - *Transducer Roll*: The mounting roll offset for transducer 1. The offset is positive when rotating the transducer away from starboard (starboard down).
    - *Transducer Roll 2*: The mounting roll offset for transducer 2. The offset is positive when rotating the transducer away from starboard (starboard down).
3. Uncertainty values used Type data as needed in the Standard Deviation section:

- *Motion Gyro*: The measurement standard deviation of the heading data in degrees.
- *Heave % Amplitude*: An additional heave standard deviation component that is the percentage of the instantaneous heave.
- *Heave (m)*: The measurement for standard deviation of the heave data. Most heave manufacturers quote heave error as being determined from *StaticHeave* or *PercentageOfHeave* depending on which value is larger.
- *Roll*: The measurement standard deviation of the roll data in degrees.
- *Pitch*: The measurement standard deviation of the pitch data in degrees.
- *Position Nav*: The standard deviation associated with the measurement of positions for the vessel. This is usually the error of the GPS sensor being used.
- *Timing Trans*: Standard deviation in transducer time stamp measurement.
- *Nav Timing*: Standard deviation in navigation time stamp measurement.
- *Gyro Timing*: Standard deviation in gyro time stamp measurement.
- *Heave Timing*: Standard deviation in heave time stamp measurement.
- *Pitch Timing*: Standard deviation in pitch time stamp measurement.
- *Roll Timing*: Standard deviation in roll time stamp measurement.
- *Offset X*: Standard deviation for the X measured offset on the vessel.
- *Offset Y*: Standard deviation for the Y measured offset on the vessel.
- *Offset Z*: Standard deviation for the Z measured offset on the vessel.
- *Vessel Speed*: The standard deviation for the vessel speed measurements.
- *Loading*: Vertical changes during the survey because of fuel consumption, etc.
- *Draft*: The standard deviation in the vessel draft measurements.
- *Delta Draft*: The standard deviation in the dynamic vessel draft measurements.
- *MRU Align StdDev Gyro*: This value is the uncertainty of the motion recording unit placement within the vessel fixed coordinate frame.
- *MRU Align StdDev Roll/Pitch*: This value is the uncertainty of the motion recording unit placement within the vessel fixed coordinate frame.
- *Comments*: Any additional information.

Vessel Coordinate System

Vessel configuration is based on a three-dimensional coordinate system, which is used to record the location of sensors and other equipment, as illustrated below.



Reference Point

Sensor positions are described using X-Y-Z axis coordinates relative to a Reference Point (RP). The RP is a location from which all other positions are derived.

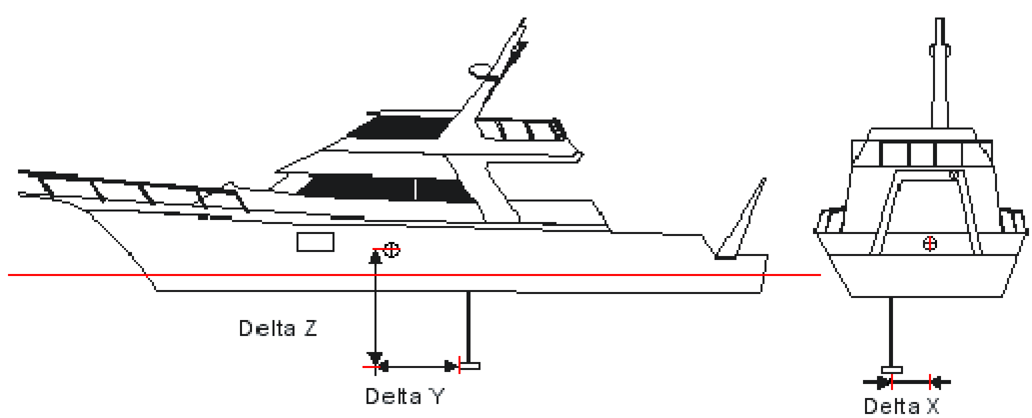
The location of the RP depends upon the type of sonar system used in the survey. Its location is usually at the centre of gravity of the ship. This position is used as the centre of the ship's rotation when applying the pitch/heave/roll parameters, and must be applied as accurately as possible.

Coordinate definitions

The axes are defined as follows:

- The Y-axis is oriented along the vessel's fore/aft axis, positive forward.
- The X-axis is oriented along the vessel's port/starboard axis, perpendicular to the Y-axis, positive to the starboard.
- The Z-axis is perpendicular to the X-Y plane, and positive into the water.

This is illustrated in the following image.



The location of the vessel's coordinate system within the local fixed coordinate system is determined by the navigation system and tide. The orientation of the coordinate system is defined by the vessel's gyro and attitude sensors (pitch and roll). Gyro, pitch and roll observations are defined as follows:

- A positive gyro observation is defined as the clockwise rotation of the vessel (from 0 and 360 degrees) within the navigation coordinate system.
- A positive pitch is observed when the bow of the vessel is down (bow down).
- A positive roll is observed when the starboard side of the vessel is up (starboard up).

Default Ellipsoid

The HVF must contain the ellipsoid and datum used by the navigation system in the survey vessel, regardless of whether or not the data contains projection coordinates or geographic coordinates.

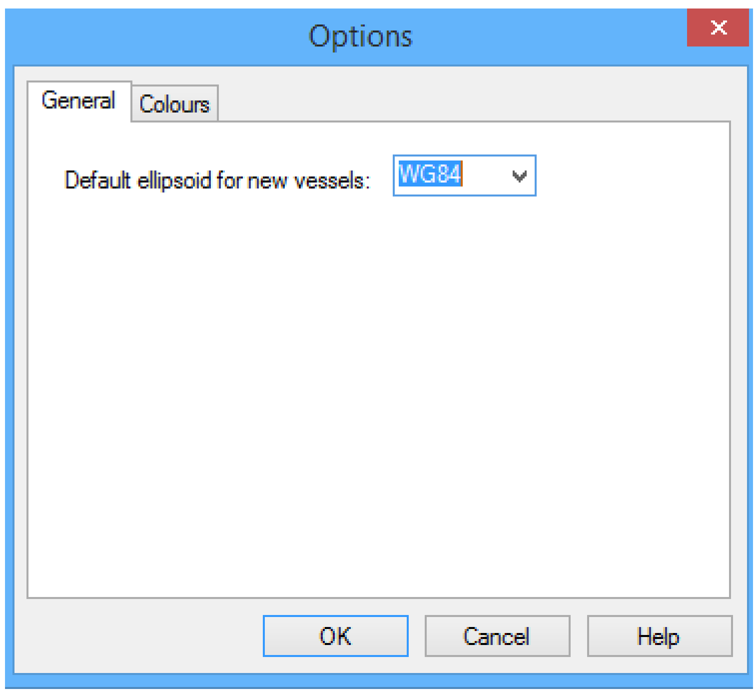
The list of available ellipsoids is maintained in the file named datum.dat referenced by the environment variable named uslXdatum. By default, this file is located in the System directory.

Menu	Tools > Options

Set the ellipsoid for a new vessel file to be the same as the one used in the survey.

1. Select the Options command.

The Options dialog is displayed.



2. Select the General tab.
3. Select an ellipsoid from the list.
4. Click **OK**.

You are returned to Vessel Editor and the name of the ellipsoid is displayed in the Navigation section.

3

Create a New Project

All project information in HIPS and SIPS is maintained in a project database. New projects are created with a wizard, which generates a *.HIPS file.

Projects in HIPS and SIPS

In HIPS and SIPS, survey data is organized into projects. The default directory for project files is `drive\Program Files\CARIS\HIPS\version\Data`.

Before any survey data can be processed, a HIPS Vessel file (HVF) must exist. This file defines the offset configurations and associated error estimates for each of the sensors, which are necessary for creating final position and depth records for survey data.

If no HVF is available, it must be defined before the new project can be created. (See "CREATE A NEW HVF" ON PAGE 31.)

Projects that were saved as *.hpf files, are converted to *.hips when you first open them in HIPS and SIPS.

New Project

Menu	File > New > Project
Tool	

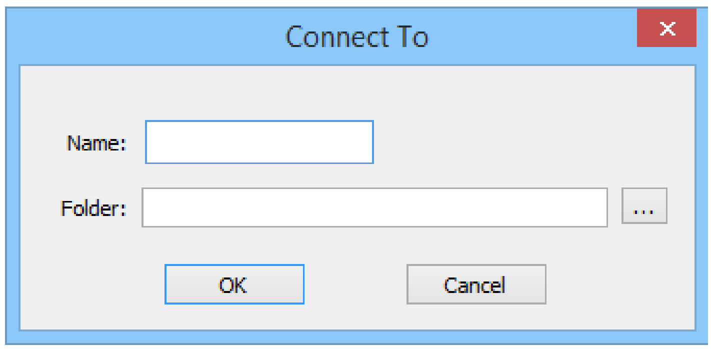
To create a new project:

1. Ensure that a vessel file has been created for this project.
2. Select the New Project command to open the New Project wizard.

If you have not defined a repository for your project data, an alert message displays the choice to do so.

3. Click **Yes** to create a project directory.

The Connect To dialog box is displayed.



4. Type a *Name* for your repository.
5. Click the browse button to open the Browse for Folder dialog box.
6. Navigate to select the location where you want the project data to be stored.
7. Click **OK**.

The path to the data will be displayed in the *Folder* field.

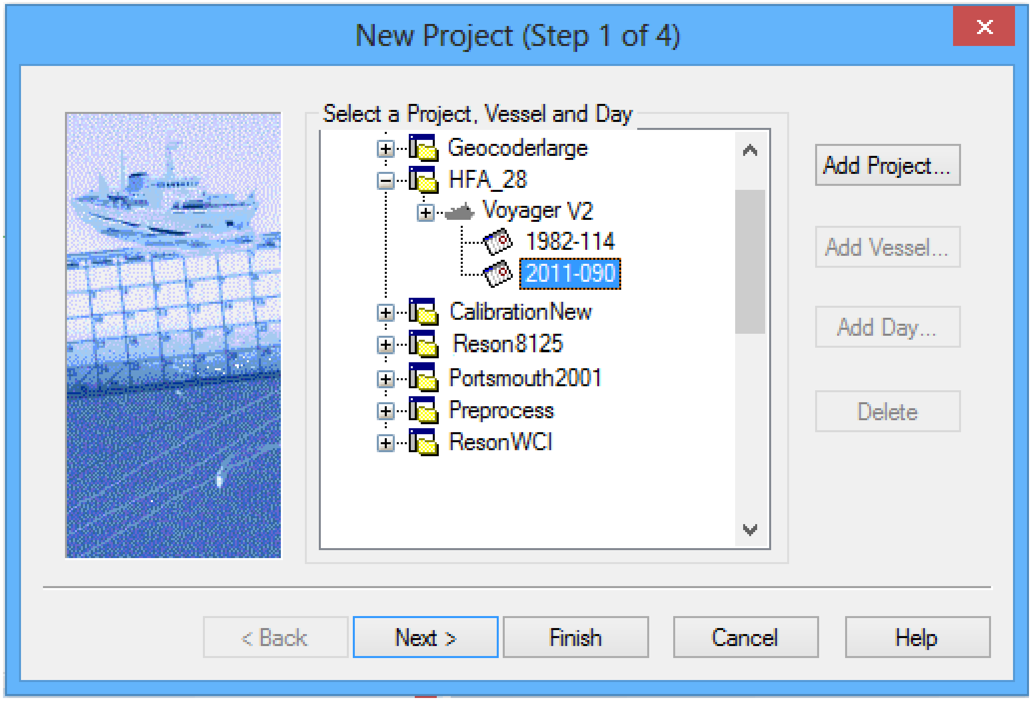
8. Click **OK** to connect.

The New Project wizard is displayed. Create the new project from these consecutive dialog boxes:

- "New Project (Step 1)" on page 63
- "New Project (Step 2)" on page 67
- "New Project (Step 3)" on page 68
- "New Project (Step 4)" on page 69

New Project (Step 1)

Project folders in HIPS and SIPS are organized in a Project/Vessel/Day hierarchy within a HIPS data repository. When creating a new project, you can add new Vessel and Day folders to an existing project, or create an entirely new project.



If an data directory has not been defined, you will be prompted to create a connection to one.

Add a new Project

1. Select an existing data folder and expand the tree to add a vessel, and day as required.
2. To create a new project folder, click **Add Project**.

The New Project Name dialog box is displayed.

3. Type a name for the new project.
4. Click **OK**.

A new Project folder is created and displayed in the dialog box in the selected data folder.

Change or add data repository

The Default directory for projects is ...\Hips*ver*\Data.

To create a new project in another project repository:

1. Right-click in the data tree area of the New Project dialog box.
2. Select Connect To… from the pop-up menu.
3. Name the new project folder and use **Browse** to set the path to the new data directory.
4. Click **OK**.

Add a new Vessel folder

The Vessel folder contains the folders for each survey day of the project. To add a Vessel folder to your project:

5. Select a project folder so it is highlighted.
6. Click **Add Vessel**.

The Available Vessels dialog box is displayed.

7. Select a vessel file from the list.
8. Click **OK**.

The Vessel folder is created and added to the new Project folder.

Add a new Day folder

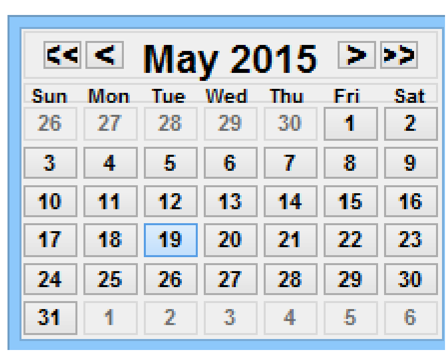
The Day folder contains track line data. For most formats the date is typically recorded with the raw data. If not, give the Day folder the date that the survey lines were recorded.

Day folders are shown in Julian Day format, for example, April 24, 2009 is shown as 2009-114.

To add a Day folder to a Vessel folder:

9. Select the Vessel folder.
10. Click **Add Day**.

The Calendar is displayed at the current year-month-day.



11. Select a different year or month, if needed, by clicking the arrow buttons at the top of the dialog box.
12. Select a new day, if needed, by double-clicking on the date.
13. Click **Add Day** again to create the Day folder with the selected dates.
14. Click **Next** to go to Step 2.

Add to an existing project

You can add new Vessel and Day folders to an existing project instead of creating a new project.

1. Select the existing project from the list in the dialog box.
2. Add vessel and day folders as needed.

> If you select an existing HIPS file (*.hips), and add a new vessel or day, the **Finish** button is displayed at this point. The original project parameters will be applied (set when the HIPS file was first created).

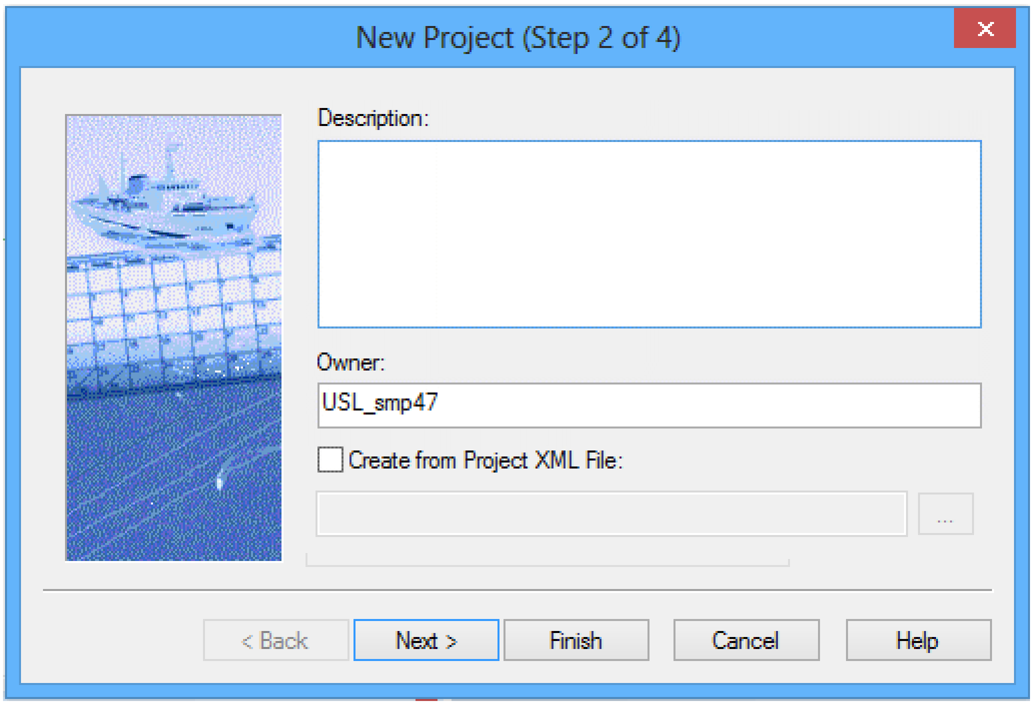
Delete empty folders

You can also delete an empty Day or Vessel folder by clicking **Delete**. **Delete** is disabled if a Vessel or Day folder already has data converted into it.

New Project (Step 2)

Enter project information and continue with the next step in the wizard, or select a pre-defined XML metadata file from which to create the project.

If you are creating a project using the New Project wizard, you can use this step to add descriptive information about the project.



1. [Optional] Type comments or identifying information about the project into the *Description* field.
2. [Optional] Type the name of the person working with the project data in the *Owner* field. The default name that is displayed is taken from the current Windows logon name.
3. Click **Next**.

Create from Project XML File

If you have a pre-defined project metadata XML file you can use it to create a project.

The XML file must follow the format of the HIPS configuration file or "schema", samples of which can be found in the folder: `HIPS\ver\System\Schemas\CARIS\chp\1.0`.

Alternatively if you have an XSLT file, HIPS can use that to translate the XML. In order to use this function, the *HIPS Program XML Transform* variable in Tools > Options > Environment must be set to the location of the XSLT file.

Sample project metadata files are available in `C:\ProgramFiles\CARIS\HIPS\`*`version`*`\Template`.

If you are creating a project using an XML metadata file:

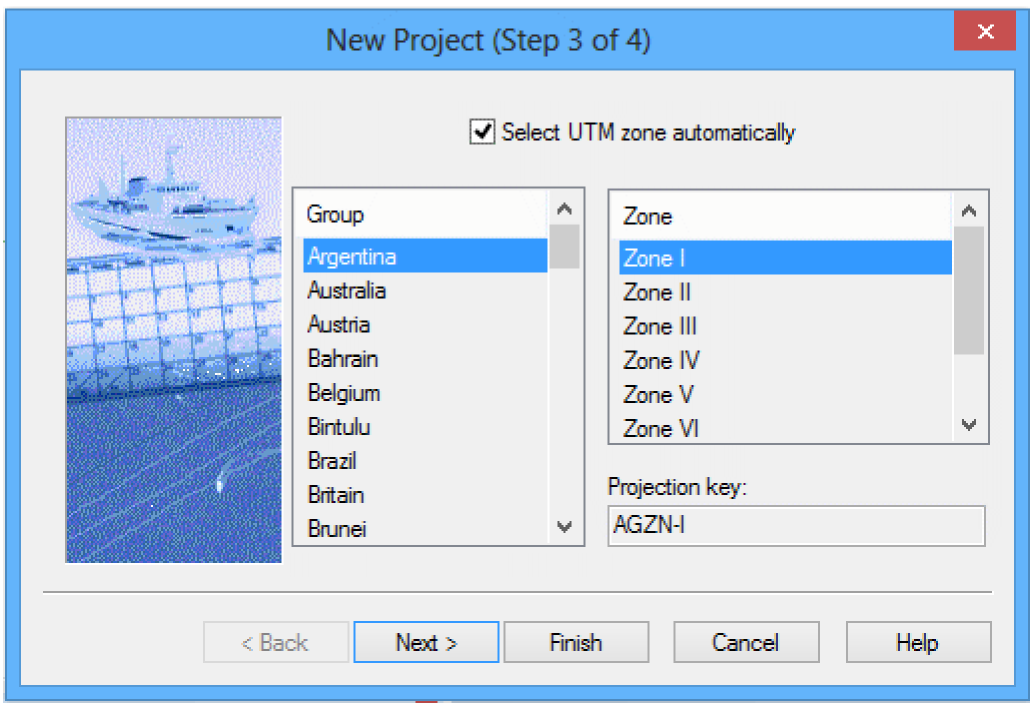
1. Select the *Create from Project X ML File* check box.

This activates a **Browse** button below it.

2. Click **Browse**.
3. In the Select Project XML File dialog box, navigate to and select the XML file.
4. Click **Open**.
5. In the New Project Step 2 dialog box, click **Finish**.

New Project (Step 3)

By default, HIPS and SIPS sets the *Select UTM Zone Automatically* check box to select the UTM zone automatically. (The country and zone options are dimmed).



To set a different projection for your project:

1. Clear the *Select UTM Zone Automatically* check box.
2. Select a country or area from the *Group Name* list box.
3. Select a zone from the *Zone* list box.

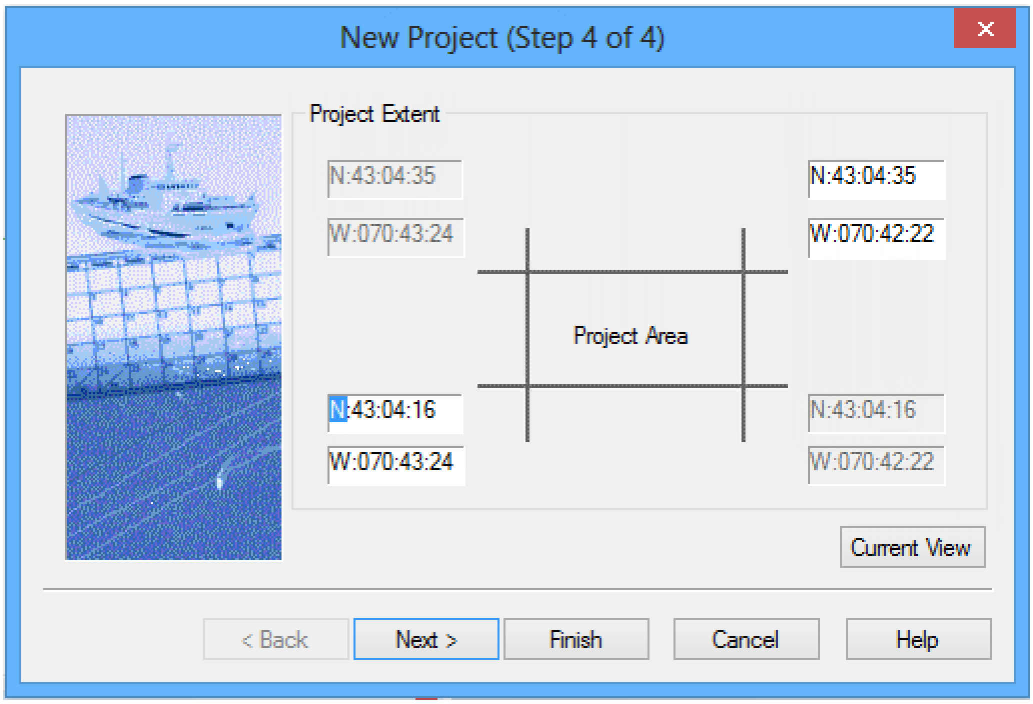
The Projection key name is automatically displayed once the country and zone have been selected.

NOTE: The world-wide map projection coordinate systems are defined in the file …\Hips\System\mapdef.dat.

4. Click **Next**.

New Project (Step 4)

Use this dialog box to set the geographic coordinates (degrees-minutes-seconds) for the project area.



By default, the project extent is set to the entire area of the earth.

If you want to use the geographic coordinates of a currently open project, at the zoom level currently displayed in the Display window:

1. Click **Current View**.

To manually enter the geographic coordinates for the project area:

1. Select a project extent box (e.g., W) and type the coordinates, or

   select the degree, minutes or second field box, and use your arrow keys to change the values.

2. Click **Finish** to complete the creation of the new project.

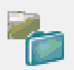
The new project has been created with Project-Vessel-Day folders according to the options selected in the wizard.

Open Projects

Projects in HIPS and SIPS can be opened using the Open or Open Project command.

The Open command will display any type of files supported in hips, including *. HIPS file.

The Open Project command filters the file type to open so that only *.hips and *.hpf files are displayed.

Menu	File > Open Project
Tool	
Key	<Ctrl + O>

1. Select the Open Project command.

The Open dialog box is displayed.

2. Select a project file.
3. Click Open.

The track lines associated with the project are opened in the Display window. The Project/Vessel/Day/Line file tree is displayed in the Project window.

The open Project command will also open *.HPF files, from projects created in earlier versions of HIPS and SIPS. In this case, the project will automatically be updated use the new *.hips database file.

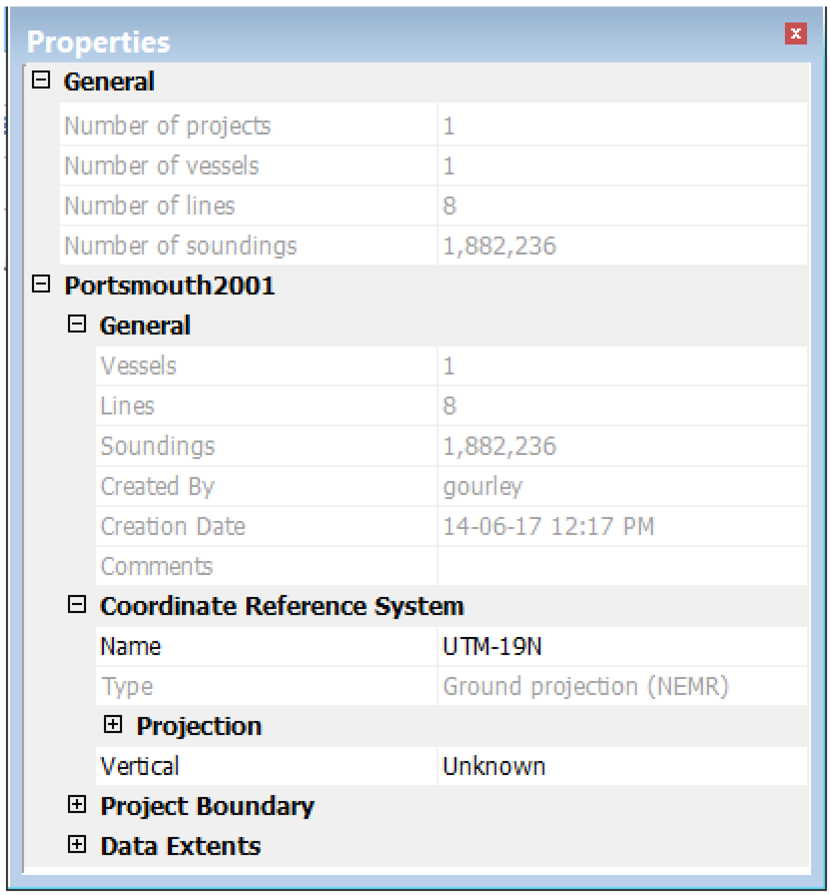
View Project Properties

Essential information for open projects can be viewed in the Properties window.

1. Select the HIPS Data layer in the Layers window.
2. Open the Properties window.

Menu	Window > Other Windows > Properties

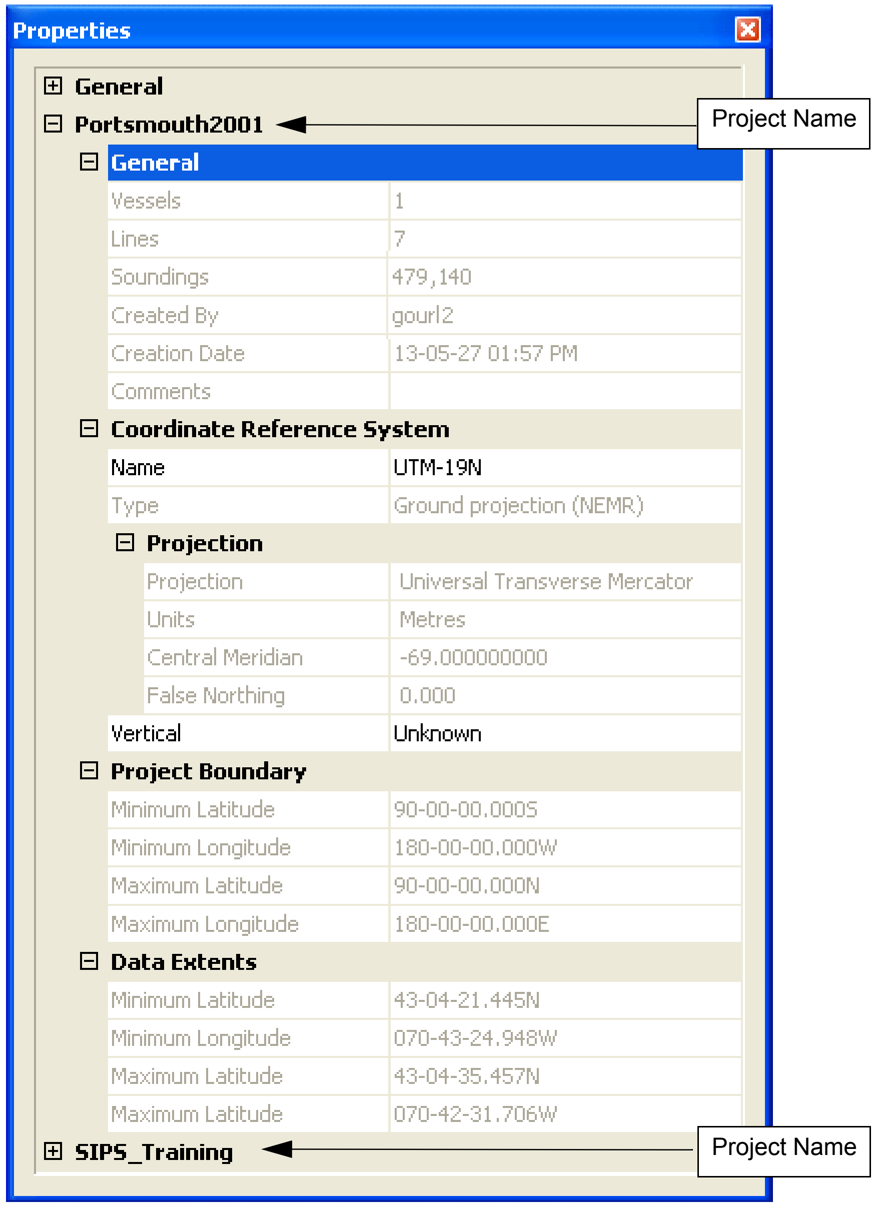
The Properties for all open projects are displayed, as in the examples below.



The General section at the top shows the total number of open projects, vessels, lines and soundings. These fields are read-only. The properties of each open project are listed below these general statistics.

With the exception of the field for the Name of the Coordinate Reference System and for the Vertical Reference System, project properties are read-only.

The properties of each open project are listed below the General statistics.



Properties	Function
Project	Name of first open project
General	
Vessels	Number of vessels in project
Lines	Number of lines in project
Soundings	Total number of soundings in the project.
Created By	Name or initials of project creator

Properties	Function
Creation Date	Date the project was created
Comments	
Coordinate Reference System	
Name	Identifies the coordinate reference system used in the project. Click in the field to open the Select Projection dialog box.
Type	Type of system, e.g., ground.
Projection	Type of projection, e.g. UTM
Units	The units of measurement of the coordinate reference system.
Central Meridian	The line of longitude used as Central Meridian in the Projection.
False Northing	The value applied as an offset to all northings.
Vertical	Identifies the vertical datum, if known. Click in the field to select a reference from the drop-down list.
Project Boundary	
Minimum Latitude	Coordinates of the geographic extents of the project
Minimum Longitude	
Maximum Latitude	
Maximum Longitude	
Data Extents	
Minimum Latitude	Coordinates of the extents of the data in the project
Minimum Longitude	
Maximum Latitude	
Maximum Longitude	
Project	Name of next open project

To change the projection for a project, use the Change Projection command on the View menu.

Set Line Properties

Ship track and towfish track lines have default display colours. Colours distinguish track lines which have been Merged from those that have not. Towfish line colours identify lines that have been slant range corrected and those which are raw side scan data only.

These default colours are set in the Display tab of the Tools > Options dialog box. See “Display window” on page 643 of the Reference guide.

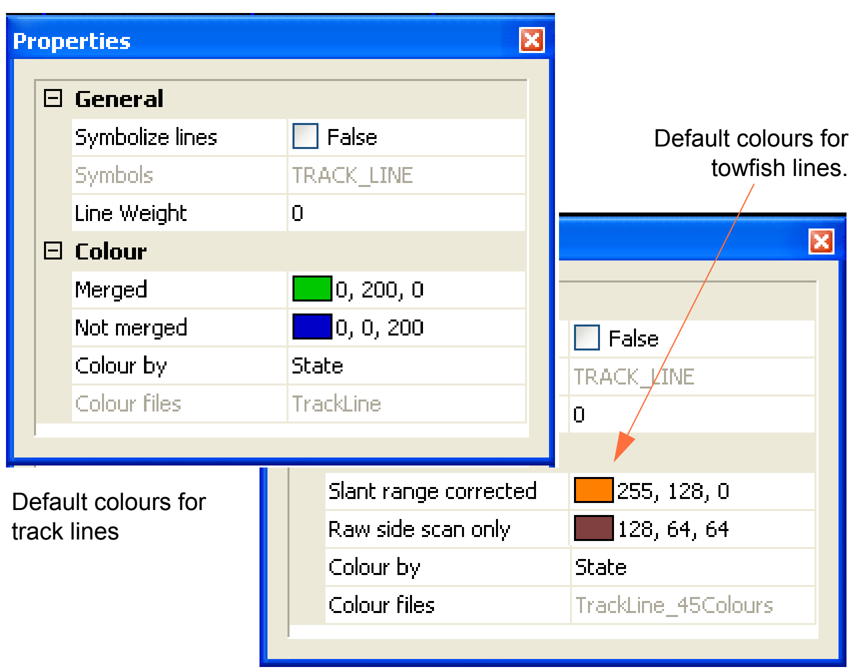
You can change the display colours and symbology for the lines in a session and save to a session file.

To change the colour and symbology of ship and towfish track lines:

Menu	Window > Other Windows > Properties

1. Select the Ship Track Lines or Towfish Track Lines layer in the Layers window.
2. Open the Properties window.

Shown below are the default properties of the lines layers.



General

These General properties apply to both ship track and towfish track lines.

1. Set the *Symbolize Lines* field to “True” to display the lines according to the symbolization specifications of a selected feature code.

    Select a feature code from the *Symbols* drop-down list. (The default symbolization is TRACK_LINE.)

2. Type a *Line Weight* value to set the size of a selected line in the Display. (A line weight can only be set if *Symbolize lines* is set to “False”).

Colour

Ship track lines and Towfish track lines have colour properties based on status as:

- Ship Track Lines: “Merged” and “Not merged”
- Towfish track lines: “Slant range corrected” and “Raw side scan only”

These default colours can be changed by highlighting the field and selecting a new colour from the drop-down list.

Lines can also be coloured by sound velocity profile or by tide file applied. Lines can also be displayed with a different colour for each line.

To change the colour for ship track or towfish lines:

3. Select an option from the drop-down list in the *Colour by* field.

    State is the default setting for both kinds of lines, e.g., towfish line is displayed in colours set in the Options for *Slant range corrected* and *Raw side scan only* status.

    SV Profile colours the ship track lines based on the sound velocity profile applied to the lines.

    Tide colours the track lines based on the tide zone file applied to the ship track lines.

    Multiple colours uses a set of 10 distinct colours to colour the track lines or the towfish lines, depending on which is selected. This can be useful if data was logged as multiple files along the same run line.

If you select to colour by multiple colours, you can also select which colour file to apply from the drop-down list in the *Colour files* field.

These settings are saved when you save your session, and will be applied when the session is re-opened.

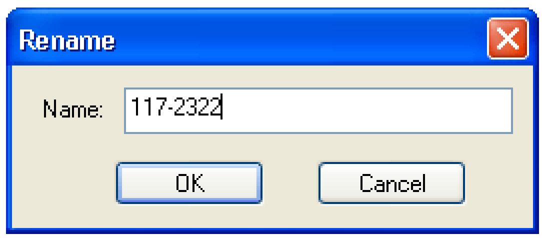
Rename Day and Line Folders

You can rename existing Day and Line folders after raw data has been converted to HIPS/SIPS format.

Menu	Edit > Rename
Pop-up	Rename

1. Select a Day folder or Line folder in the Control window, or select a track line in the Display window.
2. Select the Rename command.

The Rename dialog box is opened with the selected file or folder name displayed.



3. Enter a new name in the box.
4. Click **OK**.

The changed file name is displayed in the Control window.

Archive Project

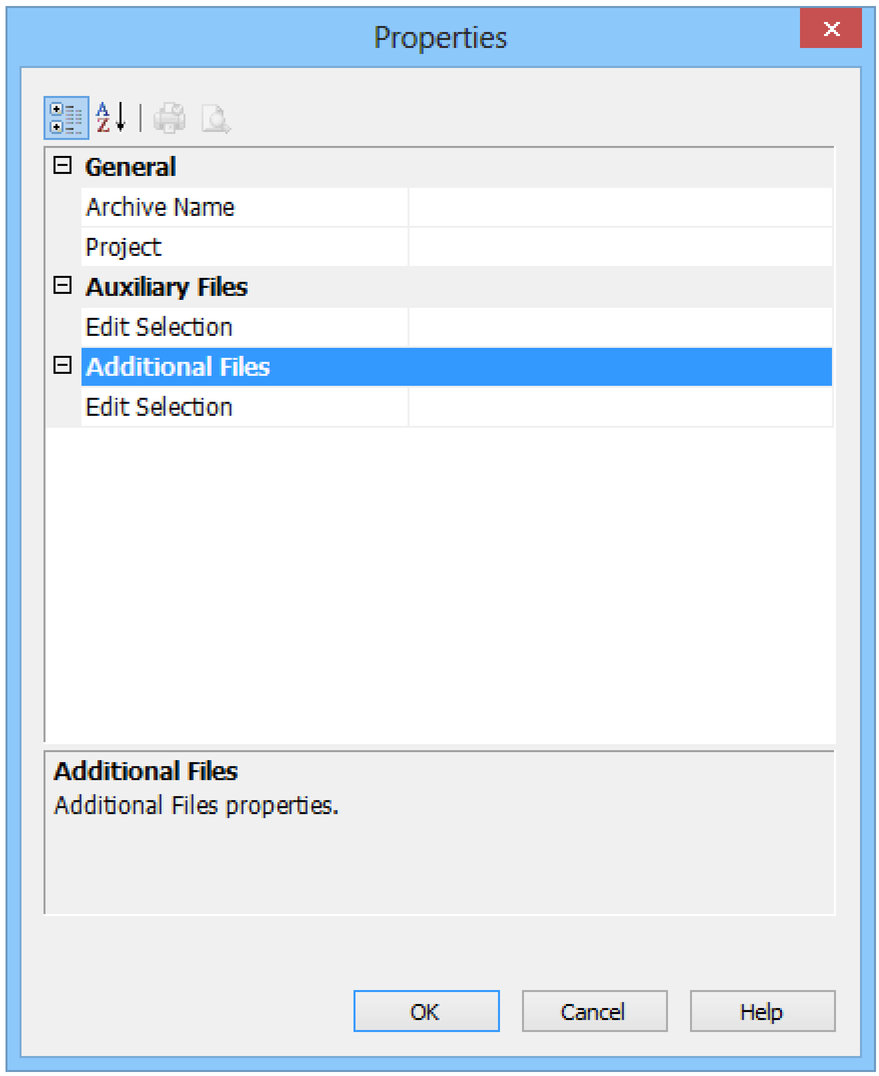
A project and its related data can be saved to a compressed format (*.ZIP) using the New Archive command.

To archive a project:

Menu | File > New > Archive

1. Select the New Archive command.

The archive Properties dialog box is displayed.



When you click in any of these property fields, a **Browse** button is activated. Use **Browse** to locate files.

2. Set paths and select files, as follows:

Field name	
General:	
Archive Name	1. Click **Browse** to open the Save As dialog box. 2. Type a name and set a destination directory for the archive file and click **OK**.
Project	1. Click **Browse** to open the Select Project dialog box. 2. Double-click on Default to open the project tree. 3. Select the project and click **OK.**
Auxiliary Files: (Related files necessary for the successful processing of data, for example, tide, sound velocity, delta draft, delayed heave files, etc.)	
Edit Selection	1. Click **Browse** to open the Select Files dialog box. 2. Click **Add** to open the standard Select Files dialog box and locate the auxiliary files to be included. 3. Repeat until list of files is complete. 4. Click **OK**. Use **Remove** and **Clear** to adjust your selection.
Additional Files: (Data that is associated with the project, but is not necessary for processing data, for example, background images and associated charts.)	
Edit Selection	1. Click **Browse** to open the Select Files dialog box. 2. Click **Add** to open the standard Select Files dialog box and locate the additional files. 3. Repeat until list of files is complete. 4. Click **OK**. Use **Remove** and **Clear** to adjust your selection.

3. Click **OK** to archive the file selected to the set destination.

The ZIP file is created and contains these folders:

- VesselConfig
- Project
- PreProcess
- Background.

4

Convert Data

Use the Data Conversion wizard to convert various types of survey data into HDCS-specific format.

In this chapter...

Converting Files to HIPS Format

HIPS and SIPS files are created from survey data using the Conversion Wizard.

Survey data in ASCII format can also be converted using the Generic Data Parser. See "Generic Data Parser" on page 93 of the Tools guide.

Data that is converted to HIPS format may be in varying stages of completion, depending on the data format options. Survey data may or may not have been corrected for factors such as heave/pitch/roll or sound velocity. Such correction can be applied during the Merge process and SVP Correction.

All HIPS and SIPS data is organized in a Project/Vessel/Day/Line directory structure. If you organize the raw data files according to this structure, then you can convert line data from entire Project or from multiple Vessel and Day folders contained in a common Project folder. Use the File Selection Type option in Step 2 of the wizard to do this (see "Select Files for Conversion" on page 84).

When entire project data has been converted, a Project/Vessel/Day/Line directory is created for the converted data. If there is already a Project folder that is identical to the Preprocess folder, then the line data in the existing folder is over-written with the new data.

Formats that can be converted to HIPS and SIPS:

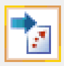
File Format	File extensions	Data types processed in HIPS and SIPS	GeoCoder Support
Atlas	*.sda, *.asd, *.acf, Surf	multibeam, single beam	No
Chirpscan3D	*.brf	multibeam	No
CMAX	*.cmx, *.cm2	multibeam, single beam	No
Coda	*.*	side scan	No
Edgetech	*.*	side scan	No
EIVA	*.sbd (generic EIVA binary format) or XTF	multibeam, single beam	No
Elac	*.*	multibeam, single beam, side scan	No
Furuno	*.*	multibeam, single beam, side scan	No
GeoAcoustics	*.rdf	Dual frequency side scan	
GSF (Generic Sensor Format)	*.gsf, *.*	multibeam	**Yes** (all formats)
Hawkeye	*.bin	LIDAR	No
Hypack	*.hsx	multibeam, single beam, side scan, sweep	**Yes** (HSX/81X, HSX/7k, HSX/R2S, HSX)
Imagenex	*.83p, *.83m	multibeam, side scan	No
Kongsberg (Simrad)	*.all, *.out, *.raw, *.depth, *.mb57	multibeam, single beam, side scan	**Yes** BA/TS (for *.all data only)
Kraken	*.til	multibeam, side scan	No
LADS	*.*	LIDAR	No
LAS (a public binary format for LIDAR data)	*.las	LIDAR	No
Marine Sonics	MS Tiff files (*.*)	side scan	No
ProSAS	*.img	swath	No
QMips	*.*	side scan	No
Teledyne Reson PDS	*.pds, *.s7k	multibeam	**Yes** (s7k only) BA/TS
Scripps	*.*	swath, side scan	No
SDF	*.sdf	side scan	No
Seabeam	*.*	multibeam	No
Seafalcon	*.*	multibeam	No
SEGY	*.*	side scan	No
SHOALS	*.out, *.hof, *.tof	LIDAR	No

File Format	File extensions	Data types processed in HIPS and SIPS	GeoCoder Support
Simrad - see Kongsberg			
Spawar	*.dat	swath	No
Swathplus	Submetrix SXP files *.sxp, *.sxr, *.sxi	multibeam, side scan	No
Teledyne	*.tdy, *.*	swath, side scan	No
UNB	swathed files (*.merged) Reson *.*	swath	No
Winfrog	*.*	single beam	No
XTF	*.xtf	multibeam, single beam, side scan, sweep	**Yes** (Reson, R2Sonic, various side scan)

BA = Beam Average

TS = Time Series

Select Data Format

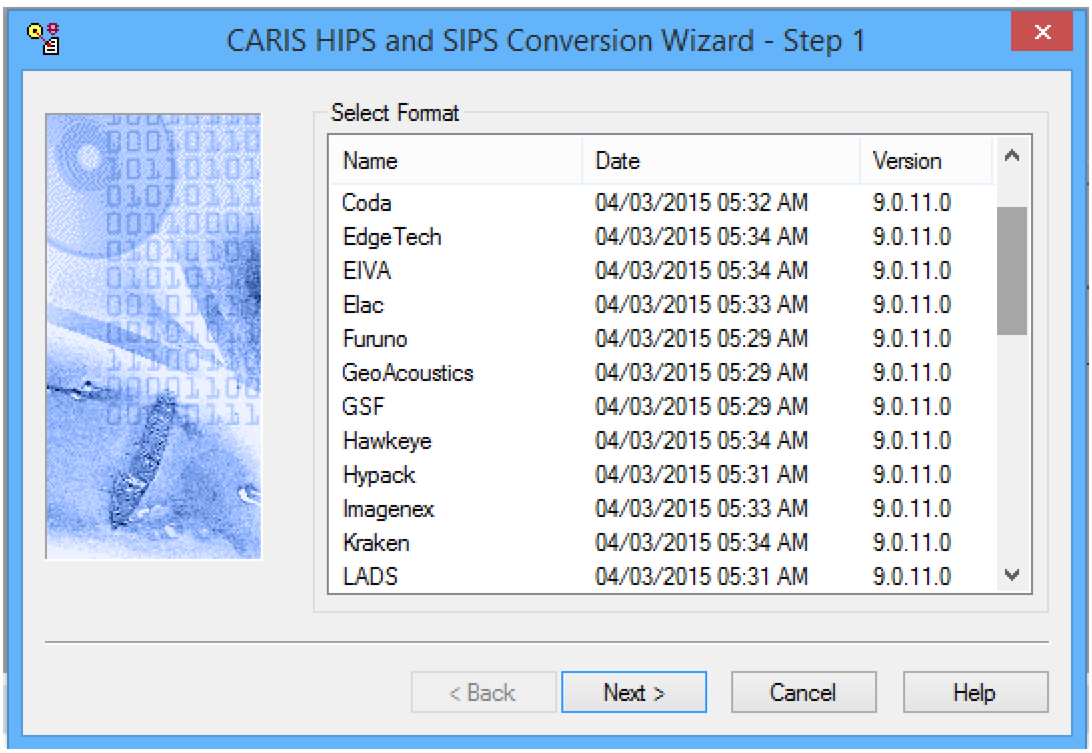
Menu	File >Import > Conversion Wizard
Tool	

To start the conversion process, activate the HIPS Conversion Wizard.

1. Select the Conversion Wizard command in the HIPS and SIPS main interface.

The first Conversion dialog box is displayed.

The Step 1 dialog box lists the data acquisition systems for which data can be converted to HIPS and SIPS files. (This list is created dynamically from all format libraries with the name `convert_*.dll` in the Hips\Bin directory.)



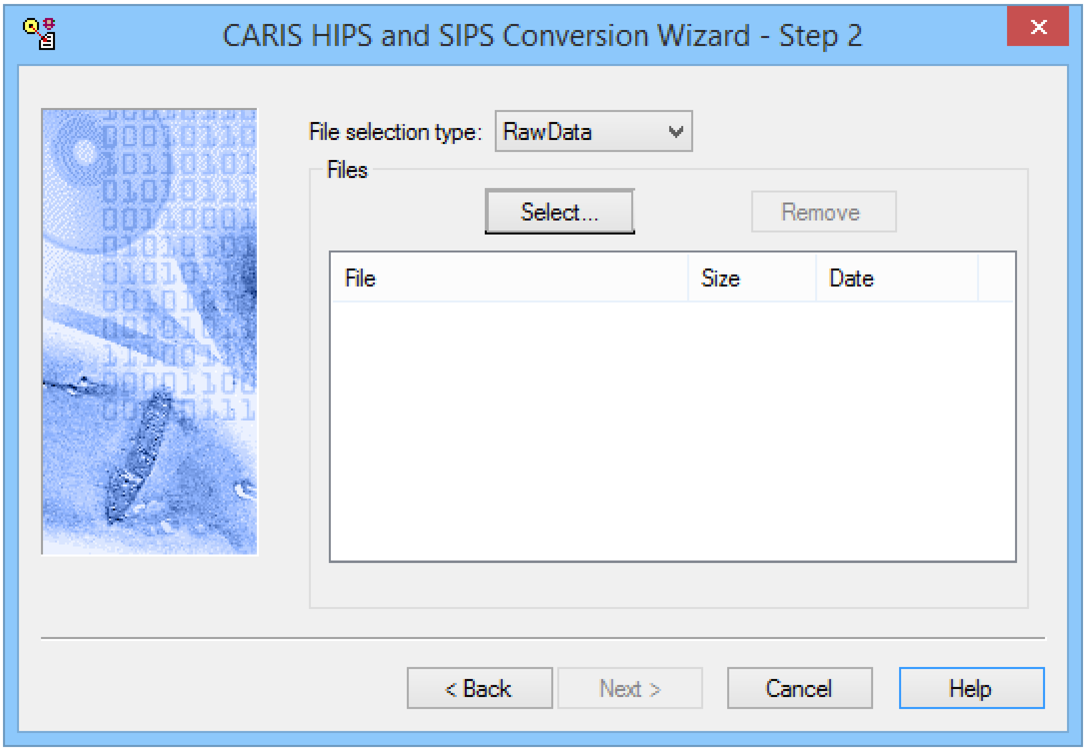
1. Select the survey data type to be converted by clicking the appropriate name.
2. Click **Next**.

"Select Files for Conversion" on page 84

"Import Options" on page 85

Select Files for Conversion

In most converters this dialog box will appear so you can select the files to be converted. Some formats have other import options to set before selecting data files. (See “Import Options” on page 85.)



1. Select the appropriate *File selection type* from the drop-down list. The default is Raw Data.
2. Click **Select** to choose the files you want to convert.

If you are converting multibeam data with the Hypack converter, select HSX files, not RAW files for conversion.

If you have selected Raw Data as the *File selection type*, clicking **Select** will open the Select Files dialog box. (The default directory for Raw Data is …\Hips\Preprocess.)

If you select Project, Vessel or Day as the *File selection type*, **Select** will open the P/V/D tree structure so you can browse for the folder containing the files you want to convert.

3. [Optional] To remove a file from the list, select it and click **Remove**.
4. Click **Next**.

> Raw data being converted into HIPS and SIPS can also be located on a read-only media such as a CD-ROM, except for Atlas SURF data, which at this time, does not support conversion from read-only files.

Import Options

Before selecting the data to be converted, some file formats have import options that must be set.

"Create or Update survey lines" on page 85

"Carry Over Raw Data Files" on page 86

"Search for Files with Temporal Overlap" on page 87

Create or Update survey lines

The conversion process creates all new data files within the project structure. However, if line directories already exist within your project, they are deleted and replaced with the new survey data.

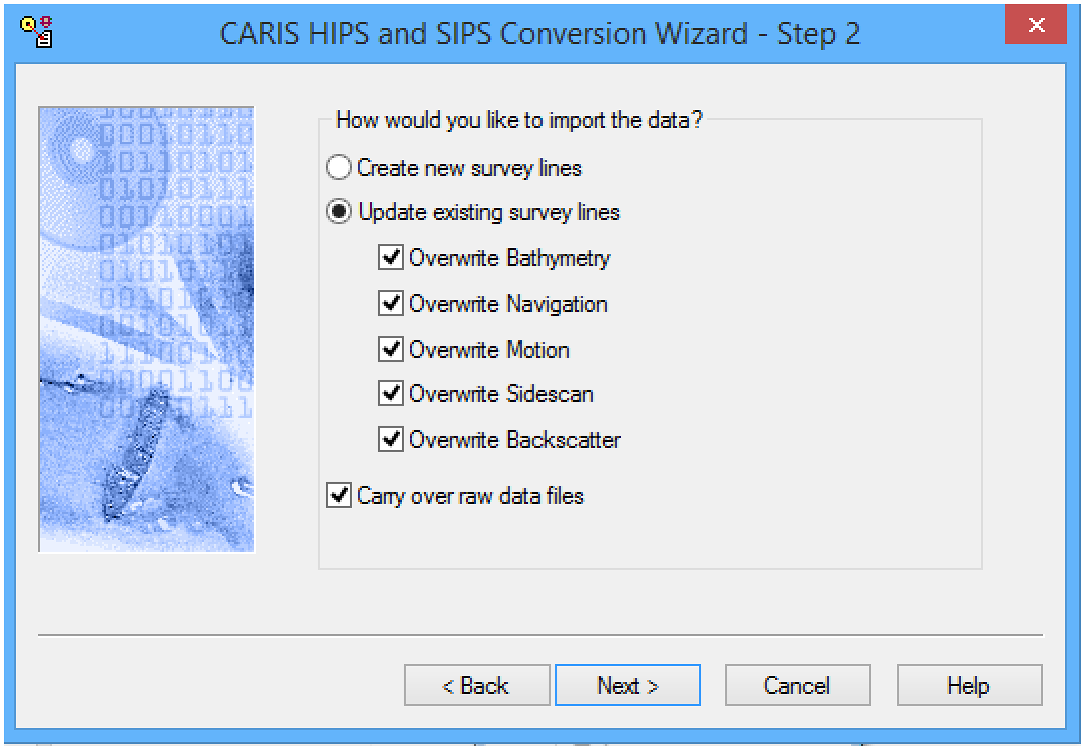
Instead of overwriting all existing data, you can choose to update only certain data. For example, you can opt to update only the navigation information on your processed survey lines, or to add backscatter imagery data to a project, without overwriting any of the rest of your processed data.

At present this option is available for these formats:

- Hypack (HSX/81X)
- Teledyne Reson PDS
- Kongsberg
- XTF

These options are presented in the Step 2 dialog box of the Conversion Wizard, before the actual data files are selected.

1. Select *Create new survey lines* to save new data to the project, overwriting any existing data. This is the default setting.
2. Select *Update existing survey lines* to choose which kinds of data to replace in your existing project. By default, all the types are set to overwrite your existing data. be overwritten. Clear the check box(es) for the data you do not want overwritten. (If the lines don't exist, no data is converted.)
   - *Overwrite Bathymetry*: When checked on, this option will cause the SlantRange, ObservedDepths and ProcessedDepths data files to be replaced by the converter.
   - *Overwrite Navigation*: When checked on, this option will cause the Navigation and SSSNavigation data files to be replaced by the converter.



- *Overwrite Motion*: When checked on, this options will cause the Gyro, Heave, Pitch, Roll, DeltaDraft, CableOut, SensorHeight data files to be replaced by the converter.
- *Overwrite Sidescan*: When checked on, this option will cause the `SideScan` and `SSSProcessedSideScan` data files to be replaced by the converter.
- *Overwrite Backscatter*: When checked on, this option will cause all 'imagery' related data files to be replaced by the converter (i.e. all data files needed for GeoCoder processing).

Carry Over Raw Data Files

The conversion process for some data formats can copy the original data files to the processed folders. This process is optional. By default, raw data files are not carried over. However, the location of the raw files is referenced in the Observed Depths file.

If you select the *Carry Over Raw Data Files* check box, your data will be converted and a copy of the raw files placed in the line folders.

3. Select *Carry Over Raw Data Files* to copy raw data files to the line folders.

If a process uses raw data, and the raw data has been carried over, HIPS will use the files in the HDCS line folder.

If the raw data file is not carried over then HIPS will search the referenced location of the raw files. If the data is not found, you will be prompted to have HIPS search for it, or to search for it yourself.

Search for Files with Temporal Overlap

When converting Kongsberg or XTF data the converter is set to scan the entire directory containing the files selected to be converted. This scan checks for navigation or motion data with time overlaps. This is done in order to synchronise the reference time stamps properly, in cases where single survey lines are split up into multiple pieces by the acquisition software.

This scanning can be time-consuming, particularly with large data sets. However, if this synchronizing is not done when a line has been split, the data in the converted line segments may not overlap correctly.

This synchronizing function is enabled, by default, with a setting of 10 seconds.

To change the overlap value:

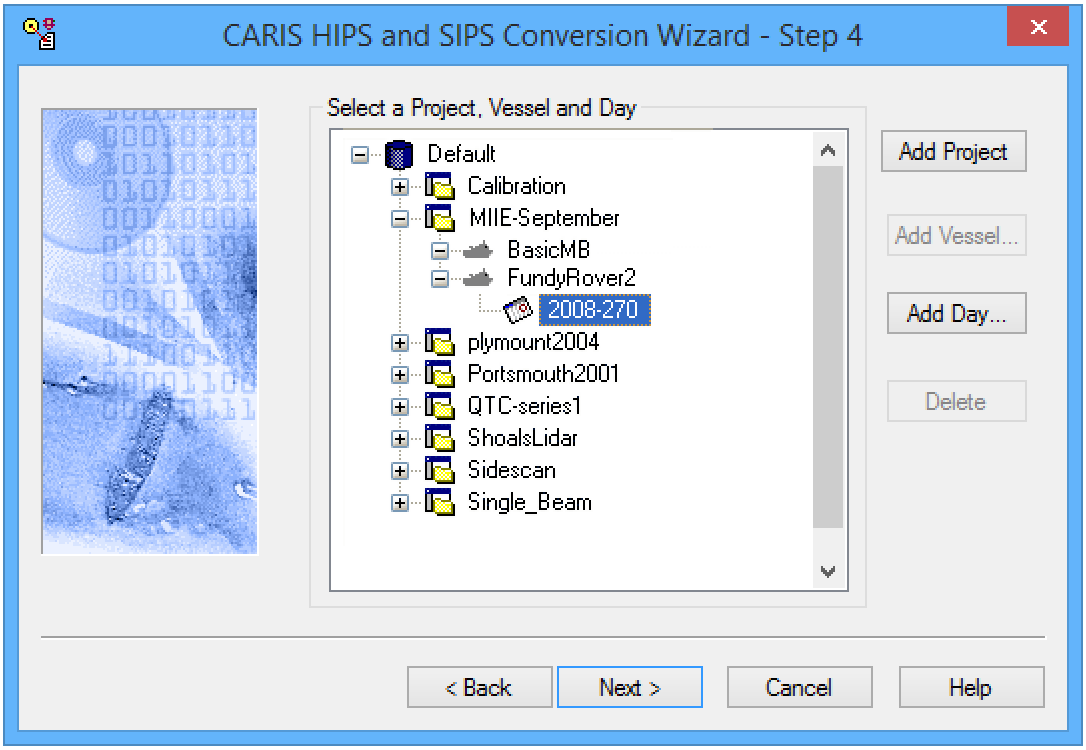
1. Type a value in seconds.

To turn off this function:

2. De-select the *Search directories for files with temporal overlap* check box.
3. Click **Next**..

Select Project, Vessel and Day

Use the Step 3 dialog box to select the Project, Vessel and Day into which your converted data will be saved.



1. Click the Expand (+) icon to expand the Project file tree so that a Day folder is visible.
2. Click the Day folder so it is highlighted.
3. Click **Next.**

To select an existing project that is not listed in the current file tree:

1. Right-click in the data tree area of the dialog box.
2. Select Connect To… from the pop-up menu.
3. Name the new project folder and use **Browse** to set the path to the new data directory.
4. Click **OK**.

To create a new project:

1. Click **Add Project** and follow the steps to “New Project” on page 63.

Add Vessel or Day

You can also use this dialog box to add Vessel or Day folders to an existing project, or to delete empty folders.

1. To add a vessel to the selected project, make sure the project directory is highlighted and click **Add Vessel**.

The Available Vessels dialog box is displayed.

2. Select a vessel folder and click **OK**.

The new Vessel folder is displayed in the selected project directory.

To add a Day folder to the project:

1. Click the Vessel folder to expand it and click **Add Day**.

The Calendar dialog box is displayed.

2. Select a year/month/day from the calendar, and click **OK**.

A Day folder with the selected date is stored in the Vessel folder.

Remove folder

3. To remove an empty Day or Vessel folder from the wizard, highlight the folder and click **Delete**.

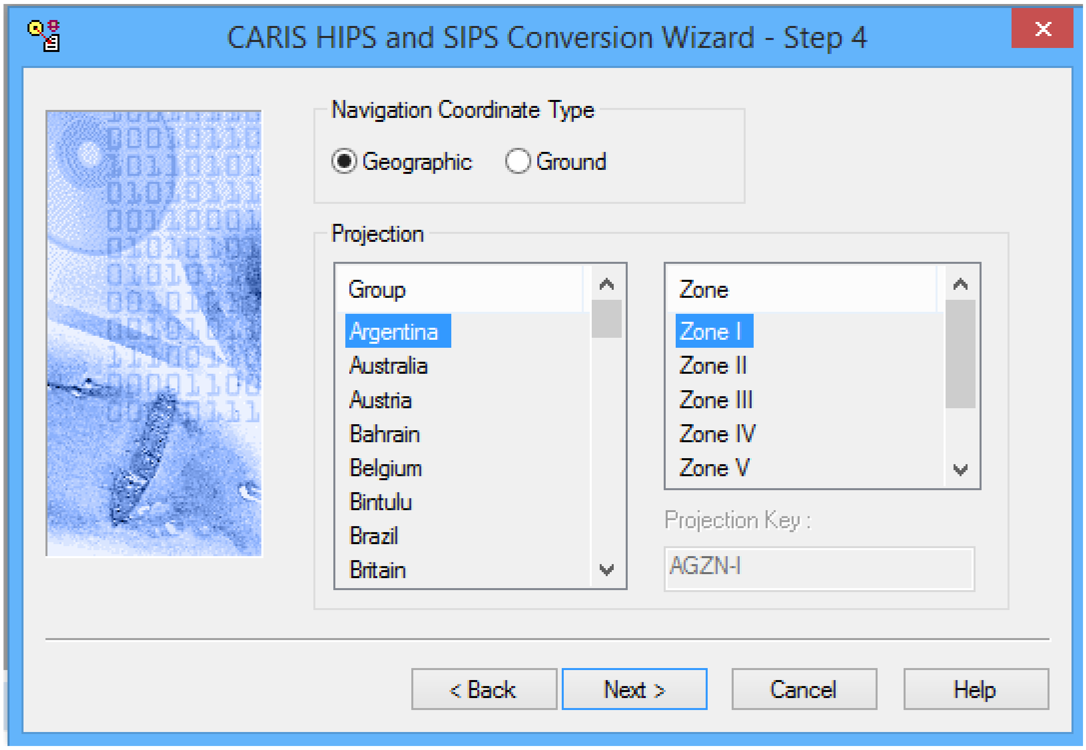
To add a Project to the root directory:

1. Click Add Project to open the New Project wizard, and follow the steps in the wizard.

When you have finished creating a new project for your data, you will be returned to the Conversion wizard to continue the conversion process.

Set Navigation Coordinate System

This step is to define the type of coordinates that were used to record the navigation data in the raw data files.



To indicate the type of coordinate system that was used for navigation data during data logging,

1. Select one of the following options:
   - *Geographic*: navigation data is recorded as latitude and longitude coordinates.
   - *Ground*: navigation data is recorded as eastings and northings.

If you select *Ground*, complete the following steps:

2. Select an area from the *Group* list.
3. Select a zone.
4. Click **Next**.

Setting Geographic as Navigation Coordinate Type for Hypack data:

Hysweep format (HSX) does not store Geographic coordinates and GPS Height information. If recorded, this data is stored in RAW files.

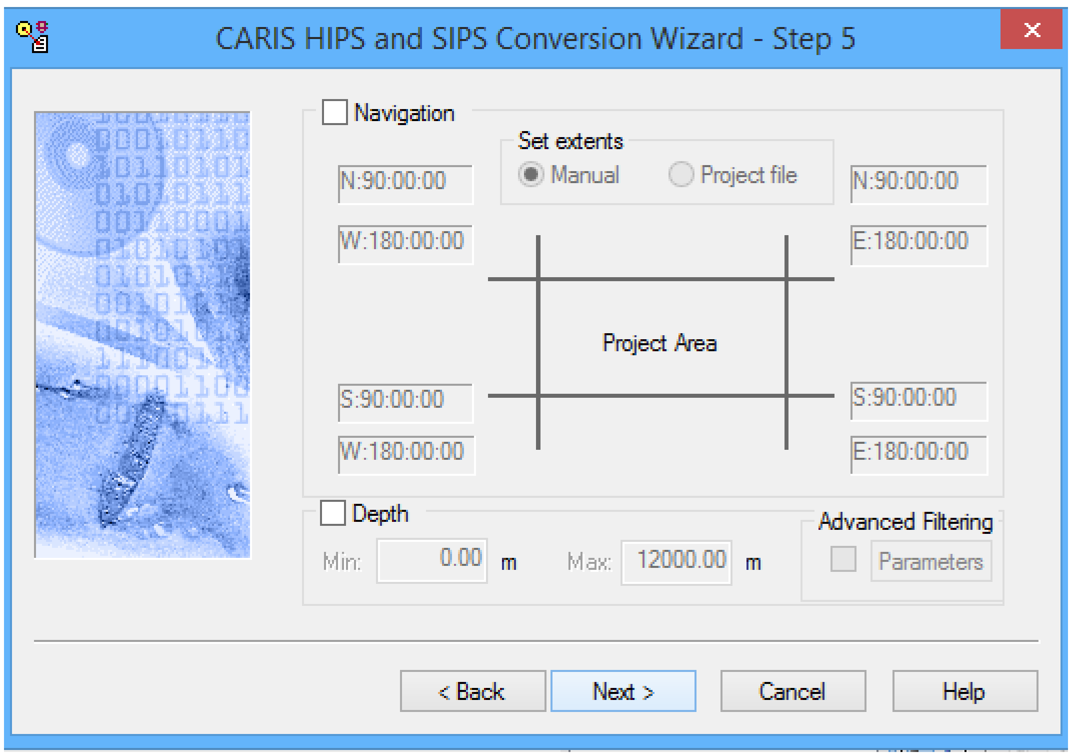
When both the Geographic option and Multibeam options are set during conversion, the converter will search for *.raw files in the same directory with the same line name to get this information.

Select Filters and Extents

During the conversion process, some soundings can be automatically rejected due to disabled beams. This is controlled by the beam status settings in the vessel configuration.

Also, several multibeam formats pre-flag some soundings as being failed detects. These soundings are also automatically rejected during conversion.

This dialog box defines basic navigation and depth filters to automatically reject extremely large errors in the recorded navigation and depth data.



1. Select the *Navigation* check box to enable the navigation filter.

The extents of the filter can be defined either by setting them manually in this dialog box, or by using the extents as defined in the project selected at Step 3 or 4.

2. Select either *Manual* option or the *Project File* option.

If you selected the *Manual* option, enter the extent coordinates in the fields. You can use the <Tab> key to move between fields and use the arrow keys to move between values in a field.

The HIPS converter for reading Klein SDF data files has been enhanced to read the bathymetry data from the Klein data format 5000 V2. As a result of this enhancement, the filtering capabilities in step 5 of the Conversion Wizard are now enabled for this conversion. These settings are functional only for the format 5000 V2 data, as no other formats processed by the SDF converter currently contain bathymetry.

3. Select the *Depth* check box to enable the depth filter.
4. Enter the minimum acceptable depth.
5. Enter a maximum acceptable depth.

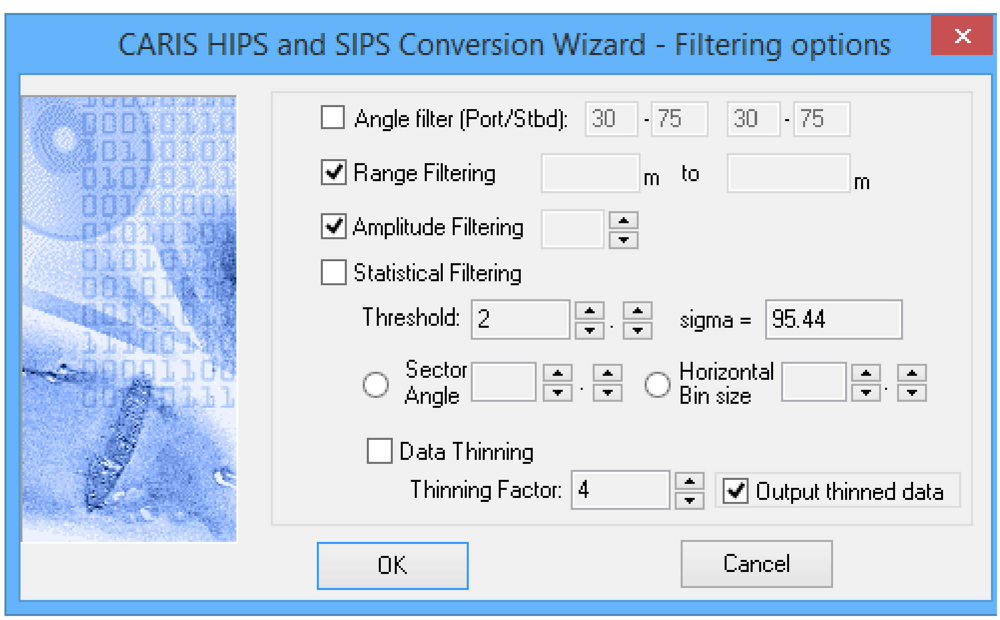
Advanced filtering is available for certain multibeam or multi-transducer data: GeoAcoustics, Klein SDF, EIVA and Hypack. If filtering is not available for the type of data you are converting, these options will be greyed-out.

To use advanced filtering:

6. Make sure that the *Depth* check box is enabled.
7. Select the *Advanced Filtering* check box and click **Parameters** to set advanced filtering options.

Advanced Filtering

Certain data can be filtered so that only the best quality data is imported to HIPS. The filter is applied to port and starboard beam angles. Currently, advanced filtering is only available for GeoAcoustics, Klein SDF, EIVA and Hypack formats.



1. Select *Angle filter (Port/Stbd)* and type a range of beam angles.

The *Range Filtering* option converts soundings within a specified distance (in metres). Soundings outside of this range are not converted.

2. Select the *Range Filtering* check box to implement this option.
3. Click the up or down arrow buttons to select a minimum and maximum distance.

The *Amplitude Filtering* option filters soundings according to amplitude value. For each ping (port and starboard pings are handled separately), the min/max amplitude values are obtained, and samples that fall below the selected percentage (0-50%) are rejected.

4. Select the Amplitude Filtering check box to implement this option.
5. Click the up or down arrow buttons to select a percentage value for rejecting soundings.

The *Statistical Filtering* option controls which soundings in each swath are considered for conversion. This option calculates the mean depth and standard deviation within a swath sector or horizontal bin. It then prevents any soundings that fall outside a multiple of the standard deviation from being converted.

If Statistical Filtering is used, then the mean within the sector is re-computed. Ultimately, soundings are sorted by their residual from the mean and those closest to the mean are converted. The actual number of soundings converted from within each sector is determined by the thinning factor (if used).

6. Select the *Statistical Filtering* check box to implement this option.
7. Determine a *Threshold* (multiples of the standard deviation) value by clicking the up and down arrow buttons.

The equivalent confidence value is displayed as a percentage.

The filtering and thinning methods are executed within a sector angle interval or a horizontal bin size. The sector angle option divides the swath into sectors according to degree angles while the horizontal bin size divides the swath into horizontal sectors based on a specified across-track distance.

Vessel motion and transducer mounting angles are considered when sector angle and horizontal bin locations are determined.

1. Select the *Sector Angle Interval* option and choose a degree level (to a maximum of 10º) by clicking the up or down arrow buttons.
2. As an alternate method, Select the *Horizontal Bin Size* option choose an across-track distance by clicking the up or down arrow buttons.

The thinning factor reduces the number of soundings converted on a per swath basis. The thinning factor uses a power of two reduction control (1/2, 1/4, 1/8, etc.) so that one of two, one of four or one of eight soundings can optionally be converted per swath sector.

3. Select the *Data Thinning* check box to implement the thinning option.
4. Select a *Thinning Factor* value by clicking the up or down arrow buttons.
5. Select *Output thinned data* to convert only non-rejected data.

You have the option not to convert soundings which have been eliminated because of data thinning. Although by doing so, a mismatch is created between the converted HIPS data and the original data file. (Since multi-transducer data is tied to the settings in the vessel file, the full dataset is always converted).

6. Click **Next**.

Enter Data Parameters

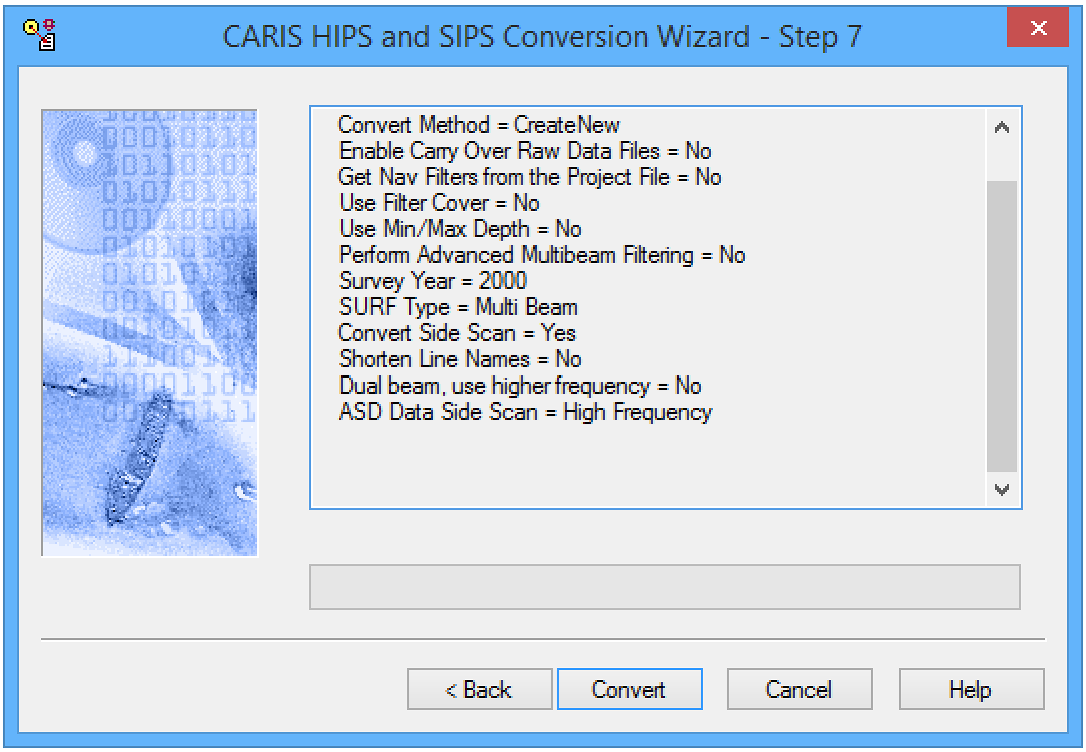
Depending on the data acquisition format you selected in the Step 1 of the wizard, one or more dialog boxes, appropriate to the format, are now displayed. See "Options for Specific Formats" on page 97 for description of options for different raw data formats.

1. Choose the settings appropriate to the data format being converted.
2. Click **Next**.

Convert Data

The final step is to launch the conversion process.

The settings and parameters you have selected are displayed in the dialog box. This information can be copied to a text file, using the Copy command on the right-click menu.

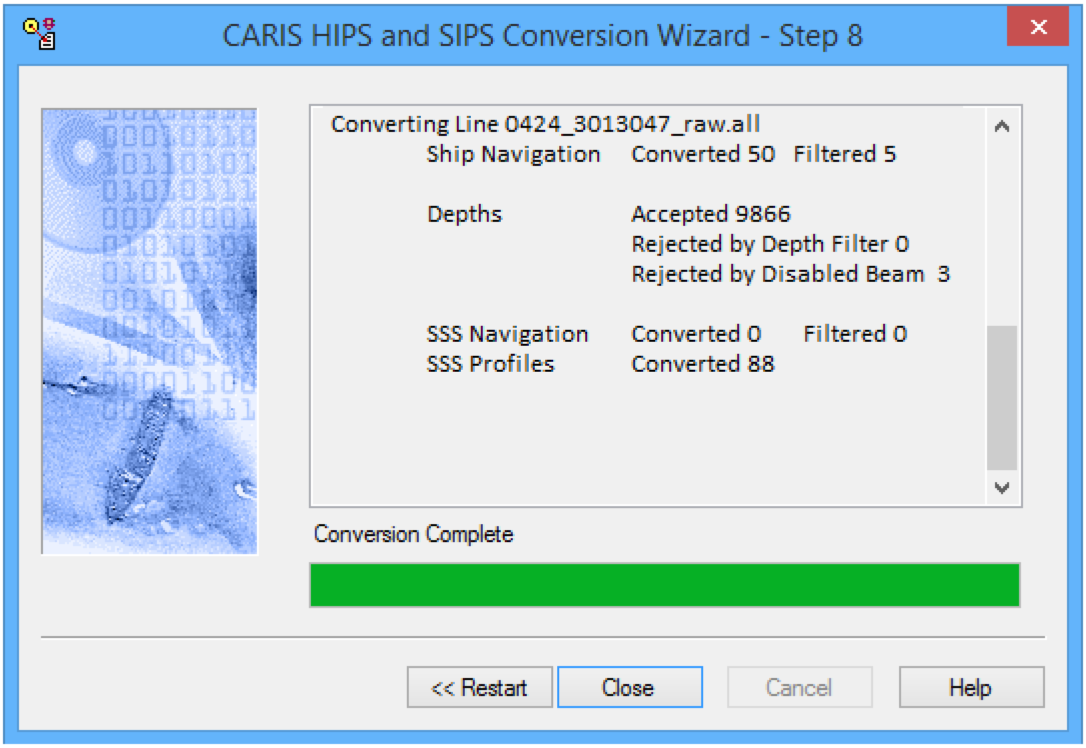


To convert the data using the settings:

1. Click **Convert**.

Conversion progress

A progress indicator bar along the bottom of the box is activated as the files are being processed. As each line is converted, the results of the conversion process are displayed in the dialog box.



This information can also be copied to a text file.

The converted files are saved in the directory you selected in Step 2 of the conversion process (see “Select Files for Conversion” on page 84).

2. Click **Close** to close the wizard.
3. [Optional] to return to the initial step of the conversion process, click **Restart.**

Options for Specific Formats

In some cases, the HIPS Conversion wizard provides import options specific to the format being converted. The number of options depends on the data acquisition format that you select in Step 1 of the Conversion Wizard.

These are the current formats which can be converted:

"Atlas" on page 98	"ChirpScan3D" on page 99
"CMAX" on page 99	"Coda" on page 99
"EdgeTech" on page 100	"EIVA" on page 104
"Elac" on page 105	"Furuno" on page 106
"GeoAcoustics" on page 107	"GSF" on page 110
"Hawkeye" on page 111	"Hypack" on page 113
"Imagenex" on page 116	"Klein (SDF)" on page 117
"Kraken" on page 120	"LADS" on page 120
"LAS" on page 121	"MarineSonics" on page 122
"Navitronics" on page 123	"Qmips" on page 123
"Teledyne Reson PDS" on page 124	"ProSAS" on page 127
"Scripps" on page 127	"Seabeam" on page 127
"Seafalcon" on page 128	"SEGY" on page 128
"SHOALS" on page 128	"Kongsberg" on page 130
"Spawar" on page 132	"SWATHPlus" on page 132
"Teledyne" on page 133	"UNB" on page 133
"Winfrog" on page 133	"XTF" on page 142

Please be advised that 64-bit versions of the following third-party libraries remain unavailable at this time. Therefore, the following capabilities will not be available in the 64-bit version of HIPS and SIPS:

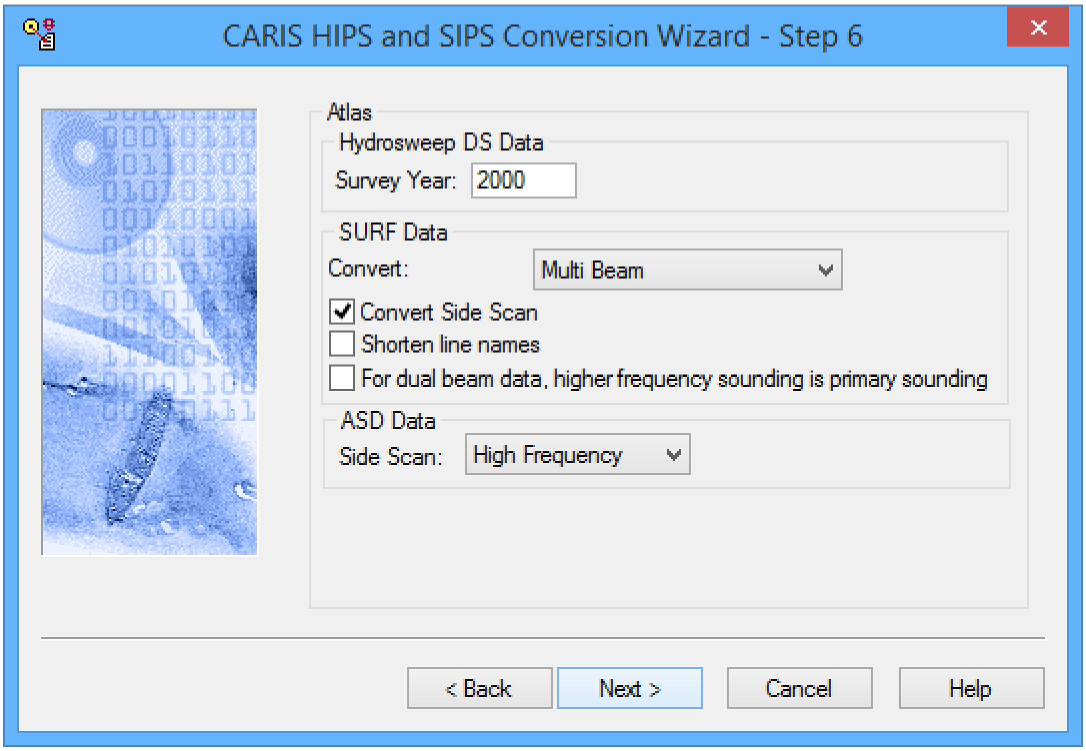
- Navitronics format conversion
- Hawkeye waveform viewer

If any of the above capabilities are required, you will need to use the 32-bit version of HIPS and SIPS.

Atlas

The ASCII versions of Hydrosweep DS files are now supported.

1. Type the year of the survey into the *Survey Year* field. You cannot enter a year prior to 1970. (The default value shown is obtained from the HIPS Day directory into which data is converted.)



2. From the pull-down menu, select one of the following SURF data types to convert:
   - multibeam
   - single beam (low frequency)
   - single beam (medium frequency)
   - single beam (high frequency)
   - dual frequency (low-medium)
   - dual frequency (low-high)
   - dual frequency (medium-high)
3. Click the *Convert Side Scan* check box to include side scan data in the conversion process.

Critical line name information can be fitted into 12 characters provided by the CARIS Source ID attribute for soundings in a CARIS map.

4. Select the *Shorten line names* check box to modify the file name to a 12-character name (YYDDD_HHMMDD).

5. Select the higher frequency data as the primary soundings (if data format is dual-frequency).
6. Select High Frequency or Low Frequency from the drop-down list to convert Atlas ASD side scan data (if available).

The Atlas SURF library cannot be converted from read-only files. Atlas files must be located on a read-write media before they can be converted into HIPS.

7. Click **OK** to continue the conversion process.

Chirpscan3D

There are no options for the Chirpscan3D format.

CMAX

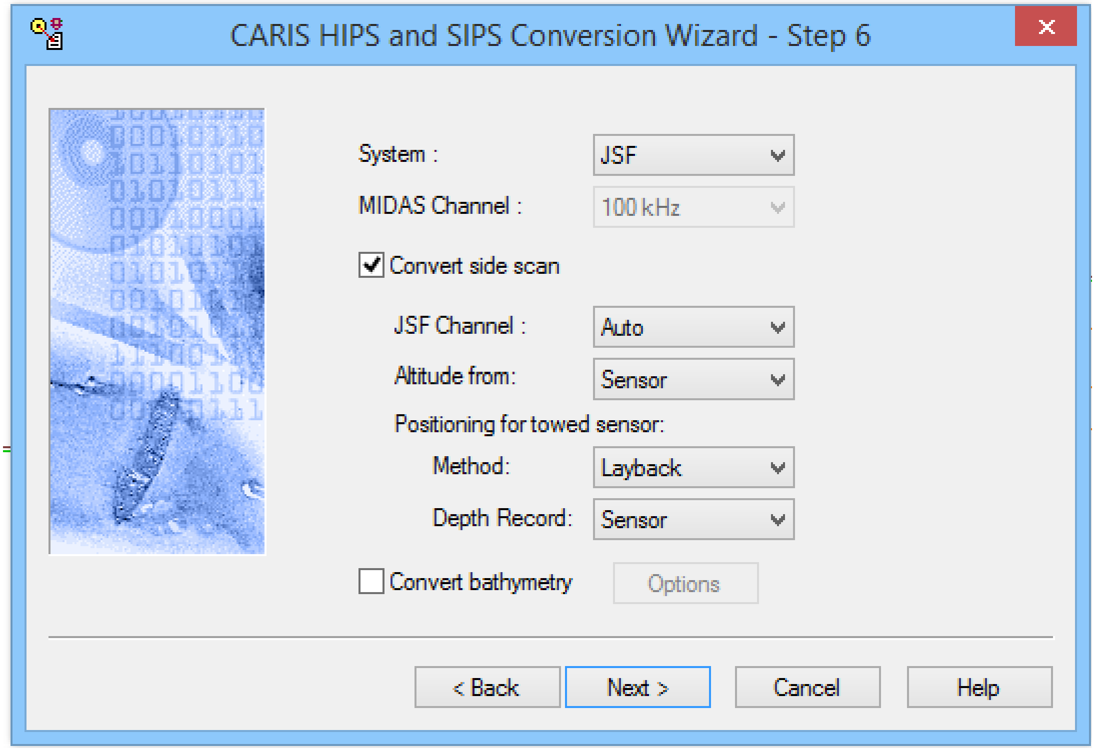
There are no options for the CMAX format.

Coda

1. Select a sonar channel.
2. Click the *Decimate Imagery* check box to apply a decimation routine to the side scan data. Decimation reduces side scan data to 1024 or fewer intensities per side for each swath.
3. Click the *Correct Imagery* check box to increase contrast in the side scan imagery.
4. Select the *Cable Out is Horizontal Layback* option to omit sensor depth during conversion. If there is no sensor depth and the towpoint has a Z-offset of zero, the cable out value is treated as the horizontal layback value during processing.
5. Click **OK** to continue the conversion process.

EdgeTech

Edgetech side scan data is imported in 16-bit format.



1. Select a side scan system from the drop-down list in the *System* field:
   - Midas
   - 260
   - JSF

If you select MIDAS the *MIDAS Channel* options are enabled.

2. Select a sonar frequency channel from the drop-down list:
   - 100 KHz
   - 500 KHz

If you select JSF, the *Convert side scan* options are enabled.

The default setting for the sonar frequency channel is set to Auto, which will automatically convert low frequency data if it exists. If it does not, high frequency data is converted.

3. Select a channel setting from the JSF Channel drop-down list:
   - All data
   - Low Frequency
   - High Frequency
4. If altitude data is present in the JSF data, select the data source:
   - Sensor

- Discover-2

5. Select either Layback or Cable Out from which to convert the JSF data.

If you select Cable Out:

6. Select the source for Depth, either Sensor or Discover-2.

To continue without using bathymetry filtering:

7. Click **Next**.

Convert JSF

If JSF data is being converted, bathymetry conversion and filtering options become available.

To automatically convert bathymetry:

1. Enable the *Convert Bathymetry* check box.
2. Click **Next**.

To set bathymetry filtering options for converting bathymetry data:

1. Enable the *Convert Bathymetry* check box.
2. Click **Options**.

The Bathymetry Filtering Options dialog box is displayed.

See "Bathymetry Filtering Options" on page 134 for settings.

- "Angle filtering" on page 135
- "Amplitude filtering" on page 136
- "Range filtering" on page 137
- "Statistical filtering" on page 139
- "EdgeTech Acquisition Options" on page 101

When all bathymetry filtering options are set:

3. Click **OK** to continue the EdgeTech conversion process.

EdgeTech Acquisition Options

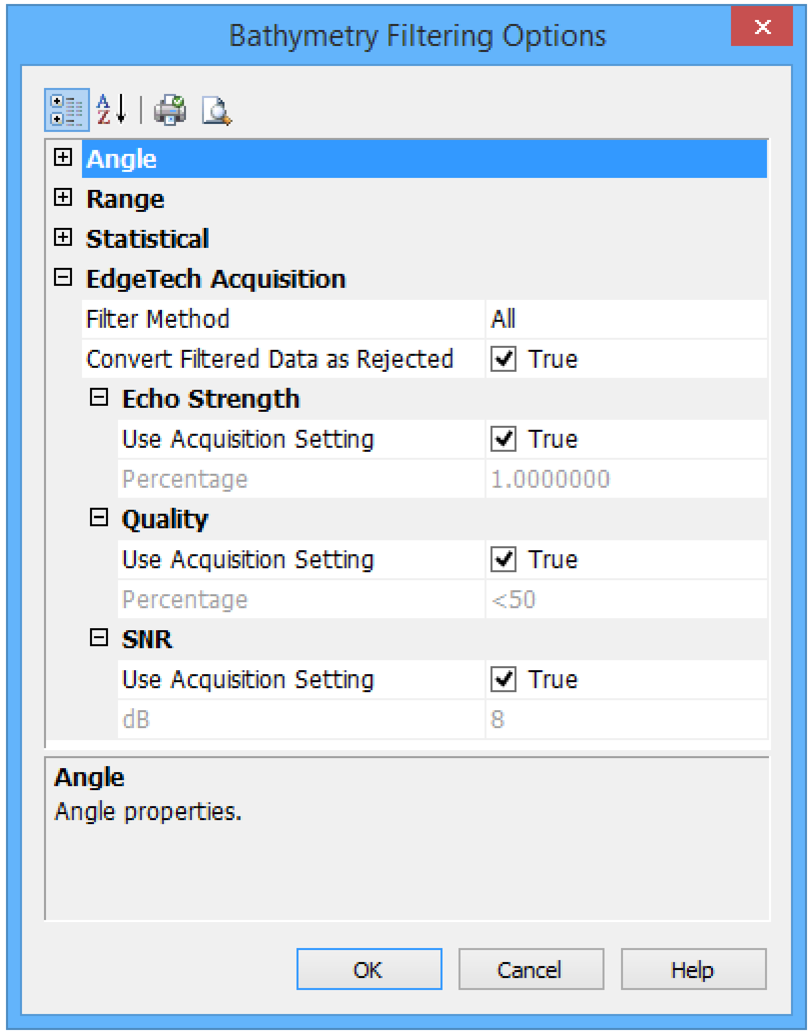
See also

- "Angle filtering" on page 135
- "Amplitude filtering" on page 136
- "Range filtering" on page 137
- "Statistical filtering" on page 139

During acquisition some soundings are flagged as having bad SNR (signal to noise ratio), quality, or echo strength. The default settings in the Bathymetry Filtering Options will filter out these flagged soundings during conversion.

You can ignore these flags, and use a specific criteria instead, by clearing the *Use Acquisition Setting* check box and manually setting filter values.

Also, the volume of data to be converted can be reduced by turning off the *Convert Filtered Data as Rejected* option.



1. Select a filter method from the drop-down list.
2. [Optional] Clear the *Convert Filtered Data as rejected* check box.

If you leave this option set to True, all data will be converted. However, data which does not meet the set filter values will be converted with “Rejected” status.

Echo Strength

Use Acquisition Setting is set to True to filter out soundings rejected for their echo strength values. To filter by a percentage instead:

3. Set *Use Acquisition Setting* to False.
4. Type a percentage value between 0 and 100. The default is set to 1.0.

Quality

Bathymetry data collected with the EdgeTech 4600 system has a quality value associated with every bathymetry sample in each ping. This quality attribute is presented as a percentage value, with a higher value representing a better quality sample.

Use Acquisition Setting is set to True, which will filter using the quality values set during acquisition. Alternatively you can select a percentage value of between <50 to <90 from the drop-down list. The default value is <50.

SNR

Use Acquisition Setting is set to True to filter using the signal-to-noise ratio values set during acquisition. Alternatively, you can set a Decibel value of between 0 and 30. The default value is 8dB.

To use a single Decibel value to filter data during conversion:

5. Set *Use Acquisition Setting* to False.
6. Type a value between 0 and 30.

When all bathymetry filtering options are set:

7. Click **OK** to return to Step 6.
8. Click **Next** to continue the EdgeTech conversion process.

Return to "Bathymetry Filtering Options" on page 134

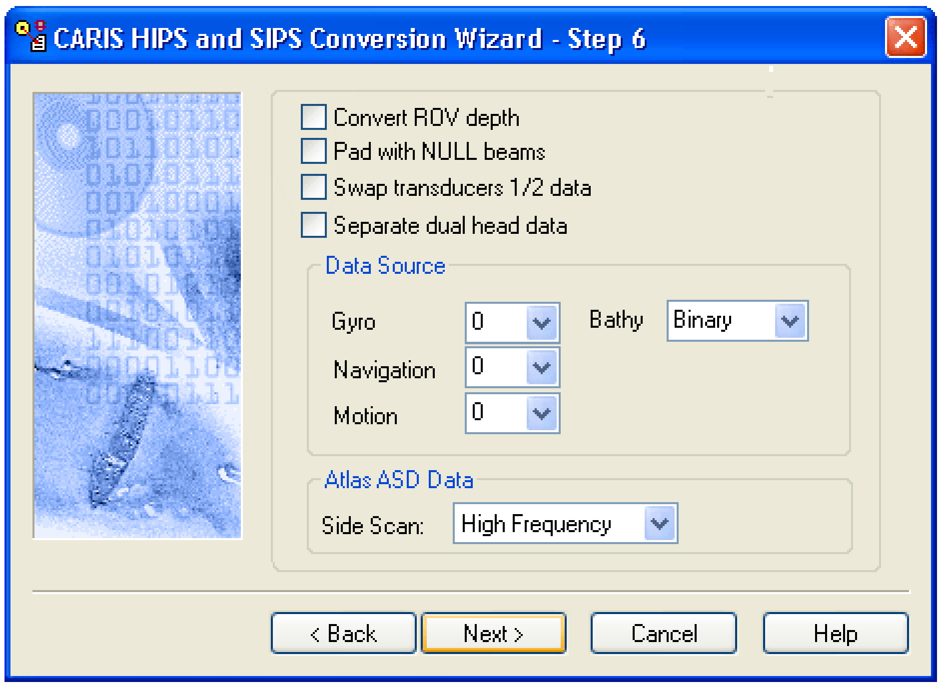
EIVA

.

At Step 5, you have the option of using the generic multibeam filtering functions built into the HIPS Conversion Wizard.

See "Advanced Filtering" on page 92.

Step 6 offers further conversion options specific to EIVA data.



1. Select the *Convert ROV depth* option to store the EIVA sub-packets as HIPS delta draft values.
2. Select the *Pad with NULL beams* option, to replace missing data with NULL (rejected) beams in the case of dual head data where the data from head #1 is missing. This results in the beam numbers for transducer 2 data remaining consistent with fully populated profiles.
3. Select *Swap transducers 1/2 data* to exchange data between heads (in dual head set-up).
4. Select the *Separate dual head data* option to specify that soundings from the dual head transducer data are not combined into a single HIPS profile. (Soundings from transducer head #2 will be identified via status bits encoding so no changes to your vessel files are needed.)

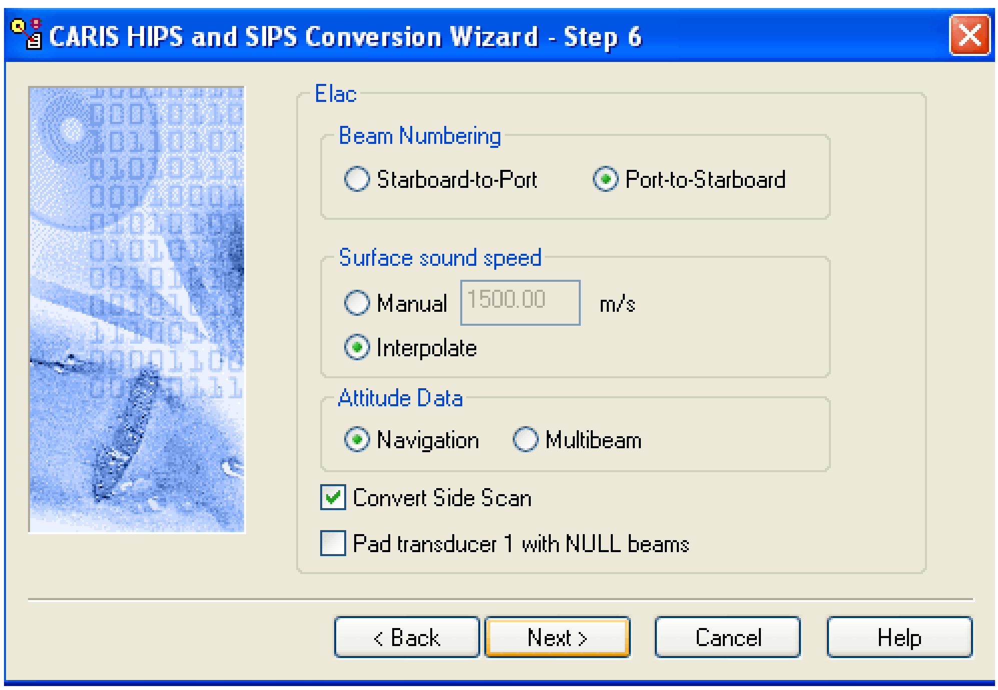
Since EIVA allows data logging from multiple devices, the *Data Source* option enables you to select the source for heading, attitude, and navigation data. The data for each device is tagged with a number (from 0 to 4), that represents one of up to five devices collecting each type of data. (If you select a number for which no device exists, no data is converted.)

5. Select a device number from the list for Gyro, Navigation and/or Motion data to convert the data from that device.

You can convert the XML portion of your Atlas ASD data, instead of the Binary (the traditional source of bathymetry data in the EIVA converter). The XML portion, if it exists, is contained within the EIVA side scan datagrams.

6. Select the source for bathymetry data from the Bathy drop-down list. The default selection is Binary.
7. Select High Frequency or Low Frequency to convert Atlas ASD data.
8. Click **Next** to continue the conversion process.

Elac



Beam numbering

During conversion, XSE-format beams are numbered according to the sounding's position along the swath (in relation to the vessel).

1. To number the converted beams along the swath from starboard to port, select the *Starboard-to-Port* check box.
2. To number the converted beams along the swath from port to starboard, select the *Port-to-Starboard* check box.

Surface sound speed

If a single Sound Velocity value was used during the survey:

3. Select the *Manual* option and type the sound velocity value.

If a Sound Velocity Profile was recorded and used by the logging system, then the *Interpolate* option must be selected.

4. Select *Interpolate* to read surface sound velocity values from a datagram.

Attitude data

You can select the source for attitude data. The Navigation option will convert heave, pitch and roll data from the navigation records in the file and multibeam will convert it from multibeam records.

5. Select either Navigation or Multibeam as the source of *Attitude Data*.

Convert Side Scan

6. To convert side scan data, select the *Convert Side Scan* check box.

Survey mode

ELAC XSE data can be recorded as 108 beams or 126 beams (depending on the survey mode). The *Pad Transducer 1 with NULL beams* writes 63 beams to Transducer 1 so that the same HIPS Vessel File (HVF) can be used for both survey modes.

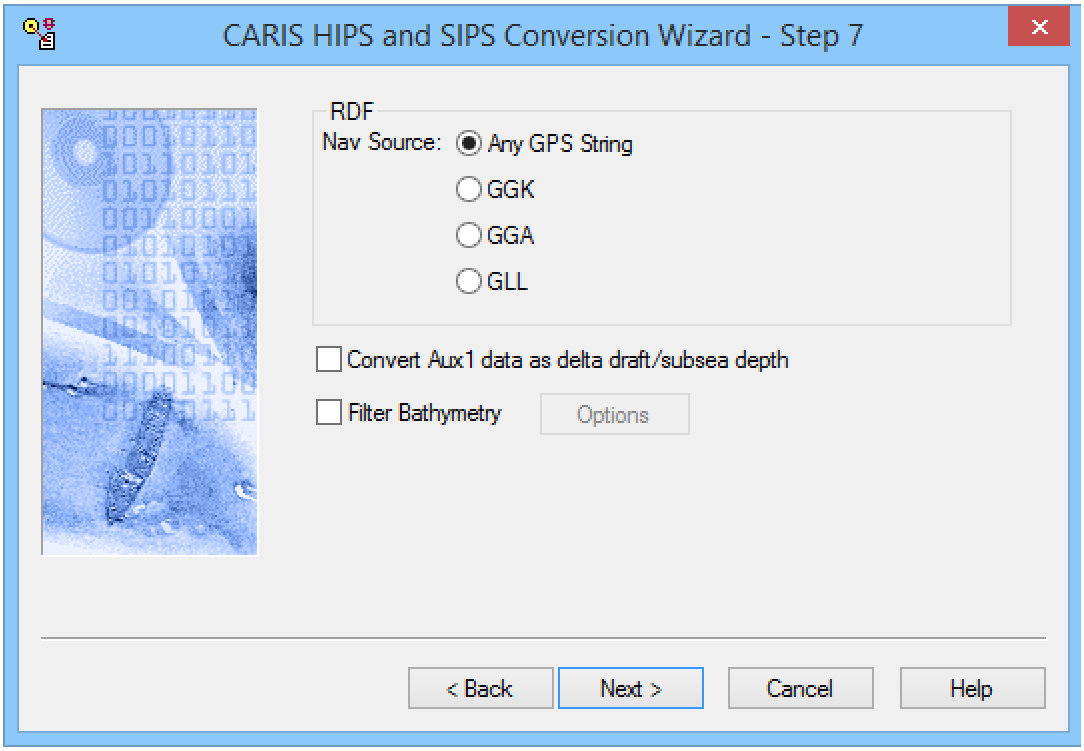
7. Click the *Pad Transducer 1 with NULL beams* check box so that the soundings associated with transducer 1 are padded to have 63 beams.
8. Click **Next** to continue the conversion process.

Furuno

1. Make sure the *Convert Side Scan* option is checked to include side scan data with the bathymetric data.

GeoAcoustics

Options for converting GeoAcoustics data at Step 7 of the conversion wizard:



1. In the *Nav Source* field, select *Any GPS String*, or *GGK*, *GGA*, or *GGL*.

If you want to convert data from Aux1 as delta draft:

2. select the *Convert Aux1 data as delta draft/subsea depth* option.

To activate options for filtering bathymetry:

1. Enable the *Filter Bathymetry* check box.
2. Click **Options**.

The Bathymetry Filtering Options dialog box is displayed.

See "Bathymetry Filtering Options" on page 134 for settings.

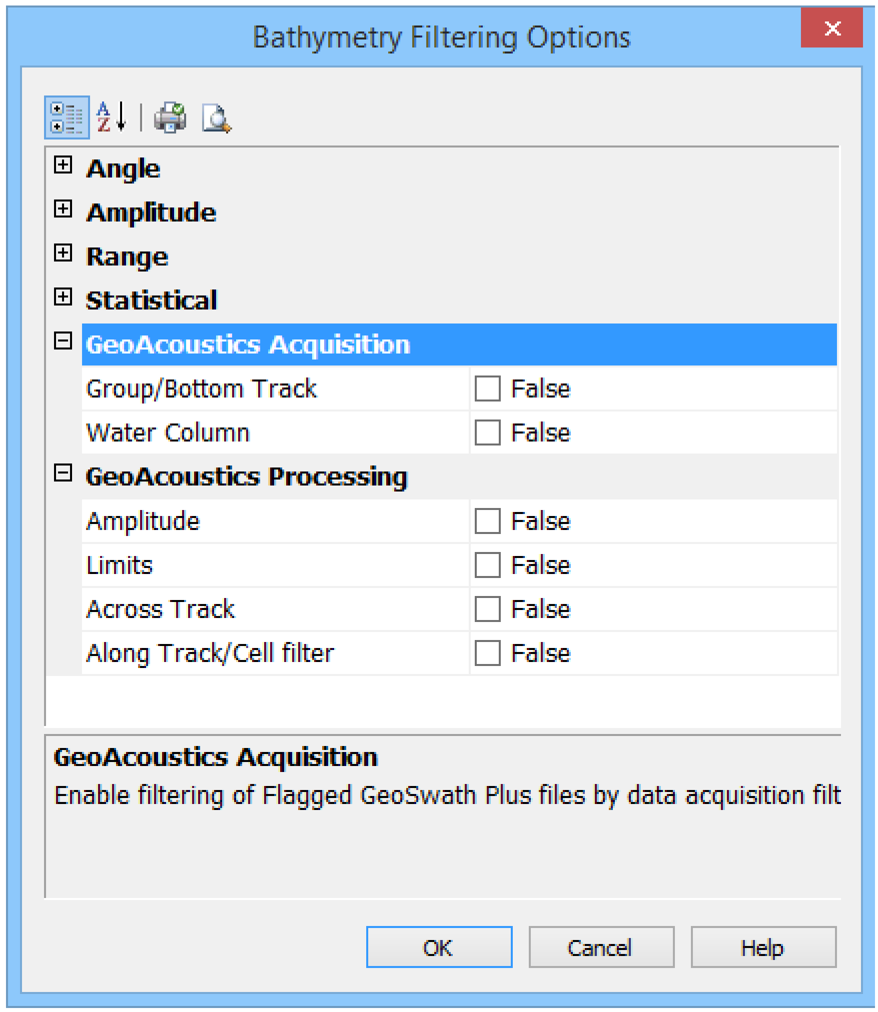
- "Angle filtering" on page 135
- "Amplitude filtering" on page 136
- "Range filtering" on page 137
- "Statistical filtering" on page 139
- "GeoAcoustics Filtering Options" on page 108

When all bathymetry filtering options are set:

3. Click **OK** to continue the GeoAcoustics conversion process.

GeoAcoustics Filtering Options

See also “BATHYMETRY FILTERING OPTIONS” ON PAGE 134 for settings for Angle, Amplitude, Range and Statistical filtering.



GeoAcoustics Acquisition

If options are set to True, data flagged during acquisition will be converted, and flagged as rejected in HIPS. The default setting (False) will convert the data and include any data flagged during acquisition.

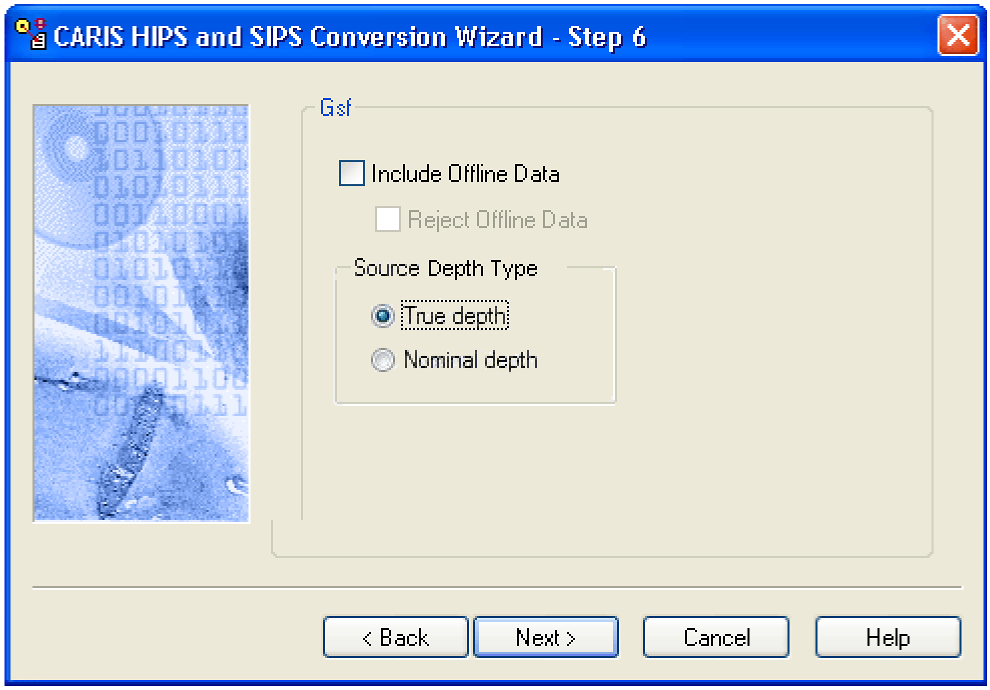
1. Set *Group/ Bottom Track* option to True to filter out flagged data in GeoSwath Plus GS4 RFF files.
2. Set *Water column* option to True to filter out flagged data in Geoswath Plus RDF and RFF files.

GeoAcoustics Processing

If options are set to True, data flagged by GS+ and GS4 filters during post-processing will be converted, and flagged as rejected in HIPS. The default setting (False) will convert the data and include any data flagged during acquisition. This applies to (RDF and RFF files).

1. Set options to True to filter out data flagged by any or all of the following:
   - Amplitude
   - Limits
   - Across-track
   - Along-track/Cell filter fields.

GSF



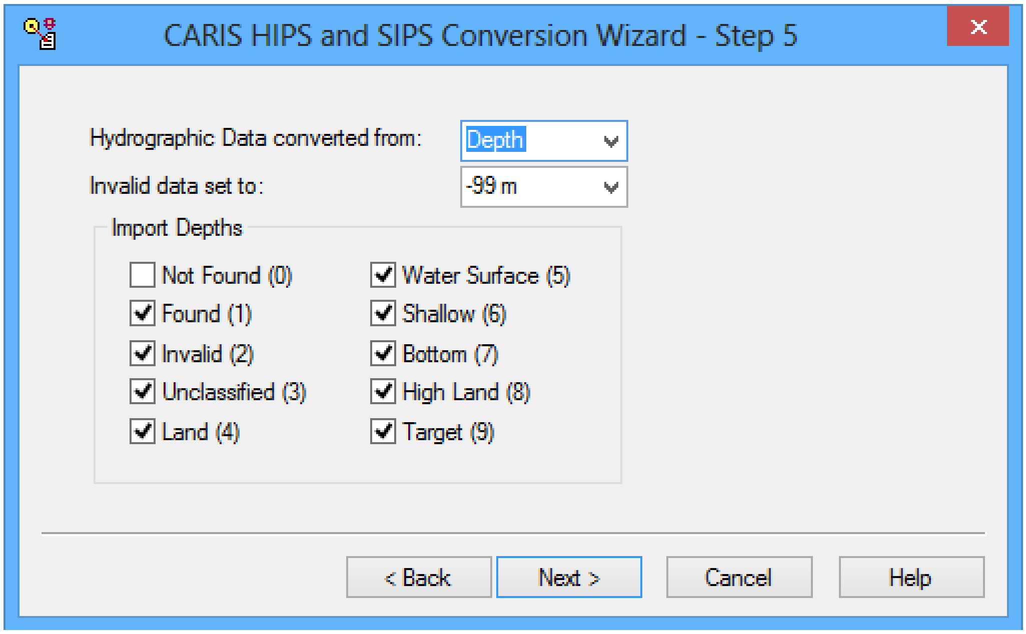
Data recorded during line turns is marked as Offline data in GSF. This data can be included or excluded during conversion. When included it can also be automatically rejected. One purpose for including the Offline data would be to maintain the same swath profile numbers in HIPS format.

1. Select the *Include Offline Data* box to convert data recorded during turns between track lines.
2. Select the *Reject Offline Data* check box to automatically reject the Offline data.
3. Select a source for depth data, either *True depth* or *Nominal depth*.

Horizontal and depth uncertainty values in the GSF file are also included during conversion.

4. Click **Next** to continue the conversion process.

Hawkeye



The Hawkeye data structure as exported from AHAB CSS Software needs to be maintained in order for processing to be successful in HIPS and SIPS.

There are two kinds of Hawkeye files which have identical data structures, but differ in contents:

- The HD files store primarily hydrographic data but can contain topographic data as well.
- The TD files store only topographic data.

Each data point in the files contains two attributes: Depth and PointAltitude.

- The Depth attribute is water depth relative to chart datum.
- The PointAltitude attribute is height relative to an ellipsoid.

Data in TD files is always be converted using the PointAltitude attribute, but where HD files are concerned, you can choose between the Depth and PointAltitude attributes.

1. Select the appropriate source for the hydrographic data from the drop-down list, either Depth or PointAltitude.

The *Invalid data set to* option will apply a substitute depth of either +99m (below chart datum) or -99m (above chart datum) for data rejected by disabled beam.

2. Select -99m or +99 m as substitute depth.

This data will be marked as “Rejected”.

3. Select the depth classes to import to HIPS. (By default all classes are set to be to be imported.)

4. Click **Next** to continue the conversion process.

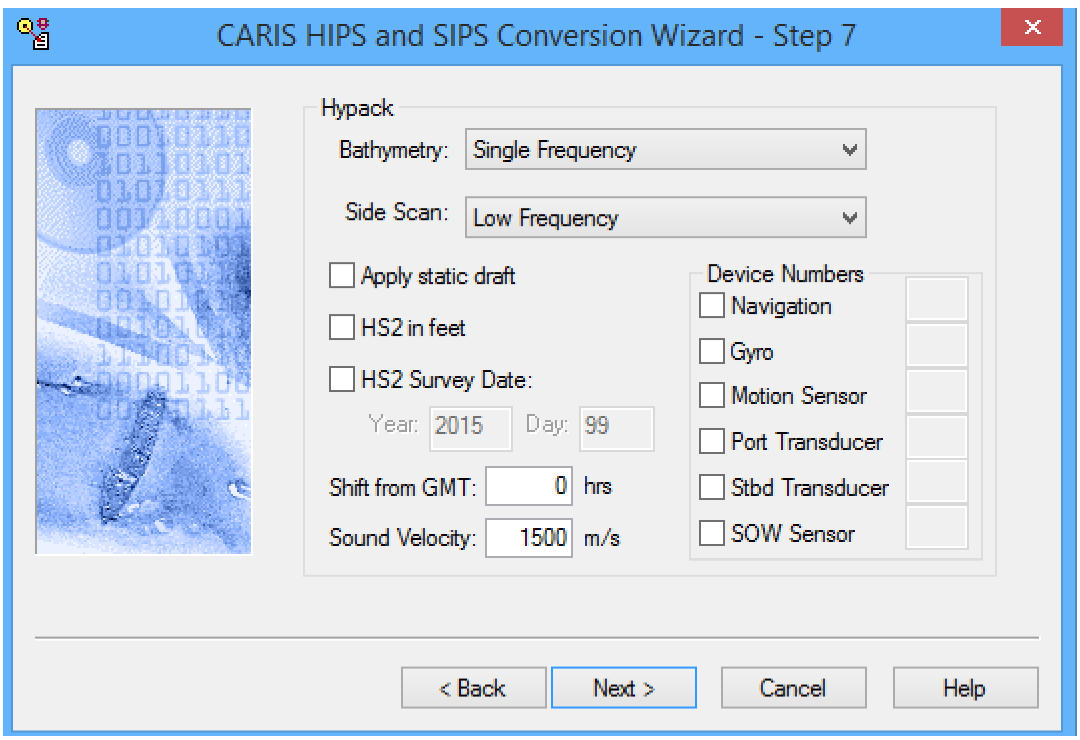
Hypack

The Hypack format records bathymetry from single beam, multibeam, and multiple transducer (sweep) systems.

It also supports recording side scan data from Hysweep format (HSX).

For dual-frequency single beam data, you must indicate if the data records in Hypack are recorded as primary/secondary or secondary/primary. The primary frequency, by default, becomes the selected data for use.

When converting Side Scan data, if the conversion wizard detects records containing 12-bit data, they will be converted to 16-bit format. Otherwise they are converted to 8-bit data.



Use the *Bathymetry* and/or *Side Scan* drop-down lists to select the type of data to be converted. (You must make a selection or the conversion process will not go forward.)

Select a *Bathymetry* setting to convert data in Hypack raw format (RAW and HSX) and processed format (HS2) from single beam, multibeam and multiple transducers systems.

If you are converting *Bathymetry*, ensure that the vessel file for the data contains a Depths section, or conversion will fail.

1. Select a *Bathymetry* sounding data type from the drop-down list:

- Single Frequency
- Dual Frequency (Primary, Secondary)
- Dual Frequency (Secondary, Primary)
- Multibeam
- Multiple transducers (for sweep data)

Hysweep format (HSX) does not store Geographic Coordinates and GPS Height information. If recorded, this data is stored in RAW files. When both the Geographic option and Multibeam options are set during conversion, the converter will search for *.raw files in the same directory with the same line name to get this information.

If you are converting multibeam data, and you set “Geographic” at Step 5 as the Navigation Coordinate Type, the converter will search the directory where the HSX files are located for *.raw files with the same name as the *.hsx files. The RAW files contain the RAW record which contains the Geographic coordinates and GPS height information.

2. Select *Side Scan* data type from the drop-down list:
   - Low Frequency, High Frequency or None.
   - If the data file contains only single frequency data then this setting is ignored, and all data is converted.
3. Select *Apply static draft* to apply the static draft during conversion.

To ensure data collected in feet is converted correctly, convert data in feet rather than metres:

4. Select *HS2 in feet* check box.

The HS2 format does not store day and year values with the time stamp (it does store a time value). When the data is converted to HIPS, by default the Day folder is used for reading the date.

If you want to include another date value:

5. Select the *Use HS2 Survey Date* check box to enable the *Day* and *Year* fields.
6. Type the year and day in the fields.

Shift from GMT

Some HSX data may contain local time. To shift the time stamps of this data to GMT use the *Shift from GMT* option. This option may be necessary to synchronize Hypack data with externally logged files, such as backscatter in 7K or Water Column imagery in R2S format.

7. Type the number of hours by which to shift to time stamps so that they show as GMT.

The value in the *Sound Velocity* field is used to compute the original travel time data from the recorded depths. Since this is used in performing sound velocity correction, it is important that the *Sound Velocity* value is correctly specified.

8. Enter the *Sound Velocity* value for:
   - single beam data (Single or Dual Frequency
   - multiple-transducer (sweep) data.

Device Numbers

The *Device Numbers* fields are used if sensor data was recorded from more than one device. To import data from a specific sensor, type the number for that device. If no number is entered then *all* sensor data is imported. In the case of the Port and Starboard Transducers, the first device found will be used.

The following sensors are affected by the device number setting:

- *Navigation* for positional records.
- *Gyro* for heading records.
- *Motion Sensor* for roll, pitch and/or heave settings.
- *Port transducer* for port multibeam records.
- *Stbd transducer* for starboard multibeam records.
- *SOW Sensor* for speed over water records.

9. If data is recorded from more than one sensor device, click the sensor check box and enter a device number.
10. Click **Next** to continue.

Imagenex

HIPS can convert dual head data stored in the Imagenex *.83m format. However, in order to properly process the data, the HIPS Vessel file must be set up as a dual head vessel.

As well, if the number of beams normally expected for head #1 is more than 200, set the number larger, for example, to 1000.

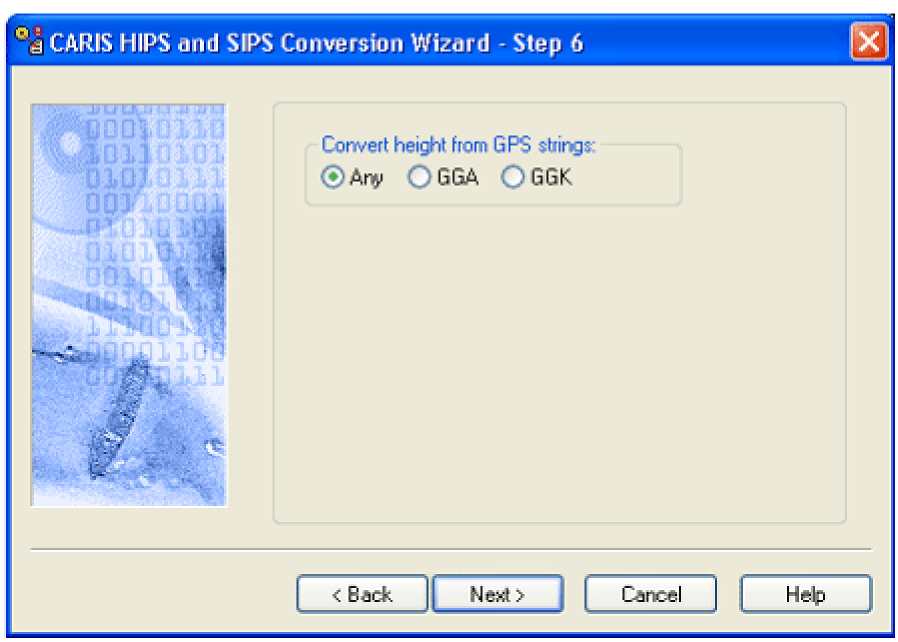
If you expect head #1 to generate no more than 200 beams, set the beam number in the HVF to 200.

Navigation data can be imported from an external *.NAV file with the same name as the Imaginex *.83p file. If the HIPS converter detects that a *.NAV file exists, it will import navigation and GPS height data from the GPS strings in the *.NAV file. In this case the stored navigation data in the *.83p file is not converted.

You can set the converter to read either the GGK or the GGA string, or have either read automatically by selecting the *Any* option. (These controls are only applied if there is an *.NAV file with the same name as the *.83p file.)

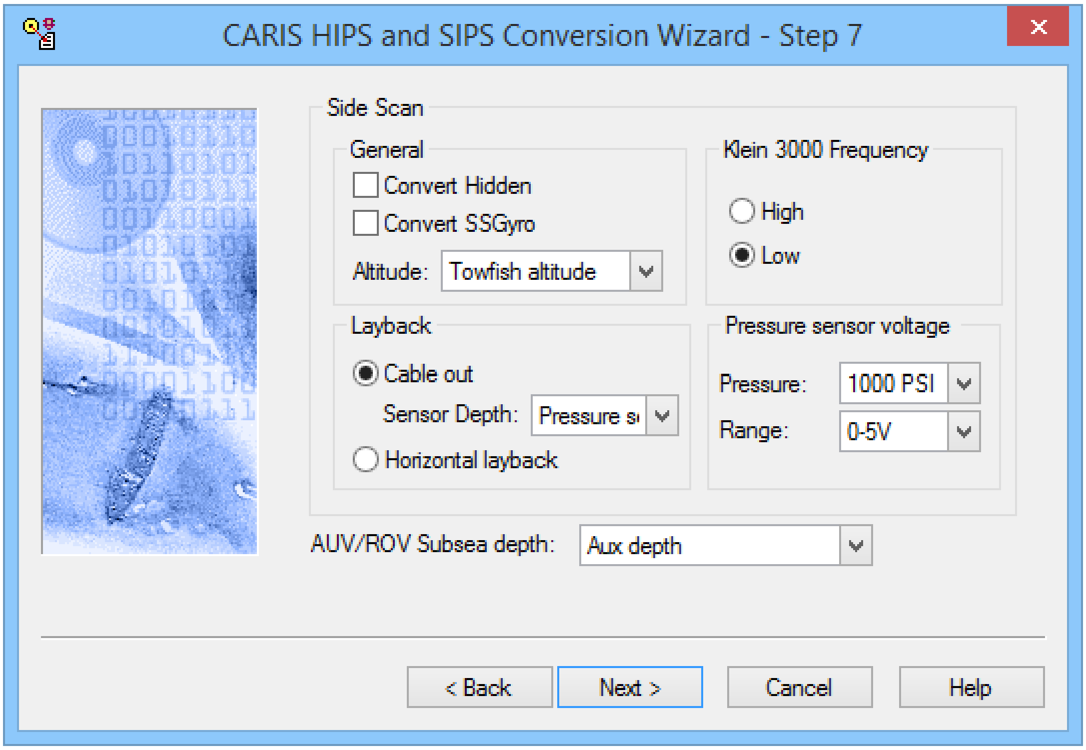
To read GPS height data from Imagenex files:

1. Select the height string to be read: *GGA, GGK* or *Any*.
2. Click **Next** to continue the conversion process.



Klein (SDF)

All sonar data is stored using 16 bits.



1. Set Side Scan options.

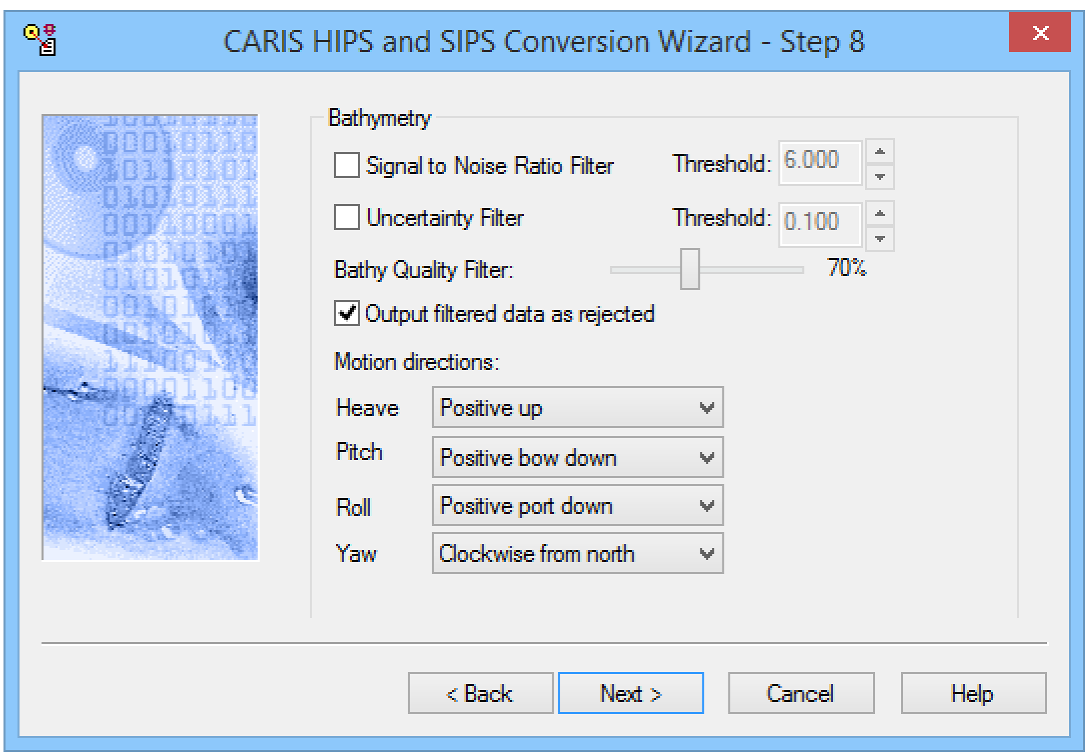
Option	Description
General	
Convert Hidden	Select the *Convert Hidden* check box to convert redundant data collected during acquisition.
Convert SSGyro	Select the *Convert SSGyro* check box to include side scan sonar gyro data.
Altitude	Select Towfish altitude from the list Select Aux altitude to convert sonar altitude from the auxiliary altitude field in the sonar record.
Klein 3000 Frequency	Set to High or Low.
Layback	Select either Cable Out or Horizontal layback. Default setting is Cable out.
Cable out	Set to apply Cable out.
Sensor Depth	If *Cable out* is active, select Aux depth or Pressure sensor voltage from the list.
Horizontal layback	If horizontal layback is selected, sensor depth is set to 0.0 metres.

Option	Description
Pressure sensor voltage	If used by the Sensor depth or AUV/ROV Subsea depth option, the settings used during acquisition will need to be set to properly convert the voltage to a depth value.
Pressure	Select a *Pressure Sensor* reading from the drop-down list to set the depth of the towfish.
Range	Select the *Range* of voltage for the pressure sensor, either 0-5V or 1-5V.
AUV/ROV Subsea depth	Default setting is Aux depth to use the data in the auxiliary depth field. If for any reason, the Aux depth setting is invalid, select the Pressure sensor voltage option (and set correct Pressure and Range values) to override the Aux depth setting.

2. Click **Next** to set Bathymetry options for SDF conversion.

Klein Bathymetry Filters

1. Set Filter options.



Option	Description
Signal to Noise Ratio Filter	Enable check-box to filter according to the signal to noise ratio (SNR) value stored during data collection.

Option	Description
Threshold	Select the value above or below the stored SNR value to filter by. • If the data has a smaller SNR than specified in the Threshold field, it is then converted and flagged as Rejected. • Minimum/maximum threshold settings are -50.0/ 50.0, respectively.
Uncertainty Filter	Enable the check-box to filter according to the Uncertainty value during data collection.
Threshold	Select the value above or below the stored uncertainty value to filter by. • If the data has a smaller uncertainty than specified in the Threshold field, it is then converted and flagged as Rejected. • Minimum/maximum threshold settings are 0.0/1.0 respectively.
Bathy Quality Filter	Use the slider to set a value between 50 and 100%. • If the stored quality value is below the set value it will be rejected, when the *Output filtered data as rejected* option is set. Otherwise it is not converted. The default value is 70%.
Output filtered data as rejected	Enable the check-box to include the data filtered by the Bathy Quality Filter in the converted dataset. The data will be marked as Rejected.
Motion directions	Select the directions that Heave, Pitch, Roll and Yaw data stored in the SDF file when created by the Klein SonarPro®software.

2. Click **Next** to continue the conversion process

Kraken

There are no specific options for the Kraken format.

LADS

LADS data is contained in two files that are copied into the HIPS survey line folder: An ASCII CAF file that contains the bathymetry data, and a binary CBF file that contains the waveform data. The CAF file can contain data from more than one survey line (or “run”). When this is the case, the converter splits the file into separate line folders for each run. The *.caf and *.cbf extensions are retained in the HIPS Line folder.

There are four sounding classifications in the LADS format:

- **S (Secondary sounding)**: the best soundings
- **P (Primary sounding)**: an unreliable result
- **N (No bottom found at depth)**: potentially a useful depth result
- **X (No bottom detected)**: no result

When the data is converted, each sounding is given a status flag according to its classification. The following table lists the HIPS status flags applied to the LADS classifications.

LADS Classification	HIPS Sounding Flag	Definition
S	Accepted	Good soundings until they are flagged as rejected during cleaning in HIPS.
P	Rejected by disabled beam	A reject flag in HIPS that can be changed to Accepted, if necessary.
N	Examined	A special flag for an Accepted sounding. The sounding can be filtered out when creating a surface or exporting. It can also be flagged as Rejected during cleaning in HIPS
X	Rejected by Depth Gate	A type of Reject flag that can be changed to Accepted, if desired.

There are no dialog box options for the LADS format.

LAS

Invert Elevation

LAS files can contain either elevation or hydrographic data. Conversion step 6 for LAS data offers the option to invert the “Z values” from positive to negative, and vice versa, so that the Z-axis is changed to match the HIPS positive down depth convention. By default this option is turned off.

GPS time reference week

LAS data that stores time as GPS seconds of the week stores the survey date in the file header. However, if the survey date was recorded incorrectly, it can be overridden by selecting the correct date using the *GPS time reference week* option.

Reference weeks are time-stamped in GPS seconds, starting at 0 at midnight on Sunday. The default setting is to take the date from the file header.

To set another reference week:

1. Click the button beside the date field, and click **Select Day**.
2. Select the reference date from the calendar picker.
3. Click Next to continue the conversion.

This option is not used during conversion if the LAS file stores timestamps using the “standard” format, which is the actual number of seconds from 6-Jan-1980.

NOTE: The LAS data format can store coordinates using the ground or geographic system. If geographic coordinates are stored, the data file should contain a “Coordinate System” record, stating this fact. If no Coordinate System record is found, the converter will assume ground coordinates have been used.

If coordinates have been stored as geographic, but no Coordinate System record is present in the file, the data will be interpreted as with ground coordinates.

To resolve this, select “Geographic (Lat/Lon)” from the Projection list in the Navigation Coordinate Type dialog box (Step 4), then select an appropriate zone, such as “World Geodetic System 1984” (WG84).

Reference week

By default, reference weeks are time-stamped in GPS seconds, starting at 0 at midnight on the start day. By default the start day is Sunday, but this can be changed. For example, if the survey week runs from Wednesday to Tuesday, Wednesday can be set as the start day for the survey week. Sunday is “rolled over” and data for the full survey week is applied.

The LAS format stores GPS time as Absolute GPS time, the seconds elapsed since January 1st 1980.

MarineSonics

There are no specific options for the MarineSonics format.

Navitronics

The 64-bit version of the libraries for this format remain unavailable at this time. Therefore, conversion of Navitronics data is available only in the 32-bit version of HIPS and SIPS.

1. Select one of the following data types to be converted:
   - *Multibeam*
   - *Singlebeam*
   - *Multiple Transducer*
2. Select the *Convert Side Scan* check box to convert side scan data.

Navitronics side scan data can be imported in 16-bit format or converted to an 8-bit format.

3. Select either the *Preserve 16-bit* option or *Convert to 8-bit* option.

If the *Convert to 8-bit option* is selected, the *Scale* and *Shift* options are enabled. The *Scale* option averages the data and the *Shift* option selects a initial bit value (between 0 and 8) and includes the next seven bit values for export. For example, if you typed 8 as your initial value, then bits 8 to 15 are exported.

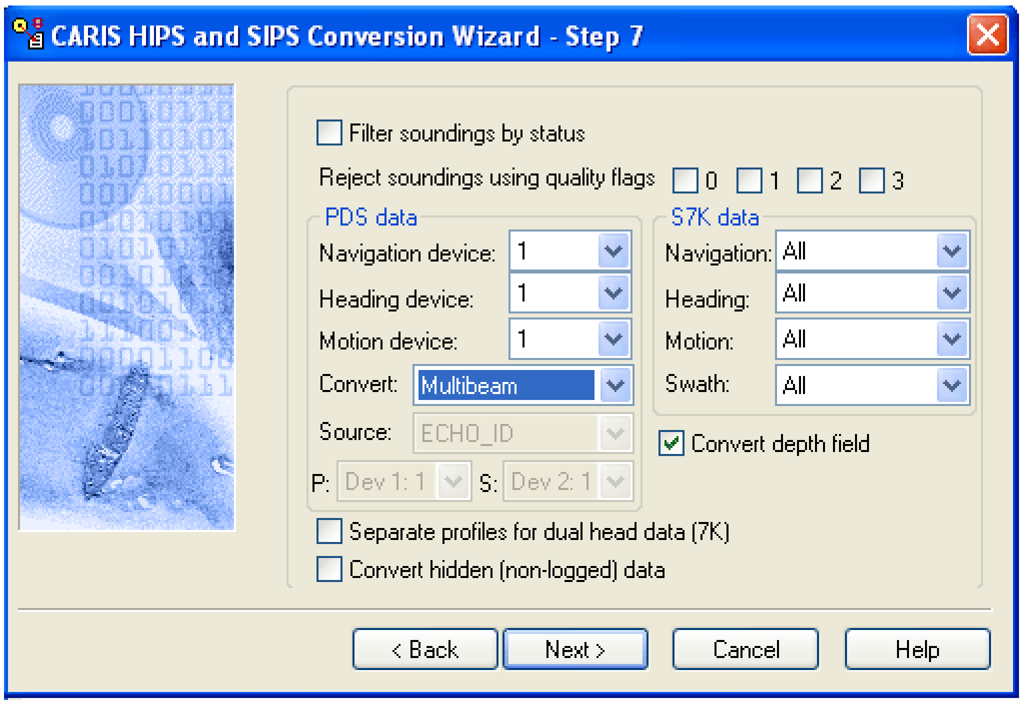
4. Select the *Scale* option if you want to average the data into 8-bit format, or select *Shift* and type the value for the first bit.
5. Click **Next** to continue the conversion.

Qmips

1. Choose a pair of sonar channels by clicking one of two options (1,2 or 3,4).

Teledyne Reson PDS

This step of the conversion process for Teledyne Reson data provides options for filtering soundings, and for identifying the devices containing the data you want to convert.



1. Select the *Filter soundings by status* option to filter using the status bits in the data file.
2. To *Reject soundings using quality flags*, select the quality bits for the soundings you want to reject during conversion. (If you select all four check boxes, all soundings will be rejected.)
3. Click **Next** to continue the conversion process.

PDS data

By default, it is multibeam data which is read and converted. You can also convert single beam or dual frequency data, by identifying the devices containing the data.

1. In the *PDS data* drop-down lists, select the number for your *Navigation*, *Heading* and/or *Motion* device, so that data will be converted from that specific device. (Devices are numbered 1 to 5.)
2. Select the type of data being converted (Multibeam, Singlebeam, Dual Frequency) from the *Convert* drop-down list.

If converting single beam or dual frequency data you will need to specify which echo sounding device was used. If you care converting single beam data, you can choose from the ECHO_ID (raw data) or CF_DEPTH (processed data) attributes from the Source drop-down list. If converting dual frequency data, the only source available is ECHO_ID.

3. From the *Convert* drop-down, select the kind of data to convert: Multibeam, Singlebeam or Dual Frequency.

Singlebeam

If you select the Singlebeam option, you can convert records from either a ECHO_ID (raw data source) or CF_DEPTH (processed source) device.

If you select ECHO_ID as the source, you can select 1 of 2 devices, with one of 4 depth attributes for either device.

If you select CF_DEPTH, you can select either device 1 or device 2.

4. Select either ECHO_ID or CF_DEPTH from the *Source* drop-down list.
5. From the Primary (P:) drop-down list, select the device and channel where the data is stored.

> If you select a device that is not present, or in which the selected depth channel is not available, no depth data is converted.

Dual Frequency

If you select Dual Frequency, you can convert data from both a primary (P:) and a secondary (S:) device, with one of four channels for each device.

6. From the Primary (P:) drop-down list, select the device and channel where the data is stored.

If you elect to convert primary data from a device that is not present, or in which the selected depth channel is not available, depths of 0.0 metres will be converted, but all data will be rejected.

If you convert secondary data from a device that is not present, or in which the selected depth channel is not available, no secondary depth data will be converted.

S7K data

It is possible to have 7004/7006 and 7027 records stored in the same file. The choices for the Swath option are Auto (default option), Bathymetry (7004/7006) and Raw Detection (7027).

If you do not apply the Auto option:

1. Specify either Bathymetry (7004/7006) or Raw Detection (7027) as the source of bathymetry in the S7K data.

The default setting Auto ensures that if Raw Detection Info (type 7027) records are present in the file, they will be converted over the Bathymetry (7004/7006) datagrams.

In the absence of any Raw Detection Info datagrams, Bathymetry (type 7004/7006) datagrams are converted.

If data is dual head and you need the data for each head stored in individual profiles:

2. Select the *Separate profiles for dual head data* option.

By default, hidden or non-logged data exported from the PDS2000 is not converted.

3. To include this data when converting Teledyne Reson PDS, select the *Convert hidden (non-logged) data* check box.

By default the depth field is not converted. To convert the depth field values:

4. Select the Convert depth field option

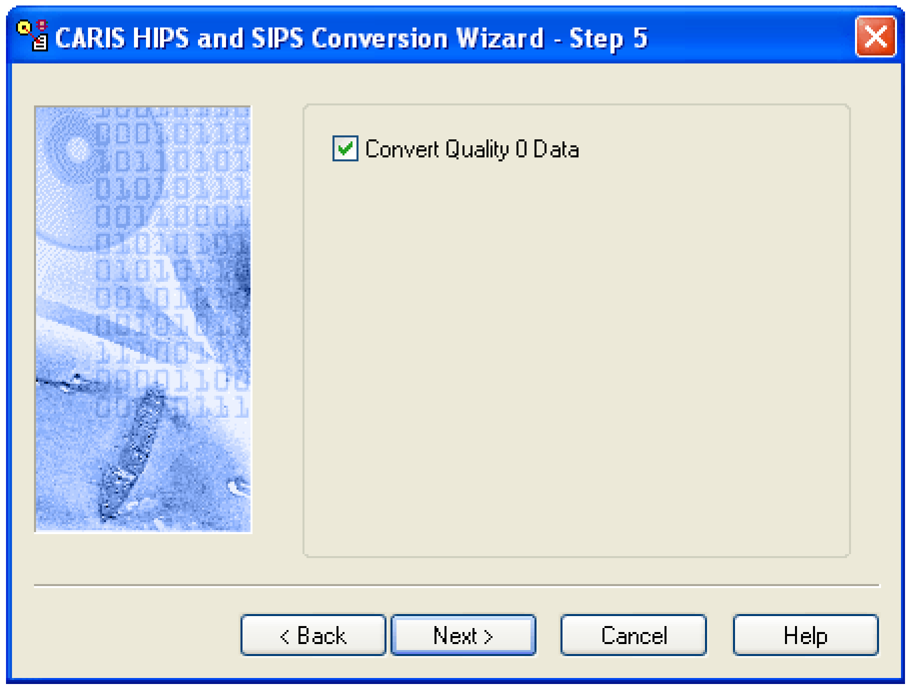
This will populate Delta Draft file in the line directory.

5. For information applying Delta Draft, see "Load Delta Draft" on page 200.
6. Click **Next**.

ProSAS

The HIPS Converter for ProSAS (Synthetic Aperture Sonar) data filters out Quality 0 data, considered “bad” data, however, it can be included in the conversion.

1. Select the check box to convert Quality 0 data.



2. Click **Next** to continue the conversion process.

Scripps

There are no options specifically for the Scripps format.

Seabeam

1. Make sure the *Convert Side Scan* option is checked to include side scan data with bathymetric data.
2. Select the *Shorten Line Names* check box to modify the file name to a 12-character CARIS name (YYDDD_HHMMDD).
3. Select the *Convert delta draft/subsea depth* option to convert this data to HIPS format.
4. Click **Next** to continue the conversion process.

Seafalcon

Seafalcon allows the option of importing a full-sounding dataset or a thinned dataset. The thinning options are based on reducing the number of beams per profile that are imported. All profiles are imported.

1. Select the thinning option, if needed.

The following two options are enabled if the thinning option is selected.

2. Select the amount of data to be processed.
   - half the number of beams are imported
   - one fourth the number of beams from each profile are imported
3. Select the sounding type to be imported into HIPS format.
   - shoal
   - average
   - deep

You can select the swaths per ping that you want to import. In deep water, the Seafalcon sonar produces five swaths with each firing of the transducer. These five swaths are in an along-track position to the ship, with swath 1 forward of the sonar and swath 5 aft of the vessel. In shallow water only one swath per ping is generated.

By default all five swaths are selected for import, but you can decide which swaths to import.

4. Select the swath number(s) for import.
5. Click **Next** to continue the conversion process.

SEGY

There are no options for the SEGY format.

SHOALS

All soundings in the SHOALS* format are given a confidence value during acquisition. These confidence values are used to flag soundings during conversion so that soundings with certain confidence values are rejected. The following status flags are assigned to soundings. These can be changed in HIPS.

- **Rejected by disabled beam**: This flag rejects soundings with a negative confidence value.

- **Rejected by depth gate**: This flag is for soundings with confidence values that range from 0 to 70. The 0 to 50 range represents soundings where no bottom was found, and the 51 to 70 represents soundings of questionable confidence.
- **Accepted**: This flag is for soundings with confidence values from 71 to 99.

When converting to HIPS, all soundings with a confidence value less than the *Confidence Cutoff Value* will be automatically rejected. The default cut-off value is 70.

1. Use the slider to set a cut-off value.

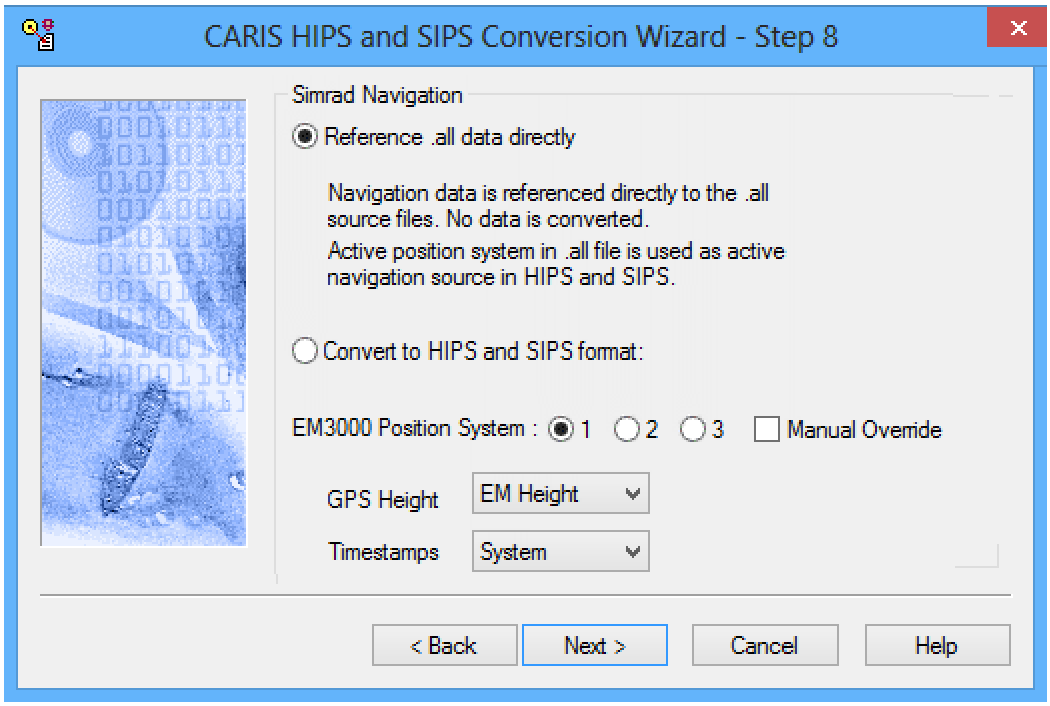
If the dataset contains land data, you can select the topographic format (TOF) options to convert this data too.

2. Select either the *First Pulse* check box to convert the return from the top of the vegetation canopy or the *Second Pulse* check box to convert the return from the ground.
3. Click **Next** to continue the conversion process.

*(Scanning Hydrographic Operational Airborne LIDAR Survey)

Kongsberg

All navigation sources from Kongsberg .all files can be referenced directly instead of converted. The primary navigation source in the .all file will be set active on conversion to HIPS. This means no navigation is converted to HIPS format, and these projects will not be compatible with previous versions of HIPS.



By default, the *Reference .all data directly* option is selected.

To convert navigation:

1. Select the *Convert to HIPS and SIPS format* option.
1. f you are using the EM3000 sonar, select the channel used for the primary position.
2. Select *Manual Override* to read from the navigation system data that is marked inactive by Kongsberg.
3. For *GPS Height*, select a source from either the *EM Height* datagrams or the *GPS String* in the navigation datagrams.

Navigation time stamps can be selected from either the logging system or from the GPS string in the navigation datagram. The default setting is *Auto.*

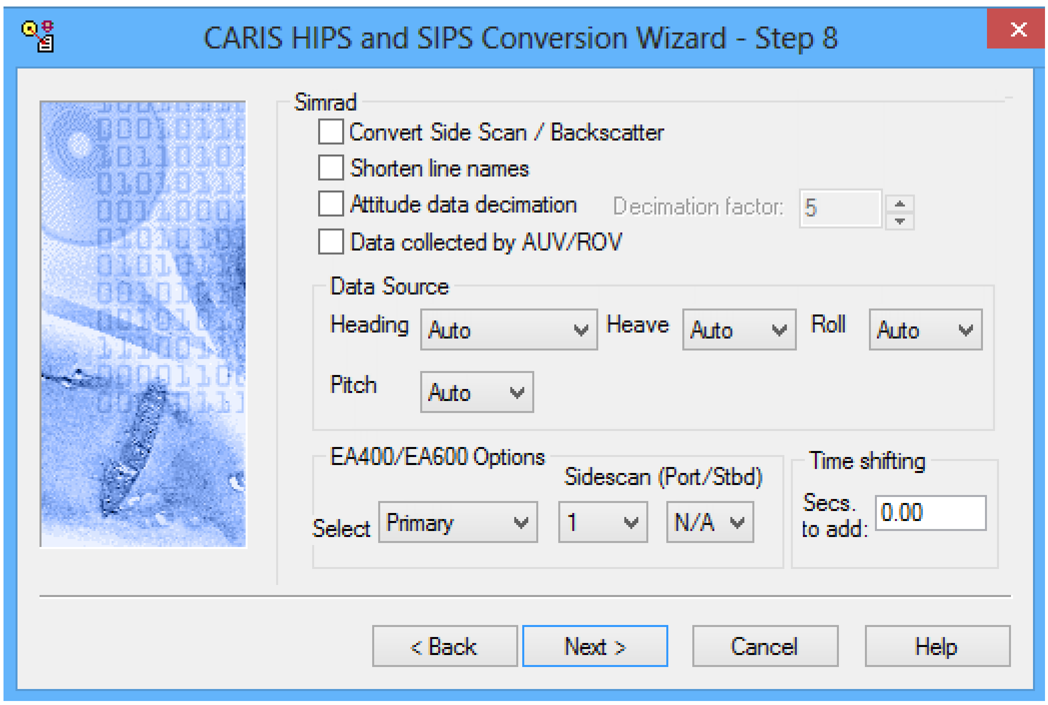
Retaining the default setting will search for GPS Height data, and if not found, will apply timestamps specified by the Installation datagram.

4. Select GPS from the Timestamps list to apply data from the GPS String. If this is not found, the conversion will fail.

5. Select System to apply time stamps from the logging system.
6. Select to select the time stamp specified by the Installation datagram.
7.
8. Click **Next** to set further Kongsberg conversion options.

Kongsberg Conversion Options 2

Another set of options for Kongsberg conversion is displayed in the next dialog box.



1. Select the *Convert Side Scan / Backscatter* check box to convert this data.

You can fit critical line information into the 12 characters provided by the CARIS Source ID attribute for soundings.

2. Select the *Shorten line names* check box to modify the file name to a 12-character CARIS name (YYDDD_HHMMDD).

Large attitude data files can result if there is a straight one-to-one import into HIPS. The *Attitude Data Decimation* option down-samples the data so that a smaller attitude dataset is converted.

The *Decimation Factor* determines the ratio of attitude data. For example, if you select 10 as the *Decimation Factor*, then one out every tenth attitude record is converted.

3. Select the *Attitude Data Decimation* check box to implement this option.
4. Select a *Decimation Factor* to set the ratio for down-sampling attitude data.

5. Select the *Data collected by AUV/ROV* check box to allow for proper processing of sub-sea depth during post-processing functions, such as Sound Velocity Correction and Merge.

Data Source

Motion data (heading, heave, roll, and pitch) is automatically read from the active motion reference unit. However, you can select another motion reference unit to read the data. Gyro data can also be read from the headings datagram.

6. For *Heading*, select the *Auto* option to read gyro data from the active motion reference unit, or select an inactive unit by selecting *MRU1/MRU2* options. Select the *H. Diagrams* option to read gyro from the headings datagram.
7. For *Heave, Roll,* and *Pitch*, select the *Auto* option to read data from the active motion reference unit, or select an inactive unit by selecting *AutoMRU1/MRU2* options.

For *GPS Height*, select a source from either the *EM Height* datagrams or the *GPS String* in the navigation datagrams.

EA400/EA600 Options

An EA400/600 dataset is composed of two parts, and side scan / amplitude data is stored in the *.raw component. However since the file format does not explicitly state which channel contains the side scan data, you need to know beforehand where the port side and starboard side data are stored.

8. If you are converting single beam data from the dual frequency EA400\600 system, select *Primary* to use the primary sounding as the selected sounding in the HIPS file, or select *Secondary* to use the secondary sounding as the selected sounding.
9. Use the *Side scan* drop-down list to select the number of the channel which contains the side scan data.
10. Use the *[Port/Stbd]* drop-by list to select the number of the port and of the starboard channels. If you select N/A for a channel, no side scan data will be converted for that channel.

Time shifting

The Time shifting option lets you add a constant number of seconds to each piece of data retrieved from the `*.all` file. This time shift value can be positive or negative and it's set to zero by default.

11. Enter the value to be added as a constant.
12. Click **Next** to continue the conversion process.

Spawar

There are no options specifically for the Spawar format.

SWATHPlus

Swathplus options:

1. Type a value in m/s in the *Speed of sound for SXR data* field to apply sound velocity to the imported data.
2. Click **Next** to continue the conversion process.

Teledyne

There are no other options specifically for the Teledyne format.

UNB

Reson options:

1. Select one or two transducers.

Swathed options:

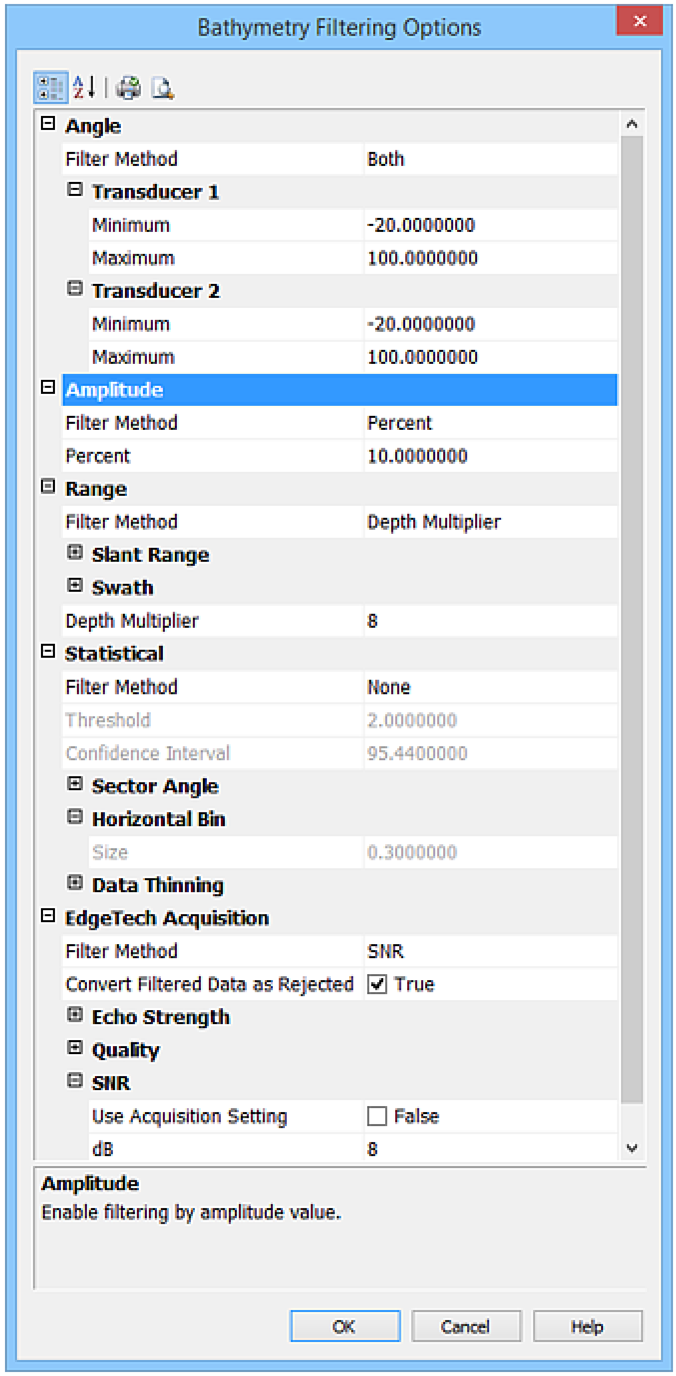
2. Select the *Convert Sonar* option to import side scan data.
3. Click **Next** to continue the conversion process.

Winfrog

Bathymetry:

1. Select *Single* or *Dual Frequency*.
2. Select attitude records to be converted, either 413 or 888 records.
3. Specify a speed of sound, so that slant range data can be generated from the measured sounding depths. Default is 1500 m/s.
4. Choose the navigation records to be converted, either 303 or 351.
5. Click **Next** to continue the conversion process.

Bathymetry Filtering Options



1. Click **OK** to accept the default settings for filtering and continue with conversion.

You can customize these filter settings. When default settings are changed, the new values are retained so they can be used the next time bathymetry filtering is applied during conversion

The types of filters which can be applied are:

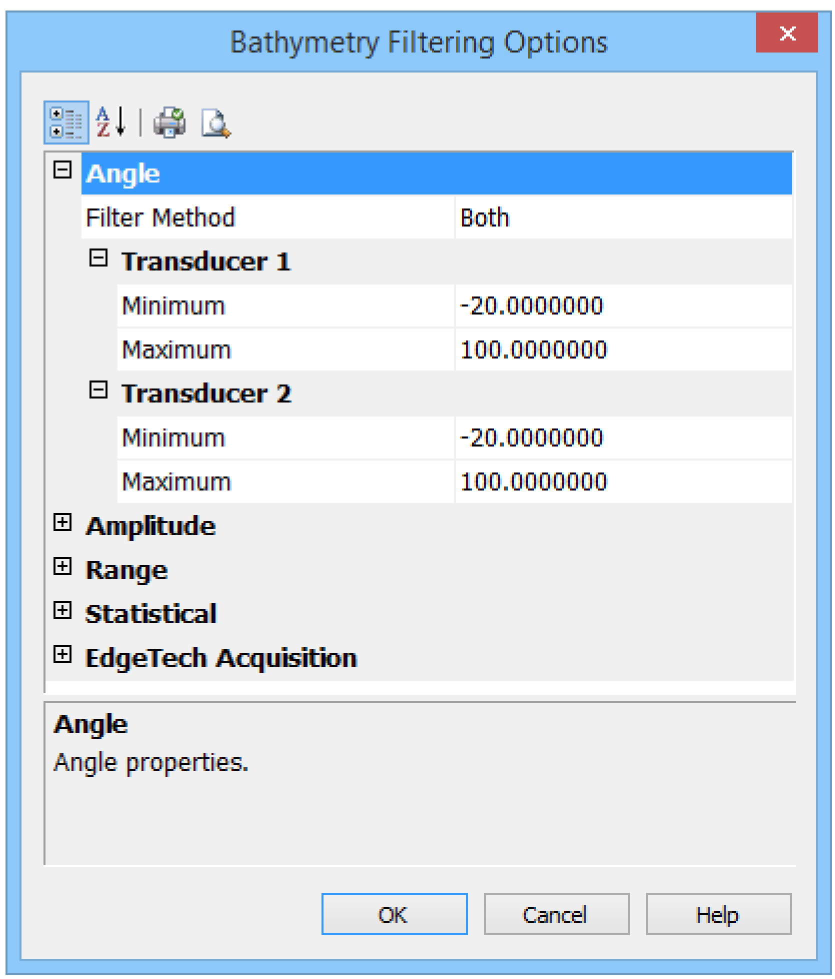
- "Angle filtering" on page 135
- "Amplitude filtering" on page 136
- "Range filtering" on page 137
- "Statistical filtering" on page 139
- "EdgeTech Acquisition Options" on page 101
- "GeoAcoustics Filtering Options" on page 108

Angle filtering

The Angle filtering uses the angles from the raw range angle information to filter out data falling above and below set values.

Filtering can be done using Transducer 1 or Transducer 2, or both. The default setting is to use both.

If you select "None", the fields are greyed out to indicate that the angle filters will not be applied.



1. Select a filtering method from the *Filter Method* drop-down list: Transducer 1 or Transducer 2 or both.
2. For each transducer, type values for the Minimum and Maximum angles, or retain the default values.
   - For Edgetech conversion the Minimum and Maximum default values are -20 and 100.
   - For GeoAcoustics conversion the values are 30 and 75

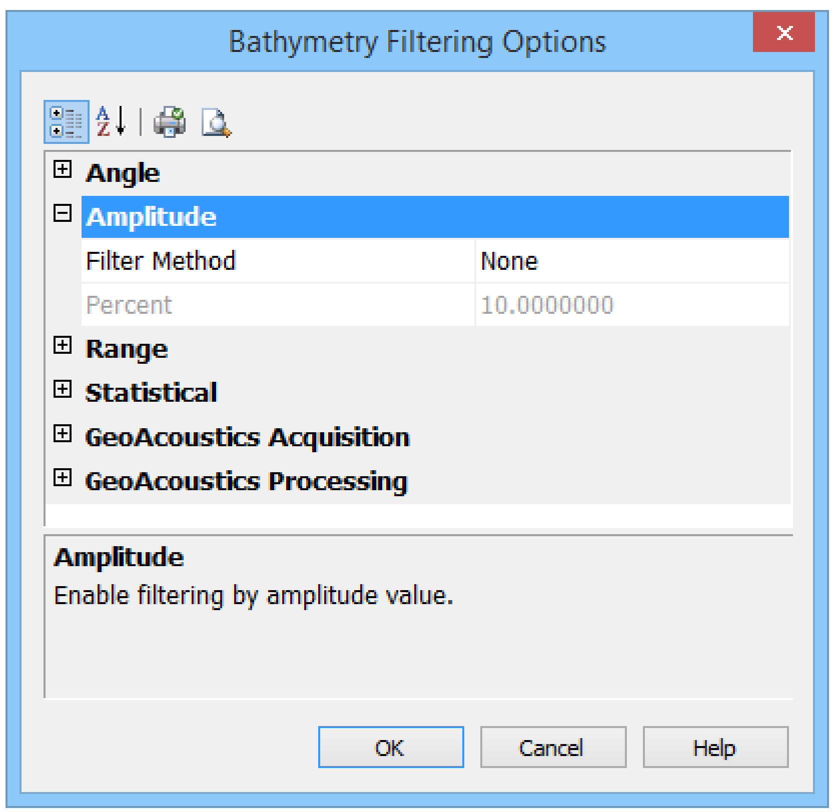
When all bathymetry filtering options are set:

3. Click **OK** to continue the conversion process.

Return to "Bathymetry Filtering Options" on page 134

Amplitude filtering

The *Amplitude Filtering* option filters soundings according to amplitude value. For each ping (port and starboard pings are handled separately), the min/max amplitude values are obtained, and samples that fall below the selected percentage (0-50%) are rejected.



To set the percentage:

4. Select Percent from the *Filter Method* field.

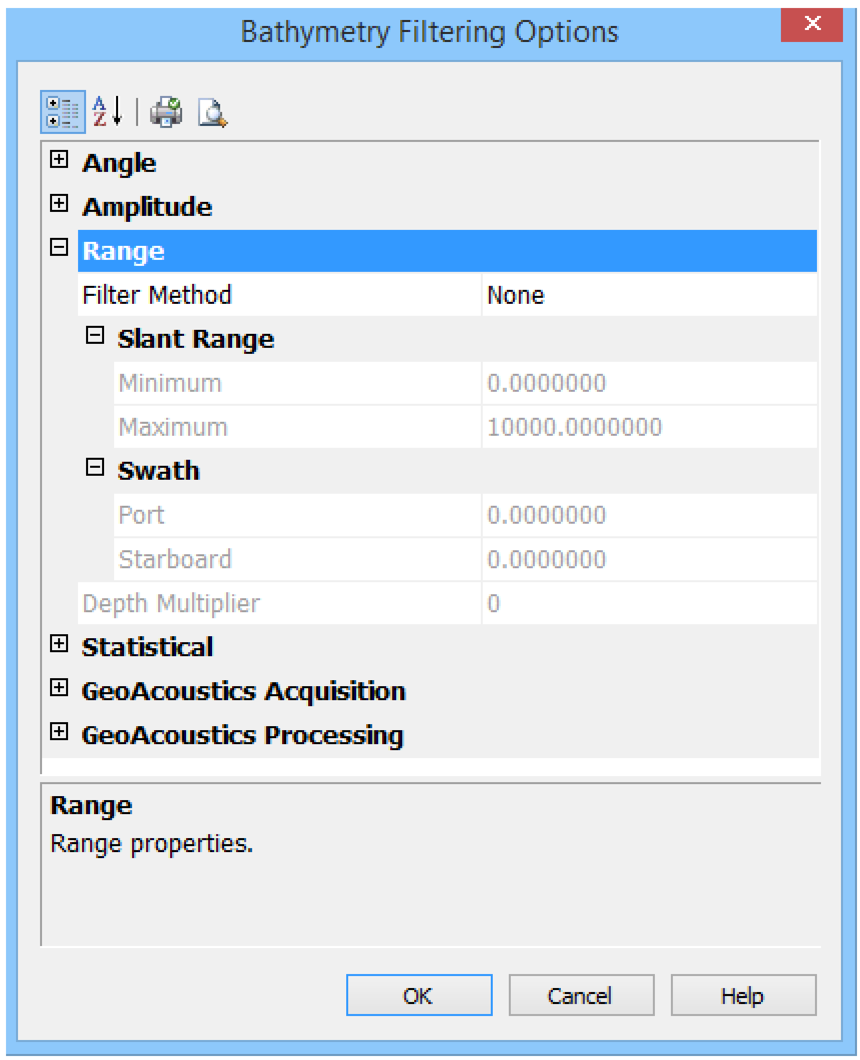
This activates the next field.

5. Type the percentage value below which data will be rejected.

Range filtering

The Range filtering converts soundings within a specified distance (in metres). Soundings outside of this range are not converted.

Filtering can be done using Slant Range, Swath or Depth Multiplier options. The default setting is “None”, and the range filter fields are greyed out to indicate that range filtering will not be applied.

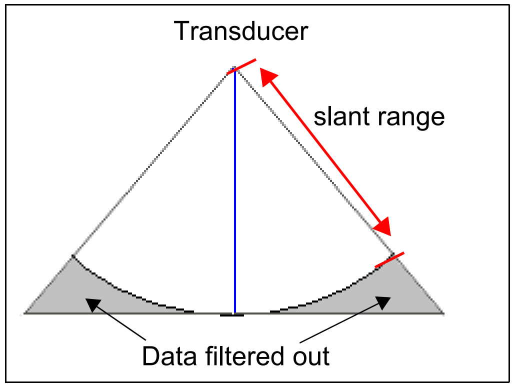


To apply range filtering:

1. Select one or all of the range filter options from the *Filter Method* drop-down list.

Slant Range

The Slant Range option will filter out all data outside a specified slant range.

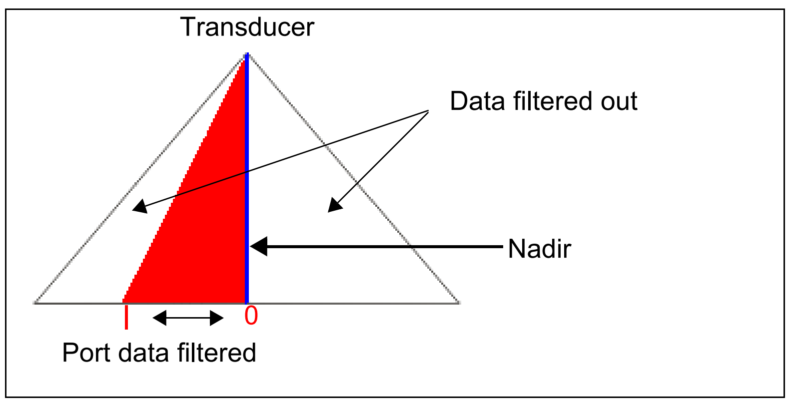


If Slant Range is selected:

2. Type minimum and maximum values for the range outside of which data will not be converted. Values can fall between 0 and 15000. The default range is 0 to 200.

Swath

The Swath option will filter all data outside the swath as determined by a set distance from nadir to port and/or nadir to starboard.



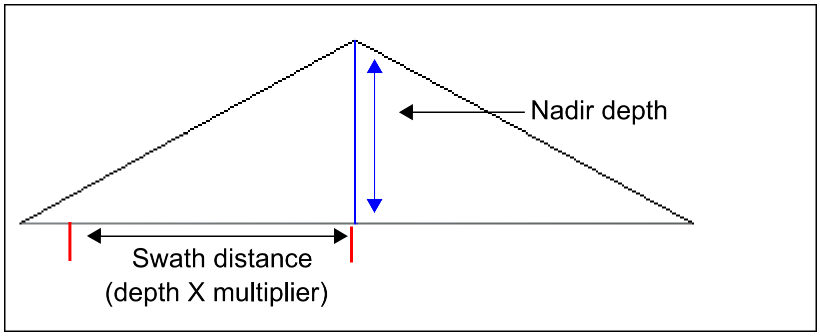
Port values define the distance from nadir to port outside of which data will not be converted. Starboard values define distance between nadir and starboard.

To use Swath filter:

1. Select Type Port and/or Starboard values between 0 and 1000 for the swath range. The default value is 50.

Depth Multiplier

Use the Depth Multiplier option to filter out data outside of a swath distance determined by multiplying the nadir depth by a selected value. The distance this generates is in the same units as the depth. For example a depth of 20 metres multiplied by a set value of 4 will filter out any data beyond 80 metres of swath distance.



To use Depth Multiplier:

2. Type a multiplier value between 0 and 50. The default multiplier is 8.

When all bathymetry filtering options are set:

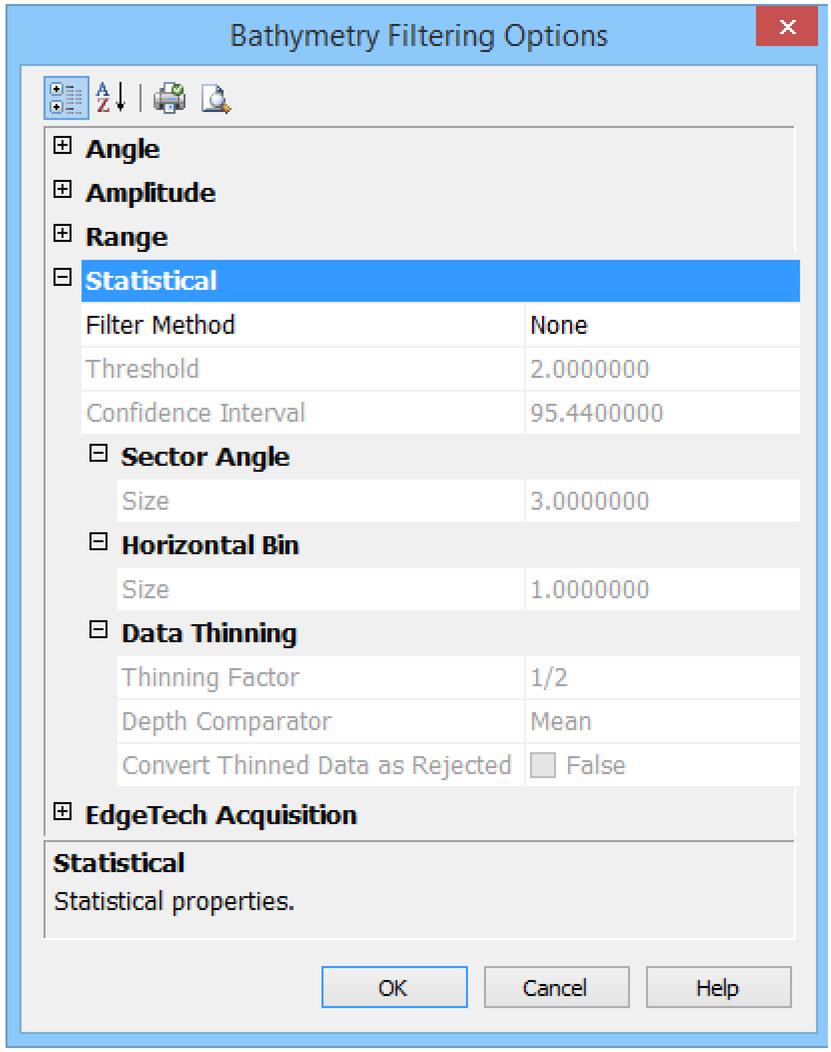
3. Click **OK** to continue the conversion process.

Return to "Bathymetry Filtering Options" on page 134

Statistical filtering

Sector Angle and Horizontal Bin filters can be used separately or in combination with Data Thinning. By default, none of these methods are selected. To apply statistical filtering:

1. Select a filtering method from the *Filter Method* drop-down list. By default there is no method set.



Statistical filters must have a threshold value or confidence level set. The threshold values can be 1, 2, and 3 (standard deviation or sigma). The default setting is 2 sigma.

2. Type a standard deviation value of 1, 2 or 3 in the *Threshold* field.

This generates the standard deviation expressed as a percentage in the *Confidence Interval* field, for example, when 2 sigma is set as the threshold, 95.44 is displayed in as the confidence interval. This field is read-only.

The Sector Angle and Horizontal Bin methods both use a defined area to determine the number of soundings to filter.

Sector Angle

Sector angle filtering uses a set angular filter area, measuring between 0.3 and 9.0 degrees. The default value is 1.

3. Type a value for the *Size* of the sector angle.

Horizontal Bin

Horizontal Bin filtering divides the swath into bins of a defined size in metres. Values between 0.03m and 5m can be set. The default value is 0.3 m.

4. Set the bin *Size* to a value representing distance on the ground.

Setting the bin size automatically calculates the number of bins of that size bins in the swath, and displays the value in the Number of Bins field.

Data Thinning

Only used with either of the other two statistical methods, thinning compares filtered values from a bin of data against either the mean or median value in the bin and retains the closest value to the mean or median.

The *Thinning Factor* uses a power of two reduction control (1/2, 1/4, 1/8, etc. to 1/256) so that one of two, one of four, or one of eight soundings can be converted per swath sector. The default value is 1/4.

5. Select a value for *Thinning Factor* from the drop-down list.
6. Select Mean or Median from the *Depth Comparator* drop-down list.

If you leave *Convert Thinned Data as Rejected* set to True, all data will be converted. Data which does not meet the filter value will be converted with “Rejected” status.

7. Set *Convert Thinned Data as Rejected* to False to not convert data that is filtered out. This is set to True by default.

When all bathymetry filtering options are set:

8. Click **OK** to continue the conversion process.

See also

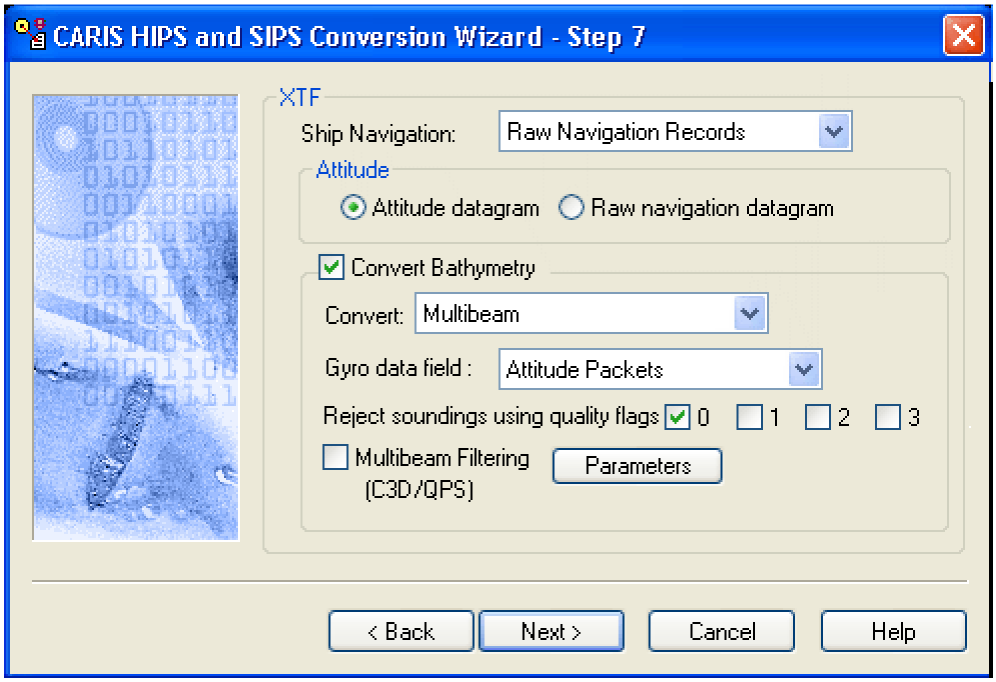
“EdgeTech Acquisition Options” on page 101

“GeoAcoustics Filtering Options” on page 108

Return to “Bathymetry Filtering Options” on page 134

XTF

The XTF format converter contains options for converting both bathymetry and side scan.



Ship Navigation and Attitude

For both sonar types, you must indicate the field used as the source for navigation and gyro/heading data.

1. Select from the drop-down list one of the following sources of navigation data:
   - Raw Navigation Records
   - Position Data Records
   - Ping Header: Ship
   - Ping Header: Sensor
   - Header Navigation Records

If you select *Raw Navigation Records*, the option to set Attitude data source is activated.

2. Select either Attitude diagram or Raw Navigation datagram.

Convert Bathymetry

To convert bathymetry data:

3. Click the *Convert Bathymetry* check box.
4. Select a bathymetric data format from the *Convert* drop-down list:
   - Multibeam
   - Multi-transducer
   - Single beam from records *{Prim=0}*, *{Prim=1}* or *{Prim=2}*:

     If there is more than one channel of data, you can identify which channel contains the primary data.

- Single beam from AUX (1,2,3, or 4).

Note: Benthos C3D data will be converted to two lines: multibeam and single beam.

The single beam HIPS line (with the addition of _SingleBeam to the line name) will have the same set of navigation and attitude data as the multibeam line, but the sounding depths will be single beam data from the XTF sensor AuxAltitude field.

Select gyro data field

The *Gyro data field* options in the pull-down list are determined by the selection made in the Ship Navigation field. (See "Ship Navigation and Attitude" on page 142.)

- If the *Ship Navigation* source is Raw Navigation Records, and the *Attitude* source is *Attitude Datagram*,
  - Select either Attitude Packets or CMG from Navigation (Course Made Good from Navigation) from the drop-down list.
- If the *Ship Navigation* source is Raw Navigation Records, and the *Attitude* source is *Raw navigation datagram,*
  - Select either Raw Navigation Records or CMG from Navigation from the drop-down list.

When any of the other Ship Navigation source options are selected, these Gyro sources options are available:

- Ping Header: Ship
- Ping Header: Sensor
- Attitude Packets
- CMG from Navigation
- CMG from SSS Navigation
- Header Gyro Records

Set reject soundings flags

The XTF format attaches quality flags to soundings. For example, with Teledyne Reson sonars, 0 represents the lowest quality and 3 represents the highest quality. You can reject soundings with a specific flag. These soundings are flagged as Rejected by Disabled Beam in HIPS.

5. Select a quality flag so the soundings with that specific flag value are rejected when converted to HIPS format.

If you are converting C3D or QPS data, you can apply filtering before moving to the next step. See "Multibeam Filtering" on page 144.

If you do not want to apply filtering

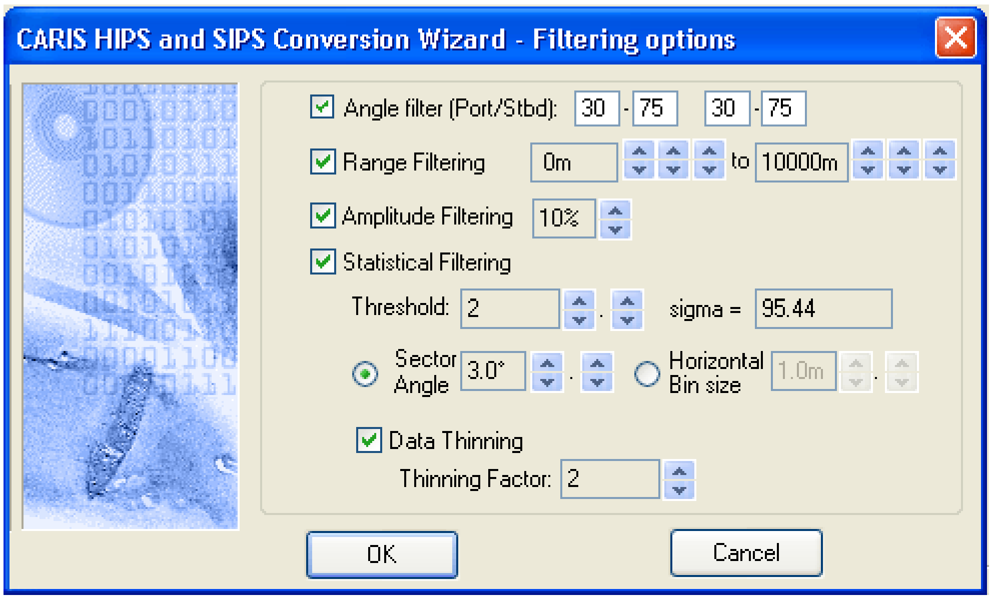
6. Click **Next** to continue to the next step.

Multibeam Filtering

To apply filters for Benthos C3D or QINSy QPS data

7. Select the *Multibeam Filtering* check box and click **Parameters** to select filtering options.

The Filtering options dialog box is displayed.



Benthos C3D data can be filtered so that only the best quality data is imported into HIPS. You can apply the filter to port/ starboard beam angles.

1. Type a range of beam angles to select the cleanest data from the port and starboard beams.

All other options are applied to QPS data only.

The *Range Filtering* option converts soundings within a specified distance (in metres). Soundings outside of this range are not converted.

2. Select the *Range Filtering* check box to implement this option.
3. Click the up or down arrow buttons to select a minimum and maximum distance.

The *Amplitude Filtering* option filters soundings according to amplitude value. For each ping (port and starboard pings are handled separately), the min/max amplitude values are obtained, and samples that fall below the selected percentage (0-50%) are rejected.

4. Select the Amplitude Filtering check box to implement this option.
5. Click the up or down arrow buttons to select a percentage value for rejecting soundings.

The *Statistical Filtering* option controls which soundings in each swath are considered for conversion. This option calculates the mean depth and standard deviation within a swath sector or horizontal bin. It then prevents any soundings that fall outside a multiple of the standard deviation from being converted.

If Statistical Filtering is used, then the mean within the sector is re-computed. Ultimately, the soundings are sorted by their residual from the mean and those closest to the mean are converted. The actual number of soundings converted from within each sector is determined by the thinning factor (if used).

6. Select the *Statistical Filtering* check box to implement this option.
7. Determine a *Threshold* (multiples of the standard deviation) value by clicking the up and down arrow buttons.

The equivalent confidence value is displayed as a percentage.

The filtering and thinning methods are executed within a sector angle interval or a horizontal bin size. The sector angle option divides the swath into sectors according to degree angles while the horizontal bin size divides the swath into horizontal sectors based on a specified across-track distance.

Vessel motion and transducer mounting angles are considered when sector angle and horizontal bin locations are determined.

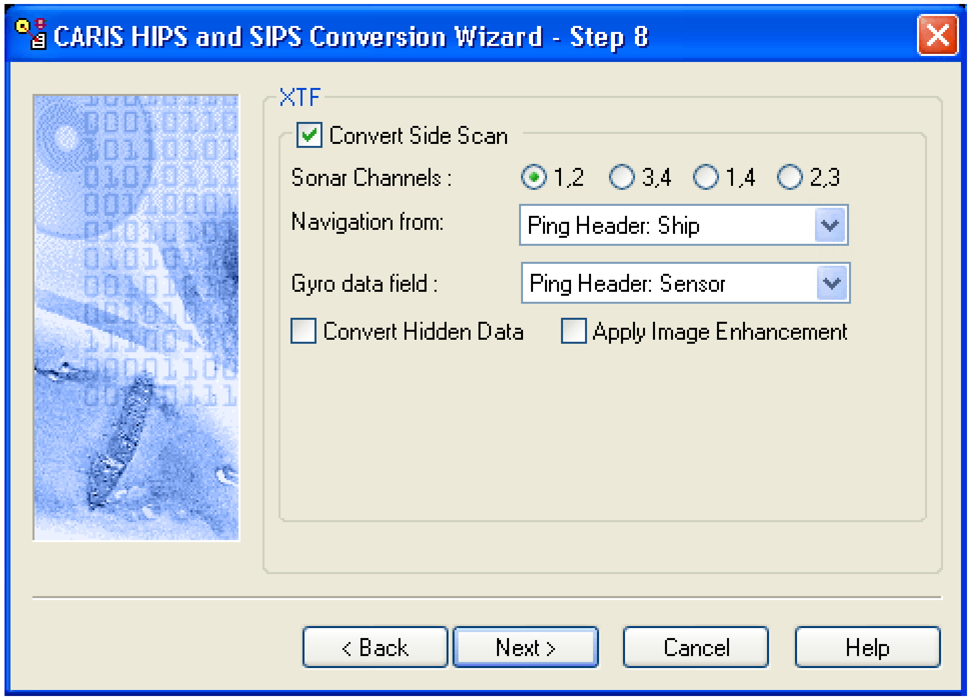
1. Select the *Sector Angle Interval* option and choose a degree level (to a maximum of 10º) by clicking the up or down arrow buttons.
2. As an alternate method, Select the *Horizontal Bin Size* option choose an across-track distance by clicking the up or down arrow buttons.

The thinning factor reduces the number of soundings converted on a per swath basis. The thinning factor uses a power of two reduction control (1/2, 1/4, 1/8, etc.) so that one of two, one of four or one of eight soundings can optionally be converted per swath sector.

3. Select the *Data Thinning* check box to implement the thinning option.
4. Select a *Thinning Factor* value by clicking the up or down arrow buttons.
5. Click **Next** to go to the Convert Side Scan Data dialog box.

Step 8: Convert Side Scan data

If you do not want to convert side scan data, click **Next** and go to “Convert Data” on page 95.



Note: Contact records created in Triton ISIS are also imported into HIPS when associated XTF data is imported.

To convert side scan data:

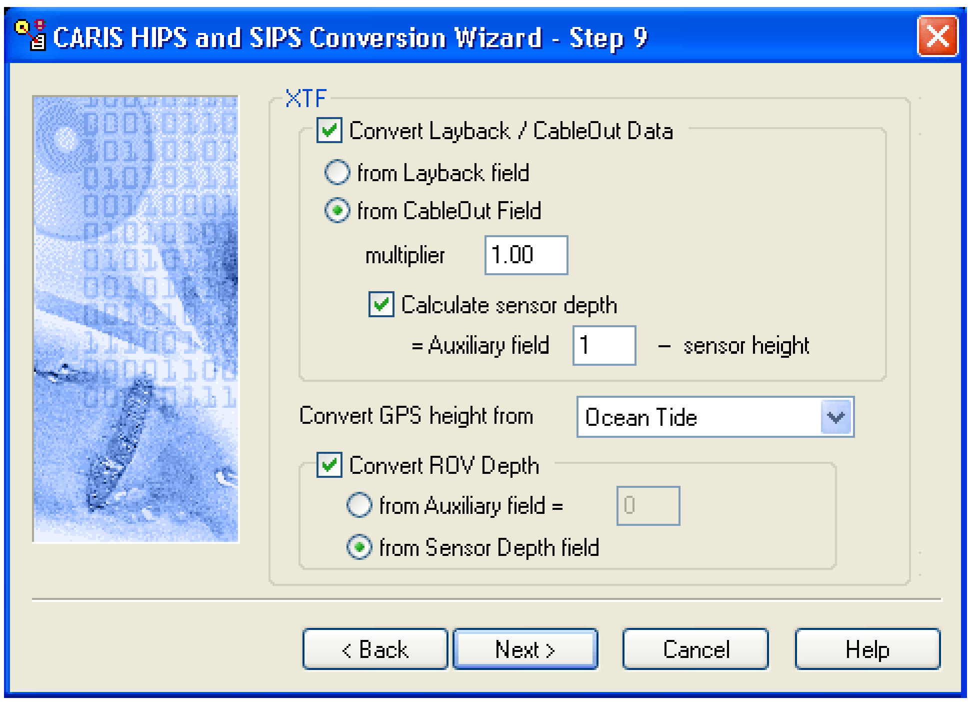
6. Click the *Convert Side Scan* check box.
7. Select one of four *Sonar Channels* pairs by clicking the radio button.
8. Select one of the following as the source for reading the *Navigation* data for the towfish:
   - Raw Navigation Records
   - Position Data Records
   - Ping Header: Ship
   - Ping Header: Sensor
   - Header Navigation Records
9. Select one of the following options from the drop-down list as the source for reading from the *Gyro data fields*:
   - Ping Header: Ship
   - Ping Header: Sensor
   - Attitude Packets
   - Raw Navigation Records
   - CMG from Navigation
   - CMG from SSS Navigation
   - Header Gyro Records

The XTF format flags repeated pings over an area as hidden data.

10. Select the *Convert Hidden Data* check box to include this data type in the SIPS project.
11. To enhance the image by scaling the intensity values, click the *Apply Image Enhancement* check box.
12. Click **Next**.

Step 9: Convert Layback

This dialog box contains options for converting horizontal tow fish layback or tow cable length distances in side scan data. There are also options for reading the dynamic depth information for a remote operated vehicle (ROV).



13. If you want to recompute side scan towfish positions in HIPS with the recorded horizontal layback or tow cable length data, make sure the *Cable Out/Layback* box is checked.
14. Select one of the following options:
    - *from Layback field*: The distance from the vessel towpoint to the towfish.
    - *from Cable Out field*: The length of the tow cable to the towfish.
15. If you selected *Cable Out*, enter a number in the *Multiplier* box to correct for integer value recording, if necessary.

If you have stored sensor depth data in an auxiliary field, you use it to calculate sensor depth. Sensor depth in this case is equal to the value in the field minus the sensor height.

16. Select the *Calculate Sensor Depth* check box.

17. Type the number of the auxiliary containing the total measured depth.

GPS Height

The total measured depth and the digitized height of the towfish are used to compute the depth of the towfish below the waterline. The towfish depth can then be combined with the tow cable length data and the vessel's tow point configuration to compute the position of the towfish.

The GPS Height field indicates the source of the GPS ellipsoid height data. The source options will vary depending on the Ship Navigation option selected in Step 7 of the Conversion Wizard.

- If ship navigation is being read from the Raw Navigation Records then the GPS height is set to Raw Navigation Records, and cannot be changed.
- If Ship Navigation set to Position Data Records, the GPS height control shows only RTK Packets, and the user cannot change it

For all other Ship Navigation selections, the options available for the source of GPS ellipsoid height data are the *RTK Packet* and the *Ocean Tide* field.

18. If data is being collected from an ROV (Remote Operated Vehicle), click the check box to enable the source options for measurement of vehicle depth.

The auxiliary and sensor depth fields are enabled.

19. Click the appropriate check box to select a source for reading ROV depth data:
    - Auxiliary
    - Sensor Depth

20. Type the number of the *Auxiliary* field (from 1 to 6), if this option is selected.

21. Click **Next** to go to the final step in the Conversion Wizard, to "Convert Data" on page 95.

5

Load Auxiliary Data

Use the Load Auxiliary Data function to augment or replace components of the converted data with data on position, heave, motion, attitude, RMS, etc.

In this chapter...

Auxiliary Data

Use the Import Auxiliary Data function to load post-processed positioning and attitude data, previously applied using the Load Navigation/Attitude, Load Delayed Heave and Load RMS commands.

These post-processed data types include:

- ASCII
- Applanix POS MV
- Applanix RMS
- Applanix SBET
- Kongsberg Navlab Position / Attitude (BIN file format)
- NovAtel
- PFree Heave
- Fugro Starfix (project folder)
- TerraPOS

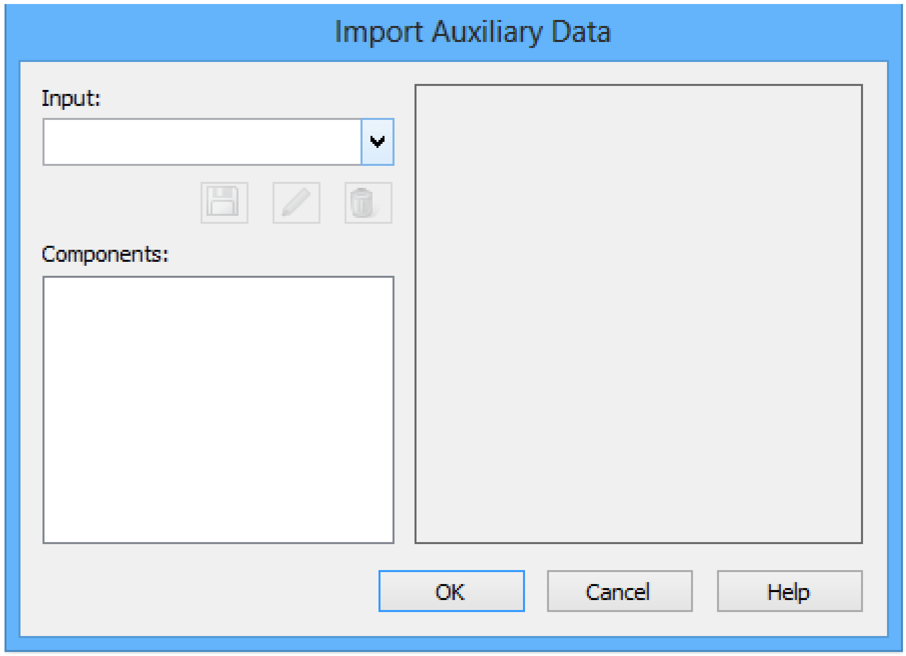
After selecting an input data type, you can set import and timing options and set the start date for a reference week. These configurations settings can then be applied, or saved as reusable templates.

Menu	File > import >Auxiliary Data
Tool	

To import auxiliary data:

1. Select one or more track lines.
2. Select the Import Auxiliary Data command.

The Import Auxiliary Data dialog box is displayed.

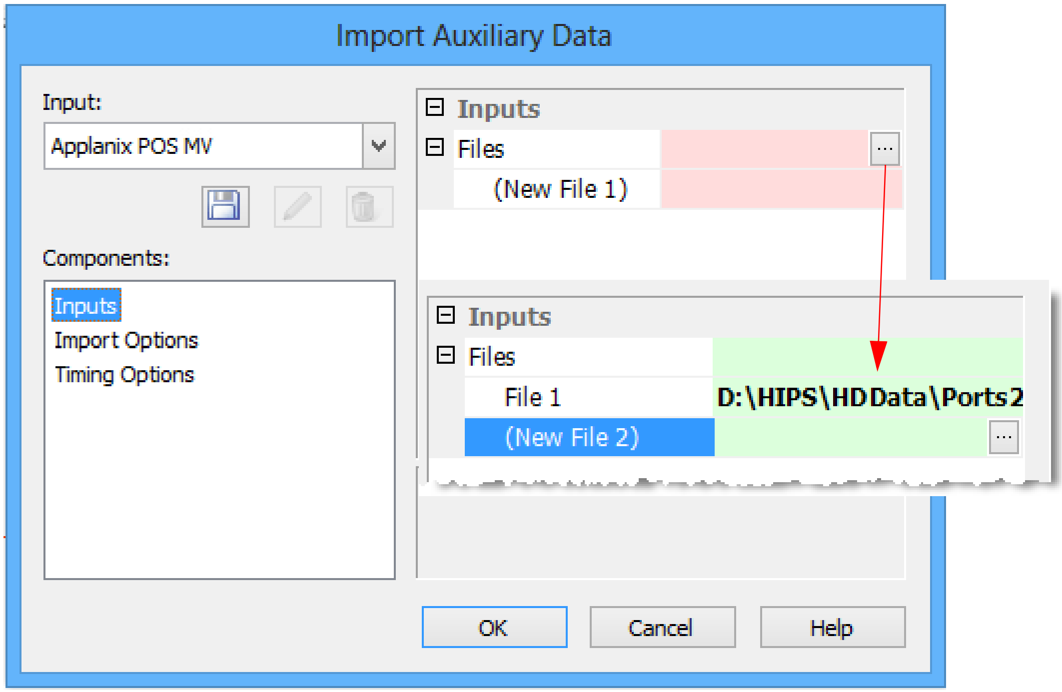


3. Select a data format from the *Input* list.

The *Inputs* component is highlighted. Depending on the format selected, a *Files* field or a *Folders* field is displayed.

4. Click Browse in the *Files* field and navigate to the location of the data file to be imported.
5. Click **Open**.

The path to the selected file is displayed in the File 1 field as illustrated below.



Multiple files can be imported at the same time, To add another file:

6. Click Browse in the *New File 2* field and select the data.
7. When you have selected all the files to import, click **OK**.

When importing Fugro Starfix data, select Folders containing the data.

Import Options

Import options enable you to set which data types are loaded, and to set values for them.

1. Select Import Options.

The Import Data fields displays the data types that can be loaded for each input format. These data types include:

Navigation	Pitch RMS
Gyro	Roll RMS
Pitch	GPS Height RMS
Roll	Delayed Heave RMS
GPS Height	Subsea Depth
Delayed Heave	Subsea Depth RMS
Navigation RMS	
Gyro RMS	

To apply specific data:

2. Select the check box for the data type you want to load.

This will activate the sampling rate value so it can be edited. The default value is .02 seconds.

> If a sampling rate of “0.0” is specified, all data is loaded with no down-sampling.

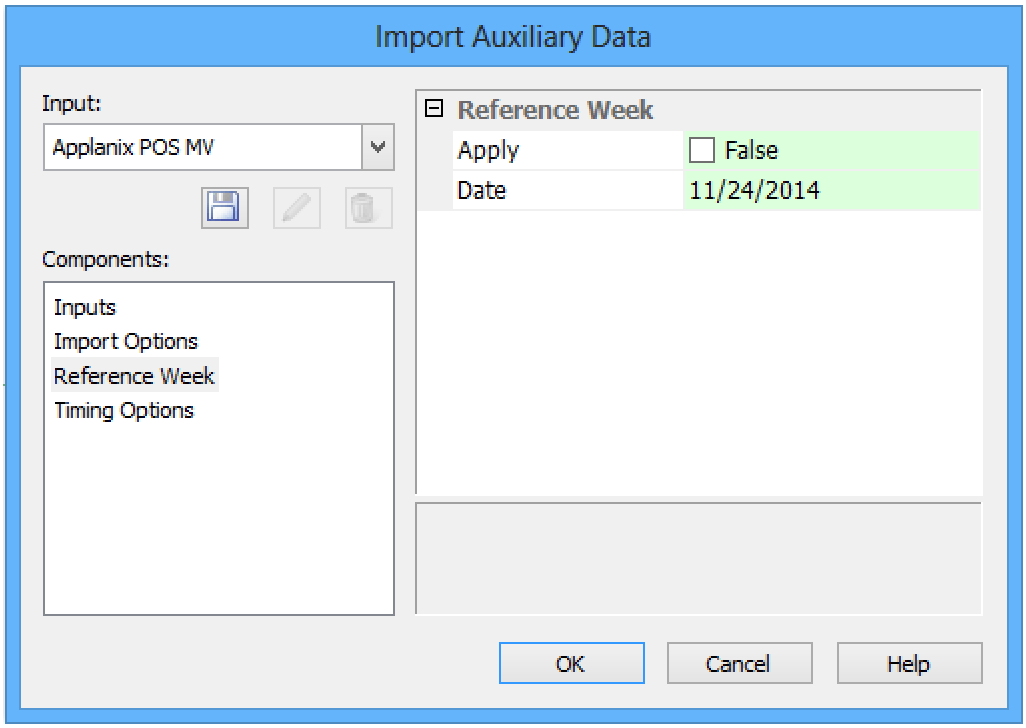
Additional options unique to ASCII and Starfix are also displayed when these are selected as Input:

Data format	Description
ASCII Options	
ASCII INFO File	Click **Browse** to select Info file.
Coordinate System	Click **Browse** to specify the coordinate system of the input data.
Starfix Options	
Coordinate System	Click **Browse** to specify the coordinate system of the input data.
Vessel	Click **Browse** to select the name of the Starfix device to be loaded.
Navigation Device	
Heading Device	
Heave Device	
Pitch/Roll Device	
Sub-sea Depth Device	

See also "Edit a Format Information File" on page 713.

Reference Week

Certain data formats, such as Applanix SBET, must be time-stamped with valid GPS entries to fix the time of data acquisition.



1. Set *Apply* to True to apply the selected GPS reference week.
2. Click in the *Date* field to pick the start date of the reference week from the calendar.

Timing Options

Timing options can be set for any of the selected formats. The options are:

- *Time Offset*: Enter a value in seconds for the offset.
- *Time Buffer*: Enter a value in seconds to apply extra data at the beginning and end of lines.
- *Maximum Allowed Gap*: Enter a value for the maximum time allowable between records.
- *Allow Partially Covered*: Set to True to load data for lines that are only partially covered.

Saving a template

Any configuration of settings can be saved as a template.To save options and value settings:

Tool	

1. Click the **Save** button.
2. In the Save Template dialog box, enter a name for the new template.
3. Click **OK**.

Menu	File > import >Auxiliary Data
Tool	

To use a saved template:

1. Select one or more track lines.
2. Select the Import Auxiliary Data command.
3. In the Import Auxiliary Data dialog box, select the template by name from the *Input* list.
4. Use the Browse function in the File and Folder fields to select files.

The template will display all other settings and properties as read-only.

To change a template:

Tool	

5. Click the **Edit** button.

This will activate all the fields.

6. Save your changes.

6

Sound Velocity Correction

Sound Velocity Correction contributes to a more accurate soundings file by applying travel time and angle information into across-track and depth values. The process combines transducer orientation and positioning data from the HVF with Sound Velocity Profile data and applies a ray-tracing algorithm.

In this chapter...

Sound Velocity Processing

Raw data formats such as XTF contain two-way travel time and beam launch angle data. The Sound Velocity Correction algorithm uses this data to calculate the length and path of the sound wave through the water column for each beam.

Sound velocity processing is not available for all types of multibeam and single beam sonar data. Some of the supported sonars are from Atlas, EdgeTech, EIVA, Elac, Imagenex, Klein, Kongsberg GeoAcoustics, Kongsberg Maritime, SEA SWATHplus, Teledyne Odom and Teledyne Reson.

Only those sonar formats that provide raw travel time and angle from the transducer are currently supported in HIPS.

Supported formats for multibeam/interferometric/SAS bathymetry include:

- ALL
- D1P
- GSF
- JSF
- HSX
- PDS
- RDF
- S7K
- SBD
- SDF
- SURF
- SXR
- TDY
- XSE
- XTF

For other types of multibeam sonar formats, it is assumed that sound velocity corrections were applied during data acquisition.

HVF

For effective sound velocity correction, the Sound Velocity section of the HIPS Vessel file must be set up to define the transducer offsets. These offsets are used in Sound Velocity Correction, together with the Sound Velocity profile data. (See "SVP" on page 52 for these sensor configurations.)

SVP file

A sound velocity profile contains data on the acoustic velocity of the water column, in a text file divided into sections defined by time stamps. This data can be viewed and modified in the SVP Editor .

> The process of Sound Velocity Correction cannot be reversed. Once it is applied, data cannot be restored to its pre-SVC state.

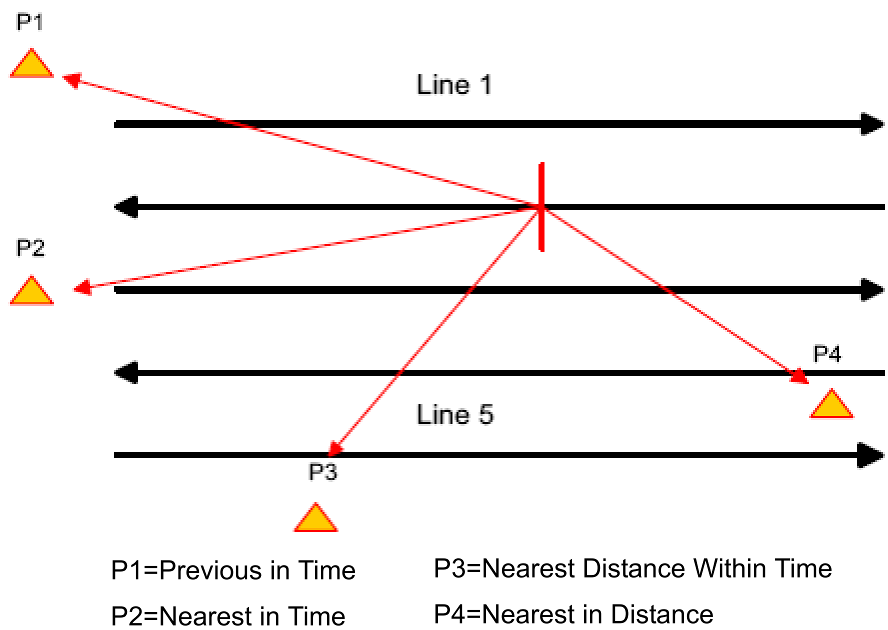
Profile selection method

When there is more than one sound velocity profile, HIPS and SIPS can use one of four options for selecting an SVP at the time of each swath:

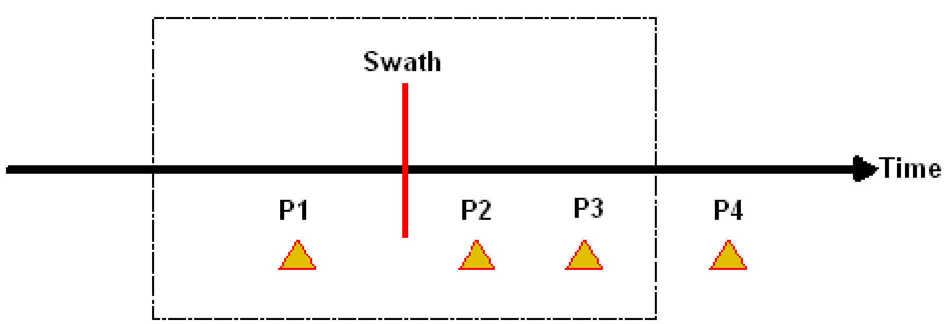
- *Previous in Time*: Select the profile with a time stamp prior and nearest to the time of the swath.

- *Nearest in Time*: Select the profile with a time stamp nearest to the time of the swath. This can either be prior to or after the time.
- *Nearest in Distance*: Select the profile with a position nearest the position of the swath. This requires that positions be included in the SVP file.
- *Nearest Distance Within Time*: Select the profile with a position nearest to the swath and has a time stamp nearest the time of the swath within a selected range.

The following diagram demonstrates the relationship of the profile options to the time/location of the swath.



P1, P2 and P3 all are calculated within a time period. P4 is calculated independently of the time period.



Sound Velocity Correction

The sound velocity correction process uses a ray tracing algorithm to apply the sound velocity profiles. You can use either regular attitude data or smoothed data for applying vessel motion data to the ray-tracing algorithm (see "Attitude Filter" on page 329).

The SVC process will select the profile to apply based on the method you select. You can also choose to apply the last method used.

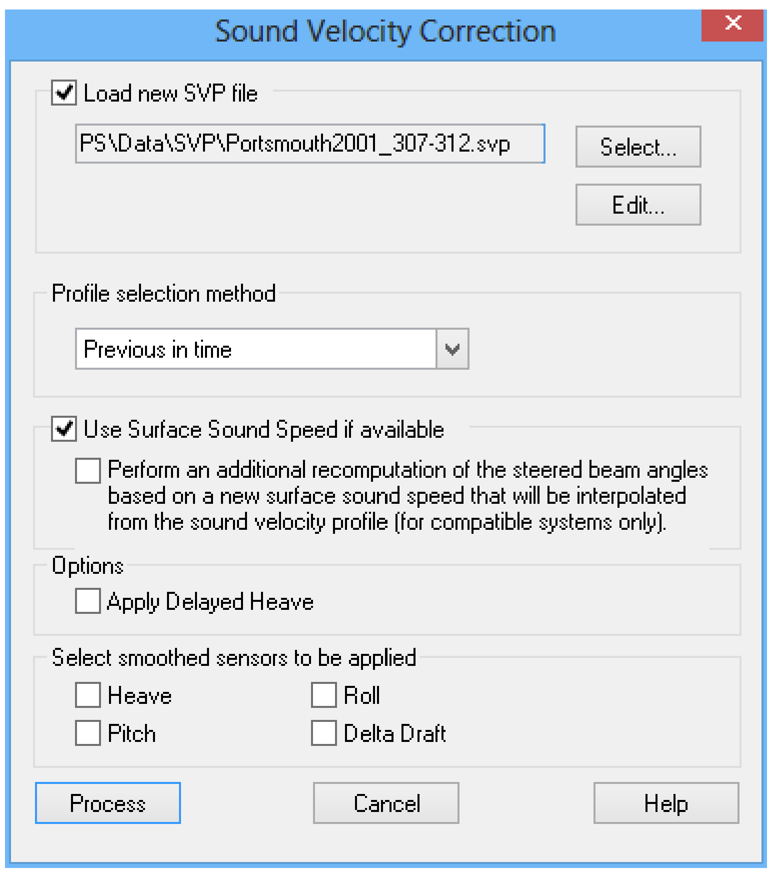
> It is recommended that sound velocity correction be applied to bathymetry before cleaning the data. Whenever SVC is applied, the Merge command should also be applied to the data

To apply Sound Velocity Correction:

Menu	Process > Sound Velocity Correction
Tool	

1. Select a track line or group of track lines.
2. Select the Sound Velocity Correction command.

The Sound Velocity Correction dialog box is displayed.



SVC will use either the sound velocity profile from your converted data, or the last SVP file applied to the data. If SVC has been previously applied, the name of the SVP file used will be displayed

You can choose to load another profile, or you can edit an existing profile.

To use a profile:

3. Select the L*oad new SVP file* check box, and click **Select** to choose an SVP file.

The “Browse for SVP file” dialog box is displayed.

4. Select the file you want, or type the file path in the File Name text box. The default directory is ...\Hips and SIPS\Data\SVP.
5. Click **Open**.

The file name and path is displayed in the text box.

To edit the data, click **Edit** to open the SVP Editor. (See “Edit SVP Data” on page 154.)

Profile selection method

When there is more than one sound velocity profile for the data, HIPS can use one of four options for selecting an SVP.

6. Select a method from the drop-down list:
   - *Previous in time* - Apply the SV profile that was taken just prior to the recording of the survey data.
   - *Nearest in time* - Apply the SV profile that was taken closest in time to the recording of the survey data.
   - *Nearest in distance* - Apply the SV profile that was taken closest to the area where the survey data was recorded.
   - *Nearest distance within time* - Apply the SV profile with a position that is nearest to the swath and has a time stamp that is nearest to the time of the swath, within a selected range.

You also have the option of applying the last correction method last used on each line, to the currently selected data.

7. Select *Last used method* to apply the last selected profile method to the current SVC.

> If a line has not had SVC applied previously, attempting to use *Last used method* for correction will cause the process to fail. In this case, choose another SVC method.

Use Surface Sound Speed

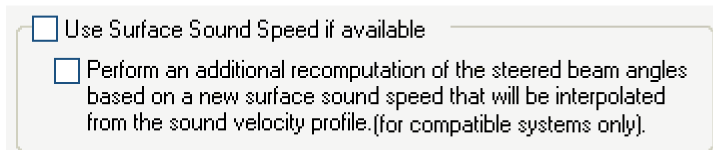
For systems which employ a flat transducer design, it is critical for the beam-forming electronics to know the exact sound speed at the location of the transducer. Therefore a sound speed probe is installed at the transducer. When this surface sound speed data is available, HIPS converts it and stores the time series data in the SSP file (in the line directory). This data can be displayed in the Attitude Editor.

When HIPS performs sound velocity correction, the SSP data, if available, is retrieved and “inserted” into the Sound Velocity Profile (SVP) at the location of the transducer depth, replacing the interpolated sound speed value at that depth. Ray-tracing then starts from that location.

> If Surface Sound Speed is available it is applied automatically when using the updated CARIS SVC for Kongsberg data.

To use available SSP data in sound velocity correction:

8. Select the *Use Surface Sound Speed if available* check box in the dialog box.



There are times, however, when the SSP data is not correct, causing the beam-forming electronics to erroneously compute beam angle data. If you determine that this has happened, you can enable the option to recompute beam angles. The recorded SSP (known to be incorrect) along with the sound speed interpolated from SVP (assumed to be correct), will be used to compute new beam angle data. Ray-tracing will then proceed without insertion of the SSP into the SVP.

To use the available SSP data only to compute new beam angle data:

9. Select the *Perform an additional recomputation* check box to re-compute the steered beam angles based on a new surface sound speed that is interpolated from the sound velocity profile.

Only certain systems support the recomputation of the steered beam angles. These include: Kongsberg EM 12/120/2040/3000-3008/3020/300/710/122/302 (except when using Kongsberg SVC Licenced `dll`), Atlas SURF, GeoAcoustics RDF, Teledyne Reson 7K (when using the 7004/7006 depth records), Benthos C3D and SWATHplus.

Apply Delayed Heave

In previous versions of HIPS, SVC used TrueHeave data in place of regular heave data when TrueHeave was available. Do not select this option if you want the original heave data to be applied.

10. Select Apply Delayed Heave to apply delayed heave values instead of regular heave.

If you choose this option, and no delayed heave data is available, there will be a warning message in the Output window, and regular heave data will be used instead.

Smoothed Sensors

11. Select which smoothed sensor data is to be applied during sound velocity correction: Heave, Pitch, Roll, Delta Draft. (Sensor data can be smoothed in the Attitude Editor. See "Data Options" on page 730).

Note: Waterline values are not interpolated during SVC.

12..Click **Process** to apply the SVP file to the selected lines.

Lines which have been SV Corrected will be displayed in the Display window in the Not Merged colour (as set in the Properties for the Ship Track Lines layer).

The *SVP Corrected* field in the Selection window will show "Yes" for any selected line to which SVC has been applied.

Query Applied SVC Profiles

You can confirm whether SVC has been applied, and see which profile has been applied to which lines, by:

- querying the SVC status of lines in the Selection window
- display the SVP file in the Display window
- colour coding line display by SV profile
- examining the logfile.xml

Query lines

To query lines which have been sound velocity corrected:

1. Select the track lines in the Display.

These selected lines will be listed automatically in the Selection window.

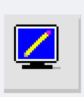
2. Scroll across the Selection window to the *SVP Corrected* column.

A “Yes” value in the column indicates that the line has been SV corrected.

Display profiles

SVP files can be opened and viewed as a layer in the Display window.

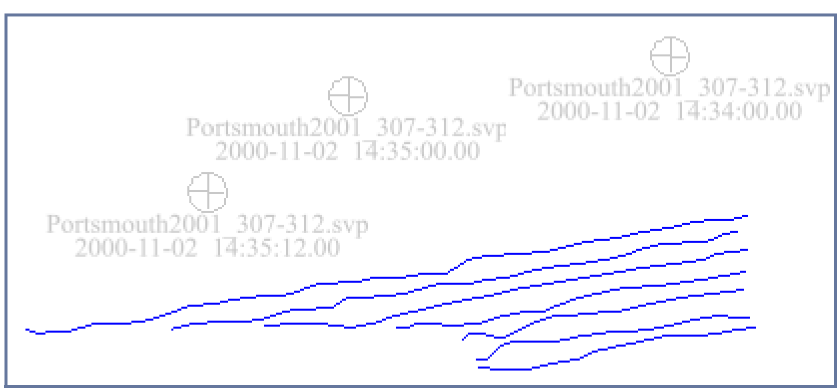
Menu	File > Open

Menu	View > Refresh
Tool	
Key	<F5>

To view the profiles:

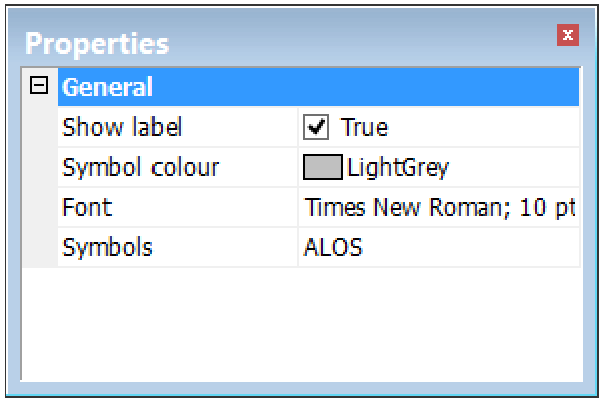
1. Select the Open command.
2. Browse to the SVP file that was applied to the lines, and click **Open**.
3. Click **Refresh.**

The open file displays the various profile locations as a symbol with a text label.



To change the display of the SV profile colour and symbology:

4. Select the SVP profile file in the Layers window.
5. Open the Properties window.



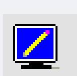
6. Set *Show label* to “False” to hide label, leaving the symbol displayed.
7. Click in the Symbol colour field to pick a new colour for the symbol and label from the chart.
8. Change the *Font* style and size by selecting from the drop-down list.
9. The default feature code for the symbol can be changed by selecting another from the drop-down list in the *Symbols* field.

Colour Lines by SV Profile

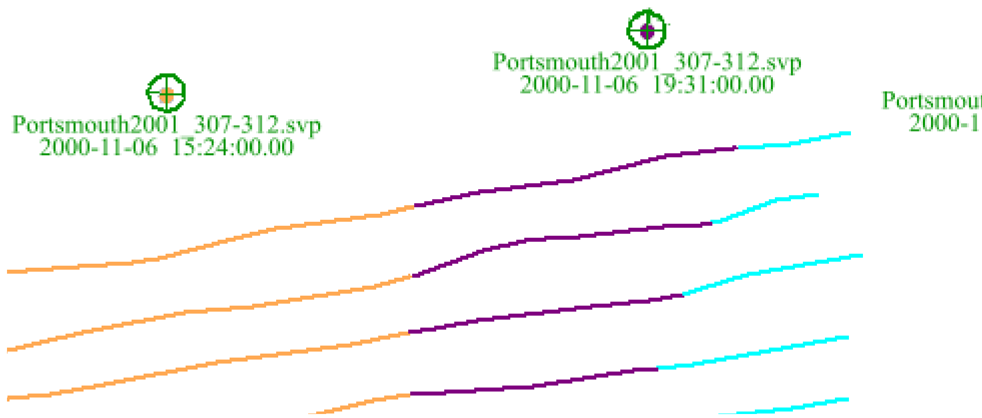
You can determine which profiles have been applied to lines by turning on the *Colour by SV Profile* property. This is located in the line properties for project.

To view the profiles that have been applied in SVC:

1. Select the Ship Track Lines layer in the Layers window.
2. Open the Properties window.
3. In the *Colour by* field, select SV profile from the drop-down list.
4. Click **Refresh**.

Menu	View > Refresh
Tool	
Key	<F5>

The track lines will display a colour for each cast, as below:



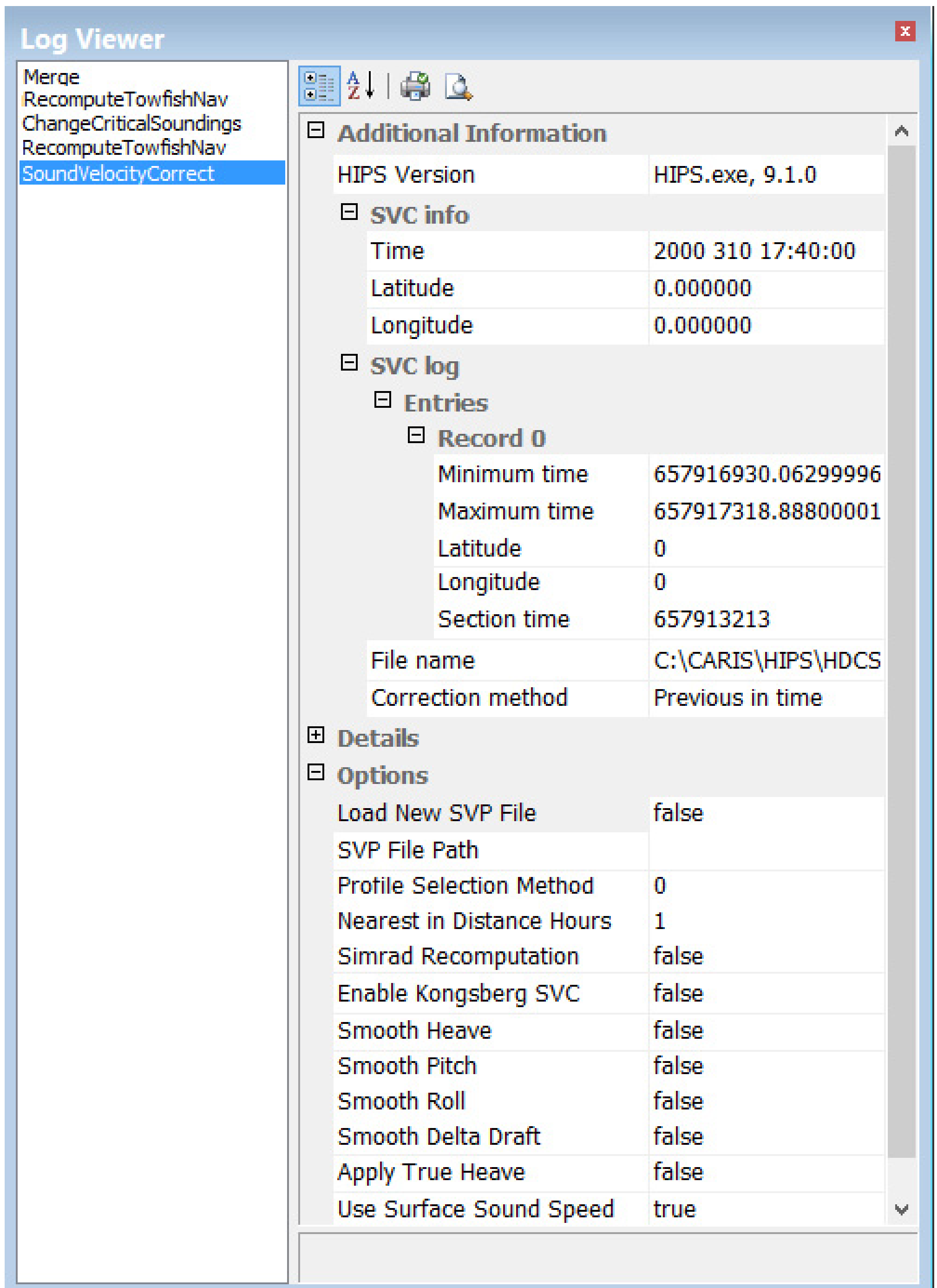
If the SVP values are not displayed, check that a colour set for SV Profile is not set to the same colour as your Display window background.

Log Viewer

To display the SV profile information in the process Log Viewer:

1. Open the Log Viewer from the Window> Other Windows menu.
2. Select `SoundVelocityCorrect` from the left column of the Log Viewer.

The SVC information is listed in the right column of the viewer.



1.

7

Correct for Tide

Tidal data must be loaded for every track line to generate final depths. Tide files can be edited or new files created using Tide Editor.

In this chapter...

Load Tide

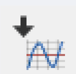
Tidal observation data must be loaded for every track line before the Merge process can be executed. Tide data is used to generate final depths relative to the tide datum by subtracting the tide from the sounding depth.

There are two options for tide loading:

- load a *.tid file to apply tide observations from a *single* station. (For information on these files, see Tide Files .)
- load a *.zdf file to apply tide observations from *multiple* tide stations to a tide zone. (For information on these files, see Tide Zone Files .)

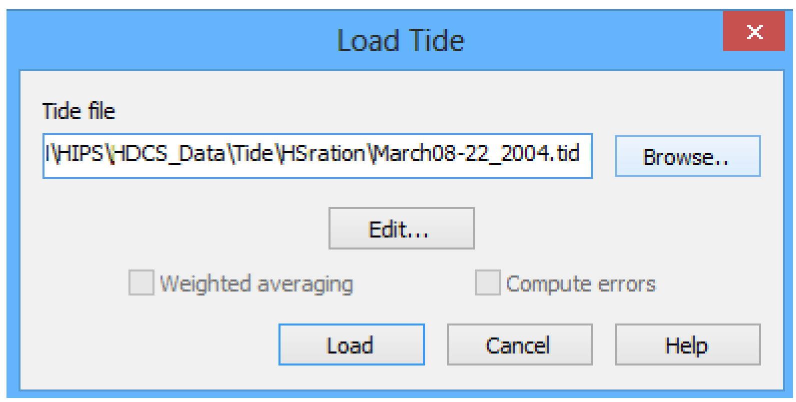
To apply tide observation data to selected HIPS track lines,

1. Select the track line(s).

Menu	Process > Load Tide
Tool	

2. Select the Load Tide command.

The Load Tide dialog box is displayed.



3. Click **Browse…** to choose a *.tid file for single station data, or a *.zdf tide zone file.
4. Click **Open**.

The tide file name is displayed in the field.

5. [Optional] To revise the contents either type of tide file, click **Edit** to open the Tide Editor.

Tide stations and weighted average

If you are loading a tide zone (.zdf) file, you have the option of applying weighted averaging from the multiple tide stations.

When an average tide is calculated from multiple tide stations, the weight given to a tide station's data is inversely proportional to its distance from the vessel at that time. In other words, the further a station is from the survey line, the less weight is accorded to the data from that station, and the greater weight is given to data from the closest tide station.

If the selected tide file doesn't contain any averaging options, this option is not available.

6. Select the *Weighted Averaging* check box to apply this option.
7. Select the *Compute Errors* check box to have the Load process save computed errors to a tide error file for each line. These tide error files will then be used in computing TPU. If the selected tide file does not contain any error parameters, this option is unavailable.
8. Click **Load**.

The tide data is loaded into the selected track lines from the tide zone definition file.

Load Tide status

To determine if tide has been loaded for a line or lines:

1. Select the track line(s).
2. Select the Selection window.

The Selection window will display line data for the selected line(s) in columns.

The *Tide Loaded* column shows "Yes" when tide has been loaded for the selected line(s).

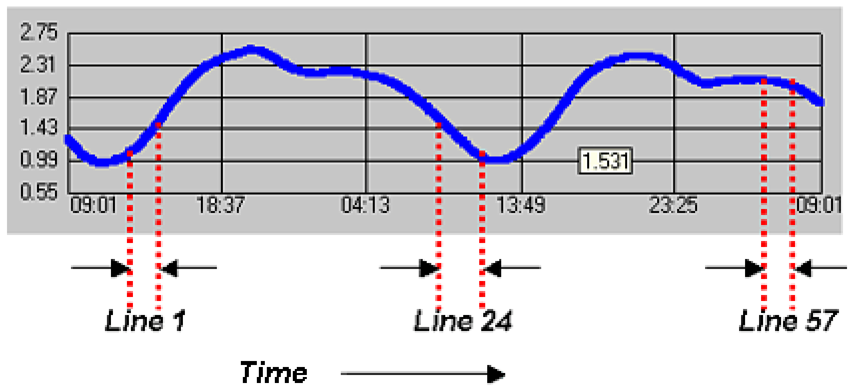
The *GPS Tide* column shows "Yes" when tide has been loaded for the selected line(s).

The *Tide Applied* column can display one of four values:

- **None**: No tide has been applied (Merge has never been run, and no ProcessedDepths exist.)
- **Observed**: The line has been Merged, and choose to apply Observed tides from loaded TID file(s).
- **GPS**: The line has been Merged, and a GPS tide file has been applied.
- **Zoned**: The line has been Merged, and a previously loaded ZDF file has been applied.

Tide Files

When tide observations from a *single* station are applied to a track line (or lines), the time extent of the line(s) is used. The illustration below shows tide values from a single station applied to three selected track lines, based on the time each line was recorded.



Tide data from a single station is loaded from files with the extension *.tid. These files can be created or edited in a text editor (such as Notepad), or with the Tide Editor.

HIPS supports a standard Canadian Hydrographic Service tide format called COWLIS, as well as two NOS/NOAA formats and the NHS file format. HIPS also supports a basic format consisting of just the required date, time, and tide values.

Track lines in areas without water level changes, such as rivers or lakes, must also have a tide file. In this case, or if tidal data cannot be obtained, a "zero tide" file—where all tide/time values are zero—is used so merging can proceed.

The zerotide.tid is located following installation in...`\Program Data\CARIS\HIPS\`*`version`*`\Template`

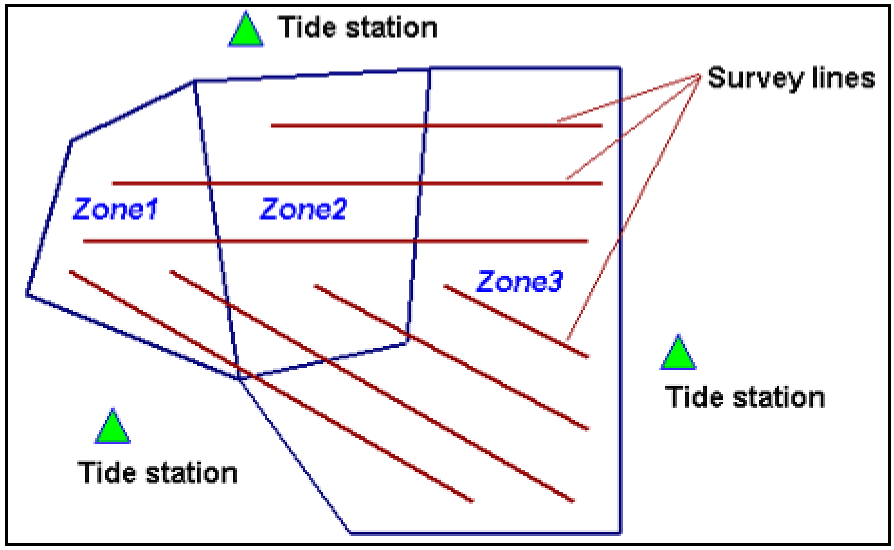
For more detail, see "Tide File Formats in the Reference Guide.

Tide Zone Files

When tide observations from *multiple* tide stations are applied to a line or lines, tide zones are used.

Where a survey line falls within two or more tide zones, the tide data to be loaded for each part of a line is determined by the zone in which that part of the line falls (as illustrated below).

The tide data file must span the entire time frame of the track line. The time zone of the tidal observation data must match the time zone of the survey data to which it will be applied.



The output from the Load Tide function (see LOAD TIDE) attaches a binary HIPS tide-time series to each track line.

The tide zone option assigns tide observations to track lines from multiple tide stations based on the time period in defined geographic zones.

Each tide zone is defined by a closed polygon with tide, time and range corrections for a primary station, plus up to three backup secondary tide stations. Tide zone should not overlap each other.

HIPS uses a Zone Definition File (with the extension *.zdf) for defining tide zone data. This is an ASCII file and can be prepared with any text editor.

For complete description of the ZDF format, see “TIDE ZONE DEFINITION FILES in the Reference Guide .

Display tide zones

Tide zones and station locations can be viewed in the Display window.

Menu	File > Open

1. Select the Open command.

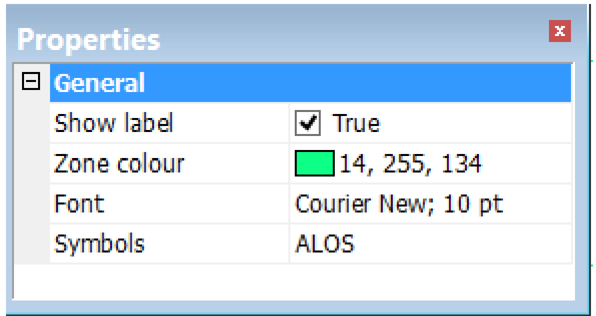
A standard Windows Open dialog box is displayed.

2. Navigate to the desired tide zone file (*.zdf) and click **Open**.

The outlines of the tide zones, and their identification labels are drawn in the Display window. The tide file is listed in the Layers window. (See TIDE ZONE FILES .)

To change the colour of the tide zone outlines and labels:

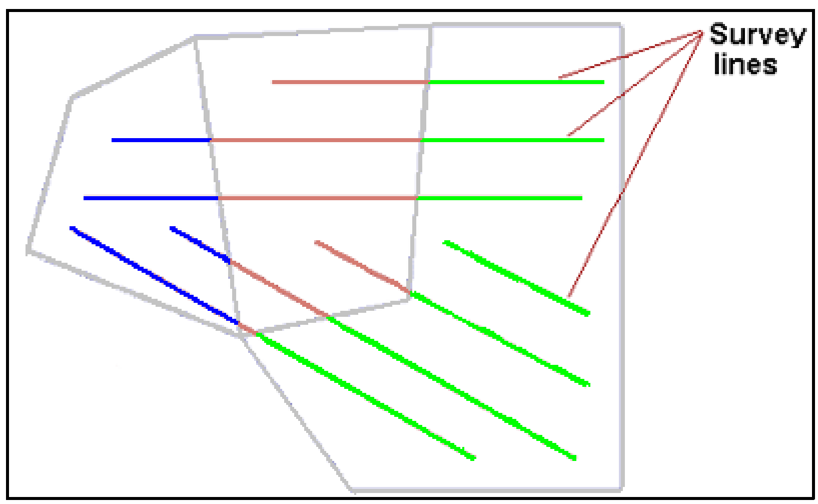
3. Select the ZDF file in the Layers window.
4. Open the Properties window.



5. Set *Show label* to “False” to hide label, leaving the symbol displayed.
6. Click in the *Zone colour* field to pick a colour for the outline and label .
7. Change the Font style and size by selecting from the drop-down list.

The Properties for track lines enable you to colour the lines by the tide applied to them. To colour by tide:

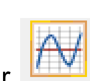
8. Select the track lines layer in the Layers Window.
9. In the Properties window, set the *Colour By* field to Tide.



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.

Tide Editor is a separate application that launched from the HIPS and SIPS interface. To open Tide Editor:

1. Click the Tide Editor button in the Tools toolbar  ,

   or select the Tide command from the Edit menu.

For information on how to use Tide Editor.

Compute GPS Tide

The Compute GPS (Global Positioning System) Tide function provides an alternative to normal tidal observation for reducing soundings to the sounding datum. A single sounding datum height, or a datum model file with a grid of datum heights, can be applied with the GPS height during the computation.

Any open surface layer or TIN layer can be selected as the input "model" for computing GPS tide. For details on sounding datum models and a sample *.info file, see GPS TIDE FORMAT and INFO FILE of the Reference Guide.

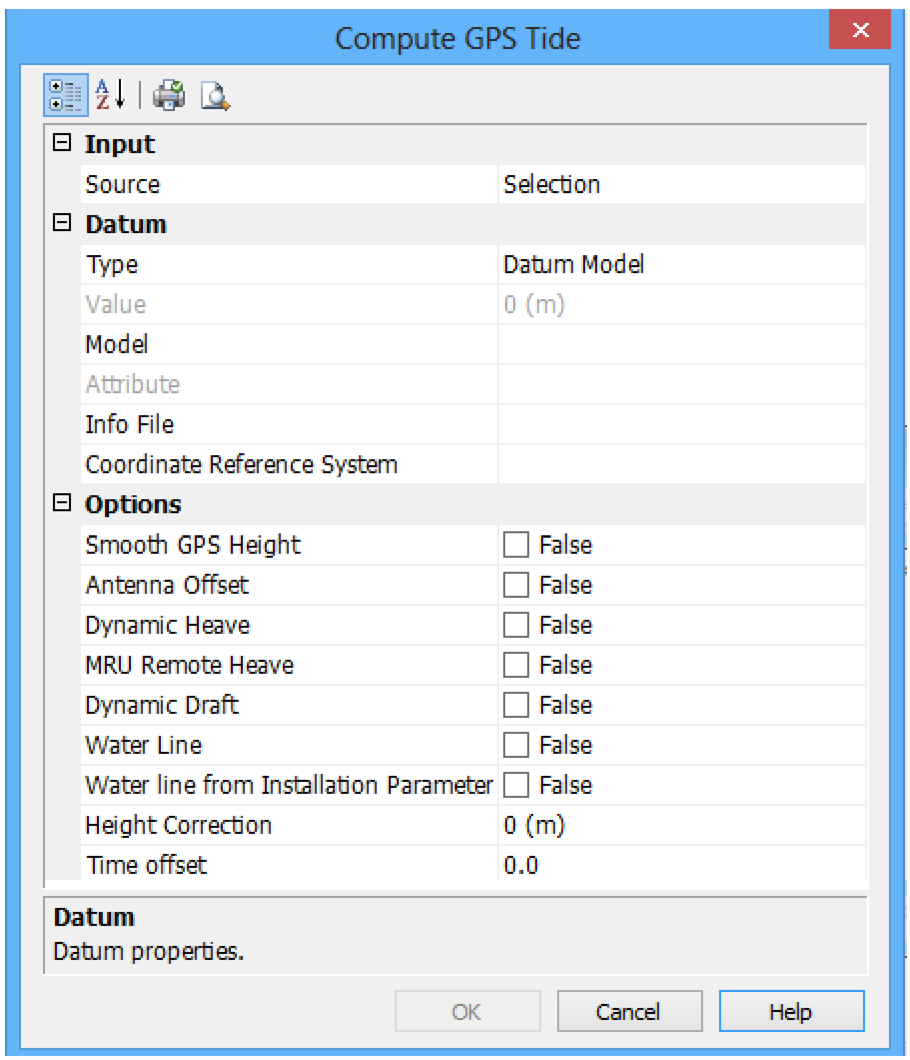
To compute GPS tide:

1. Select one or more track lines.

Menu	Process > Compute GPS Tide

2. Select the Compute GPS Tide command.

The Compute GPS Tide dialog box is displayed and the *Source* field contains "Selection".



Input

To use data from the entire project and not just to the selected lines:

3. Select "All Track Lines" from the *Source* drop-down list.

Datum

When a surface is used as the datum model, an elevation attribute must be set. This can be selected from a drop-down list in the *Attribute* field.

When a parsable ASCII file is used as the datum model, but is not recognized as a known format, you must also open a *.info file that will control the parsing of the ASCII data. As well, a coordinate system must be selected. The *Info File* and *Coordinate Reference System* fields are enabled so that you can enter this information.

4. In the *Type* field, select either Single Value or Datum Model as the type of datum to be applied.
5. If Single Value is selected as *Type*, enter a single datum height value in the *Value* field to apply a single distance from the ellipsoid to the antenna.
6. If Datum Model is selected as *Type*, use the drop-down list in the *Model* field to select a currently open surface, or to select <Browse> and open a model file. The field will display the surface name, or the file name of the model file.

    If you have selected a surface as the datum, select an appropriate attribute from the list in the *Attribute* field.

    If you have selected a datum model file of an unknown format, click **Browse** button in the *Info File* field to select the appropriate .info file. This field is disabled if the selected file is a known format with an available info file (which will be applied by default).

    In the *Coordinate Reference System* field, click the Browse button to open the Select Projection dialog box and set the relevant projection.

Options

7. Select the check boxes or type in values to apply the following options during the process:

Smooth GPS Height	Set to "True" to apply smoothing to GPS height.
Antenna Offset	Set to "True" to apply an offset of the GPS antenna from the reference point.
Dynamic Heave	Set to "True" to apply dynamic heave (either regular vessel heave, or delayed heave, if it exists).
MRU Remote Heave	Set to "True" to apply remote heave resulting from vessel roll/pitch on an offset-mounted MRU.
Dynamic Draft	Set to "True" to apply dynamic draft (as a result of interpolating the draft table in the HIPS Vessel File (HVF), or as stored in HIPS as time series data).
Water Line	Set to "True" to apply the waterline offset in the HVF.
Water Line from Installation Parameters	Set to "True" to apply the waterline value from the InstallationParameters.xml file generated by the converter.
Height Correction	Type an amount as a static offset to the GPS antenna.

Time Offset	Type a value in seconds to apply as a time offset.

8. Click **OK** to apply.

The GPS Tide is calculated for the selected line(s). The settings in the dialog box are saved in a GPS Tide file in the line directory.

You can open and view the GPS tide data in the Attitude Editor (ATTITUDE EDITOR).

Sounding datum model files in gridded binary (.bin) or ASCII (.xyz) format can be opened in HIPS and SIPS as background data. See OPEN DATA in the Reference Guide.

Compute Separation Model

Generate a separation model based on traditional tide inputs as well as other vessel dynamics. The model is saved in a CSAR format and opened as a background layer.

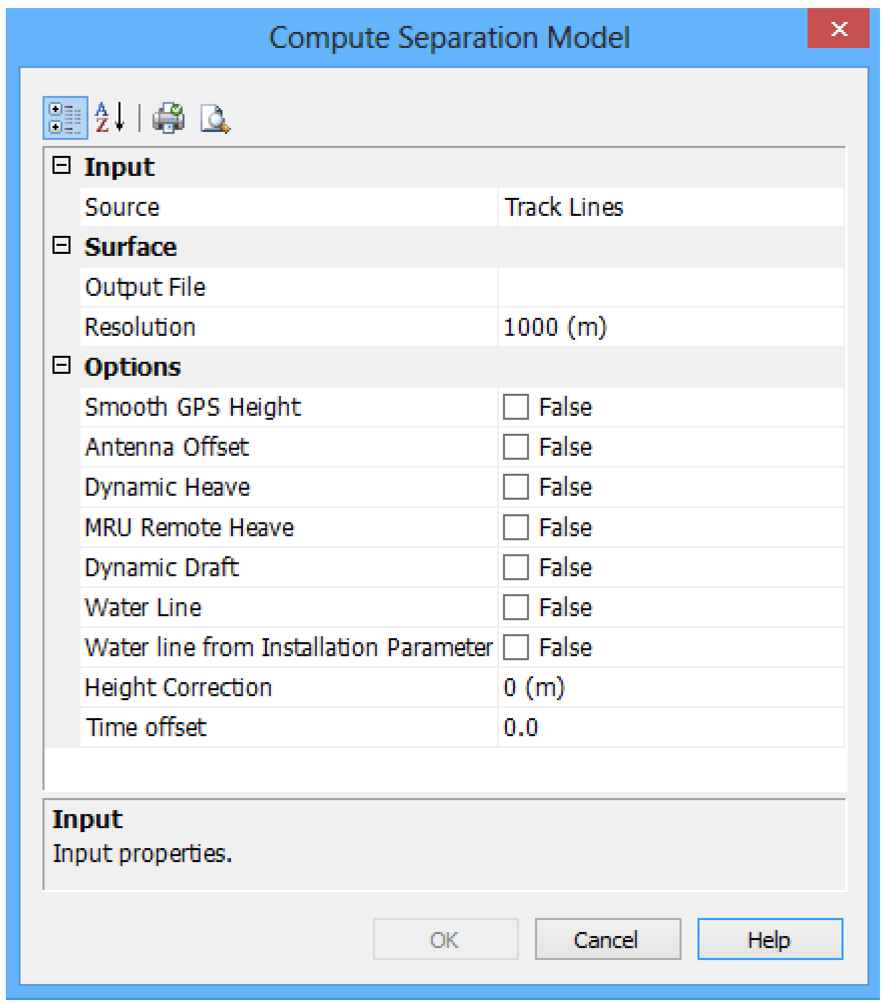
To compute a separation model:

1. Open a project containing Tide and GPS Height data.
2. Activate the Ship Track Lines layer in the Layers window.
3. [Optional] Select a line or lines of data.

Menu | Process > Compute Separation Model

4. Select the Compute Separation Model command.

The Compute Separation Model dialog box is displayed.



The Input *Source* field will display either:

- "Track Lines", if no data is selected, or
- "Selection", if you have selected a line or lines.

To compute the model based on the entire project and not just on the selected lines:

5. Select “Track Lines”: from the drop-down list in the *Source* field.
6. Click in the *Output File* field and click **Browse** to set or select an output file.
7. Set a resolution value in meters in the *Resolution* field.
8. Set appropriate *Options* to “True” to apply them in the model.
9. Click **OK**.
10. A separation model will be created using all of the data in the project, or only the selected data if you set “Selection” in the *Source* field.

The layers making up the model will be displayed in the Layers window, and a CSAR file will be created in the output folder.

Options

Option	Function
Smooth GPS Height	Set to “True” to apply smoothing to GPS height.
Antenna Offset	Set to “True” to apply an offset of the GPS antenna from the reference point.
Dynamic Heave	Set to “True” to apply dynamic heave (either regular vessel heave, or delayed heave, if it exists).
MRU Remote Heave	Set to “True” to apply remote heave resulting from vessel roll/pitch on an offset-mounted MRU.
Dynamic Draft	Set to “True” to apply dynamic draft (as a result of interpolating the draft table in the HIPS Vessel File (HVF), or as stored in HIPS as time series data).
Water Line	Set to “True” to apply the waterline offset in the HVF.
Water line from Installation Parameters	Set to “True” to apply the waterline value from the `InstallationParameters.xml` file generated by the converter
Height Correction	Type an amount as a static offset to the GPS antenna.
Time offset	Type a value in seconds to apply as a time offset.

8

Compute TPU

Total Propagated Uncertainty (TPU) is used to assign a horizontal error estimate (HzTPU) and a depth error estimate (DpTPU) to each sounding.

In this chapter...

Computing Total Propagated Uncertainty

Total Propagated Uncertainty (TPU) is derived from a combination of all individual error sources. The following errors (among others) contribute to TPU:

- nav/gyro/heave/pitch/roll/tide errors
- latency error estimate
- sensor offset error estimates
- individual sonar model characteristics

TPU is essential for the following functions:

- creating S-44 or S-57 compliant datasets (see "TPU Filtering" on page 190)
- calculating surfaces weighted by uncertainty (see "Uncertainty Weight" on page 205)

Static values are stored in the vessel file for the survey. Sonar error values are stored in the DeviceModels.xml file in `...\HIPS\System`. Other static errors can be directly entered in the Compute TPU dialog box. For some formats, real-time errors are stored for each line.

The following sonar error-models in the DeviceModels.xml have been tested by the University of New Hampshire (UNH).

- Atlas Hydrosweep DS
- Elac Nautik 1180
- Seabeam 2112
- Teledyne Reson SeaBat 8101
- Teledyne Reson SeaBat 9001
- Teledyne Reson SeaBat 9003
- Teledyne Reson SeaBat 8125
- Kongsberg EM300
- Kongsberg EM1000
- Kongsberg EM1002
- Kongsberg EM3000
- Kongsberg EM3000D

Other sonars listed in the file may make assumptions that could produce less rigorous results than these error models supplied by UNH.

Compute TPU must be performed after Sound Velocity correction. Whether it is applied before or after Merge will depend on the workflow you are following.

Sources of Uncertainty Data

The Compute TPU command computes Horizontal TPU (HzTPU) and Depth TPU (DpTPU) values for each sounding in the observed depths.

- DpTPU: The uncertainty associated with the depth value of a sounding. It is a random error and is scaled to 95% (1.96 sigma).
- HzTPU: The uncertainty associated with the position of a sounding. It is a random error and is scaled to 95% (1.96 sigma).

The HzTPU and DpTPU values are stored line-by-line with indexing to profiles and beams. You can view these specific error values for individual soundings in the Selection window, by querying the soundings in Swath, Single Beam, or Subset Editor.

Static errors, such as motion or position uncertainty, that are used to compute TPU are read directly from the project vessel file, sonar uncertainty is computed from the sonar settings maintained in the `DeviceModels.xml`, and tide and sound velocity from values set in the Compute TPU dialog box.

When present, real-time sonar uncertainty computed by the acquisition system, is recognized and automatically converted into HIPS. All other dynamic errors, such as from a motion sensor, can be loaded using the Import Auxiliary Data function.

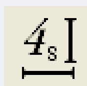
If “Realtime” is specified and real-time sources are not found, Compute TPU will default back to the vessel file, and if static sources are not found there, TPU will use a value of “0” for that component.

Compute TPU

Use the Compute TPU dialog box to either read all uncertainty values from the vessel file, or to read all from real-time data.

Alternatively, the Custom option enables you to set the uncertainty source for each component that contributes to the TPU computation. For example, *Position* error could be set to read from the vessel, while *Roll* and *Pitch* could be set to use real-time data, and a static value is set for *Tide* error.

TPU can be applied to a selection of lines or to the entire project, using settings in the *Input Source* field. The Ship Track Lines layer must be active to compute TPU.

Menu	Process > Compute TPU
Tool	

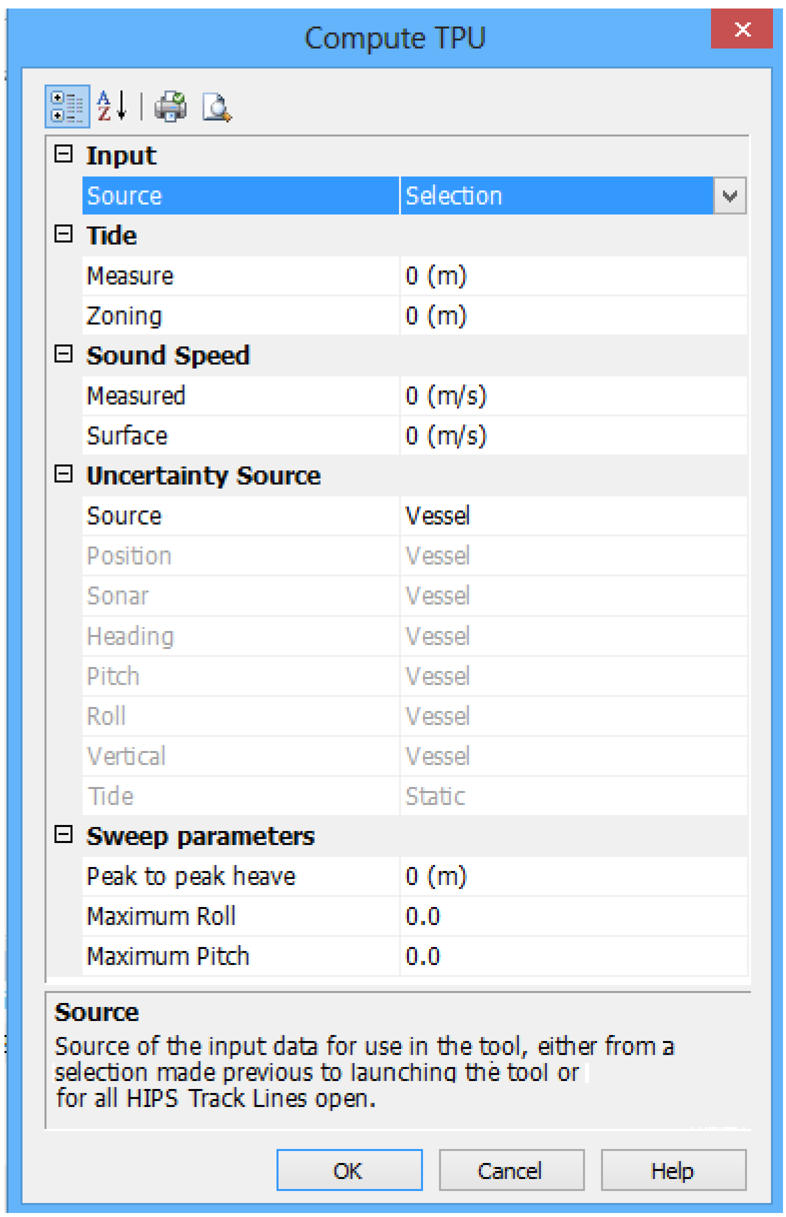
To compute TPU:

1. Highlight the Ship Track Lines layer.
2. [Optional]: Select a track line or lines in the Display window.
3. Select the Compute TPU command.

The Compute TPU dialog box is displayed and the *Source* field contains Selection.

To apply TPU to the entire project and not just to the selected lines:

1. Select “All Track Lines” from the drop-down list.
2. Set the error sources and values to be applied.
3. Click **OK** to apply TPU to processed depths.



Error sources

Error sources for computing TPU		
Input		
	Source	If there is no data selected, the Source field displays “All Track Lines”. TPU will be applied to the entire project. If you have data selected, the Source field will display “Selection”. Selecting “Track Lines” will apply TPU to the lines open in the project.
Tide: Enter the vertical uncertainty values due to tide. These will be applied to all lines being processed.		
	Measured	The uncertainty value for the tide station. It is equivalent to the standard deviation of the Tide Gauge measurements.
	Zoning	The vertical uncertainty value in range calculation for a tide zone file. (HIPS does not model the TPU for the tide zone. Instead a single user-defined single value may be used here.)
Sound Speed: The error offsets for Sound Speed Values.		
	Measured	The offset value used in computing range error to compensate for inaccuracies in SVP measurements.
	Surface	The offset value is used to account for errors in surface sound speed measurements that alter the beam angle. This parameter is only applicable to systems that require accurate surface sound speed measurements.

Uncertainty Source	Designate the uncertainty source for all components as either “Vessel” or “RealTime”, or set a “Custom” source for each individual component.	

	Source	Set the sources for uncertainty data:
		• If *Source* is set to “Vessel”**:** all individual source fields are set to “Vessel” (read-only).
		   • Selecting “Vessel” will ignore RMS data and use vessel settings.
		• If *Source* is set to “Realtime”: all individual sources are set to “Realtime” (read-only).
		   • Selecting “Realtime” will use previously loaded Down/Heave RMS data
		
		If “Custom” is selected, individual sources may be set as follows.
		• “Vessel” or “Realtime” can be set as the source of the uncertainty values for:
		   • *Position*
		   • *Sonar*
		   • *Pitch*
		   • *Heading*
		   • *Roll*
		
		The source of *Vertical* uncertainty values can be set to:
		   • Vessel
		   • Realtime Heave (Vertical RMS -GPS Height RMS/Heave RMS)
		   • Delayed Heave (Delayed Heave RMS)
		
		The source of *Tide* uncertainty values source can be set to
		   • Static (use values as set in the *Measure* or *Zoning* fields)
		   • Realtime (from tide error files)

Sweep parameters	Enter transducer motion offsets for sweep surveys	
	Peak to peak heave	Enter the uncertainty value for the estimated observed heave of the transducer boom
	Maximum Roll	Enter the Roll offset of the transducer boom
	Maximum Pitch	Enter the Pitch offset of the transducer boom.

If you choose “Realtime” as the uncertainty source configuration, and Delayed Heave RMS has been loaded (using the Import Auxiliary Data command), Compute TPU will use the Delayed Heave RMS data loaded with Import Auxiliary Data and override the Vertical RMS values in the source files.

If you choose “Vessel” or “Custom” this doesn't apply.

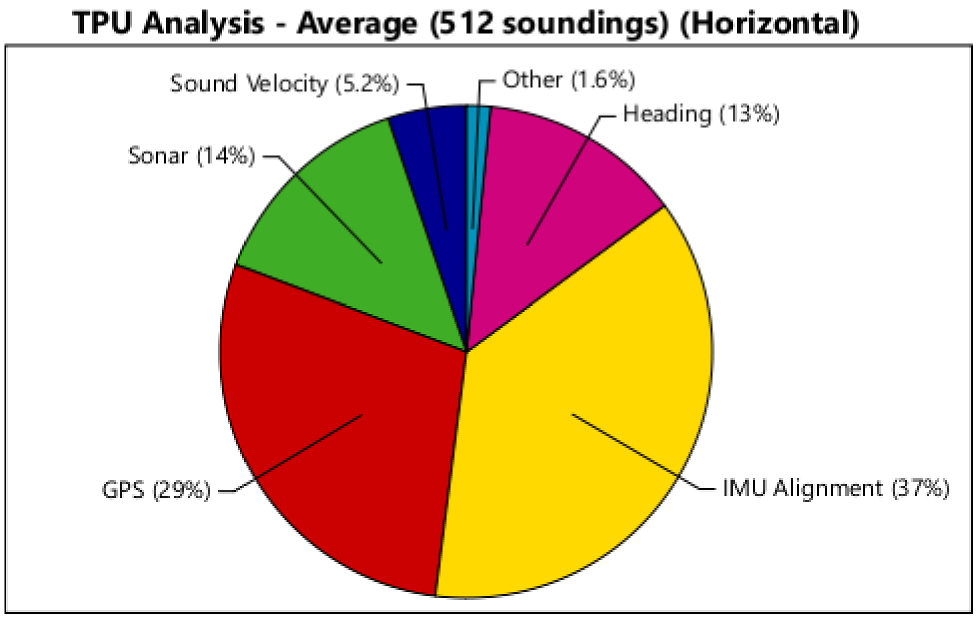
When the computation is finished, the Output window will list the error sources that were applied. This output can be saved to a text file using the Save All command on the right-click menu.

A graphical view of the results can be seen in the "TPU ANALYSIS WINDOW" ON PAGE 188

TPU Analysis Window

The TPU Analysis window enables you to visualize the proportional effect of the various error sources that make up the TPU. These component sources are displayed in pie chart, bar chart or scatter plot graphs.

The TPU Analysis window example below displays a pie chart with the proportionate contributions of six error sources to horizontal TPU, as averaged over 512 selected soundings.



The options for the TPU Analysis window enable you to view:

- component sources as contributions to horizontal or vertical TPU
- sources as a percentage of the total contributions, or as component values

As well, the various error sources can be toggled on or off to see their impact.

This information can be exported to a comma separated value text file, or to a TIFF, BMP, PNG or JPG image.

To view TPU analysis for data:

Menu	Window > TPU Analysis

1. Open the data to which TPU has been applied.
2. Open the TPU Analysis window.
3. Open Swath Editor, or load a subset of data in Subset Editor.
4. In an editor window, make a selection of data to which TPU has been applied, for example, in the Plan view in Swath Editor.

The TPU window will display a chart of the type last viewed. The default type is a pie chart.

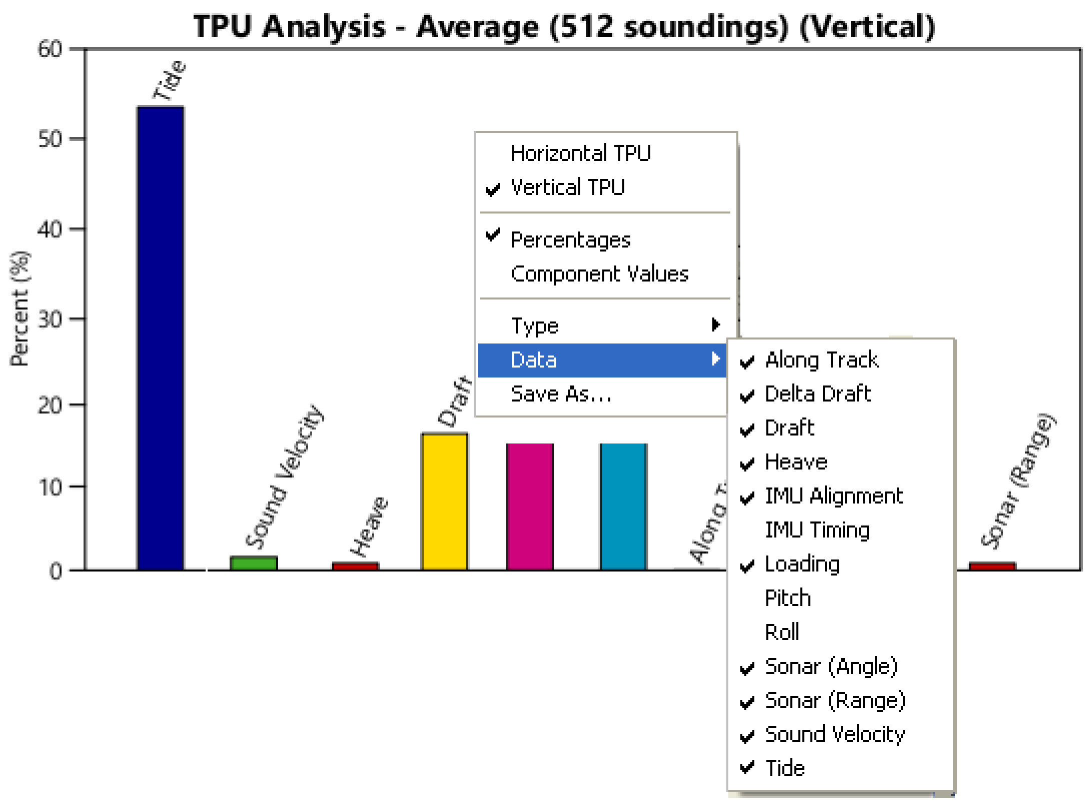
To change the type of graph:

1. Right-click in the TPU Analysis window and select the type.

To change the data that is displayed in the graph:

1. Right-click in the TPU Analysis window and select or de-select the data to display.

The example below shows selected sources contributing to vertical TPU.



TPU Filtering

TPU filtering compares the HzTPU and DpTPU values for each sounding to the depth and horizontal error limits for a specific IHO S-44 survey order, or for an S-57 zone of confidence (CATZOC) attribute.

These limits are standards set by the International Hydrographic Organization (IHO). All soundings with HzTPU and DpTPU values outside those limits are rejected or accepted, depending on which option you select.

Filtering parameters are set and saved using the Set Filters dialog box, and then applied using the Apply Filters command.

To set TPU filters:

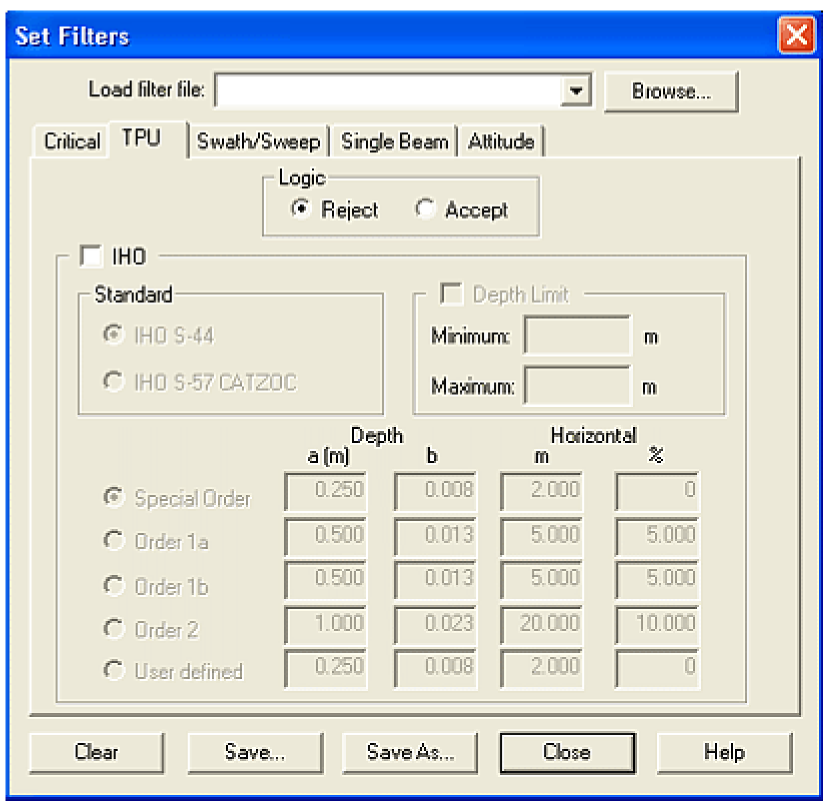
1. Open a project.
2. Select the Set Filters command.

Menu	Tools > HIPS Data Filters > Settings
Tool	

The Set Filters dialog box is displayed.

3. Click the TPU tab.

If you have previously entered values in the TPU filter fields, they are displayed. Parameters you enter into these fields are retained until you change them or until load a saved filter file.



4. [Optional] Select an existing HIPS Filter File from the list or click **Bro**wse to select a file.
5. Under *Logic*, select either *Reject* or *Accept* as the filter criteria.
6. Enable the *IHO* check box.

This will enable the TPU filter so the values will be applied to data when the Apply Filters command is used.

7. Select the standard to apply, either *IHO S-44* or *IHO S-57 CATZOC.*

    IHO S-44 displays options for filtering according to an S-44 survey order.

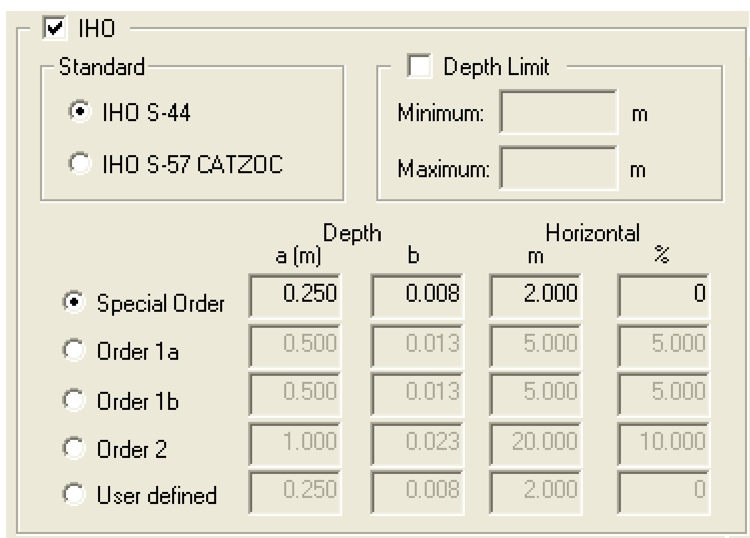
    IHO S-57 CATZOC displays options for filtering an S-57 zone of confidence (CATZOC) attribute.

8. [Optional] Enable the *Depth Limit* check box, and define a range of depths to which filtering will be applied by values typed into the *Minimum and Maximum* fields.

If no minimum/maximum depth range is set, filtering is applied to all soundings.

9. Select select a standard to apply, either the S-44 survey order or CAZOC zone of confidence.

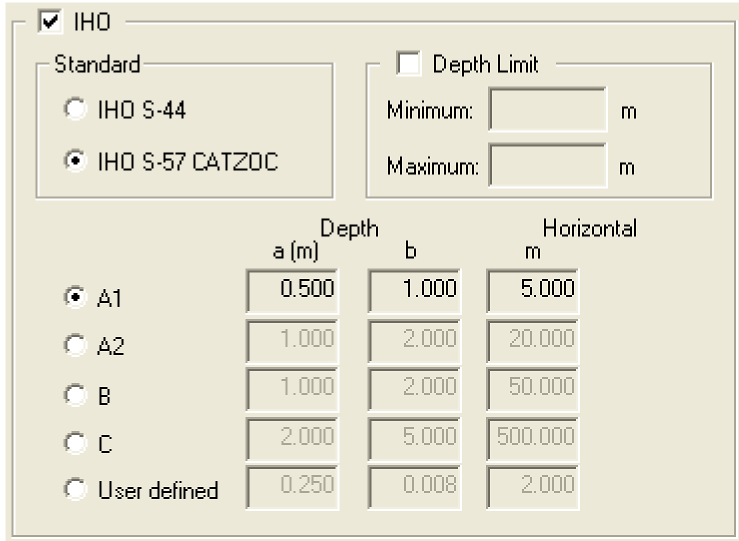
The dialog box displays the values associated with the standard you selected. The left column lists the IHO specification type and the other columns list the horizontal and vertical uncertainty values for the selected standard.



S-44

Both standards contain the following fields:

- *a*: constant depth error (i.e. the sum of all constant errors)
- *b*: factor of the depth dependent error
- *m*: position limit for horizontal errors
- *%*: percentage of depth used to calculate horizontal error (S-44 only).



S-57

Depth error limit

To calculate the error limits for depth accuracy, the constant depth error (a) and the factor of the depth dependent error (b) are combined with depth (d) and the depth dependent error (b * d) in the following formula:

Error limit for depth accuracy = $\pm\sqrt{a^2 + (b \times d)^2}$

When the filter is run, soundings with a vertical uncertainty outside the error limit for the survey order or zone of confidence are rejected (or accepted).

Horizontal error limit

To calculate horizontal accuracy, the limit for horizontal error (m) and (if applicable) the depth percentage (%) for the selected survey order or zone of confidence are compared against a sounding's HzTPU. Soundings with a horizontal uncertainty outside the limits are rejected (or accepted).

Values entered in the TPU filter will be retained when the dialog box is closed so that you can use them again and again.

You can also save the settings in a HIPS Filter File (.hff), for example, one file for the S-44 filtering and one for CATZOC.

The table displays the pre-set vertical and horizontal values for each Survey Order or Zone of confidence. These values are fixed values and cannot be changed.

To use custom values:

10. Select the User defined option.

When the *User defined* option is selected are the relevant fields are activated so you can type *Depth* and *Horizontal* values

11. Type *Depth* and *Horizontal* values.

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

13. Click **Close**.

Your saved filter can now be applied to your data using the Apply Filters command.

Apply Filter

TPU filtering can be applied to a single line or the multiple lines. To apply the filtering values from the Set Filters dialog box:

To apply filtering to a range:

1. Select line or lines.
2. Run one of the Filtering commands.

Menu	Tools > Apply Filters > Selected Lines Tools > Apply Filters >/All Lines
Tool	

See also

- “Attitude Filter” on page 329
- “Automatic Filtering” on page 373
- “Single Beam Filtering” on page 810
- “Protect Critical Soundings” on page 347

9

Merge

The Merge process converts along track/across track depths into latitude, longitude, and depth by combining the ship navigation with the horizontal and vertical offsets from the HIPS vessel file. This geographically references the sounding position and depth.

In this chapter...

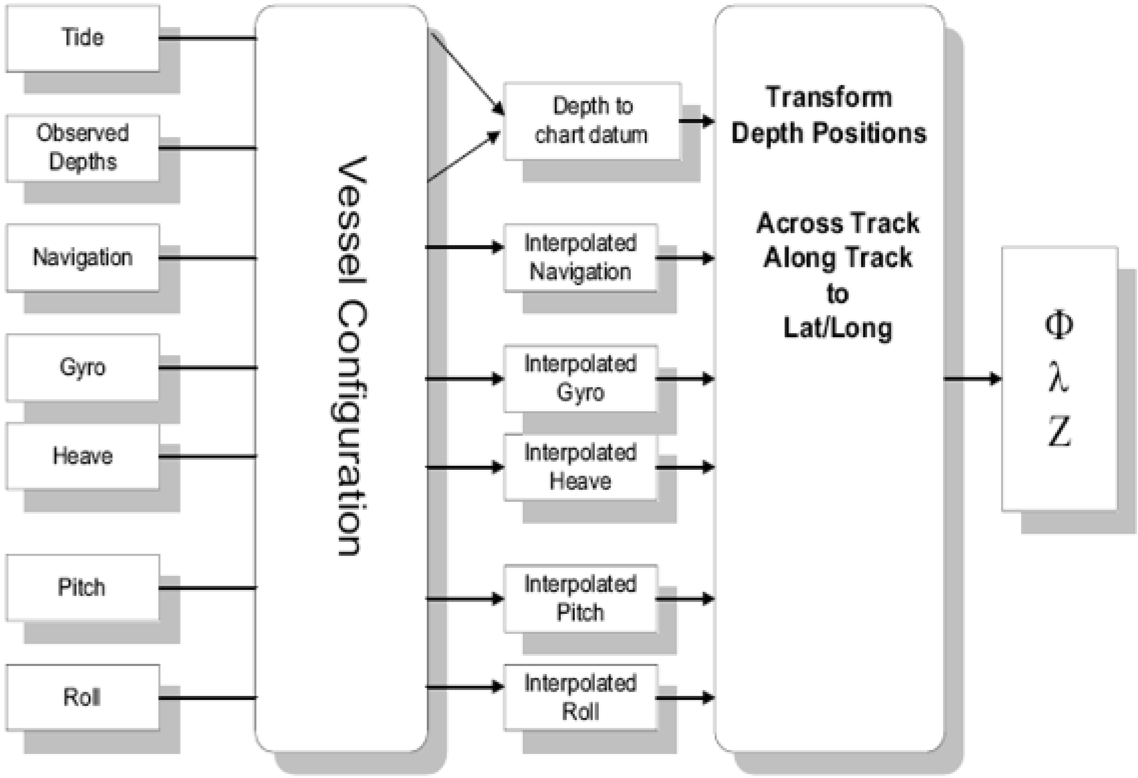
Merge Process

Once Sound Velocity Correction has been applied, (see "Sound Velocity Correction" on page 160) the final sounding depth and geographical position needs to be calculated. This is done in the Merge process.

Track line data must be Merged before BASE surfaces can be created and before surface cleaning can be performed.

Merge takes into consideration pertinent vessel configuration offsets and the following values, many of which are set in the HIPS Vessel file:

- Navigation
- Gyro - based on the settings in the HIPS Vessel file.
- Dynamic draft - based on the settings in the HIPS Vessel file.
- Smoothed sensor data
- Waterline (if not applied in the SVC process)
- Motion data: Heave, Pitch, and Roll (if not applied in the SVC process)
- Tide or GPS tide
- Delta draft
- Refraction coefficients
- Observed depths



The following operations take place when merging:

- Recorded sensor information is compensated for constant time errors as noted in the vessel file.
- For each time-tagged depth record, the position for the centre of the swath profile is calculated based on the interpolated position fix at that time.
- For each depth record, the position of the sounding is calculated, based on the profile centre position and any gyro/heave/pitch/roll corrections that are appropriate (this is echo sounder system dependent).
- Tide is interpolated and applied to each depth record.
- Draft is applied.

A processed depths file is created for each line. This file contains the final computed geographic position for each depth record.

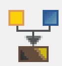
If any of these offsets or parameters is changed the data must be Merged again.

Apply Merge

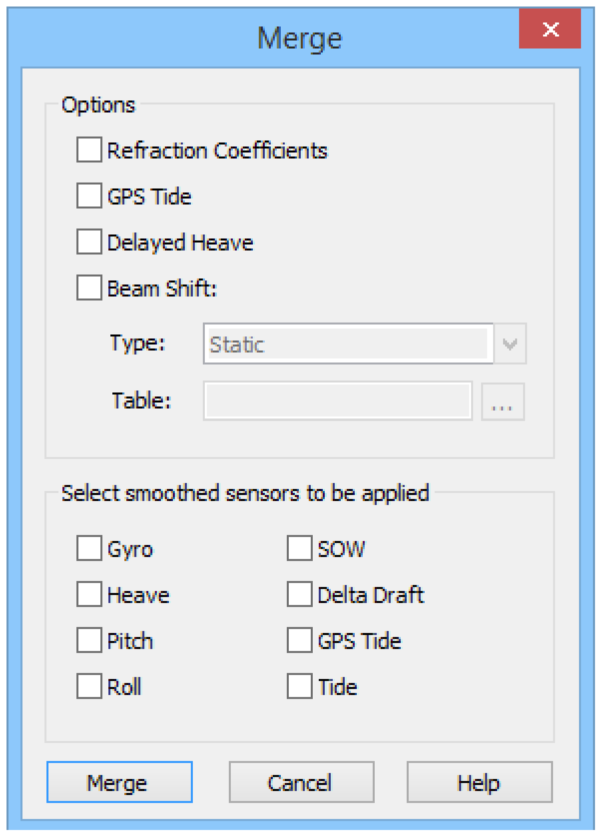
Lines must have tide loaded before they can be merged.

To apply Merge:

1. Select a line or group of lines.
2. Select the Merge command from the Process menu or toolbar.

Menu	Process > Merge
Tool	

The Merge dialog box is displayed.



3. Select the *Apply refraction coefficients* check box to apply any data created in Refraction Editor. (See “Correction for Refraction Artifacts” on page 383)
4. Select the *Apply GPS tide* check box to use GPS tide data instead of tidal observation data.
5. Select the *Apply Delayed Heave* check box to apply delayed heave values instead of regular heave.

If you choose this option, and no delayed heave data is available, there will be a warning message in the Output window, and regular heave data will be used instead.

Beam Shift

The beam shift option applies values in a table to shift depths.

The beam shift table consists of two columns, one contains the beam number and the other contains the value of the shift. Only the beams listed in the table will be shifted, by the amount specified.

For example:

```
1,2
2,2
3,2
22,3
75,1
```

If the *Static* option is selected, the value is applied as a straight shift (positive down, depths get deeper). If *Relative* is selected, the value is applied as a percentage of the depth being shifted.

6. Select the *Beam Shift* check box to apply a per-beam vertical correction to the data during Merge.
7. From the *Type* list, select Static or Relative as the type of look-up table to be applied.
8. To select the table to be applied, click **Browse** and set the location of the table file.

If you have a `SmoothedCoefficients` file for a sensor, you can apply this file during merge to smooth selected sensor data.

9. Click the check box beside a sensor to select which data that is to be smoothed during Merge.
10. Click **Merge**.

You can view the progress of the Merge process in the Output window.

Functions requiring geo-referenced soundings can now be applied.

> If a line is merged and then changes are made to the sensor data used by merge, the track line is marked in the Project window with the Outdated icon. As well, the Outdated field for that line is set to “Yes” in the Selection window. An Outdated track line must have Merge applied again.

Load Delta Draft

The Load Delta Draft function lets you import your model for dynamic changes in draft and have these changes applied to the soundings during the Merge process. Delta Draft represents a change in draft that is positive as draft increases and negative as draft decreases. The data, which is loaded from a ASCII text file, overrides the Dynamic Draft information in the vessel file. The Delta Draft value is interpolated between time stamps in Merge and added to the Observed Depth to obtain the Final Depth.

The input format for the Delta Draft text file is similar to the COWLIS tide format. Each record is in the following format:

```
YYYY/MM/DD HH:MM:SS.SSS 1.234
```

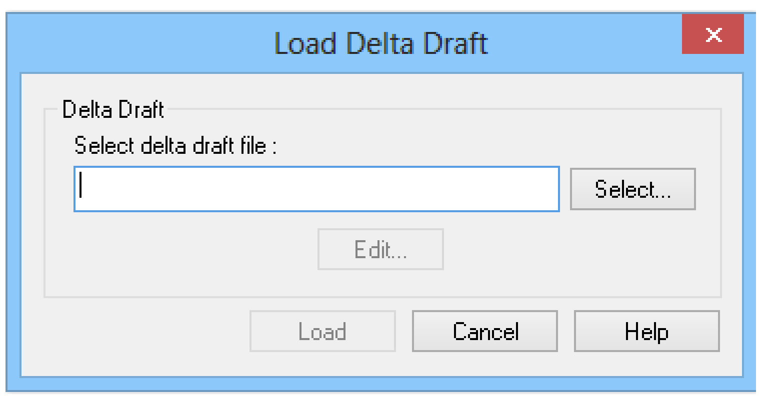
The Delta Draft function can also be used to load recorded depth versus time data for an underwater platform on which the multibeam sonar is mounted, such as a ROV (Remote Operated Vehicle) or AUV (Autonomous Underwater Vehicle), or a towed vehicle.

1. Select the track line(s).

Menu	Process > Load > Delta Draft

2. Select the Load Delta Draft command.

The Load Delta Draft dialog box is displayed.



3. Type the path of the file or click **Select** to locate the file.
4. To make changes to the file, click **Edit**.

The file is opened in the default text editor (such as Notepad). Make any changes and save the file.

5. Click **Load**.

The delta draft is loaded into the line folder.

10

Create Surfaces

Surface can be generated for a detailed examination of the seafloor or to view data for safety of navigation.

Four kinds of surfaces can be created:

- Swath Angle
- Shoalest Depth True Position
- Uncertainty
- CUBE (Combined Uncertainty and Bathymetry Estimator)

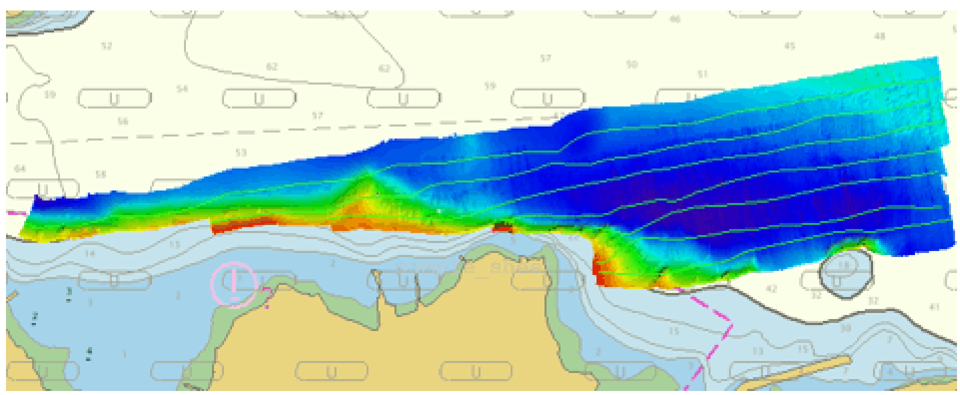
In this chapter...

Overview

Surfaces are georeferenced images that can contain a visual representation of horizontal and vertical uncertainty. Surfaces are used

- as a background visual aid when cleaning data interactively or automatically.
- as a quality control feature for data, by identifying bad data and determining if an area needs to be re-surveyed.
- as a product surface that can be used to build contours, sounding selections, profiles, and other data layers. This data can later be used to create an ENC or raster chart.
- to create a georeferenced TIFF images that can be exported to other software such as real-time navigation systems.
- to export data as an XYZ text file so it can be imported as soundings in a CARIS map or used to build digital terrain models in other software

An example of a surface is shown below.



A surface is saved as a CSAR file, using the CARIS Spatial ARchive framework. This file contains the data and metadata for the surface.

Types of surfaces

Four different surface types can be created:

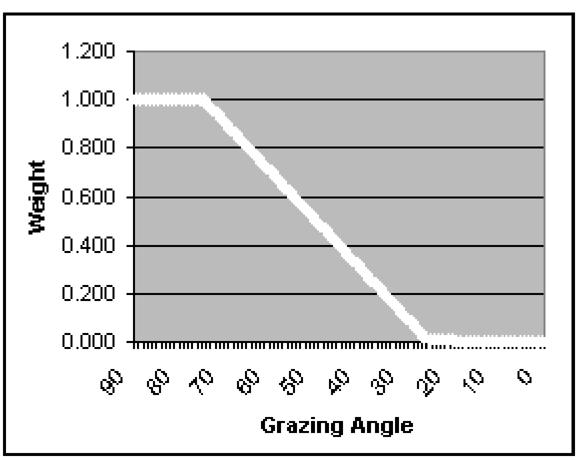
- Swath Angle "SWATH ANGLE WEIGHT" ON PAGE 203 uses a weighting scheme based on a beam's intersection angle with the seafloor.
- Shoalest Depth True Position "SHOALEST DEPTH TRUE POSITION" ON PAGE 204 stores the shoalest depth within a given node in the depth layer. The true position of this depth is also stored.
- Uncertainty "UNCERTAINTY WEIGHT" ON PAGE 205 uses a weighting scheme based on depth uncertainty as specified by a selected S-44 survey order.
- CUBE (Combined Uncertainty and Bathymetry Estimator) "CUBE" ON PAGE 205 surface uses multiple hypotheses to represent potential depth variances along the seafloor (see OVERVIEW to CUBE PROCESSING).

Swath Angle Weight

The weighta sounding contributes to the surface varies with the grazing angle of the sounding with the seabed.

This weighting value is important in areas with adjacent or overlapping track lines. The swath angle weight ensures that higher weight is given to beams from the inner part of a swath than to outer beams from adjacent track lines.

In this graph, beams with a grazing angle between 90 and 75 degrees are given a weight of 1.0. The weight decreases linearly to 0.01 as the angle with the sea floor decreases to 15 degrees.



This is the default swath angle weight scheme as defined by the GrazingAngleWeights.txt file in `C:\ Program`

Files\CARIS\HIPS\<*version*>\System. To use another weighting scheme, customize this file.

Range weighting

Range weighting is used in creating Swath Angle and Uncertainty surfaces to determine how a sounding value is applied to a node.

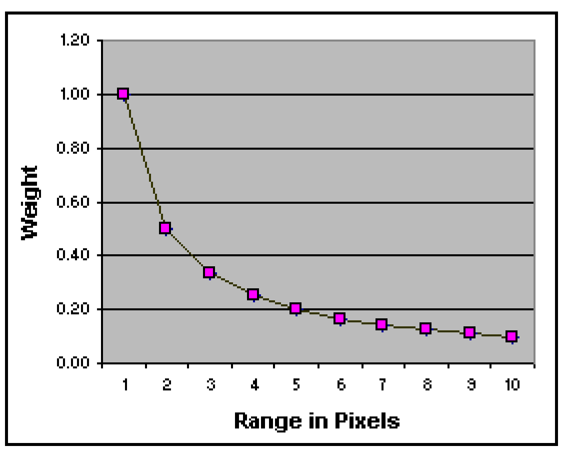
The range weight is inversely proportional to distance from the node: soundings closer to a node are given a greater weight than soundings further away. The calculated node positions are determined using the lower left corner of the area defined in the create Surface Wizard.

The number of nodes each sounding is applied to is determined by the size of the beam footprint. The beam footprint is calculated using depth, sonar beam width, and the grazing angle.

The sonar that is used, with the appropriate beam width, is defined in the Surface wizard.

A list of multibeam systems and corresponding beam widths is listed in C:\ Program Files\CARIS\HIPS\<version>\DeviceModels.xml.

The following graph demonstrates range weighting using distance from a node in units of pixels (multiples of the surface resolution).



Shoalest Depth True Position

This surface stores the shoalest depth within a given node in the depth layer. It can create a true position surface layer for which horizontal and vertical uncertainty can be specified as optional output layers.

If uncertainty data does not exist for the lines selected, the surface creation will continue without it. If uncertainty data becomes available, the surface can be recomputed and updated accordingly.

Uncertainty Weight

The weight that a sounding contributes to a node is inversely proportional to the predicted depth uncertainty of the sounding.

The sounding's depth uncertainty is scaled as a function of the sounding's distance from the node. The propagation of the depth uncertainty to the node takes into account the sounding's distance from the node AND the sounding's horizontal uncertainty. Thus, both depth uncertainty and horizontal uncertainty play a role in determining the weight a sounding contributes to a node.

The area of influence is a radius that determines the number of nodes to which the sounding can be applied. This radius is determined by the depth uncertainty of the selected IHO S-44 survey order, using the formula below.

$$\text{Depth Uncertainty} = \sqrt{a^2 + (b \times d)^2}$$

Once the depth uncertainty is propagated beyond the S-44 survey order requirement that has been selected for depth accuracy, it has reached its area of influence limit.

All distance computations are based on the ground coordinate system defined for the surface—they are not simplified to cell distances. All of the nodes within a sounding's area of influence are located rigorously. No simplified weight matrix scheme is used.

CUBE

The Combined Uncertainty and Bathymetry Estimator (CUBE) generates a surface that contains multiple hypotheses representing potential depth variances on the seafloor. If you examine these hypotheses in Subset Editor, you can determine if they are valid or not. These hypotheses can be replaced by alternative hypotheses.

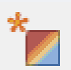
If you use a surface filtering tool, soundings that fall outside specified parameters can be rejected and another CUBE Surface generated. In this way, CUBE can be used in an iterative process to filter data to produce the best-possible representation of the surveyed area.

For production of CUBE surfaces and hypothesis editing, see "Generating CUBE Surfaces" on page 287

Create a New Surface

Surfaces are created using a wizard. You can select specific track lines to include in the surface or have the surface cover the extents of the data automatically.

> All track lines must be merged before a surface can be created.

Menu	Tools > Surfaces > New
Tool	

1. Select the New Surface command.

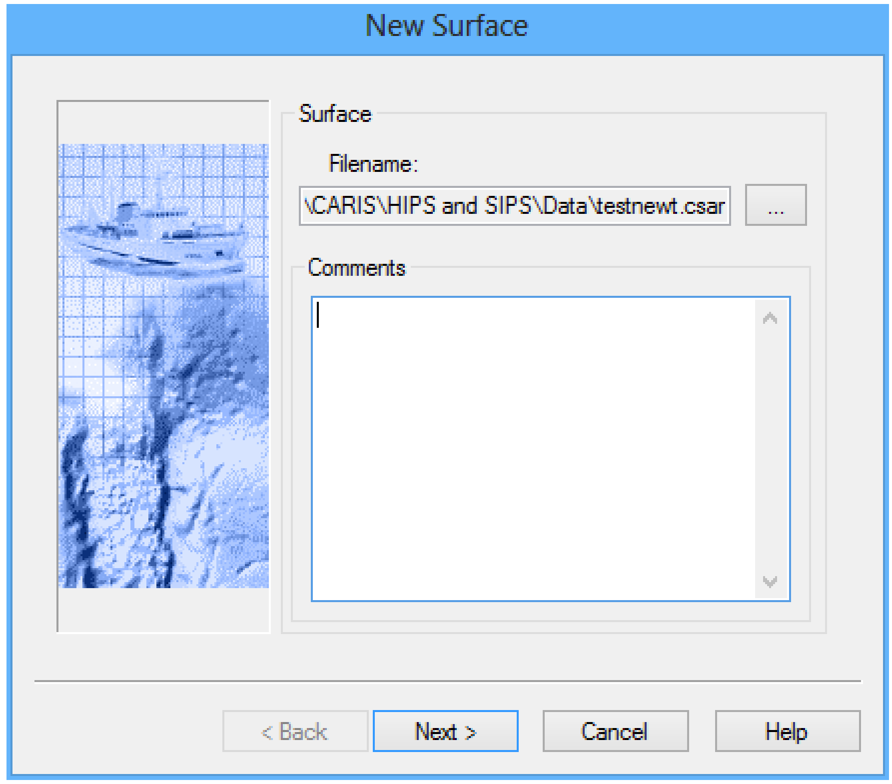
The number of steps in the wizard is determined by the type of surface you are creating.

Steps to create a surface:

- Name the surface ON PAGE 207
- Set extents ON PAGE 208
- Set Surface Resolution, Surface Type and Vertical Datum ON PAGE 211
- Set options for adding data to surface for:
  - Swath Angle Weighting ON PAGE 203
  - Shoalest Depth True Position ON PAGE 204
  - Uncertainty ON PAGE 205
  - CUBE ON PAGE 291
  - Set CUBE parameters ON PAGE 294

Name the Surface

The first New Surface dialog box is displayed.

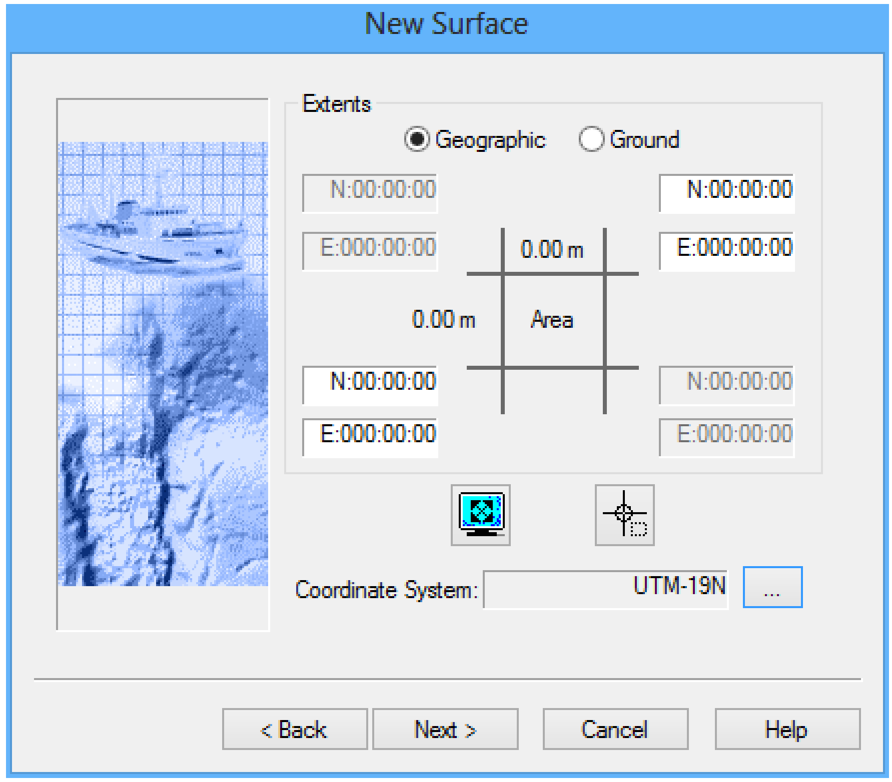


1. Click Browse and select a folder to save the surface file to.
2. Type a name for the surface file.
3. Click **Save**.
4. Type any necessary *Comments* in the text box.
5. Click **Next.**

See also "Overview" on page 202 and "Types of surfaces" on page 203

Set Extents

Set the extents and coordinate system of the new surface.



Using geographic coordinates in degrees-minutes-seconds, set the extents. By default, the extents are set to zero.

1. [Optional] Set coordinates to *Ground* option.
2. Define the extents of the surface by one of three methods:
   - use the current extent of the Display window, or
   - use a bounding box to define the extent, or
   - enter the geographic or ground coordinates.

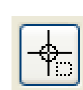
Use current display

1. Zoom in on the area you want to include so that it fills the Display window.
2. Click the Current Display Extent button. 

The extents field display the coordinates for the current display.

OR

Draw a bounding box

1. Click the Bounding Box button. 

The cursor in the Display window is shown as a cross-hair.

2. Press and hold the mouse button, and drag the cursor across the area where you want to create the surface.

A rectangular box is drawn across the area and the coordinates are displayed in the extents fields.

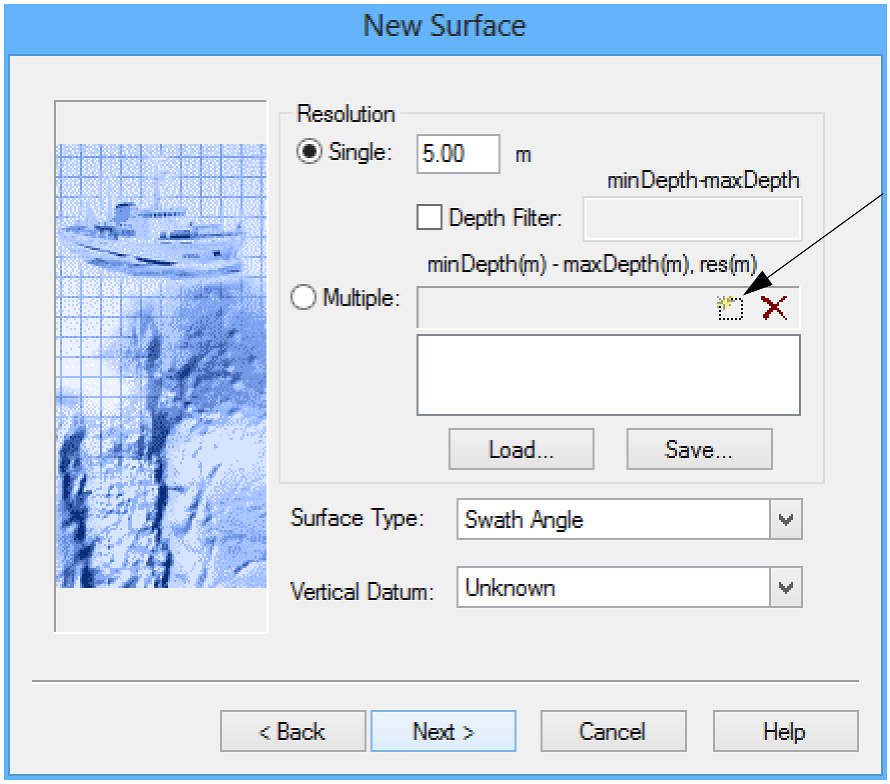
- To move the bounding box, position the cursor inside the box so it becomes a four-headed arrow. Press and hold the mouse button, and then drag the box to a new location.

- To resize the bounding box, click on the box handles at NE or SE and drag to resize the box.

OR

Enter coordinates	1. Enter the northeast and southwest extents by clicking in one of the appropriate fields and typing the coordinate data, or by using the arrow keys to toggle the values into the entry fields.
Coordinate system	The Coordinate System field displays the system used for the open project. To apply a different coordinate system to the surface: 1. Click **Browse**. 2. Select an alternate coordinate system from the Coordinate System dialog box. 3. Click **Next**.

Set Surface Resolution and Type



Click to activate depth range and resolution fields.

The *Resolution* value sets the distance(s) between surface nodes. You can use the same single resolution value for the entire surface, or use different resolutions for each range of depths.

If you select the *Single* option, you can determine the range of depths to include in the surface by using the *Depth Filter* option.

To use a single resolution value:

1. Select *Single* and enter a resolution value
2. Select *Depth Filter* and enter a range of depths.

If you select the *Multiple* option, the depth ranges and related resolution values can be saved as a template to apply when creating other surfaces.

1. select the *Multiple* option and enter a range of values.
2. Click Load to use a saved file.

   OR

3. Click to activate the fields for depth ranges and the desired resolution in metres, and type the depth and resolution values.
   - Click to delete contents of the field.
4. Click **Save** to save the settings as a template.

Values are entered in the following format:
`minimum depth - maximum depth, resolution` (all in metres). For example, `0.0-10.0,5.0` which associates depths between zero and 10 metres with a resolution of 5 metres.

5. Select a *Surface Type* from the list: Swath angle, Shoalest Depth True Position, Uncertainty or CUBE.

The next dialog box will contain the specific options for the type of surface selected here.

6. Select the *Vertical Datum*, if known, from the list.

The Vertical Datum set for the surface will be maintained in products created from it, such as finalized or interpolated surfaces. The Vertical Datum setting for a selected surface can be seen and changed in the Properties window.

7. Click **Next**.

"Swath Angle Weighting Options" on page 213

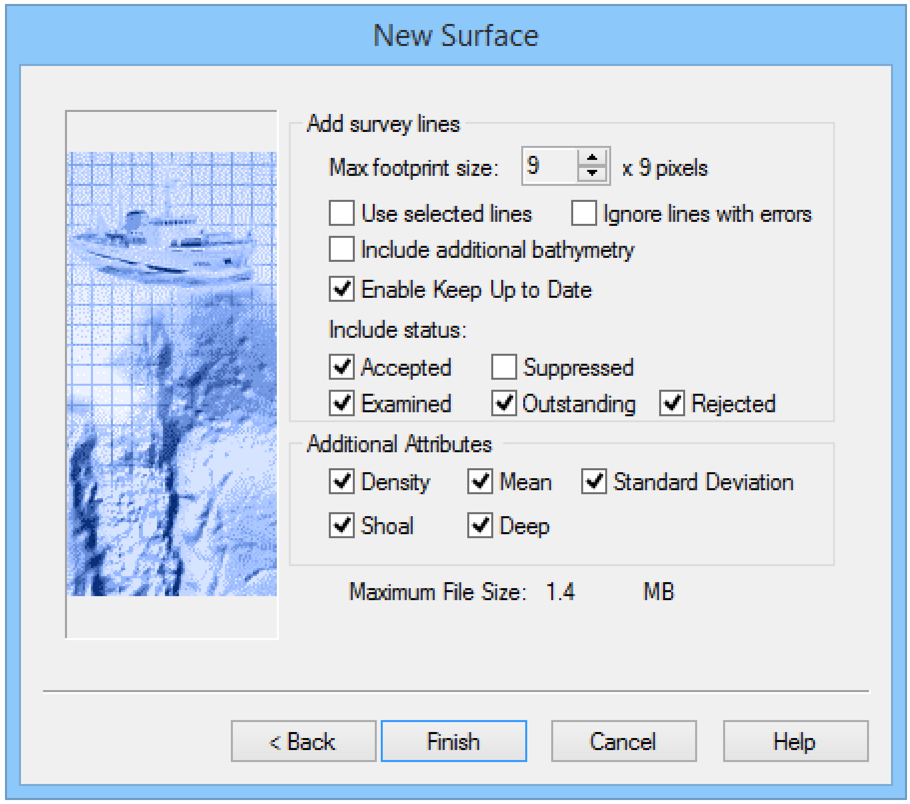
"Shoalest Depth True Position Options" on page 215

"Uncertainty Weighting Options" on page 217

"New Surface - CUBE Options" on page 291

Swath Angle Weighting Options

If you are creating a *Swath Angle Weighting* type of surface, options for adding survey lines and displaying attributes can be set before creating the surface.



Add Survey Lines:

The *Max Footprint Size* defines the maximum area in the Surface to which a sounding is applied. Use this value to prevent over-expansion of the footprint due to large depth spikes or very shallow grazing angles.

1. Click the up and down arrow buttons in the *Max Footprint Size*.

If one or more track lines were selected before the surface creation process was started, the *Use Selected Lines* check box is enabled.

2. Select *Use Selected Lines* if you want to create the surface only on the selected line or lines.

Use the *Ignore Lines with errors* option so that the surface creation is not interrupted if it encounters bad data. The process will skip the part of the line containing bad data and continue creating the surface with the next line. Once the surface is complete you can decide whether to remove the partial line.

3. Select *Ignore Lines with errors* to have bad line data omitted from the surface.

4. Select *Include Additional Bathymetry* to include water column bathymetry added to project.

The *Enable Keep Up to Date* option is set to track the outdatedness of a surface. This option must be set during surface creation or afterwards in the properties, in order to use the *Automatic Surface Update* function in Tools > Options >General.

5. If you do not want to track outdatedness, de-select the *Enable Keep up to Date c*heck box.
6. Select or de-select the check boxes for specific status to include data in or hide it from the new surface.

   By default, data with Accepted, Examined or Outstanding status are set to be included in the surface. Suppressed or Rejected data can be included in the surface by selecting the check box.

Additional Attributes

7. Select one or many of the *Additional Attributes* to create a surface layer that displays that attribute.

Density	creates an attribute layer that displays the density of soundings contributing to a node.
Mean	creates an attribute layer that displays the mean of all soundings contributing to a node.
Standard Deviation:	creates an attribute layer that displays the standard deviation from the mean.
Shoal	creates an attribute layer that displays the shoalest soundings contributing to a node
Deep	creates an attribute layer that displays the deepest soundings.

8. Click **Finish**.

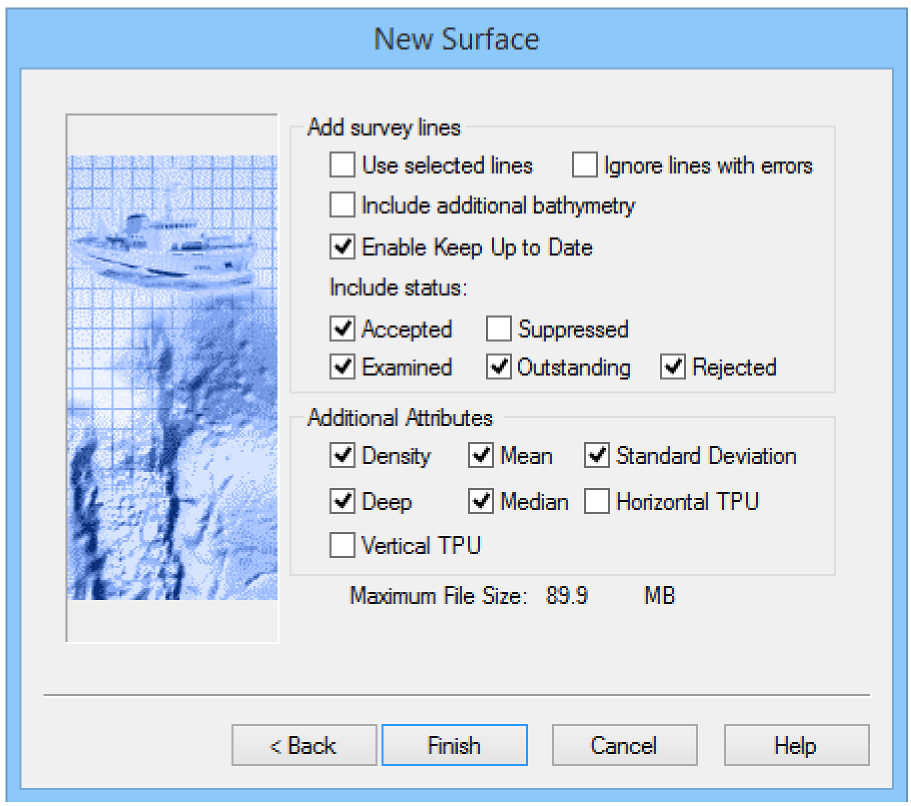
The Swath Angle surface is created.

Shoalest Depth True Position Options

If you are creating a Shoalest Depth True Position surface, this dialog box displays options specific to this surface type. In addition to options related to lines, you can also create surface attribute layers from these kinds of data:

- Median
- Horizontal Uncertainty
- Vertical Uncertainty

If such data does not exist for the lines selected, the surface creation will continue without these attributes. If, at a later time, uncertainty data becomes available, you can recompute the surface and it will be updated accordingly.



Add Survey Lines

1. Check *Use Selected Lines* if you want to create the surface only on the selected line or lines.

Use the *Ignore Lines with errors* option so that the surface creation is not interrupted if it encounters bad data. The process will skip the part of the line containing bad data and continue creating the surface with the next line. Once the surface is complete you can decide whether to remove the partial line.

2. Check *Ignore Lines with errors* to have bad line data omitted from the surface.

3. Check *Include Additional Bathymetry* to include water column bathymetry added to project.

The *Enable Keep Up to Date* option is set to track the outdatedness of a surface. This option must be set during surface creation or afterwards in the properties, in order to use the *Automatic Surface Update* function in Tools > Options >General.

4. If you do not want to track outdatedness, de-select the *Enable Keep up to Date c*heck box.
5. Select or de-select the check boxes for specific status to include data in or hide it from the new surface.

   By default, data with Accepted, Examined or Outstanding status are set to be included in the surface. Suppressed or Rejected data can be included in the surface by selecting the check box.

Additional Attributes

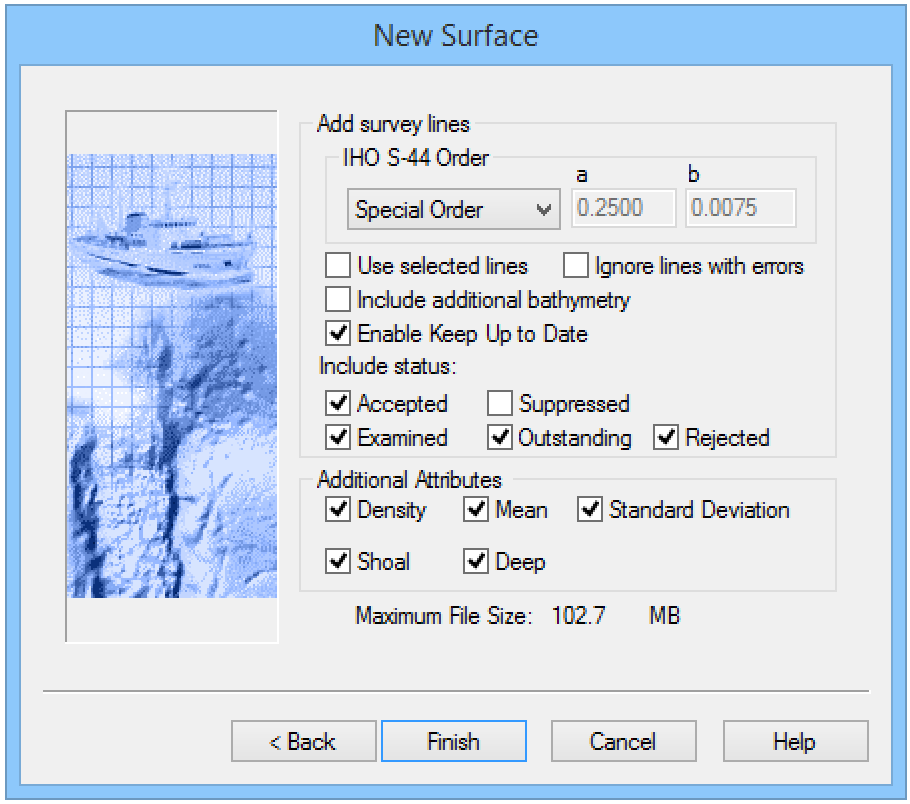
6. Select one or many of the *Additional Attributes* to create a surface layer that displays that attribute.

Density	creates an attribute layer that displays the density of soundings contributing to a node.
Mean	creates an attribute layer that displays the mean of all soundings contributing to a node.
Standard Deviation:	creates an attribute layer that displays the standard deviation from the mean.
Deep	creates an attribute layer that displays the deepest soundings.
Median	creates an attribute layer that displays the median values of soundings contributing to the node
Horizontal TPU	creates an attribute layer that displays the Horizontal TPU value for the shoalest sounding used for the Depth Attribute layer.
Vertical TPU	creates an attribute layer that displays Depth TPU value for the Shoalest sounding used for the Depth attribute layer

7. Click **Finish**.

Uncertainty Weighting Options

If you are creating an *Uncertainty* surface, the options for it are displayed.



Add Survey Lines

1. Select an *IHO S-44 Order* from the drop-down list.

The *a* (constant depth error) and *b* (factor of depth dependent errors) fields are automatically filled when a survey order is selected. The values are read from `C:\Program Files\CARIS\HIPS\<version>\System\IHO_Standards.xml`.

2. If one or more track lines were selected before the surface creation process was started, the *Use Selected Lines* check box is enabled. Select this check box if you want to create the surface for the selected line or lines.

Use the *Ignore Lines with errors* option so that the surface creation is not interrupted if it encounters bad data. The process will skip the part of the line containing bad data and continue creating the surface with the next line. Once the surface is complete you can decide whether to remove the partial line.

3. Check *Ignore Lines with errors* to have bad line data omitted from the surface.

4. Check *Include Additional Bathymetry* to include water column bathymetry added to project.

The *Enable Keep Up to Date* option is set to track the outdatedness of a surface. This option must be set during surface creation or afterwards in the properties, in order to use the *Automatic Surface Update* function in Tools > Options >General.

5. If you do not want to track outdatedness, de-select the *Enable Keep up to Date* check box.
6. Select or de-select the check boxes for specific status to include data in or hide it from the new surface.

   By default, data with Accepted, Examined or Outstanding status are set to be included in the surface. Suppressed or Rejected data can be included in the surface by selecting the check box.

Additional Attributes

7. Select one or many of the *Additional Attributes* to create a surface layer that displays that attribute.

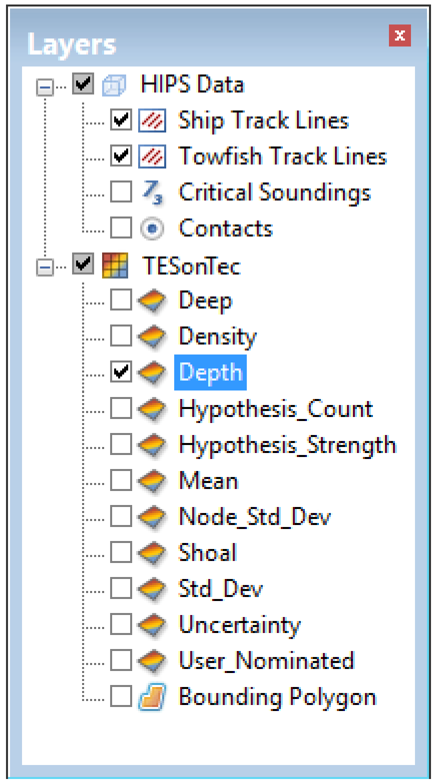
Density	creates an attribute layer that displays the density of soundings contributing to a node.
Mean	creates an attribute layer that displays the mean of all soundings contributing to a node.
Standard Deviation:	creates an attribute layer that displays the standard deviation from the mean.
Shoal	creates an attribute layer that displays the shoalest soundings contributing to a node
Deep	creates an attribute layer that displays the deepest soundings.

8. Click **Finish**.

The Uncertainty surface is created.

View Surfaces

When a surface is created it is listed in the Layers window. Attributes such as Depth are displayed as child layers of the surface.



To view a surface, or its layers:

1. Expand the data tree in the Layers window.
2. Select the check box beside the surface layer.

The following attributes layers can be displayed in the surface (depending on the weighting method you selected when creating the surface):

- **Deep:** Displays the deepest soundings contributing to the node.
- **Depth**: Depth at the node.
- **Density**: The number of soundings contributing to a node.
- **Standard Deviation**: The number of standard deviations that the node is from the mean.
- **Uncertainty**: The depth uncertainty assigned to each node.
- **Mean**: The mean depth calculated from all soundings contributing to a node.
- **Shoal**: The shoalest sounding contributing to a node.

3. To view or hide layers, select or clear the check boxes beside the layer.

Properties of Surfaces

Use the Properties window to view information about a selected surface or to set the display options for its attribute layers.

"View Surface Properties" on page 220

"Attribute Layer Properties" on page 222

"Group Multiple Surfaces" on page 244

View Surface Properties

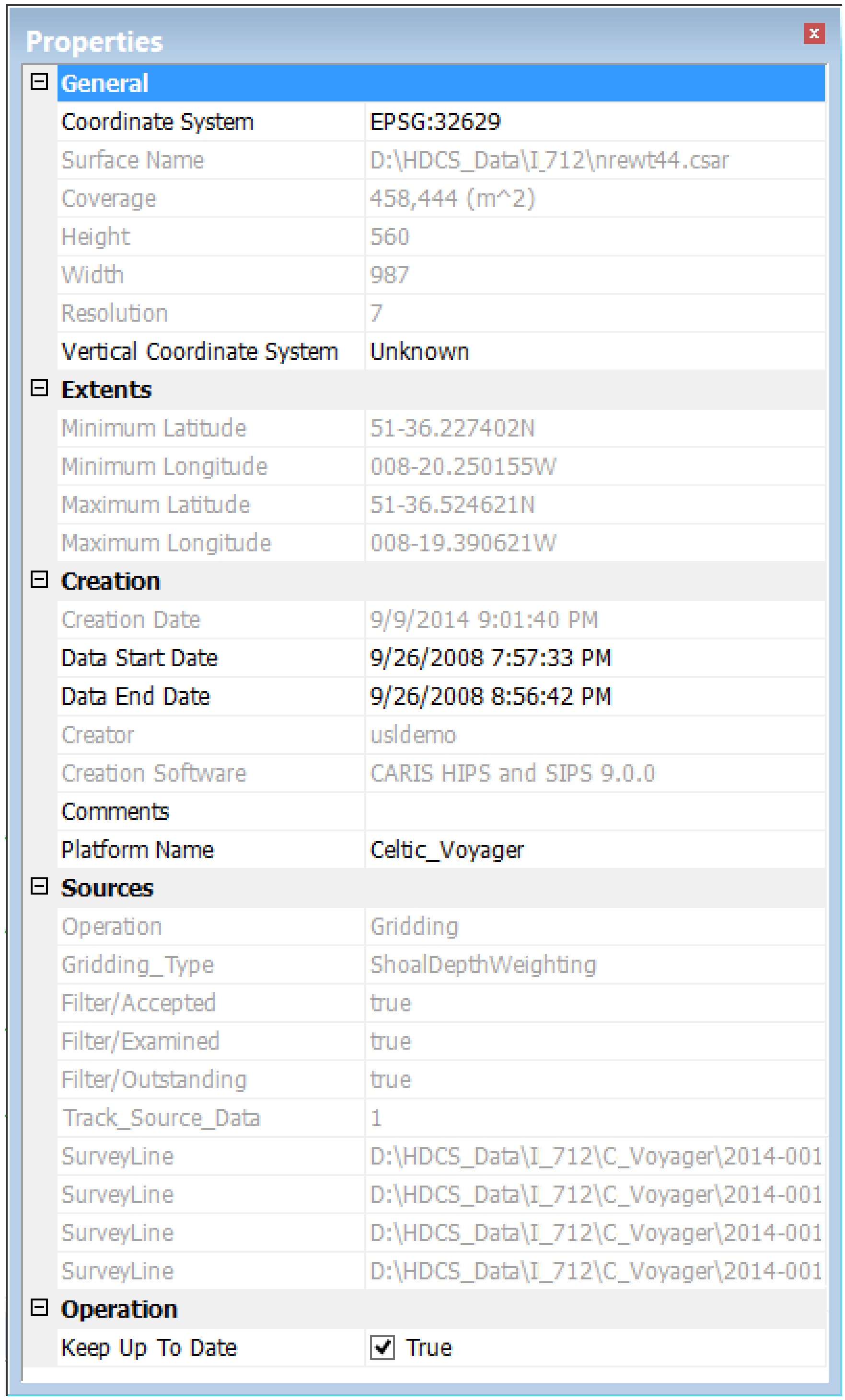
1. Select a surface layer in the Layers window.
2. Select the Properties window command.

Menu	Window > Other Windows >Properties

Most of this information is read-only, however, the following fields can be edited:

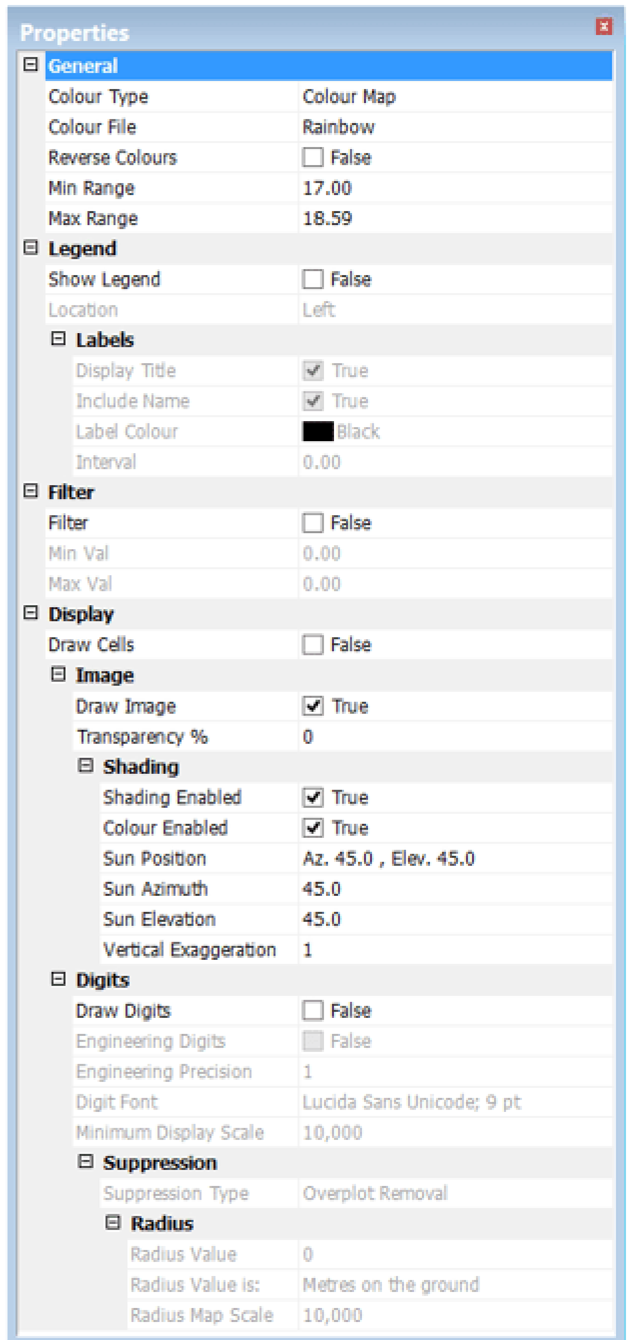
Properties	Function
General	
Coordinate System	Displays the name of the coordinate system used. To see further details, click **Browse**.
Vertical Coordinate System	Select the appropriate vertical datum for the surface data from a drop-down list.
Creation:	
Data Start Date	Date can be modified using the drop-down calendar
Data End Date	Date can be modified using the drop-down calendar
Comments	This field contains the contents of the Comments field from the first step in creating a surface. This can be the mandatory <abstract> element for 19115 metadata, or other information about the surface entered during creation. This content can be edited.
Platform Name	Displays the vessel name.
Operation:	
Keep Up To Date	This option is set to track the outdatedness of a surface. In order to use the *Automatic Surface Update* in Tools > Options >General, *Keep Up to Date* must be set during surface creation or afterwards in the properties.

The Properties window displays the surface information, as illustrated below.


Properties
General
Coordinate System
EPSG:32629
Surface Name
D:\HDCS_Data\I_712\nrewt44.csar
Coverage
458,444 (m^2)
Height
560
Width
987
Resolution
7
Vertical Coordinate System
Unknown
Extents
Minimum Latitude
51-36.227402N
Minimum Longitude
008-20.250155W
Maximum Latitude
51-36.524621N
Maximum Longitude
008-19.390621W
Creation
Creation Date
9/9/2014 9:01:40 PM
Data Start Date
9/26/2008 7:57:33 PM
Data End Date
9/26/2008 8:56:42 PM
Creator
usldemo
Creation Software
CARIS HIPS and SIPS 9.0.0
Comments
Platform Name
Celtic_Voyager
Sources
Operation
Gridding
Gridding_Type
ShoalDepthWeighting
Filter/Accepted
true
Filter/Examined
true
Filter/Outstanding
true
Track_Source_Data
1
SurveyLine
D:\HDCS_Data\I_712\C_Voyager\2014-001
SurveyLine
D:\HDCS_Data\I_712\C_Voyager\2014-001
SurveyLine
D:\HDCS_Data\I_712\C_Voyager\2014-001
SurveyLine
D:\HDCS_Data\I_712\C_Voyager\2014-001
Operation
Keep Up To Date
True

Attribute Layer Properties

All of the properties for an attribute layer can be edited, however some fields are only activated when others are set. For example, *Sun Position* and *Vertical Exaggeration* fields are greyed out unless *Shading Enabled* is set to “True”.



The properties of the Bounding Polygon layer are different from those of the other attribute layers. See "Bounding Polygon Properties" on page 221.

The properties of surface attribute layers are organized into:

"General options" on page 223

"Legend options" on page 224

"Filter options" on page 225

"Display options" on page 226

General options

General properties control the colour display for the attribute layer. Use either a colour map file or a colour range file to display data values as colours in the display. Colours can be reversed when using a colour map file.

See "Colour Map Editor" on page 44 for more information.

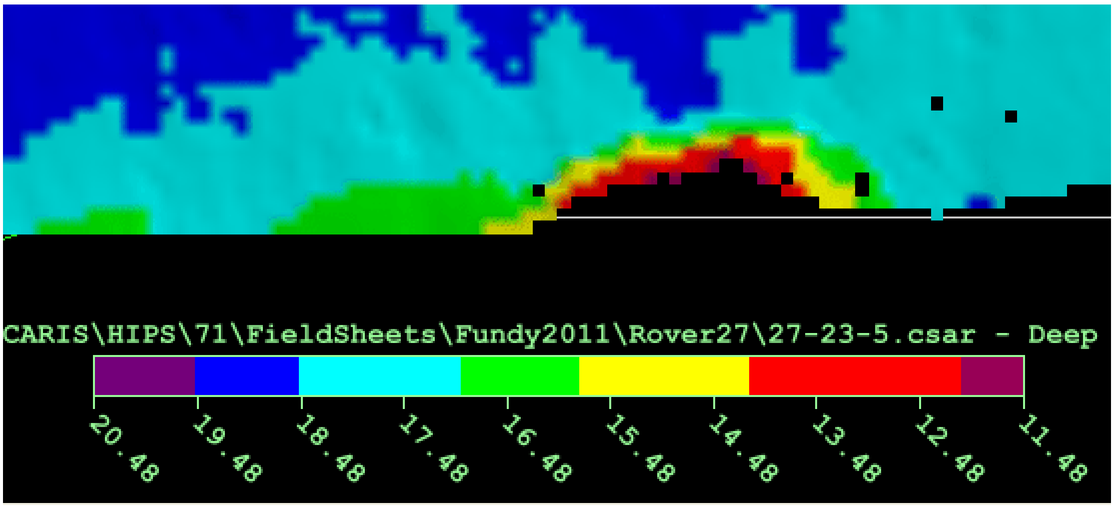
Property	Function
General	
Colour Type	Select Colour Map or Colour Range (Colour Map activates other fields).
Colour File	Displays the active colour file based on the *Colour Type* selected above. The default colour map is Rainbow. If you chose *Colour Range*, but no colour range files exist in the directory, the surface will automatically be coloured grey. By default, the first colour range file in the drop-down list will be selected.
Reverse colours	Select to map colour values 1-255 as maximum to minimum values instead of from minimum to maximum.
Min range	The minimum value for the attribute layer to which the colour file is applied, e.g., the shoalest data on the Depth layer or the minimum number on Density layer.
Max range	The maximum value for the attribute layer to which a colour is applied e.g., the deepest sounding on the Depth layer or highest number on the Density layer.

The *Min Range* and *Max range* values determine the upper and lower extents to which the colour file is applied. The Minimum of the range is mapped to the first value in the colour file, and the maximum to the last value.

The Min and Max range is taken from the data for the selected layer, and can be adjusted. For example, if the minimum of the range is increased, all data falling below that value will be mapped to the first colour defined in the colour file. Similarly, if the maximum in lowered, all data falling above that maximum will be coloured with the last colour in the colour file.

Legend options

You can add a raster legend to the display, to identify which attribute values (e.g., depths) are represented by which colours.

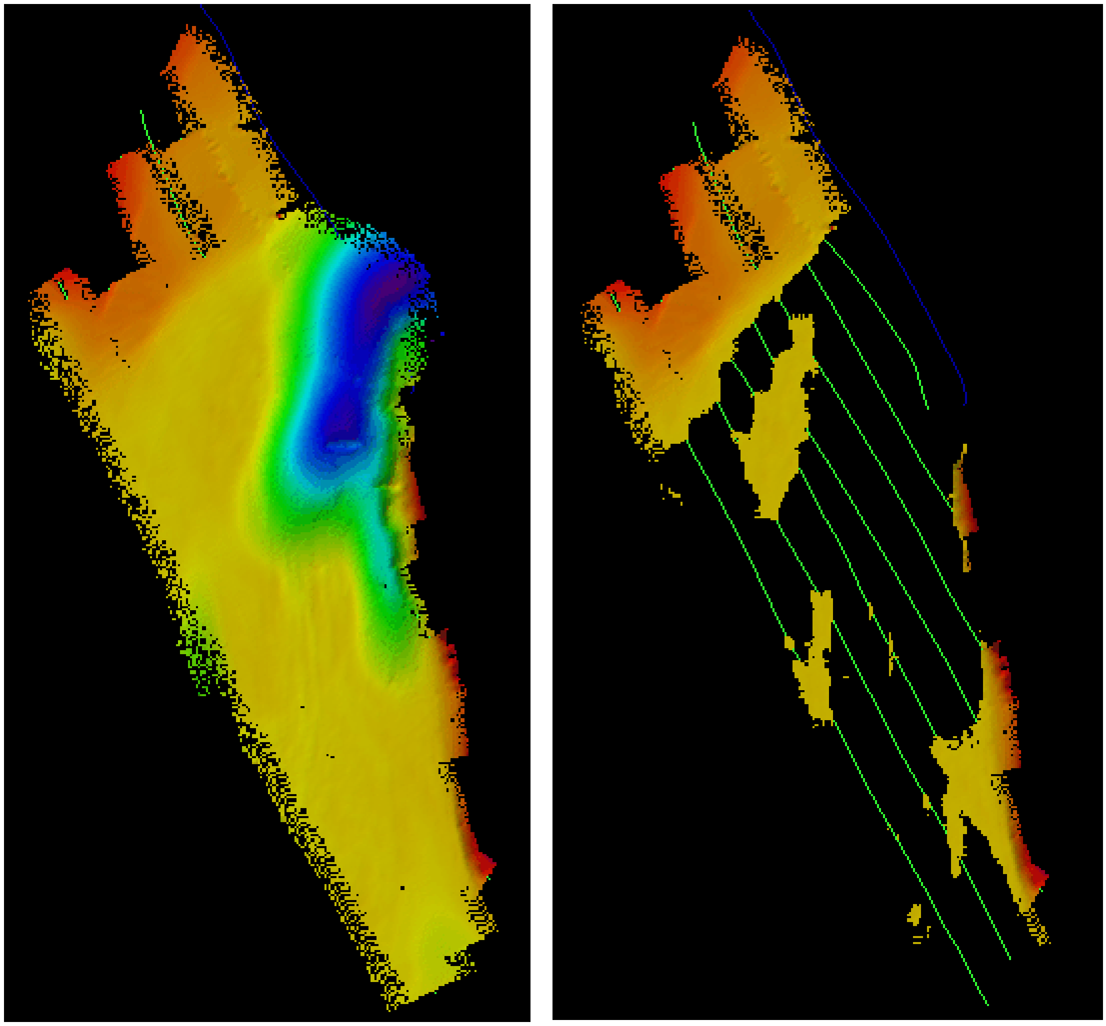


See also "RASTER LEGEND" ON PAGE 43

Property	Function
Legend:	
Show Legend	Set to True to display a legend to identify the range of values to which each colour is mapped.
Location	Select the position of the legend, e.g., Left of the display, bottom of the display etc.
Labels:	These are options for descriptive text on the legend.
Display Title	Set to True to display the name of the attribute layer and units, e.g., Depth (m).
Include Name	Set to True to display the full path and name of the attribute layer.
Label Colour	Set the colour of the labels on the legend. The default colour is black.
Interval	Set an interval between the values displayed on the legend scale

Filter options

Use filter controls to limit the display of data to that which falls between the minimum and maximum values. The image below illustrates filtering to display a range of shoalest depths.



Property	Function
Filter:	
Filter	Set to True to display only the surface data with values between the Filter minimum and maximum values (as set in the fields below).
Min val	Set a minimum value below which data will not be displayed.
Max Val	Set a maximum value above which data will not be displayed.

Display options

Display properties include controls for display of images and digits. The Draw Cells property will display a grid representing the nodes of the surface layer. With large data sets, you may need to zoom in to see this grid, as in the illustration below.

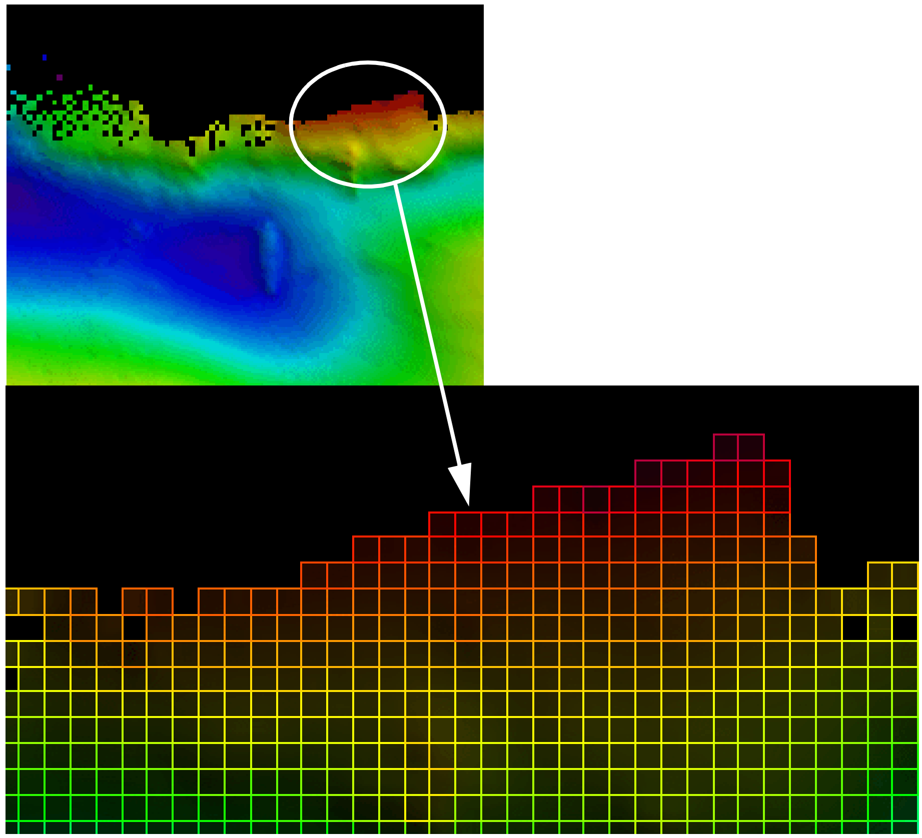


Image properties control whether or not the selected attribute layer is displayed, and options such as transparency and shading. Shading effects reflect the sun elevation and azimuth values.

Properties for display of digits include the option to display data in engineering format instead of hydrographic format. For example, hydrographic format would show a depth as 12_7. Engineering format would display the same value as 12.7 (to as many decimal places as set in the Engineering Precision field).

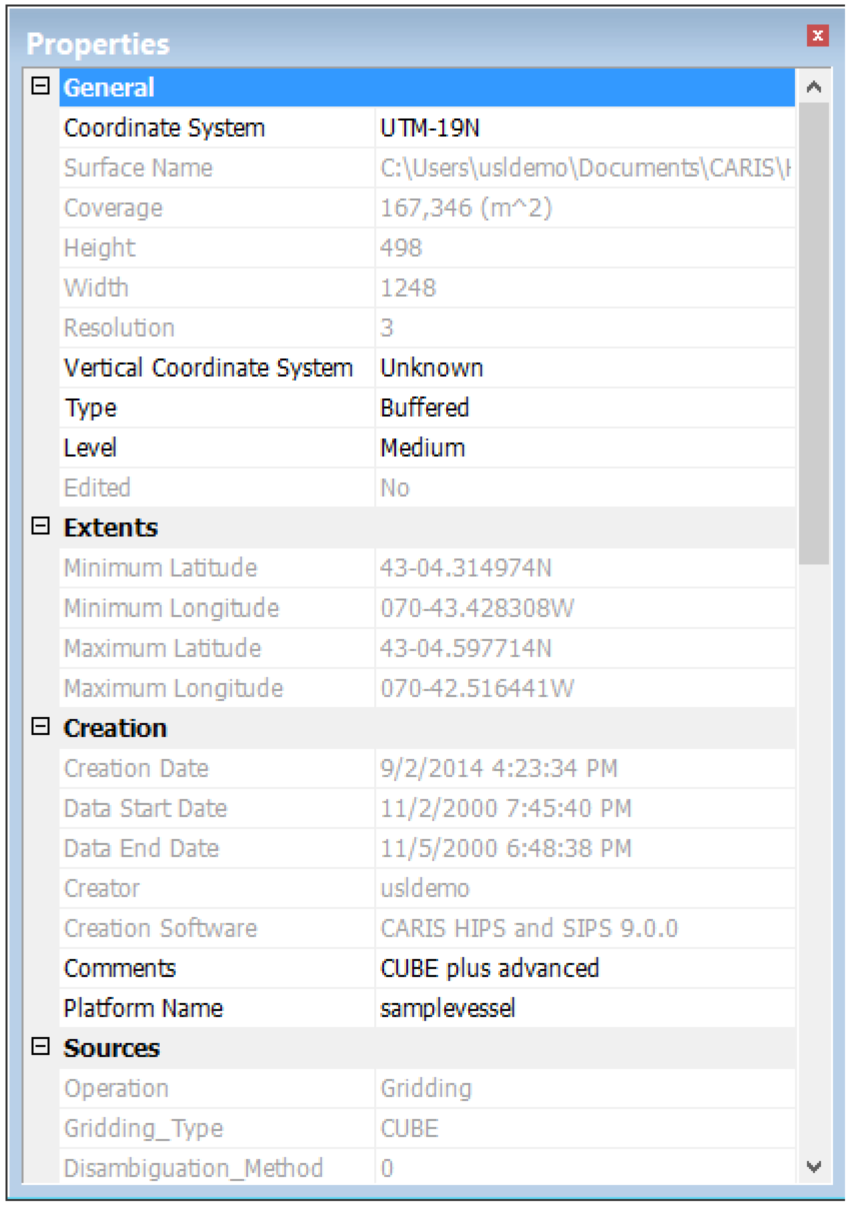
When using engineering digits for depth values, sounding rounding rules are applied. However, only the last digit in the sounding will be rounded to ensure that the desired precision is not lost.

Property	Function
Display:	
Draw Cells	Set to True to display a grid representing the nodes of the surface.
Image:	
Draw Image	Set to True to display the selected layer or grid. Default setting is True.
Transparency	Type a *Transparency* percentage for the selected layer. The higher the percentage value, the more transparent the layer will appear.
Shading:	
Shading Enabled	Set to True to turn on shading to heighten the ridge detail for fine-scaled features.
Colour Enabled	Set to display the attribute layer using the full colour map or file. If set to False, the layer will be displayed as grey only.
Sun Position	5. Click in the field to display the sun compass. 6. Move the symbol to shift the light direction. 7. Click in the display to see the change in shading. When the sun symbol is in the centre of the compass, the light is represented as directly above the image. The height and direction from which the light is displayed is shown by the *Sun Elevation* and *Sun Azimuth* values.
Sun Azimuth	An adjustment here is reflected in the *Sun Position* field.
Sun Elevation	An adjustment here is reflected in the *Sun Position* field.
Vertical Exaggeration	Increase the exaggeration value to heighten the vertical detail.
Digits:	
Draw Digits	Set to True to display data values.
Engineering Digits	Set to True to display the digits of the data nodes with decimal places instead of subscripts, and in regular (non-slanted) text.
Engineering Precision	Set the number of decimal places to include when values are displayed in engineering digits. Default is 1 decimal place.
Digit Font	Click the Browse button to select the font from the Fonts dialog box.
Minimum Display Scale	Set the value of the smallest scale at which digits will be displayed.

Property	Function
Suppression:	Filter out specific data based on the number of soundings in a selected area. Fields are activated when Draw Digits is set to “True”.
Suppression Type	Apply Overplot Removal (to remove overlapping soundings), Radius (filter out soundings within a set radius), or None.
Radius:	Size of the area to which the suppression is applied.
Radius Value	The radius defined as metres on the ground, or as the number of millimetres between soundings at map scale.
Radius Map Scale	Value to apply to scale.

Bounding Polygon Properties

A bounding polygon displays properties similar to those of a parent surface (Extents, General, Creation, and Sources). (See "PROPERTIES OF SURFACES" ON PAGE 220.)



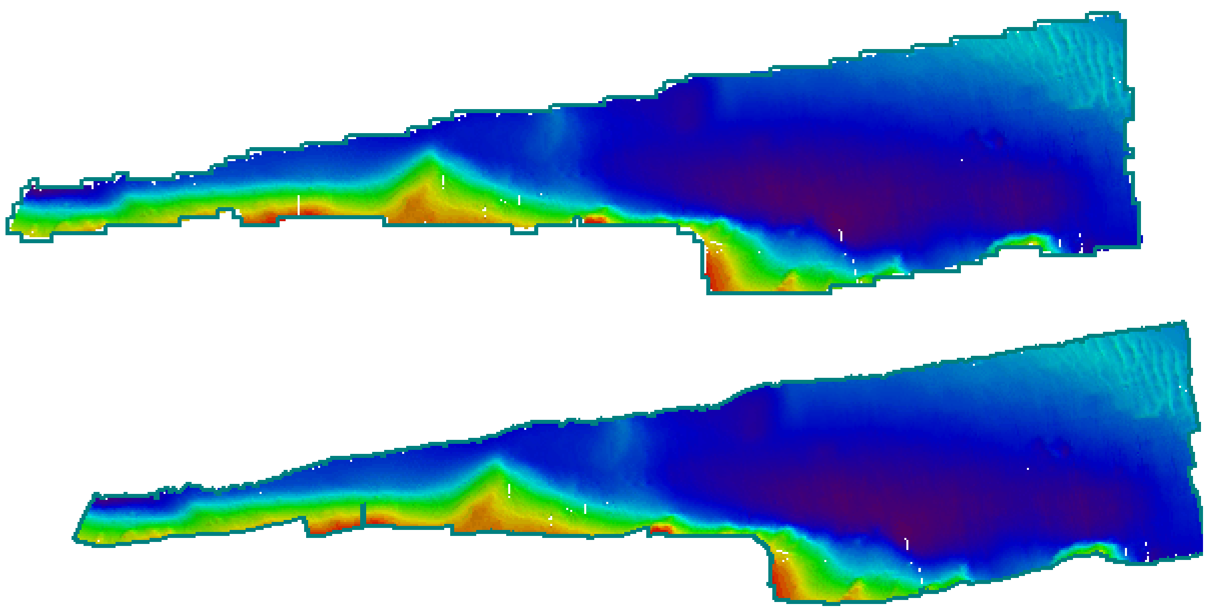
How closely the bounding polygon fits the extents is determined by the level of detail used to create the polygon. The polygon can be adjusted so that it more closely delineates the extent of the surface.

Settings in the *Level* field in the Properties window can be used to rebuild the polygon with a different resolution. (By default, polygons are created at a Medium level.) The polygon can be recreated using a Coarse level (less detail), or a Fine level (more detail).

To change the Level property:

1. Select the Bounding Property layer in the Layers window.
2. Select the *Level* field in the Properties window.
3. Choose a different setting from the drop-down list.
4. Right-click on the Bounding Polygon layer, and select Rebuild Automatically from the pop-up menu.

The bounding polygon will be rebuilt. The example below shows the difference between the polygon drawn at the default level and rebuilt at a Fine level.



Vertical Reference System

A vertical reference system is a 3D coordinate reference system in which position is defined by latitude, longitude, height or depth and a linear unit of measure, for example, feet or metres.

When the Z values in a vertical reference system represent depths, the Z axis direction is “positive-down”. (This is consistent with the HIPS and SIPS depth convention.)

In HIPS and SIPS the identification of the vertical reference system information used for a project can be added to a surface during its creation. Alternatively, it can be added to the Vertical Coordinate System field in the Properties of an existing surface.

A database of defined vertical reference systems is available in HIPS and SIPS through the Vertical System Editor. This information is drawn from the European Petroleum Survey Group (EPSG) database of reference systems, which can be found at: http://www.epsg-registry.org.[1]

Vertical Reference System Editor

The existing datum, ellipsoid and reference system entries in the database are read-only. Each entry has a unique name and numerical identifier.

“Vertical Reference System” on page 231

User-defined entries can also be added to the database using the Editor.

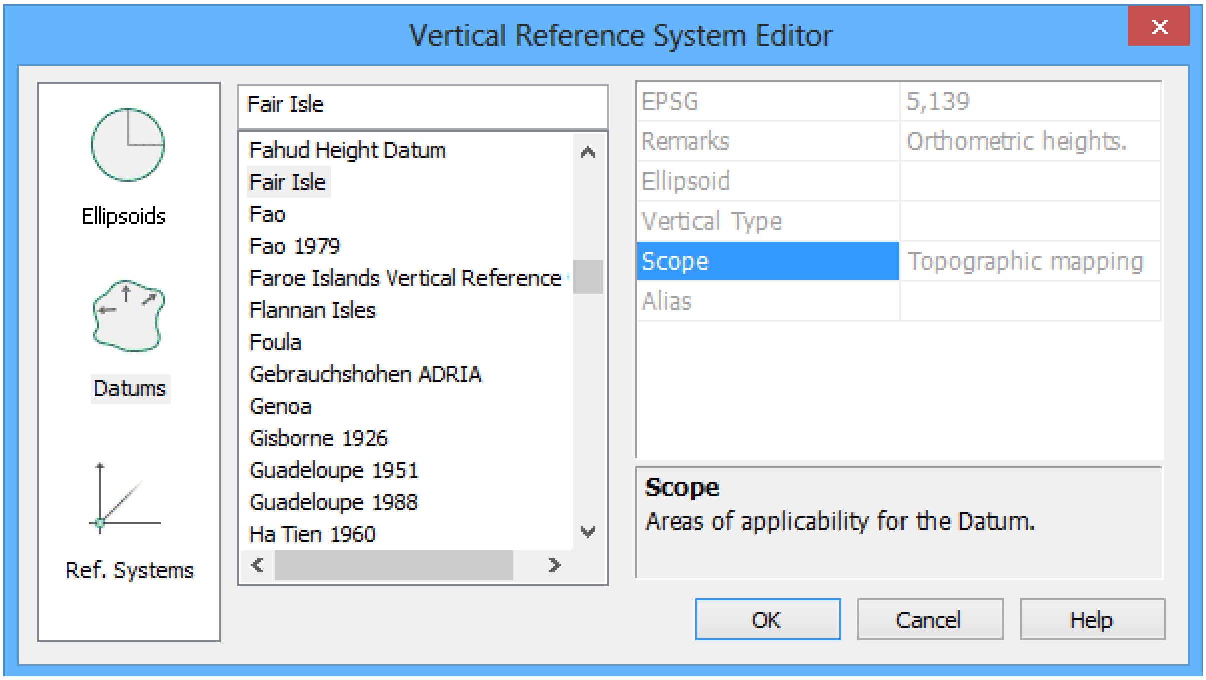
To open the Vertical Reference System Editor:

Menu	Tools > Editors > Vertical Reference System

1. Select the **Reference System Editor** command.

The Vertical Reference System Editor is displayed.

1. Use of material from OGP Surveying and Positioning Committee’s EPSG Geodetic Parameter Registry does not imply endorsement by OGP Surveying and Positioning Committee of CARIS products or services.



The leftmost column lists the types of coordinate system elements for which you can view properties.

2. Select the category of data you want to view, for example, Datums.

This will activate the listing of all the entries for that category. to view the properties of a specific entry,

3. Select an item from the list. (The example above shows the Fair Isle datum selected.)

The properties for the selected datum (or ellipsoid or vertical reference system) are displayed in the pane on the right. As you select each property field, a brief description of the field is displayed below the properties. In the example illustrated above, the content of the *Scope* field show “Topographic mapping and geodetic surveying” as the areas of applicability for the selected datum.

The first property field contains the EPSG code number that identifies the selected datum, ellipsoid or reference system.

Editing the Vertical Reference database

You can add your own entries to the database, provided the names of the entries do not duplicate existing names. User-defined entries can be deleted, but original entries cannot.

To add an item:

1. Select the icon for the category of data that you want to add. For example, to add an ellipsoid, click the Ellipsoids icon.
2. Right-click anywhere in the ellipsoids listing, and select New from the pop-up menu.

A New entry is added to the list.

3. Click on the new entry and type the name for your entry, over-writing “New”.

In order for your new entry to be saved to the database, it must have values defined for the properties. If you do not define the properties, the new item will be removed from the list when the editor is closed.

4. Type appropriate information in the properties fields in the list on the right.
5. Click **OK** to save your changes to the database and close the dialog box.

Deleting entries

While the original items in the Vertical Reference System database are read-only, any user-created entries can be deleted.

To delete a user-defined entry:

1. Select the entry.
2. Right-click in the pane and select “Delete” from the pop-up menu.
3. Click OK in the dialog box to confirm the deletion.

Compute Surface Statistics

The Compute Statistics command generates statistics on the nodes in an attribute layer of a surface.

The statistics calculated are:

- Minimum value in the attribute layer of the surface
- Maximum value in the attribute layer
- Mean of the values in the layer
- Area
- Standard Deviation (Std_dev)
- Total count

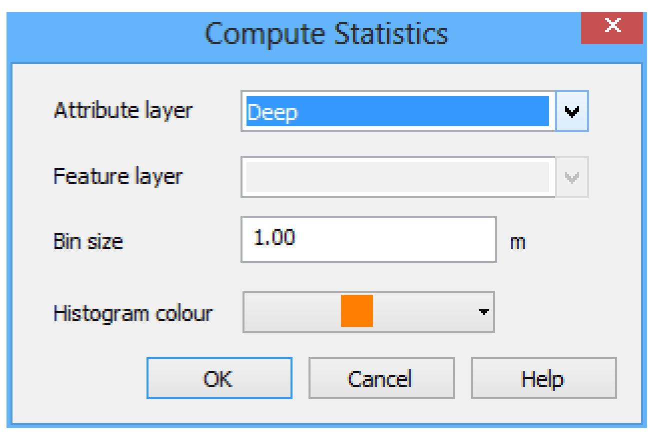
These statistics are reported together with a histogram depicting the distribution of the points/nodes within the minimum/maximum range of values.

To compute the statistics of an attribute layer:

1. Select the parent layer of the attribute layer.
2. Select the Compute Statistics command.

Menu	Tools > Surfaces > Compute Statistics
Pop-up	Compute Statistics (Layers window)

The Compute Statistics dialog box is displayed.



3. Select the *Attribute layer* for which you want to compute statistics, for example, the Depth or Std_Dev layer.
4. From the *Feature layer* list, select the method you want to use to compute statistics:
   - *Entire Dataset*, which will compute statistics for the entire area of the surface
   - *By Digitizing* a polygon around the area of the surface that will be used to compute statistics, or

- *Using Feature Layer*, which will compute statistics on a limited area of the surface based on a polygon in a feature layer.

> To use the feature layer method, you must have a feature layer open that contains `cvrage` features, prior to selecting the command.

If you select the digitizing option, you must digitize a coverage (`cvrage`) area object around the section of the surface for which you want to compute statistics.

Bin Size

The Bin size field defines the range of each grouping of data that will be displayed as a column in the histogram. Each bin or column will contain only data within the range set by this field.

The unit of measure for the Bin size field is controlled by the type of data in the selected attribute layer. For example, Depth layers are reported in the units set for the data in Tools > Options > Display Units. Density and Hypothesis Count are unitless.

5. Enter a *Bin size* for the histogram.

> The Min and Max Range values in the Properties for the selected attribute layer can help you set a useful bin size for the histogram.

For example, if the range of values for the Std_Dev layer is 0 to 16, a bin size of .02 m will yield a more useful histogram than would 2m bin size.

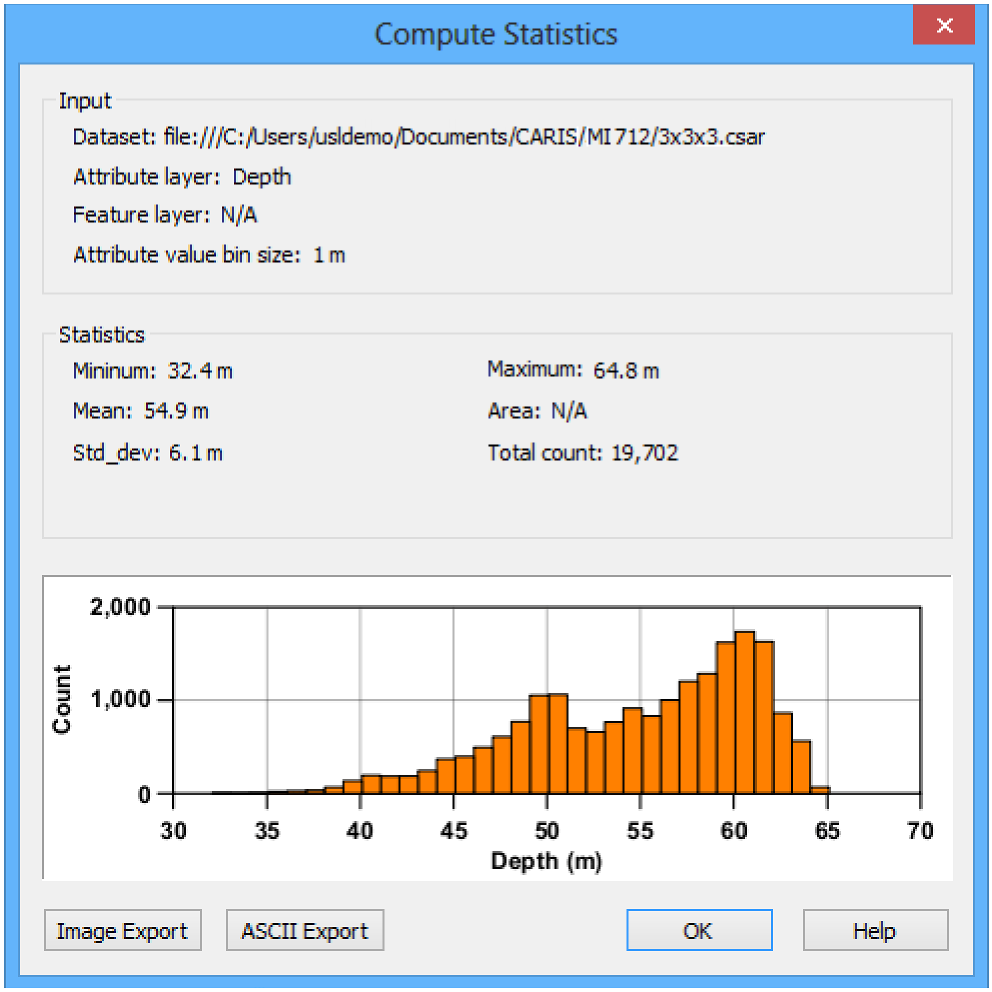
6. Select a colour for the content of the columns from the *Histogram colour* drop-down list. The default colour is white (outlined in black).
7. Click **OK**.

Statistics output

The Compute Statistics dialog box displays the input data and the computed statistics as values and as a histogram.

- The *Input* section displays the settings that were used to calculate the statistics: data set, attribute layer and bin size.
- The *Statistics* section provides the results of the analysis of the data in the selected attribute layer.
- The histogram displays the distribution of the data. The X axis is labelled with the source layer name and unit of measure. The Y axis contains the count of the data points in each bin or column.

The following image displays the statistics generated for the standard deviation layer of a surface.



To save these statistics, export them to image or ASCII formats. When exporting to an image format, only the histogram is exported.When exporting to ASCII, the *Input*, *Statistics* and histogram data will all be exported.

See "Export Histogram to Image" on page 236 and"Export Statistics to ASCII" on page 239 for more information.

8. Click **OK** to close the dialog box when finished.

Export Histogram to Image

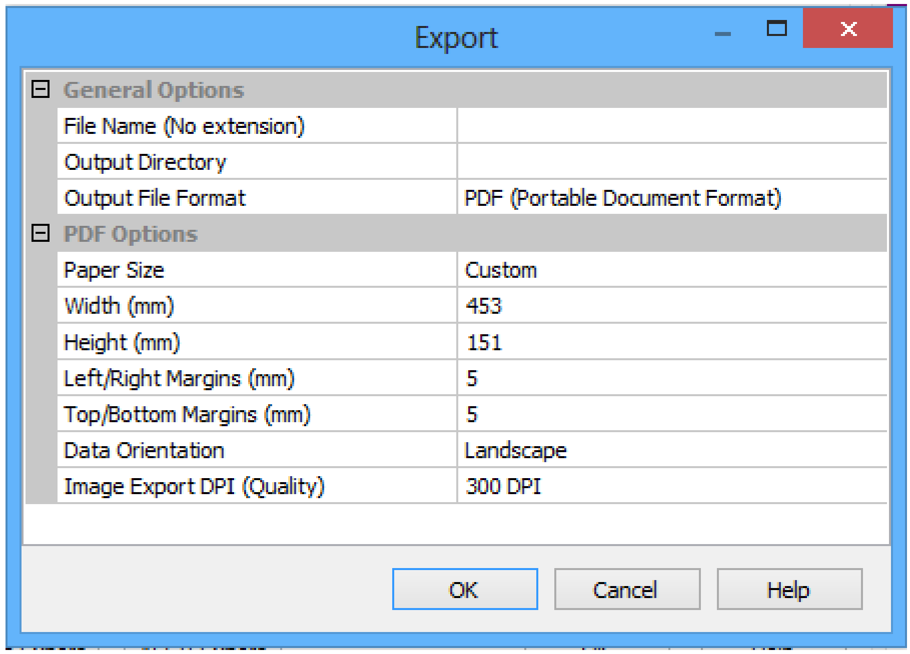
Use the Image Export option in the statistics results dialog box to export an image of the current histogram as:

- PDF (Portable Document Format)
- PS (PostScript)
- SVG (Scalable Vector Graphics)
- TIFF (Tagged Image File Format)

To export an image of the histogram:

1. Click **Image Export**.

The Export dialog box is displayed.



Use the Export dialog box to define the settings for exporting the image. Click on each row to activate the options for that field. The example above displays the options for export to PDF format.

By default, the *File Name* field is populated with a name combining the names of the surface and the attribute layer that was computed (`<surfacename>_<layername>`). You can change this file name.

2. [Optional] Type a *File Name* for the image file.
3. Click within the *Output Directory* field to enable the **Browse** (...) button.
4. Click **Browse** and select a location for the exported file.
5. Select the *Output File Format* from the drop-down list.

Once you select a format, specific options for that format are displayed in the lower section of the dialog box.

6. [Optional] Enter or select a value for the options as needed.

Export options:

PDF and PostScript options	
Paper size	Set the page size of the exported file.
Width/Height	Set the width and height of the histogram in the resulting file.
Data Orientation	Set the orientation of the histogram in the resulting file. Portrait is a vertical position; Landscape is a horizontal position. Landscape is selected by default.
Margins	Set the width of the area between the histogram and the edge of the file. The default is 5mm.
Image Export DPI (Quality)	Set the resolution used to export the histogram. The higher the resolution, the better the quality of the image when zoomed in. If the file will be used in the electronic form with the image being zoomed in tightly, a higher resolution is recommended. If the file will simply be printed, a smaller resolution is adequate.
SVG format options	
Width/Height	The width and height of the histogram in the resulting file.
Image Export DPI (Quality)	Set the resolution used to export the histogram. The higher the resolution, the better the quality of the image when zoomed in. If the file will be used in the electronic form with the image being zoomed in tightly, a higher resolution is recommended. If the file will simply be printed, a smaller resolution is adequate
TIFF format options	
Image Resolution	The resolution (quality) at which to export the image. The higher the resolution, the closer the image can be zoomed, but the larger the file.
Width/Height (Pixels)	Set the width and height of the resulting image in pixels. These fields are controlled by the resolution of the image. As the DPI is increased or decreased, so is the number of pixels.
Image Depth	Set the number of values applied to each pixel in the image (one value for each colour, plus one for transparency if using 32 Bit). If you would like a transparency setting applied to the background colour, the 32 Bit (RGBA) option must be selected.
Background Colour	The colour displayed in the background of the histogram. The default is black.
Background Alpha	(This option is only available when the Image Depth is set to 32 Bit RGBA). Set the degree of transparency applied to the background colour. The default setting is zero transparency.

Compression	Compress the resulting image file during export to decrease the file size. This option is only available when using the TIFF format. There are various compression methods available, each of which performs differently. By default, LZW is used.

7. Click **OK**.

The image file is saved to the specified location.

Export Statistics to ASCII

Use the ASCII Export option in the Compute Statistics results dialog box to export all of the statistical information to an ASCII text file. This file will include:

- Information about the computed surface
- The resulting statistics
- The centre value of each bin and the count of data points within the bin.

To export to an ASCII file:

1. Click **ASCII Export**.

A standard Save As dialog box is displayed. By default, the *File Name* field is populated with a name combining the names of the surface and the attribute layer that was computed (`<surfacename>_<layername>`).

2. [Optional] Enter a new *File name.*
3. Select a location for the file and click **Save**.

An ASCII text file is created in the specified location.

Surface QC Report

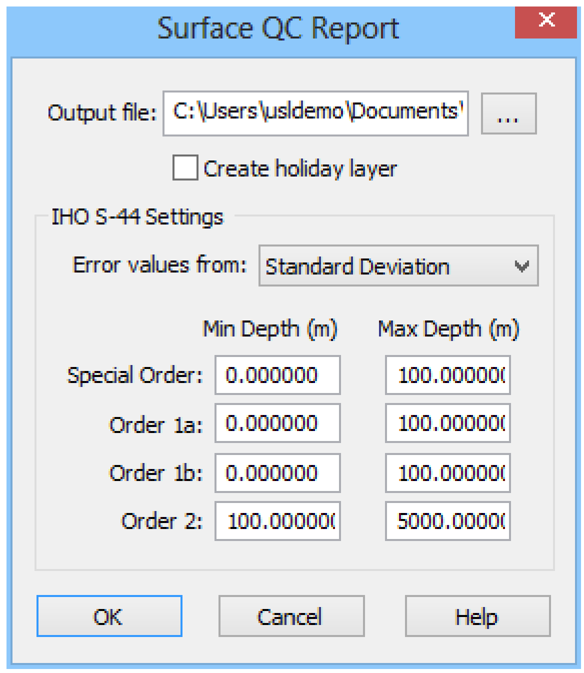
The Surface Quality Control report lists significant data gaps within the surface. It also analyzes how much of your data conforms to the confidence levels set by the S-44 standard for the four orders of survey.

1. Select the surface parent layer in the Layers window.

Menu	Tools > Surfaces > QC Report
Pop-up	QC Report (Layers window)

2. Select the Surface QC Report command.

The Surface QC Report dialog box is displayed.



1. In the *Output file* field, click **Browse** and set a directory path and name for the QC report.

Significant data gaps are known as "holidays". A holiday is formed when enough gaps are clustered together within a specified radius. By default, the layer will be created which displays these holidays.

2. To create the layer, select the *Create holiday layer* check box.

IHO S-44 Settings

Use these settings to compare uncertainty values contained in the surface to the S-44 standard. These values are drawn from the Standard Deviation and Uncertainty layers. This comparison will show how many nodes in the surface fall within the S-44 categories.

3. Select the source of *Error values from* the drop-down list:
   - Uncertainty layer (which contains the uncertainty "model" values entered when the surface was created),
   - Standard Deviation (the values showing how the data varies from the "model"),

- Greater of the two (uncertainty or standard deviation, whichever is larger value)
- Lesser of the two.

For a swath angle surface, the error values come from the Standard Deviation layer only. (The option will be greyed out, but it is applied.)

4. Enter the minimum and maximum depth levels for each survey order, from the minimum and maximum depths in your surface. (You can have overlapping depth levels.)
5. Click **OK**.

The report is shown in the Output window, and in the Output file you designated. A Depth and Holiday layer is displayed in the Layers window under the other layers for the surface.

QC Report File example

```
Surface QC Report
Date and Time: 2/4/2014 3:39:49 PM
Surface:
C:\Users\usldemo\CARIS\HIPS\90\FundyData\PASG1\Jn317.csar
Holiday layer created: Yes
Error values from: Standard Deviation

Number of nodes processed: 8859
Number of nodes populated: 8858 (99.99%)
Number of holidays detected: 3
IHO S-44 Special Order:
    Range:  0.000 to 100.000
    Number of nodes considered: 8858
    Number of nodes within: 2523 (28.48%)
    Residual mean: 0.743
S-44 Order 1a:
    Range:  0.000 to 100.000
    Number of nodes considered: 8858
    Number of nodes within: 3995 (45.10%)
    Residual mean: 0.473
S-44 Order 1b:
    Range:  0.000 to 100.000
    Number of nodes considered: 8858
    Number of nodes within: 3995 (45.10%)
    Residual mean: 0.473
S-44 Order 2:
    Range: 100.000 to 5000.000
    No depths within the specified range
```

Sections of a QC report file

The first section contains information about the report file and the data holiday options set in the dialog box:

- The data and time that the QC report file was created.
- The surface on which the QC report is based
- The minimum number of nodes required to omit a node (without data) from being included in a holiday.
- If a holiday layer was created or not.
- The source of error values (the layer).

The second section contains information on the number of nodes containing data:

- The total number of nodes processed in the surface.
- The total number of nodes that contains data.
- Percentage of the surface that contains data.
- Number of data holidays located.

The third section contains information about each S-44 survey order:

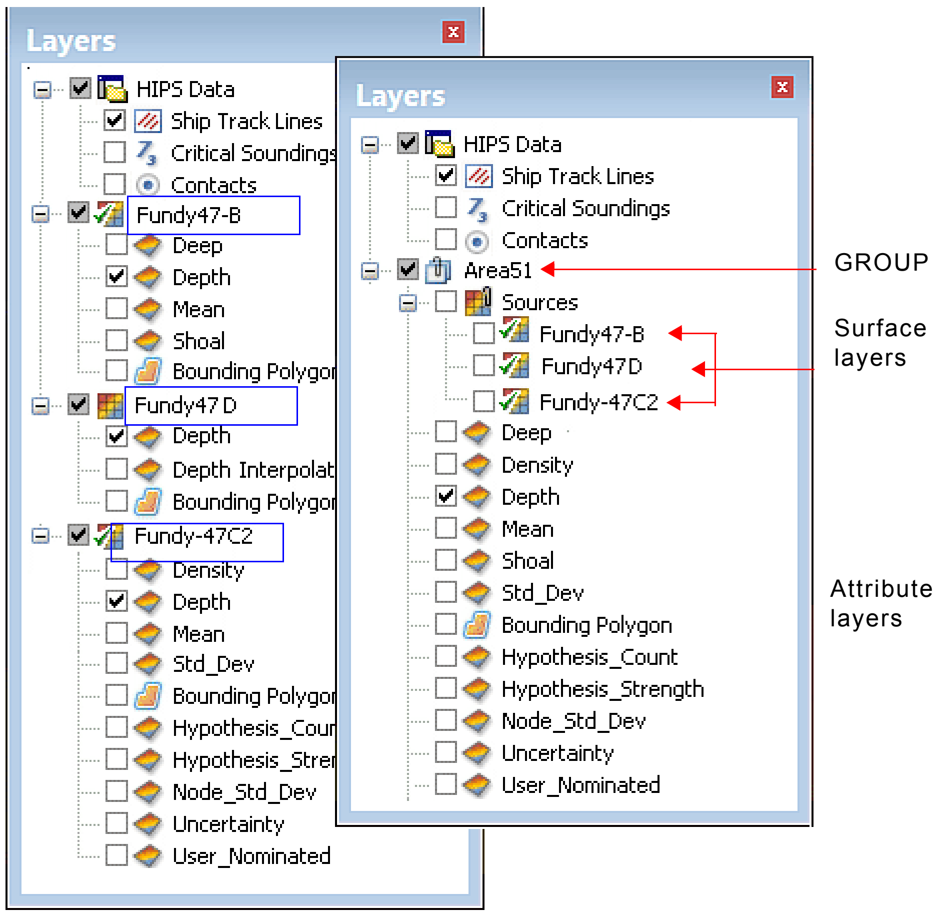
- The depth range for the survey order (set in the dialog box).
- The number of nodes within that depth range with uncertainty values that were considered for the survey order.
- The number of nodes that fall within the depth error limits for that survey order.
- The residual mean is derived from the computed and reported uncertainty.

  If the value is negative, the uncertainty values meet the requirements for the selected survey order. If the value is positive, the uncertainty values do not meet the requirements for the selected survey order. Therefore, if no nodes were considered for the survey order, the residual mean would have to be above zero.

Group Multiple Surfaces

Multiple surfaces can be grouped together, so that the same display properties can be applied consistently to all the grouped surfaces. See "Properties of Surfaces" on page 220.

Grouped surfaces are listed together under a new Sources layer, followed by their combined attribute layers, as in the example below.



When a surface is added to an existing group, the properties of the first of each type of attribute layer included in the group are applied to all other attribute layers of the same type.

For example, if the Depth layer for the grouped surfaces has Filter values set, the Depth layer for a surface added to the group will display data within the filter range, regardless of its settings before it was added to the group.

Create a group

To group multiple surfaces:

1. Right-click on a surface parent layer in the Layers window.

Pop-up	Group > Create New

2. Select the Create New command from the pop-up menu.

The New Group dialog box is displayed.

3. Type a name for the group layer in the Group Name field.
4. Click **OK**.

The group is created in the layers tree, and the selected surface is added to the group, under the Sources layer.

Add surface to group

To add another surface to the group:

1. Right-click on the surface to be added.

Pop-up	Group > *group name*

2. Select Group from the pop-up menu.

The pop-up sub-menu will list any created and available groups.

(If a selected surface already belongs to a group, that group is greyed out in the pop-up menu, to prevent a surface from being added to a group twice.)

3. Select the group you want to add the surface to.

The surface is added to the group, under the Sources layer.

Group all like types

Group all datasets to the currently selected group that:

- are of the same type as the datasets in this group; and,
- are not already included in a different group of datasets.

1. Select a dataset.

Pop-up	Group All Like Types

2. Right-click on the group and select Group All Like Types from the pop-up menu.

Unassigned datasets of the selected group type are now included in this dataset group.

Order grouped surfaces

Datasets in a group can be arranged in ascending or descending order according to one of the following metadata attributes:
- Data Start Date
- Data End Date
- Creation Date

This ordering controls both the order of the datasets in the Layers window and their draw order in the Display window.

It is common to draw the oldest data first, having the newest data displayed on top. To achieve this order, sort By End Date (ASC).

Pop-up	Sort > By Start Date (ASC/DSC) > By End Date (ASC/DSC) > By Creation Date (ASC/DSC)

To sort grouped datasets:

1. Right-click the group layer in the Layers window.
2. Select the **Sort** command for the desired ordering.

The datasets listed under the Sources layer will be ordered according to the selected command.

Remove a surface from a group

You can remove a surface from a group. It will be listed separately in the Layers window.

To remove a surface from a group:

1. Right -click on the surface you want to remove from the group.
2. Select Group from the pop-up menu.
3. Select None from the sub-menu.

Pop-up	Group > None

The surface is removed from the group and restored to a position in the Layers window.

Ungroup

To disband the group and restore the component surfaces, use the Ungroup command:

1. Right-click on the group layer.
2. Select Ungroup from the pop-up menu.

Pop-up	Group > Ungroup

The surfaces are listed separately in the layers window.

Surface Filtering

Surface filtering uses a scaled standard deviation or uncertainty threshold to reject soundings that fall outside that threshold. You can also filter sounding depth by a direct threshold amount, for example, 2 metres from the surface.

Soundings that fall outside the threshold are assigned a Rejected by Surface Filter flag and are no longer available for processing (unless they are given an Accepted flag again).

This can greatly reduce the number of manual edits required to produce a clean sounding set.

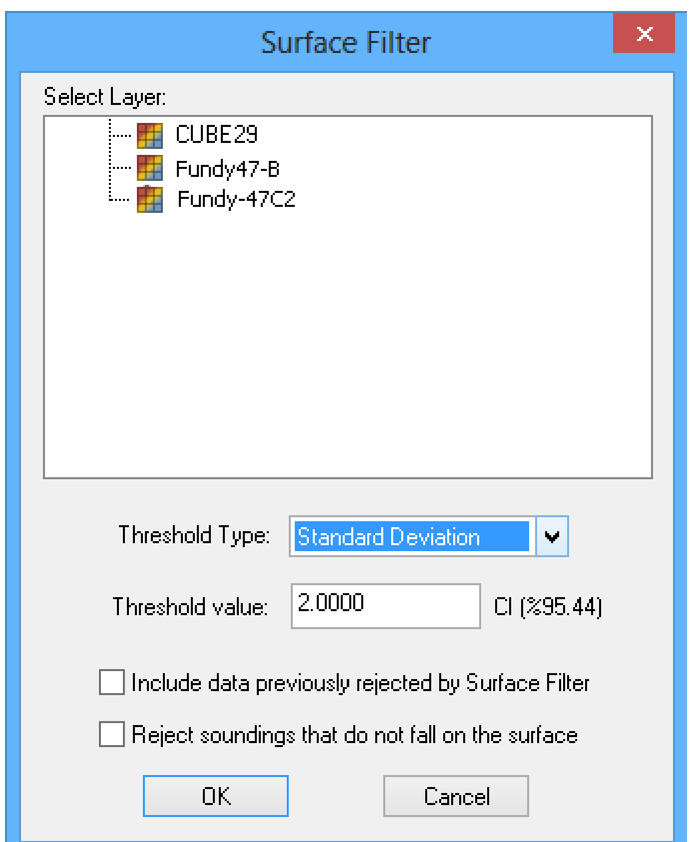
Select data to filter in two ways:

- Select specific lines to apply the filter to then choose a surface that overlaps those lines from the list in the Surface Filter dialog box, or,
- Select the surface from the Surface Filter dialog box, which will then use and filter all lines in the metadata for that surface.

Menu	Tools > HIPS Data Filters > Surface

1. Select the Surface Filter command.

The Surface Filter dialog box is displayed.



2. Select the surface to be filtered from the *Select Layer* tree.

3. Select the *Threshold Type* from the drop-down list:
   - Standard Deviation
   - Uncertainty
   - Greater of the two
   - Lesser of the two.
   - Static Value
4. Set the desired *Threshold value.*
   - For standard deviation or uncertainty thresholds, the value entered will show the confidence interval next to the field.
   - Static values are entered in units set in Tools > Options > Display > Units > Vertical units, e.g., metres.
5. Select the *Include data previously rejected...* check box to include rejected data when running the filter.
6. Select the *Reject soundings* check box to reject soundings that are offset from the surface.
7. Click **OK**.

Surface filtering can also be used in Subset Editor, where you can apply filters to a *subset* of a surface. See "SUBSETS AND SURFACES" ON PAGE 403

11

Edit Surfaces

In this chapter...

Surface Editing Tools

Swath Angle, Shoalest Depth True Position, Uncertainty and CUBE surfaces, can be edited using a number of surface tools.

You can:

- add or remove layers from a surface
- update source data and recompute the surface
- combine surfaces
- compare the differences between surfaces
- identify holidays and interpolate surfaces to fill empty nodes
- vertically shift a surface

Open

To open a surface:

Menu	File > Open

1. Select the Open command.

The Open dialog box is displayed.

2. Browse to locate the surface file.
3. Click **Open**.

The Surface is now visible in the Display window.

Delete

Permanently delete a surface.

Pop-up	Delete

1. Right-click on a surface layer in the Layers window, and select the Delete command from the pop-up menu.

The surface layer is permanently deleted from the project.

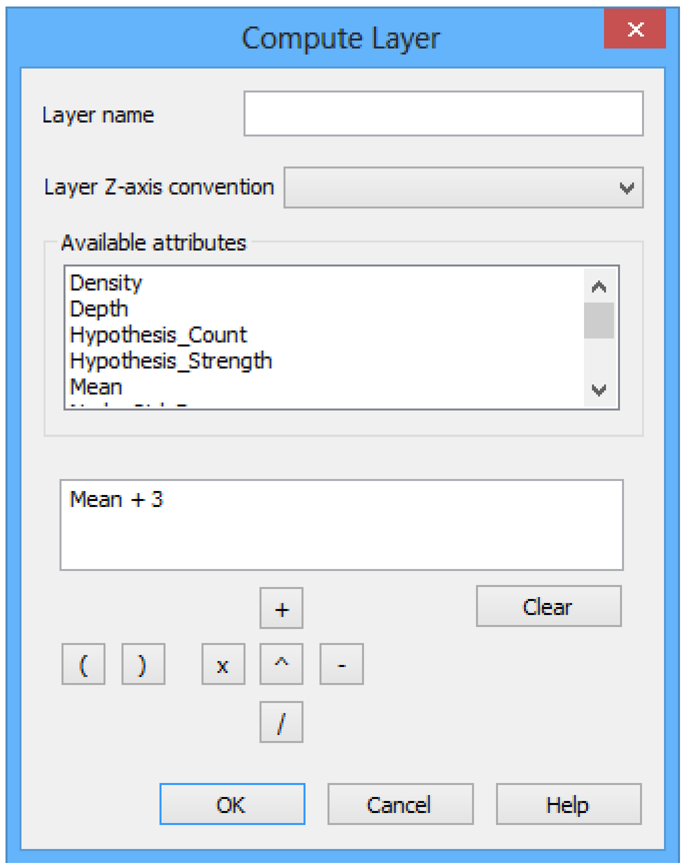
Add Layer

Use the Add Layer command to compute a new layer for an existing surface. The new attribute layer can be created from one or more existing attribute layers, numerical operators and numeric values in an equation. You cannot create a layer for an attribute that is not present in the surface.

Menu	Tools > Layers > Compute Layer
Pop-up	Add Layer (in Layers window)

1. Select a surface (parent) layer in the Layers window.
2. Select the Add Layer command from the right-click menu.

The Compute Layer dialog box is displayed.



3. Type a name for the new surface layer in the *Layer Name* field.

The *Layer Z-axis convention* field defines the Z-axis convention for the data in the new layer. The options are:

- Depth with Z-axis positive down: to be used if the data in the new layer will contain depth values that are positive down.
- Height with Z-axis positive up: to be used if the data in the new layer will contain elevation values that are positive up.
- Not applicable: to be used if the data in the new layer will represent generic, non-Z values (e.g., Std_Dev).

4. Select a Layer Z-axis convention from the drop-down list.

The Available Attributes list displays the attribute layers currently present in the selected surface.

5. Double-click an attribute in the *Available Attributes* list to insert it into the conditions field below.

If there are spaces in the name of the selected attribute, place quotation marks around the attribute in the conditions field, (e.g., “attribute name”).

Use your keyboard and the condition buttons to create the equation for the new layer. The example above shows the equation to create a new depth layer with values increased by 3 metres.

6. Click any of the operators to add it to the field.

Operator	Function	Example
( )	bracketed operation	(Depth+1) -
+	addition	Depth + 4
x	multiplication	Depth * 4
–	subtraction	Depth - 4
/	division	Depth / 4
^	exponentiation ($Depth^X$)	Depth ^ 4

To add values:

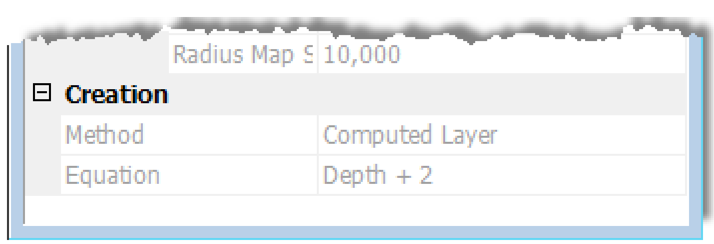
7. Click at the point in the equation where you want to enter a numerical value, and type the number.

Use Backspace to remove parts of an equation, or click **Clear** to remove everything from the conditions field.

8. Click **OK** to compute the new layer.

The new layer is drawn in the Display window and is listed as a child layer of the surface in the Layers window.

The properties of the new layer can be viewed in the *Creation* fields of the Properties.The *Method* field shows it is a computed layer, and the *Equation* field shows the variables in the equation.



Add to Surface

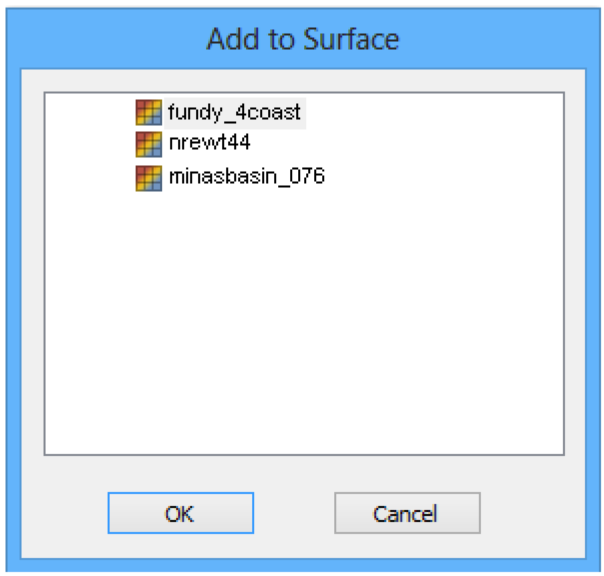
The Add to surface command applies new track lines to an existing surface without the need to regenerate the surface. You can add specific lines to the surface or you can let the program select the lines to be added automatically.

1. Select the track line layer in the Layers window.
2. Select the track line(s) you want to add to the surface so it is highlighted.

Menu	Tools >Surfaces > Add to
Tool	

3. Select the Add to Surface command.

The Select Surface dialog box is displayed.



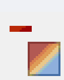
4. Select the surface to which the line is to be added.
5. Click **OK**.

The surface is updated to include new track lines.

Remove from a Surface

Remove a survey line from a Swath Angle, Shoalest Depth True Position or Uncertainty surface.

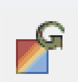
1. Select the survey line in the Project window.
2. Select a surface layer in the Layers window.

Menu	Tools >Surfaces > Remove From
Tool	

3. Select the Remove from surface command.

The surface is regenerated without the data from the selected survey line.

Recompute surface

Menu	Tools > Surfaces > Recompute
Tool	
Pop-up	Recompute

The Recompute function rebuilds a surface and regenerates the surface image.

If you have updated the surface since it was interpolated, you can apply Recompute to rebuild the interpolated surface.

1. Highlight the surface name in the Layers window.
2. Select the Recompute command.

Only the track lines used previously in the surface are processed during rebuilding.

The Recompute operation checks to see if the surface makes use of HIPS data that is no longer present.

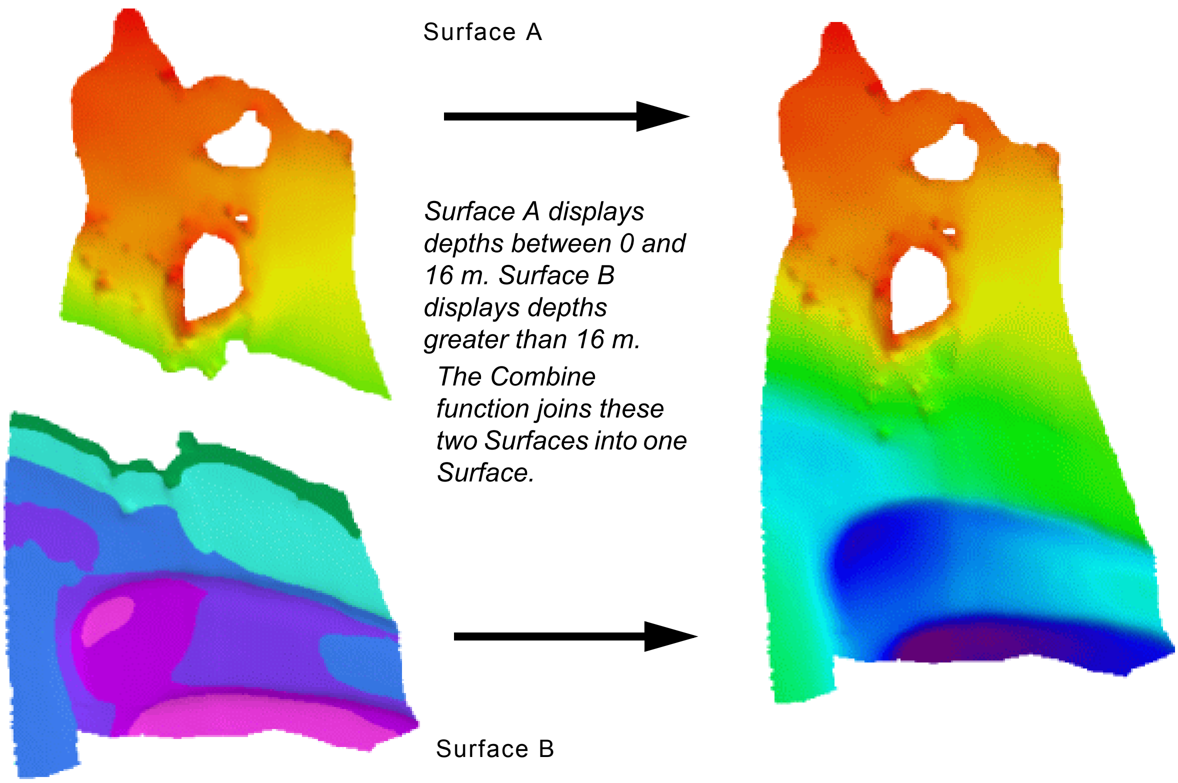
If data is missing, a dialog box lists what is missing. You then have the option of terminating the Recompute process, or continuing without the missing data. If you choose to continue, the replacement surface will have the missing data lines removed from its list of contributing lines.

When recomputing a CUBE surface, all CUBE surface editing, including nominations, hypothesis and node rejections, are considered no longer valid and will thus not be retained after re-computation of the surface.

Combine

The Combine tool stitches two or more surfaces together to form one surface. This option can be used when you have created finalized surfaces with different depth thresholds (see “Finalize a Surface” on page 270), or when you have two adjacent surfaces with different resolutions, or to combine data from overlapping surfaces into one continuous surface.

In areas where two or more source surfaces overlap, Combine will use the shoalest value in the resulting surface.

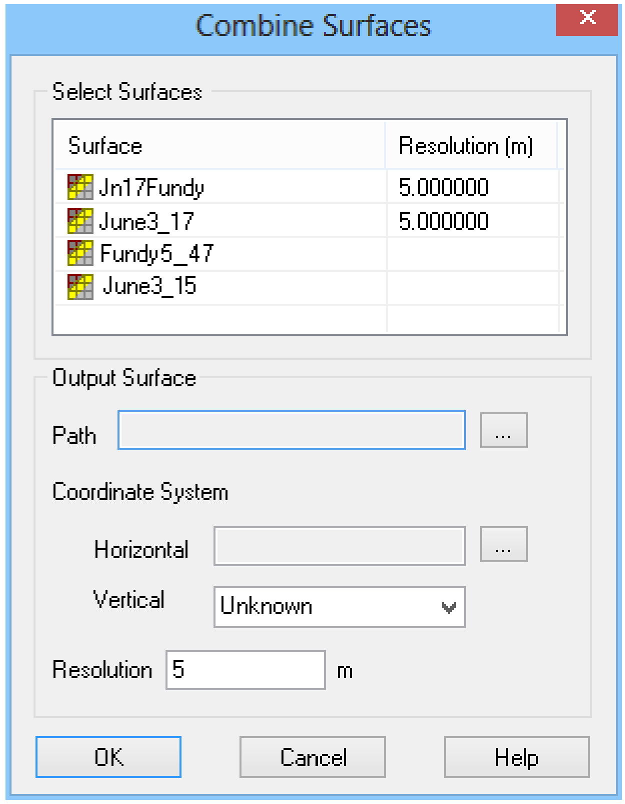


To combine two surfaces:

Menu	Tools > Surfaces > Combine

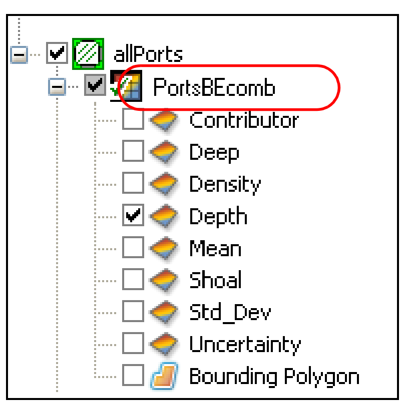
1. Select the Combine command.

The Combine Surfaces dialog box is displayed.



2. Select the surfaces to combine from the list in the Select Surfaces field.
3. In the *Path* field, click **Browse** to name and save the combined surface.
4. Click **Save**.
5. Type a resolution for the combined surface.
6. Click **OK**.

A new surface is generated that combines the selected surfaces. The new combined surface is listed in the Layers window.



The combined surface reproduces all the attribute layers contained in the source surfaces.

Difference Surfaces

Create a surface showing the differences in attribute values between two surfaces.

The Difference Surface function is useful for comparing changes to an area. A surveyed area can change considerably over time due to the redistribution of sediment with the currents. This redistribution can cause changes in depths. A difference surface can be used to find these changes by comparing two surfaces for the same area that were created at different times.

To use this function, the surfaces to be compared must cover part or all of the same area, and the data in the attribute layers being compared must be of the same type.

Data type

The data types include:

Type	Size/Format	Range
Short	16-bit (2-byte), integer	-32,768 to 32,767 (no decimal points)
Integer	32-bit (4-byte), integer	-2,147,483,648 to 2,147,483,647 (no decimal point)
Float	single-precision, IEEE 754 32-bit (4-byte), floating-point	1E-44 to 3.4E+38 (positive or negative)
Double	double-precision, IEEE 64-bit (8-byte), floating-point	negative values: -1.79769313486231570E+308 to -4.94065645841246544E-324 positive values: 4.94065645841246544E-324 to 1.79769313486231570E+308

For example, Depth layers can be compared, but you can't compare a Density layer to a Depth layer, since the former contains Integer data and the latter is Float.

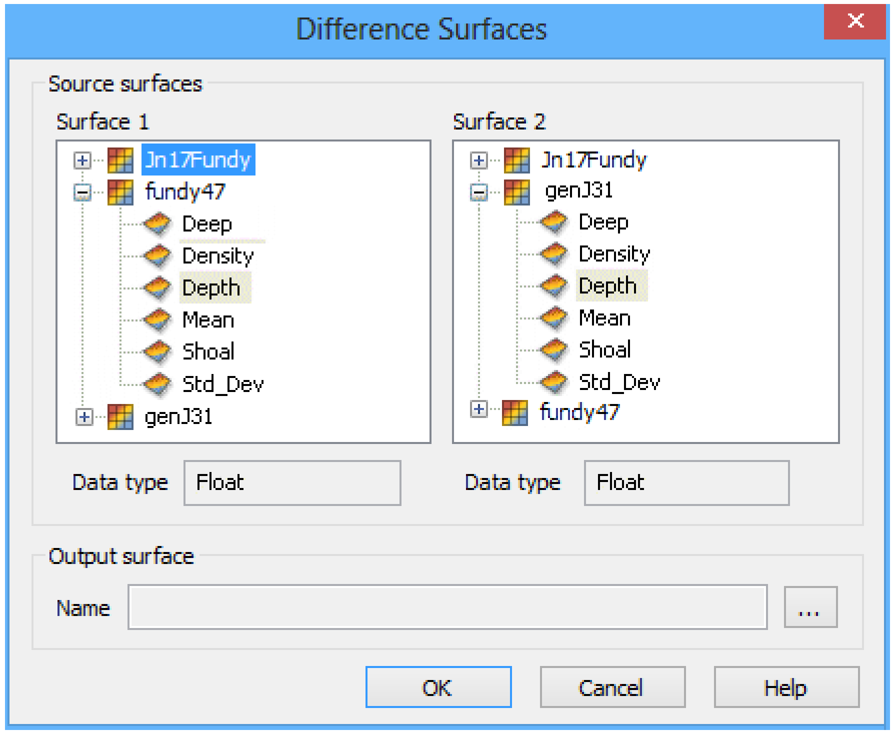
When you select a layer in the Difference Surface dialog box, its data type is displayed in the *Data Type* field.

To compare two open surfaces:

1. Select a surface or attribute layer in the Layers window.
2. Select the Difference Surfaces command.

Menu	Tools > Surfaces > Difference

The Difference Surface dialog box is displayed.



Any open surfaces are listed in the Surface 1 and 2 fields.

3. Click the expand (+) icon beside a layer in the *Surface 1* list to view the available attribute layers.
4. Select an attribute layer to use as the data source for the difference calculation.

The *Data Type* field displays the data type of the selected layer.

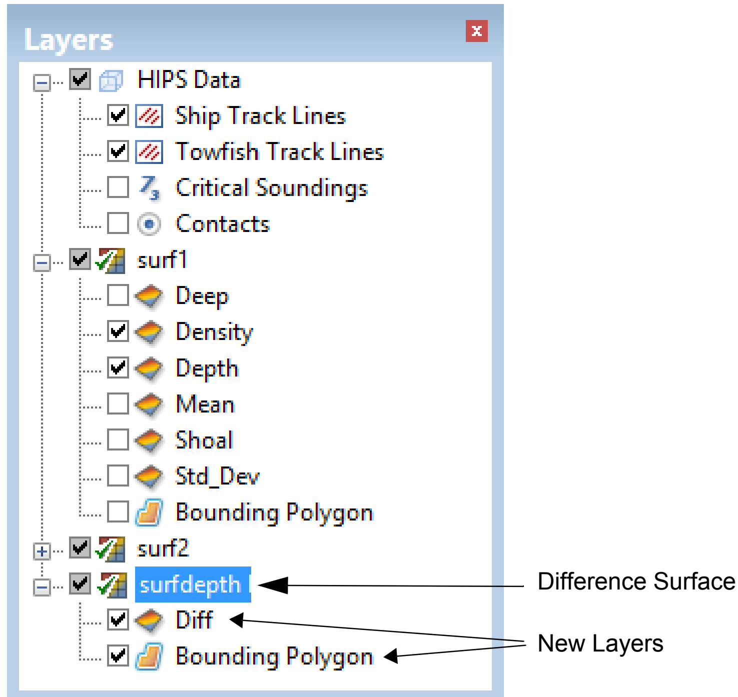
5. Expand the *Surface 2* list and select an attribute layer with the same data type as the layer selected for *Surface 1*.

Values in the Data type fields must match.

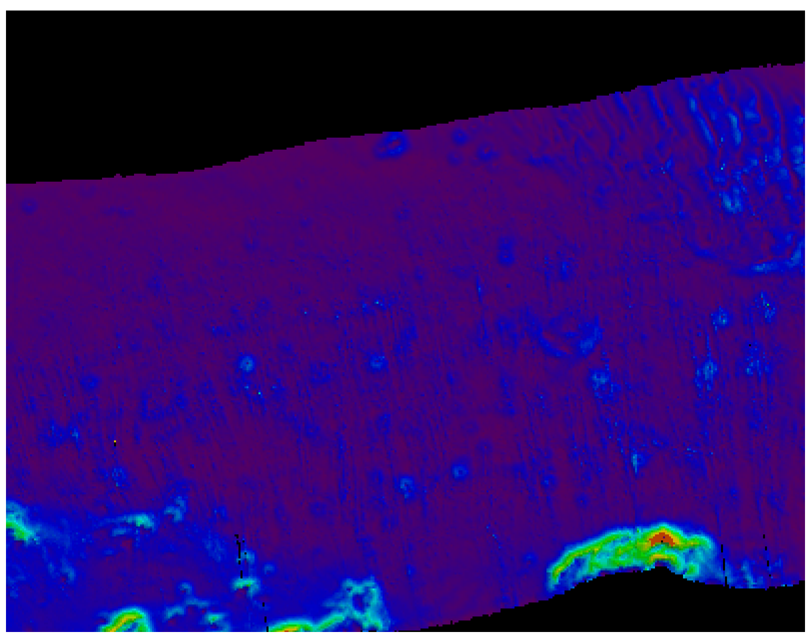
The *Data Type* field displays the data type of the second selected surface attribute.

6. Click the Browse button (**...**) to define the name and location for the new surface file, which is saved in CSAR format.
7. Click **OK**.

A new surface is generated, with an attribute layer named Diff and a Bounding Polygon layer. The resolution of the new surface will be the same as the resolution of the surface selected as Surface 1.



An example of a Difference Surface is shown below.



The colours represent the range of discrepancies between the two compared surfaces. The dark areas of this surface represent greater depth differences than the light areas. If the contrast is not clear, you can reverse the colours, or change the colour map settings or set a colour range, in the Properties window. (See "Colour Map Editor" on page 44 and "Colour Range Editor" on page 50 of the Tools guide.

Extract Surface

Extract a portion of a surface or point cloud to create a new surface with the selected attributes. The extraction can be done by digitizing a line around the desired area, or by applying a feature layer containing coverage features for the desired area(s).

"Extract Surface by Digitizing" on page 260

"Extract Surface Using a Feature Layer" on page 263

Extract Surface by Digitizing

The Extract by Digitizing command is used to create a new surface by copying a section of an existing surface. You identify the section that you want to copy by digitizing a line around the data. A copy of that section will be saved as a new surface, using the attribute values you selected from the existing surface. The resulting surface will have the extents of the digitized section.

To extract a surface by digitizing:

1. Select a surface or a point cloud in the Layers window.
2. Select the (Extract) By Digitizing command.

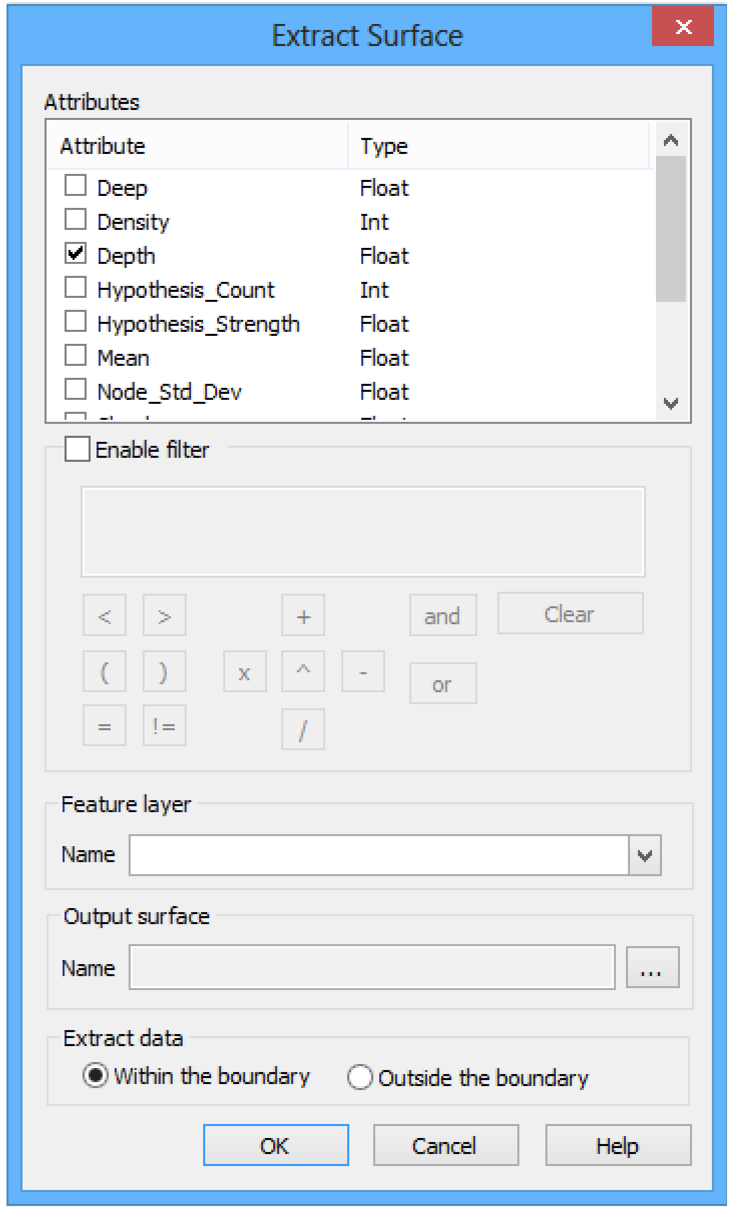
Menu	Tools > Surfaces > Extract > By Digitizing
Pop-up	Extract (root layer)

The cursor changes to indicate you are in digitize mode.

To digitize a line, you must place vertices around the area you want to select. To place the vertices, click the display in the necessary locations. Each time you click a vertex will be added to the display and a line will be drawn between the vertices. Point-to-Point line is the default digitizing tool, but there other digitizing modes available. For more information, see "Digitizing Features" on page 572.

3. Digitize a line around the section to extract from the existing surface.
4. To finish the line, right-click and, from the pop-up menu, select Area Digitizer > Close Area.

The Extract Surface dialog box is displayed with a list of attributes associated with the selected surface. These attributes can be saved with the extracted area. When finished, the values of the attributes selected from the existing surface will be present in the extracted surface.



5. Click the check boxes to select the *Available Attributes* you want to include in the extracted surface.

The primary Z layer (Depth, Height, etc.) is mandatory and cannot be turned off. Also, any designated points within the selected area will automatically be included in the extraction.

You may choose to use filters to extract only specific data from the selected area. When the *Enable Filter* option is selected, you set conditions for the filter and only the nodes within the selected area that meet the conditions will be extracted.

For example, with the depth units set to metres, if you only want nodes with a Depth value of less than 8 metres, you would:

- Double-click *Depth* in the *Available Attributes* list to add it to the filter.
- Use the keyboard and/or filter buttons to add " < 8" after the attribute (including the spaces).

The resulting filter displayed in the field will be "Depth < 8".

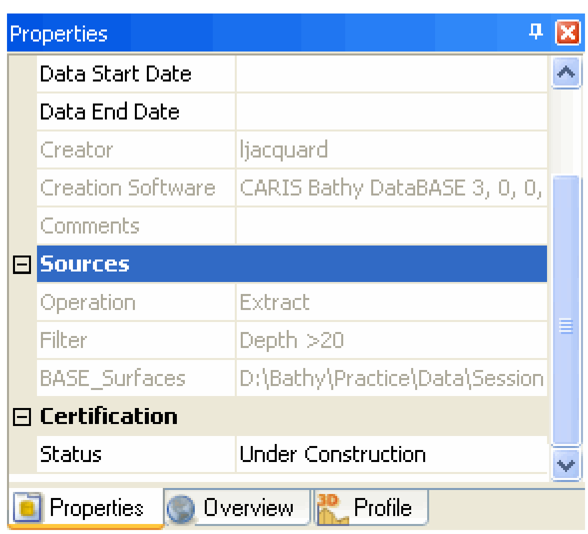
6. [Optional] To apply filters to the surface data, select the *Enable Filter* option and enter the filter conditions.
7. Click Browse (**...**) to locate a path and specify a file name for the *Output Surface.*

The *Extract data* option enables you to either include or exclude the surface data within the digitized area. If you were to select *Within the boundary*, only data within the digitized line would be extracted. If you were to select *Outside the boundary*, only data outside the digitized line would be extracted, resulting in a surface with a hole in it.

8. Select an *Extract data* option.
9. Click **OK**.

The selection is extracted to a new surface.

In the Properties window you can see the source data from which the surface was extracted, the operation that created the surface, and any filters that may have been applied during extraction, as well as any other properties that were populated in the source surface.



Extract Surface Using a Feature Layer

The Extract Using Feature Layer command is used to create a new surface by copying one or more sections of an existing surface. The sections to be copied are identified by coverage area features in a feature layer associated with the data. These features can be boundary type or hole type. Boundary type suggests that there is data in the area. Hole type suggests that there is no data in the area, or that the data in the area of the hole is not valid.

The copied sections will be saved as new surfaces, using the attribute values you selected from the existing surface. The resulting surfaces will include the extents of each coverage area in the feature layer, including disjointed areas. If there are any hole type coverage areas, they will be reflected in the new surfaces.

You can use an existing feature layer and objects, or create a new layer and cvrage objects specifically for the extraction. See "Create a Feature Layer" on page 565 for information on creating a feature layer. See "Create Features" on page 567 for information on creating features.

> Points with a status value of Rejected are not extracted.

To extract a surface using a feature layer:

1. Open a feature layer containing `cvrage` features, or create a feature layer and add `cvrage` features.
2. Select a surface or point cloud in the Layers window.
3. Select the (Extract) Using Feature Layer command.

Menu	Tools > Surfaces > Extract > Using Feature Layer

The Extract Surface dialog box is displayed.

The attributes from the existing surface can be saved with the extracted area. When finished, the values of the attributes selected from the existing surface will be present in the extracted surface.

4. Click the check boxes to select the *Available Attributes* you want to include in the extracted surface.

> The primary Z layer (Depth, Height, etc.) is mandatory and cannot be turned off. Also, any designated points within the extents of the feature layer will automatically be included in the extraction.

You may choose to use filters to extract only specific data from the selected area(s). When the *Enable Filter* option is selected, you set conditions for the filter and only the nodes within the selected area(s) that meet the conditions will be extracted.

For example, with the depth units set to *Metres*, if you only want nodes with a Depth value of less than 8 metres, you would:

- Double-click *Depth* in the *Available Attributes* list to add it to the filter.
- Use the keyboard and/or filter buttons to add " < 8" after the attribute (including the spaces).

The resulting filter displayed in the field will be "Depth < 8".

5. [Optional] To apply filters to the surface data, select the *Enable Filter* option and enter the filter conditions.

The Feature Layer field provides a list of all feature layers currently available.

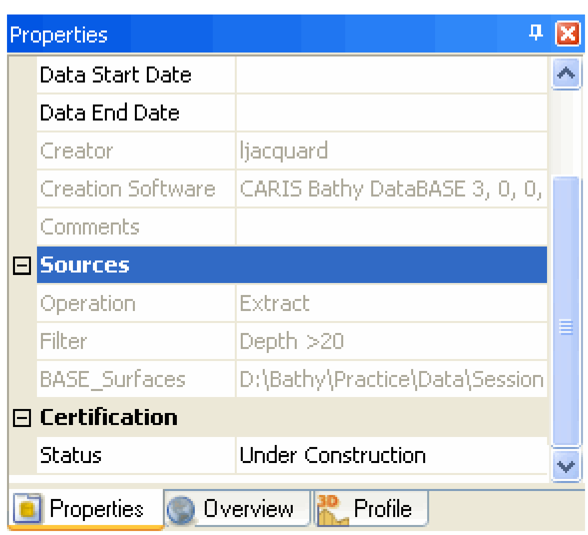
6. Select the *Feature Layer* containing the cvrage features you want to use to define the extents of the extraction.
7. Click Browse (**...**) to specify a path and file name for the *Output Surface*.

The *Extract data* option enables you to either include or exclude the surface data within the `cvrage` features. If you were to select *Within the boundary,* only data within the boundary type objects would be extracted and data within the hole type objects would be removed. If you were to select *Outside the boundary*, only data within the hole type objects would be included and data within the boundary type objects would be discarded.

8. Select an *Extract data* option.
9. Click **OK**.

The data covered by the feature objects is extracted to a new surface.

In the Properties window you can see the source data of the extracted surface, the operation that created the surface, and any filters that may have been applied during extraction, as well as any other properties that were populated for the source surface.



Interpolate

The Interpolate commands fill empty nodes in a surface grid using values from surrounding nodes that contain data.

The Interpolate function attempts to fill all holes in a selected surface. The process begins by classifying all nodes in the surface as “holes” (gaps in data), “data” (areas contain data) or “no-data” (areas outside of the surface area). These classifications are used to determine the locations of holes that need to be interpolated. See “Identify Holidays” on page 267 for more information on this process.

A new “holiday” layer is added to the surface to contain the classification information. The new layer name uses the name of interpolated attribute layer and adds the word “Holiday” as, for example, Depth and Holidays.

> If the surface being interpolated already contains a Holidays layer, the existing layer will be used and will be updated to reflect the newly interpolated nodes.

Then the populated nodes surrounding holes are analysed to determine if there are enough neighbouring values to interpolate a value for the hole based on the user-specified criteria. If sufficient data is present, the process runs as a single iteration, filling all holes. Holes are filled as far as possible from the outer edge to the centre, given the populated data and specified criteria.

Large holes may be only partially filled as there may not be sufficient neighbouring data to completely fill a hole. The Interpolate command can be run multiple times, however, to continue to fill holes using a single iteration each time. Each successive run will use the interpolated values from the previous run as populated neighbouring nodes.

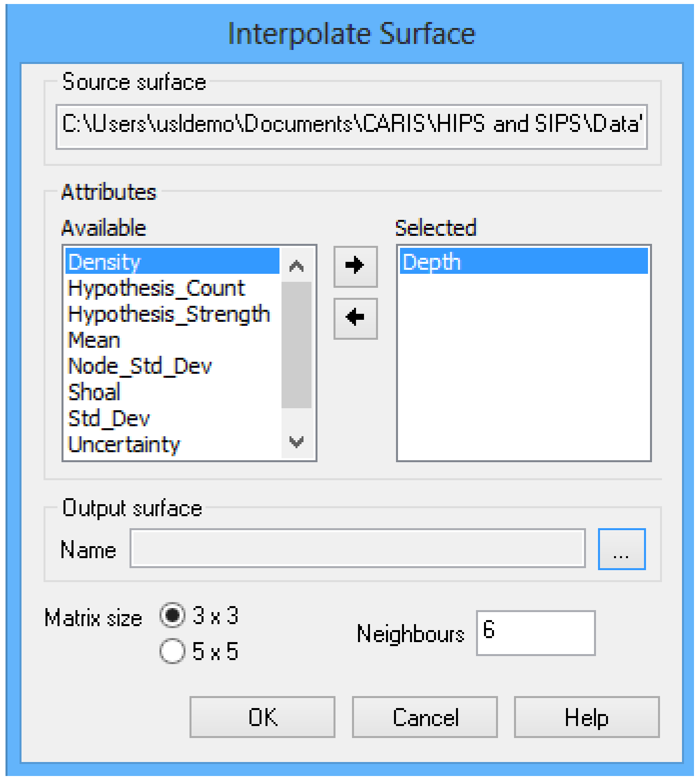
The original surface is not affected by this command. A copy of the selected surface is created and the interpolation is performed on that data.

To interpolate a surface:

1. Select a surface layer in the Layers window.
2. Select the Interpolate command.

Menu	Tools > Surfaces> Interpolate > Entire Dataset
Popup	Interpolate (in Layers window)

The Interpolate Surface dialog box is displayed.



The *Source surface* field displays the path to and name of the selected surface. This cannot be changed.

The surface attribute layers are listed in the *Attributes* fields. The primary Z attribute layer is displayed in the *Selected* field. This cannot be removed from the list.

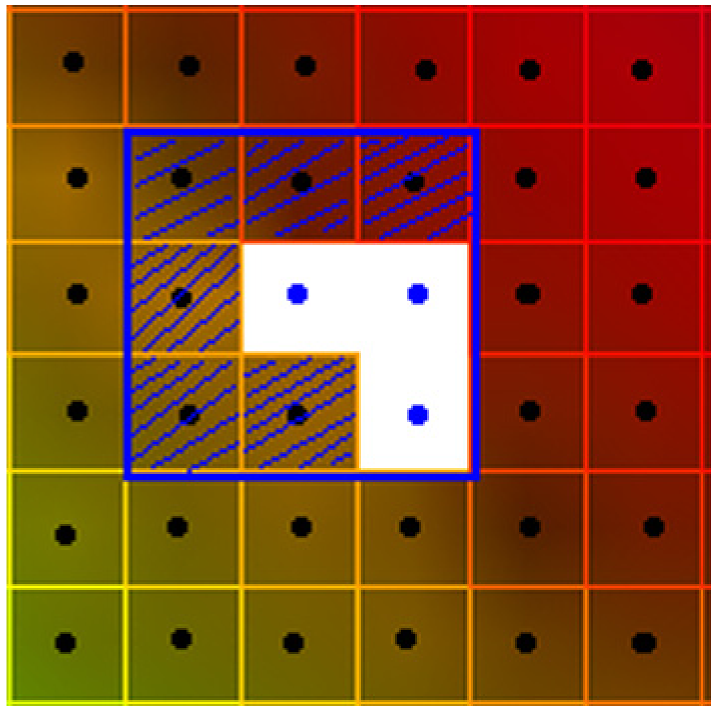
3. From the *Available* list, select each attribute you want to interpolate.
4. Click the right arrow to move attributes to the *Selected* list.
5. Click Browse in the *Output Surface* field, and type a name for the interpolated surface.

The *Matrix Size* is the number of nodes surrounding a no-data hole. The size is either 3x3 or 5x5, meaning the application will analyze a square area around each pixel of the hole that is either 3 pixels high and 3 pixels wide, or 5 pixels high and 5 pixels wide.

6. Select a *Matrix size* option.

Neighbours are the number of pixels within the matrix area that must contain data in order for the hole to be interpolated. For example, if you select the 3x3 matrix size option and enter a neighbours value of 6, the area must have at least 6 populated data nodes in order for a pixel in the hole to be interpolated.

In the example below, the blue square represents the matrix area. Each dot within the square is a node. Of those nodes, only the 6 shaded nodes contain data. For the criteria specified, this would be sufficient to interpolate a value for the pixel at the centre of the square. If the square were shifted to centre on each pixel in the hole, you can see the entire hole could be interpolated using the specified criteria.



7. Type the number of neighbouring nodes used to interpolate pixels.
8. Click **OK**.

The interpolated Surface is created and is displayed in the data tree in the Layers window.

The *Operation* field under Sources in the Properties for the new surface will confirm that the surface was created by Interpolation.

> An interpolated surface cannot be updated using the *Automatic Surface Update* setting in the Options dialog box.

Identify Holidays

The Identify Holidays tool enables you to determine where there are gaps in data which need to be addressed. Identify Holidays analyzes raster surfaces to classify each pixel node as a “hole”, as “data” or as “no-data”. This information can be used to interpolate the holes in the surface. See “Interpolate” on page 265.

A numerical value is assigned to each node to represent its classification. These classification values are:

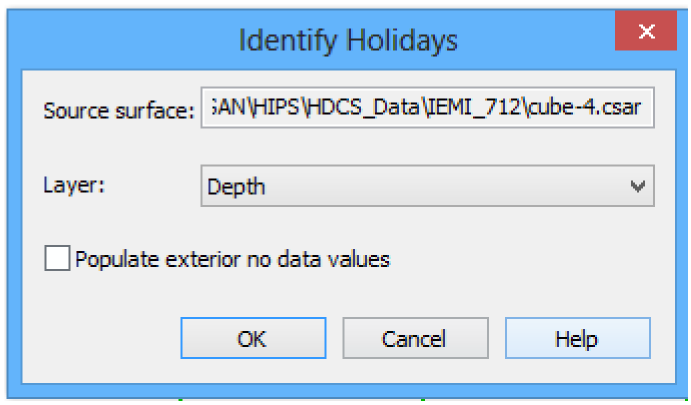
- 0: assigned to exterior, non-data values. This includes any area within the extents of the surface, but beyond the actual data of the surface.
- 1: assigned to valid data values (areas containing data).
- 2 and up: assigned to holes (gaps in data). Each hole is uniquely labelled, starting with 2 and incrementing by one for each hole until the maximum is reached for the 32bit Integer data type range. See "Data type" on page 257.

To identify holidays in a surface:

1. Select the surface in the Layers window.
2. Select the Identify Holidays command.

Menu	Tools > Layers > Identify Holidays
Pop-up	Identify Holidays (Layers window)

The Identify Holidays dialog box is displayed.



The *Source surface* field displays the name and location of the surface selected in the Layers window. This field is read-only and cannot be used to change the source surface.

The *Layer* field determines which layer of the source surface will be classified. If an attribute layer was selected in the Layers window when the tool was opened, this layer is automatically selected. To select another layer:

3. Select a source layer from the *Layer* drop-down list.

By default, classifications are only assigned to holes and valid surface data. Setting the *Populate exterior no data values* option will also classify the area outside of the surface data but within the extents of the surface.

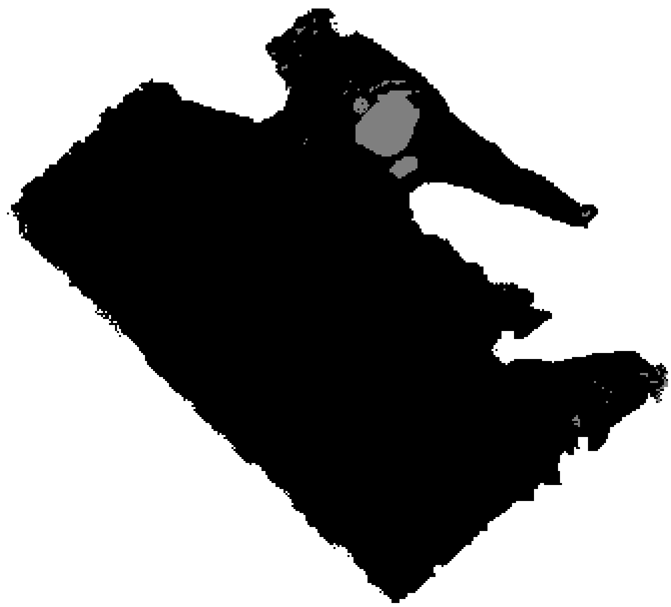
4. [Optional] Click the check box to enable the *Populate exterior no data values* option.
5. Click **OK**.

A new layer is added to the surface. All holiday layers are named according to the layer that was classified and appended with the word "Holidays". For example, if the Depth layer of a surface were selected, the resulting layer would be called "Depth and Holidays".

6. Select the layer in the Layers window to view the classifications in the Display window.

By default, the holidays layer is displayed in the Rainbow colour map. You can select another colour map file from the Properties for the holidays layer.

There is also a colour range file called Holidays available for displaying holiday data. Selecting this colour file in the Properties will colour data nodes black, exterior no-data nodes white and holes grey, as in the example below.



Finalize a Surface

A finalized surface is a finished version of the surface that is ready for export or for further processing (for example, to add contours). Most importantly, finalizing a surface ensures that designated soundings are included in the final surface, to be carried through to bathymetric products.

These three operations can be applied when finalizing a surface:

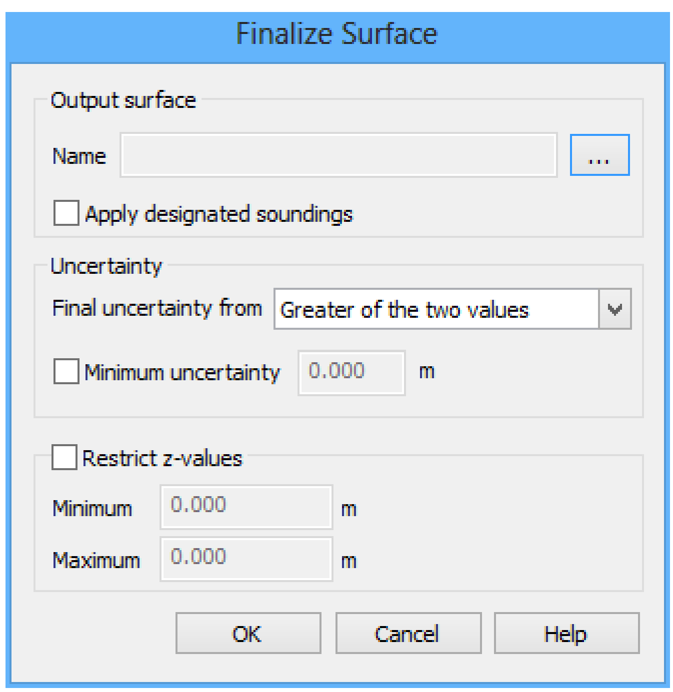
- **Minimum uncertainty**: Vertical uncertainty can fall to almost zero if a node contains too many soundings. This function applies a user-defined vertical uncertainty to each node to compensate for this situation.
- **Designated soundings**: If the data set contains designated soundings, the surface is regenerated and the designated soundings' depth values are applied to the nearest nodes. (See "Critical Soundings" on page 337).
- **Depth thresholds**: A finalized surface can be modified to represent a range of depths levels. When the finalized surface is generated, only these depths are displayed.

To create a finalized surface:

1. Select a surface layer in the Layers window.
2. Select the Finalize command.

Menu	Tools > Surfaces > Finalize...

The Finalize Surface dialog box is displayed.



1. Click Browse to name and save the finalized surface.

2. Select the *Apply Designated Soundings* check box to ensure that designated soundings are retained in the resulting surface,
   - If this option remains un-selected, designated soundings are processed no differently from other soundings.

Some surfaces have calculations for approximated values. For these surfaces, options from the Uncertainty section may be available. The *Final uncertainty from* drop-down list gives three options as the source of values for calculating the uncertainty of the finalized surface:

- Uncertainty: the vertical uncertainty values from the Uncertainty layer of the source surface.
- Std Dev.(scaled to 95% CI): the standard deviation values from the source surface (scaled to 95% confidence interval).
- Greater of the two values: the larger of the two above values.

3. Select a final uncertainty option from the drop-down list.
4. Select the *Minimum Uncertainty* check box and type a depth value to represent the minimum vertical uncertainty value.

The Restrict Z-values fields enable you to limit the nodes that will be included in the finalized surface by specifying a range of values to include. Only nodes that fall within this range will be included in the resulting surface.

5. Select the *Restrict Z-values* check box to create a finalized surface that displays only node depths between the set range.
6. Type minimum and maximum depth values in their respective fields.
7. Click **OK**.

A finalized surface is generated and visible in the Display window, and listed in the Layers window.

Vertical Shift

Apply a vertical shift to one or more layers (Depth, Height, etc.) of an existing surface or point cloud, to create an output surface with an increased or decreased Z value. This function is not only applied to update data, it may also be used to apply tide.

There are three shift types available:

- Single Shift: shift the entire surface by a single value, see "Shift Using a Single Value" on page 272
- ASCII/BIN Shift: apply a file containing a grid of sounding datum heights, see "ASCII File Shift" on page 275 or "BIN File Shift" on page 276
- Tide Correction: apply tide correction using data from a single station or multiple stations, see "Shift Using Tide Correction Data" on page 279

Shift Using a Single Value

A surface can be shifted by a single value. If you select this option, the value you enter in the *Shift value* field is used to shift the selected layers. The direction of the shift depends on the current application settings for the Z-axis Convention attribute.

For example, if the elevation is 12 metres below the datum and a shift of (positive) 10 metres is applied:

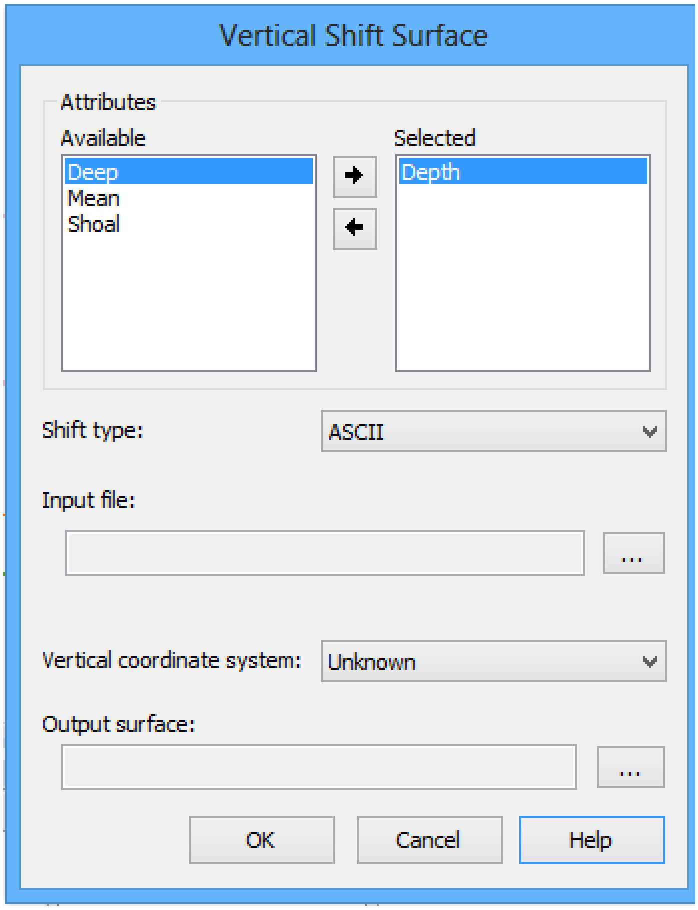
- if the Z-axis Convention is set to "Down is positive," the result is 22 metres below the datum.
- if the Z-axis Convention is set to "Up is positive," the result is 2 metres below the datum.

To apply a vertical shift using a single value:

Menu	Tools > Surfaces > Vertical Shift
Pop-up	Vertical Shift (root layer)

1. Select a surface root layer in the Layers window.
2. Select the **Vertical Shift** command.

The Vertical Shift Surface dialog box is displayed.



In this dialog box, you define the attribute layers you want shifted and the amount by which to shift them. The resulting surface will contain all of the same layers as the source surface, but only those selected for shift will have their values shifted.

Any layer with a Z value can be shifted.

> If the source surface contains a computed layer, that layer will lose its dynamic status and will no longer be affected by changes to the primary elevation layer.

3. To select a layer, click the layer name in the *Available* list, then click the Add (**right-arrow**) button to move it into the *Selected* list.
4. [Optional] Repeat step 3 for each layer you want to add.
5. [Optional] You can also remove layers from the *Selected* list by clicking the layer name in the list, then clicking the Remove (**left-arrow**) button to return the layer to the *Available* list.

> The primary elevation layer is required and cannot be removed from the *Selected* list.

6. Select *Single Shift* from the *Shift type* drop-down list.
7. Type a value greater than zero in the *Shift value* field.

You have the option of selecting the vertical reference system that will appear in the CSAR metadata for the resulting surface. The options available in the list are controlled by the Vertical Reference System database. This list can be edited using the Reference System Editor tool.

8. Select a *Vertical Datum* option from the drop-down list.
9. To choose the name and location of the resulting surface, click the *Output surface* Browse (**...**) button.

A Save As dialog box is displayed.

10. Select a path and type a name for the shifted surface, then click **Save**.
11. To finish, click **OK**.

A new surface is created and displayed in the Layers window. The attributes and properties from the source surface are carried over to the shifted surface. Additional properties are also populated under the *Sources* group.

- *Operation:* This property identifies the tool that was used to create the surface. For a shift, it will read "Vertical Shift".
- *Vertical Coordinate System:* The vertical coordinate system that was used to create the shifted surface.
- *Additional Attributes:* This property identifies the attributes from the source surface that were shifted.
- *Shift Method:* This property identifies the type of shift that was applied to the surface.
- *Shift Value:* This property differs depending on the type of shift applied. For a single-value shift, it identifies the value by which the surface was shifted.
- *Original Depth Update:* This property states whether or not the original depth value of any designated soundings was also updated by the shift. This property is only present if the source surface contained designated soundings.
- *Source:* This property identifies the name and location of the source surface.

The Z values in the selected layers should be shifted according to the selected method. To confirm that the shift was successful, compare the data in the new surface to the data in the original surface. All elevations should be adjusted by the specified amount.

Shift Using a File of Predefined Values

If you are using a file of predefined values, you can choose between an ASCII file or a BIN file.

ASCII File Shift

The ASCII file option uses a file containing points that represent a boundary around the entire surface, or the section of the surface you wish to shift. Using this boundary, a TIN will be calculated according to the values in the ASCII file. These values will then be applied to the values in the area of the surface encompassed by the boundary.

If using an ASCII datum model file, the file must follow a specific format:

- The data must be in comma delimited format.
- The coordinate type must be Geographic and the format must be decimal degrees (Latitude, Longitude, Z).
- The hyphen symbol (-) must be used to designate coordinates in the Southern or Western hemispheres.

An example of an ASCII shift file is shown below:

```
-33.848326,151.192435,10.4
-33.849484,151.192370,10.1
-33.849237,151.193173,9.7
-33.849506,151.194227,9.8
-33.848298,151.193916,10.0
-33.848402,151.193149,10.2
```

Values in the ASCII file are always Z positive down by design. If your ASCII file contains negative values, this will be reflected in the resulting shift values. For example, a file with a value of -10 applied to a surface that is positive down with a depth of 20m, would result in a depth of 10m. If you were to apply that same file to a surface that is positive up, the result would be a height of 30m.

Using the example above, the following table shows what the resulting values would be with a given set of original values:

Original Value	Shift Value	Resulting Value
8.2	10.4	18.6
8.6	10.1	18.7
7.3	9.7	17.0
7.4	9.8	17.1
9.1	10.0	19.1
9.2	10.2	19.4

BIN File Shift

The BIN format option used for vertical datum transformations is based on the US NGS format for GEOID99, published here: http://www.ngs.noaa.gov/GEOID/GEOID99/.

The BIN format is described here in the Frequently Asked Questions section of the GEOID99 web site: http://www.ngs.noaa.gov/GEOID/USGG2009/faq_2009.shtml.

A BIN file consists of a 44-byte header followed by "nla" rows of data, each row being "nlo" elements long and each element being a 4-byte floating point number. This format is known in FORTRAN lingo as "direct access binary".

The exact ordering of the bytes is shown below:[1]

Bytes	Data Type	Variab Name	Variable Description
1-8	real*8	glamn	Southernmost Latitude of grid (decimal degrees)
9-16	real*8	glomn	Westernmost Longitude of grid (decimal degrees)
17-24	real*8	dla	Latitude spacing of grid (decimal degrees)
25-32	real*8	dlo	Longitude spacing of grid (decimal degrees)
33-36	int*4	nla	Number of rows in grid
37-40	int*4	nlo	Number of columns in grid
41-44	int*4	ikind	Set to "1", meaning the gridded data is "real*4"
45-48	real*4	data (1,1)	Gridded value at element 1,1 (Southwest corner)
. . .			

The remainder of the file continues as 4-byte real values. The south row is filled in first, with data(1,nlo) being the last variable in the south row, and then proceeds northward for each consecutive row.

> Note: Values in a BIN file are always Z positive up by design.

The total number of bytes in a BIN file is found using the formula: *44 + 4*nla*nlo*.

The actual numbers of rows (nla) and columns (nlo) for each file is the same within a region, but varies between regions, as shown below:[2]

REGION	ROWS (nla)	COLUMNS (nlo)
Conterminous United States	1081	1141
Alaska	721	1921
Hawaii	361	421
Puerto Rico and the Virgin Islands	361	301

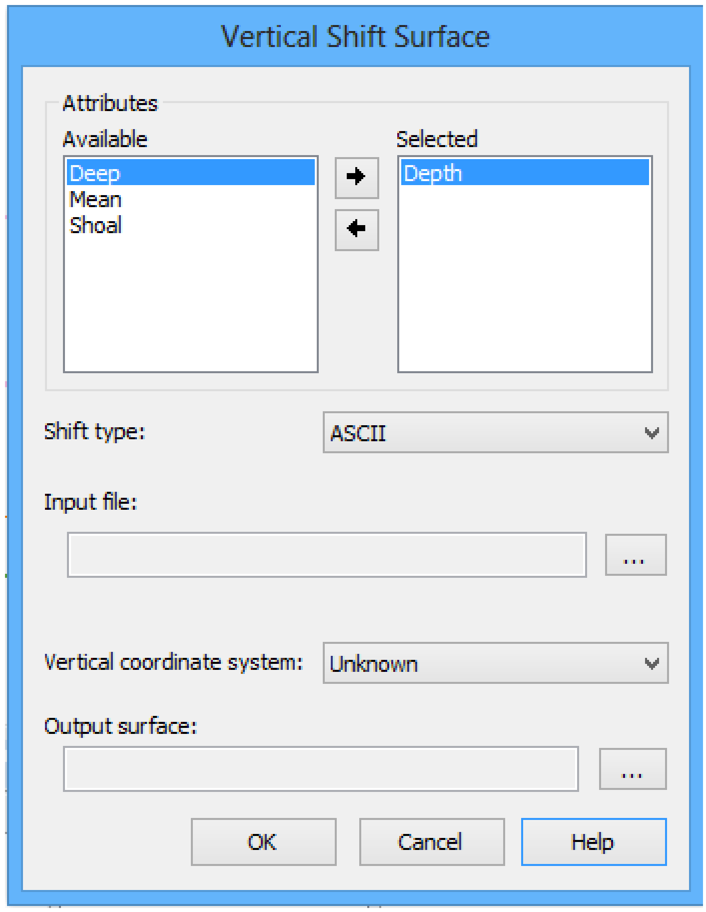
1. Use of material from NOAA's National Geodetic Survey (NGS) is for descriptive purposes only and does not imply endorsement by NGS of CARIS products or services.
2. Use of material from NOAA's National Geodetic Survey (NGS) is for descriptive purposes only and does not imply endorsement by NGS of CARIS products or services.

Menu	Tools > Surfaces > Vertical Shift
Pop-up	Vertical Shift (root layer)

To apply a vertical shift using an ASCII or BIN file:

1. Select a surface root layer in the Layers window.
2. Select the **Vertical Shift** command.

The Vertical Shift Surface dialog box is displayed.



In this dialog box, you define the attribute layers you want shifted and the amount by which to shift them. The resulting surface will contain all of the same layers as the source surface, but only those selected for shift will have their values shifted.

Any layer with a Z value can be shifted.

If the source surface contains a computed layer, that layer will lose its dynamic status and will no longer be affected by changes to the primary elevation layer.

3. To select a layer, click the layer name in the *Available* list, then click the Add (**right-arrow**) button to move it into the *Selected* list.
4. [Optional] Repeat step 3 for each layer you want to add.

5. [Optional] You can also remove layers from the *Selected* list by clicking the layer name in the list, then clicking the Remove (**left-arrow**) button to return the layer to the *Available* list.

The primary elevation layer is required and cannot be removed from the *Selected* list.

6. Select either *ASCII file* or *BIN file* from the *Shift type* drop-down list, depending on the type of file being used.
7. Click the *Input file* Browse (**...**) button and select an available file for the selected shift type and click **Open**.

The selected file is displayed in the *Input file* field.

You have the option of selecting the vertical reference system that will appear in the CSAR metadata for the resulting surface created for the shift. The options available in the list are controlled by the Vertical Reference System database. This list can be edited using the Reference System Editor tool. For more information, see "VERTICAL REFERENCE SYSTEM" ON PAGE 231.

8. Select a *Vertical Datum* option from the list.
9. To choose the name and location of the resulting surface, click the *Output surface* Browse (**...**) button.

A Save As dialog box is displayed.

10. Select a path and type a name for the shifted surface, then click **Save**.
11. To finish, click **OK**.

A new surface is created and displayed in the Layers window.

The attributes and properties from the source surface are carried over to the shifted surface. Additional properties are also populated under the *Sources* group.

- *Operation:* This property identifies the tool that was used to create the surface. For a shift, it will read "Vertical Shift".
- *Vertical Coordinate System:* The vertical coordinate system that was used to create the shifted surface.
- *Additional Attributes:* This property identifies the attributes from the source surface that were shifted.
- *Shift Method:* This property identifies the type of shift that was applied to the surface.
- *Shift File:* This property differs depending on the type of shift applied. For ASCII and BIN shifts, it identifies the file of depth values that was used to shift the surface.
- *Original Depth Update:* This property states whether or not the original depth value of any designated soundings was also updated by the shift. This property is only present if the source surface contained designated soundings.

- *Source:* This property identifies the name and location of the source surface.

The Z values in the selected layers should be shifted according to the selected method. To confirm that the shift was successful, compare the Z values at various coordinates in the new surface to the values in the selected datum file. The elevation values should match.

Shift Using Tide Correction Data

Tide data is used to generate final depths relative to tide datum by subtracting the tide from the sounding depth. This correction provides a more accurate depth value for the surface. When available, tide correction should be applied to all Surfaces before using them to create charts.

You can select to load observations from one station or many tide stations. The single station option (*Single tide station observations*) would be used if your surface falls entirely within a single tide zone. The multiple stations option (*Multiple tide station observations*) is used if your surface spans multiple tide zones.

Tide zones are closed polygons with tide, time and range corrections for a primary station, plus up to three backup secondary tide stations.

Observations are loaded from a tide file (.tid) or from a zone definition file (.zdf).

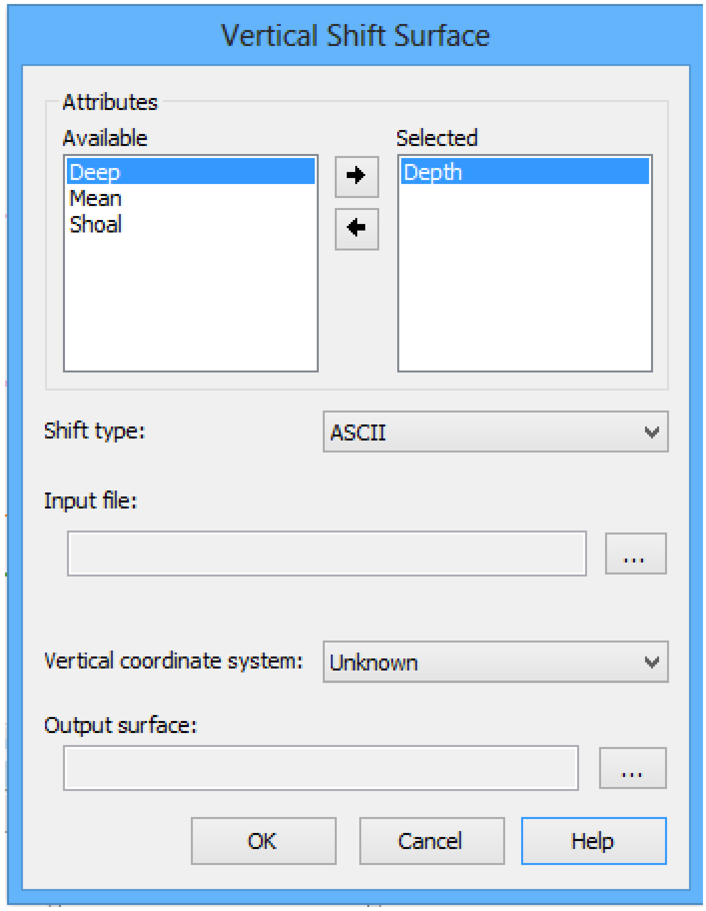
- Tide files contain observations from a single tide station. Applying this file will apply tide observations for the surface for the specified time period. Tide files can be created or edited in a text editor.
- Zone definition files contain data that defines tide zones. Applying this file tells the application where the zones are and in turn, which observations to apply for your surface. Zone definition files are in ASCII format and can be edited in a text editor.

To apply a vertical shift using tide correction data:

Menu	Tools > Surfaces > Vertical Shift
Pop-up	Vertical Shift (root layer)

1. Select a surface root layer in the Layers window.
2. Select the **Vertical Shift** command.

The Vertical Shift Surface dialog box is displayed.



In this dialog box, you define the attribute layers you want shifted and the amount by which to shift them. The resulting surface will contain all of the same layers as the source surface, but only those selected for shift will have their values shifted.

Any layer with a Z value can be shifted.

If the source surface contains a computed layer, that layer will lose its dynamic status and will no longer be affected by changes to the primary elevation layer.

3. To select a layer, click the layer name in the *Available* list, then click the Add (**right-arrow**) button to move it into the *Selected* list.
4. [Optional] Repeat step 3 for each layer you want to add.
5. [Optional] You can also remove layers from the *Selected* list by clicking the layer name in the list, then clicking the Remove (**left-arrow**) button to return the layer to the *Available* list.

The primary elevation layer is required and cannot be removed from the *Selected* list.

6. To apply tide correction, select a *tide station observations* option.

7. Click the *Input file* Browse (**...**) button and select an available file for the selected shift type.
8. Click **Open**.

The selected file is displayed in the *Input file* field.

When applying tide correction, you need to specify the date and time of the data to apply. Tide values for a particular area differ depending on the time of year and the time of day. This information is contained in layers in the surface.

9. Using the drop-down lists, identify the layers that contain the *Date* and *Time* values.

You have the option of selecting the vertical reference system that will appear in the CSAR metadata for the resulting surface created for the shift. The options available in the list are controlled by the Vertical Reference System database. This list can be edited using the Reference System Editor tool. For more information, see "VERTICAL REFERENCE SYSTEM EDITOR" ON PAGE 231.

10. Select a *Vertical Datum* option from the list.
11. To choose the name and location of the resulting surface, click the *Output surface* Browse (**...**) button.

A Save As dialog box is displayed.

12. Select a path and type a name for the shifted surface, then click **Save**.
13. To finish, click **OK**.

A new surface is created and displayed in the Layers window. The attributes and properties from the source surface are carried over to the shifted surface. Additional properties are also populated under the *Sources* group.

- *Operation:* This property identifies the tool that was used to create the surface. For a shift, it will read "VerticalShift".
- *Vertical Coordinate System:* The vertical coordinate system that was used to create the shifted surface.
- *Shift Method:* This property identifies the type of shift that was applied to the surface.
- *Shift File:* This property differs depending on the type of shift applied. For a tide shift, it identifies the file of depth values that was used to shift the surface.
- *Date/Time Attribute:* These properties identify the layers from which date and time values were obtained. These properties are only present for a surface with tide data applied.
- *Original Depth Update:* This property states whether or not the original depth value of any designated soundings was also updated by the shift. This property is only present if the source surface contained designated soundings.

- *Source:* This property identifies the name and location of the source surface.

The Z values in the selected layers should be shifted according to the selected method. To confirm that the shift was successful, compare the data in the new surface to the data in the original surface, referencing the adjustments in the tide file. The elevations should have been adjusted according to the data.

12

CUBE Processing

A Combined Uncertainty and Bathymetry Estimator (CUBE) surface uses multiple hypotheses to represent potential depth variances along the seafloor.

In this chapter...

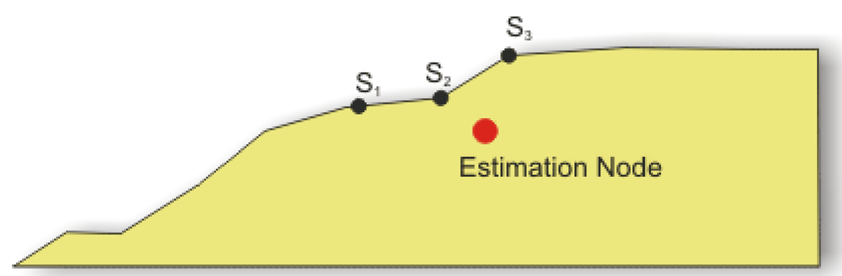
Overview

The CUBE process uses sounding propagation along with disambiguation to create and select hypotheses.

When soundings are propagated to a grid of estimation nodes:

- Soundings with a low vertical uncertainty are given more influence than soundings with high vertical uncertainty
- Soundings with a low horizontal uncertainty are given more influence than soundings with a high horizontal uncertainty.
- Soundings close to the node are given a greater weight than soundings further away from the node.

Generally, as soundings are propagated to a node, a hypothesis (depth value) is developed at that node. If a sounding's value is not significantly different from the previous sounding then the same or modified hypothesis is used. If the value does change significantly, a new hypothesis is created. A node can contain more than one hypothesis.



In the above graphic, two soundings—S_1 and S_2—have similar values and therefore are part of the same hypothesis at the estimation node. However, sounding three (S_3) has a significantly different value so it forms a new hypothesis.

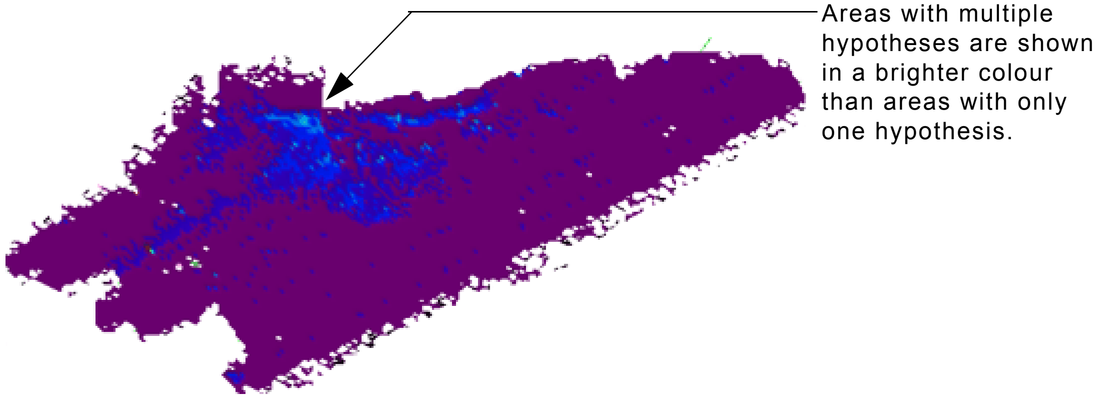
Disambiguation

The final process in CUBE is disambiguation. Disambiguation selects one hypothesis over others. There are four disambiguation options:

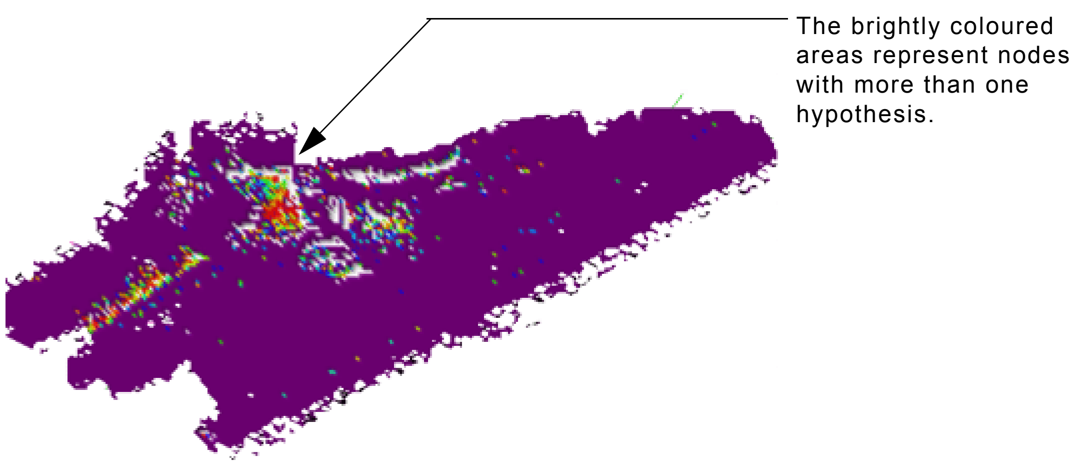
- Density: Select the hypothesis with the greatest number of sounding samples.
- Locale: Select the hypothesis that is most consistent with the surrounding nodes that have only one hypothesis.
- Locale and Density: Select the hypothesis the contains the greatest number of soundings and is also consistent with neighbouring nodes.
- Initialization: Select the hypothesis that is nearest to a node value of a previously created surface. Initialization differs from the other methods because it filters potential outlier soundings just prior to disambiguation.

When a surface using the density and locale options is generated, two layers specific to the CUBE Surface are displayed.

- The Hypothesis Count layer is a visual representation of hypothesis density at a node. A Surface with a Hypotheses Count layer is displayed below.



- The Hypothesis Strength layer is a visual representation of the mathematical confidence of a chosen hypothesis. Each node is given a value ranging from 0.0 (high confidence) to 5.0 (low confidence). Nodes with one hypothesis have a confidence value of 0.0 while nodes with multiple hypotheses will have confidence values greater than 0.0. A Hypothesis Strength layer is displayed below.



More than one hypothesis or a low confidence value does not necessarily mean an error. Uneven areas (slopes, for example) show more than one hypothesis because of the changing terrain. Nodes with multiple hypothesis should be examined in the Subset Editor. See "Hypotheses Cleaning" on page 308.

When you select the Initialization method of disambiguation, two additional layers are created:

- `Guide_Depth`: Depths from the initialization surface.
- `Guide_Uncertainty`: Vertical uncertainty values from the initialization surface.

The User Nominated layer displays the nominated hypotheses that were chosen over the hypotheses selected by CUBE disambiguation (see "Nominate Alternative Hypotheses" on page 305).

Generating CUBE Surfaces

CUBE Surfaces are created using the New Surface wizard.

All track lines must be merged and TPU must be computed before a CUBE Surface can be created.

You can select specific track lines to include in the surface or have the surface cover the extents of the data automatically.

Menu	Tools > Surfaces > New
Tool	

To create a CUBE Surface:

1. Select the New Surface command.

The number of steps in the wizard is determined by the type of surface you are creating.

Steps to create a CUBE Surface:

"New Surface" on page 288

"Set Extents" on page 288

"Set Surface Resolution and Type" on page 290

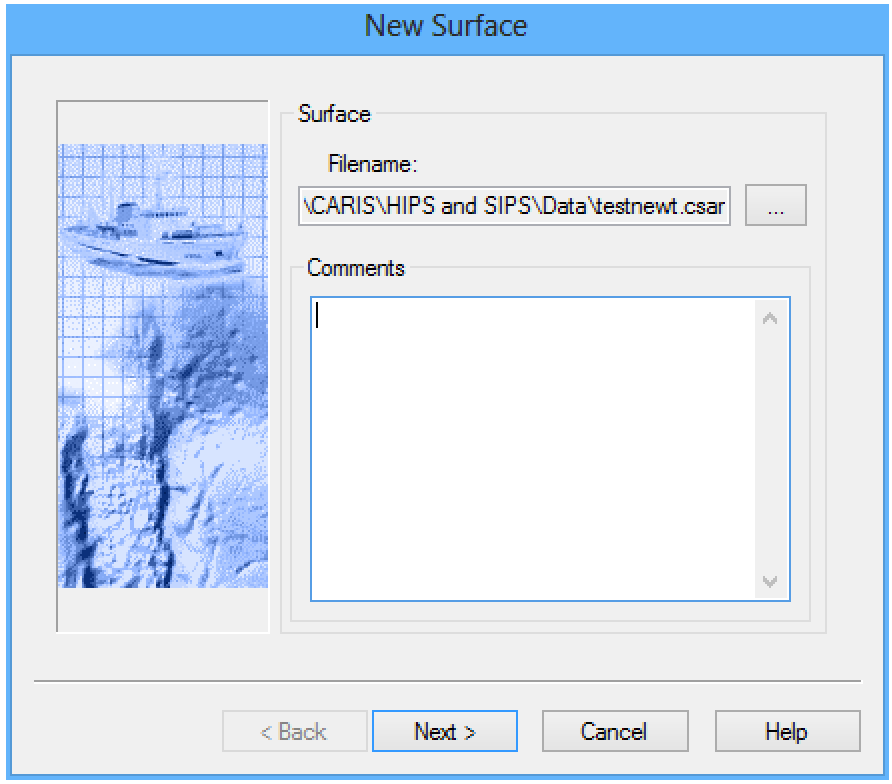
"New Surface - CUBE Options" on page 291

"New Surface - CUBE Parameters" on page 294

"Advanced Options for configuration" on page 295

New Surface

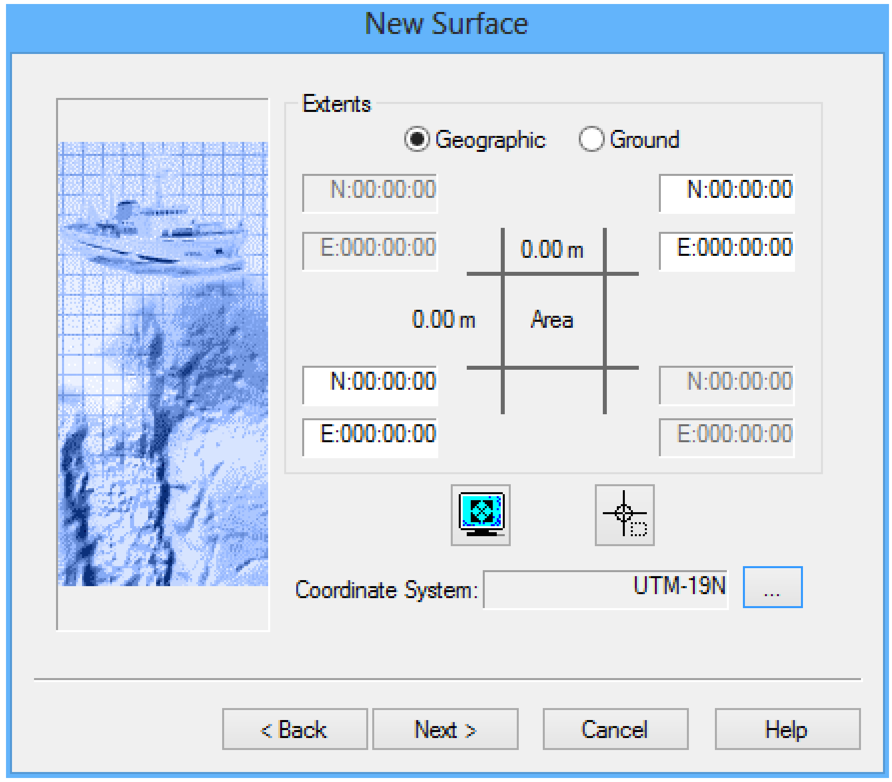
The New Surface Wizard (Step 1) dialog box opens.



1. Type a *Name* for the surface.
2. Type any necessary *Comments* in the text box, and click **Next**.

Set Extents

Set the extents and coordinate system of the new surface.



Using geographic coordinates in degrees-minutes-seconds, set the extents. By default, the extents are set to zero.

1. [Optional] Set coordinates to *Ground* option.
2. Define the extents of the surface by one of three methods:
   - use the current extent of the Display window, or
   - use a bounding box to define the extent, or
   - enter the geographic or ground coordinates.

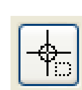
Use current display

1. Zoom in on the area you want to include so that it fills the Display window.
2. Click the Current Display Extent button. 

The extents field display the coordinates for the current display.

OR

Draw a bounding box

1. Click the Bounding Box button. 

The cursor in the Display window is shown as a cross-hair.

2. Press and hold the mouse button, and drag the cursor across the area where you want to create the surface.

A rectangular box is drawn across the area and the coordinates are displayed in the extents fields.

- To move the bounding box, position the cursor inside the box so it becomes a four-headed arrow. Press and hold the mouse button, and then drag the box to a new location.

- To resize the bounding box, click on the box handles at NE or SE and drag to resize the box.

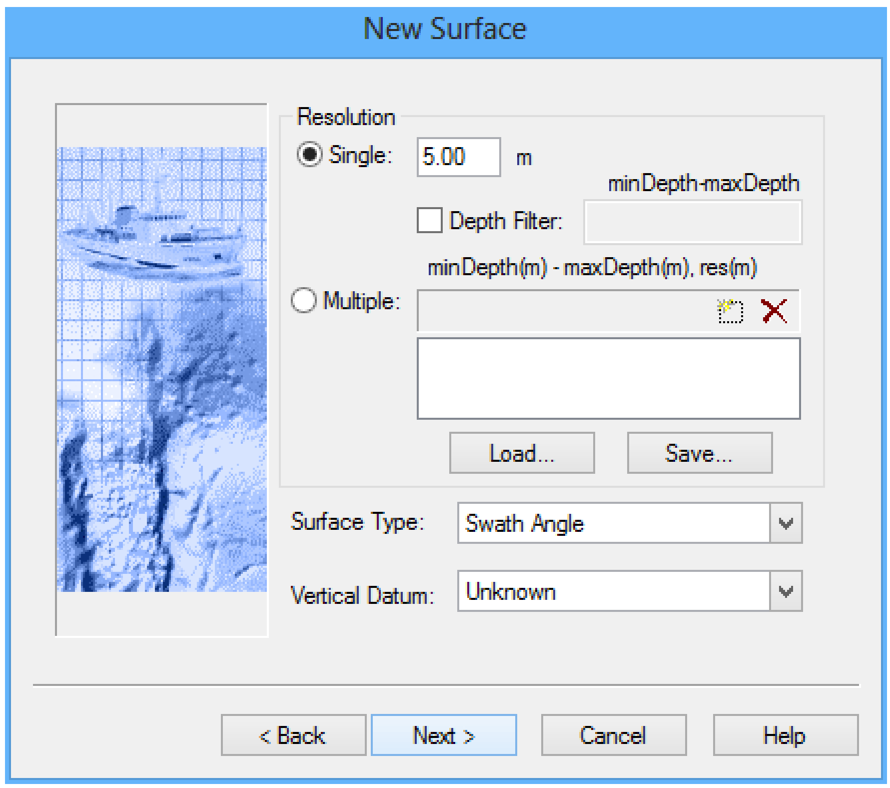
OR

Enter coordinates

1. Enter the northeast and southwest extents by clicking in one of the appropriate fields and typing the coordinate data, or by using the arrow keys to toggle the values into the entry fields.

Set Surface Resolution and Type

The step 3 dialog box is displayed.



The *Resolution* value(s) sets the distance(s) between surface nodes. You can use the same single resolution value for the entire surface, or use different resolutions for each range of depths.

1. Select *Single* and enter a resolution value, or select the *Multiple* option and enter a range of values.

If you select the *Single* option, you can determine the range of depths to include in the surface by using the *Depth Filter* option.

2. Select *Depth Filter* and enter a range of depths.

If you select the *Multiple* option, the depth ranges and related resolution values can be saved as a template to apply when creating other new surfaces.

3. Enter the series of depth ranges and the desired resolution in metres, and click **Save** to save the settings as a template.

   OR

4. Click Load to use a saved file.

Values are entered in the following format:
minimum depth - maximum depth, resolution (all in metres. For example, 0.0-10.0,5.0 which associates depths between zero and 10 metres with a resolution of 5 metres

5. Select CUBE from the *Surface Type* list.
6. Select the *Vertical Datum*, if known, from the list.
7. Click **Next**.

The content of the next dialog box is determined by the choice of surface type made in step 5:

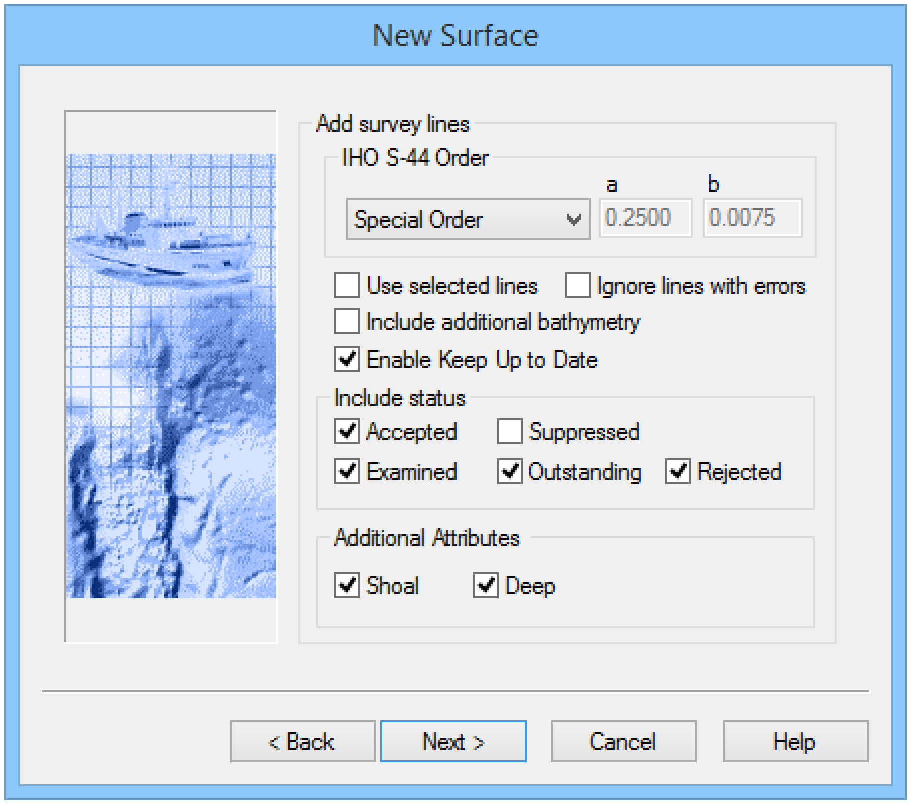
“NEW SURFACE - CUBE OPTIONS” ON PAGE 291

New Surface - CUBE Options

The S-44survey order is used to determine the area of influence that a sounding can apply to the nodes.

A sounding’s area of influence (the number of nodes to which a sounding is applied) is determined by the vertical uncertainty limit in the selected S-44 survey order. When a sounding’s vertical uncertainty increases beyond the requirement for the survey order then it cannot contribute to the node.

The *a* (constant depth error) and *b* (factor of depth dependent errors) fields are automatically filled when a survey order is selected. These values are read from ..\HIPS\System\IHO_Standards.xml.



Fields related to the *IHO S-44 Order* are used to determine the area of influence that a sounding can applied to the nodes.

The values in the “a” field show the constant depth error value associated with the selected S-44 survey order.

The values in the “b” field show the factor of the depth dependent errors value associated with the selected S-44 survey order.

These values are automatically displayed as for each order selected from the list.

1. Select an *IHO S-44 Order* from the drop-down list.

To apply other values:

2. Select User Defined from the list and type the values in the a and b fields.

By default a surface is created from all the lines within the extents of the data.

3. Check the *Use selected lines* check box if you want to create the surface only on the selected lines.

Use the *Ignore Lines with errors* option so that the surface creation is not interrupted if it encounters bad data. The process will skip the part of the line containing bad data and continue creating the surface with the next line. Once the surface is complete you can decide whether to remove the partial line.

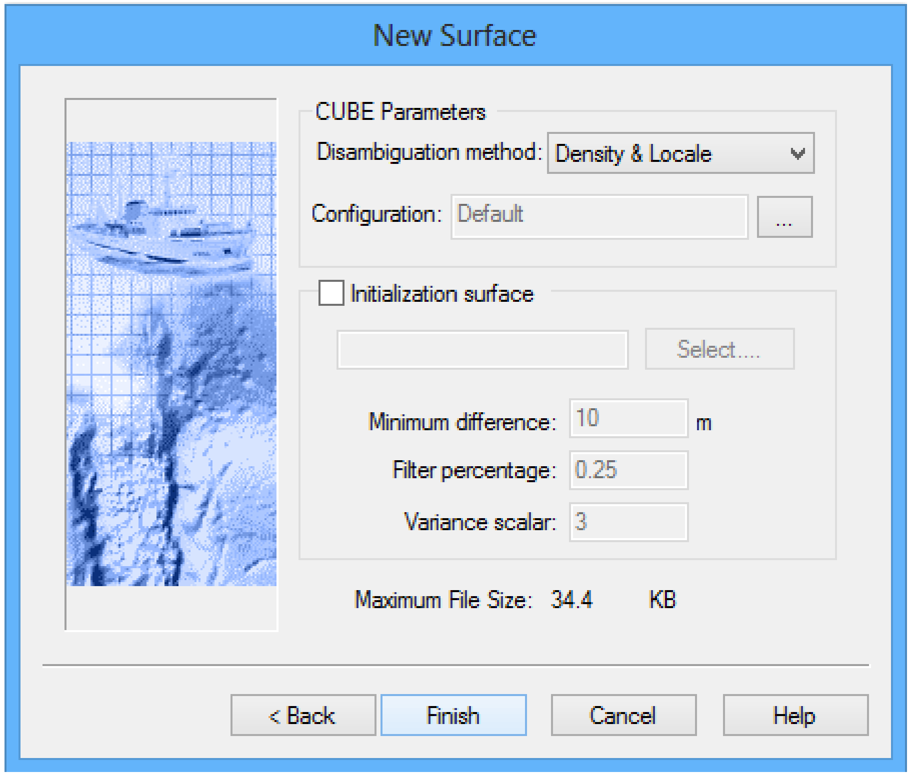
4. Check *Ignore Lines with errors* to have bad line data omitted from the surface.
5. Check *Include Additional Bathymetry* to include water column bathymetry added to project.

In order to use the *Automatic Surface Update* function set in Tools > Options > General, the *Keep Up to Date* option must be set during surface creation (or afterwards in the surface properties).

6. Enable *Keep Up to Date* to track the outdatedness of the surface.
7. Select or de-select the check boxes for specific status to include data in or hide it from the new surface.

   By default, data with *Accepted*, *Examined* or *Outstanding* status are set to be included in the surface. *Suppressed* or *Rejected* data can be included in the surface by selecting the check box.
8. Select *Shoal* or *Deep* (or both check boxes) to include an attribute layer in the CUBE Surface that contains the shallowest or deepest soundings.
9. Click **Next**.

New Surface - CUBE Parameters



1. Select a disambiguation method from the drop-down list:
   - Density
   - Locale
   - Density and Locale
   - Initialization Surface

   (These methods are described in “Disambiguation” on page 284.)

Default Configuration values will be applied unless you select otherwise. To apply other configuration options, click the [...] **Browse** button to open the Advanced configuration settings. (See “Advanced Options for configuration” on page 295.)

If you selected *Initialization* as the method for disambiguation, the *Initialization Surface* check box is automatically selected, so you can set values for the filtering tests.

2. Type the path and name of an existing surface, or click **Select** and browse to the surface.

If you have selected a disambiguation method based on density or locale, the *Initialization Surface* tests can be used as an optional filtering method.

Initialization Surface filtering uses the depth and vertical uncertainty values in an existing surface (the Initialization

Surface) to exclude potential outliers from the new surface. There are three filtering tests:

- *Minimum Difference* is a set distance from an existing surface node. For example, if the *Minimum Distance* is set at 10 metres, all soundings that are not within 10 metres of the node are excluded when a new Surface is generated.
- *Filter Percentage* is a percentage of depth at a node. For example, if the depth at a node is 20 metres and the percentage value is 0.25, all soundings that are more than 5 metres from the node (20x0.25) are excluded from the new Surface.
- *Variance Scaler* is a multiplier of vertical uncertainty at a node. For example, if the scaler is set to three and depth uncertainty for a node is one metre then soundings more than three metres (1x3) from the node are excluded from the processing of the new surface.

The shoalest values from the three tests are then used as the threshold to filter outlier soundings.

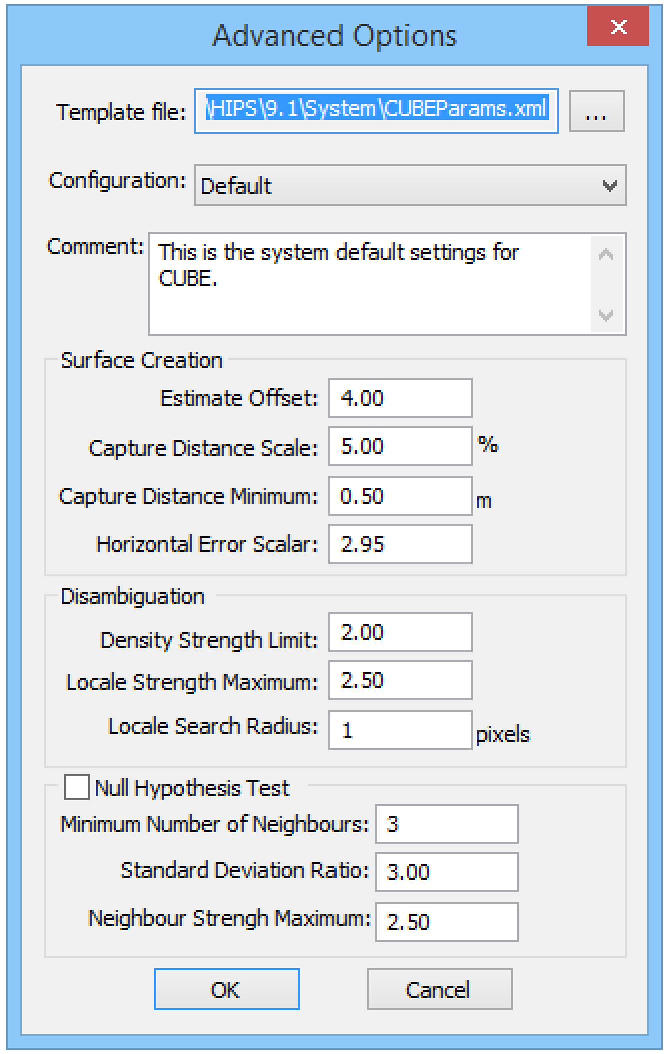
3. [Optional] Changes the default values set for the filtering tests.
4. Click **Finish**.

A CUBE Surface is shown in the Display window and the layers associated with the Surface are listed in the Control window. See “View Surfaces” on page 219 for more information on display options.

Advanced Options for configuration

The Advanced Options dialog box displays the detailed configuration settings applied to the CUBE surface. These settings are contained in the CUBEparams.xml file, found in the installed Template folder.

You can use the settings in this file or select a custom configuration file. If you use a custom file, use the same syntax and structure as the CUBEparams.xml file.



1. [Optional] Click **Browse** to select a configuration file. The file path is displayed in the *Template File* field.
2. Select a configuration file from the *Configuration* drop-down list: Default, Deep or Shallow.

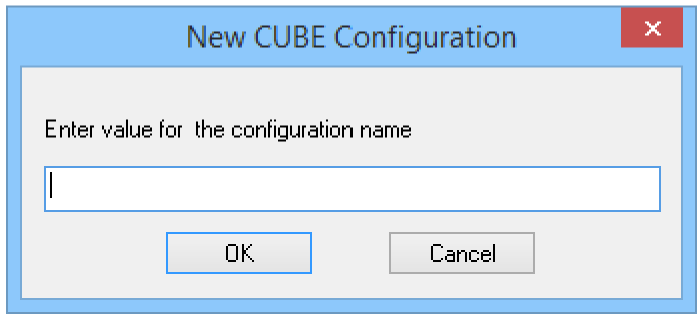
You can adjust the configuration settings can in these files, but you must save changes before they can be applied. If you do not want to overwrite the CUBEparams.xml file, select New and create a new template file.

- Deep: This configuration is intended to be used in areas where small features are not likely (such as shifting sand shoals), not important (steep grades and deep water), or separately located with side scan. This parameter set corresponds with NOAA's 2006 Complete gridding standard.
- Shallow: This file defines the parameters for the CUBE algorithm and are intended to be used in areas of critical under-keel clearance, in areas with numerous small features, and multibeam that can stand on its own without feature-by-feature side scan

correlation. This parameter set corresponds to NOAA's 2006 Object Detection gridding requirements.

- New: select New to save any changes to the default settings, or to create a new configuration. Otherwise the changes will be saved to the CUBEparams.xml file.

3. If you select New, the New CUBE Configuration dialog box is displayed.



4. [Optional] Type the name for the new configuration setting and click **OK**.
5. [Optional] Type any additional information in the *Comments* field.
6. The default values in the following fields can be adjusted, as needed. You will be prompted to save your changes before the configuration can be applied. *Estimate Offset*: The threshold for significant offset from current estimate to warrant an the creation of a new hypothesis. The value must be between 0.1 and 10.0.
   - *Capture Distance Scale*: Scale on predicted or estimated depth for how far out to accept data. Value is a percentage of depth used to limit the radius of influence a sounding may have on the grid. Value must be between 1.00 and 100.00.
   - *Capture Distance Minimum*: The minimum value (in metres) on predicted or estimated depth for how far out to accept data. This value is used in conjunction with Capture Distance Scale to limit the radius or influence of a sounding. Value must be between 0.0 and 100.00.
   - *Horizontal Error Scalar*: The value used to scale the horizontal error of each sounding when used in the radius of influence computation. Value must be between 0.0 and 10.00.
   - *Density Strength Limit*: The strength value used to switch from the 'density' disambiguation method to the 'locale' version when using the density & locale algorithm. Value must be between 0.00 and 5.00.
   - *Locale Strength Maximum*: The maximum strength value allowed as part of the mean in the locale algorithm. Value must be between 0.00 and 5.00.
   - *Locale Search Radius*: The radius of the search when computing the trimmed mean. The values are in pixels and must be greater than zero. The original system default value is 1.

- *Null Hypothesis Test*: Flag used to control the application of the NULL hypothesis test. Value must be either True or False. Default value is False.
- *Minimum Number of Neighbours*: During the Null Hypothesis test, this controls the minimum number of neighbours that a node must have in order to be considered for the Standard Deviating Ratio test. If the node has less the specified number, it is automatically marked as 'Null'. Values must be between 2 and 8.
- *Standard Deviation Ratio*: During the Null Hypothesis test, this represents the cut-off limit for the standard devotion ratio including the node in question to the standard deviation of the qualified neighbouring nodes. If the computed ratio exceeds the specified value, the node is marked as 'Null'. Values must be between 0.0 and 10.0.
- *Neighbour Strength Maximum*: The maximum strength value that is allowed to be considered as part of the standard deviation computations in the NULL hypothesis test. Value must be between 0.00 and 5.00.

7. Click **OK** to save and apply settings.

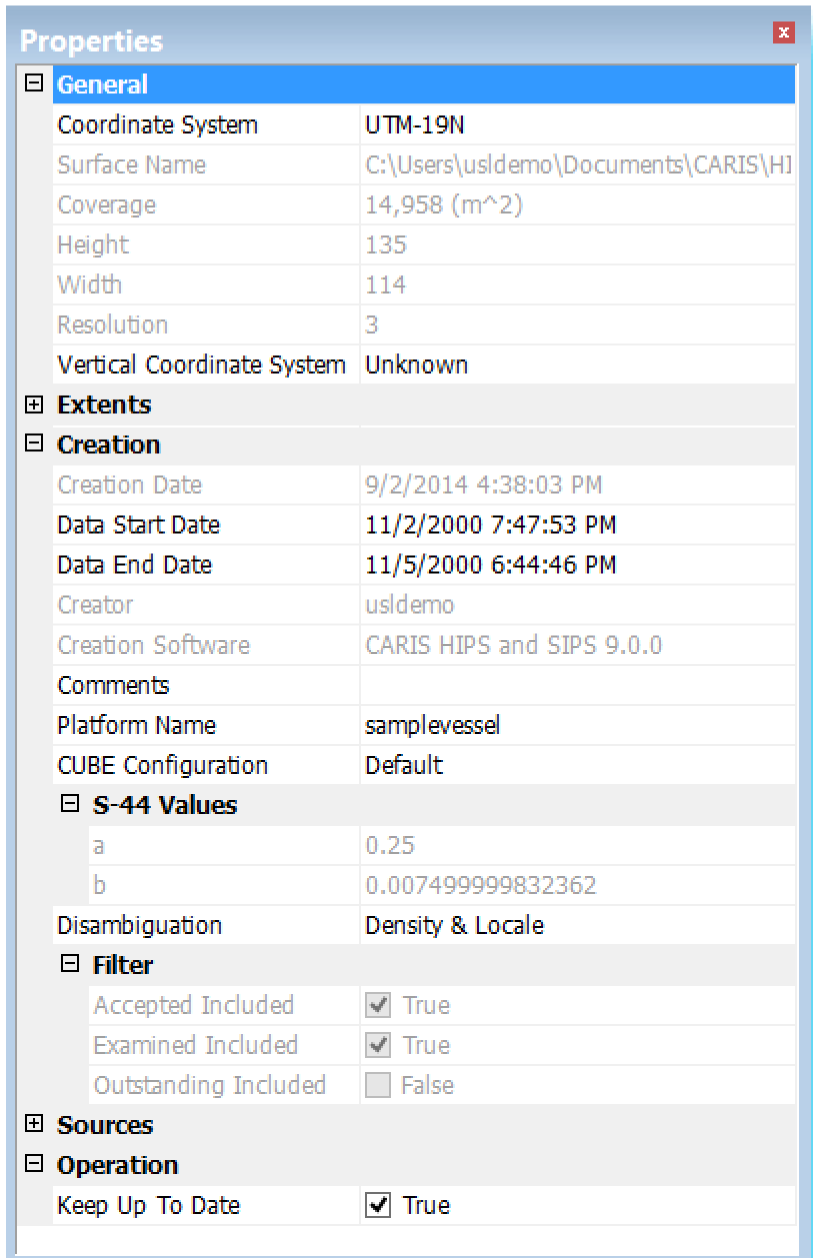
CUBE Surface Properties

Menu	Window > Other Windows >Properties

1. Select a CUBE surface layer in the Layers window.
2. Select the Properties window command.

The Properties window displays the surface information.

The image below shows the properties of a CUBE surface parent layer.



Most of this information is read-only, however, the following fields can be edited:

Properties	Function
General	
Coordinate System	Displays the name of the coordinate system used. To see further details, click **Browse**.
Vertical Coordinate System	Select the appropriate vertical datum for the surface data from a drop-down list.
Creation:	
Data Start Date	Date can be modified using the drop-down calendar
Data End Date	Date can be modified using the drop-down calendar
Comments	This field contains the contents of the Comments field from the first step in creating a surface. This can be the mandatory <abstract> element for 19115 metadata, or other information about the surface entered during creation. This content can be edited.
Platform Name	Displays the vessel name.
CUBE Configuration	Displays the configuration selected in the last step of creating a CUBE surface. If a configuration other than Default is displayed, click on the Browse button [...] to see the values applied in the configuration that was selected. (Viewed in the Advanced Options dialog box.) (See also "Advanced Options for configuration" on page 295.)
S-44 Values:	(Present only if selected surface is a CUBE surface.)
Disambiguation	Displays the process used to create the CUBE surface. To see the effect of another disambiguation method, select it from the drop-down list.
Operation:	
Keep Up To Date	This option is set to track the outdatedness of a surface. In order to use the *Automatic Surface Update* in Tools > Options >General, *Keep Up to Date* must be set during surface creation or afterwards in the properties.

Hypothesis Editing

CUBE surfaces can be displayed in the Subset Editor for examination and editing, using an iterative process to choose which of multiple hypotheses of depth values are the best ones to represent the sea floor.

When a CUBE surface is created, soundings are weighted and contribute to surface grid nodes based on TPU values and distance from the nodes. The CUBE surface allows for multiple depth estimates or hypotheses to exist at a single grid node, depending on the variation of the sounding data. CUBE then uses "Disambiguation" to determine which hypothesis at each node is the most "correct".

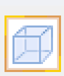
You can verify and, if necessary, override, a CUBE decision in Subset Editor, by nominating an alternative hypothesis as the depth. Once these kinds of changes have been made to the CUBE surface, the surface is updated.

A surface filter can then be applied to the data. Any sounding data that is not in agreement with the selected hypotheses will be flagged as rejected. By applying this surface filtering, the number of manual edits required by the hydrographer to produce a clean sounding data set is greatly reduced.

> When following a CUBE workflow, this hypothesis editing process is performed instead of other filtering such as Swath or Subset Editor filters.

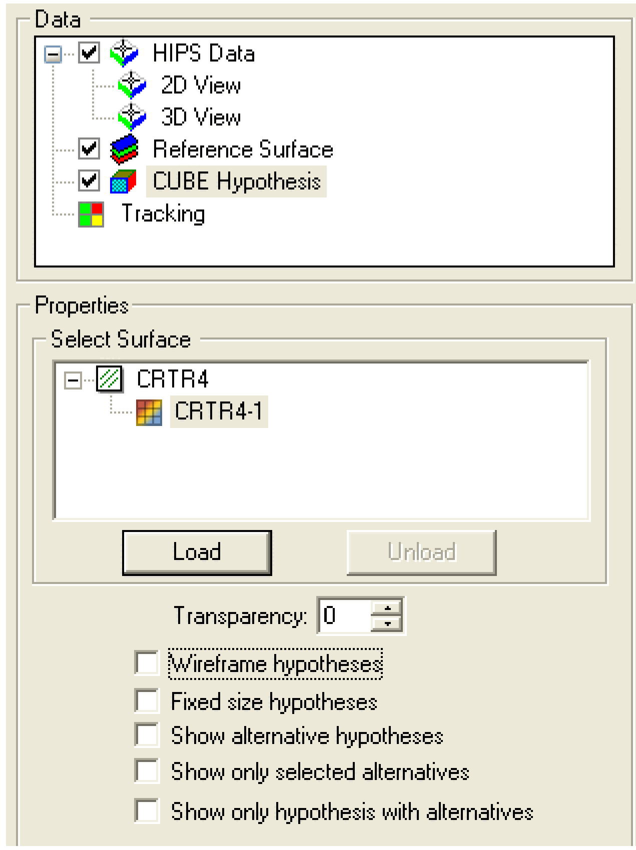
To examine an open CUBE Surface in Subset Editor:

1. Select the Subset Editor command.
2. Define the subset with a bounding box.
3. Load the subset.

Menu	Tools > Subset Editor
Tool	

Data contained within the bounding box is loaded into the 3D Subset view.

4. In the Subset Control window, select the CUBE Hypothesis layer.



5. Expand the Select Surface data tree and select a surface containing CUBE data.
6. Click **Load** to view the CUBE surface.
7. To return to the normal display in the 3D and 2D Views, click **Unload**.

The display options for the hypotheses are listed at the bottom of the Subset Control window.

"Wire frame Hypotheses" on page 303

"Fixed Size Hypotheses" on page 304

"Nominate Alternative Hypotheses" on page 305

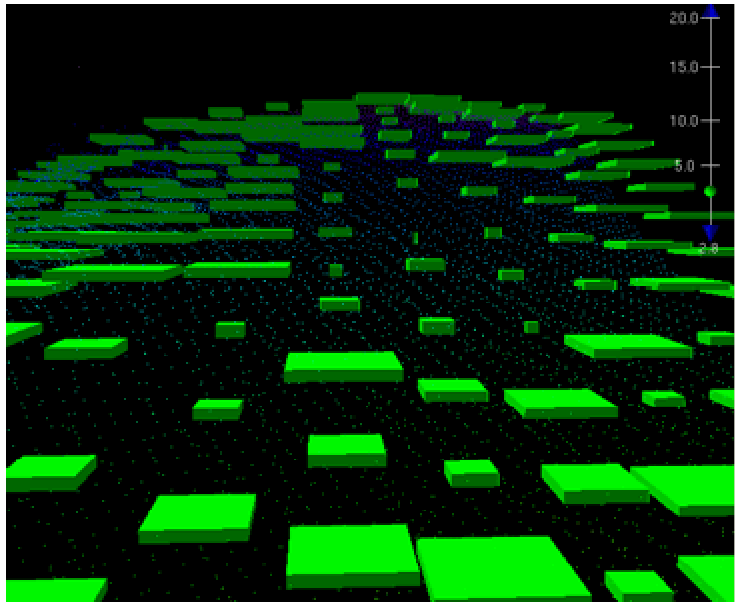
"Show only Selected Alternatives" on page 306

"Show Only Hypotheses with Alternatives" on page 306

8. View and edit the hypotheses. See "Hypotheses Cleaning" on page 308
9. Update the surface. See "Updating a CUBE Surface" on page 309

Hypotheses views

Below is an example of an enlarged section of a CUBE surface loaded in the 3D View.



Hypotheses are represented as squares. The squares have different dimensions to reflect the confidence level given to each hypothesis. These confidence levels range from 0.0 (the highest value) to 5.0 (the lowest value). The size of a square is related to the numerical confidence level: large squares represent nodes with the high confidence values while smaller squares represent nodes with low confidence values.

The thickness of the square represents the vertical uncertainty assigned a node (to a 95% confidence interval). The thicker the square, the greater the vertical uncertainty.

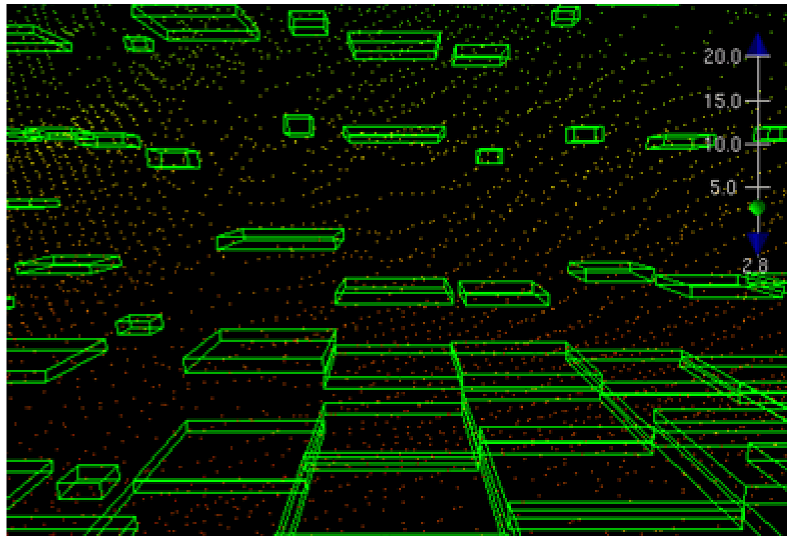
More than one hypothesis or a low confidence value does not necessarily mean an error. Uneven areas (slopes, for example) show more than one hypothesis because of the changing terrain.

Wire frame Hypotheses

Hypothesis squares can also be displayed without a fill so that they are transparent except for the outlines of the squares.

1. Select the *Wireframe Hypotheses* check box.

The 3-D View is refreshed to display the squares without fill, as illustrated below.

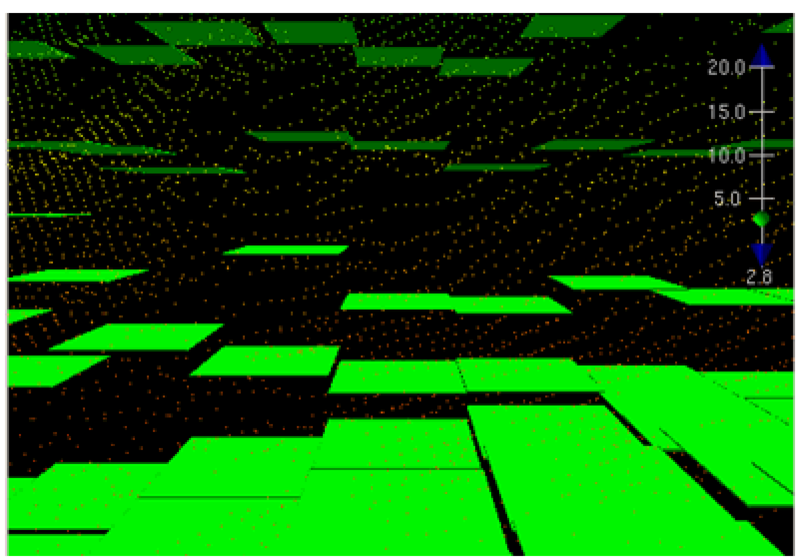


Fixed Size Hypotheses

The *Fixed Size Hypotheses* option flattens the hypotheses squares and redraws them to a standard size. This option is useful if you want to examine the dataset for holes or to see the exact position of a square in the subset.

1. Select the *Fixed Size Hypotheses* check box.

The 3-D View is refreshed to display the squares as flattened and in a standard size.



Nominate Alternative Hypotheses

Some nodes have alternative hypotheses that were not selected during disambiguation You can view alternative hypotheses and compare them against the selected ones. If needed, the alternative hypotheses can be nominated to replace the established hypotheses.

"Show alternative hypotheses" on page 305

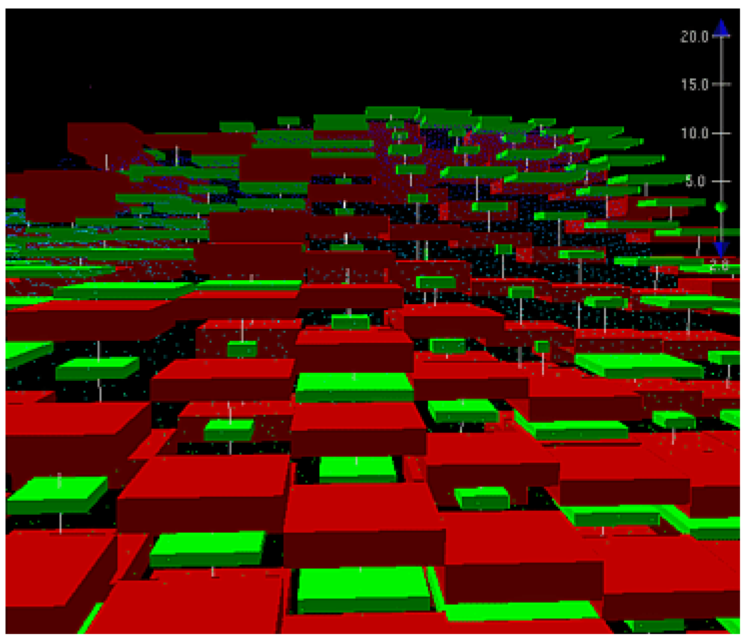
"Show only Selected Alternatives" on page 306

"Show Only Hypotheses with Alternatives" on page 306

Show alternative hypotheses

1. Select the *Show Alternative Hypotheses* check box and click **Apply**.

The 3-D and 2-D Views are refreshed to highlight the alternative hypotheses. The following image shows a 3-D View with the alternative hypotheses displayed in red.



The red squares represent hypotheses that were not selected during disambiguation. The vertical lines that run between the nodes are viewing aids to match these alternative hypotheses to the ones selected by CUBE.

1. To replace an hypothesis, select an alternative hypothesis in the 2D or 3D window.

The Nominate and Clear commands are now active.

2. Select the Nominate command.

Menu	Tools > Subset Editor > Nominate
Tool	

The alternative hypothesis is now highlighted in blue. This means that the nominated hypothesis has replaced the established one.

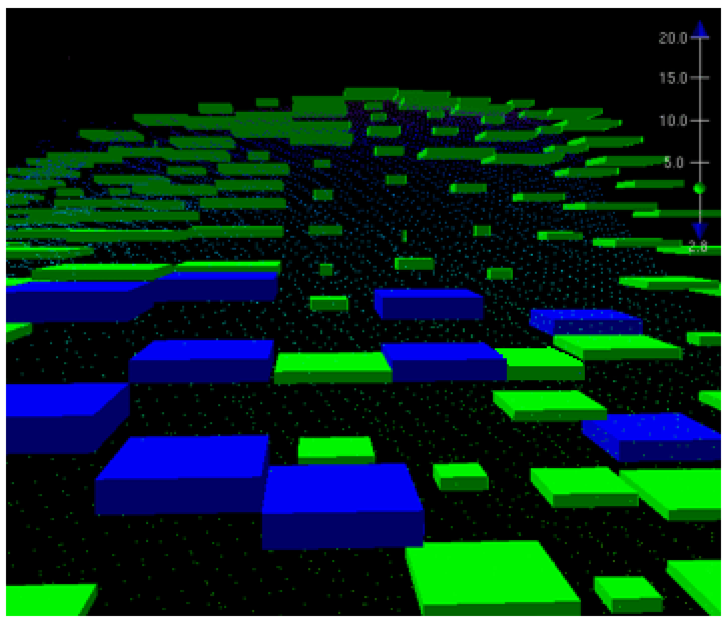
All nominated hypotheses are given the highest confidence value of 0.0.)

Show only Selected Alternatives

To compare nominated nodes against the existing nodes,

3. Clear the *Show Alternative Hypotheses* check box
4. Select the *Show Only Selected Alternatives* check box.

The 2D and 3D Views are refreshed to show only the nominated and established hypotheses. The following image shows the 3-D View with only nominated and established hypotheses displayed.

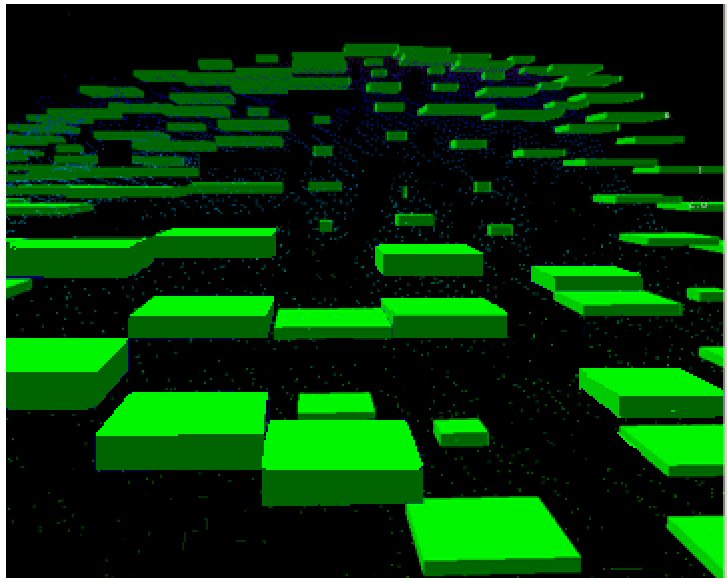


Show Only Hypotheses with Alternatives

Some hypotheses will not have alternatives. To view only those hypotheses selected by CUBE that *do* have alternative hypotheses:

5. Select the *Show Only Hypotheses with Alternatives* check box.

The Views are refreshed to show only the hypotheses for which there are alternative hypotheses, as illustrated below.



When you have finished nominating alternative hypotheses to replace the hypotheses selected by CUBE disambiguation, the User Nominated layer will display these nominated hypotheses.

Menu	Tools > Subset Editor > Clear
Tool	

6. Select the Save command to save your data.
7. To remove the nomination flag, select the Clear command.

Hypotheses Cleaning

Like soundings in Swath and Subset Editor, hypotheses can be given a rejected flag. When a hypothesis is rejected, it is retained in the Surface but is excluded from processing when disambiguation is run again or when filtering is applied.

Menu	Tools > Subset Editor > Reject
Tool	

1. Select a hypothesis.
2. Select the Reject command.

The hypothesis is now flagged as rejected.

To change rejected back to the accepted status (and therefore available for further processing).

Accept

Menu	Tools > Subset Editor > Accept
Tool	

1. Select a hypothesis.
2. Select the Accept command.

The hypothesis is now flagged as accepted and is available for further processing.

Reject Node

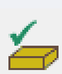
Menu	Tools > Subset Editor > Reject Node
Tool	

To reject all hypotheses at a node:

1. Select a hypothesis.
2. Select the Reject Node command.

All hypotheses associated with the node are rejected.

Accept Node

Menu	Tools > Subset Editor > Accept
Tool	

To return hypotheses associated with a node to their normal status.

1. Select a hypothesis.
2. Select the Accept command.

All hypotheses associated with the node are flagged as accepted.

Updating a CUBE Surface

Menu	Tools > Surfaces > Recompute
Tool	

If changes are made to the data contributing to the surface, the CUBE Surface can be rebuilt to show these changes using the following methods:

- Select the Recompute Surface command. (See “Recompute Surface” on page 254).

When recomputing a CUBE surface, all CUBE surface editing, including nominations, hypothesis and node rejections are considered no longer valid and *will not be retained* after re-computation of the surface.

- Select the Automatic Surface Update check box in Tools> Options > General. After edits are made, this will automatically update the area of the CUBE surface contained within the subset bounding box. (See General for more information).

When using the Automatic Surface Update option, all CUBE surface editing, including nominations, hypothesis and node rejections *could be retained* if no soundings were rejected in the proximity of the nodes.

If soundings have been rejected after CUBE surface editing, these editing adjustments will be lost.

13

Data QC

Use a surface to highlight problems with sensor data that may exist, and edit problem areas.

In this chapter...

Workflow

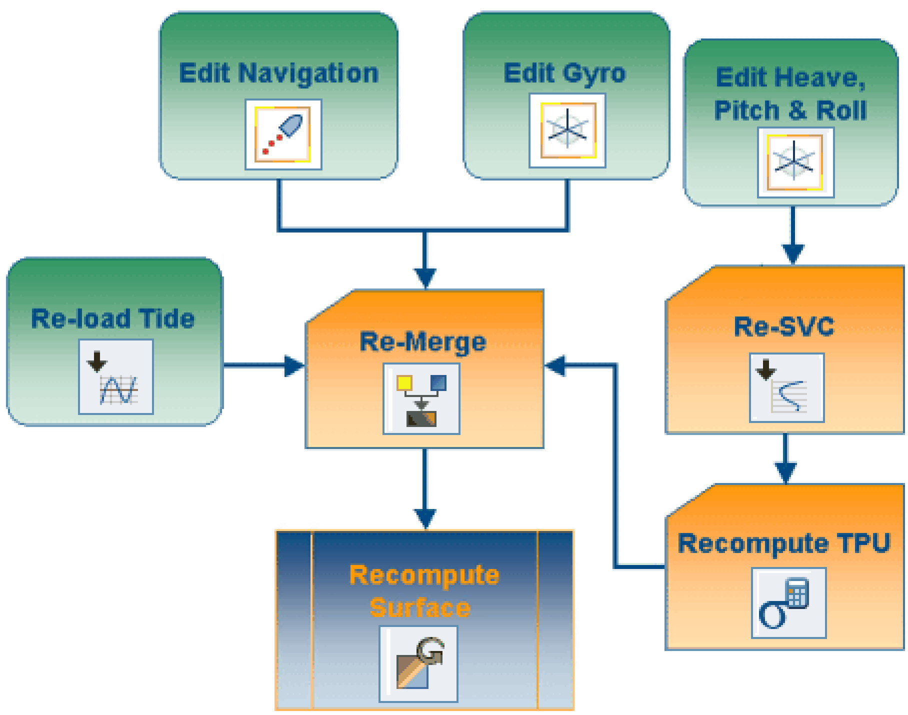
Problems with sensor offsets, auxiliary sensor data, and so forth, become readily apparent in the surface. Relying on a surface to highlight problems in the auxiliary sensors can improve processing efficiency, because you are no longer required to investigate the motion and navigation for each survey line.

In the HIPS multibeam workflow described here, the auxiliary sensor data was not investigated prior to creating the surface. So at this point, you can use the surface to highlight problem areas that may exist.

Certain edits will require that data be Merged again. For example, if you reload tide, edit navigation, or edit gyro, you will need to Merge the modified lines again.

If you edit heave, pitch or roll, you will need to re-apply Sound Velocity Correction. If you re-apply SVC, recompute TPU before Merging again.

If any project lines are re-Merged, the surface will need to be re-generated. See “Recompute surface” on page 254.



After data QC, the following processes can be applied to further clean data:

- “Subset Cleaning” on page 389
- “Swath Cleaning” on page 372

- filters
  - "Protect Critical Soundings" on page 347,
  - "Attitude Filter" on page 329,
  - "TPU Filtering" on page 190,
  - "Single Beam Filtering" on page 810.

Tools for examining and editing data

There are a number of tools available to assist examining soundings, and in tracking the progress of data cleaning. For example, you can:

- examine information about one or more selected track lines ("Query Data" on page 315).
- set a flag to indicate the type of survey line ("Classify Lines" on page 332).
- reject an entire track line to remove it from processing ("Reject Line" on page 333)
- show selected types of rejected soundings ("View Soundings Status" on page 334)
- preserve and highlight shallowest soundings ("Critical Soundings" on page 337)
- easily view Designated, Examined and Outstanding soundings when working with a large data set ("Critical Soundings" on page 337)
- reverse previous depth cleaning by re-setting status flags from "Rejected" to "Accepted" ("Restart Cleaning" on page 350)

See also "Surface Filtering" on page 247.

Query Data

Line information, such as project and date of survey, and status of corrections such as SVC and Merge, can be viewed in the Selection window and the Detailed Line Query window.

This line profile information is displayed in columns. The columns which are displayed, and the order in which they are displayed, is controlled by a pop-up menu. (See "Column Settings" on page 521 of the Reference guide for details on adjusting the columns displayed.)

Similar information for data in HIPS editors can be viewed when selected and queried using the Query or Query Line command.

Queried data can be saved to a text file from the display, or a line report can be generated using the Process > Line Report command.

"View Line Data" on page 315

"Query Line Data" on page 317

"Query Data in Editors" on page 316

"Line Report" on page 317

View Line Data

To view data profiles for one or more track lines selected in the Display window:

1. Select a track line in the Display window or in the Project window.
2. Open the Selection window.

The Selection window automatically displays the data profile for each selected line. As well as Project, Vessel, Day and Line identification, the profile data displayed can include this information:

- Min Time
- Max Time
- Total Time
- Merged
- Outdated
- Speed
- Line Reject
- Line Class
- Heading
- Length
- SR Corrected
- GPS Tide
- Tide Loaded
- Raw Range
- SVP Corrected
- TPU Computed
- Nav Examined
- Del Dft Loaded
- Tide Applied
- Locked
-
-

The information is read-only and cannot be modified.

3. [Optional] Select Save As from the right-click menu to save the line data to a text file.

(See “Column Settings” on page 521 of the Reference guide for details on selecting and adjusting the columns displayed.)

Detailed Line Query

To view details of line data in addition to that displayed in the Selection window, use the Detailed Line Query function to display an additional 30 columns of data in the Detailed Line Query window.

To view this detailed line information:

1. Select a line or lines in the Display window.
2. Select the Detailed Line Query command.

Menu	Process > Report > Detailed Line Query

The Report window opens and displays more than 50 columns of detailed line information for the selected lines.

If you are processing a large number of lines, a progress bar will display in the left side of the Status bar. You can cancel loading of the selected lines by selecting **Cancel** on the Processing dialog box.

To query another line or lines:

3. Select the lines.
4. Select the Detailed Line Query command again.

The display in the Report window will be refreshed to show your new selection.

As with the Selection window, the Report window will display the line information in a table. These read-only records can be sorted by column by clicking on a column header.

Which data is visible and in what order it is displayed is controlled using the Column Settings dialog box, opened from the right-click menu. (See "Column Settings".) Changes made to the column selection and order are retained until you change them.

This detailed information can be saved to a text file.

5. [Optional] Select Save As from the right-click menu to save the line data to a text file.

You can also save a customized selection of the data using the “Line Report” on page 317.

Query Data in Editors

Use the Query function to examine data selected in any HIPS and SIPS interactive editor. For example, position data selected in Navigation Editor, or pings selected in Side Scan Editor.

Menu	Edit > Query
Tool	
Pop-up	Query

1. Select the data so it is highlighted in one of the editor windows.
2. Select a Query command.

The selected records are displayed in the Selection window. For example, in Navigation Editor, this data is displayed for each numbered record:

- Time
- d-Time
- Lat (DMS)
- Lon (DMS)
- Distance (m)
- d-Distance (m)
- Speed (m/s)
- d-Speed (m/s)
- CMG
- d-CMG
- Status

This data is read-only. However, individual records can be rejected or accepted using the pop-up menu in the Selection window.

(See "Column Settings" on page 521 of the Reference guide for details on selecting and adjusting the columns displayed.

Query Line Data

To view records for entire lines in a HIPS and SIPS Editor, use the Query Line command.

Menu	Edit > Query Line
Pop-up	Query Line

1. Select data in any editor window.
2. Select a Query Line Command.

The records for the entire line are displayed in the Selection window, in the same tabular format as the Query function provides. This data can be rejected or accepted from the pop-up menu in the Selection window.

Line Report

Use the Line Report command to save line profiles directly to a text file.

The saved report will contain a table of values under the same column headings that can be viewed in the Detailed Line Query window.

As well as the detailed line data, the report also shows the totals of the values for these specific columns:

- Total Time
- Length
- Total Nav
- Accept Nav
- Reject Nav
- Total Depth
- Accept Depth
- Reject Depth

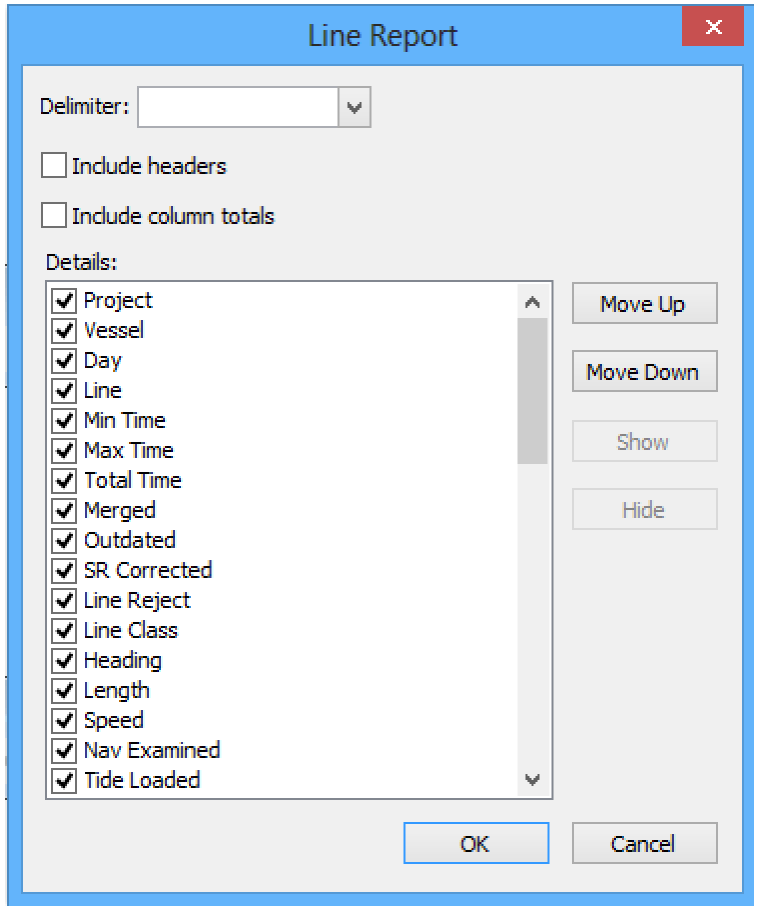
The text file can be opened in a text editor, but column data can be best viewed in a spreadsheet.

To create a line report:

1. Select a line in the Display window.
2. Select the Line Report command.

Menu	Process > Report > Line

The Line Report dialog box is displayed.



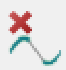
3. Select a *Delimiter* to use to indicate the separate columns. The default is Tab.
4. Select *Include column totals* to have this information included in the text file.
5. From the *Details* list select the column headings to include in the text file.
6. Click **Move Up** and **Move Down** to change the order of the columns.
7. Click **OK**.
8. In the Save As dialog box, set a name and destination folder for the text file.
9. Click **Save**.

Your selection of Details and the order of the columns, are retained for the next time you use the line report tool.

Rejecting and Accepting Data

Data can be rejected or accepted in HIPS and SIPS editors. All data is marked as “Accepted” until flagged otherwise.

Reject Data

Menu	Edit > Status Flag > Reject-With Interpolation/Reject-Break Interpolation
Tool	 
Pop-up	Reject > Reject-With Interpolation/Reject-Break Interpolation

1. Select and highlight the data in the View window of the editor.
2. Select a Reject command (Reject with Interpolation or Reject_ Break Interpolation).

The selected data is now flagged as rejected.

Accept Data

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

Use this command to revert the status of rejected data back to “Accepted”.

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

The previously rejected data is now flagged as Accepted.

Examine Navigation Data

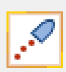
Navigation outliers can affect the final positions of soundings during the Merge process. Therefore, the navigation data should (but is not required to be) be examined and cleaned for outliers.

In Navigation Editor you can examine navigation data track lines to accept or reject, and query data, as necessary.

To locate navigation outliers, examine the track lines and time graphs that show the computed speed, distance, and course made good between navigation points.

Multiple navigation sources can be associated with a line.

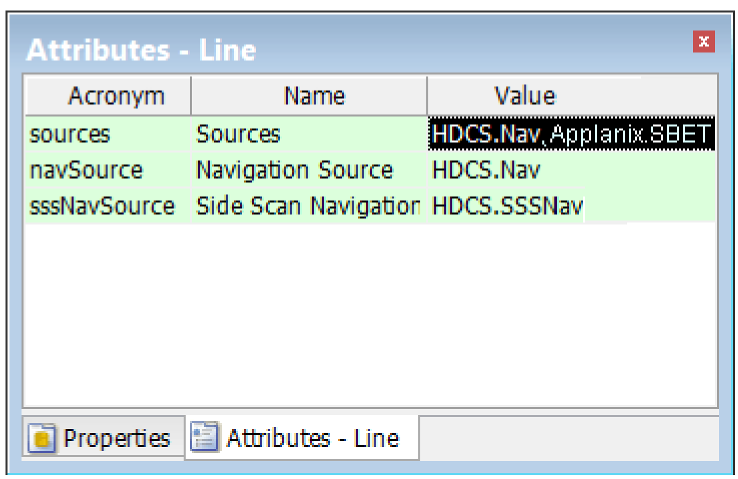
To edit lines in Navigation Editor:

Menu	Tools > Navigation Editor
Tool	

1. Select a track line in the Display window.
2. Select the Navigation Editor command.

Navigation Editor time graphs open in the HIPS and SIPS interface and display relevant speed, distance, and course-made-good data for the selected line.

3. Open the Attributes window to view the navigation attributes for the selected line.



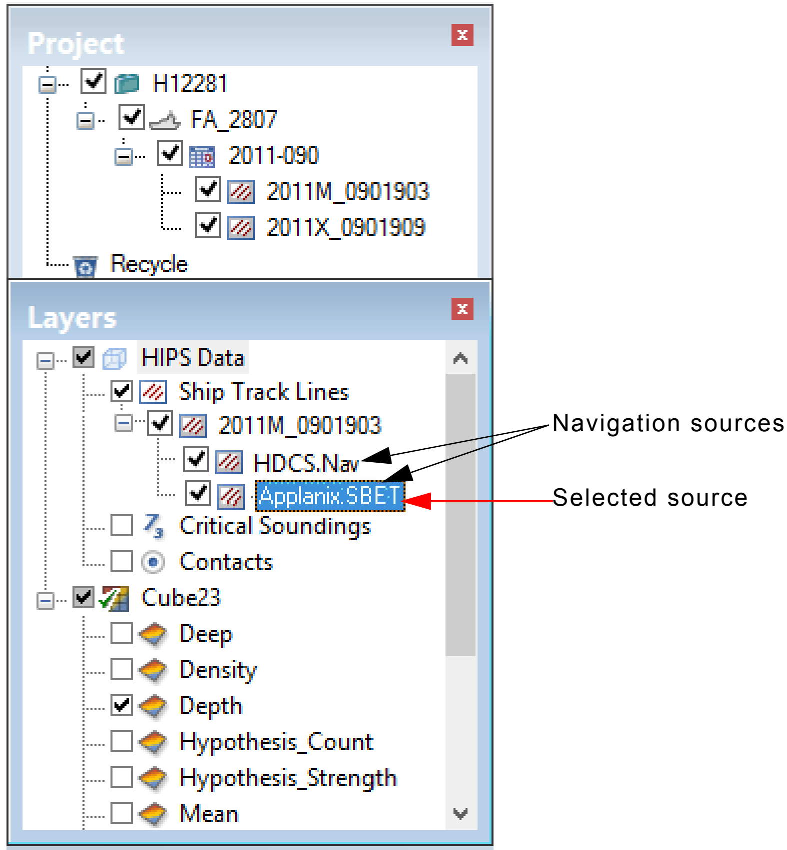
The Value column for the *Sources* field lists the available navigation sources. The *Navigation Source* field shows the active navigation source.

The Layers window will display the available navigation sources for the line. Only the selected, i.e. active source can be edited.

To make a source active:

4. Select a navigation source in the Layers window.

In the following example the Applanix source is the active source.

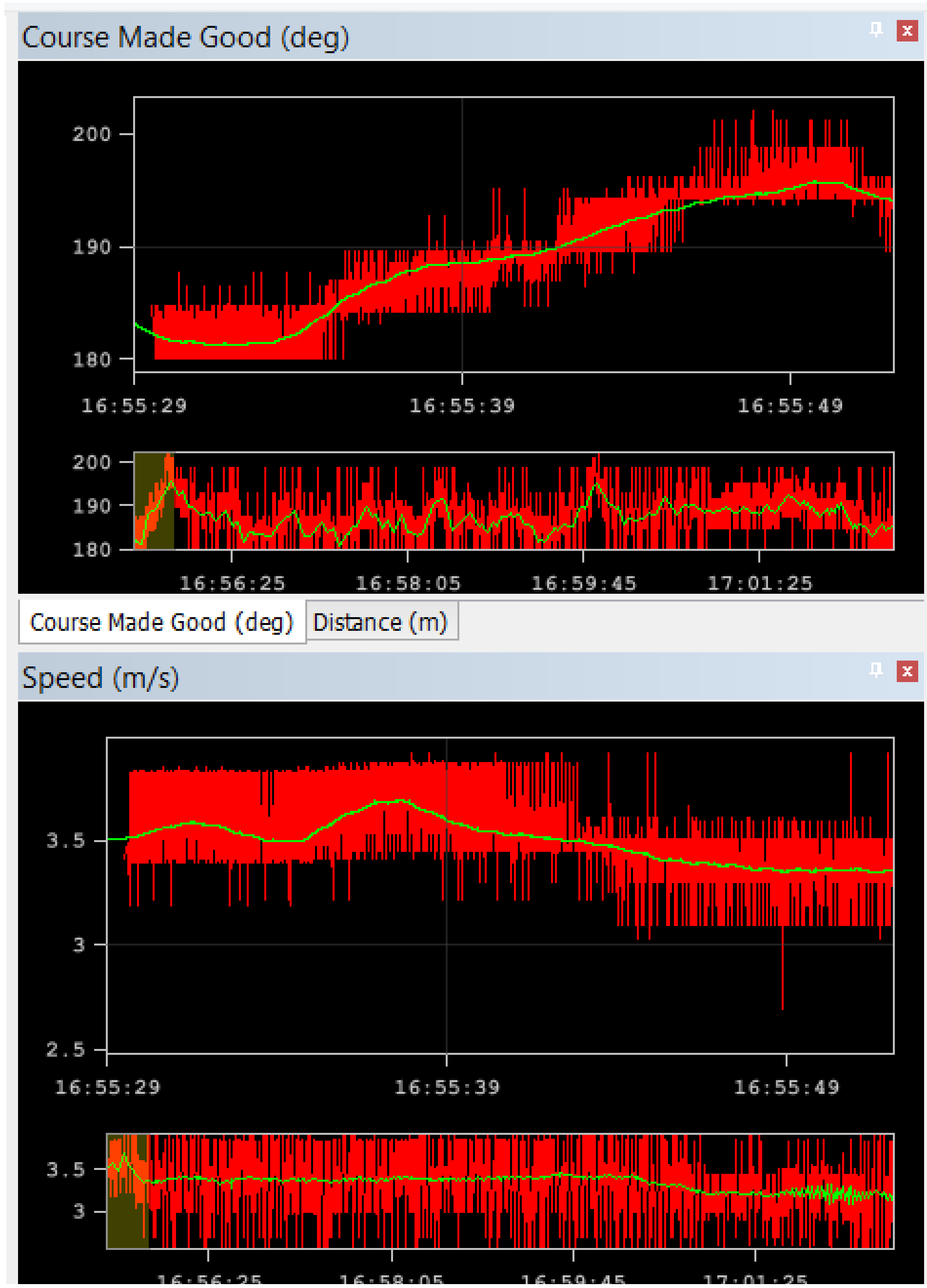


When lines with multiple navigation sources are open in Navigation Editor, the data sources are displayed simultaneously in the Navigation graphs.

To differentiate between sources, set different colours for each using the Properties:

1. Expand the Ship Track Lines layer to see the navigation sources layers.
2. Select a navigation source layer.
3. Open the Properties window.
4. Click in the Colour field to select a colour from the colour palette.
5. Repeat for each source as necessary.

In the following illustration, the HDCS navigation data is plotted in red. The green line is the Applanix SBET data that is set as the active navigation source.

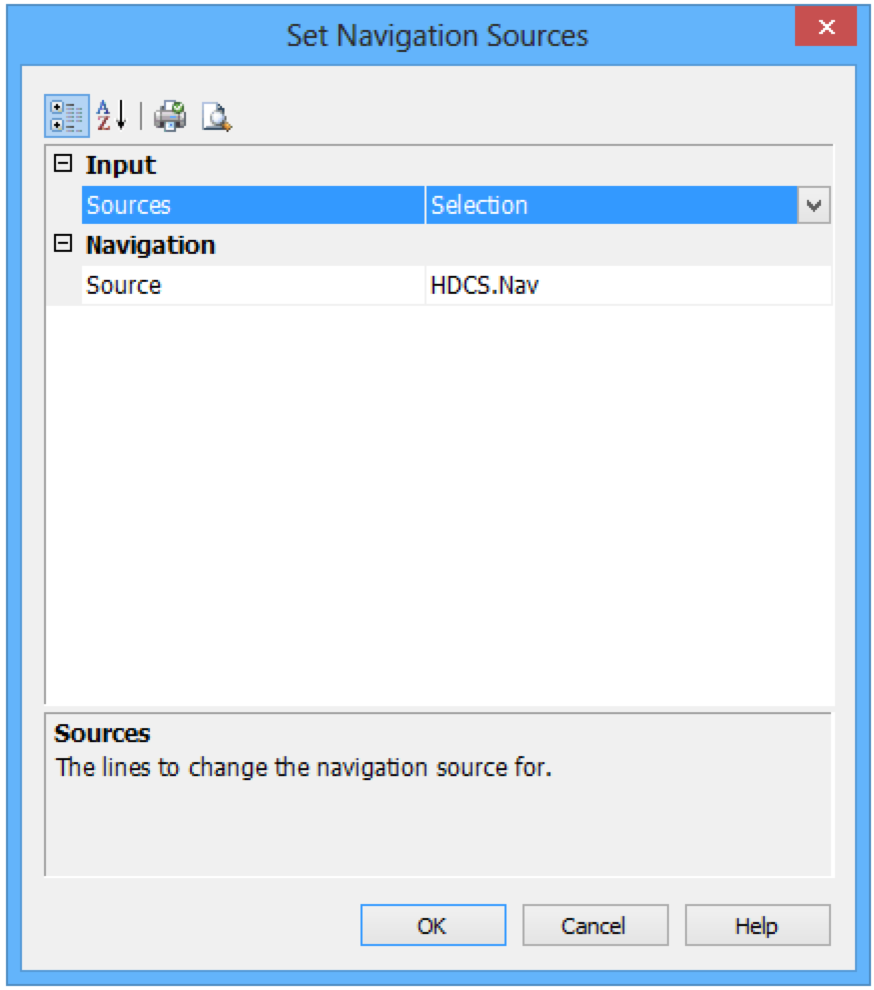


To change the navigation source setting:

Menu	Process > Navigation > Set Source

6. Select the Set Navigation Sources command.

The Set Navigation Sources dialog box is displayed.



When the Navigation editor is open, Set Navigation Sources will only apply to the line opened in Navigation Editor.

If Set Navigation Sources is used when there are no editors open, you can set the navigation source for all the open lines or for a selection of lines.

7. To set the source for all open lines, select Ship Track Lines from the Input Sources list.
8. To set for sources for some lines, select Selection from the list.
9. Choose the navigation source to be set for the line(s), from the list in the Navigation Source field.
10. Click **OK.**

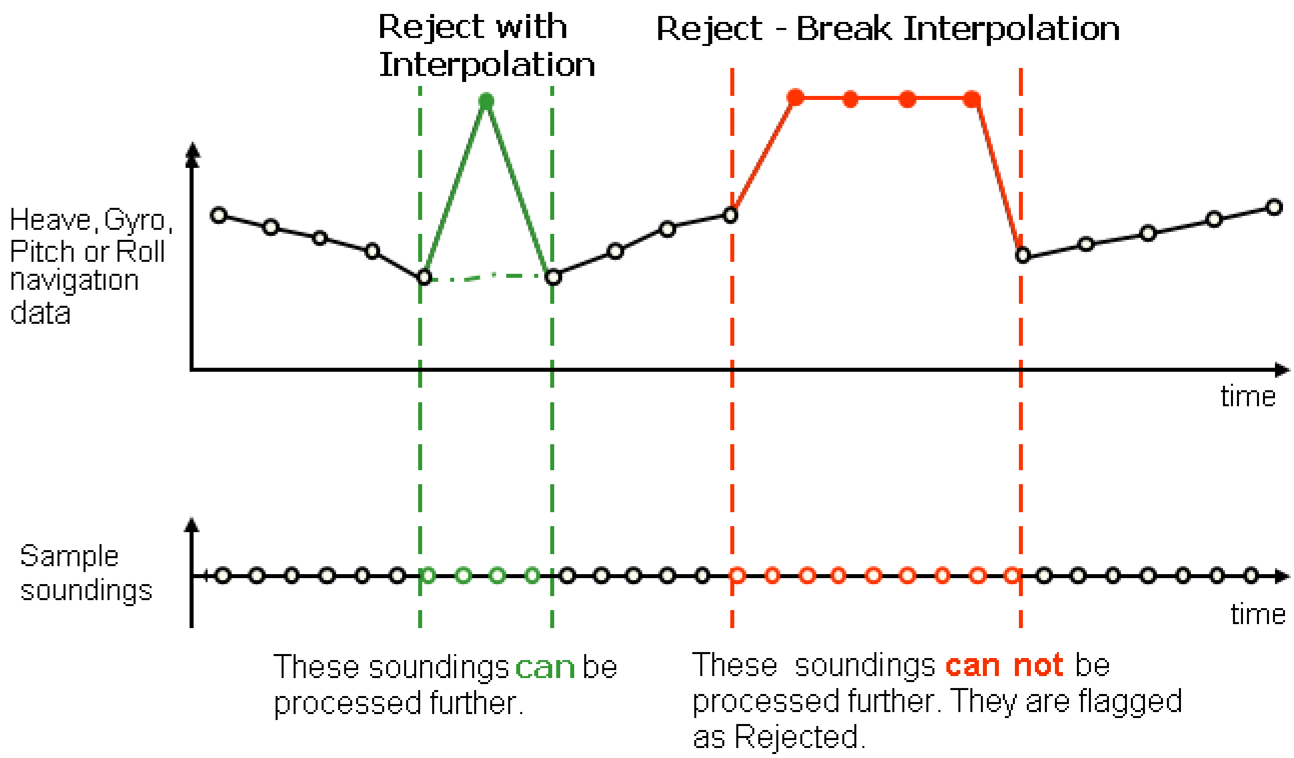
Rejecting Navigation Data

All data is marked as Accepted until flagged otherwise. Data can be queried for position fixes, and rejected.

Data can be rejected using either

- **Reject - With Interpolation:** Soundings associated with this navigation data are processed further since they have positions interpolated for them.
- **Reject - Break Interpolation:** Soundings associated with this navigation data are flagged as rejected and cannot be processed.

These alternatives are illustrated below.



Line Interpolation

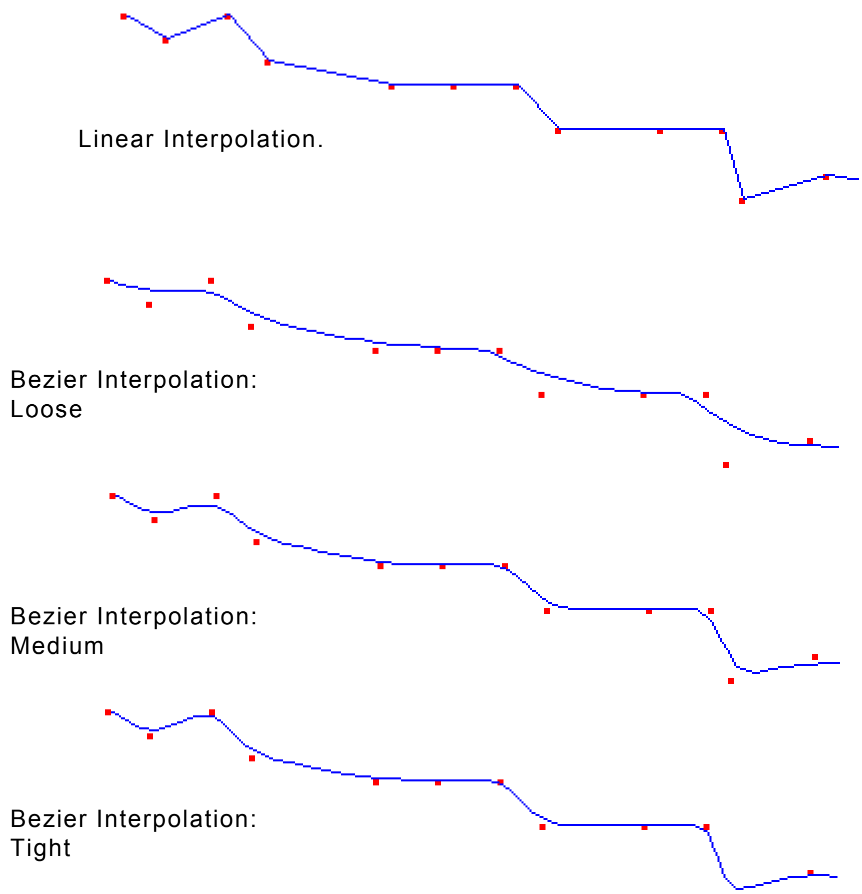
Position observations do not usually occur at exactly the same instant as a depth observation (ping). Thus it is unlikely that there is position data for every ping.

For example, positions observations may be taken every second, (1 Hz frequency), while pings may be observed 10 times a second (10 Hz frequency). So, in most cases it will be necessary to interpolate positions to match the time for each ping.

You use either linear or Bezier method of interpolation.

- **Linear**: Calculate new positions by connecting consecutive positions using straight lines. This is the default method.
- **Bezier**: Calculate new positions by connecting points using a Bezier curve that does not necessarily connect all navigation positions. Bezier curves are available in three types: tight, medium and loose.

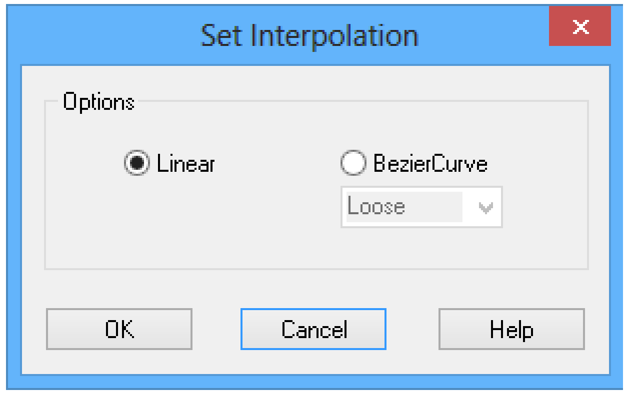
Linear interpolation is suitable if the original navigation positions are clean and do not significantly deviate from neighbouring positions. Bezier interpolation is suitable if the original data is noisy.



In the Navigation Editor the interpolation method is set in the Set Interpolation dialog box.

Menu: Process > Navigation > Set Interpolation

1. Select Set Interpolation command from the Tools > Navigation menu.
2. Select either *Linear* or *Bezier Curve* by selecting the appropriate check box.
3. If the *Bezier Curve* option is enabled, select either a *Loose*, *Medium* or *Tight* format from the drop-down menu.


Set Interpolation
Options
Linear
BezierCurve
Loose
OK
Cancel
Help

4. Click **OK**.

Shift Navigation Data

Use this to apply a shift in latitude and longitude to an entire survey. Latitude and longitude can be shifted independently to a maximum of 1 minute (0.016667 in decimal degrees and 1800 metres).

After data is shifted this way, Merge must be applied.

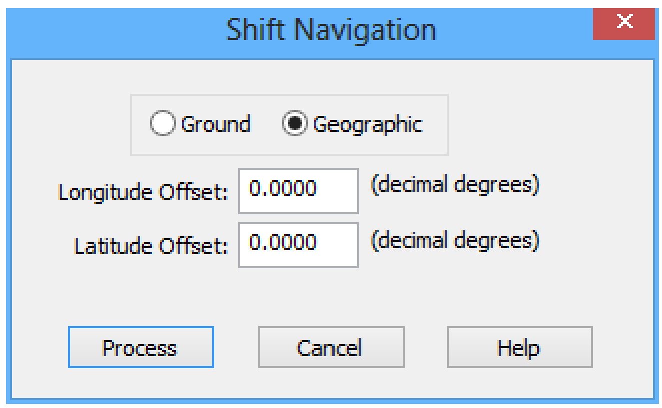
To shift survey data to a new position:

1. Select one or more lines.

Menu	Tools > Shift Navigation

2. Select the Shift Navigation command.

The Shift Navigation dialog box is displayed.

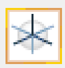


3. Select either Ground or Geographic units.
4. Type the amount of shift in one or both of the *Offset* fields. (Offset value can be positive or negative.)
5. Click **Process**.
6. [Optional] Click **Cancel** to cancel the process, and click **Yes** to confirm the cancel.

Attitude data

You can examine and clean the motion data of the vessel or towed transducer using the Attitude Editor, within the HIPS and SIPS interface. (See also "ATTITUDE EDITOR INTERFACE" ON PAGE 716.)

To open the Attitude Editor:

Menu	Tools > Attitude Editor
Tool	

1. Select a track line so it is highlighted.
2. Select the Attitude Editor command.

The Attitude Editor is displayed, showing graphs for Gyro, Pitch and Roll (in degrees) and Heave (in metres) for the selected line.

- Gyro data is displayed as a positive value when a clockwise rotation is experienced.
- Pitch data is displayed as positive when the vessels bow is down.
- Roll data in is displayed as positive when the vessels starboard side is up.
- Heave data is displayed as a positive value when the vessel is heaved upwards.

Other sensors can be added by selecting them from the Available Sensors list on the Sensor Layout dialog box.

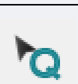
Menu	Tools > Sensor Layout

1. Select the Sensor Layout command.

Use the space bar to move forward along a track line and <CTRL> + <SPACEBAR> to move back.

Query Data

Use the Query command to display information about selected data.

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

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

The following data is displayed in the Selection window:

- time stamp
- d-time difference in seconds from last time stamp
- sensor value in either degrees or metres/feet
- d-value difference from last value
- status

You can change the data status flag to rejected or accepted.

1. Select the data in the Selection window so it is highlighted.

2. Select a Reject or Accept command.

The data is marked as accepted or rejected, depending to the option you selected.

Filter Attitude Data

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

Menu	Tools > Attitude Editor
Tool	

1. Select a track line so it is highlighted.
2. Select the Attitude Editor command.

The Attitude Editor is displayed. (See “Attitude Editor Interface” on page 716.)

Attitude Filter

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).

The Attitude filter can be used to:

- Smooth data: The Smooth function is used to even out localized variability. The parameters for creating the smoothed data are saved to the SmoothedCoefficients file in the ...\Project\Vessel\Day\Line folder.

  This file can be applied to the track line during any process that supports smoothing.
- Filter data: The Filter function is used to reject attitude that falls outside of defined boundaries. Soundings with the same time stamp as the rejected data are also flagged as rejected. You can choose to use interpolation when rejecting data.

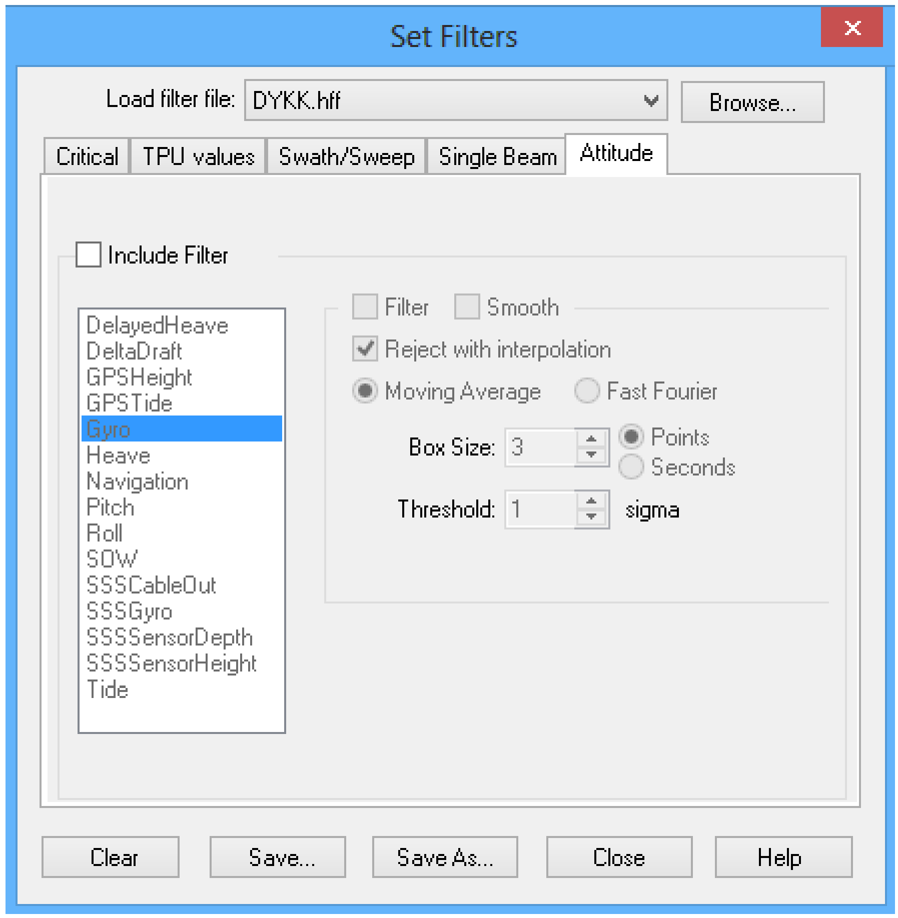
(These filter and smooth functions are also available in the Attitude Editor.)

Menu	Tools >HIPS Data Filters > Settings
Tool	

To set filtering options for attitude:

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

The Set Filters dialog box is displayed.



3. Select a sensor from the list.
4. Select the *Filter* check box.

By default, the filter is set to “Reject with interpolation” so that soundings with the same time stamp as the rejected data are not also flagged as rejected.

5. Clear the Reject with Interpolation check box if you want soundings associated with the rejected sensor data to also be rejected.

There filtering options are: Moving Average and Fast Fourier.

- The *Moving Average* calculates a mean for each data point, by calculating 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.

- *Fast Fourier* applies a common wave-smoothing algorithm to a data 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.

6. Select either the *Moving Average* or *Fast Fourier* filtering option.
7. Select the Box Size parameter by clicking the up or down arrow buttons. When using the Moving Average, the Box Size determines the size of the window for averaging the data point. When using the Fast Fourier, the Box Size determines the cut-off frequency that is applied to all data.
8. Determine if the Box Size is in (data) points or time, by selecting the Points or Seconds option.
9. Click the Box Size up or down arrow buttons to select the number of adjacent data points.
10. Click the *Threshold* up or down arrow keys until you reach a desired value. Threshold is a multiple of the standard deviation (sigma).
11. Click Save As to save the filter settings to a HIPS filter file.

Menu	Tools >HIPS Data Filters >Attitude
Tool	

To use the filter:

12. Select an Apply Filter command from the Tools menu.

The corresponding attitude data is rejected.

For other automated filters, see “Automatic Filtering” on page 373, “TPU Filtering” on page 190, “Single Beam Filtering” on page 810, and “Protect Critical Soundings” on page 347.)

Classify Lines

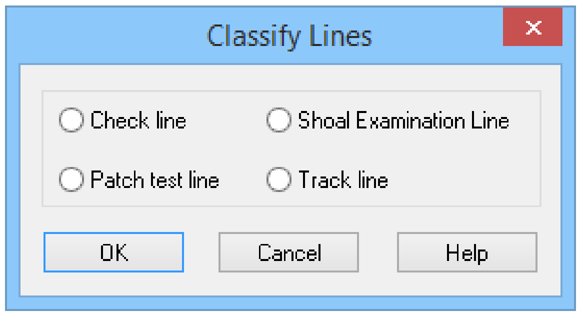
Use this command to set or change a classification flag for a track line, to help distinguish between line types.

1. Make sure the Ship Track Lines layer is selected in the Layers window.
2. Select a track line in the Display window or Control window.

Menu: Edit > Classify Lines

3. Select the Classify Lines command.

The Classify Lines dialog box is displayed.



4. Select from four options for setting the line classification flag:
    - *Check Line*: A line that is run perpendicular to the survey and is used to check the validity of survey data.
    - *Patch Test Line*: A line that is run over a surveyed area to calibrate sensor and sonar offsets.
    - *Shoal Examination Line*: A line that is run over a surveyed area to re-check possible shoals on the sea floor.
    - *Track Line*: A line that is part of the main survey grid.
5. Click **OK**.

The line is now flagged according to one of four categories.

Reject Line

Use the Reject Line command to remove an entire track line from data processing.

When a rejected line is merged, the profiles are marked as rejected, which prevents the soundings from being used in surface creation, surface cleaning, and mosaicing. However, status flags on the individual soundings are maintained. A rejected line is not processed in the tiling operations or exported to a CARIS map or ASCII file.

1. Select a track line.

Menu	Edit > Reject Line

2. Select the Reject Line command.

The rejected line is displayed in the Project window with a red **X** beside the file name.

To change status of a rejected line back to accepted:

Menu	Edit > Accept Line

3. Select the Accept Line command.

The line is displayed in the Project window with a red exclamation point (**!**) beside the file name. This icon indicates the line has been flagged as outdated. You must merge the soundings again before they are available for further processing (see "Merge" on page 195).

View Soundings Status

Menu	View > Display Filter
Tool	
Key	<F4>

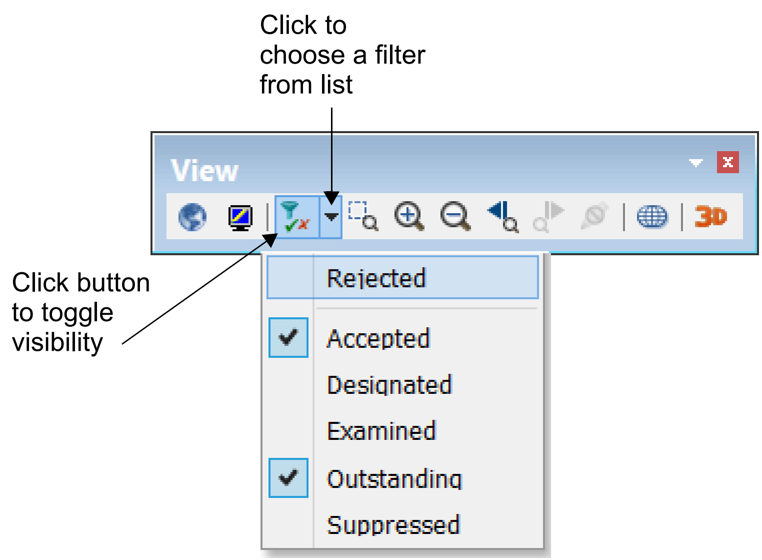
You can display or hide data in HIPS editors based on assigned status flags, using the Display Filter command. This command controls the display of Accepted and Rejected soundings, as well as Critical Soundings.

For example, the two-part Display Filter button on the View toolbar can be used to display or hide Rejected data in an editor.

The left part of the Display Filter button activates the display of the types of data selected from the list on the right-hand part of the button. By default the Display Filter is not enabled and all data is displayed.

In the example below, the Display Filter button is activated, and Accepted and Outstanding soundings have been selected for Display.

.



For example, to toggle between displaying *all* the data open in a HIPS editor, and displaying only Rejected data:

1. Open data in one of the HIPS editors.
2. Click the arrow part of the Display Filter button and select the *Rejected* check box from the drop-down list.
3. Click the Display Filter button to make the Rejected data visible.
4. Click the button again to see all data.

The status of individual soundings can be seen by querying data in an editor, and looking at the Status column in the Selection window.

Reason for Rejection

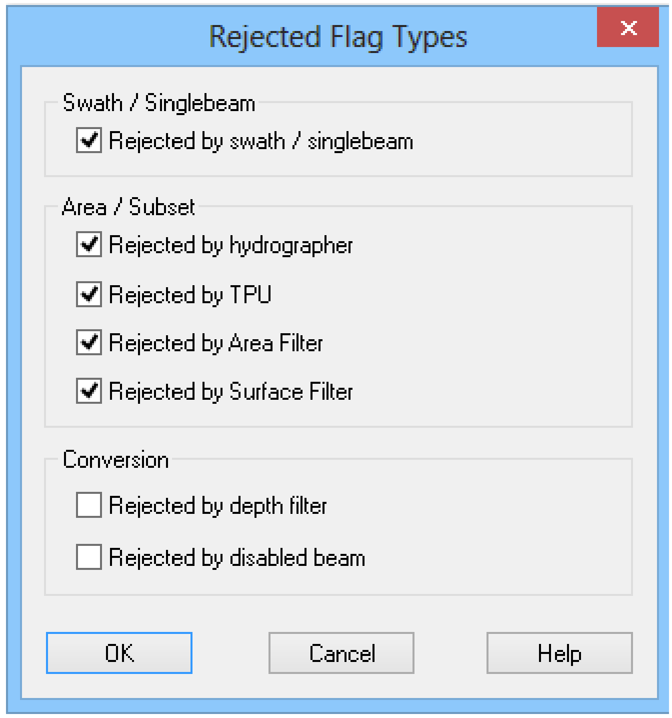
You can also control which type of rejected soundings are displayed in the Single Beam, Swath, and Subset Editors.

To display specific types of rejected data:

Menu | View > Rejected Flag Types

1. Select the View Rejected Flag Types command.

The Rejected Flag Types dialog box opens.



2. Place a check mark beside the box with the type of rejected data you want to view.
   - *Rejected by Swath/Single Beam Editor*: View soundings rejected in Swath or Single Beam Editor or by the Swath or Single Beam filters.
   - *Rejected by Hydrographer*: View data rejected interactively in the Subset Editor by the hydrographer.
   - *Rejected by TPU*: View soundings rejected through the Total Propagated Uncertainty filtering process.
   - *Rejected by Area Filter*: View soundings rejected using a Boundary Polygon filter.
   - *Rejected by Surface Filter*: View soundings rejected through surface filtering.
   - *Rejected by Depth Filter*: View soundings rejected by the depth filter in the Conversion Wizard or the Generic Data Parser when it was converted to HIPS format.
   - *Rejected by Disabled Beam*: View soundings rejected due to a disabled beam status during conversion. This occurs when the beam is disabled in the (legacy) vessel configuration or if the sonar system flagged the beam as a bad detection in the raw data file.
3. Click **OK**.

The rejected data types you selected are displayed in the editors.

Critical Soundings

Three types of critical soundings can be flagged in HIPS:

- *Designated*: soundings that have been flagged in Swath or Subset Editor as being the shoalest sounding in an area.
- *Outstanding*: soundings that have been flagged in Subset Editor as needing further examination.
- *Examined*: soundings that have been flagged in Subset Editor as having been examined and verified

The Designated flag identifies the shallowest (also called “shoalest”) sounding on important features. For example, upright masts on a significant wreck would be flagged as Designated.

The purpose of the Designated flag is to ensure that the shallowest depths over significant seabed features are maintained for standard hydrographic products.

When these soundings are applied to a surface, the surface is regenerated so that the Designated sounding’s depth value is applied to the nearest node (see “Finalize a Surface” on page 270).

Identify Designated Soundings

Designated Soundings can be identified in two ways:

- automatically with the Critical Sounding Detection command (see “Designate Critical Soundings from a Surface” on page 339)
- manually as part of processing in Single Beam, Swath and Subset Editor (see “Find and Designate Soundings in HIPS Editors” on page 344)

Designated soundings can exported to ASCII, CARIS, GSF, and HOB formats.

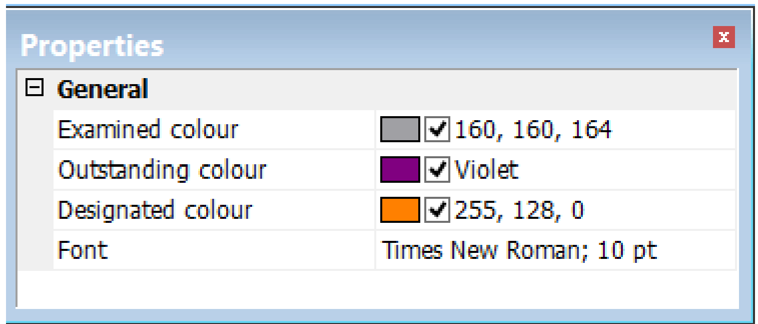
The location of designated soundings can be viewed in the Display window. They can also be viewed in the HIPS editor in which they are designated. (See “View Critical Soundings in HIPS Editors” on page 345 and “View Critical Soundings in the Display” on page 345.)

Menu	Window > Other Windows > Properties

To view and edit the display properties of critical soundings:

1. Select the Critical Soundings layer in the Layers window.
2. Select or open the Properties window.

As displayed below, the properties have different colour settings to indicate the status of selected soundings as Examined, Outstanding or Designated. The *Font* property controls the style and size of the font in which the depth values of the soundings are displayed.


Properties
General
Examined colour
160, 160, 164
Outstanding colour
Violet
Designated colour
255, 128, 0
Font
Times New Roman; 10 pt

Designate Critical Soundings from a Surface

The Critical Soundings Detection tool automatically finds and designates shoal or deep soundings in a surface attribute layer.

The detection process first contours the selected surface. By default, a step interval equal to half the resolution of the surface is used. Contouring can also be done with a user-defined value, or with specific levels.

Then any isolations (closed contours which have no contours inside them) are identified. The direction within these isolations is known (i.e. either shoaling or deepening). Based on the direction within the isolation, a shoal or deep is flagged on the HIPS data. You can define these flags, and possible attribute values as well, in the Critical Soundings Detection dialog box.

As well, radius filtering can be employed to help identify the critical sounding within clusters of soundings around detected features.

Also, a Maximum Isolations for Detection option enables the detection process to stop if a set maximum number is exceeded.

"Critical Sounding Detection" on page 339

"Attribute Configuration" on page 341

"Contours Options" on page 342

"Radius Filters" on page 343

"Change Critical Soundings" on page 348

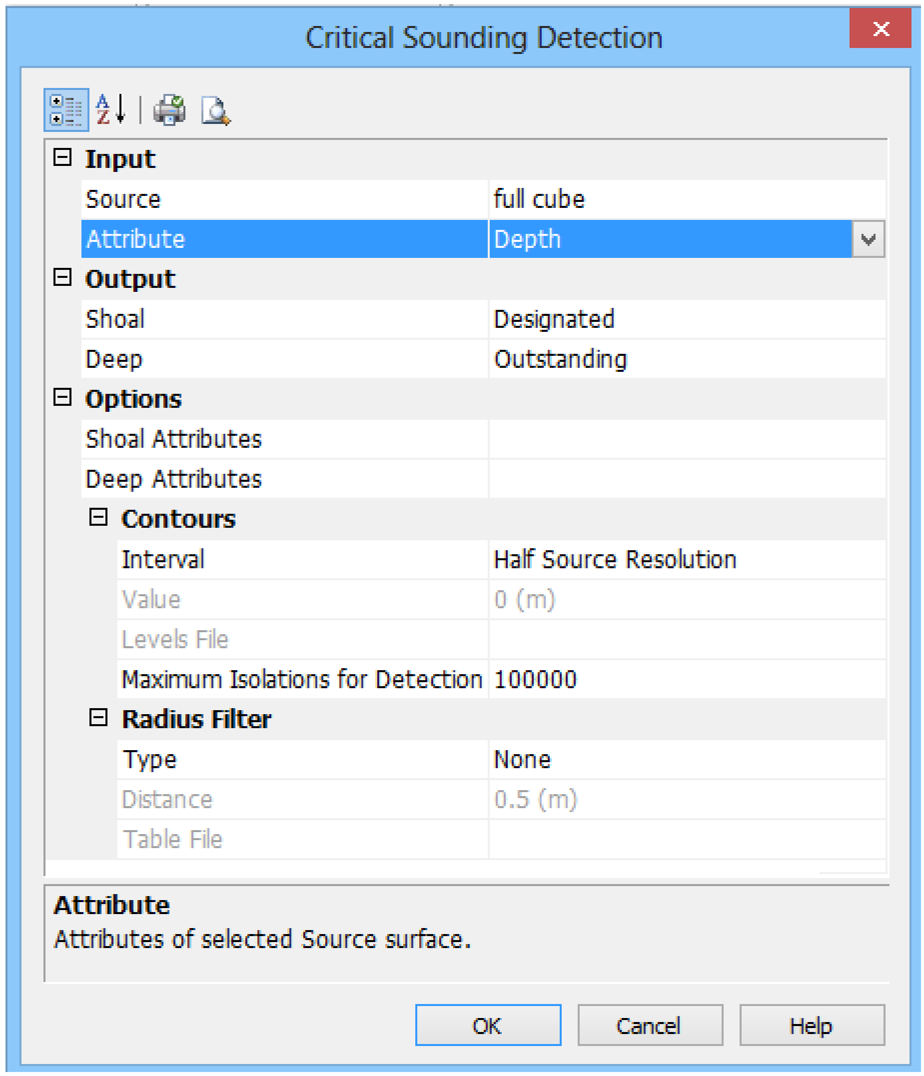
Critical Sounding Detection

To automatically designate soundings as critical:

Menu	Tools > Critical Sounding Detection

1. Open a surface.
2. Select the Critical Sounding Detection command.

The Critical Sounding Detection dialog box is displayed.



In the dialog box, define the *Input* to the process:

3. Click in the *Source* field to select a surface by name from the list of surfaces currently open in HIPS.
4. Click in the *Attribute* field to select a surface attribute layer, such as Depth, from the drop-down list.

Next, define the *Output*:

5. Click in the *Shoal* field to select Designated, Outstanding or Examined from the drop-down list. This will determine which soundings will be flagged.
6. [Optional] Click in the *Deep* field and do the same.

At least one Output type needs to be set to run the process.

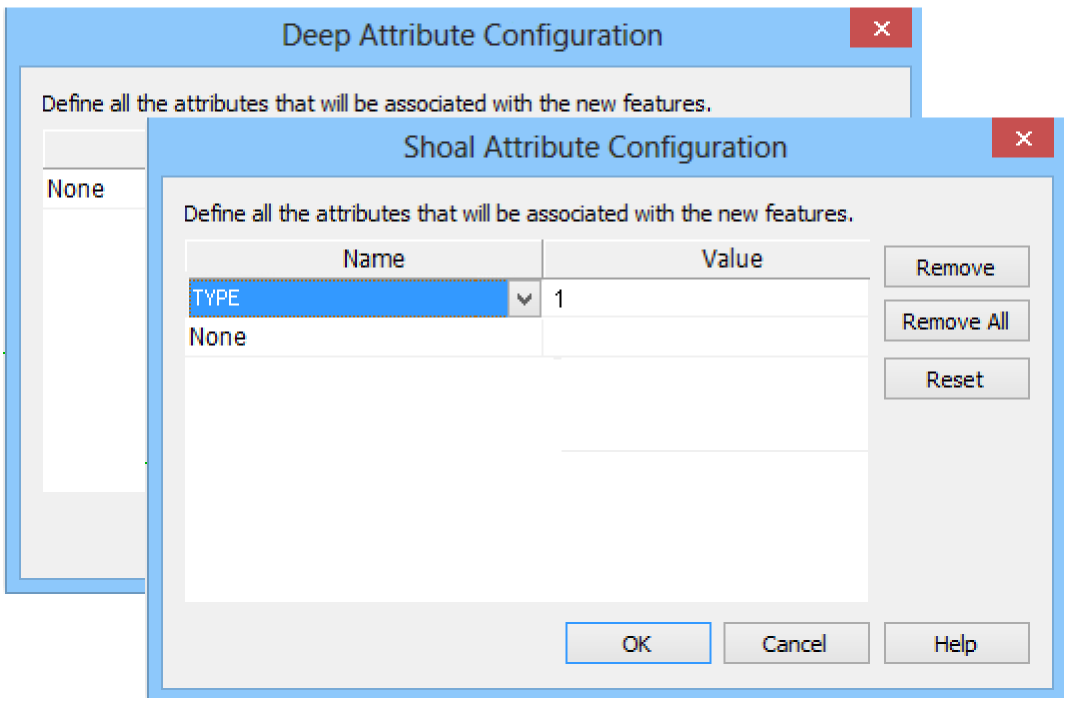
7. [Optional] Set attributes associated with the detected soundings. See "Attribute Configuration" on page 341 for more detail.
8. [Optional] Set contour intervals or filters to detect soundings more precisely. See "Contours Options" on page 342

9. Click **OK** to run the detection process with default settings and no radius filtering.

Attribute Configuration

Additional user-defined attributes can be set for Shoal and Deep soundings, to be applied to detected soundings when the process is run.

1. Click in the *Shoal Attributes* field to open the Shoal Attribute Configuration dialog box.



Attributes in this list and their values are defined in the Catalogue Editor and contained in the catalogue for the current project. (See "Catalogue Editor" on page 116 of the Tools guide.)

In the image above, an attribute called "Type" has been selected from the drop down list. If this "Type" were to represent, as an example, the type of process which designated the sounding, then the Value that would be set could be either 0 if detected automatically, or 1 from manual processing.

The attributes defined will be listed in the Shoal Attributes field in the previous dialog box, when you click **OK**. When the Detection process is run, the attributes and their set values will be saved with the detected soundings.

2. Click in the Name field and select an attribute from the drop-down list.
3. Click in the Value field to set a value.

To clear an added attribute row, click **Remove.** Remove All will clear all the entries from this dialog box and from the Advanced Options fields.

Reset will reset this list to the entries listed when the Shoal Attributes Configuration dialog box was opened.

4. Click **OK** to return to the Critical Soundings Detection dialog box.
5. [Optional] Repeat the process to set additional attributes for Deep.

Contours Options

By default, the detection process applies a contour interval, between a minimum and maximum depth, at value equal to half the resolution of the surface.

> For instance, if a surface has a min/max depth of 5/40 metres, with a resolution of 1 metre, the process will contour from 5m to 40m at every 0.5m, resulting in contours at 5.0, 5.5, 6.0 …, 39.5, 40.0.

You can optionally set a specific value as the contour interval, or use a levels file. Contour levels files, or depth list files, are text files that contain a list of specific depths for contouring data. In a level file, values are negative when representing heights above the datum, and positive when representing depths below the datum. (See "Depth Range Files" on page 485 for an example.)

6. From the drop-down list in the *Interval* field, select either:
   - Half Source Resolution: the interval is a value equal to half the resolution of the selected surface (default)
   - User-defined Value: apply a set value to use as the contour interval
   - Levels File: apply a depth list file

If you select the User-defined Value option to apply to the contour interval, the *Value* field is activated. In this case:

> Type a value for the distance between contours.
>
> This value will be displayed using the default *Horizontal Length* units as set in Tools > Options > Display > Units.

If you select the Levels File option, the *Levels File* field and its Browse button are activated. In this case:

> Browse to the location of the contour levels file, and select the file to apply.

The Maximum Isolations for Detection option enables the detection process to stop if a set number of detected isolations is exceeded. (Isolations are closed contours which have no contours inside them.)

7. Set a value for the maximum isolations or accept the default value of 100,000.

Radius Filters

The Critical Soundings Detection function can result in clusters of critical soundings around detected features. To more precisely designate the critical sounding, radius filters can be used to define an area to filter such clusters.

The filter will be applied to whichever sounding type you have selected in the *Output* field. Shoal and Deep can both be set, and detected soundings of each type will be filtered separately.

You can set a single value as the radius of the area around the features you want to filter, or use a text file containing a table of values for:

minimum depth of the range

maximum depth of the range

minimum radius between soundings for the depth range.

(See "Sounding Radius Table" on page 479 for an example.)

From the drop-down list in the *Type* field, select:

- None - no filtering is applied
- Distance - applies the radius filter value set in the *Distance* field below.
- Table File - applies a file containing a table of values

If you set Distance as the type of filter, the *Distance* field is activated.

8. Type the length of the radius

   This value is displayed in the default *Horizontal Length* units as set in Tools > Options > Display > Units.

If you select Table File as the type of filter, the field is activated.

9. Browse to the location of the filter table file. Click **OK** to run the detection process.

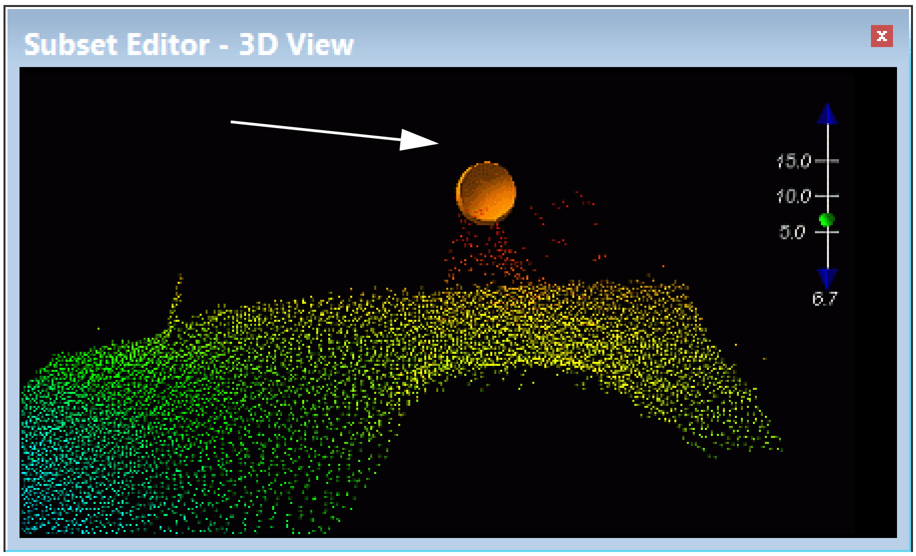
Find and Designate Soundings in HIPS Editors

Menu	Edit > Status Flag > Designated
Tool	
pop-up	<Designated>
Key	<D>

To flag a shallow sounding as the shoalest:

1. Open either Single Beam, Swath or Subset Editor, and load data.
2. Select the shallowest sounding from a cluster of soundings around a shoal feature.
3. Select the Designated command.

The sounding will be flagged as Designated in the Selection window, and highlighted in the editors by a special symbol (as shown in the zoomed-in view below).

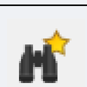


Find Designated Soundings

The Find Designated command automatically selects the shallowest sounding in cluster of highlighted soundings. This feature reduces the time needed to designate shallowest soundings.

To automatically find and designate the shallowest sounding:

1. Use the cursor to highlight soundings in Swath, Single Beam or Subset Editor.
2. [Optional] Use the Query command to view the soundings in the Selection window.
3. Select the Find Designated command.

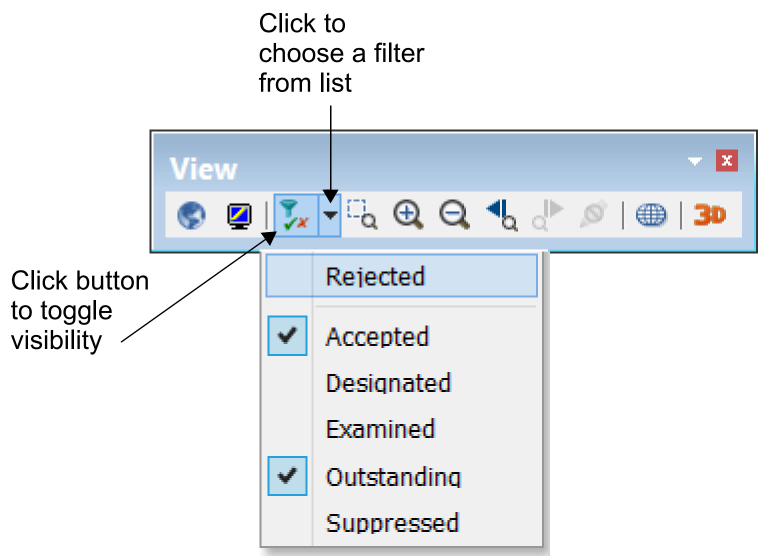
Menu	Edit > Status Flag > Find Designated
Tool	
pop-up	<Find Designated>

Menu	View > Display Filter
Tool	
Key	<F4>

View Critical Soundings in HIPS Editors

To view critical soundings in an editor such as Swath Editor, use the two-part Display Filter button on the View toolbar. This control will display data in Designated, Outstanding and Examined flags.)

Use the right half of the button to set which data to display. Click the left half of the button to display the selected data type. When the button is not active, all data is displayed.



For example, to toggle between displaying all the data open in a HIPS editor, and displaying only Designated soundings:

1. Open data in an Editor.
2. Click the arrow part of the Display Filter button and select the *Designated* check box from the drop-down list.
3. Depress the Display Filter button to make the Designated data visible.
4. Click the button again to see all data.

View Critical Soundings in the Display

Critical soundings that have status flags set during subset and swath cleaning are contained in a HIPS Data layer in the Layers window.

To view these critical soundings in the Display window:

1. Select the Critical Soundings layer in the Layers window.
2. Refresh the display.

The Designated, Outstanding and Examined soundings will be displayed.

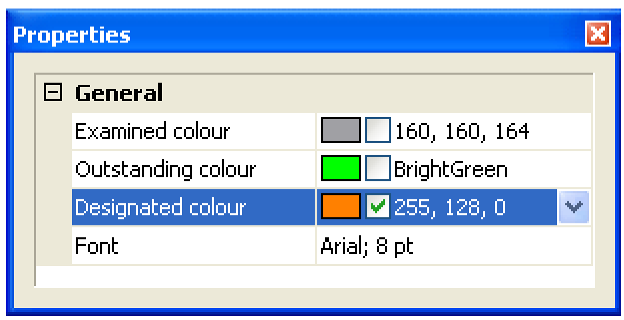
Each type has a default display colour, based on status. These colours by status are set in the Display window of the Tools > Options dialog box. See "Status" on page 660 in the Reference guide.

Properties

You can over-ride these default colours using the Properties for the Critical Soundings layer. Settings changed in the Properties window are retained in the session file.

1. Highlight the Critical Soundings layer in the Layers window.
2. Open the Properties window.
3. Highlight a status field (in the example below, Designated is selected).
4. Set a new colour from the drop-down list.
5. Refresh the Display.

These settings are saved when you save your session, and will be applied when the session is re-opened.



To limit the type of critical soundings visible in the Display window, for example, to display only the Designated soundings:

1. Highlight the Critical Soundings layer in the Layers window.
2. Open the Properties window.
3. Remove the check mark from the check box for the status types you don't want to view, leaving the desired status type active (as in the illustration above).

To change the type face and font size of the soundings:

4. Highlight the Font field in the Properties window, and click the browse button to select from the standard Font dialog box.

These settings are saved when you save your session.

Outdated layer

If changes are made to the Critical Soundings layer, the icon for that layer will indicate that the layer is outdated.

To update the Critical Soundings layer:

1. Right-click on the layer, and select Regenerate from the pop-up menu.

(See also "Subset Cleaning" on page 389 and "Swath Cleaning" on page 372.) For more information on sounding status flags, see "Status Flags and Values" on page 664 in the Reference Guide.

Protect Critical Soundings

You can set a protective sphere around critical soundings to protect any other soundings within the sphere from being rejected when you apply automated filters.

The size of this protective area is determined by setting a radius. You can also select which type of critical soundings the protection is applied to.

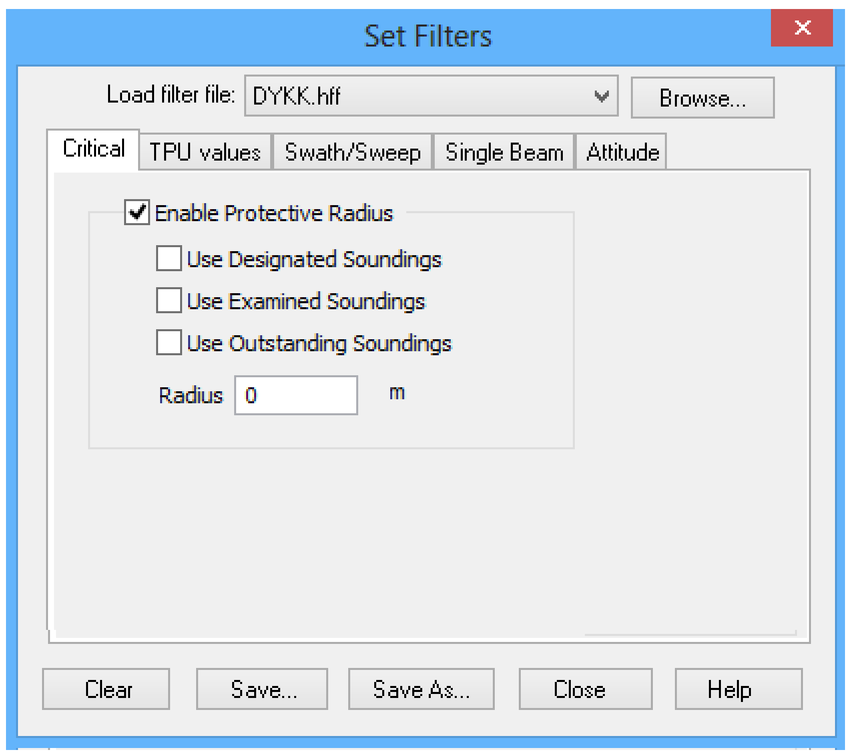
Data must be merged before critical sounding filtering is applied.

Menu	Tools > HIPS Data Filters > Settings
Tool	

To prevent the neighbours of critical soundings from being rejected during automated filtering:

1. Select the Set Filters command.

The Set Filters dialog box is displayed.



2. Select the Critical tab.
3. Select the *Enable Protective Radius* check box.
4. Select which type(s) of soundings to apply the radius to.
5. Type the dimensions of the protective area in the *Radius* field. This value is in the default units set in Tools > Options > Display > Units.
6. Click **Save** or **Save As** to save your filter file so it can be applied with the Apply Filters command.

Change Critical Soundings

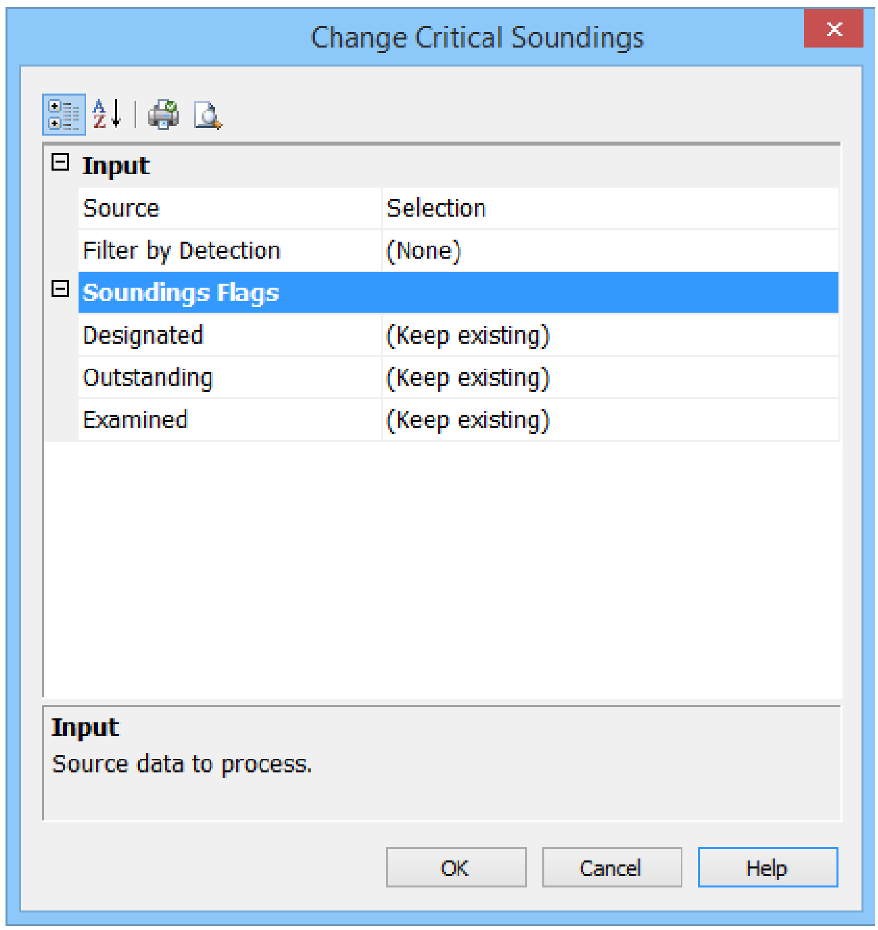
The status flags on soundings designated as critical can be changed to other flags, for example, Designated can be set to Outstanding or Examined. These soundings can also be set to Accepted or Rejected.

To change the status of critical soundings:

Menu	Edit > Change Critical Soundings
Pop-up	Change

1. Select a Critical Soundings layer in the Layers window
2. Select a Change command.

The Change Critical Soundings dialog box is displayed.



Input

1. In the *Source* field, select either Selection or All Critical Soundings.
   - The Selection option is only available if soundings are selected prior to opening the dialog box. If no soundings are selected, all critical soundings open in the data will be changed.
2. From the Filter by Detection field, select the method by which the critical sounding flags were set to Designated, Outstanding or Examined:
   - Manual - individual soundings were assessed and flagged in a HIPS and SIPS editor

- Automatic - sounding flags were set using the Critical Soundings Detection command on the Tools menu.

Soundings Flags

3. In each or all of the Soundings Flags fields, select the flag you want the critical soundings set to, or keep existing.
4. Click **OK**.

Changes to critical soundings are logged and can be viewed in the process Log Viewer.

Restart Cleaning

Edits made to sounding, attitude, and navigation data using the various data cleaning tools can be undone by using the Restart Cleaning function. The Restart Cleaning function resets status flags for each selected track line from “Rejected” to “Accepted”.

You can set which editing process will be reversed by selecting the appropriate check boxes in the dialog box.

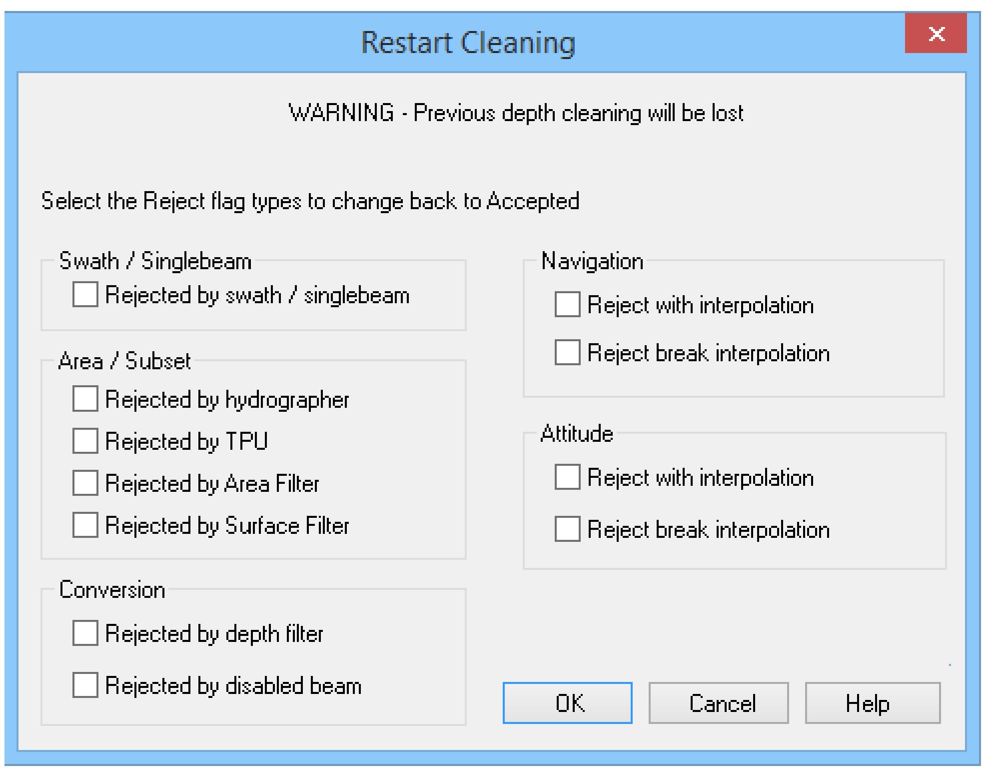
> Restarting cleaning will set depth flags to original values.
>
> All previous depth cleaning is lost once this command is implemented.

Menu	Edit > Restart Cleaning
Tool	

To reset Rejected status flags to Accepted and restart the editing process:

1. Select the track line(s).
2. Select the Restart Cleaning command.

The Restart Cleaning dialog box lists the types of criteria by which data can be rejected.



3. Select any or all of the check boxes to select a rejected data criteria.

Criteria that rejected data	Data reset to "Accepted"
Rejected by Swath Editor or Single Beam Editor	Any sounding data that was rejected in Swath or Single Beam Editor, or by the Swath or Single Beam filters.
Rejected by Hydrographer	Any sounding data that was interactively rejected in Subset Editor.
Rejected by TPU	Data that was rejected during Total Propagateda Uncertainty (TPU) filtering.
Rejected by Surface Cleaning	Data that was rejected during Surface Cleaning.
Rejected by Surface Filter	Data that was rejected during CUBE filtering.
Rejected by depth filter	Data that was rejected by the depth filter during conversion.
Rejected by disabled beam	Data that was rejected due to the disabled beam flag during conversion.
Rejected with interpolation (Navigation)	Any navigation data rejected with interpolation in Navigation Editor.
Rejected with break Interpolation (Navigation)	Any navigation data rejected without interpolation in Navigation Editor.
Rejected with Interpolation (Attitude)	Any attitude data that was rejected with interpolation in Attitude Editor.
Rejected break Interpolation (Attitude):	Any attitude data that was rejected without interpolation in Attitude Editor.

4. Click **OK**.

The data in the selected track lines select is now flagged as Accepted.

Quality Control Reports

The Quality Control (QC) process in HIPS is typically used to statistically compare soundings recorded from check lines against selected attribute values from a surface in the same survey area.

Statistical variance between the check-line soundings and the attribute values are shown in a table in the Reports window once the Line QC report command is run.

To create a Line QC report:

1. Open data.
2. Select a survey line in the Display (or you can select a Project, Vessel, or Day folder in the Project window).

Menu	Process > Report > Line QC

3. Select the Report command.

This will open the QC Report Wizard.

"Line QC Report - Step 1" on page 352

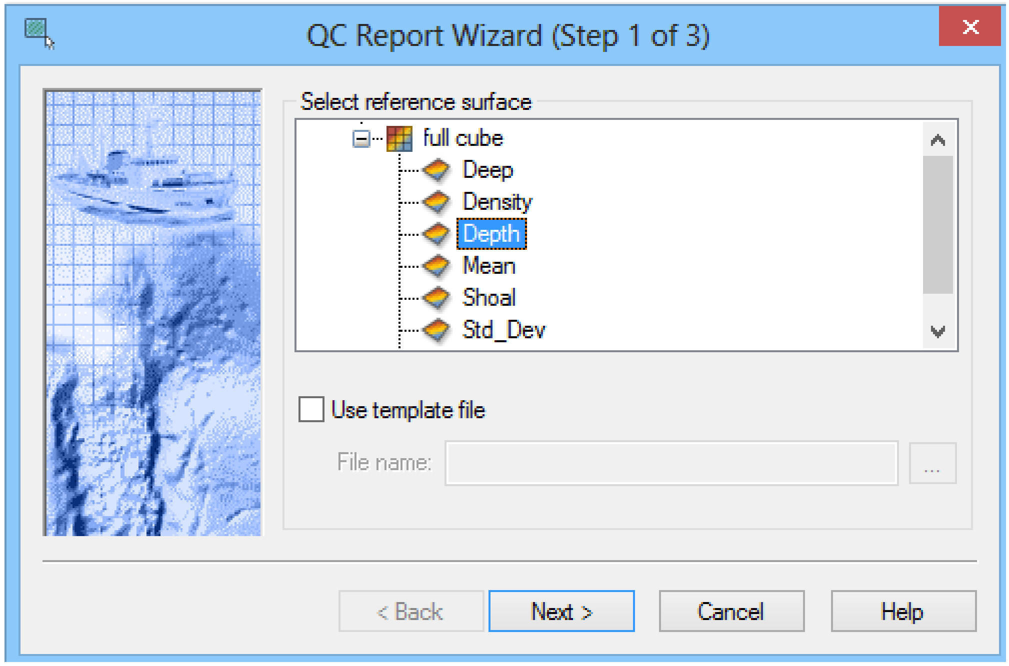
"QC Report - Step 2" on page 353

"QC Report - Step 3" on page 355

Line QC Report - Step 1

Use the QC Report Wizard (Step 1) dialog box to select the surface to be examined. By selecting ***Use template file*** you can create a report template for future use, or select an existing template for the current report.

Template files in XML format are saved to the Templates folder. A template is created using the settings in the wizard used to create the output file. If choose to use an existing template file, the settings saved in that file are applied to the options in the following step of the wizard.

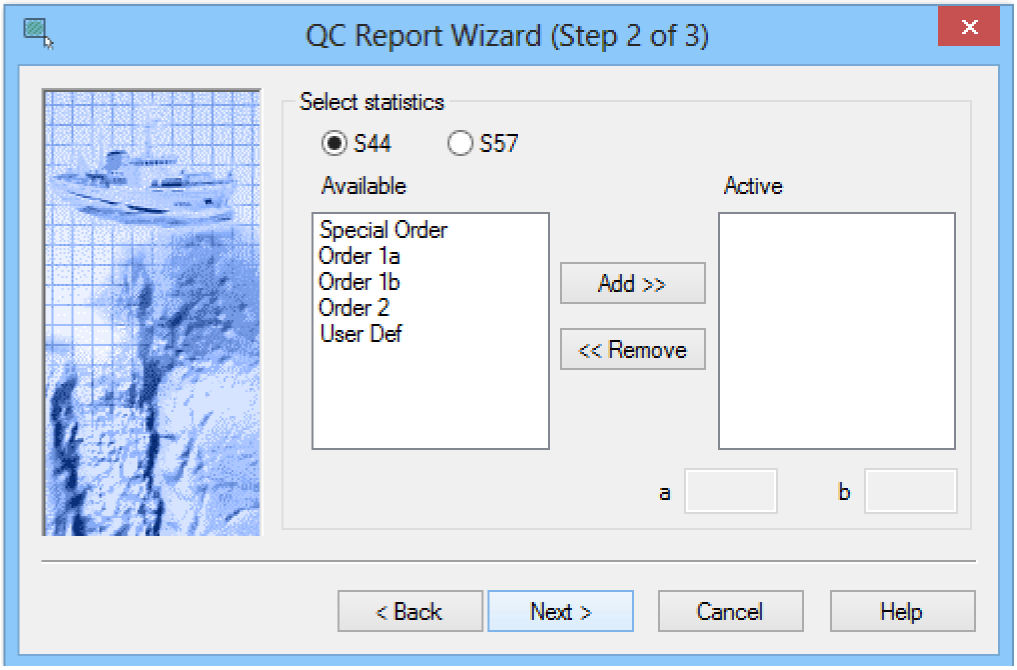


1. Select a surface attribute from the *Select Reference Surface* list.
2. Select the *Use template file* check box to create a new, or to open an existing, template file.
3. Click **Browse** to set a location and name for the new file or to select an existing file.
4. Click **Next**.

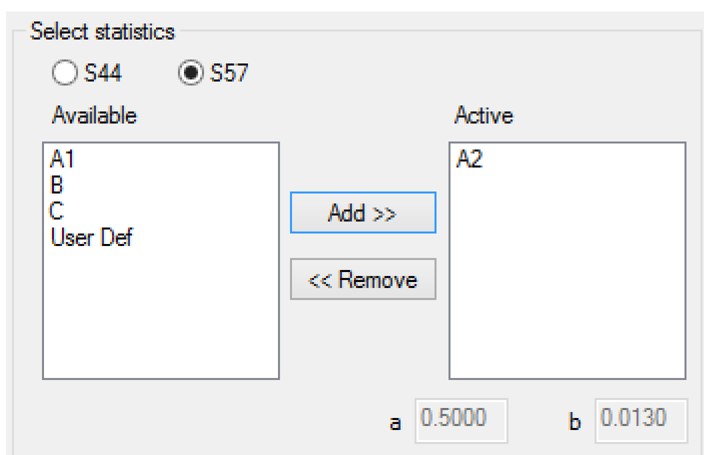
QC Report - Step 2

The QC Report - Wizard Step 2 dialog box is used to select the IHO standard for determining what percentage of soundings fall within a selected error limit for depth accuracy.

You can use the S-44 Ed. 4 survey orders or S-57 CATZOC zones of confidence. (For more information on how error limit for depth accuracy is calculated, see "TPU Filtering" on page 190).



S44 is selected by default. If you select *S57*, the dialog box is refreshed to show the s-57categories relevant to S-57CATZOC zones of confidence.



1. Select *S44* or *S57*.
2. To display a category in the QC report, select it in the *Available* list and click **Add** to move to the *Active* list.

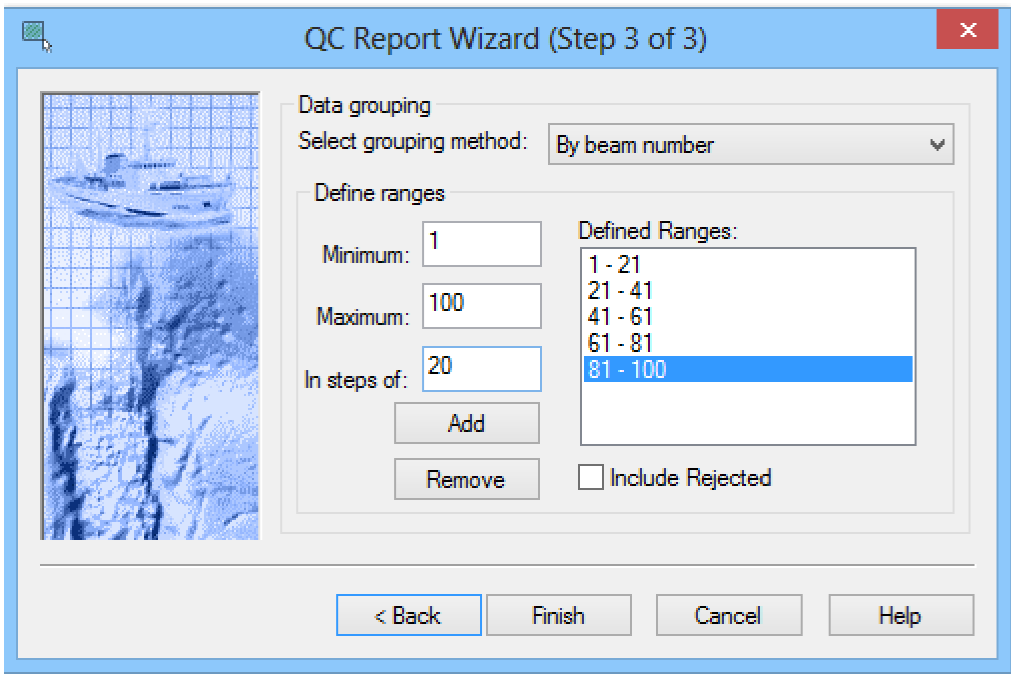
If you are using the user defined (*User Def*) category, the *a* (constant depth error) and *b* (factor of the depth dependent error) fields become active once it is moved to the *Active* list.

3. [Optional] Type the values for the user defined category in the *a* and *b* fields.
4. To remove a category, select it in the *Active* list and click **Remove** to move it back to the *Available* list.

5. Select a category and click the up or down arrow buttons at the top of the *Active* list to determine its order in the *Active* list. This will determine the order that data is displayed in the Report window when the QC Report is generated.
6. Click **Next**.

QC Report - Step 3

The QC Report - Wizard Step 3 dialog box is used to define how soundings are grouped in the QC Report.



There are three options available:

- *Beam number*: Soundings are grouped according to across-track beam number.
- *Angle*: Soundings are grouped according to beam angle from nadir.
- *Distance*: Soundings are grouped according to across-track distance from nadir.

1. Select a method for grouping soundings from the drop-down list.

Soundings can be further organized by range. The range values must correspond to grouping method selected above.

2. Type the range in the *Minimum* and *Maximum* fields.

Ranges are incremented according to the value entered in the *In steps of:* field. For example, for beam numbers 1 to 100 with an increment of 10, the beams are then displayed in the QC Report window in 10 rows.

3. Type a value to increment the ranges.
4. Click **Add** to display the range in the *Defined Ranges* list.
5. To remove a range, select it in the *Defined Ranges* list and click **Remove**.
6. To include rejected soundings, select the *Include Rejected* check box.
7. Click **Finish** to complete the wizard and close the dialog box, or click **Cancel** to close the wizard without implementing changes.

Quality Control Statistics

The quality control report is displayed in the Report window.

Field	Definition
Beam Number	Range grouping as defined in the third dialog box in the wizard.
Count	Total soundings in range.
Max	Maximum distance of soundings above surface.
Min	Maximum distance of soundings below surface.
Mean	Mean difference of soundings to surface.
Std Dev	Standard deviation of mean differences is presented at one sigma or 68% Confidence Interval.
(Selected S-44 or S-57 categories)	Percentage of soundings that fall within the selected S-44 survey orders or S-57 CATZOC zones of confidence categories.

14

Statistical Surface Cleaning

The surface cleaning function is used to perform area-based statistical data cleaning on multibeam data in HIPS.

The process uses advanced tiling technology for subdividing the entire area into variable size and density cells. It then applies polynomial regression statistics to compute the surface.

In this chapter...

Process

In surface cleaning, tiling is used to subdivide the area into manageable cells where each tile is considered separately during the polynomial regression and statistical threshold testing. The tile split criteria is based either on the density of soundings within the tile, or the tile size, or both.

Tiling is based on either the full extents of the open dataset, or on a selected surface or other data layer. .

For each tile, a polynomial solution is computed using iterative least-squares re-weighted regression. The objective of the regression is to generate a surface defined by a set of polynomial terms, which closely matches most of the real depths. However, since the number of terms defining the polynomial surface are less than the number of points considered, then the points do not fit the surface exactly. Using the residual values of each point as a goodness measure, outliers can be detected by testing for high residual values. See "Polynomial Surface Types" on page 359

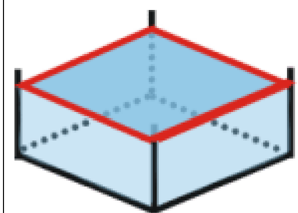
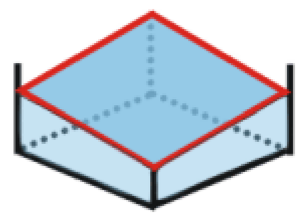
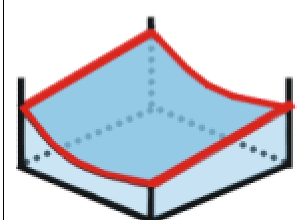
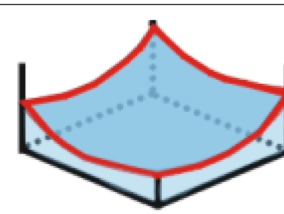
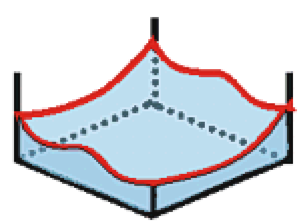
The process is iterative within each tile, and for each new iteration the soundings are re-weighted inversely proportional to their previous residual. You can fix the re-weighting scheme to apply full weight to all soundings within a defined limit from the surface. This re-weighting limit can be based on a fixed distance or a multiple of the standard deviation of the residuals within the tile.

You can also define the maximum number of iterations. The buffer around each tile determines how much extra data is used to compute the tile polynomial surface. This helps eliminate edge effects during statistical testing.

The second stage of surface cleaning is the testing of the residuals once the surface has been computed. The relevant parameters are the rejection threshold scaling values in both the positive and negative directions (above and below the surface), and the minimum absolute residual of an outlier. The latter parameter specifies that no point within this distance (either above the surface or below) are rejected. This is an important parameter if the local standard deviation of a tile is small, because the scaling values are multiplied by a small standard deviation and may reject points which are only slightly removed from the surface. This parameter is location dependent and maybe used, for example, to prevent the rejection of rocks of a specific height. The rejection threshold values are also location dependent and surface dependent..

Polynomial Surface Types

The following table lists each of the polynomial surface types with the corresponding equation and a graphical example of each surface type.

Surface parameter	Equation	Graphic representation
Mean	$f(xy) = b0$	
Tilted Plane	$f(xy) = b0 + b1x + b2y$	
Curved Tilted Plane	$f(xy) = b0 + b1x + b2x + b3xy$	
Quadratic	$f(xy) = b0 + b1x + b2x + b3xy + b4x2 + b5y2$	
Cubic	$f(xy) = b0 + b1x + b2x + b3xy + b4x2 + b5y2 + b6x2y + b7xy2 + b8x3 + b9y3$	

Surface Cleaning

The surface cleaning process uses tiling to subdivide the entire data area into variable size and density cells, and then applies polynomial regression statistics to compute the surface.

This kind of statistical surface cleaning can be applied either to the extents of data, to a feature layer or to a selected surface.

For the background to this process see "Process" on page 358.

> Data must be merged before surface cleaning can be performed.

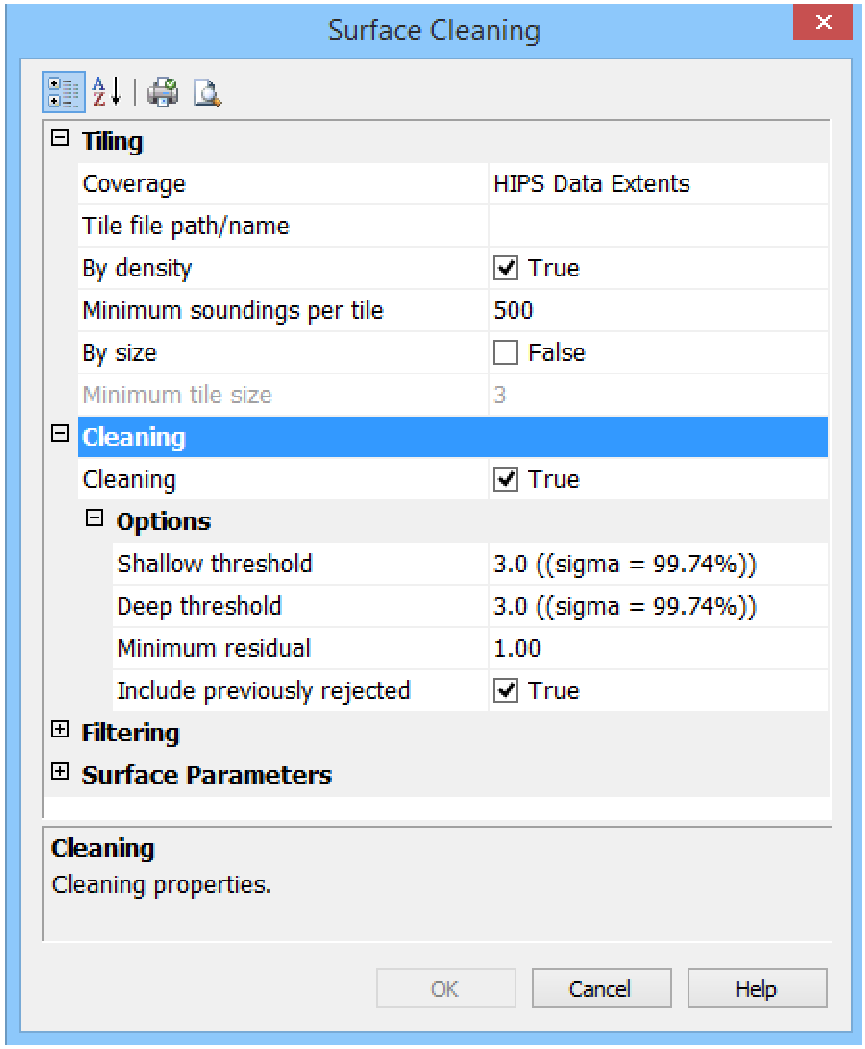
To apply surface cleaning:

1. Open the data to be processed. If using a surface as the area for cleaning, open the surface.

Menu	Tools > Surface Cleaning

2. Select the Surface Cleaning command.

The Surface Cleaning dialog box is displayed.



Tiling

Surface Cleaning cannot be applied unless tiling is done first. The **OK** button to run the process does not become active until the tile file name and location are set.

All tile files are assigned an mts extension.

1. In the *Coverage* field set the area to be cleaned.
   - By default, HIPS Data Extents is selected.
   - To use an open surface, select it by name from the drop-down list.

To set the tile name:

2. Click the **Browse** button.
3. In the Save As dialog box, type a name for the tile file.
4. if necessary, navigate to another folder. Click **Save**.

Tiling takes the largest dimension of the surface or data extents and then applies one or both of these tiling methods:

- Tiling *By Density:* Each tile is continuously sub-divided until one of the sub-tiles has less than the minimum number of soundings.

- Tiling *By Size:* This will continuously sub-divide each tile until the sub-tiles reach the minimum size.

5. Select a tiling method by setting the field to True.
   - If you select *By Density*, type a *Minimum soundings per tile* value for the minimum number of soundings that will be covered by each tile.
   - If you select *By Size*, set the *Minimum size of tile*, in default measurement units.

Cleaning

If the *Cleaning* option is set to False, no soundings will be rejected.

6. Set the *Cleaning* check box to *True* to enable the surface cleaning functions.
7. Set Cleaning options:

Option	Description
Shallow threshold	Set the value of the threshold *above* the polynomial surface. The default value is 3.0((sigma = 99.74%)).
Deep threshold	Set the value of the threshold *below* the polynomial surface. The default value is 3.0((sigma = 99.74%)).
Minimum residual	A sounding that fails the threshold test will not be rejected if the absolute value of the residual is less than the minimum set here. Default value is 1.0.
Include previously rejected	Set to True to include any data previously rejected by Surface Cleaning in this computation.

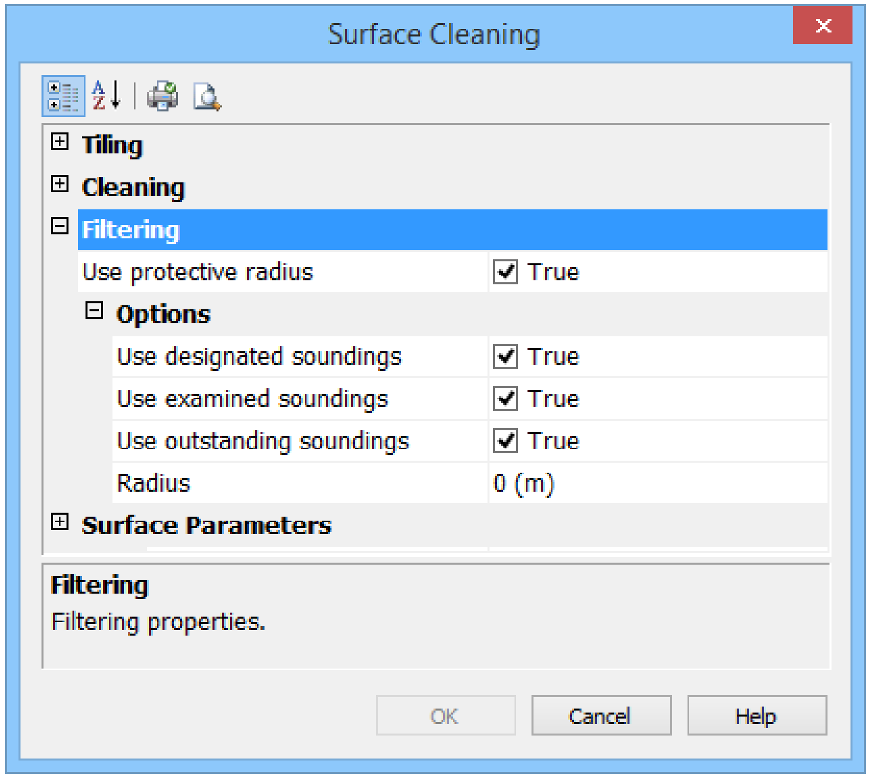
The sigma value is a multiple of the standard deviations of residuals in the tile. The equivalent confidence values are displayed as a percentage.

Filtering

Use the Filtering options to protect critical soundings from being rejected when you apply automated filters. (See “Critical Soundings” on page 337.)

This is done by setting a protective area, defined by a radius, around soundings, and setting which type of soundings are protected. Designated, examined, and outstanding soundings can all be protected.

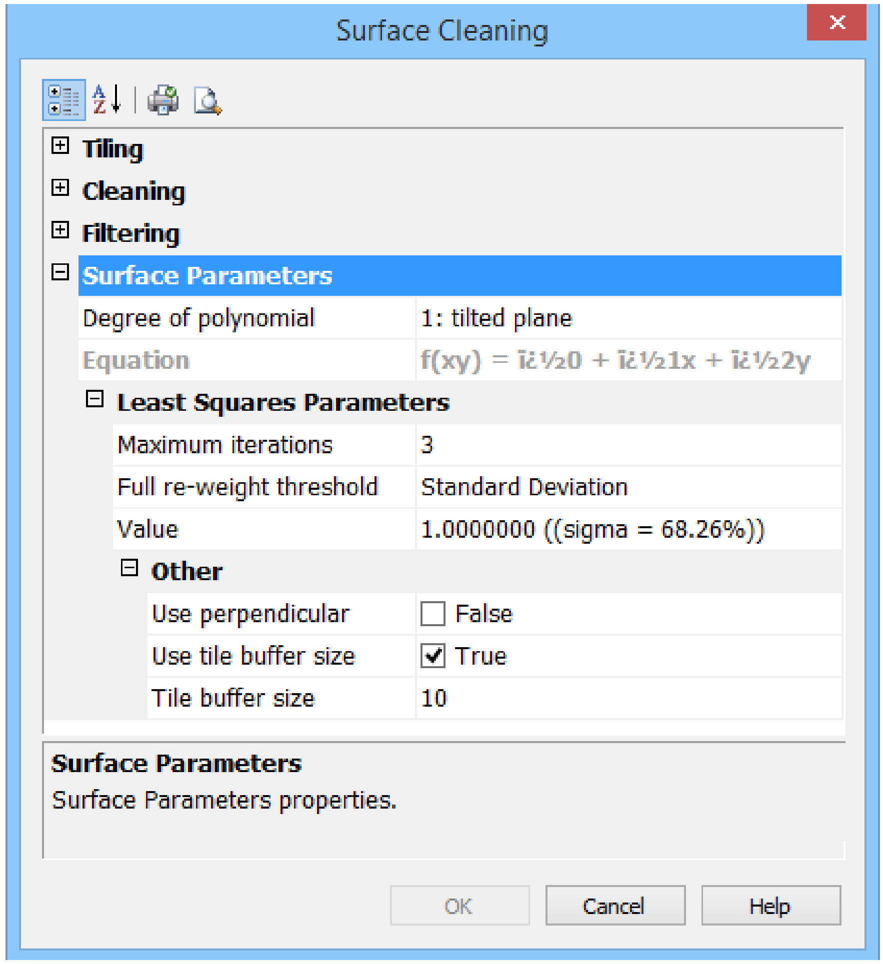
o



8. Set *Use protective radius* to True (Default).

To ensure that soundings of a specific type are included in the surface:

9. Set the *Use ...soundings* option for the type to True.
10. In the *Radius* field, type the length of the radius (in metres) to define the protective area.



Surface Parameters

11. From the *Degree of polynomial* drop-down list, select one of the available polynomials that best matches the expected complexity of the sea floor.
    - The equation for the selected degree will be displayed in the *Equation* field. (Refer to "Polynomial Surface Types" on page 359 for polynomial descriptions.)
12. Set the *Least Squares Parameters*:

Option	Description
Maximum iterations	Set the number of times the least squares computation of the polynomial will process the soundings. This re-weighting continues until there are no additional rejected soundings or the maximum number of iterations is reached. Default value is 3 iterations.

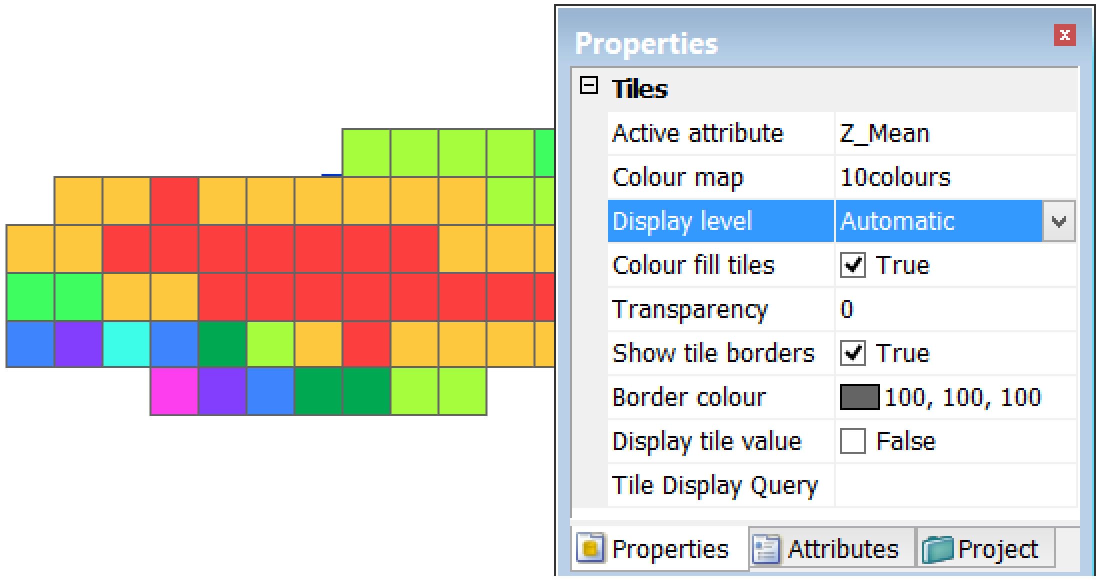
Option	Description
Full re-weight threshold	*Full re-weighted threshold*. Select either by: • *Standard deviation*: The full re-weighting of soundings for each iteration can be a multiple of the standard deviation of the residuals. The *Value* field will show a multiple of the standard deviations of residuals in the tile. The equivalent confidence values are displayed as a percentage.) • *Residual surface value*: The full re-weighting of soundings for each iteration can be fixed as a distance from the surface.
Value	Set the value of the full re-weight threshold selected above. • *Standard deviation*: Default is 1.0 ((sigma = 68.26%)). • *Residual surface value*: Default is 0.3 metres.
Other	Other least square options
Use perpendicular	When this option is set to True, the residual is calculated as the perpendicular distance between the sounding and the nearest point on the polynomial surface. The default calculation for the residual uses the depth difference between the sounding and the polynomial surface at the location of the sounding. Default is set to False.
Use tile buffer size	When this is set to True, extra width around the tile is included in the surface computation. Default is True.
Tile buffer size	Set the size of the buffer around the tile as a percentage of the tile size. Default is 10%.

13.When all options are set, click **OK**.

The tiles are drawn in the Display window and the tile surface is listed in the Layers window, using the file name set in the *Tile file path / name* field.

Tile Display

The tiles are automatically displayed as squares of equal size.



To display the applied tiling:

1. Select the tile layer in the Layers window.
2. In the Properties window, set the Display level to Maximum.
3. Refresh the display.

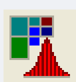
Other display properties of the tiles can be also be set in the Properties window.

Tile Histogram

The Tile Histogram dialog box displays the distribution of residuals within a tile.

If the histogram resembles a bell curve then the selected polynomial is well suited for the area. However, if the histogram has many peaks or a wide centre, then the polynomial surface is not suited to the area. If there are severe shifts in data distribution, then the area may have to re-tiled using a higher degree of polynomial.

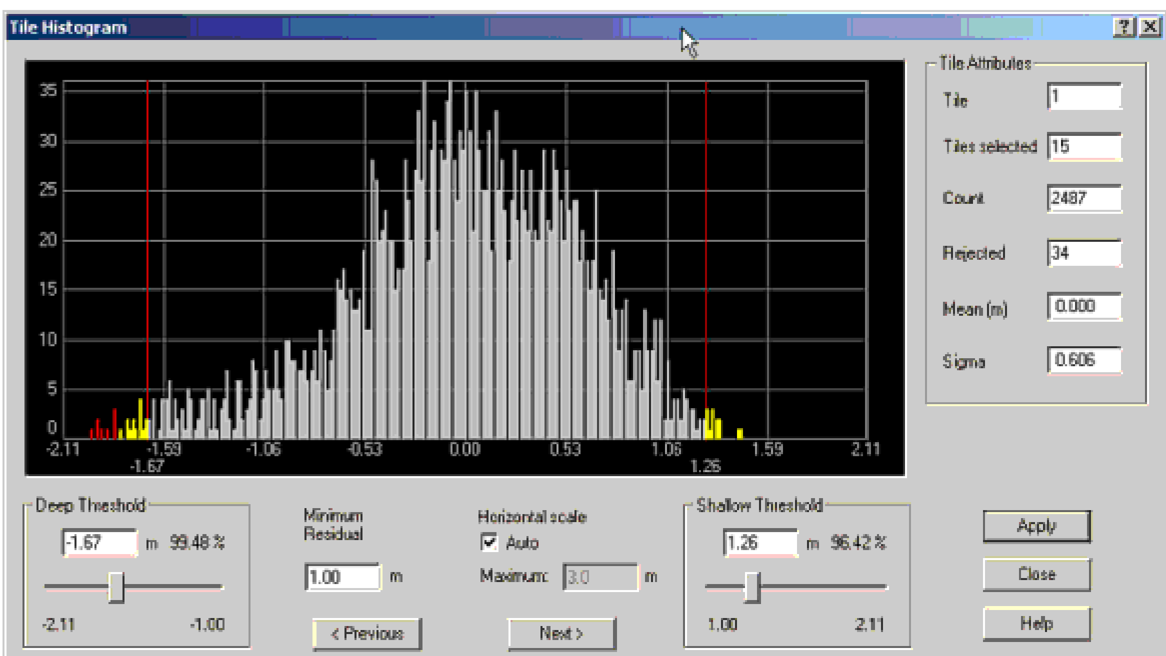
You can also use the histogram to change the threshold values and reject soundings.

Menu	Edit > Tiles > Tile Histogram
Tool	

1. Select the tile layer in Layers window
2. In the Properties window, set the *Display Level* to Maximum.
3. Select the tiles you want to examine.
4. Select the Tile Histogram command.

The Histogram dialog box is displayed.

Attributes of one of the selected tiles is displayed on the right-hand side of the dialog box.



5. To view data for another tile, click **Next Tile** or **Previous Tile** to see the attributes of each of the rest of the selected tiles.
6. Use the *Shallow Threshold* or *Positive Threshold* sliders to adjust these values.

The corresponding confidence intervals and Maximum Residual Distance values change as the bars are moved.

7. Click **Apply**.

The rejected residuals are displayed in red.

Query data in tiles

There are three ways to examine data in tiles:

- "Select and View Tile Data" on page 369
- "Query Soundings in Tiles" on page 369
- "Query Tile Display" on page 370

Select and View Tile Data

Data such as the number of soundings within the tiled surface cleaning area and the size of tiles can be displayed in the Selection window. To see this data:

1. Open surface cleaning tiles in the Display window.
2. Select the tile layer in the Layers window.
3. In the Display window, select tiles to be queried.

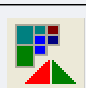
The data for each selected tile is automatically displayed in the Selection window, including:

- X and Y coordinates for the centre of the tile
- Z mean, Z minimum and maximum
- Res mean and Res Sigma
- Count of the number of soundings within the tile area, and the number of Rejected soundings within the tile area
- Tile level and tile size

Query Soundings in Tiles

To view data on the soundings in selected tiles:

1. Open surface cleaning tiles in the Display window.
2. Select the tile layer in the Layers window.
3. In the Display window, select the tiles that lie over the soundings to be queried.
4. Select the Query Tile Soundings command.

Menu	Edit > Tiles > Query Tile Soundings
Tool	

The progress of the query is indicated in the status bar. When finished the data for each sounding in the selected tiles is displayed in the Selection window.

As well as the Project/Vessel/Day/Line data for each sounding, other attributes are also displayed, such as:

- *X* and *Y* coordinates for the soundings
- Profile, Beam and Status data

- *Depth*: depth of sounding.
- *Surface Depth*: depth of polynomial surface at the location of the sounding.
- *Residual*: percentage difference between the sounding and the polynomial surface.
- *Shallow Threshold*: depth of rejection threshold above the polynomial surface.
- *Deep Threshold*: depth of rejection threshold below the surface.

Query Tile Display

You can also use the Tile Display Query function to create an SQL-type query that will display only those tiles with selected attributes. For more information, see "Tile Display Query" on page 398.

15

Cleaning Swath Data

Data cleaning in Swath Editor consists of interactively selecting and rejecting soundings and using proven and efficient filtering functions to automatically detect and reject outliers.

You can also flag Designated Soundings.

(For descriptions of the Swath Editor interface and controls, see "Swath Editor" on page 843 in the Editors guide.)

In this chapter...

Swath Cleaning

During data conversion, the original data file is separated into a number of distinct files, each storing a different type of information. All are related by time. One of these files is the Observed Depths file. This file contains measured single beam, sweep, or swath soundings as reported by the data acquisition system.

The Swath Editor displays the contents of the file for sweep and swath systems in the Display window in graphic form so you can perform cleaning and filtering operations on the data.

Each soundings begins with Accepted status. If rejected (for any reason) during processing it can be restored to Accepted status without loss of data.

Sounding outliers can be rejected by applying automatic swath filters (see "Automatic Filtering" on page 373) or interactively using manual cleaning tools (see "Manual Cleaning" on page 377).

For information on the Swath Editor interface, see Swath Editor in the HIPS and SIPS Editors Guide.

Automatic Filtering

When dealing with high-volume bathymetric datasets, it is advantageous to be able to reject outliers automatically.

Using Swath filters can be an effective way to reduce the time it takes to clean large volumes of data.

There are three basic types of swath filters:

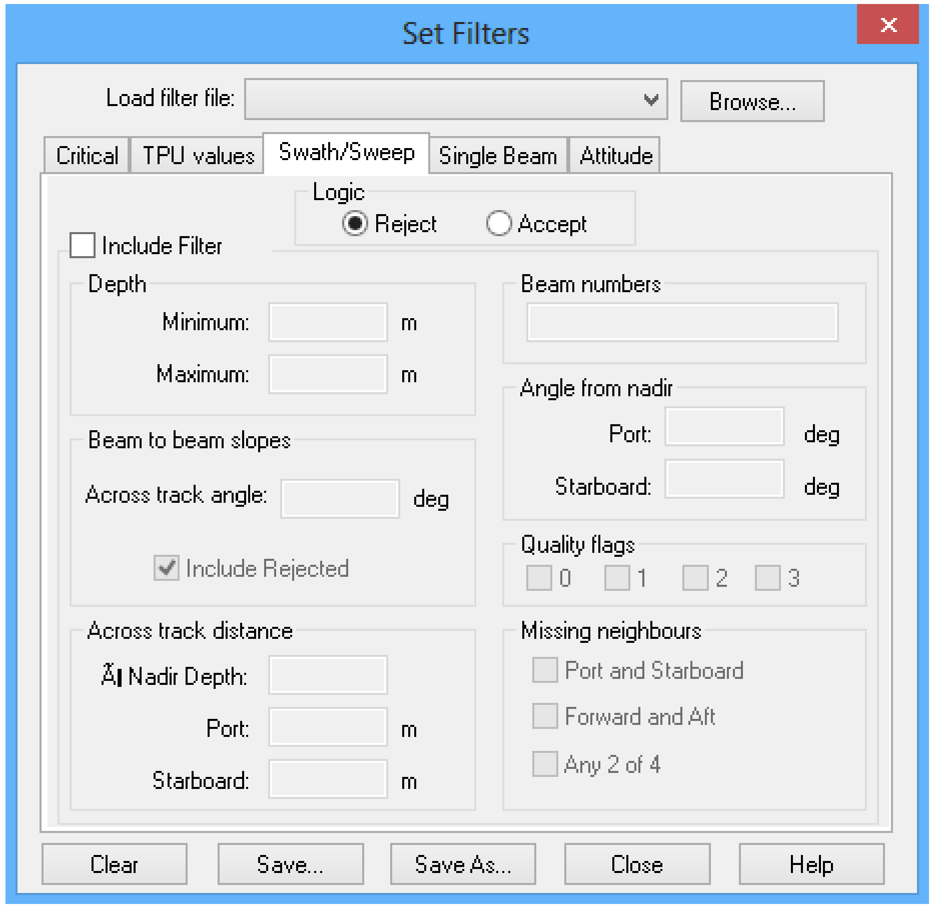
- **Spike detection**: The minimum and maximum depth filters and the beam-to-beam slope filters.
- **Swath reduction**: across track distance filters and nadir angle filters.
- **Sonar quality**: Quality values set by the sonar and missing neighbours

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

Menu	Tools > HIPS Data Filters > Settings
Tool	

1. Select the Set Filters command.

The Swath/Sweep tab of the Set Filters dialog box is displayed.



1. [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 the values used in that file are shown in the fields. You can filter data or change any of the values in the fields.

2. Set the logic to *Accept* or *Reject*.
3. Select the *Include Filter* check box to activate the filtering options.
4. Select the filter parameters used to flag data (see below).

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.

5. Click **Close**, or click **Clear** to remove data from the fields.
6. Click **Save As** to save the settings to a filter file.

The following filters are available.

- *Minimum Depth*. Reject any soundings that are shallower than the depth specified (e.g., 10 m).
- *Maximum Depth*. Reject any soundings that are deeper than the depth specified (e.g., 50 m).

- *Across track angle* Beam-to-beam slopes, across track. For each beam on the same swath, calculate the slopes in degrees to the prior and post beams, and if both slopes exceed the defined value and are of opposite sign then reject the beam.

  The *Include Rejected* option includes previously rejected soundings when recalculating the slopes during multiple runs of the filter.

- *Across track distance: X Nadir Depth*. Any soundings with an across track distance, from the centre beam, greater than the nadir depth times this number (e.g,. 3.0) are rejected. For example, in 30 metres of water, with a setting of 3.0, all soundings with an across track distance greater than 90.0 metres are rejected.
- *Across track distance: Port*. Filter a sounding if the across track distance on the port side is greater than the value entered in this field.
- *Across track distance: Starboard*. Filter a sounding if the across track distance on the starboard side is greater than the value entered in this field.
- *Beam Numbers*. Specify beams to be rejected. Separate individual beams with a space and specify a range of beams with a hyphen. For example, entering 12 32 56-60 72 rejects beams 12, 32, 56, 57, 58, 59, 60, and 72.
- *Angles from Nadir: Port and Starboard*. Filter beams that are outside of the designated angles (e.g., 60 degrees). Angles are computed from the nadir using depths and across track distances (roll corrected).
- *Quality Flags*. If your system attaches quality codes to soundings, and those codes are transferred to HIPS during conversion, a sounding can be rejected according to its value.

> Quality flags are currently supplied by Teledyne Reson, Elac, Hypack, and GSF multibeam systems/formats.

Missing neighbours are beams rejected during the conversion process into HIPS. This occurs if the beam is disabled in the vessel configuration or if the sonar system flags the beam as a bad detect in the raw data file.

- *Port-Starboard*. Beams are rejected if beams on each side of the swath are rejected.
- *Forward-Aft*. Beams are rejected if the adjacent beams in the previous and next swath are rejected.
- *Any 2 of 4*. Beams are rejected if two of the four neighbouring beams are rejected.

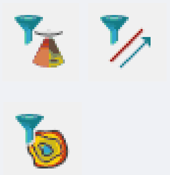
During filtering, only accepted soundings are examined. Soundings rejected after one filter pass are not considered in the next.

"Apply Filters" on page 376

Apply Filters

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

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

Menu	Tool > Apply Filters > 1 Screen/To End of Line/Bathymetry
Tool	

You have four options for applying the filters:

- **Filter 1 Screen**: Apply the filters only to the swaths that are currently visible in the Plan View of Swath Editor.
- **To End of Line**. Apply the filters to the track line currently open in Swath Editor, from the first swath currently visible in the Plan view to the end of the line.

Manual Cleaning

Menu	Tools > Swath Editor> Open
Tool	

1. Select the Ship Track Lines layer in the Layers window.
2. Select a track line.
3. Select an Open Swath Editor command.

The Swath Editor opens with all views displayed.

(For a complete description of the Swath Editor interface, and ways of viewing data display options and controls, see SWATH EDITOR in the HIPS and SIPS Editors Guide.)

4. Select soundings to be cleaned. (When soundings are selected they are highlighted in yellow.)
5. Apply either the Reject or the Reject Swaths command to the sounding as appropriate.

Tools

To assist with manual cleaning, use:

"AUTO CURSOR MODE" ON PAGE 377

"FIND SOUNDING" ON PAGE 377

"QUERY DATA" ON PAGE 380

Auto Cursor Mode

Auto Cursor Mode can be used to speed up the interactive process. This mode combines the selection and Accept/Reject/ Query functions into a single procedure.

Menu	Edit > Status Flag > Auto Cursor
Tool	

1. Select the Auto Cursor command.
2. Select a Reject/Accept/Query command.
3. Press the mouse button and draw a bounding box over the data.
4. Release the mouse button.

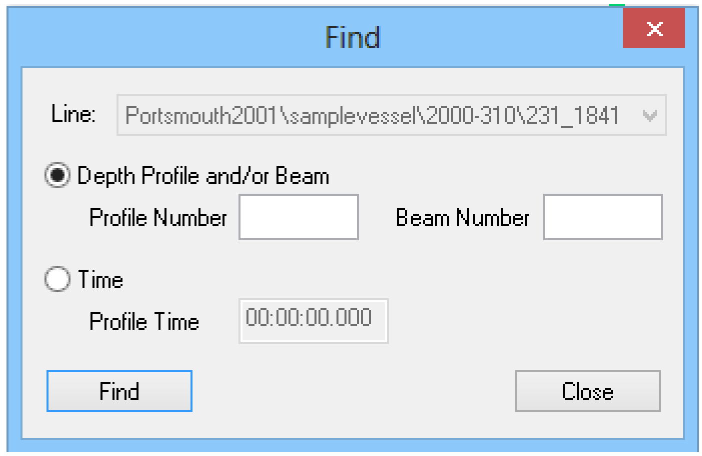
Find Sounding

Use the Find function in Swath Editor to search a survey line for a specific sounding(s) by beam and/or profile number.

Tool	
Pop-up	Find

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

The Find dialog box is displayed.



3. To search by the swath or beam number, select the *Profile and/or Beam* option.
4. Type the swath or beam number in the *Profile Number* or *Beam Number* field.
5. To search for a swath that was recorded at a specific time, select the *Time* option.
6. Click inside the *Profile Time* box and type the time in the relevant fields (hour:minutes:seconds:decimal fractions of a second), using the 24-hour clock.
7. Click **Find**.

The selected beam or swath is highlighted in the Plan, Rear and Side Views.

8. Click **Query** to display the data in the Selection window.

Rejecting Data

A sounding rejected within Swath Editor has the Reject by Swath Editor flag turned on. This flag is displayed in the Status field of the Selection window when the sounding is queried.

To reject a sounding:

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

1. Select the data you want to reject.
2. Select a Reject command.

The Reject by Swath Editor status flag is now on and the rejected sounding may or may not be displayed depending whether the Display Filter function is turned on. (See "View Soundings Status" on page 334 for description.)

A rejected sounding can always be returned to the "Accepted" status without loss of data.

To revert rejected soundings back to "Accept":

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

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

The soundings are now flagged as accepted.

Reject Swaths

In some instances it may be necessary to reject all soundings in one or more swaths. This can be done with the Reject Swaths command without having to select all of the data in the swaths.

Menu	Edit > Status Flag > Reject Swaths
Tool	
Key	<S>

1. Select some sounding in the swath you want to reject.
2. Select the Reject Swaths command.

All soundings in the swath (whether individually selected or not) now have the Rejected by Swath Editor status flag turned on. The sounding will not be displayed unless the Display Filter tool is turned on (see "View Soundings Status" on page 334).

Query Data

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

Select soundings to view status and other information.

1. Select the data in one of the Swath Editor windows.
2. Select a Query command.

The following data fields for the selected soundings are displayed in the Selection window:

- time of the swath profile
- d-time from the previous profile
- profile number
- beam number
- across track distance
- along-track distance
- depth
- d-depth from the previous profile
- d-depth from the previous beam
- amplitude / phase detect
- quality value
- status flag
- horizontal error value
- vertical error value

You can reject queried soundings selected in the Selection window.

1. Query a selection of soundings.
2. Highlight certain soundings in the Selection window.
3. Select an Accept or Reject command.

You can also use the keyboard to set the flag: “A” key for Accept, “Q” for Query, “R” for Reject, and “S” for Reject Swath.

Update Backscatter

This command updates the backscatter imagery for a line after swath cleaning is complete.

Backscatter data is referenced directly to the individual beams of bathymetry data. After soundings have been rejected in Swath Editor, Update Backscatter is applied to remove the intensity values for the rejected soundings from the backscatter image.

Menu	Tools > Update Backscatter

1. Select a track line.
2. Select the Update Backscatter command.

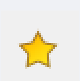
Flag Designated Soundings

The Designated Sounding flag identifies the shallowest, or “shoalest” sounding in a feature. For example, in a cluster of soundings surrounding an outcrop of rocks, the shoalest sounding among that cluster is identified as Designated.

The purpose of the Designated Sounding flag is to ensure that the shoalest depths over significant seabed features are maintained in charts and other standard hydrographic products.

Soundings can be designated in both Swath Editor and Subset Editor.

For more information, see “Critical Soundings” on page 337.

Menu	Edit > Status Flag > Designate
Tool	
Key	<D>

1. Select the shoalest sounding from a cluster of soundings around a feature in Swath Editor.
2. Select the Designate Soundings command.

The sounding is flagged as Designated.

Correction for Refraction Artifacts

Since it is not possible to completely control the sound speed variations in water we apply sound velocity profiles to correct for this. However, refraction artifacts may still remain, especially if incorrect or insufficient sound velocity profiles are applied either during acquisition or during post processing.

To correct for this, you could choose to limit the usable swath angle (and therefore survey more lines), or you can use the Refraction Editor, which simulates the effects of altering the SVP to derive a better refraction solution.

Since refraction errors are most pronounced in the outer parts of the survey line coverage, they tend to create characteristically curved (upward or downward) refraction artifacts.

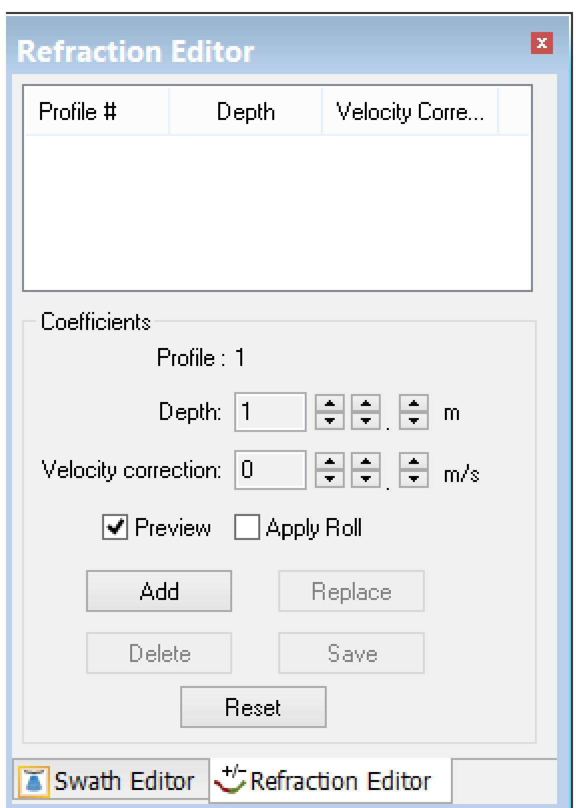
Apply the velocity correction at a user-defined depth. As you adjust the values, you can observe the changes in the shape of the swath profiles. When you arrive at the values which give you the desired correction, save these coefficients. Later, during the Merge process, you can apply these coefficients.

To open the Refraction Editor:

1. Select a track line.
2. Open Swath Editor, then select the Refraction Editor command.

Menu	Tools > Editors > Swath > Refraction Editor

The Refraction Editor window is displayed.



3. Set the values for depth and velocity correction until the desired effect is observed.
4. Click **Add** to add the new entry to the list of coefficients.
5. Click **Reset** before clicking **Add** to set the values to their original setting.
6. To change an existing coefficient, select it, adjust the settings and click **Replace** to update it.
7. To remove an entry from the list select it and click **Delete**.
8. Click **Save** to save your entries.

As you change the values in the Refraction Editor window you will be able to see the effects of these changes in the Swath Editor display.

9. Select the *Preview* check box to view the effects of your changes.

If the *Preview* option is selected, you can also enable the *Apply Roll* option. When you first open the Refraction Editor, this control is set to be the same as the setting in the Roll section of the Vessel file (HVF). This control enables you to quickly observe the effects of applying (or not applying) dynamic roll to the sounding data.

> If there is no roll data, or if the appropriate Roll section cannot be found in the HVF, or if the survey line has already been sound velocity corrected, toggling the *Apply Roll* switch will have no effect.

10. Select the *Apply Roll* check box to see the effects of roll on the data

The velocity correction and depth are recorded with the current swath profile number. While paging through the track line during cleaning, new swath/correction/depth values can be recorded.

Interpolation

When you add the first Refraction Coefficient, the correction starts at that profile and goes to the end of the line with that same correction. If you add a second Refraction Coefficient after the first, the correction applied between the two profiles is interpolated. This is done so that there is no “jump” in the profile when the correction at the second Refraction Coefficient is applied to the data.

This interpolation is linear. That is, if the selected profile is between two refraction entries, the depth and delta velocity are both interpolated from these entries.

Querying

Querying the data in Swath Editor will always give the results from the display, so if you change the display with the *Preview* option of the Refraction Editor turned on, the query will return the results of the change. The Observed Depth data (across track, along track and depth) does not change.

The effects of the Refraction Editor can be turned on or off during Calibration. As well, the effects on the final merged depths can be modified any time by editing the coefficients and re-merging, or the effects can be removed by re-merging and choosing not to apply the coefficients.

16

Process Data in Subsets

Create subsets to efficiently visualize and clean data from multiple track lines at the same time. Subset tiles to track the progress of processing.

In this chapter...

Data Cleaning Workflow

A subset is a user-defined rectangular area that contains some of the soundings of a survey project. Dividing the area of the survey into subsets makes the cleaning of a large data set more manageable.

The following workflow is a suggested method for cleaning data with Subset Editor.

1. Create Subset Tiles.

   (See "Track Progress with Subset Tiles" on page 394)

2. Open Subset Editor and create a subset area within the tiled area. (See Subset Editor Interface in the HIPS and SIPS Editors Guide.)
3. Use the 3D View Controls to adjust the view of the data. (See Subset 3D View in the HIPS and SIPS Editors Guide.)
4. Clean subset data.

   (See "Subset Cleaning" on page 389)

5. Classify data in subset area.

   (See "Tracking Cleaning Status" on page 402)

6. Define another subset and repeat the process.

If you are processing a small amount of data, using subset tiles to track progress can be omitted (Steps 1 and 2), and Subset Editor can be opened directly with a subset area over the track lines.

However, if you are processing large areas of data, the subset tiling method is a useful way to track changes.

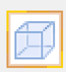
(For hypothesis editing a CUBE surface in Subset Editor, see "Hypothesis Editing" on page 301.)

Subset Cleaning

Soundings in the 2D and 3D Views can be interactively examined and their status flags can be turned on and off as needed. To start with, a sounding is flagged as Accepted.

Status flags for selected soundings can be turned on from the right-click menu in the Subset Editor 2D or 3D View. Flags can also be set for queried soundings from the right-click menu in the Selection window.

When you select data in the 3D View, all data in the highlighted area is selected, not just the visible data.

Menu	Tools > Subset Editor
Tool	

To set status flags for data in Subset Editor:

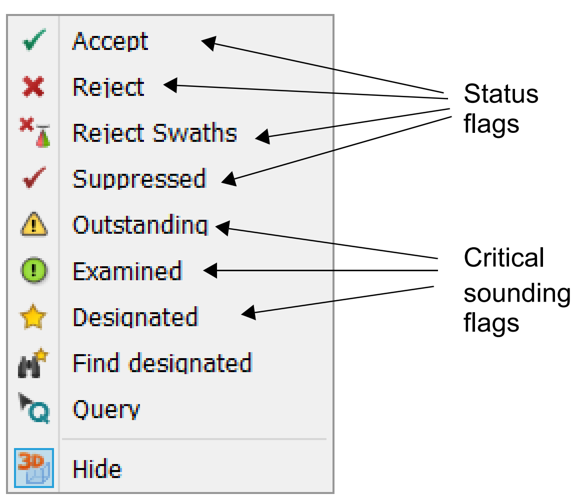
1. Select the Subset Editor command.
2. Define the area in the Display window to be included in the subset. See “Open Subset Editor and Load Data” on page 813 in the Editors Guide.
3. Select the Load Subset command to the load the data in the defined area into the Subset Editor windows.

Menu	Tools > Subset Editor > Load
Tool	

The subset of your data is generated and displayed in the 2D and 3D windows in Subset Editor.

If data for any line has not been merged, the depth information for data may not be accurate. In this case, the subset will be loaded, but the Output window will warn you that the processed depths for a specified line are outdated and require Merge. (See “Merge” on page 195.

4. Select data to be edited.
5. Select the appropriate setting from the Status Flag menu, from the Subset Editor toolbar or the pop-up menu (illustrated below).



The flagged data will be displayed in the colour set for the specific status in Tools > Options > Display > Status.

"ACCEPT" ON PAGE 390
"REJECT" ON PAGE 390
"REJECT SWATHS" ON PAGE 390
"SUPPRESSED" ON PAGE 391
"OUTSTANDING" ON PAGE 391
"EXAMINED" ON PAGE 391
"QUERY" ON PAGE 392
"DESIGNATE SOUNDINGS" ON PAGE 393

Accept

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

Change rejected soundings back to the accepted status.

1. Select the rejected data that you want to return to accepted status.
2. Select the Accept command.

The soundings are now marked as accepted.

Reject

If rejected during processing, the status of a sounding can be returned to the Accepted status without loss of data.

In either the 2D or 3D View:

1. Select the data you want to reject so it is highlighted.

> When you select data in the 3D View, all data in the highlighted area is selected, not just the visible data.

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

2. Select the Reject command.

If you Query the rejected soundings, you will see that they now display the status flag "Rejected (Subset-Hydrographer)".

As well, the soundings may or may not be displayed depending on whether the Display Filter is set to show Rejected soundings.

See "VIEW SOUNDINGS STATUS" ON PAGE 334.

Reject Swaths

In some instances it may be necessary to reject all soundings in one or more swaths. This can be done with the Reject Swaths command without having to select all of the data in the swaths.

1. Select some sounding in the swath you want to reject.

Menu	Edit > Status Flag > Reject Swaths
Tool	
Key	<S>

2. Select the Reject Swaths command.

All soundings in the swath (whether individually selected or not) now have the Rejected by Swath Editor status flag turned on. These soundings will not be displayed unless the Display Filter tool is turned on.

See "View Soundings Status" on page 334

Suppressed

Set the status of soundings so that they are hidden or suppressed when creating a surface, or exporting the surface, but are not Rejected.

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

1. Select the data that you want hide but not reject.
2. Select the Suppressed command.

The soundings are now flagged.

To view suppressed soundings:

3. Select the check box for Suppressed from the Display Filter list. By default, suppression is turned off in Display Filters.

   See "View Soundings Status" on page 334.

The displayed colour for suppressed soundings is set in Tools > Options > Display > Status.

Outstanding

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

Flag a sounding as requiring further examination.

1. Select the data that you want flag as Outstanding.
2. Select the Outstanding command.

Examined

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

Flag questionable soundings as having been examined and verified.

1. Select the data that you want to flag as Examined.
2. Select the Examined command.

Query

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

View information on selected data.

1. Select the data that you want to query.
2. Select the Query command.

The data is displayed in the Selection window.

Note: When you select data in the 3D View, all data in the highlighted area is selected, not just the visible data.

Designate Soundings

Shoal soundings over significant seabed features can be flagged as “Designated” to ensure that these depths are maintained in charts and other standard hydrographic products.

Use the Designate commands to flag, for example, the top of a mast in a wreck, or the shoalest sounding in a cluster of soundings identifying an outcrop of rocks.

For more information, see “Critical Soundings” on page 337.

Menu	Edit > Status Flag > Designate
Tool	
Key	<D>

1. Select the shoalest sounding from a cluster of soundings around a feature in Subset Editor.
2. Select the Designated Soundings command.

The sounding is flagged as Designated.

Shoalest soundings can also be designated in Swath Editor.

Find and Designate

The Find and Designate command automatically selects the shallowest sounding in cluster of highlighted soundings. This feature reduces the time needed to designate shallowest soundings.

1. Use the cursor to highlight soundings in Swath or Subset Editor.
2. [Optional] Use the Query command to view the soundings in the Selection window.
3. Select the Find and Designate command.

Menu	Edit > Status Flag > Find and Designate
Tool	

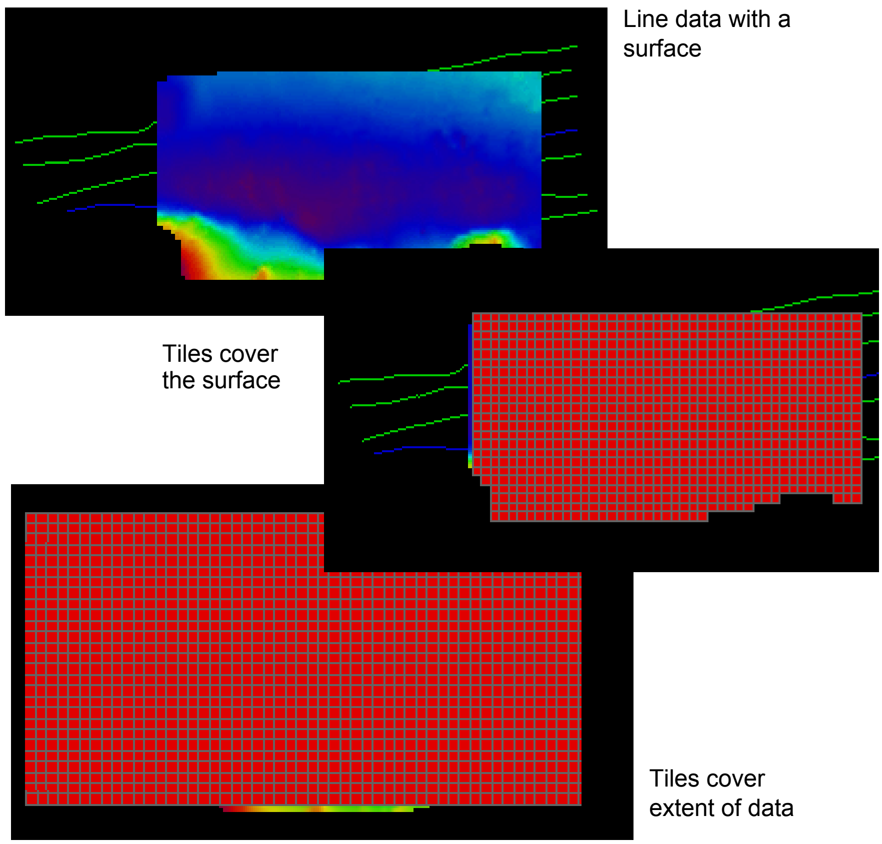
The shoalest sounding among the cluster of highlighted soundings is now flagged as Designated in the Selection window, and displays the Designated symbol when viewed in Swath or Subset Editor.

Track Progress with Subset Tiles

Subset Tiles are used to track the process of examining and cleaning areas within the survey project. Each tile within a Subset Tile layer has one of three tracking settings: Incomplete, Partially Complete, or Completed (see “Tracking Cleaning Status” on page 402).

The extent of the subset tile area can be determined by setting the tiles to cover:

- the extents of the open data, or
- the area of an open surface, or
- the area covered by an open ENC *.000 file.



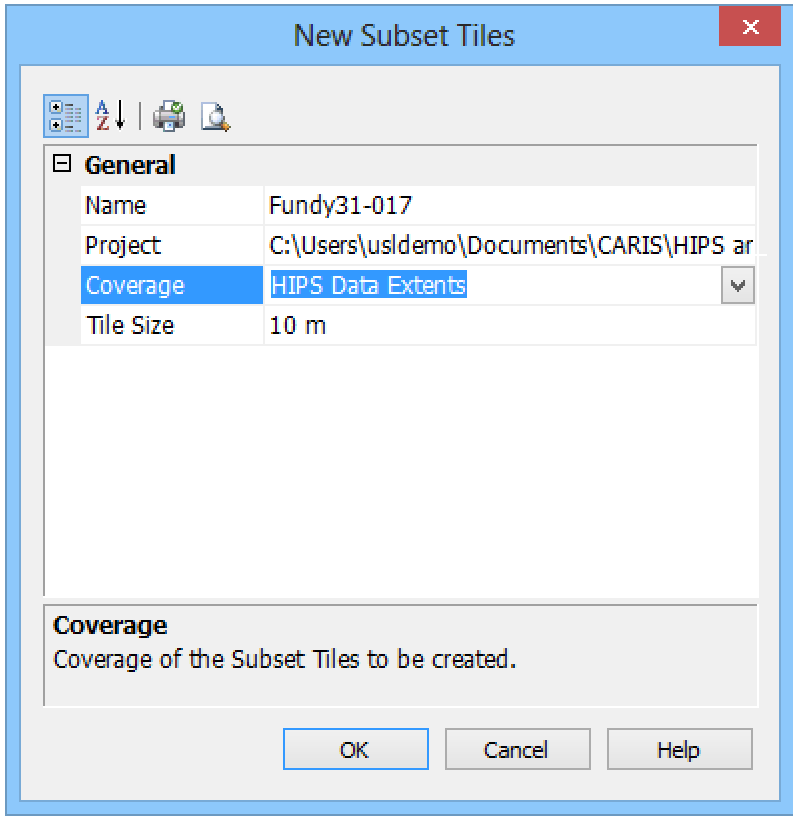
The option to cover the extent of the data is useful if you are cleaning newly acquired data with Subset Editor, while the area is being surveyed. You would not have to create a new subset tile layer when data is added to your project.

Menu	File > New > Subset Tiles

To create subset tiles for tracking:

1. Select the New Subset Tiles command.

The New Subset Tiles dialog box is displayed.



2. In the *Name* field, type a name for the subset tiles layer.

The drop-down list in the *Project* field will display the names of the currently open projects.

3. Select the project to be tiled from the drop-down list.

The *Coverage* field gives you the option of tiling the entire extents of the open data, or limiting the subset tile area to the selected surface.

4. Select either HIPS Data Extents or a surface by name from the drop-down list in the *Coverage* field.
5. Type a value for the size of a tile.
6. Click **OK**.

A subset tile layer is displayed, and listed by name in the Layers window. The tile layer is saved to the project file.

Subset tile properties

You can set display properties for tiles in the Properties window. For example, you can set colours to display the progress of subset cleaning, or make the tile layer transparent so the features below it are more visible.

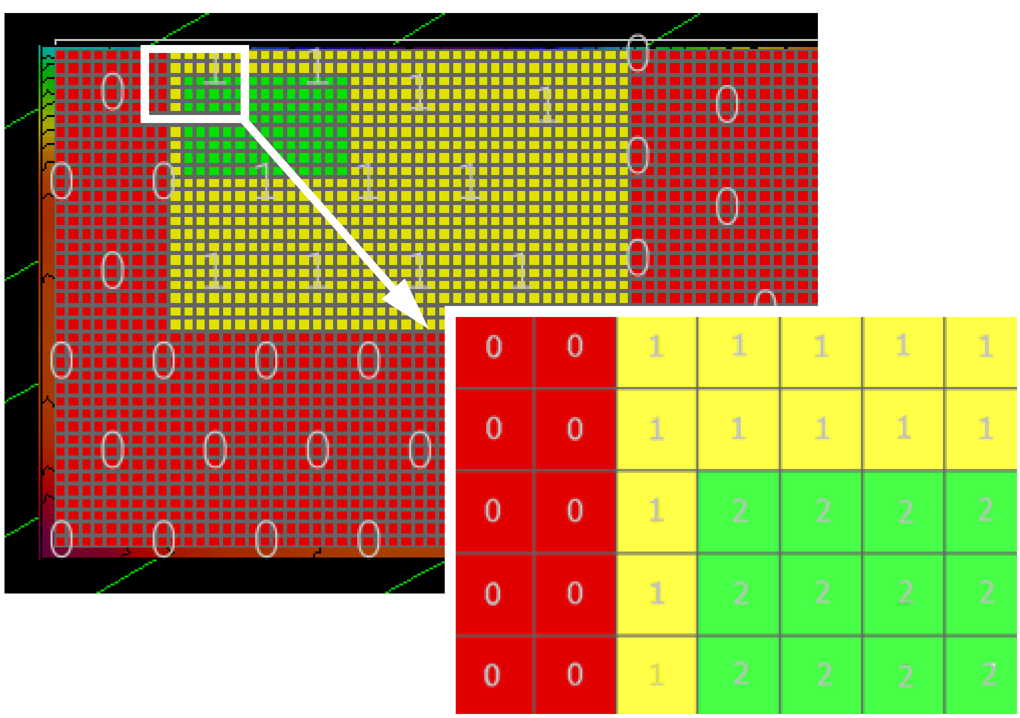
The *Active Attribute* field lists all attributes included in the tiles. When you select an attribute, such as cleaning status, tiles are displayed according to the values of those attributes.

For example, if `Cleaning_Status` is selected as the active attribute, and the Colour fill tiles is set to “True”, the tiles will automatically be coloured as you edit, as follows:

- red, if the tiles have not been examined. (status = Incomplete)
- yellow, if the tiles are partially complete. (The tiles were only partially covered by a subset that has been marked as complete.)
- green, if the cleaning of the tiles is complete.

As well, the numeric value for the attribute status can be displayed, using the *Display tile value* field.

When this field is set to “True”, the tiles will display the value 0 for Incomplete, 1 for Partially complete or 2 for Completed. The example below shows tiles with “Cleaning status” set as the active attribute, and both *Colour fill tiles* and *Display tile value* set to “True”.

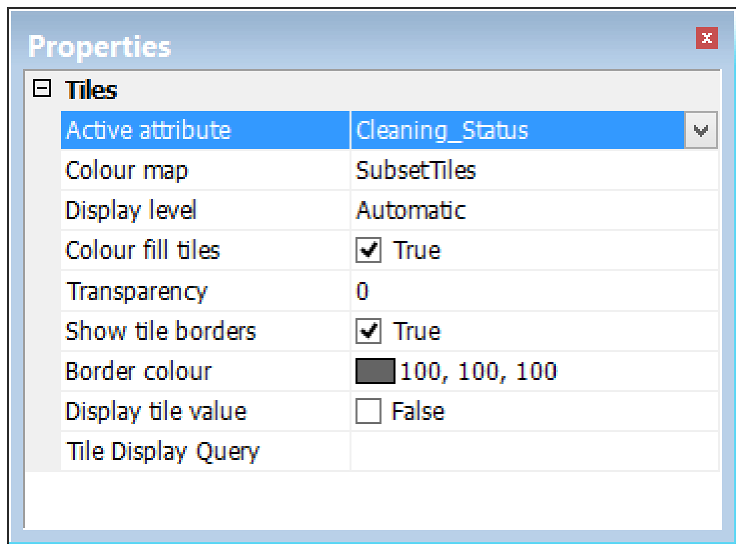


These cleaning status values can also be viewed in the Selection window when tiles are selected.

To set properties for the tiles layer,

1. Select a tile layer in the Layers window.
2. Open the Properties window.

The Properties window shows the display options for tiles.



3. Select an attribute from the drop-down *Active Attribute* list, either Cleaning Status or Last Modified.
4. Select a colour map from the *Colour Map* drop-down list.
5. Change how tiles are rendered to the Display Window by selecting the *Display Level*, either:
   - *Maximum*: Draw all tiles
   - *Automatic*: Draw tiles to fit the current extent of the Display window.

The *Colour Fill Tiles* check box determine how attribute values are drawn in the Display window.

- If the *Colour Fill* check box is set to “True”, the tiles are drawn according to the selected colour map.
- If the check box is unchecked and set to “False”, the tiles are drawn in outline only.

6. Select a *Transparency* percentage (0, 50, 100) to make features visible through the tiles.

The *Show Tile Border*s check box determines how the border lines dividing the tile areas are displayed.

- If the *Show Tile Borders* check box is checked, the value is set to “True” and the lines are colour-coded according to the colour selected in the *Border colour* field.
- If the *Show Tile Borders* check box is clear, the value is set to “False”, and the borders of the tiles are not displayed.

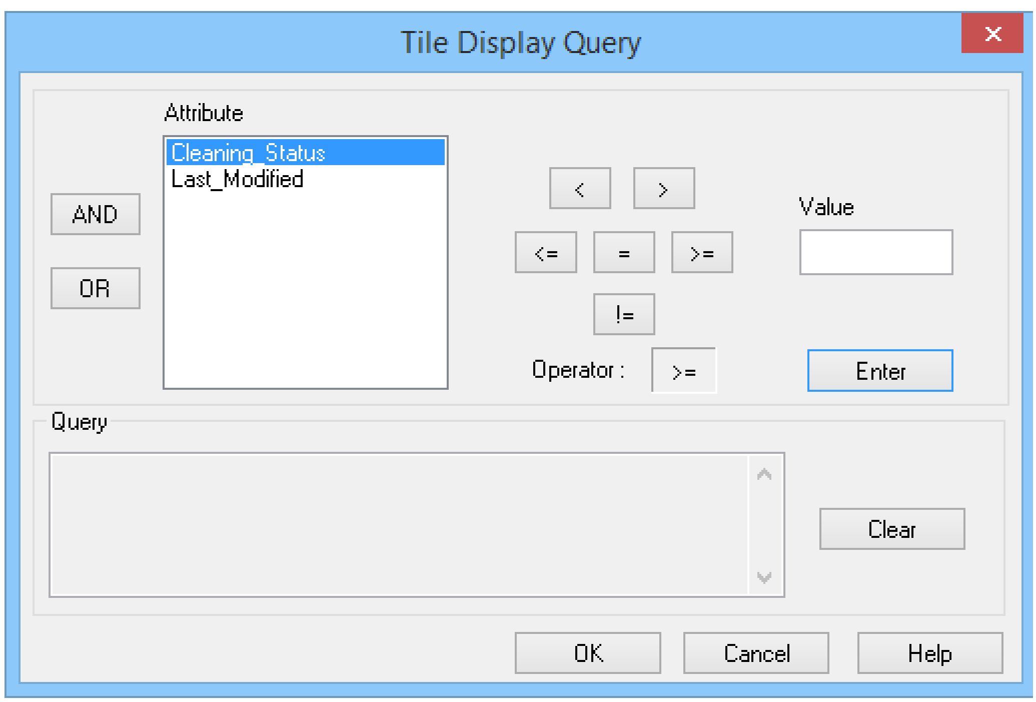
7. [Optional] Select an alternative colour for the border lines from the colour picker in the *Border colour* field.
8. Set the Display tile value check box to “True” so the appropriate cleaning status value (0=Incomplete, 1=Partially Complete and 2=Complete) is displayed in each tile.

Tile Display Query

The last field in the Properties for subset tiles enables you to create an SQL-type query based on selected tile attributes and defined values. Only those tiles that meet the search criteria will be shown in the Display window. The query parameters are saved in the Properties window when the session is saved.

The Tile Query dialog box is opened from the Properties window, when the tile layers are selected.

1. Select the tile layer in the Layers window.
2. Open the Properties window for the tiles layer.
3. In the Tile Display Query field, click **Browse (...)** to open the Tile Display Query dialog box.



The Attribute fields displays the attributes of the selected tile layer.

4. Select an attribute from the list to filter by.
5. Click one of the following operator buttons:
   - (<) less than (e.g., to select attributes that have a value less than the entered amount)

- (>) greater than
- (<=) less than or equal to
- (=) equals
- (>=) greater than or equal to
- (!=) not equal to

6. Type a value for the attribute in the *Value* field, and click **Enter**.

The search criteria you selected in the dialog box are displayed in the *Query* section.

7. To add other search criteria, click either of these logic operators:
   - **AND**: This operator combines two queries together so the results of both are shown in the Display window.
   - **OR**: This operator searches and displays either one of two search criteria in the Display window.

For example, if one of the attributes is "Count", to set a query string to select only the tiles with more than 2500 soundings that also have a resolution of 500, use:

```
Count < 2500 AND MinRes = 500
```

8. Repeat Steps 3 through 6 to add as many search criteria as needed.
9. Click **OK**.

The Display window is refreshed to show only the tiles that meet the criteria you entered.

The Query string is displayed in the Tile Display Query field in the Properties window.

10. Save your session so the query will be saved and restored when that session is loaded again.

To clear the Tile Display Query field:

11. Select the tile layer.
12. Click the **Browse** button in the Tile Display Query field.
13. In the Tile Display Query dialog box, click **Clear**.
14. This will remove the criteria and display all the attributes for the selected tile layer.

Edit tiled subset data

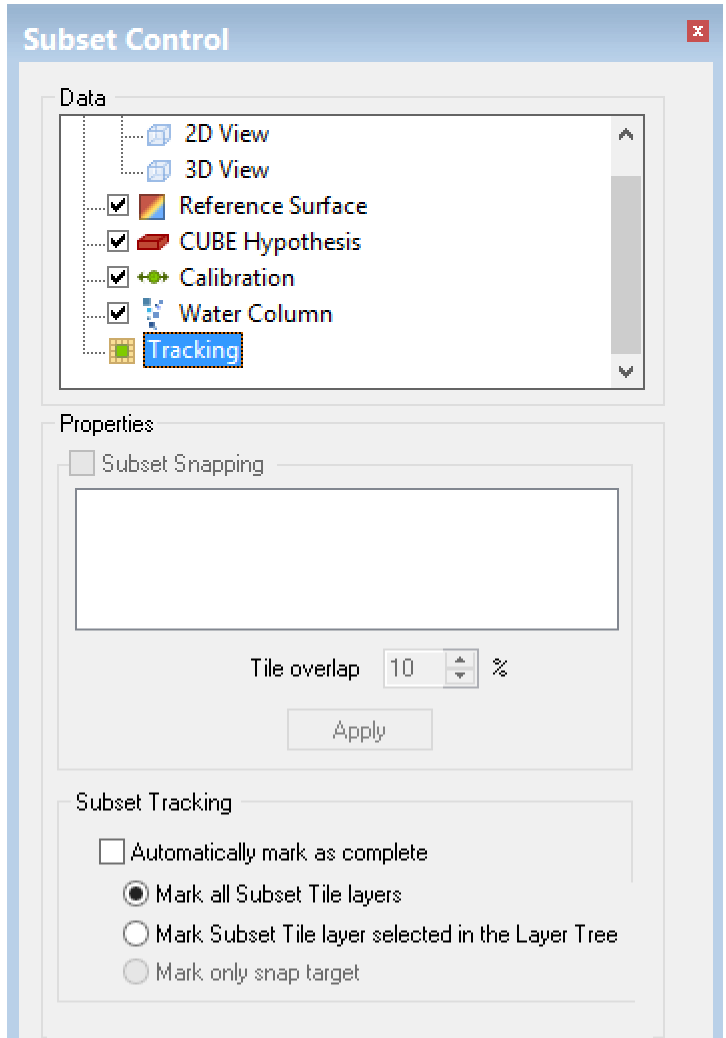
Once the tracking tiles are created, you can examine and edit the data by opening Subset Editor and defining a subset. (See "Open Subset Editor and Load Data" on page 813). As you clean data from the open subset (see "Subset Cleaning" on page 389), you can track the progress of cleaning by setting status from the Subset Editor toolbar to Incomplete, Partially Complete or Complete.

Tracking Layer Options

The layer containing the tracking tiles has options which can be set in the Subset Control window (see "" on page 817).

1. Select the Tracking layer in the Subset Control window.

The window is refreshed to display the tracking options.



Activate the Subset Snapping check box so that the subset bounding box encloses entire tiles.

2. Select the Subset Snapping check box.
3. Select the tile tracking layer.
4. Click **Apply**.

You can also set an amount by which the subset snapping will overlap tiles as you move from completed to unprocessed tiles.

This helps ensure coverage of all of the data that lies within tiles along the edges of the subset.

5. Select a percentage in the *Tile Overlap* field.
6. Click **Apply**.

7. In the Subset Tracking options, select the *Automatically mark as complete* check box to flag the data as examined and cleaned. This can be changed later (see "Tracking Cleaning Status" on page 402).

When the *Automatically mark as complete* check box is selected, two other options are made active.

8. Select the *Mark all Subset Tile layers* option to flag the tiles in ALL tile layers as completed, or select *Mark only snap target* to select tiles only in the selected tile layer as completed.

Tracking Cleaning Status

Once you have examined and cleaned a subset, the subset area can be assigned one of three classifications:

- **Complete**: All data in the selected area is clean and ready for further processing or export.
- **Partially Complete**: Not all the data within the area has been viewed and verified as clean in Subset Editor.
- **Reset**: Data has not been verified or cleaned.

The area is colour-coded in the Subset Tile according to the selected cleaning status classification (and the settings in the Properties on the Subset Control window):

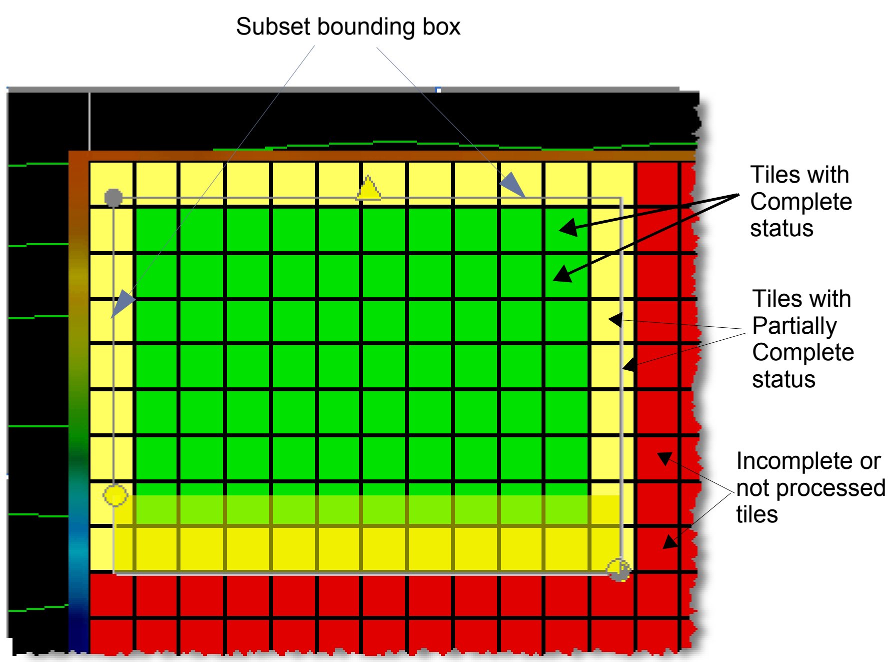
- Complete = green (2)
- Partially Complete = yellow (1)
- Incomplete or not processed = red (0).

To assign a cleaning status to a subset area,

1. Select the appropriate status from the Subset Editor toolbar or from the Tools >Subset Editor sub-menu.

The flagged tiles will be displayed with the colour appropriate to their status. Only tiles totally encompassed by the subset bounding box are flagged with the selected status.

Tiles only partially covered by the bounding box are flagged as Partially Complete, as in the example below.



Subsets and Surfaces

Surfaces can be opened in Subset Editor and displayed in the 2D and 3D Views. This enables you to determine the position of Designated soundings against an existing surface, and whether Designated soundings have been correctly applied to a Finalized surface. (See "Critical Soundings" on page 337.)

You can use the Surface Filter on a subset of any surface in Subset Editor.

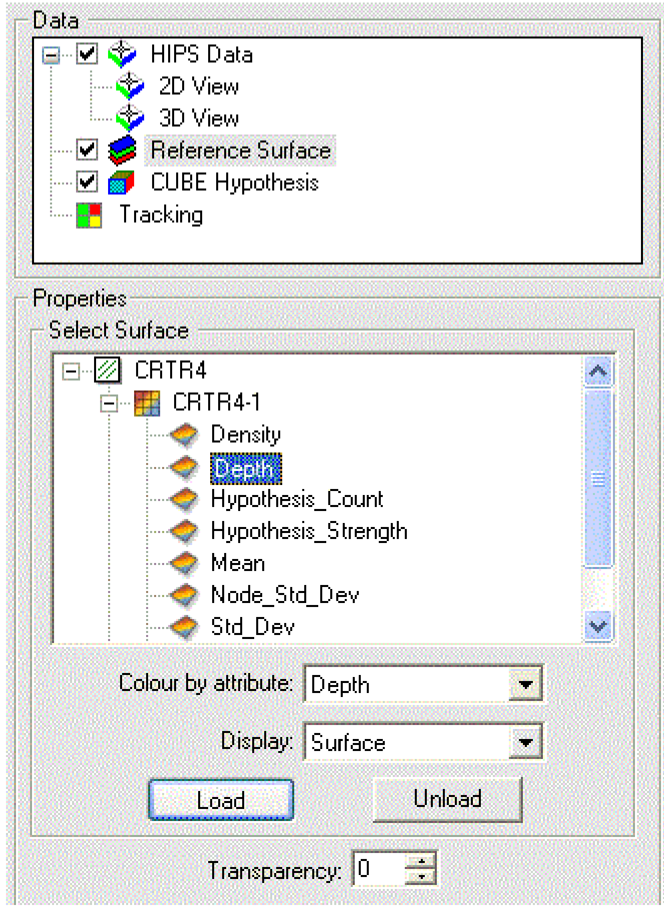
As well, you can use subset filters to filter soundings by depth, beam number and beam-to-beam slope values.

To use Subset Editor for hypothesis editing in a CUBE surface, see "Hypothesis Editing" on page 301.

Open a Surface in Subset Editor

1. Open the surface for your project.
2. Open Subset Editor and define a subset over the surface.
3. Load the subset.
4. Select the *Reference Surface* layer in the Subset Control window.

The Subset Control window is refreshed to show the property options for the Reference Surface layer.



5. Select a surface layer from the *Select Surface* file tree.

Although the Depth attribute is used to display the surface, you can colour the surface by another attribute by selecting that attribute from the *Colour by Attribute* drop-down list.

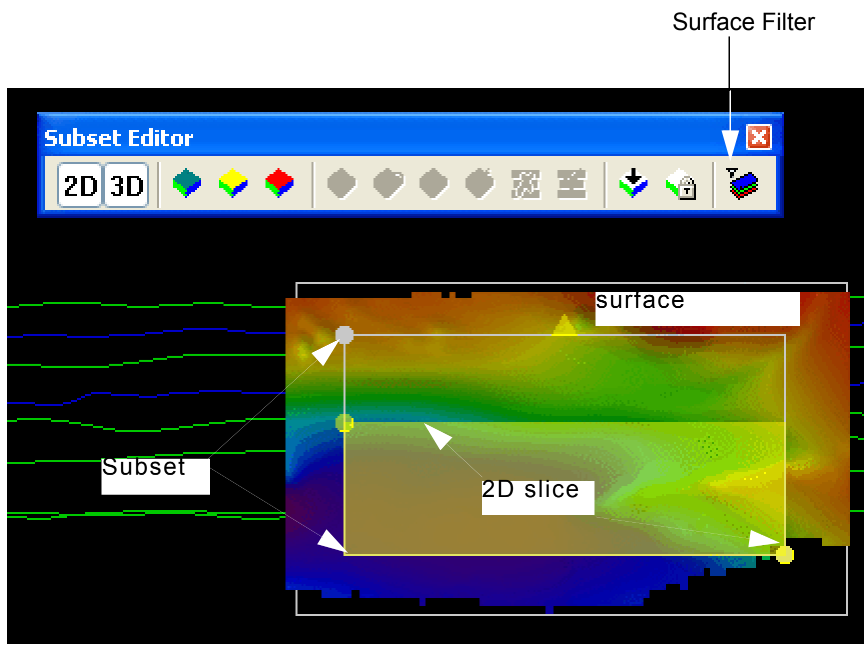
6. Display the surface as points, wire frame or a complete surface by selecting an option from the *Display* drop-down list.
7. Select a transparency level for the Surface, using the up and down arrow buttons.
8. Click **Load** in the bottom of the control page.

The surface is displayed in the Views according to the above options.

9. [Optional] Click **Unload** to remove the surface from the view windows.

Using the Surface Filter in Subset Editor

You can load any type of surface to Subset Editor and apply filtering to the part of the surface within the subset slice, or to the part of the surface defined within the full subset.



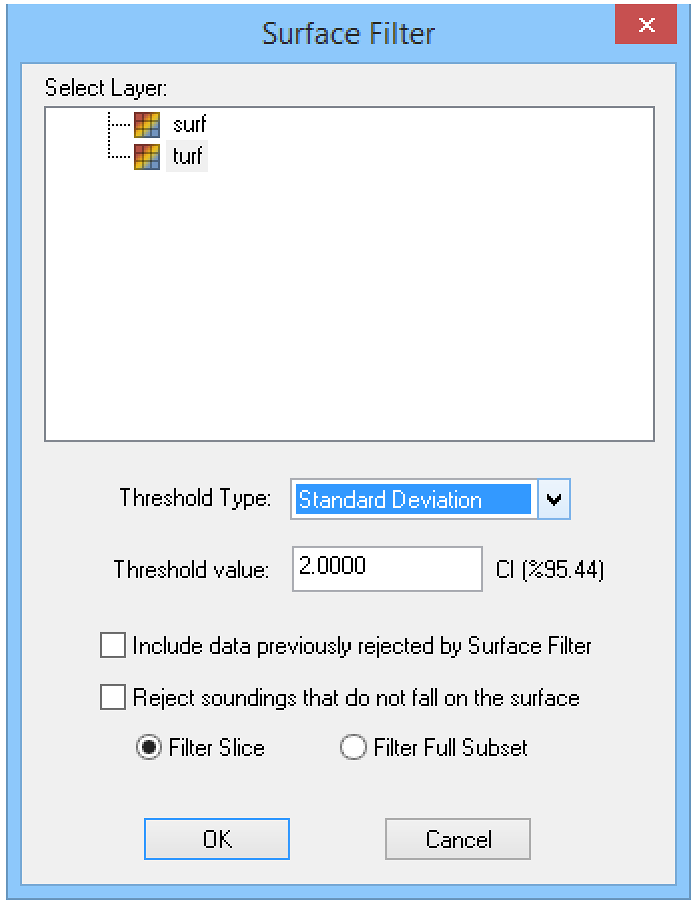
1. Open a surface.
2. Open Subset Editor and define a subset over the surface. (See "Open Subset Editor and Load Data" on page 813.)
3. Load the subset.
4. Select the *Reference Surface* layer in the Subset Control window.
5. Select the surface Depth layer from the *Select Surface* file tree in the Properties.
6. Click **Load**.

When the surface is loaded, the Surface Filter button on the Subset Editor toolbar is activated.

Menu	Tools > Subset Editor > Surface Filter	
Tool		Subset Editor toolbar

7. Select the Surface Filter command.

The Surface Filter dialog box is displayed.



In the Surface Filter dialog box:

8. Select the *Threshold Type* from the drop-down list.
9. Set the desired *Threshold value.*
   - For standard deviation or uncertainty thresholds, the value entered will show the confidence interval next to the field.
   - Static values are entered in units set in Tools > Options > Display > Units > Vertical units, e.g., metres.
10. Select the *Include data previously rejected* ... check box to include rejected data when running the filter.
11. Select the *Reject soundings that do not fall on the surface* check box to reject soundings that are offset from the surface.
12. Select *Filter Slice* to filter only the soundings within the subset slice, or
13. Select *Filter Full Subset* to filter all the soundings within the subset bounding box.
14. Click **OK** to apply the filter.

Subset Filters

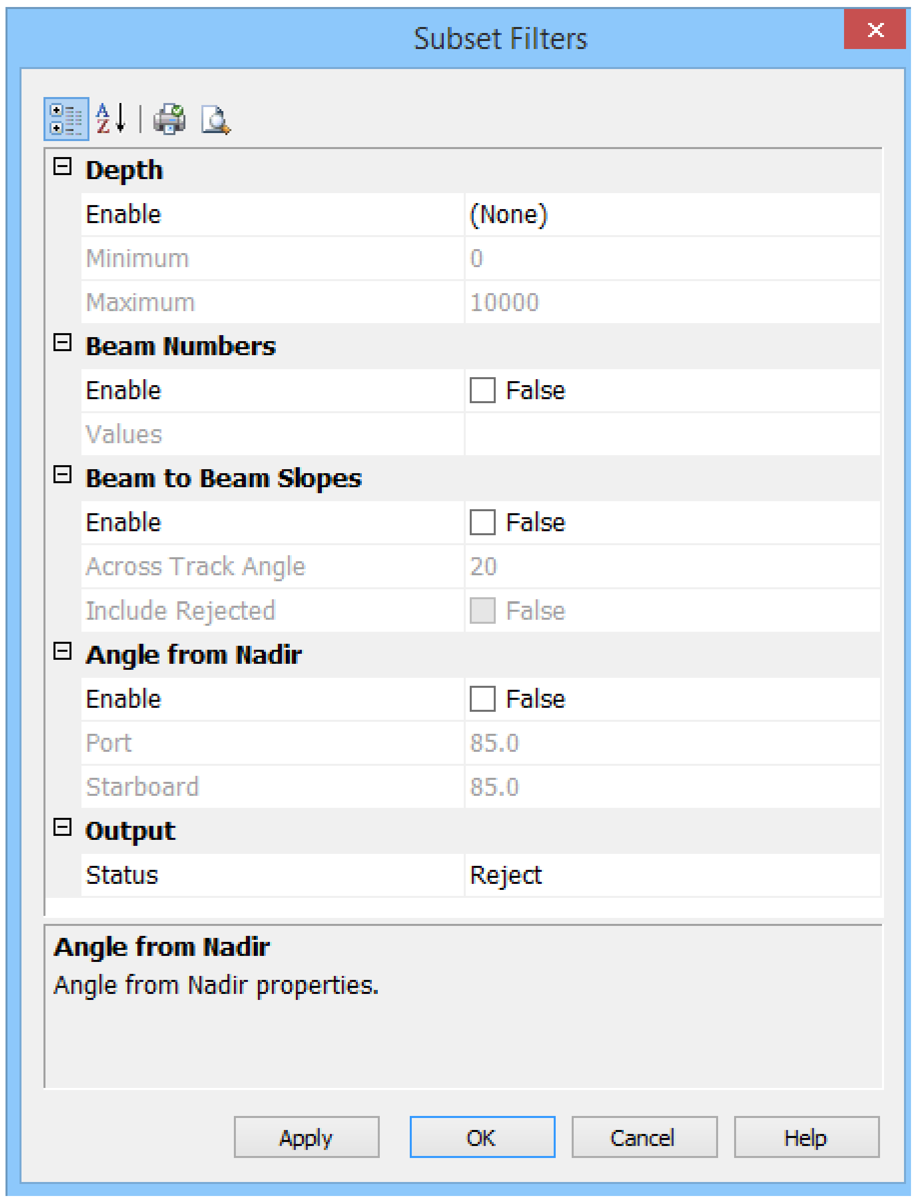
Menu	Tools > Editors > Subset > Filters
Tool	

Apply depth, beam number and beam-to-beam slope filters to soundings in Subset Editor.

To use the Subset filter:

1. Open Subset Editor.
2. Define a subset of the data, and Load data.
3. Select the HIPS Data layer in the Subset control window.
4. Select the Subset Filters tool.

The Subset Filter dialog box is displayed.



1. Enable options and set filter values, as follows:

Field	Description
Depth	Set minimum and/or maximum depth values to filter out soundings above and below the specified values.
Enable	From the drop down list: • Select *Both* to apply min/max filtering. • Select *Minimum* to filter only shallow values. • Select *Maximum* to filter only deep values. Default is *None*.
Minimum	Type a value above which shallower soundings will be filtered out.
Maximum	Type a value above which deeper soundings will be filtered out.
Beam Numbers	Filter single or multiple numbered beams.
Enable	Set check box to *False* to skip beam number filtering. Default is set to *True*.
Values	Enter the numbers of the beams to be rejected. • Separate individual beam numbers with a space e.g., values `12 32 46` will reject beams `12, 32` and `46`. • Specify a range of beams with a hyphen e.g.,`56-60` will reject beams `56, 57, 58, 59`, and `60`.
Beam to Beam Slopes	Slopes before and after a beam value across track are used to filter outliers.
Enable	Set check box to *False* to skip outlier detection. Default is set to *True*.
Across Track Angle	For each beam on the same swath, calculate the slopes in degrees to the prior and post beams. If both slopes exceed the defined value and are of opposite sign, then filter the beam. Default value is 20 degrees.
Include rejected	This option includes previously rejected soundings when recalculating the slopes during multiple runs of the filter.
Angle From Nadir	Filter beams that are outside of the designated angles (e.g., 60 degrees) in the Port and Starboard side of the swath. Angles are computed from the nadir using depths and across track distances (roll-corrected).
Enable	Set to True to apply the filter settings. Default is set to False.
Port	Type the value of the Port angle from nadir to filter the beams that are outside of the designated angle.
Starboard	Type the value of the Starboard angle from nadir to filter the beams that are outside of the designated angle.
Output	
Status	Select a status flag to specify whether data is accepted or rejected by the filter process.

17

Process Imagery Data

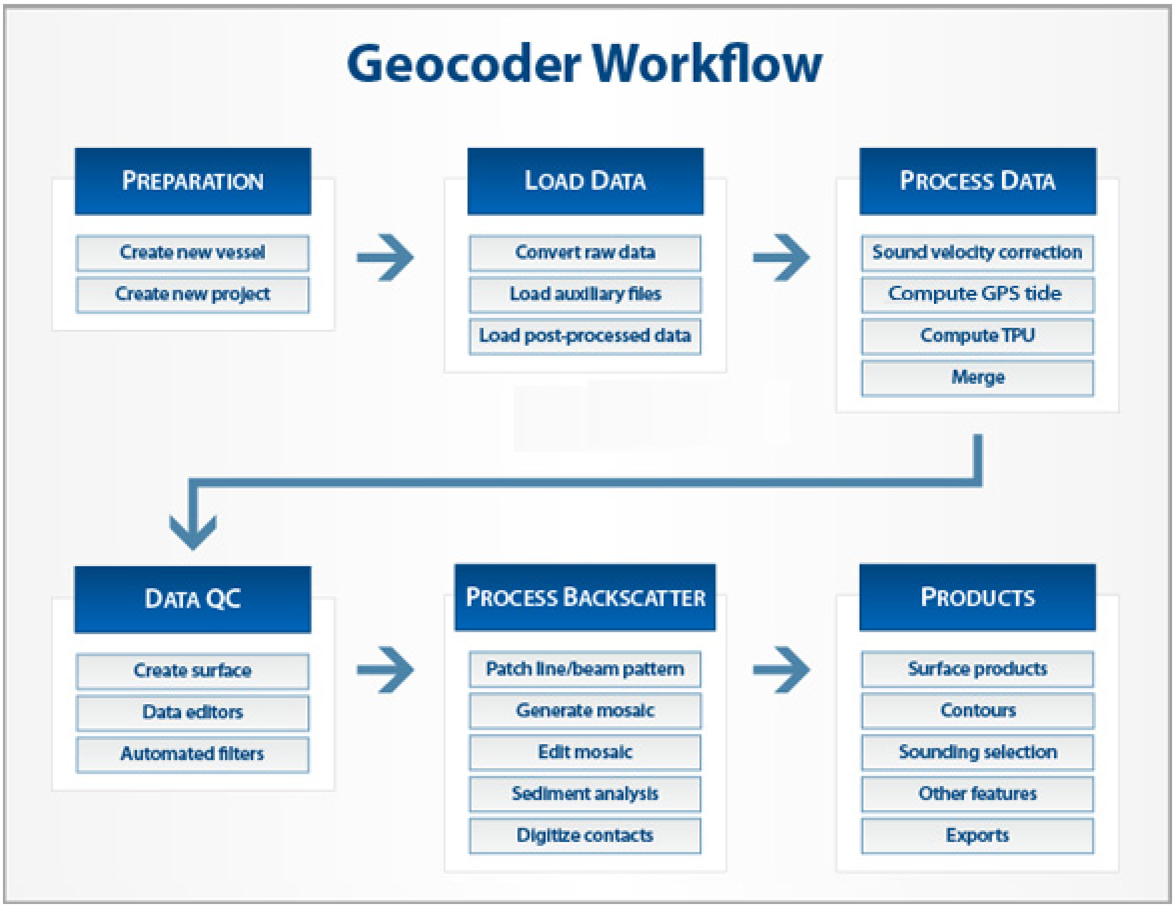
In this chapter...

Side scan and multibeam backscatter imagery data can be processed directly into mosaics, which can then be examined for anomalies and corrected.

Overview

Multibeam backscatter and side scan sonar data can be used to create mosaics. But as well as being end products, mosaics can be used as tools to more efficiently process imagery data, using mosaic properties.

Multibeam backscatter must be converted to HIPS format, processed with the bathymetric editors, and Merged (see "Merge" on page 195) before imagery correction is done.



If you:

- need to edit altitude data for side scan data, or
- will be digitizing and editing contacts, or
- prefer to use a waterfall display for editing, or

then these functions must be performed in Side Scan Editor. (This is described in the Side Scan Editor section of the Editors Guide.)

Processing Engines

Mosaics and beam pattern files can be created using these processing engines:

SIPS Backscatter

This engine processes multibeam backscatter, employing such corrections for backscatter intensity as TVG, transmit and receiver gain, and beam pattern correction.

This processing engine will directly read Reson *. s7k, Hypack *.hsx/*.7k and Kongsberg *.all files to create a beam pattern from the beam value stored or extracted from the time series datagrams.

"Create SIPS Backscatter Beam Pattern file" on page 413

GeoCoder

GeoCoder processes multibeam backscatter, both beam-averaged and the higher resolution time-series returns, as well as side scan sonar data.

"Create GeoCoder Beam Pattern File" on page 417

SIPS Side Scan

SIPS Side Scan engine processes side scan using imagery corrections such as TVG, gain and despeckle as well as beam pattern correction to create mosaics from selected track lines.

"Create SIPS Side Scan Beam Pattern File" on page 415

Beam Pattern Correction

Beam Pattern Correction is applied to remove acoustic artifacts from the imagery caused by imperfections in the sonar. Beam pattern correction relies on a user-generated beam pattern file to identify and remove this effect.

Once the beam pattern file is created it can be applied to the survey data to uniformly correct the selected data. This also results in a more consistent appearance in imagery.

For consistent results with GeoCoder beam patterns, you should create your beam pattern file over an area of known sediment type and featureless an area as possible, preferably sand. For best results, it is recommended that a “patch test” be done during a survey over an area known to have such properties.

For SIPS beam patterns, for consistent results you should use as much data as possible to create the beam pattern.

Beam Pattern correction can be applied during the creation of a mosaic, or in Side Scan Editor using the settings in the Properties window.

“Create Beam Pattern file” on page 413

See also “Beam Pattern Correction” on page 91 in Side Scan Editor.

Create Beam Pattern file

Beam pattern files can be created from SIPS backscatter, SIPS side scan or GeoCoder data. A beam pattern file can be created from a selected line or lines for SIPS backscatter or side scan data, and from a selected line segment when using the GeoCoder processor.

A SIPS Side Scan beam pattern file can also be created in Side Scan Editor from a line or a selection of pings.

Menu	Beam Pattern > Create >
Tool	

To create beam pattern file:

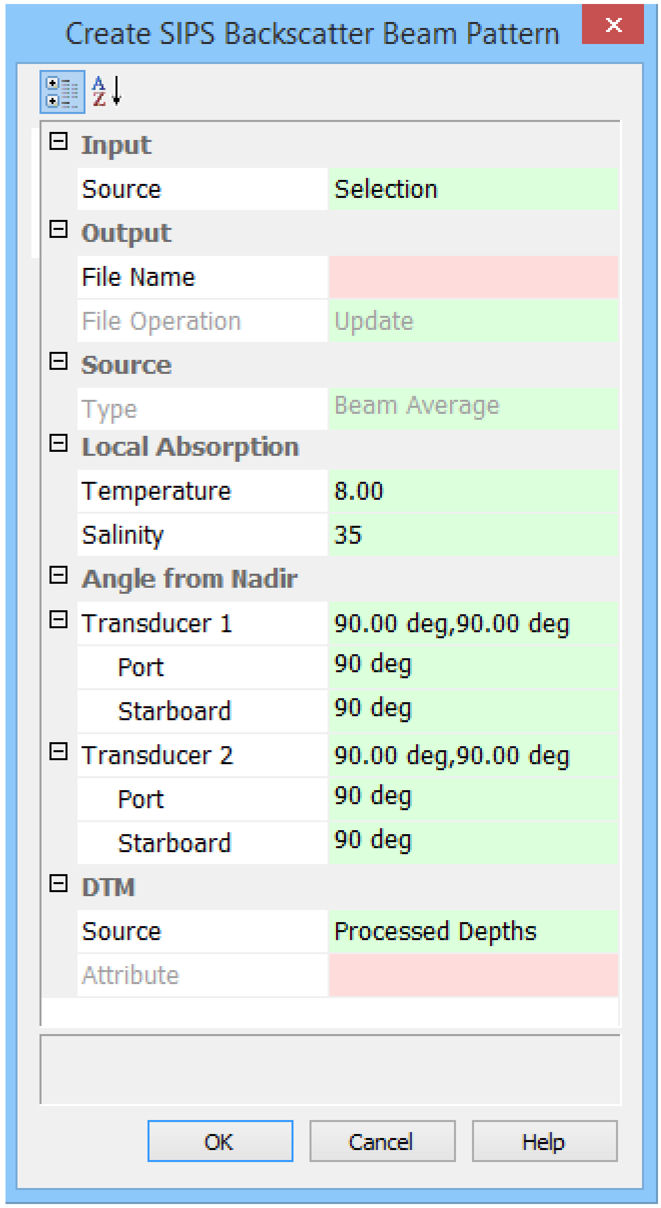
1. Select a line.
2. Select a Create Beam Pattern command:
   - SIPS Backscatter,
   - SIPS Side Scan or
   - GeoCoder.

The Create Beam Pattern dialog box is displayed.

- "Create SIPS Backscatter Beam Pattern file" on page 413
- "Create SIPS Side Scan Beam Pattern File" on page 415
- "Create GeoCoder Beam Pattern File" on page 417

Create SIPS Backscatter Beam Pattern file

Beam pattern files can be created from SIPS backscatter data from a line or lines selected in the Display window.



1. Set options.

Input	The source of the data.
Source	• To create the beam pattern file from the selected line or lines, set the *Source* field to Selection. • To create the beam pattern file from all open lines, select the All Track Lines option in the *Source* field.
Output	Name and destination of the beam pattern file.
File Name	1. Click **Browse** in the *File Name* field and enter a name for the beam pattern file. 2. Click **Save**.
File Operation	If a beam pattern file already exists, this field displays options to Update or Overwrite the existing file.

Source	The type of imagery that will be used to create the beam pattern file.
Type	Beam Average is the only type supported at this time.
Local Absorption	Correction for transmission loss using temperature and salinity values.
Temperature	Set temperature value in degrees. Default value is 8.00
Salinity	Set salinity value as parts per thousand. Default value is 35 parts per thousand.
Angle from Nadir	Use an angle across-track from directly below the transducer (0 degrees) to set how much data is included in the beam pattern file.
Transducer1	This field displays the angle from nadir values set for Transducer1 in *Port* and *Starboard* fields.
Port	Set values in degrees of the angle to port from nadir.
Starboard	Set values in degrees of the angle to starboard from nadir.
Transducer2	This field displays the angle from nadir set for Transducer2 (in a dual system) in *Port* and *Starboard* fields.
Port	Set values in degrees of the angle to port from nadir.
Starboard	Set values in degrees of the angle to starboard from nadir.
DTM	Settings for the digital terrain model used to compute the local bottom slope used in the calculations of real incidence angle and ensonified area.
Source	Select the DTM source to use.
Attribute	Select the surface attribute to use.

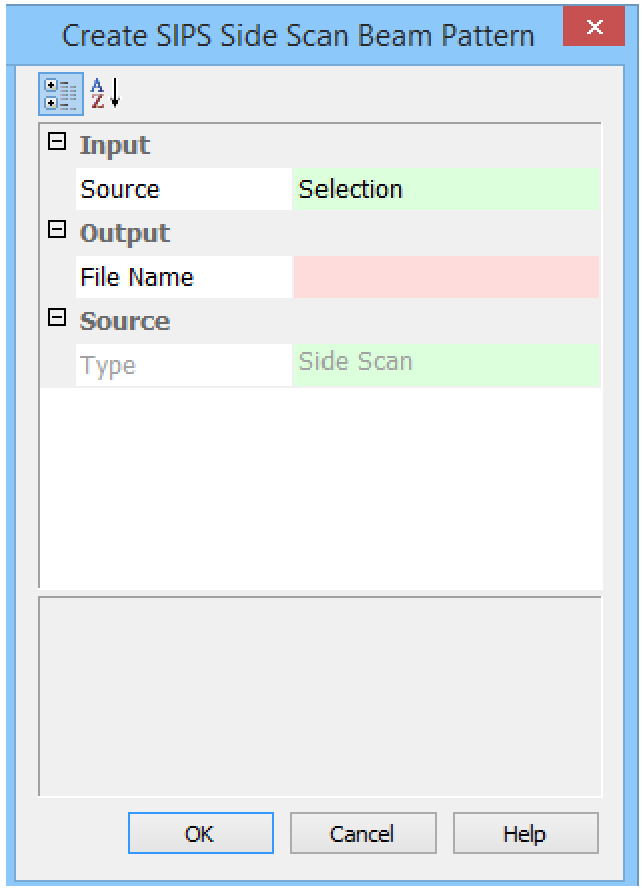
2. Click **OK** to create file.

The new beam pattern file will have a `*.bbp` extension.

Create SIPS Side Scan Beam Pattern File

Beam pattern files can be created from lines selected in the Display window, or from all open track lines.

Beam pattern files can also be created from a range of pings selected in the waterfall view in Side Scan Editor.



1. Set options.

Input	The source of the data:
Source	• To create the beam pattern file from the selected line set the *Source* field to Selection. • To create the beam pattern file from all open lines, select the All Track Lines option in the *Source* field.
Output	Name and destination of the beam pattern file.
File Name	1. Click **Browse** in the *File Name* field and enter a name for the beam pattern file. 2. Click **Save**.
Source	The type of data to be read.
Type	Set to Side Scan by default.

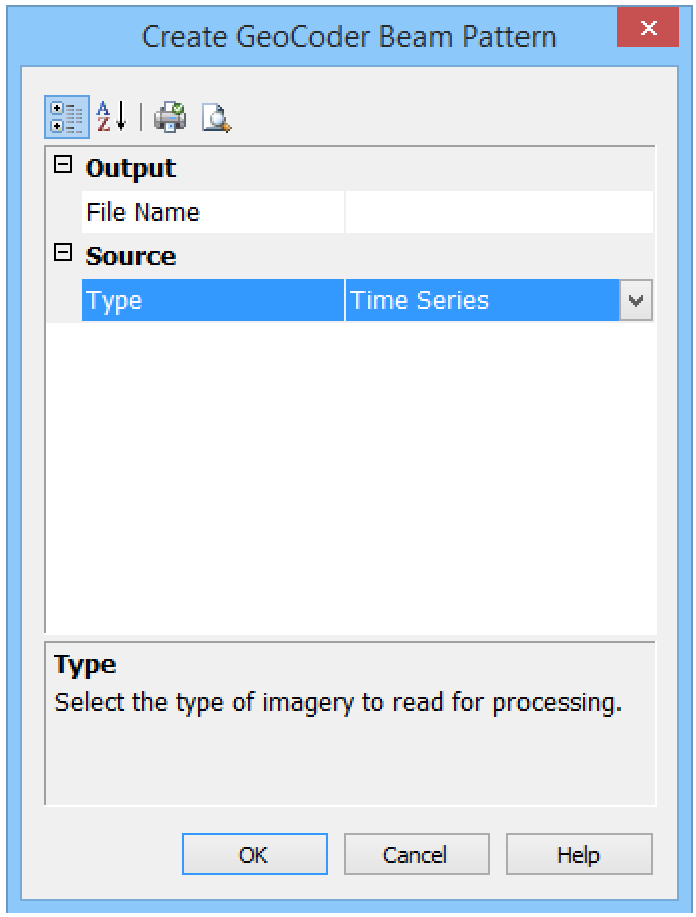
2. Click **OK** to create the side scan beam pattern file.

The new file will have a *.bp extension.

You can view a loaded beam pattern in the Signal Display window, where it will appear as a yellow line. See “SIGNAL DISPLAY WINDOW” ON PAGE 80.

Create GeoCoder Beam Pattern File

For consistent results with GeoCoder beam patterns, beam pattern files should be created over a homogeneous area of known sediment type. The Sediment Analysis tool can be useful in determining sediment type. See "Sediment Analysis" on page 440.



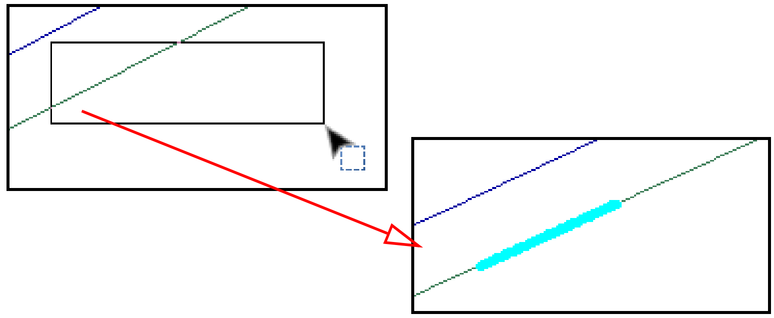
1. Set Options:

Output	Name and destination of the beam pattern file.
File Name	1. Click **Browse** in the *File Name* field and enter a name for the beam pattern file. 2. Click **Save**.
Source	The type of data to be read.
Type	1. Select Side Scan, Beam Average or Time Series. Set to Time Series by default.

2. Click **OK**.

The dialog box closes and the cursor changes shape.

3. Use the cursor to range-select a segment of a line.



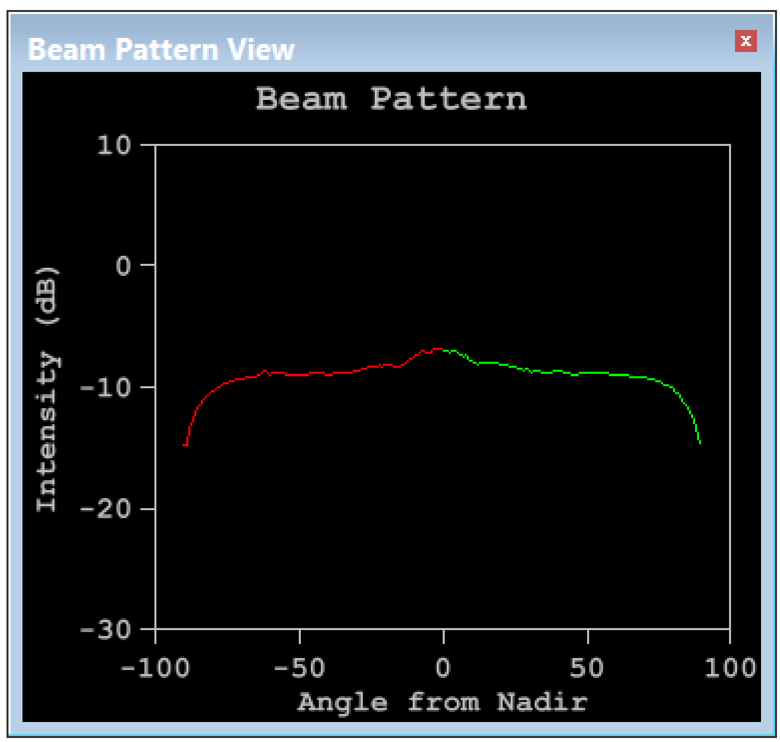
When you release the cursor, the selected segment of the line is highlighted. HIPS will then automatically create the beam pattern from the segment. The Output window and progress bar show the status of the beam pattern creation.

The GeoCoder beam pattern file will be saved with a `*.bpt` extension.

Menu	Process > Beam Pattern > View
Tool	

4. Select the View command to see the beam pattern.

The Beam Pattern View displays the beam pattern graph.



To complete the GeoCoder beam pattern creation:

Menu	Process > Beam Pattern > Close
Tool	

5. Click the Close Beam Pattern command.

The current beam pattern must be closed before another GeoCoder beam pattern can be created.

See also "Set Sediment Class Options" on page 419.

Beam Pattern Offset

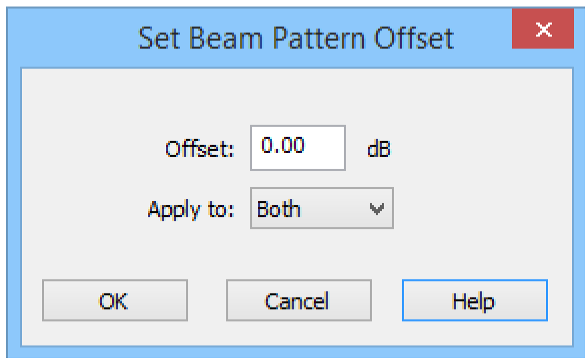
If there is a need to adjust the beam pattern to the observed data, you can use a dB offset.

The Beam pattern file must be visible in the Beam Pattern View in to order to see the effect of the offset on the curve.

Menu	Beam Pattern > Offset
Tool	

1. Select the Set Beam Pattern Offset command.

The Beam Pattern Offset dialog box is displayed.



2. Enter an offset in dB. Values can be entered in negative numbers to move the curve.

Menu	Process > Beam Pattern > Close
Tool	

3. Set to apply the offset to Port, Starboard or Both by selecting the values from the list.
4. Click **OK** to apply.
5. Click **Close** to close the beam pattern.

Set Sediment Class Options

When a beam pattern is created using the GeoCoder engine, it removes the sediment response component, based on the internal sediment response model, to obtain a “pure” beam pattern that can be used across multiple sediment types.

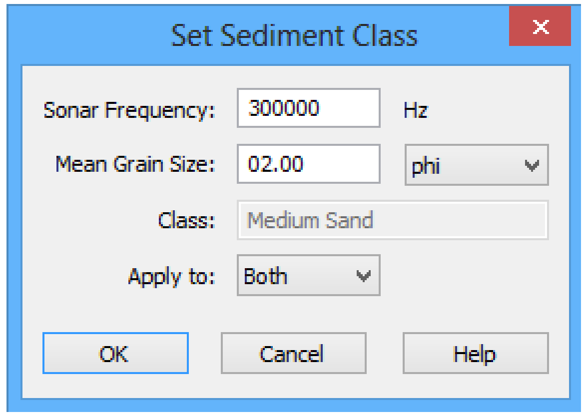
After the beam pattern is generated, you can define the sediment type in the selected area to properly correct said beam pattern. See also “Sediment Analysis” on page 440.

To set sediment class:

Menu	Process > Beam Pattern > Sediment Class
Tool	

1. Load a beam pattern
2. Select the Sediment Class command.

The Set Sediment Class dialog box is displayed.



3. Adjust the *Sonar Frequency* value.
4. Set the *Mean Grain Size* in units set from drop-down list: `phi, mm or mil.`

The *Class* field displays a short description of the sediment indicated by the mean grain size. This will change with the values in the Mean Grain Size field. These values are drawn from the Grain Size table. See "Grain Size Table" on page 443.

5. In the *Apply to* field, select to apply the class to: Port, Starboard or Both.
6. Click **OK**.

New Mosaic

A mosaic can be created directly from a single track line, from a selection of two or more track lines, or from all track lines open in the project.

Mosaics can be created using one of these processing engines:

- SIPS Backscatter
- SIPS Side Scan
- GeoCoder

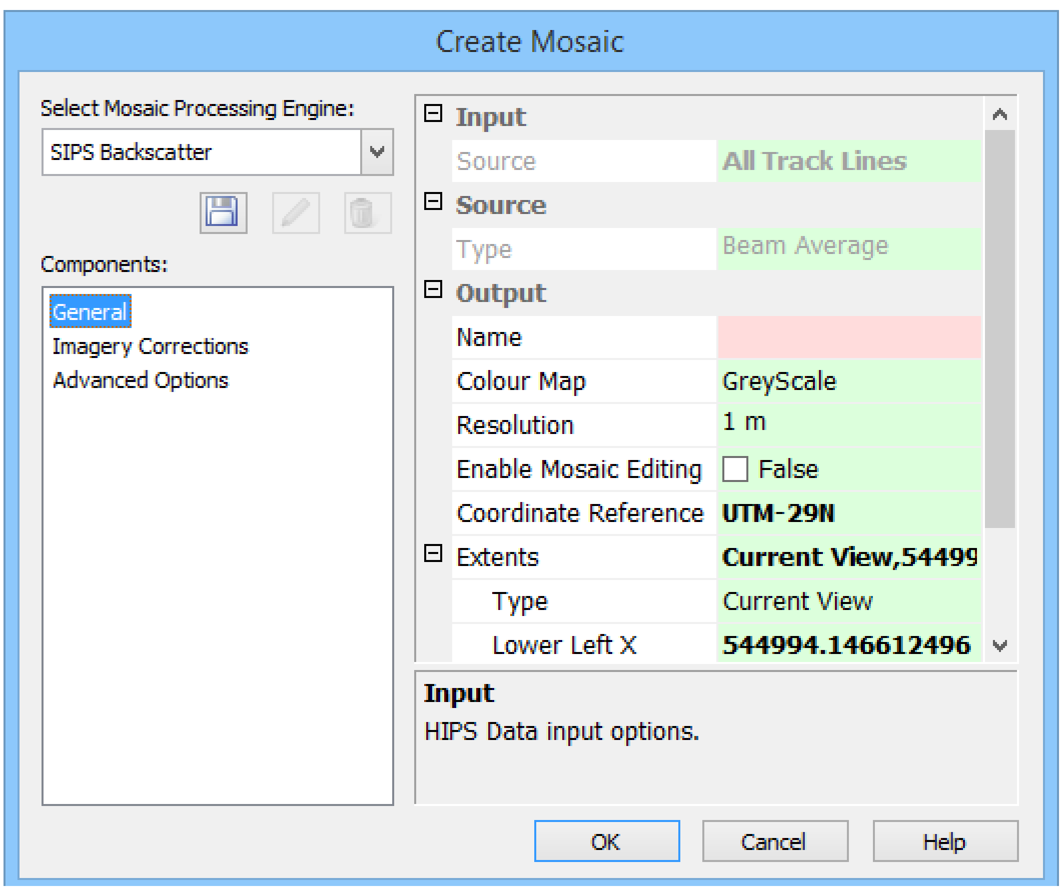
For more information see “Processing Engines” on page 411

To create a mosaic:

1. Open a project in HIPS.
2. Select the Ship Track Lines layer in the Layers window.
3. Select one or more track lines.
4. Zoom/pan to display the extents of the data that you want included in the mosaic.
5. Select a New Mosaic command.

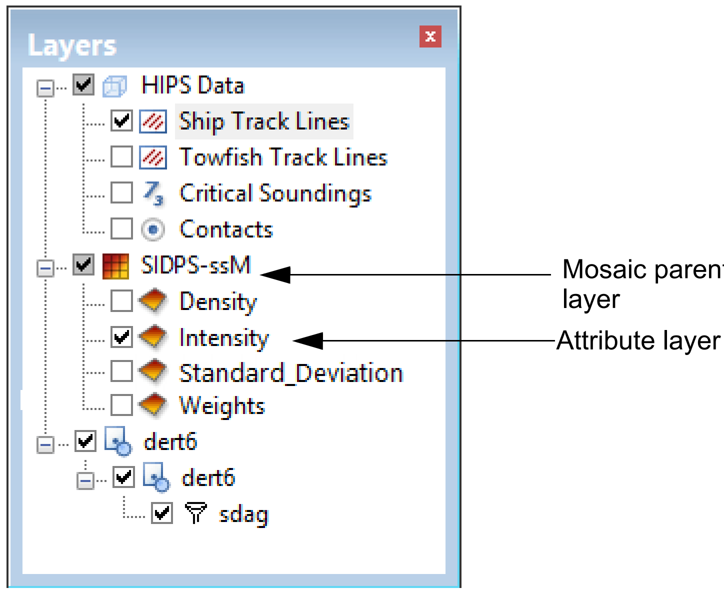
Menu	Tools > Mosaics > New
Tool	

The Create Mosaic dialog box is displayed.



6. Select a Mosaic Processing Engine from the list.
7. Set “General options” on page 423, “Apply Imagery Corrections” on page 425 and “Advanced Options” on page 428.
8. [Optional] Click **Save Template** to save settings as a template to use in future mosaic creation.
9. Click **OK** to create the mosaic.

The new mosaic is displayed in the Display window and is listed in the Layers window.



A new Standard Deviation layer has been added to mosaics. This layer contains the standard deviation of all samples contributing to each cell in the mosaic.

General options

Input	The source of the data:
Source	• To create the mosaic from the selected line or lines, set the *Source* field to Selection. • To create the mosaic from all open lines, select the All Track Lines option in the *Source* field.
Source	The type of data to be read for processing into a mosaic
Type	For SIPS Backscatter the type of data is set to Beam Average. For GeoCoder, select Time Series, Side Scan or Beam Average.
Channel	For SIPS Side Scan select Port channel, Starboard or Both to be read.
Output	Name and destination of the mosaic file.
Name / File Name	Click **Browse** in the *File Name* field and save a name for the mosaic. There must be a defined name or the mosaic will not be created.
Colour Map	The default setting will produce a mosaic in grey scale. To change colour, click in the field and select another mapping from the list.
Resolution	Type a numerical value for the resolution of the mosaic image. It will be displayed in the units as set in Options > Display Units > Distance.
Auto-resolution	Applies only to mosaics created with GeoCoder engine. If set to True, a resolution will be set based on density of data. In this case, Resolution option is disabled.
Enable Mosaic Editing	If set to True, the mosaic can be edited after creation.
Anti-aliasing	Anti-aliasing can minimize distortion artifacts when representing the high resolution imagery at a lower resolution. Applies only to mosaics created with the GeoCoder engine. If set to True the mosaic image will be smoothed using anti-aliasing.
Coordinate Reference System	Displays the coordinate reference system for the mosaic. Click **Browse** to select another reference system.

<table>
<tr><td>Extents</td><td>Displays the data area that will to be used for the mosaic.</td></tr>
<tr><td align="right">Type</td><td>Current View displays the coordinates of the current view of data in the Display window. These are displayed as Lower Left and Upper Right X and Y values. This is the default setting.

To change the area to be included in the mosaic:
1. Select Custom, and zoom/pan to the extents of the data that you want to include.
2. Select Current View again and the coordinates will be updated.

Alternatively:
1. Select Pick from screen from the Type field. This changes the shape of the cursor.
2. Use the cursor to drag a box around an area to mosaic. This box can be resized using the controls on the lower right and upper left corners of the box. It can also be dragged to a new location.

Coordinates for the area will update as you draw, resize or move the box.</td></tr>
<tr><td align="right">Lower Left X</td><td rowspan="4">Ground coordinates for the data at the lower left and upper right corner extents of data.

For Current view the extents are that of the Display window.

For Pick from screen, the extents are that of the box. f</td></tr>
<tr><td align="right">Lower Left Y</td></tr>
<tr><td align="right">Upper Right X</td></tr>
<tr><td align="right">Upper Right Y</td></tr>
</table>

10. Enter values for *General*, *Imagery Corrections* and *Advanced* options.

Apply Imagery Corrections

Various imagery corrections can be applied, depending on the type of data and the processing engine. See "Types of Imagery Correction" on page 432 for more information

Imagery Corrections for SIPS Backscatter

AVG	Use a moving average filter to remove the angular response of sediment from the imagery. See "Angle-Varying Gain" on page 433.
Enable	Set to True to apply AVG correction. Set to False by default.
AVG Window Size	*Window Size* option is activated if AVG option is set to True. Set the number of across track samples to include in the moving average filter.
Beam Pattern	Use a beam pattern file to uniformly remove angular artifacts from sonar data. See "Beam Pattern Correction" on page 412
Enable	Set to True to apply the selected beam pattern file.
File Name	1. Click **Browse** in the *File Name* field and enter a name for the beam pattern file. 2. Click **Save**.
File Operation	If the beam pattern file selected above already exists, this field displays options to Update or Overwrite the existing file.
Local absorption	Correction for transmission loss using temperature and salinity values.
Temperature	Set temperature value in degrees. Default value is 8.00
Salinity	Set salinity value as parts per thousand. Default value is 35 parts per thousand.
Angle From Nadir	Use an angle across-track from directly below the transducer (0 degrees) to set how much data is included in the beam pattern file.
Transducer1	This field displays the angle from nadir values set for Transducer1 in *Port* and *Starboard* fields.
Port	Set values in degrees of the angle to port from nadir.
Starboard	Set values in degrees of the angle to starboard from nadir.
Transducer2	This field displays the angle from nadir set for the second transducer in a dual system in *Port* and *Starboard* fields.
Port	Set values in degrees of the angle to port from nadir.
Starboard	Set values in degrees of the angle to starboard from nadir.
DTM	Settings for the digital terrain model used to compute the local bottom slope used in the calculations of real incidence angle and ensonified area.
Source	Select the DTM source to use.
Attribute	Select the surface attribute to use.

Imagery Corrections for SIPS Side Scan

TVG	Time-Varying Gain: Gain varied by time values is applied so that inner-most samples have the least gain and the outer-most samples have the highest gain correction. See also "Time-Varying Gain" on page 432.
Enable	Default setting is None. (No TVG correction is applied.) • Select Linked to apply the correction value set for the Port channel to the Starboard channel as well. • Select Both to apply the values set in the Port and Stbd fields. • Select Port only to apply correction only to that channel. • Select Stbd Only to apply correction only to that channel.
Port	Set a correction value for port side imagery.
Stbd	Set a correction value for starboard side imagery.
Gain	Applies a constant value to all data. see also "Gain" on page 432.
Enable	Default setting is None. No Gain correction is applied. • Select Linked to apply the correction value set for the Port channel to the Starboard channel as well. • Select Both to apply the values set in the Port and Stbd fields. • Select Port only to apply correction only to that channel. • Select Stbd Only to apply correction only to that channel.
Port	Set a correction value for port side imagery.
Stbd	Set a correction value for starboard side imagery.
Gain Normalization	Compensates for the along-track motion of the sonar which can cause variations in intensity. See also "Gain Normalization" on page 433.
Enable	Set to True to apply normalization. (Set to False by default).
Window Size	*Window Size* option is activated if *Enable* is set to True. Set the number of pings to include in the moving average filter.
Beam Pattern	Use a beam pattern file to uniformly remove angular artifacts from sonar data. See also "Beam Pattern Correction" on page 412.
Enable	Select either Port, Starboard or Both channels to apply beam pattern correction to. Default setting is None. (No correction is applied)
File Name	1. Click **Browse** in the *File Name* field and enter a name for the beam pattern file. 2. Click **Save**.
File Operation	If the beam pattern file selected above already exists, this field displays options to Update or Overwrite the existing file.
Despeckle	Smooths imagery by removing "noise". Pixels are removed If they have an intensity level outside a specified strength compared to their neighbouring intensity levels. See also "Despeckle" on page 432.

Enable	Set to User Defined to apply correction. Default setting is None. (Despeckle is not applied.)
Strength	Value field is activated when Despeckle option is enabled. Set the strength of the filter as a percentage from 10 to 90.

Imagery Corrections for GeoCoder

Imagery Correction	Options which can be applied during mosaic creation.
TVG	Time-Varying Gain: correction so that soundings from any depth will have comparable signal strengths after TVG is applied. Gain applied varies so that inner-most samples have the least gain and the outer-most samples have the highest gain correction. Select: • None (no correction will be applied) • Automatic - an automatic correction is applied
Gain	Applies a constant value to all data. Select: • None (no correction will be applied) • Automatic - an automatic correction is applied
AVG	Angle Varying Gain correction removes a low frequency noise caused by angular gain artifacts, using a moving average window to calculate mean intensity as a function of angle. Select one: • None – no AVG correction is applied. • Flat – All samples are normalized to a single value resulting in a flat mean intensity. • Trend – Samples with grazing angles between 30° and 60° contribute to a linear least square method to fit total backscatter curve. • Adaptive – Uses Flat option for grazing angles smaller than 15°, otherwise Trend option is used.
AVG Window Size	Activated if AVG option is set to True. Set the number of across track samples to include in the moving average filter.
Despeckle	Smooths imagery by removing “noise”. Pixels are removed If they have an intensity level outside a specified strength compared to their neighbouring intensity levels. Select either: Weak, Moderate, Strong, Very Strong
Beam Pattern	Use a beam pattern file to uniformly remove angular artifacts from sonar data. See “Beam Pattern Correction” on page 412.
Enable	Set to True to apply a selected beam pattern file.
File Name	1. Click **Browse** in the *File Name* field and enter a name for the beam pattern file. 2. Click **Save**.

Advanced Options

Advanced Options for SIPS Backscatter Mosaic

Option	Description
Filter	
Intensity	A min/max range filter for the final compensated intensities from the mosaic. Displays the values entered below.
Minimum	Set the minimum of the range.
Maximum	Set the maximum of the range.
Options	
Sound Velocity	Settings for the source of sound velocity and its value in distance per second. Displays the selections set in the following two fields.
Source	Set the source of the surface sound velocity: • Select Surface Sound Speed to apply the value of the Surface Sound Speed data in each line. • Select User Defined to apply the value set in the Value field. If Surface Sound Speed is selected, but not available, the value in the Value field is used.
Value	Type a sound velocity value to be used if User-Defined option is set as *Source*, or if a surface sound speed is not available in the raw data files..

Advanced Options for SIPS Side Scan mosaic

Option	Description
Filter	
Acrosstrack/ Altitude Ratio	This option filters data starting from nadir extending across-track to a distance determined by the towfish altitude and the given ratio, i.e. Acrosstrack/Altitude = ratio and distance to filter Acrosstrack = ratio * Altitude. This field displays the values set in the Minimum and Maximum fields.
Enable	Enables the ratio filter and applies it according to the option selected. • None - Acrosstrack/Altitude Ratio filter is not applied. • Minimum - values below the minimum ratio value will be filtered. • Maximum - values greater than the maximum ratio value will be filtered. • Both - values below and above the Minimum and maximum will be filtered.
Minimum	The ratio value below which the data be filtered out. • For example, if the minimum fish altitude limit is 5 metres, then the Min Acrosstrack/Altitude Ratio option is not used on any sonar profiles where the altitude is less than 5 metres.
Maximum	The ratio value above which the data be filtered out. • For example, if the maximum fish altitude limit is 10 metres, then the Max Acrosstrack/Altitude Ratio option is not used on any sonar profiles where the altitude exceeds 10 metres.
Limit	Cut-off limits for the altitude value used in the ratio. The filter is not applied when these values are exceeded.
Enable	Enables a cut-off limit and applies according to the option selected. • None - Limit is not applied. • Minimum - minimum altitude cut-off value is applied. • Maximum - maximum altitude cut-off value is applied. • Both - values below and above the Minimum and maximum will be applied.
Minimum	The minimum altitude cut-off
Maximum	The maximum altitude cut-off.
Across-track Distance	Filter to excluded a fixed distance across-track from nadir, regardless of altitude. This field displays values set in the Minimum and Maximum fields.

Option	Description
Enable	Enables the distance filter and applies it according to the option selected. • None - Acrosstrack Distance filter is not applied. • Minimum - values below this distance from nadir will be filtered. • Maximum - values greater than this distance from nadir will be filtered • Both - values below and above the Minimum and maximum will be filtered.
Minimum	The distance in value below which the data be filtered out. Distance units set in Tools > Options > Display Units.
Maximum	The distance value above which the data be filtered out. Distance units set in Tools > Options > Display Units.
Intensity	A min/max range filter for the final compensated intensities from the mosaic. Displays the values entered below.
Minimum	Set the minimum of the range.
Maximum	Set the maximum of the range.
Options	
Use Bathymetry to Register Image	Set to True to enable registration of the imagery using bathymetry location data. Set to False (default value) a flat sea floor assumption will be used.
Gyro	Select a heading source for processing. • Automatic - uses Towfish Gyro, then Ship Gyro, then Course Made Good, in that order. Course Made Good - direction (in degrees) calculated from one navigation fix to the next. Ship Gyro - the heading orientation of the vessel Towfish Gyro - heading orientation of the towfish
Smooth gyro	Set to True to apply gyro that has been smoothed in Attitude Editor.
Sound Velocity	Set the sound velocity value applied using slant range correction.
Extrapolate	Extrapolate heading and navigation at the beginning and end of lines in the mosaic to a set time.
Enable	Set to True to extrapolate data at beginning and end of lines. Set to False to omit extrapolation.
Time	This option is activated if Enable is set to True. Set maximum extrapolation value in seconds.
Altitude Offsets	
Height	Set height value in the units of measure set in Tools > Options > Display Units > Elevation.
Time	Set time in seconds.

Advanced Options for GeoCoder mosaic

Option	Description
Filter	
Angle From Nadir	Use an angle across-track from directly below the ship (0 degrees) to set how much data is included in the beam pattern file. Value applies to both port and starboard angles.
Minimum	Minimum value in degrees of the angle from nadir. Default is 0 degrees.
Maximum	Maximum value in degrees of the angle from nadir. Default is 87 degrees.
Intensity	Filter the final compensated intensities of the mosaic using range values.
Minimum	Minimum value for the range (default value is -100dB)
Maximum	Maximum value for the range (default value is 0dB.
Options	
Smooth Gyro	Set to True to apply smoothing coefficients for gyro data. Default setting is False. No smoothing is applied.
DTM	Settings for the digital terrain model used to compute the local bottom slope used in the calculations of real incidence angle and ensonified area.
Source	Select the DTM source to use.
Attribute	Select the surface attribute to use.

Types of Imagery Correction

These corrections to imagery can be applied during the creation of a mosaic.

Time-Varying Gain

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

Gain varied by time values is applied so that inner-most samples have the least gain and the outer-most samples have the highest gain correction.

TVG is calculated as:

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

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

Gain

Apply a uniform gain correction without applying the any time-dependent gains by using only the Gain controls.

The calculation for gain correction is:

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

where gain is the value from the property field.

Beam Pattern Correction

Beam pattern correction uniformly removes angular artifacts from sonar systems. Correction is applied using a beam pattern file. See “Create Beam Pattern file” on page 413.

Despeckle

Despeckle 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.

Angle-Varying Gain

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

Gain Normalization

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

Properties of Mosaics

Most of the information in the Properties for a Mosaic parent layer is read-only, however, the following 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
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 mosaic are the same as for other surface attribute layers. See "Attribute Layer Properties" on page 222.

Edit Mosaics

Mosaics can be edited for brightness and contrast through the Properties window. To add or removed component lines, see "Add Lines to a Mosaic" on page 437.

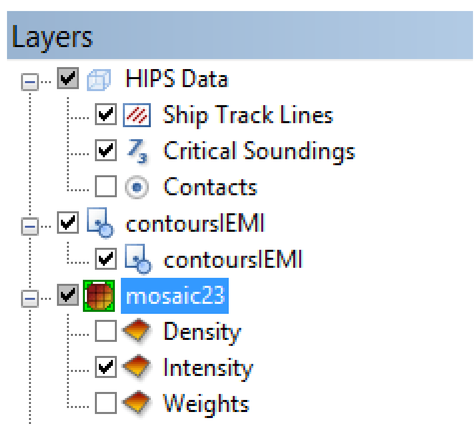
While you may have more than one mosaic open in the display, only one mosaic can be edited at a time.

> Mosaics created as field sheet layers in HIPS and SIPS prior to version 9.0, can be opened comparison purposes, but they cannot be edited.

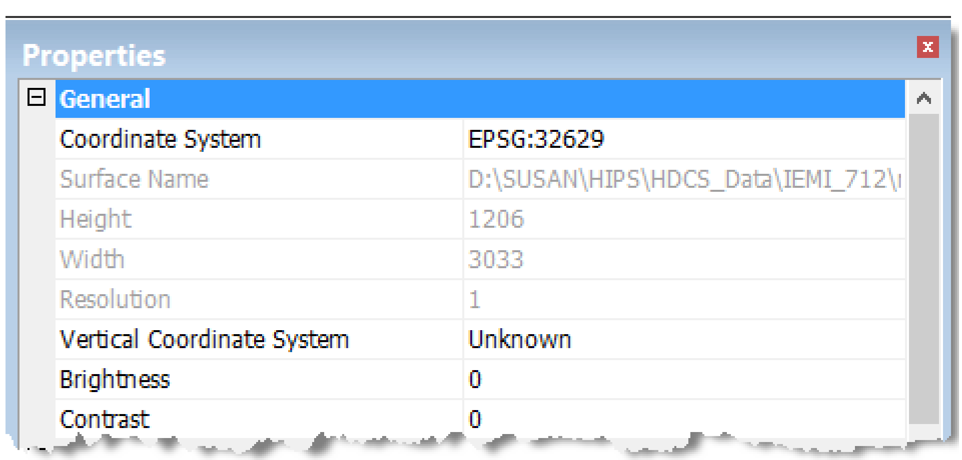
To edit a mosaic:

1. Open the mosaic.
2. In the Layers window, right-click on the parent layer, and select Edit from the pop-up menu.

The icon for the selected mosaic layer icon in the Layers will be highlighted in green indicating that the mosaic is now in Edit mode.



3. Open the Properties window.



Change the values in the Brightness and Contrast fields to adjust the display of the mosaic.

4. To adjust brightness of the image, enter negative values to increase the brightness, positive to decrease.

   Using small increments, for example, 0.5 may give the best results.

5. To adjust contrast, enter positive values to heighten the contrast, and negative values for less contrast.

See also "View Surface Properties" on page 220.

You can also change properties for a mosaic layer using the Properties window. See "Attribute Layer Properties" on page 222.

Add Lines to a Mosaic

To add line data to an existing mosaic:

1. Open the mosaic.
2. Select a line or lines to be added to the mosaic.

Menu	Tools > Mosaics > Add to

3. Select the Add to Mosaic command.

The Add to Mosaic dialog box displays the options and settings that were used to create the original mosaic.

By default, the Source field is set to Selection, which will add only the selected lines.

4. If you are adding to a side scan mosaic, select the Channel to be added to the mosaic: Port, Starboard or both channels.
5. Click **OK** to add the lines with the same option settings used to produce the original mosaic.
6. Otherwise, adjust other options and click **OK** to add the lines with different settings.

The mosaic will be regenerated to include the selected line or lines.

> If you add a line that is already part of the mosaic, and this line data has been updated since the mosaic was created, the line is effectively removed and then replaced in the mosaic with its updated values.

Remove line from mosaic

To remove a component line from a mosaic:

1. Select the Track Lines layer.

Menu	Tools > Mosaics > Remove from

2. Select the line to be removed.
3. Select the Remove From Mosaic command.
4. Select the mosaic name to remove the line from.

The line is removed.

Delete Mosaic

To delete a mosaic:

1. Right-click on the mosaic layer in the Layers window.
2. Select Delete Mosaic from the pop-up menu.

Compute Statistics

You can use Compute Statistics command to generate the statistics for a selected mosaic, and output them to an ASCII file or to image. See “Compute Surface Statistics” on page 234.

Classify Mosaic

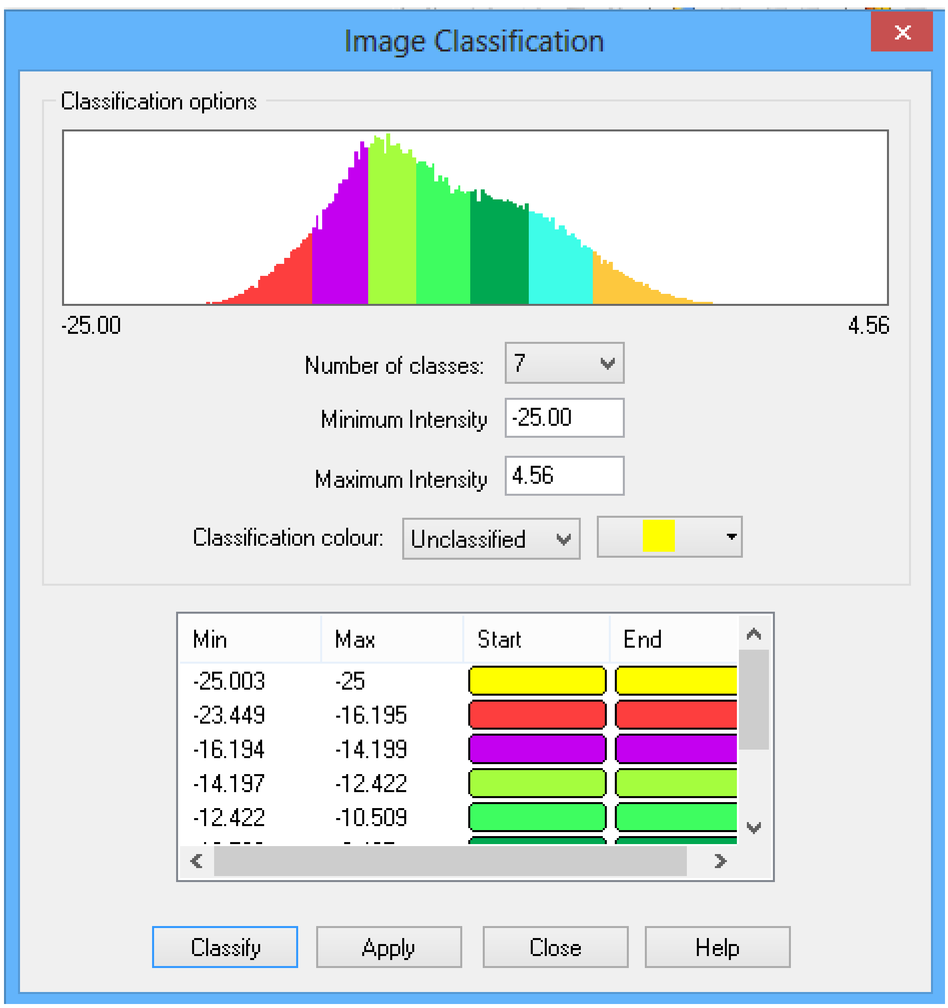
Classify mosaic image groups return-signal intensities (represented in the mosaic as pixel values) into classes. These classes are displayed in the mosaic using specific colours.

Menu	Tools > Mosaics > Classify
Popup	Classify

To determine the range of intensities:

1. Select the Mosaic in the Layers window.
2. Select the Classify command.

The Image Classification dialog box is displayed.



The histogram at the top of the image classification dialog box shows the distribution of pixels in the selected mosaic. The pixels represent return signal intensities. The numbers at the bottom corners of the histogram indicate the range of values in the mosaic.

To determine the range of intensities:

1. From the *Number of classes* drop-down list, set the total classes for the mosaic.

2. Set a range of values for a class by typing values for *Minimum intensity* and *Maximum Intensity* in their respective fields.
3. [Optional] Select a colour for a class by selecting a class from the *Classification* colour drop-down list, and the colour picker.

This colour will be displayed beside the class name.

4. [Optional] Select another colour from the colour picker, or create a custom colour from the standard Windows colour palette.
5. Repeat Steps 4 to 7, as needed.
6. Click **Classify**.

A colour table is generated according to the number of classes you selected. Pixels are grouped according to class starting from the minimum to the maximum pixel value.

The histogram is refreshed to show the pixels coloured according to the class designation. When the dialog box is closed, the intensity values in the mosaic will be displayed with these colour classifications.

To save the colours and their classes to a `.CRF` (colour range file) for future use:

7. Click **Apply**.
8. In the Save As dialog box, type a name for the file and click **Save**.
9. Click Close.
10. Refresh the Display and the mosaic will be displayed in the colour values set in the Image Classification.

Sediment Analysis

Use the Sediment Analysis tool to determine sediment type, by analyzing the angular responses of data processed in the GeoCoder engine.

Data is analysed with respect to the sediment angular response models in order to determine an average grain size. This average grain size is then cross-referenced to the customizable look-up table in order to provide a textual response. (See “Grain Size Table” on page 443.)

The sediment analysis results can be seen in the Sediment Analysis Graph window that displays the average angular response for a given range of data. (See “Sediment Analysis graph” on page 442.)

Sediment analysis results can also be exported to ASCII. (See “Sediment Analysis to ASCII” on page 540.)

“Advanced Mode” on page 443

“Grain Size Table” on page 443

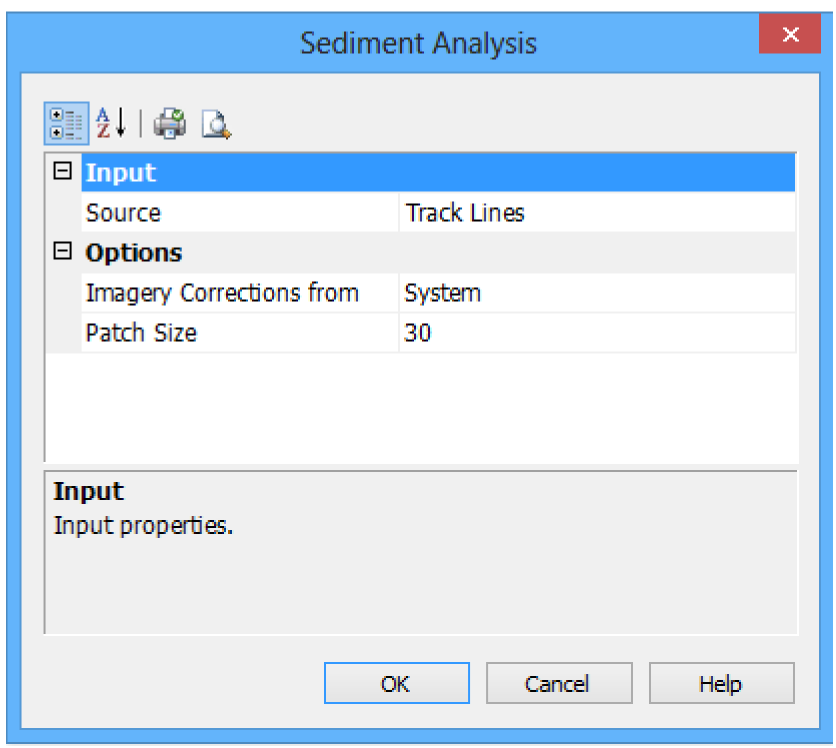
“Sediment Analysis graph” on page 442

To run a sediment analysis:

Menu	Process > Compute > Sediment Analysis
Process	

1. Select the Sediment Analysis command.

The Sediment Analysis dialog box is displayed.



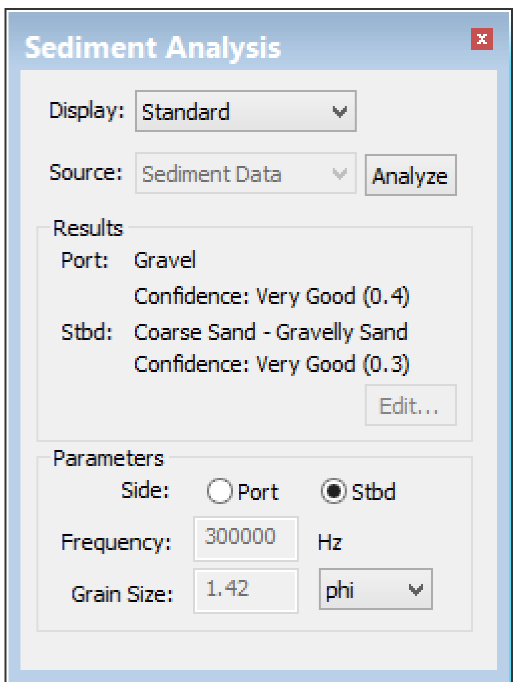
2. Set the following options:

Option	Description
Input	
Source	Selection: analyze selected lines Track Lines: apply to all open track lines.
Options	
Imagery Corrections	The corrections that will be used when determining sediment type. Set by the system by default.
Patch Size	Type the number of pings over which the sediment analysis will be performed.

Menu	Tools > Editors > Sediment Analysis
Tool	 on Process menu

3. Click **OK**.
4. Select the Sediment Analysis Editor command.

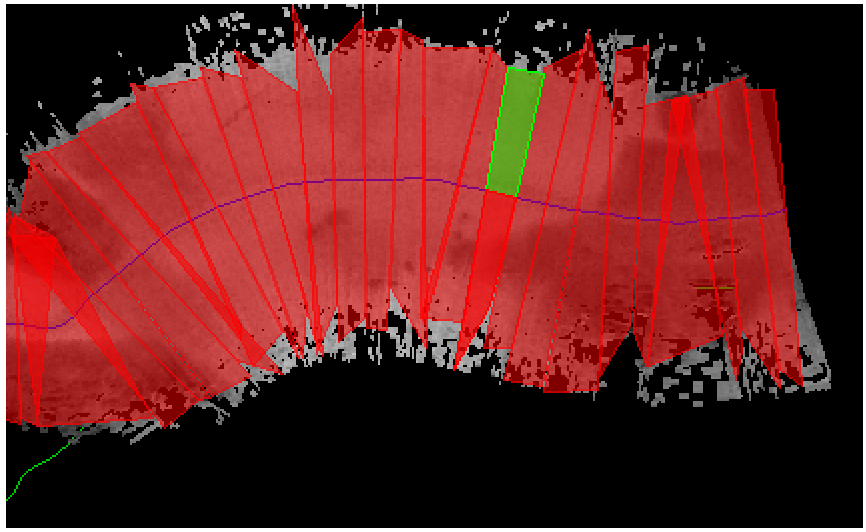
The Sediment Analysis Editor is opened.



The results of the analysis are displayed in the Results section of the editor page. Displayed are the type of sediment, based on the values set in the Grain Size table. Also shown is the confidence level in the result (the smaller the value, the higher the confidence level).

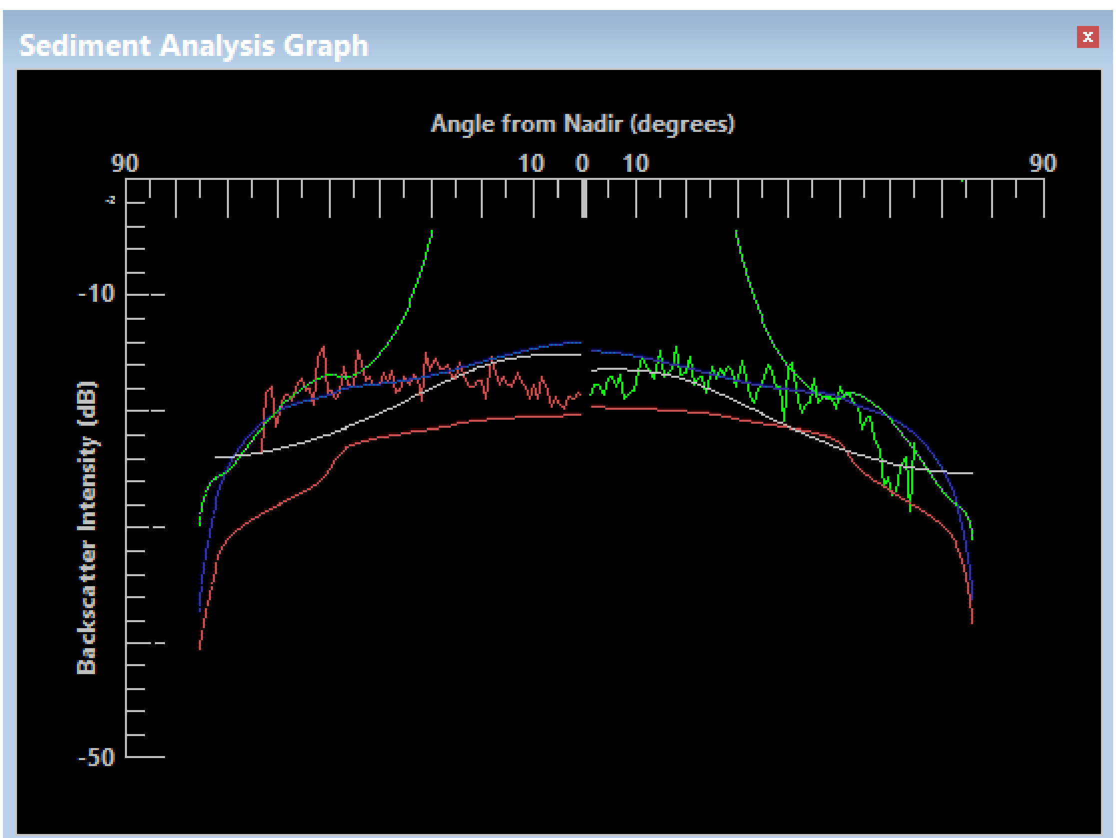
Further analysis of sediment data can be done in Advanced mode. See "Advanced Mode" on page 443.

The sediment analysis patches are displayed on the selected line or lines in the Display window. In the image below the sediment patches are in red. One selected patch shows in port and starboard colours.



Sediment Analysis graph

The Sediment Analysis Graph is also displayed.



The graph can display three kinds of backscatter, or the total backscatter or all four, colour coded as follows:

- Light green line is Interlaced Backscatter: occurring at the sea floor/water interface, this is the main component of the initially returned acoustic energy.
- Light red line is Volume Backscatter: secondary in time, this is sound energy returned or scattered from within the sediment. The less homogeneous the sediment, the more the sound wave is disturbed.
- Yellow line is Kirchhoff Backscatter: This model accounts for the roughness of the sea floor particularly for grazing angles close to 90 degrees.
- Blue line is Total Backscatter: the combined backscatter from the three backscatter sources. This is analyzed to determine approximate grain size.

The line colours can be changed from these defaults in the Tools > Options > Display > Sediment Analysis dialog box.

Also shown are port and starboard lines in their traditional colours.

5. Select sediment patches in the Display window to view their data displayed in the sediment analysis graph.

Advanced Mode

Advanced mode is an extension of the Standard mode of sediment analysis and displays more parameters. This analysis compares the grain size from the data to the values on a customizeable lookup table. (See "Grain Size Table" on page 443.)

(The Advanced Mode is likely to be of more interest to users doing scientific research rather than processing a workflow.)

1. Highlight a sediment analysis patch.
2. Select Advanced from the *Display* field.
3. Select Sediment Data from the *Source* field in the Sediment Analysis Editor.
4. Click **Analyze**.

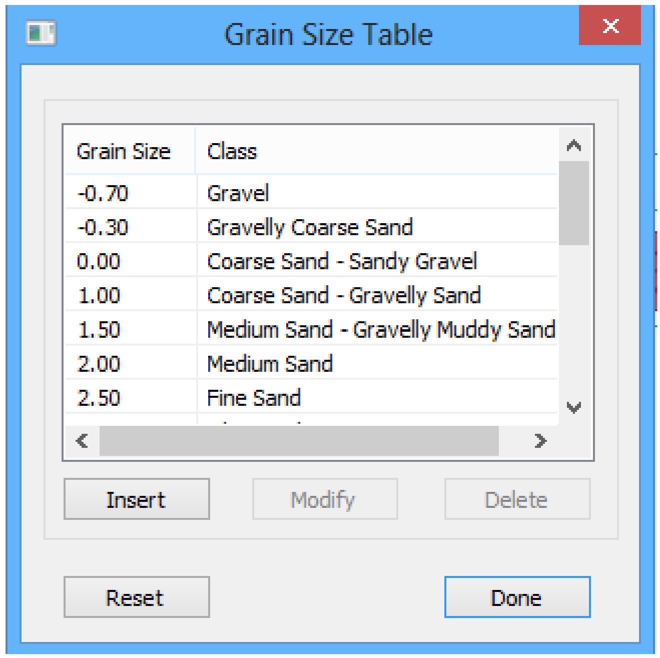
The sediment analysis results for that patch are displayed in the *Results* field. The input parameters for the results are shown in the *Parameters* fields. These values are read-only.

Grain Size Table

The classes of sediment which are displayed by the sediment analysis are based on values in the Grain Size table. This table can be customized while in Advanced mode.

1. Select the Advanced option from the Display field in Sediment Analysis Editor.
2. In the Results field, click **Edit** to open the Grain Size table.

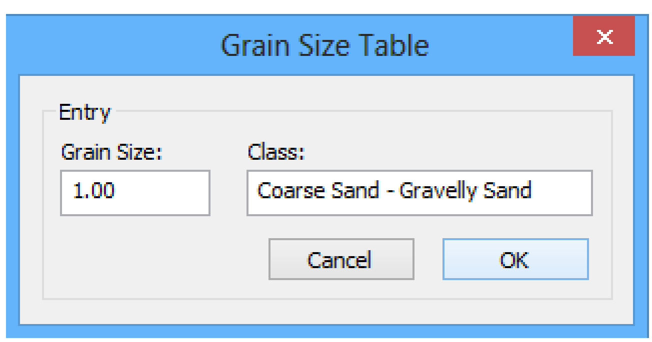
The Grain Size Table dialog box is displayed.



Grain size values and their descriptions can be edited using the **Modify** button. To modify a value:

3. Select a line in the table, and click Modify.

The Grain Size edit dialog box is displayed.



The selected grain size and its class are displayed.

4. Edit the field values and click OK.

New classes can be added using the Insert button, which will open the edit dialog box so a new entry can be added. The Delete button will remove a line from the table.

Use Reset to restore the default values.

Contacts

Contacts are representations of objects and features on the sea floor that are visible in the sonar data, for example, wrecks, pipe lines or other shoal features.

In Side Scan Editor you can georeference contacts and attach attributes attached to them using the Add Point or Add Line Contacts tools.

Contacts are stored in the HIPS file. The object created for each contact includes sufficient information to be exported to a supported format.

Contacts are placed directly in the waterfall view. Contacts are described in "Contacts" on page 98 in the Side Scan Editor section of the Editors Guide.

18

Create Product Surfaces

Create products such as contours, selected sounding layers and profiles. These products are stored on a feature layer and are saved in HOB format.

In this chapter...

Product Surfaces

Create products from finalized surfaces. Product surfaces maintain the designated soundings from the finalized Surface and can be used to create more cartographically correct contours. The finalized surface is smoothed into a generalized product surface that can be used for building contours, sounding selections and profiles. It can also be used to compile features.

Types of products that can be created on a product surface:

- Contours: Use a surface to output contours to a contour layer.
- Profiles: draw a cross-sectional graphic view of a height source
- Sounding Selection: Use a height source to selection a representative sounding set.

Generalize a Surface

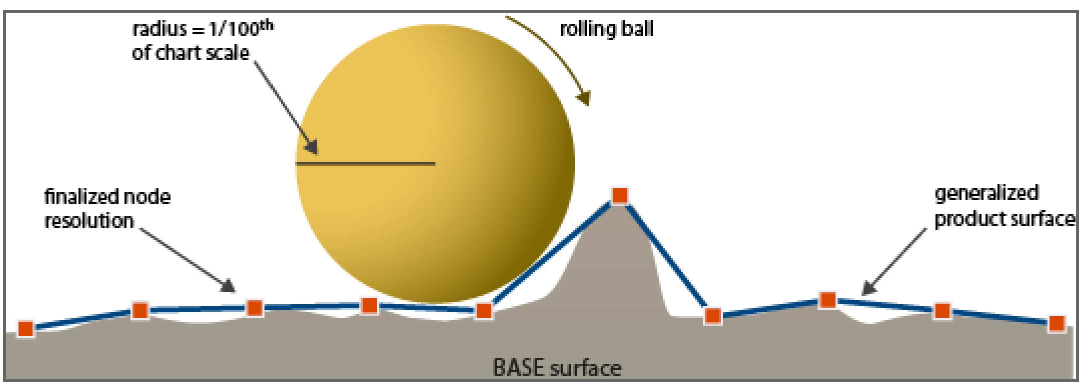
A generalized surface is a shoal-biased surface that can be used for creating more cartographically correct contours for navigation aids (for example, Electronic Navigational Charts).

These surfaces are created from a finalized surface, t is down-sampled so that some of the finer surface details are no longer visible, while the shoals are preserved. This surface can be used to directly generate contours and selected soundings.

Process

A surface is generalized through the process of 3D double buffering. This process smooths the surface to reduce clutter and highlight significant shoals.

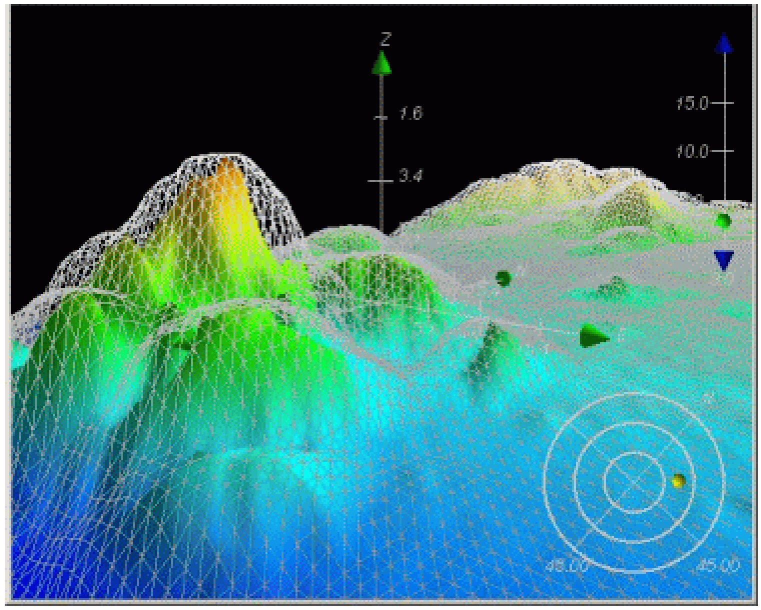
Double buffering is like rolling a ball over the surface at an interval determined by the surface's node resolution. The radius of the ball is determined from the chart scale (radius = 1/100 of chart scale). As the ball is rolled over the surface, the surface is smoothed, but the shoals are retained.



Defocusing

This optional process can also be applied to the generalization process. Defocusing spreads each shoal depth over an area defined by the horizontal error. The process uses an ellipsoid of rotation around each node with the horizontal error as the semi-major axis. Nodes within the ellipse are adjusted up to the surface of the ellipse.

In the following image, the webbed surface overlaying a surface represents the generalized area after defocusing has taken place. The shoals are preserved, but without the sharply defined detail of the original surface.

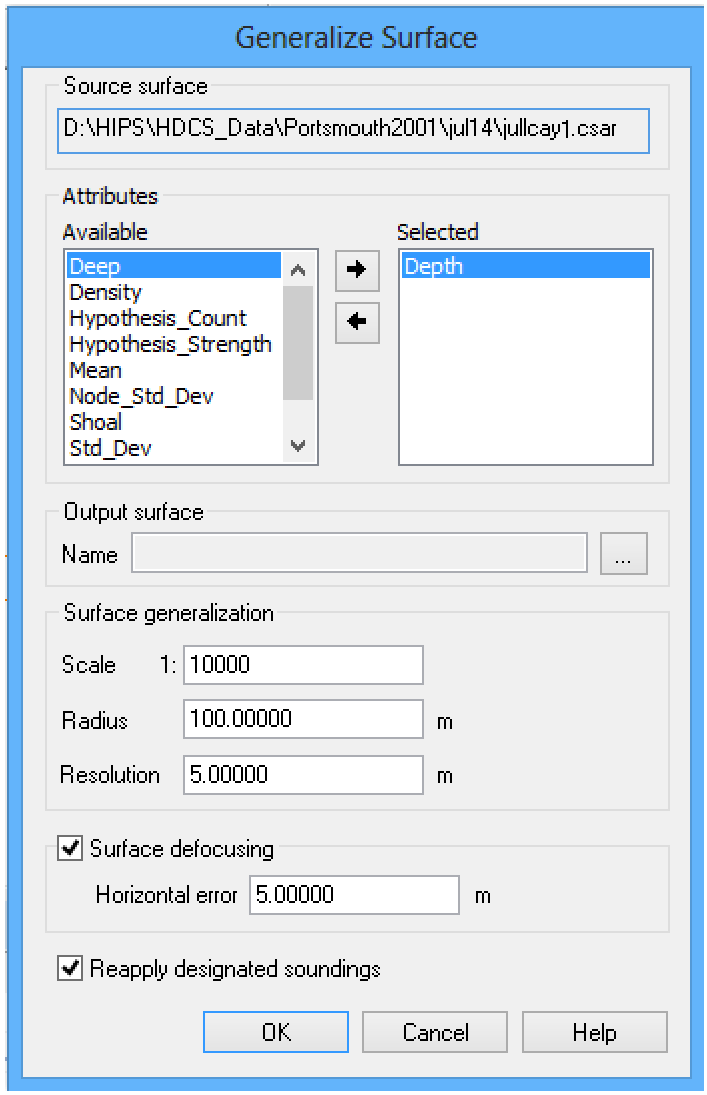

Z
1.6
3.4
15.0
10.0
45.00
45.00

Create a Generalized Product Surface

Menu	Tools > Surfaces > Generalize
Pop-up	Generalize

1. Create a product surface by generalizing a finalized surface
   Open the finalized surface.
2. Select the Surface layer in the Layers window.
3. Select the Generalize command.

The Generalize Surface dialog box is displayed.



The path and file name of the selected surface is displayed at the top of the dialog box.

4. [Optional] Select one or more attribute layers to add to the Depth layer in your product surface.
5. In the *Output* field, Browse to save a name and location for the new product surface.
6. Type a scale ratio that best suits the type of surface.

The scale of the Surface should match the scale of the product being created. For example, if you are creating a product surface that is going to be used for an ENC approach to a harbour, then use the appropriate scale for an approach.

The scale determines the radius used for generalizing the contours of the Surface. The *Radius* value changes as the *Scale* value is changed.

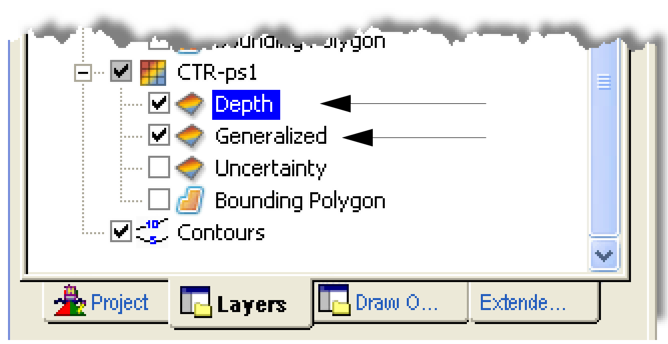
The *Resolution* value sets the node spacing of the generated surface.

7. Type a *Resolution* value, if needed.
8. [Optional] Select *Surface Defocusing* to implement this option.

The defocusing operation requires you to apply a horizontal error value. This value must be derived from the errors values associated with the data.

9. Type a *Horizontal Error* value.
10. Select the *Reapply Designated Soundings* check box to make sure that designated soundings are applied to the surface.
11. Click **OK**.

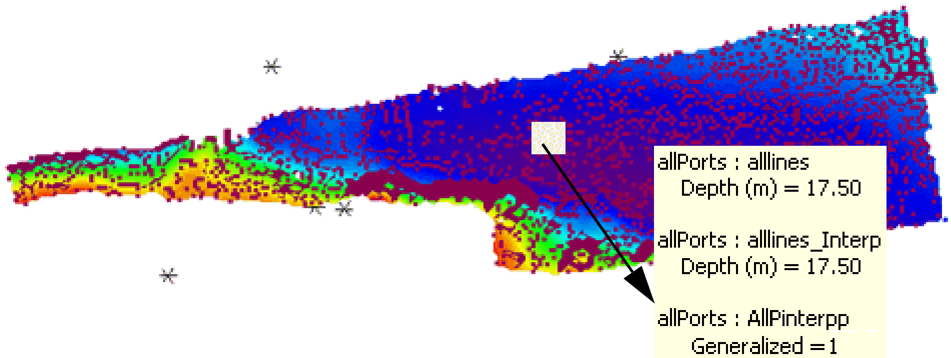
When the generalized surface is created, it is listed in the Layers window and shown in the Display window.



Also created is a Generalized layer which displays the nodes that were affected during the product surface-creation process.

- Nodes that have been modified are given a status flag of 1.
- Nodes that retain their original values are given a status flag of 0.

The status of the nodes is displayed in the tool tip, as illustrated below, which shows Generalized = 1.



In this example, the filter in the Properties for the Generalization layer has been set to show only the nodes which have been modified by the product surface. These are displayed as red dots in the example above.

When you create selected soundings from the Generalization layer, you can use SQL-type filters to omit nodes that have been modified. See "Enable Filter" on page 481.

Contours

Contours are created using the elevation values of a dataset as displayed in a surface. Contours are created on a feature layer.

In order for the contouring command to be enabled, you must have a feature layer or a HOB file open.

Menu	Tools > Contouring
Pop-up	Contouring… (parent layer in Layers window)

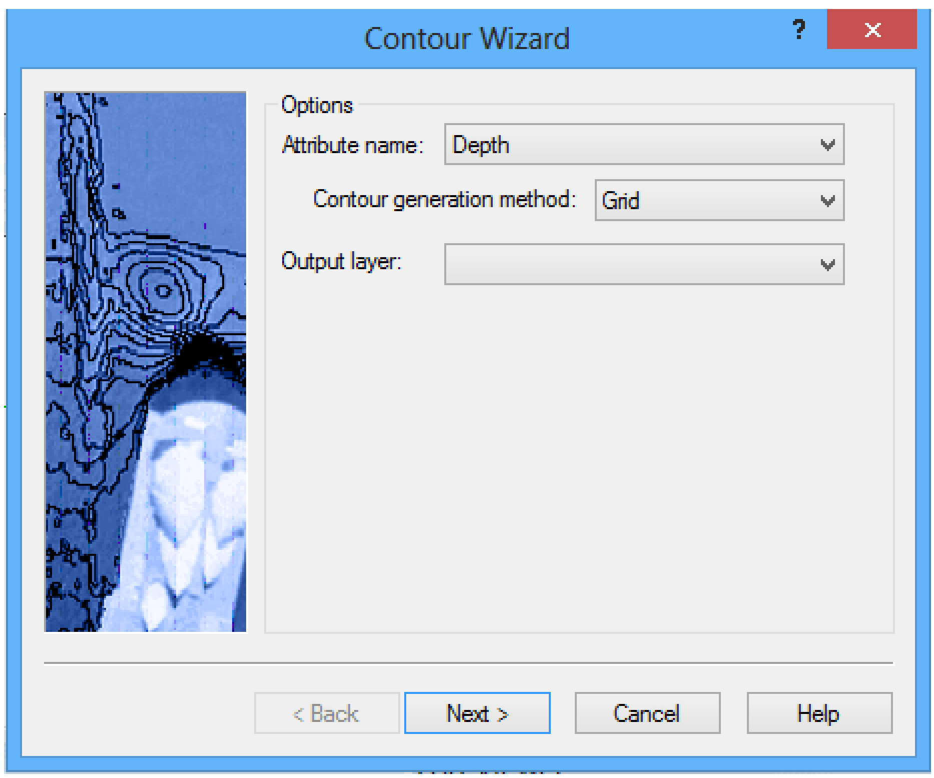
To create contours:

1. Create a feature layer, or open a HOB file.

For details on creating a new feature layer, see “Create a Feature Layer” on page 565 of the Reference Guide.

2. From the Layers window, select the layer you want to contour.
3. Select the Contouring command.

The Contour Wizard dialog box is displayed.

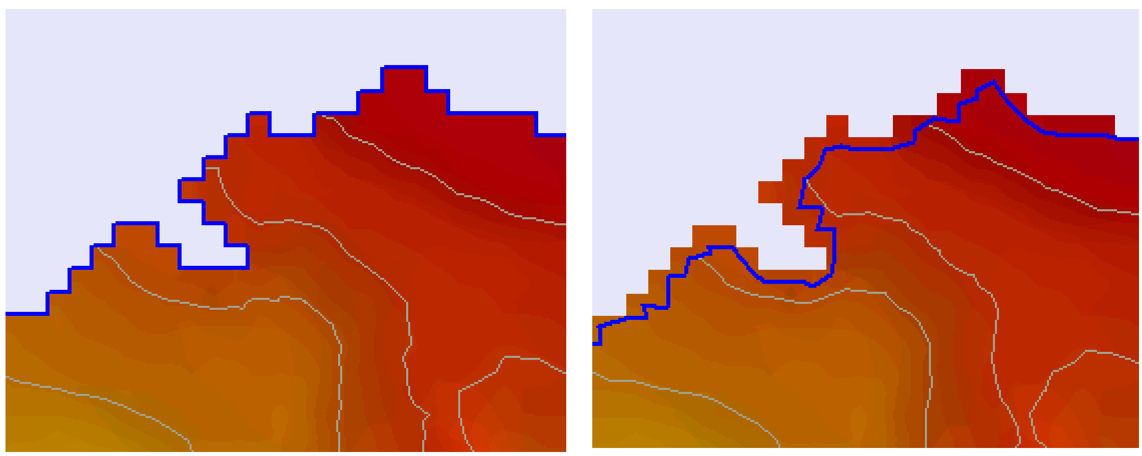


At this step specify the input and output settings for the contours. The input is the attribute layer of the source surface that will be used as the height source for creating the contours.

4. Select a layer to use as the height source from the *Attribute name* list.

The Contour generation method defines how the contour boundaries will be created. The options available in the drop-down list are dependent on the selected source data.

- For a TIN layer, this option is disabled as the contour boundaries will automatically be built using the points in the TIN.
- For an attribute band without true position data, contour boundaries can only be generated using the outer edges of the grid cells. In this case, *Grid* will be the only option in the list.
- For an attribute band with true position data, contour boundaries can be created using either the true positions for the data, or the outer edges of the grid cells. In this case, *True Position* and *Grid* will both be available in the list.



Grid contour boundary | True position contour boundary

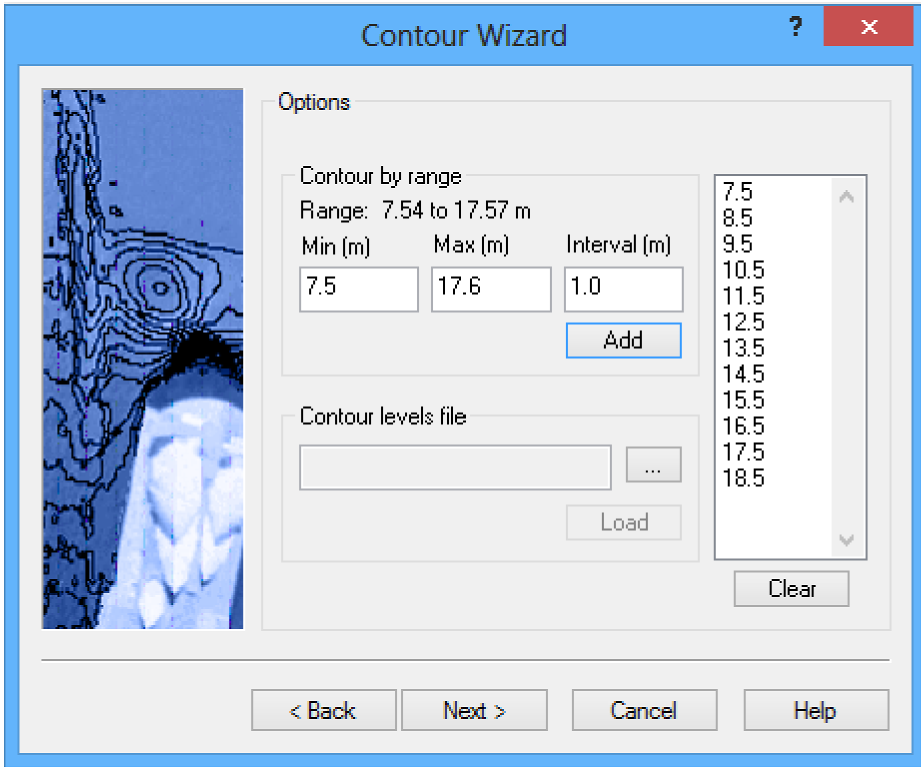
5. Select an option from the *Contour generation method* drop-down list.

The *Output layer* list displays feature layers or HOB file layers that can be used to store the contours.

6. Select a layer for storing the contours from the *Output layer* list.
7. Click **Next**.

Contour Wizard - Options

The Contour Wizard - Options dialog box is displayed.



Contours are created according to the elevation values in the source surface. You can have contours created at regular intervals in the elevation or you can load a file of predefined levels to have contours created at specific values. If you are creating contours by range, continue with the next step. If you are loading an existing file of contour levels, go to “Contour Using an Existing File” on page 457.

Contour by Range

The *Range* field provides the range of elevation values present in the source. When contouring by range, you can use the entire range, or you can specify a range of values for which to create contours. The units for the ranges settings are set in the Tools > Options > “Display Units” on page 630

8. In the *Min* field, type the minimum value to include in the range.
9. In the *Max* field, type the maximum value to include in the range.

The *Interval* is the elevation difference between each contour in the range of values.

10. Type an *Interval (m)* value.
11. Click **Add** to generate the contour levels for the specified range and interval.

The list box at the right will be populated with the contour levels. This list can be edited using a pop-up menu, if necessary. See "Contour Levels List Pop-Up Menu" on page 457 for more information.

12.Click **Next**.

Contour Using an Existing File

A contour levels file is a text file that contains a list of predefined levels at which to create contours. The following is an example of a contour levels file.

```
13.00
13.10
13.20
13.30
13.40
13.50
13.60
13.70
13.80
13.90
14.00
14.10
14.20
14.30
14.40
14.50
14.60
14.70
14.80
14.90
15.00
15.10
```

13.Click Browse (**...**) to select a contour levels file.

14.Click **Load** to apply the contour levels from the file.

The list box at the right will be populated with the contour levels. The list can be edited using a pop-up menu, if necessary. See "Contour Levels List Pop-Up Menu" on page 457 for more information.

15.Click **Next**.

Contour Levels List Pop-Up Menu

Once the contour levels list is visible, a pop-up menu is available for the list. The following commands are provided:

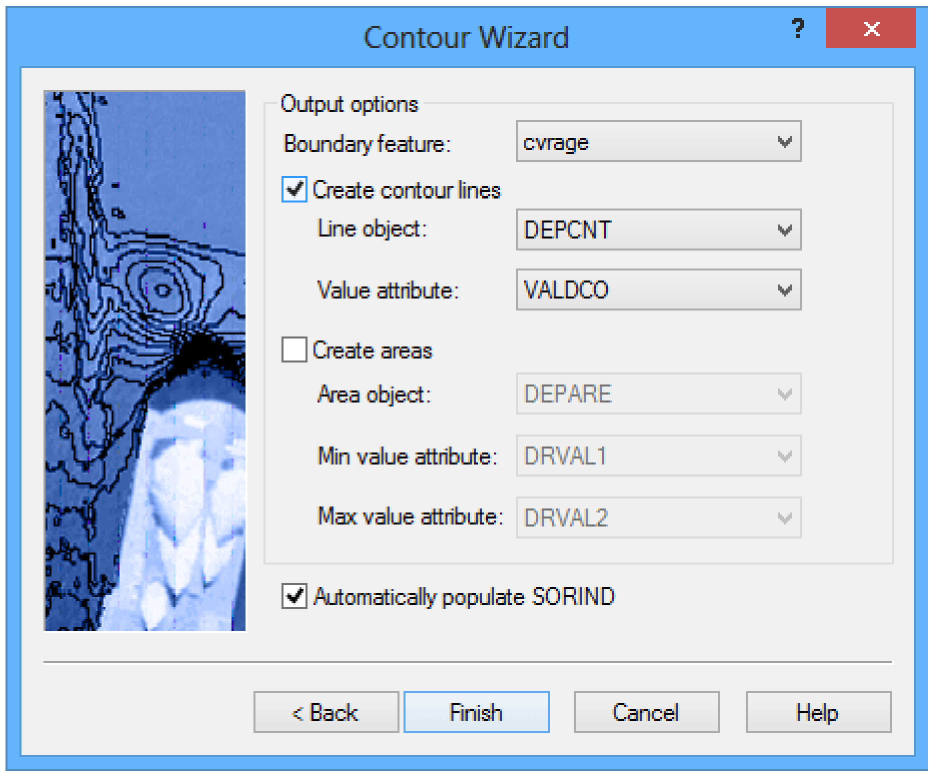
- *Insert*: Display the Add Contour Level dialog box to manually add a specific contour level to the list. This option is useful if a contour is needed at a specific level, but the level was not generated automatically.
- *Delete*: Remove the currently selected level from the list. To delete multiple levels, press and hold <Ctrl> while selecting the levels and then select this command.
- *Save*: Save the current contour levels list to a text file for future use with the *Contour levels file* field. Specify a name and

location for the file in the Save As dialog box that is displayed.

- *Clear*: Remove all current levels from the list. This functionality is also available through the **Clear** button.

Contour Wizard - Output Options

The Contour Wizard - Output Options dialog box is displayed.



In this dialog box, you define the object and attribute acronyms that will be used to create the features resulting from this procedure.

Boundary feature

A by-product of generating contours is a bounding polygon feature, which represents the inside and outside extent of the data used to derive the contour lines. By default, this feature object is created using the `cvrage` object acronym (Coverage Area); however, the possible acronyms are controlled by the catalogue of the feature layer being used to create the contours. If the `cvrage` acronym does not exist in the catalogue of the feature layer being used, the first acronym in the list will be selected instead. All open contours are connected to the boundary by connected nodes.

For the bMIO format, a metadata object such as M_QUAL can be used for the boundary object.

1. Select an object acronym for the *Boundary feature* from the list.

You have the option of outputting either contour lines or areas, or both.

Create contour lines

To create contour lines:

2. Select the *Create contour lines* option.

If using the Bathy DataBASE catalogue, the *Line object* field is set to DEPCNT by default. The default acronym then selected for *Value attribute* when using DEPCNT is VALDCO.

3. Select a *Line object* option. This is the object acronym that will be used to create the contours.
4. Select a *Value attribute* option. This is the attribute acronym that will be used to assign values to the contours.
5. Click **Finish** to create the contour lines.

Create areas

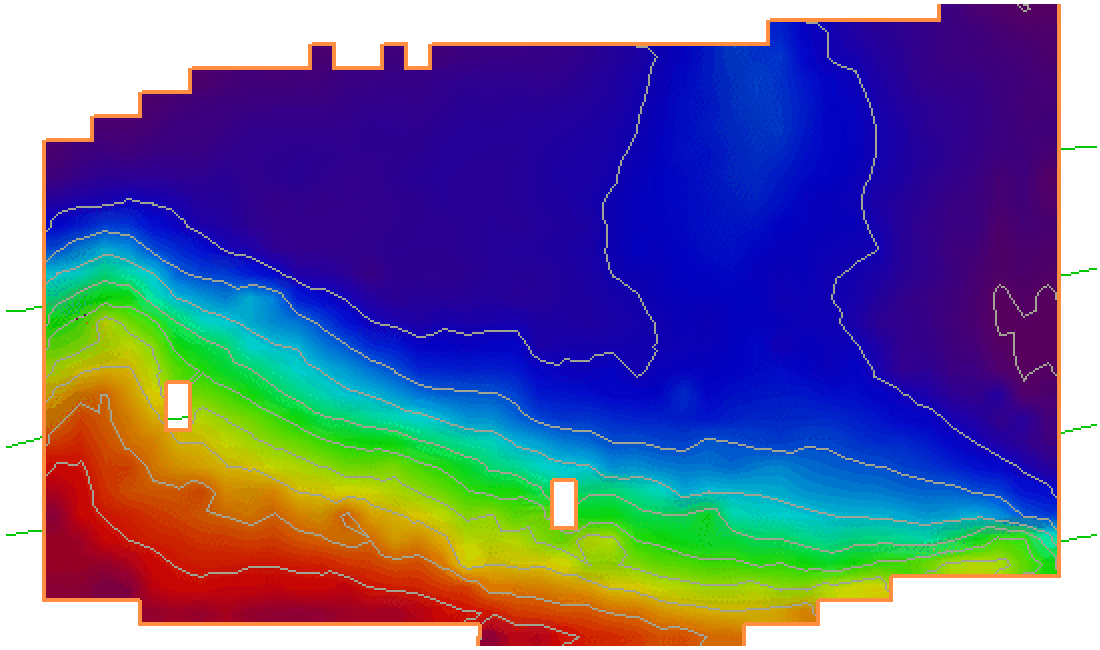
To create areas:

6. Select the *Create areas* option.

If using the Bathy DataBASE catalogue, the *Area object* is set to DEPARE by default. The default settings for *Min value* and *Max value* when using DEPARE are DRVAL1 and DRVAL2, respectively.

7. Select an *Area object* option. This is the object acronym that will be used to create the areas.
8. Select a *Min value attribute* option. This is the attribute acronym that will be used to assign minimum values for the areas.
9. Select a *Max value attribute* option. This is the attribute acronym that will be used to assign maximum values for the areas.
10. [Optional] Select *Automatically populate SORIND* to automatically populate the source attribute value for the contours and areas.
11. Click **Finish** to create the contour areas.

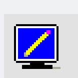
The contour line and/ or depth area features are created and visible in the Display window.



If the output layer is using the Bathy DataBASE catalogue, any isolation type contours will have the Isolation Type (`isotyp`) attribute populated automatically.

This attribute defines whether the contour contains a shoal (shoaler than the surrounding elevation values) or a deep (deeper than the surrounding elevation values), or is unknown. This attribute can be used to filter the contour features or to resolve isolations when creating depth areas. The `isotyp` value for a selected contour can be updated in the Attributes window.

If the number of contours to be created exceeds the amount of memory available to complete the procedure, an error message will be displayed. You may be required to restart the application to clear your virtual memory if this happens.

Menu	View > Refresh
Tool	
Key	<F5>

To view the new contour features:

1. Select the **Refresh** command.

The feature layer with the contour and area features can be saved to a HOB file using the File > Save command. You can change the colour display of the contours using the Properties for the feature layer.

Edit Contours

The commands in the Edit > Contours menu can be used to edit existing contour features.

Merge Contours

Merge selected contours into a single contour feature. Only contours with the same attribute can be merged into a single contour feature.

1. Select a layer with contours.
2. Select two or more adjacent contour features.
3. Select the Merge Contours command.

Menu	Edit > Contours > Merge Contours

The selected contours are merged.

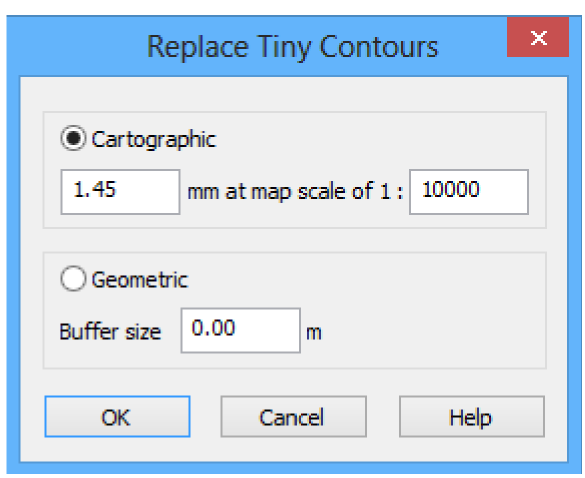
Replace Tiny Contours

Tiny contours, or “isolations”, tend to slow down data processing without adding value to the final product. You can use this tool to modify tiny contours and make them more useful to your products.

1. Select a layer with contours.
2. Select and superselect a tiny contour feature.
3. Select the Replace Tiny Contours command.

Menu	Edit > Contours > Replace Tiny Contours

The Replace Tiny Contours dialog box is displayed.



Two replacement methods are available:

- *Cartographic*: replace the original contour with one that encompasses a sounding object at the specified size and map scale.

- *Geometric*: using the specified buffer size, replace the original contour with one that is larger.

4. Select the replacement method to use.

Cartographic Replacement Method

5. Type a *mm* display size.
6. Type a *map scale* value.
7. Click **OK**.

Geometric Replacement Method

6. Type a *Buffer size* value.
7. Click **OK**.

The selected contours are replaced.

Contour Smoothing

Smooth out and filter contour lines with tightly curved or angular line segments.

Smoothing is performed using the `bSpline` algorithm. When using this option, the distance between vertices is shortened by adding additional vertices. By having shorter edges, smoother curves are achieved.

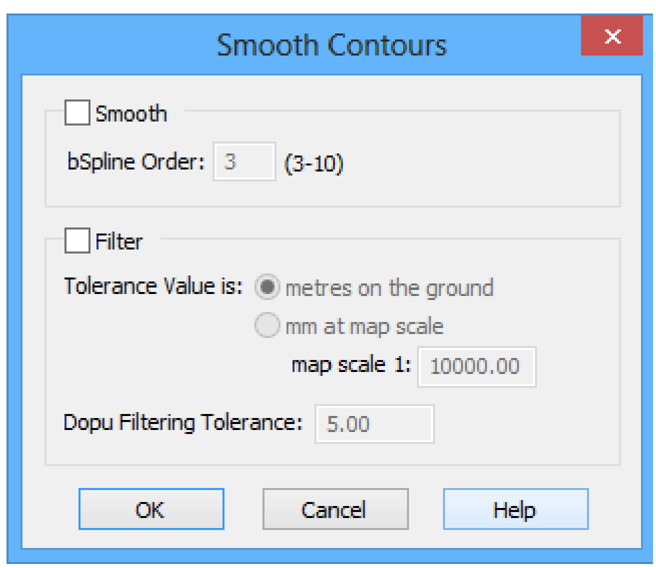
Filtering is based on the Douglas-Peucker algorithm. When using this option, vertices are removed from the lines if they are not necessary.

Select a layer with contours.

Menu	Edit > Contours > Contour Smoothing

8. Select a contour line.
9. Select the Contour Smoothing command.

The Contour Smoothing dialog box is displayed.



Smooth

10. [Optional] Click the check box for the *Smooth* option if you want to smooth the lines.

11. If you chose to smooth lines, enter the degree of smoothing you wish to apply in the `bSpline` *Order* field, with 10 being the largest possible value.

Filter

12. [Optional] Click the check box for the *Filter* option if you want to remove unnecessary vertices.

If you chose to filter lines, you must specify a tolerance to use for the filtering. You can choose to filter by *millimetres (mm) at map scale*, which requires you to enter a scale, or by *metres on the ground*.

13. Select the tolerance type to use.
14. [Optional] If you chose map scale, enter a value in the *map scale* field.
15. Enter the number of units by which to perform filtering in the *Dopu Filtering Tolerance* field.
16. Click **OK**.

The selected line is smoothed/filtered according to the settings defined.

Profiles

Use the Profile command to draw a cross-sectional view of a height source, such as a surface attribute layer.

Profile graphs are created from a line digitized on a surface, either manually drawn or along a superselected track line.

A profile is graphed in a dock-able window, and can be viewed in real-time as the profile is digitized or edited. Profile lines can be edited, and exported as an image or to an ASCII file.

Profiles are saved as layers.

- "Digitize a Profile" on page 464
- "Create Profile by Superselection" on page 467
- "Profile Options" on page 468
- "Export Profiles" on page 471

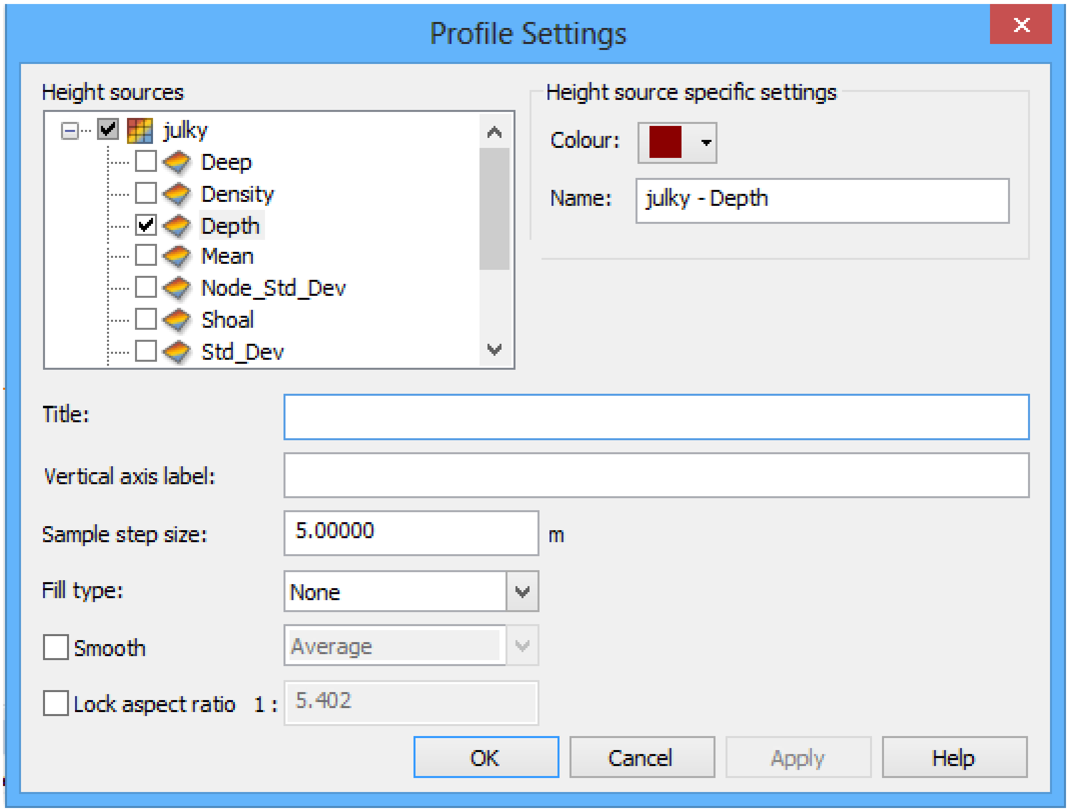
Digitize a Profile

Menu	Tools >Profile > By Digitizing
Pop-up	New >Profile

To create a profile by digitizing:

1. Select surface to be profiled.
2. Select the Profile by Digitizing command.

The Profile Settings dialog box is displayed.



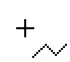
In this dialog box you can set all the options for the profile, or, you can select one or more height sources, generate the profile, then re-open this dialog box to adjust settings while viewing the effect in the Profile window.

3. Select the height source check box for the layer(s) you want to profile.

You can select more than one layer as height sources for the profile. Each source that you select will generate its own profile line in the graph. Lines with the same values will overlap.

4. Set a *Sample step size*. Default value is 5m.
5. Set other options such as a different colour setting for each different source. (See "Profile Options" on page 468.)
6. Click **OK**.

The Profile window opens and displays the message "Empty profile line".

The cursor changes to digitizing mode.  The default line type is a Point to Point line. Other types of lines can be used. See "Digitizing Lines" on page 572 of the Reference Guide.

Digitize the profile line

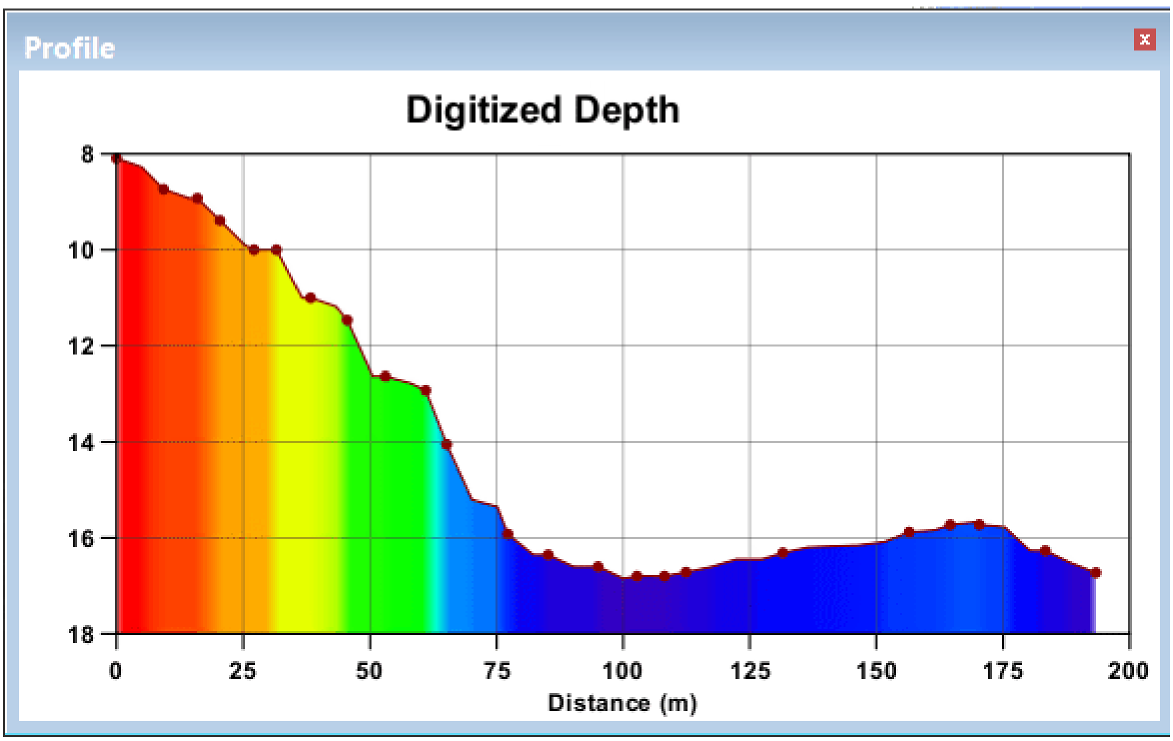
7. Click once on the surface to fix the anchor point for your profile line.
8. Click to add a series of points to define the line.

The profile line is automatically drawn between the points. The Profile window dynamically displays the profile as you add more points to the digitized line.

9. To remove points before the line is completed, right-click and select Remove Last from the pop-up menu. Repeat as needed.
10. Click the **Enter** key to complete the line, or right-click on the line and select End line from the pop-up menu.

The Profile is displayed in the Profile window, and Profile layers are added to the Layers window.

The example below shows the profile on one height source, generated by the digitized line. The dots in the graph represent the digitized points of the line. The vertical fill option was turned on.



Display settings can be adjusted in the Profile Settings dialog box. "Profile Options" on page 468.

To change the graph display options:

11. Right-click in the profile window and select Settings from the pop-up menu.

The profile can be saved as an image or as ASCII file.

12. Right-click in the profile window to select export options. See "Export Profiles" on page 471.

Create Profile by Superselection

You can create a profile that follows a superselected line, such as a track line.

To create a profile for superselected track line:

1. Open the surface to be profiled.
2. Select the Ship Track Lines layer in the Layers window.
3. Select a track line.

Menu	Tools >Profile > Superselection

4. Select the Profile by Superselection command.
5. In the Profile Settings dialog box, select the height source check box for the layer(s) you want to profile.

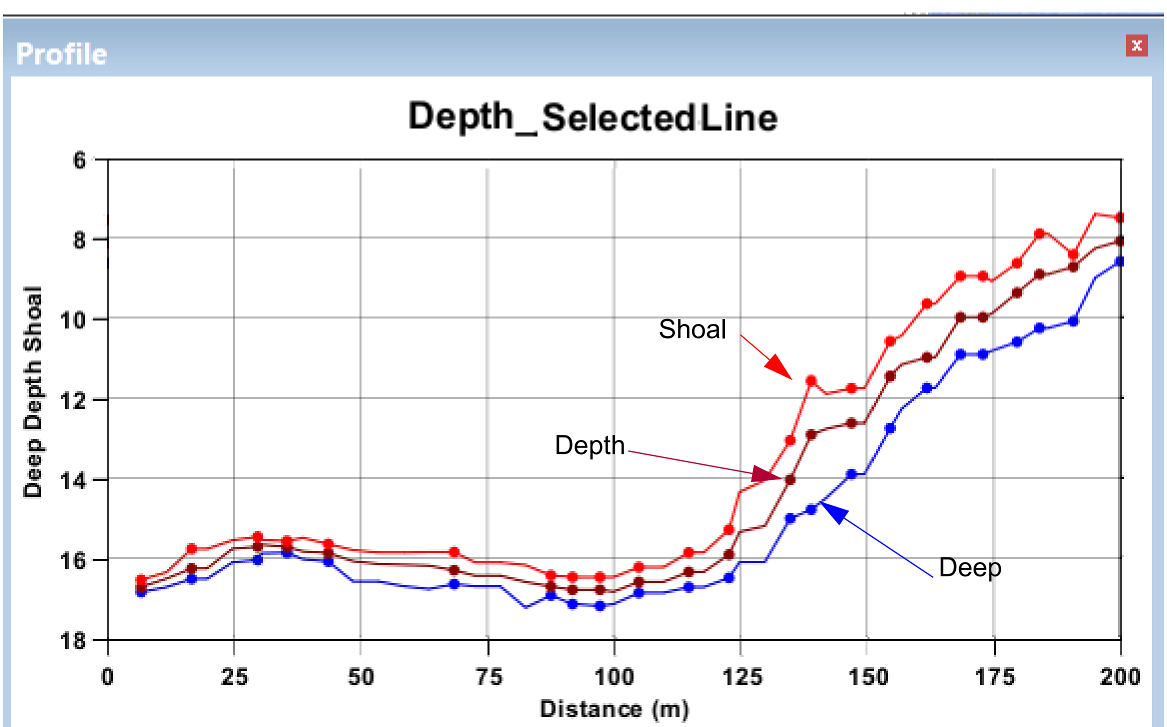
You can select more than one height source for the profile. Each layer that you select will have its own profile line in the graph. Profile lines with the same values will overlap.

You can also generate the profile and then re-open this dialog box to dynamically adjust settings.

6. Set a *Sample step size*. Default value is 5m.
7. Set other options as desired. See "Profile Options" on page 468.
8. Click **OK**.

The profile is displayed in the Profile window.

The example below shows the profiles on different height sources, generated by the same superselected line.



When more than one height source is selected the vertical and horizontal fill options are disabled.

Display settings can be adjusted in the Profile Settings dialog box. To change the graph display options:

9. Right-click in the profile window and select Settings from the pop-up menu. See "Profile Options" on page 468

The profile can be saved as an image or as ASCII file.

10. Right-click in the profile window to select export options. See "Export Profiles" on page 471.

Profile Options

Profile colour

Use the *Colour* field to assign a different colour to each height source /line graphed in the Profile window.

1. Highlight a layer in the *Height sources* list.

The *Name* field is automatically populated with the name of the surface and the layer you selected (e.g., `SurfaceName - LayerName`). You can edit this name.

2. [Optional] Replace the automatically generated name.
3. Select a colour from the colour picker. (Default colour is black.)
4. Repeat to assign a different colour for each source.

Graph labels

5. Enter a *Title* for the profile graph.

The vertical axis in the graph shows the attribute values from the surfaces. The horizontal axis is always distance along the profile.

6. Enter a *Vertical axis label* for the profile graph.

Sample step size

The *Sample step size* field controls the frequency with which the surface is sampled. HIPS samples the data at the specified interval and uses it to draw the profile line between the digitized points. The smaller the step size, the more detail in the profile.

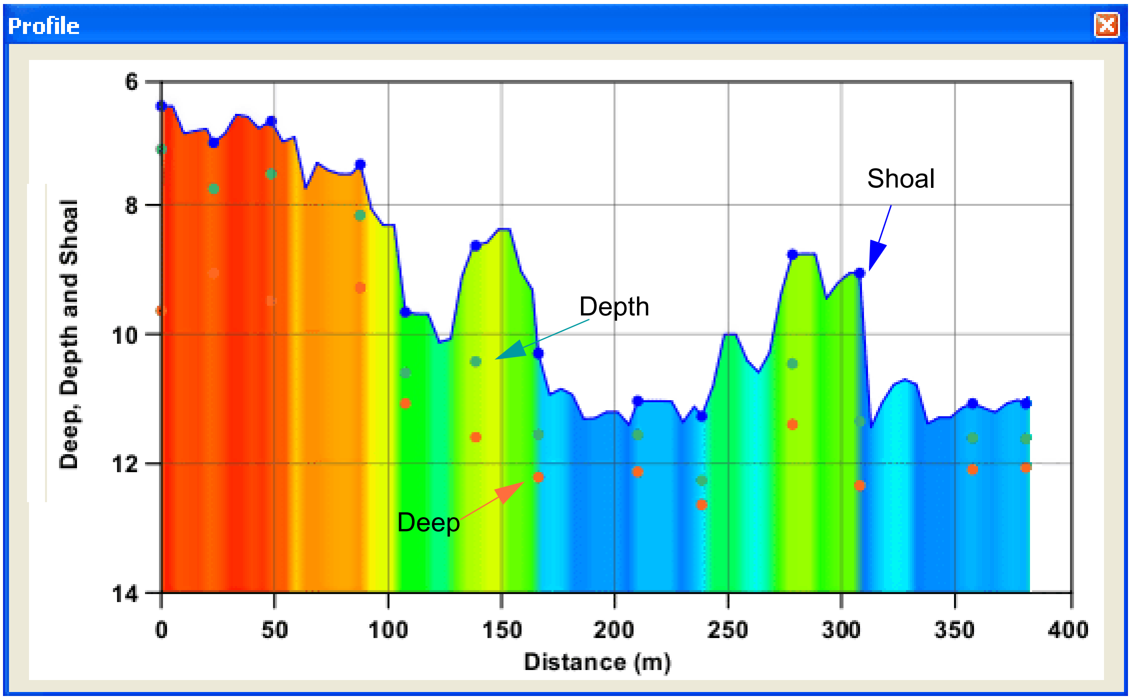
7. Enter a value for *Sample step size*.

Fill

The *Fill type* option enables you to colour the profile according to the elevations in the data. You have the option of applying the colours horizontally or vertically.

When more than one height source is selected the vertical and horizontal fill options are disabled.

This is an example of a profile with vertical fill.



8. Select an option from the **Fill type** drop-down list.

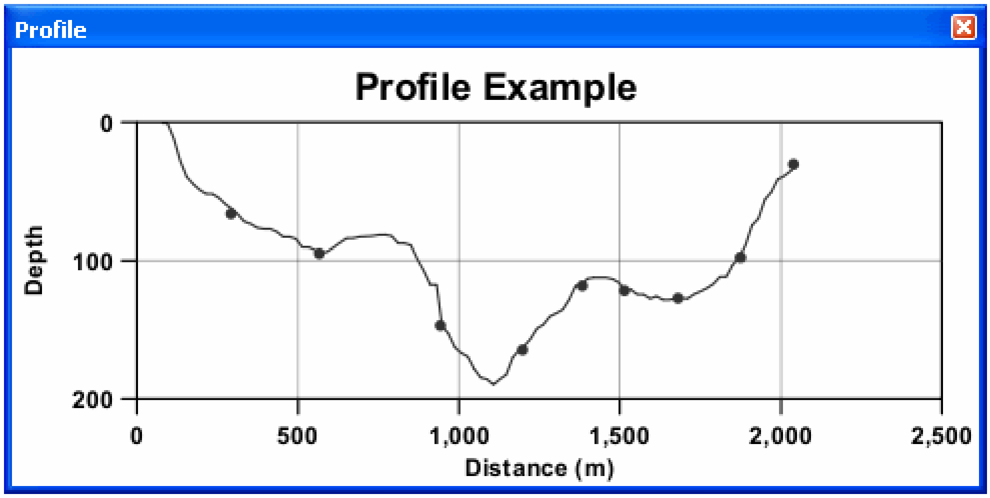
Smooth

The *Smooth* option enables you to smooth the profile line in the profile graph. Smoothing is applied reduce the number of peaks in the graph if it has a high number of sample points. The number of sample points is based on the *Sample step size* setting. There are three types of smoothing to choose from:

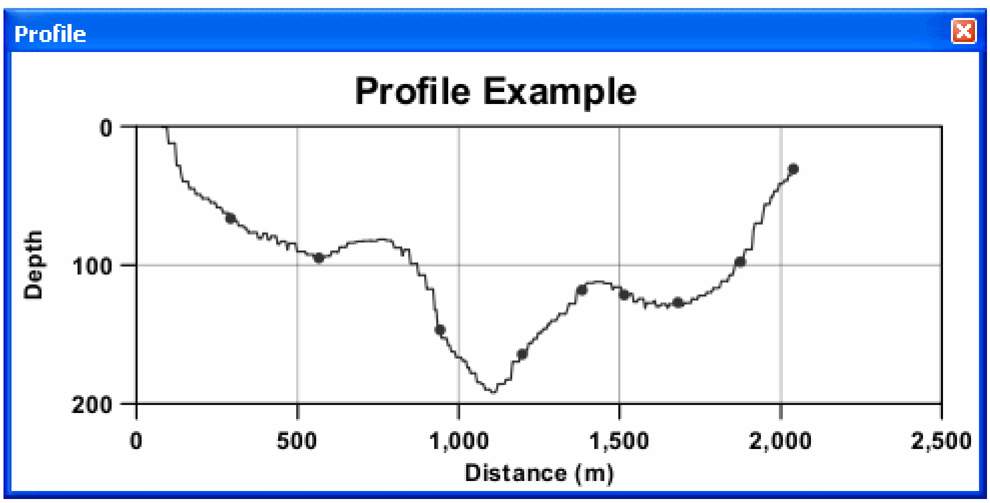
- *Average*: This method will create the graph using the average elevation values within each sample distance.
- *Shoal*: This method will create the graph using the shoalest values within each sample distance.
- *High/Low*: This method will create two lines in the graph; one for the minimum values in each sample distance and one for the maximum values in each sample distance.

Below is an example of a profile with and without shoal smoothing applied.

With Shoal Smoothing



Without Smoothing



9. Click the check box to enable the **Smooth** option.
10. Select a smoothing type from the drop-down list.

Lock aspect ratio

Use *Lock aspect ratio* option to maintain the ratio of the width of the profile graph to its height, if the Profile window is re-sized.

11. Click the check box to enable the **Lock aspect ratio** option.

The current ratio of the graph will be displayed in the aspect ratio field.

12. [Optional] Enter a new ratio value for the graph.

> The settings for Vertical axis label, Fill type, Smooth and Lock aspect ratio will be remembered the next time the dialog box is opened.

Resize Profile window

The size of the Profile window can be changed to provide a better view of the profile. As the window is re-sized, the view is automatically scaled and the axes values updated dynamically.

- The vertical axis represents the height source of the profile and can be assigned a name in the Profile Settings dialog box.
- The horizontal axis represents the length of the line feature being used to generate the profile. The unit of measure for the values on this axis are controlled by the *Horizontal Length* setting in Tools > Options > Display > Units.

Export Profiles

The profile can be saved as an ASCII file or as an image.

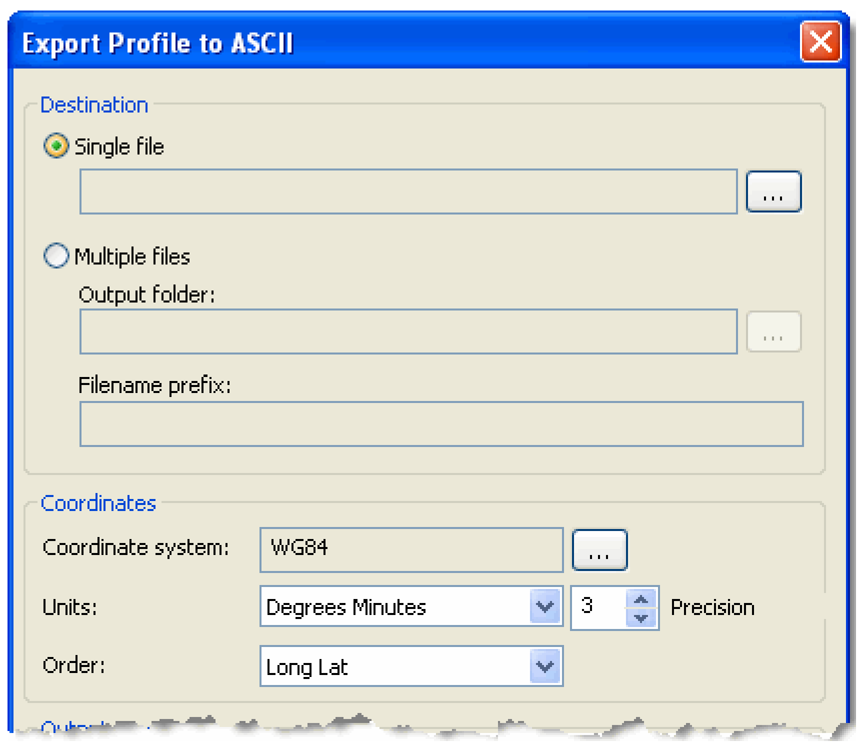
Export Profile to ASCII

Use the ASCII Export option in the Profile window pop-up menu to export profile lines to ASCII files.

To export to ASCII:

1. Right-click the Profile graph in the Profile window.
2. Select the **ASCII Export** command.

The Export Profile to ASCII dialog box is displayed.



Destination

If you have created a profile line on multiple layers, you have the option of exporting the profile as one file or to one file for each layer.

To export to a single file:

3. Select *Single File* and click **Browse** to select the destination folder.
4. Type a name for the file.

When you export to multiple files, the ASCII files will be named with a combination of the surface name and layer name. You can add a prefix to the file name. The format of the file name with this option enabled is:

```
Prefix-SurfaceName - LayerName.txt
```

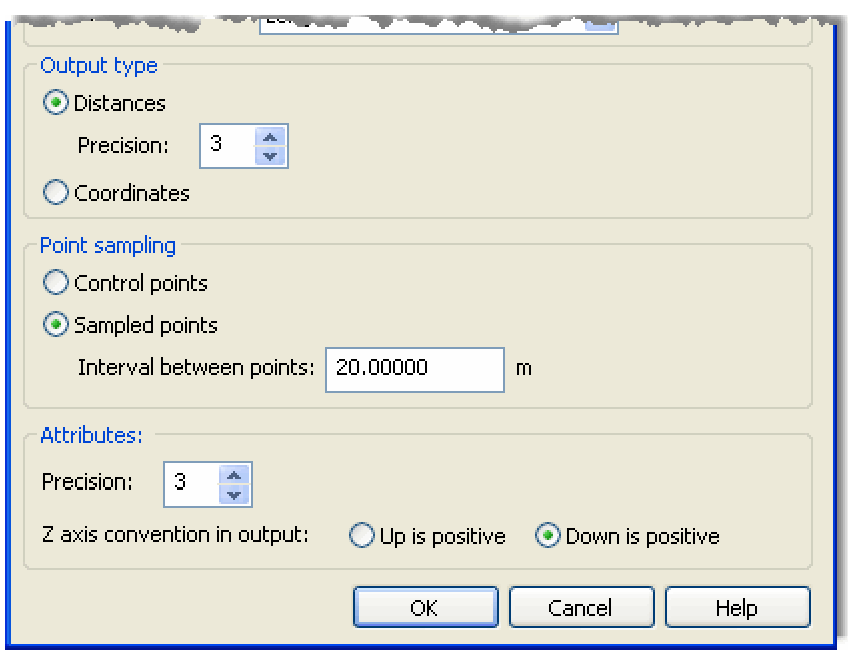
5. Select the *Multiple Files* output option.
6. Click **Browse** (...) to select the output folder.
7. [Optional] Enter a *Filename prefix* to add to each file name.

Coordinates

By default, the Geographic (Lat/Lon) - WGS84 coordinate system is set as the coordinate system for the output.

If you change the coordinate system here, this new system will be used the next time the Export to ASCII command is used.

8. [Optional] Click **Browse (...)** to select another output coordinate system.
9. Select an option from the *Units* drop-down list.
10. Select a *Precision* value to define the number of decimal places to apply to the exported coordinate values.
11. Use the *Order* drop-down list to specify the order of the coordinate columns in the output file, e.g., Long/Lat or Lat/Long.



Output type

The ASCII file will contain the depth values at specific points along the profile and the locations of those points. Depending on the Output type selected, the locations may be reported as distances from the start of the profile (in metres, feet, etc.), or as geographic coordinates.

12. Select an *Output type* option.

If exporting locations as *Distances*, you can also define the number of decimal places to apply to the distance values.

13. Select a Precision value for the *Distances* field.

You have the option of exporting depths at all control points in the profile, or at a sampling of points.

- The *Control points* option exports depth values at the points that were digitized to create the profile line.
- The *Sampled points* option selects points at a specified interval and exports the values at those locations.

14. Select a *Point sampling* option.
15. [Optional] If exporting sampled points, enter a value for the *Interval* between points.

> The Interval setting defaults to the Sample step size setting of the profile graph, but you can export sampled points at a different interval if desired.

The profile was created using attribute values from the selected source layer, e.g., Depth. You can define the precision for these values as well as the Z-axis convention to use in the output.

16. Select a Precision value to define the number of decimal places to include in attribute values.
17. Select the Z-axis convention to use in the exported values.
18. Click **OK** to perform the export.

The data in the Profile window is exported to an ASCII file. Header information includes:
- The name of the profile line.
- The coordinate projection of the data.
- The unit of measure for the attribute values.
- The Z-axis convention of the data.
- The unit of measure for distance values.
- The start and end coordinates of the profile line.
- The headings for the column order.

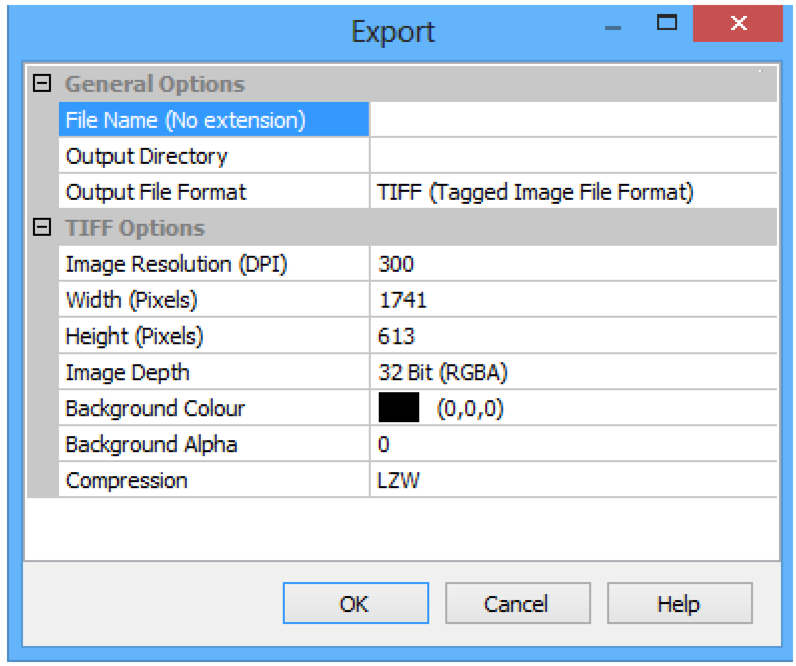
Export Profile to Image

Use the Image Export command in the pop-up menu of the Profile window to export an image of the current profile graph. The profile image can be exported to:
- PDF (Portable Document Format)
- PS (PostScript)
- SVG (Scalable Vector Graphics)
- TIFF (Tagged Image File Format)

To export an image of the profile graph:

1. With the profile displayed, right-click the Profile window and select choose **Image Export** from the pop-up menu.

The Export dialog box is displayed.



Use the Export dialog box to define the settings for exporting the image. The example above displays the options for export to TIFF format.

2. Type a *File Name* for the image file.
3. Click within the *Output Directory* field to enable the **Browse** (...) button.

4. Click **Browse** and select a location for the exported file.
5. Select the *Output File Format* from the drop-down list.

Once you select a format, specific options for that format are displayed in the lower section of the dialog box.

6. [Optional] Enter or select a value for the options as needed.

Export options

PDF and PostScript options	
Paper size	Set the page size of the exported file.
Width/Height	Set the width and height of the histogram in the resulting file.
Data Orientation	Set the orientation of the histogram in the resulting file. Portrait is a vertical position; Landscape is a horizontal position. Landscape is selected by default.
Margins	Set the width of the area between the histogram and the edge of the file. The default is 5mm.

Image Export DPI (Quality)	Set the resolution used to export the histogram. The higher the resolution, the better the quality of the image when zoomed in. If the file will be used in the electronic form with the image being zoomed in tightly, a higher resolution is recommended. If the file will simply be printed, a smaller resolution is adequate.
SVG format options	
Width/Height	The width and height of the histogram in the resulting file.
Image Export DPI (Quality)	Set the resolution used to export the histogram. The higher the resolution, the better the quality of the image when zoomed in. If the file will be used in the electronic form with the image being zoomed in tightly, a higher resolution is recommended. If the file will simply be printed, a smaller resolution is adequate
TIFF format options	
Image Resolution	The resolution (quality) at which to export the image. The higher the resolution, the closer the image can be zoomed, but the larger the file.
Width/Height (Pixels)	Set the width and height of the resulting image in pixels. These fields are controlled by the resolution of the image. As the DPI is increased or decreased, so is the number of pixels.
Image Depth	Set the number of values applied to each pixel in the image (one value for each colour, plus one for transparency if using 32 Bit). If you would like a transparency setting applied to the background colour, the 32 Bit (RGBA) option must be selected.
Background Colour	The colour displayed in the background of the histogram. The default is black.
Background Alpha	(This option is only available when the Image Depth is set to 32 Bit RGBA). Set the degree of transparency applied to the background colour. The default setting is zero transparency.
Compression	Compress the resulting image file during export to decrease the file size. This option is only available when using the TIFF format. There are various compression methods available, each of which performs differently. By default, LZW is used.

7. Click **OK**.

The image file is saved to the specified location.

Sounding Selection

Survey data contains so many soundings that they cannot all be displayed legibly on a chart or map. Sounding selection enables users to make a meaningful selection of soundings to be displayed.

The selection of soundings are created on a feature layer. The selection is made using either radius or overplot removal criteria, and are shoal- or deep-biased. A template of selection criteria can be created, loaded and re-used for sounding selection.

Soundings are selected from a gridded surface or from a point cloud. Use the Sounding Selection wizard to set criteria and options for selection.

To create a layer of selected soundings:

1. Create or open a feature layer. "Create a Feature Layer" on page 565
2. Select a surface layer.
3. Select the Sounding Selection command.

Menu	Tools > Soundings Selection
Pop-up	New > Selected Soundings…

The Sounding Selection Wizard is displayed.

Selected Soundings uses the values from an attribute layer of the selected dataset as the height source for the sounding selection.

By default, the Attribute name field displays the layer selected when the Selected Soundings tool was last used.

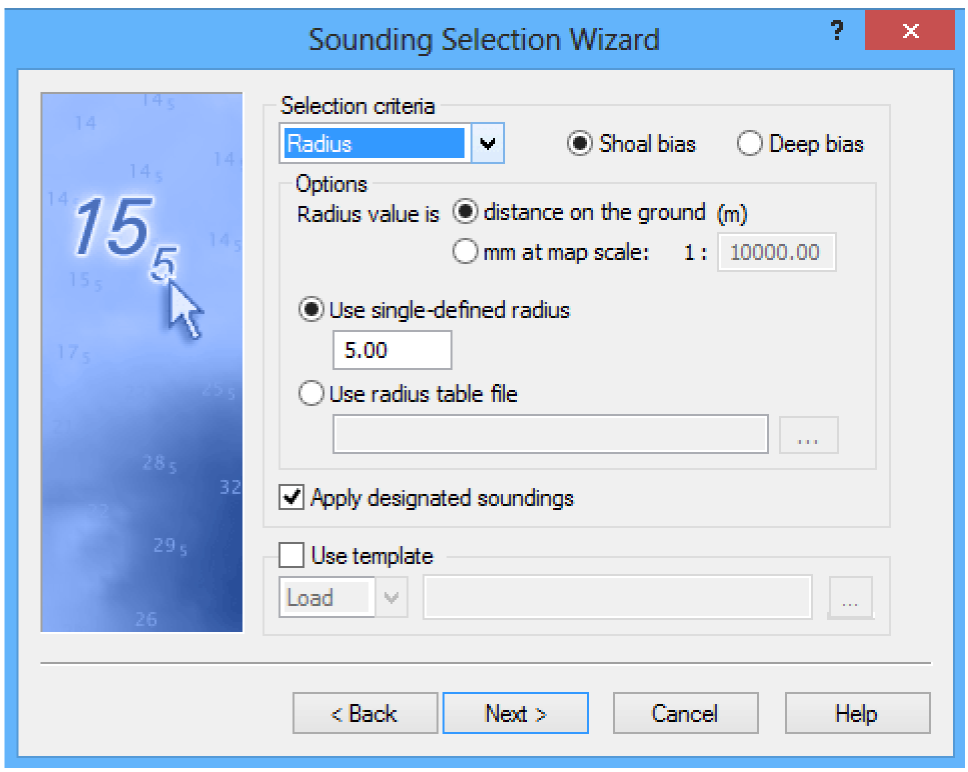
4. [Optional] Select another surface attribute layer from the drop-down list.

The *Output layer* is the destination layer for the selected soundings.

5. From the *Output layer* list select the feature layer that you opened or created.
6. Click **Next**.

Selection Criteria

The settings in this dialog box determine the density of the selected soundings in the layer.



Selection is based on two main options:

- a sounding criteria method (e.g., radius or overplot removal)
- a conflict resolution method (shoal bias or deep bias)

Selection criteria

You can apply these methods for sounding selection:

- *Radius:* selects only the (shoal or deep) soundings within a set distance

- *Overplot Removal:* selects only certain soundings in a dense data set so that they can be plotted without overlapping.

Conflict resolution

You can set whether the shoalest soundings or the deepest soundings will be included when conflicts are encountered during selection. For example, if you select *Shoal bias,* and two soundings overlap, the shoalest of the two will be selected.

1. Set either *Shoal Bias* or *Deep Bias* as the method of resolving any conflicts during the selection process.
2. Select one of the criteria from the *Selection Criteria* drop-down list.

The dialog box will display the options associated with those criteria.

This conflict resolution is also applied to designated soundings.

See "Radius" on page 478

See "Overplot Removal" on page 480

Apply designated soundings

This is set by default to ensure that all designated soundings are included in the sounding output. To omit designated soundings from the output:

3. De-select the apply designated soundings check box.

Selection Template

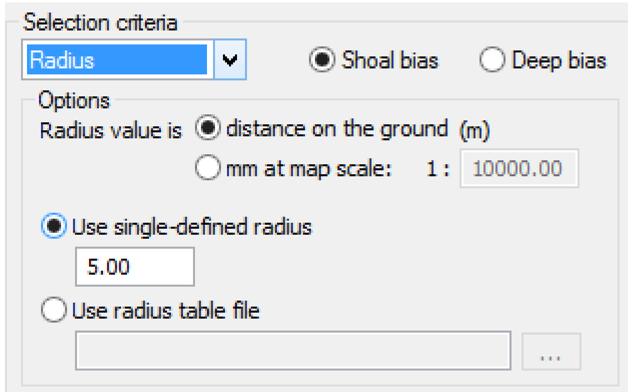
Settings in the wizard can be saved to a Sounding Selection Template (CSST) file. This file can be opened and reused.

4. Select the *Use template* option to open or create a template file that contains sounding selection options.
5. Select *Load* to open an existing template file, or

   select *Save* to create a new template file from the options you are setting in the wizard.
6. Click the Browse button to open a file, or create a new file.
7. Click **Next**.

Radius

If you choose *Radius* as your selection criteria, the following option fields are displayed:



The radius value you set (either as a specific value or from a radius table file) will either be applied as metres on the ground, or as the number of millimetres between soundings at map scale.

Once you have set how the radius value will be applied, you can either set the same interval value to be applied to each radius, or have varying values applied from a radius table file.

For example, if you selected Shoal bias, and set a single-defined radius of 5, then only the shoalest sounding within each 5m radius on the ground, or within 5mm at map scale, will be displayed.

Sounding Radius Table

A radius table sets the minimum radius between soundings for various depth ranges. Each depth range will have its own radius value.

The sounding radius table is a text file containing three columns:

- minimum depth of the range
- maximum depth of the range
- radius value for the depth range.

For example, using the example radius table below, 8m soundings would be a minimum of 1mm apart at map scale, while 80m soundings be 10mm apart.

Min depth	Max depth	Radius value for range
0.0	10.0	1.0
10.0	20.0	2.0
20.0	50.0	5.0
50.0	500.0	10.0

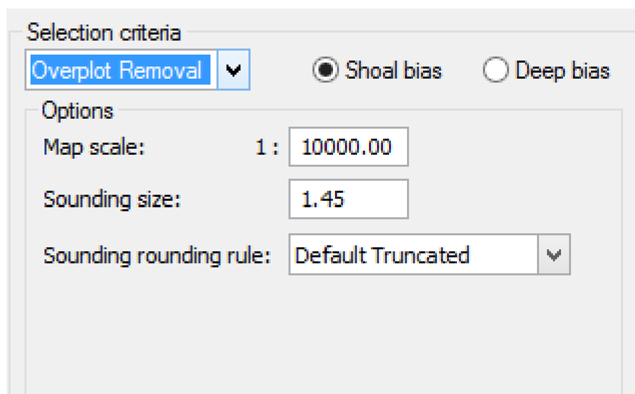
1. Set the *Radius Value*.
    - Select distance on the ground (m), or

- Select *mm at map scale*, and type a scale value in the *Map Scale 1:* field (default is 1:10000).

2. To set a standard minimum distance between soundings, select the *Use single defined radius* and enter the interval.
3. To use a radius table to set the minimum distance between soundings, select the *Use radius table file* option and click **Browse** to select the file.
4. Click **Next**.

Overplot Removal

If you choose the *Overplot Removal* option from the *Selection Criteria* list, the dialog box is refreshed to show the following options.



Overplotting occurs when soundings overlap or are so close together that they are plotted on top of each other. Two soundings with depths of 11 and 22 might look like 1212 if they were overplotted.

Overplot Removal calculates how big the displayed sounding will be, and places a buffer zone around each sounding, based on sounding size, then suppresses either the deepest or shoalest soundings which overlap these buffer zones.

5. Type the map scale for the selected soundings (default is 1:10000).
6. Type the size of the soundings (that you want them to appear in the Display window) in the *Sounding Size* field.
7. Select a *Sounding Rounding Rule* from the drop-down list.
8. Click **Next**.

Enable Filter

The *Enable Filter* option enables you to extract and display only the soundings that meet the conditions you set for the filter.

1. Select the *Enable Filter* check box.
2. Use a combination of attributes, operators and values to construct your filter.
3. Use the Backspace key to delete a character or **Clear** to remove the entire filter string

For example, to display only the soundings with a Depth less than 10 metres, and a hypothesis count of 2:

- Double-click on Depth in the list of available *Attributes* to place it in the filter field.
- Click the "**<**" button, and type 10.
- Click the **and** button and double-click on the `Hypothesis_Count` attribute.
- Click the "**=**" button, and type 2.

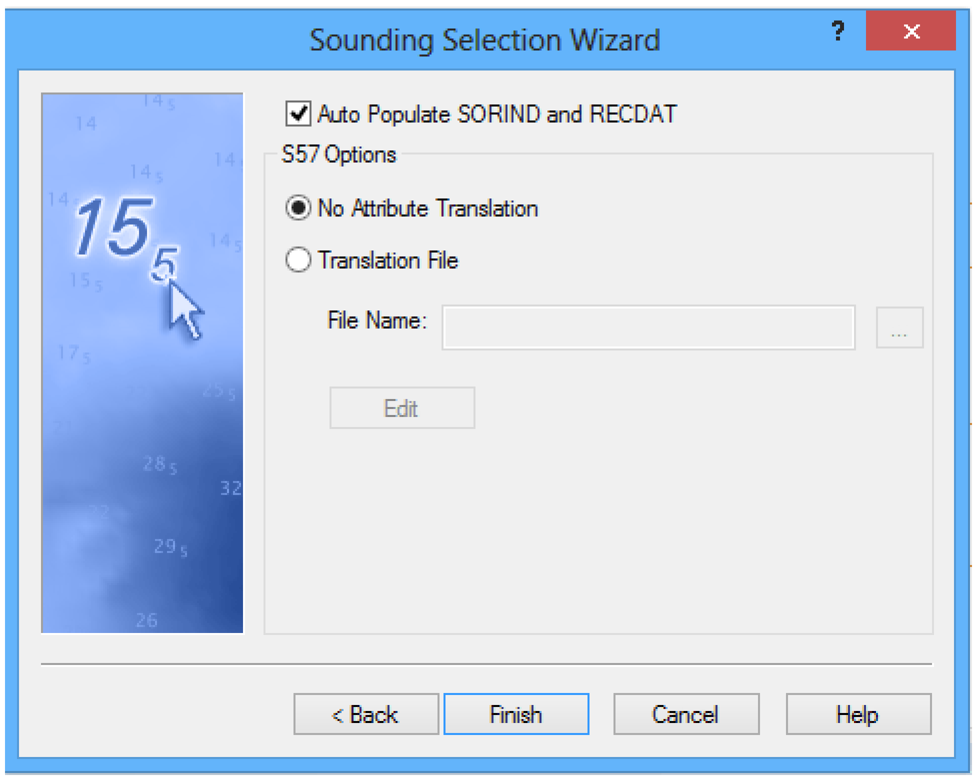
HIPS and SIPS automatically adds the necessary spaces between the parameters in the filter string. However, if you choose to type the filter parameters into the filter field, you must manually add those spaces.

If there are spaces in the name of the selected attribute, place quotation marks around the attribute in the conditions field, (e.g., “attribute name”).

4. Click **Next**.

S-57 Options

By default the options here are set to automatically enter information into *SORIND* and *RECDAT* attribute fields, and to map surface attributes to specific S-57 attributes.



Auto Populate SORIND and RECDAT

These attributes can be automatically populated:

- SORIND (information about the source of the sounding) and
- RECDAT (date the object was captured, edited or deleted).

1. [Optional] Disable the *Auto Populate SORIND and RECDAT* check box so that information on these attributes is not added to the soundings in the HOB file.

Auto populating the SORIND and RECDAT attributes can increase save and export times as well as file size, particularly with large, high-density data sets.

S-57 options

The other S-57 option is to map the surface attributes to S-57 attributes, using a translation file. The translation file has a BSST file extension and is in XML format. See "TRANSLATION FILE" ON PAGE 483.

The *No Attribute Translation* option is set by default.

To continue without mapping any attributes:

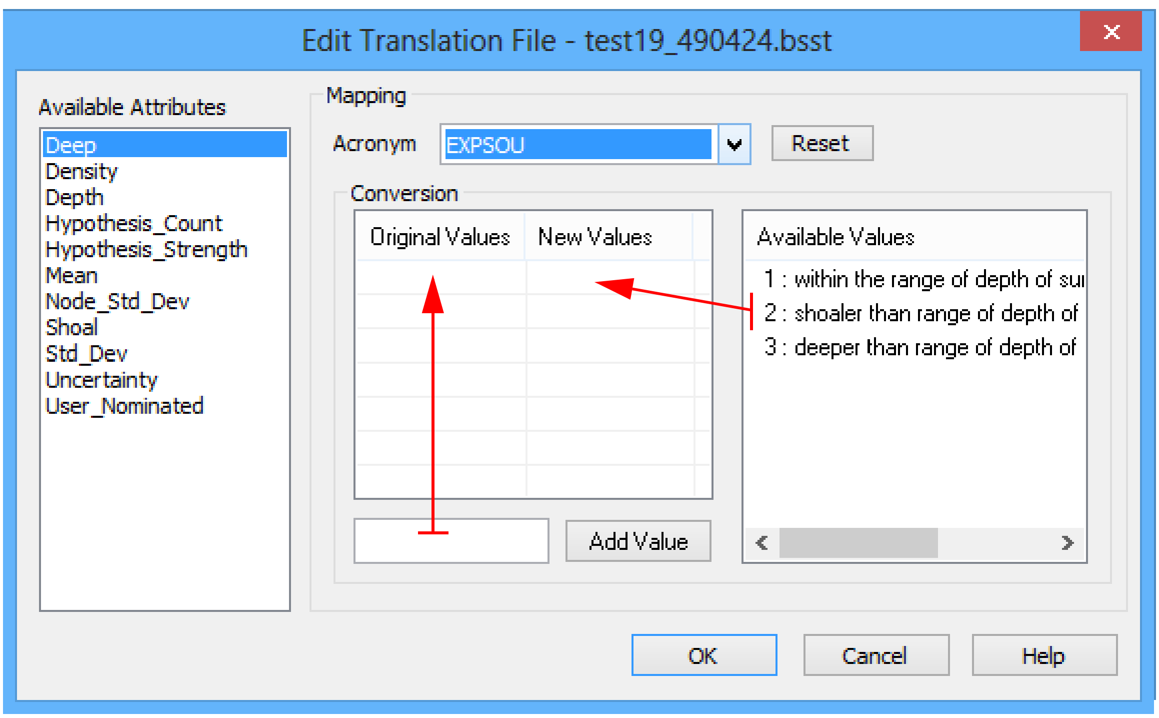
2. Click **Finish**.

Translation file

To use a translation file to map attributes:

1. Select the *Translation File* option.
2. Click **Browse** to select an existing BSST translation file (or to specify a name for a new translation file).
3. Click **Edit** to set the mapping for surface attributes to S-57 attributes.

The Edit Translation File dialog box is displayed.

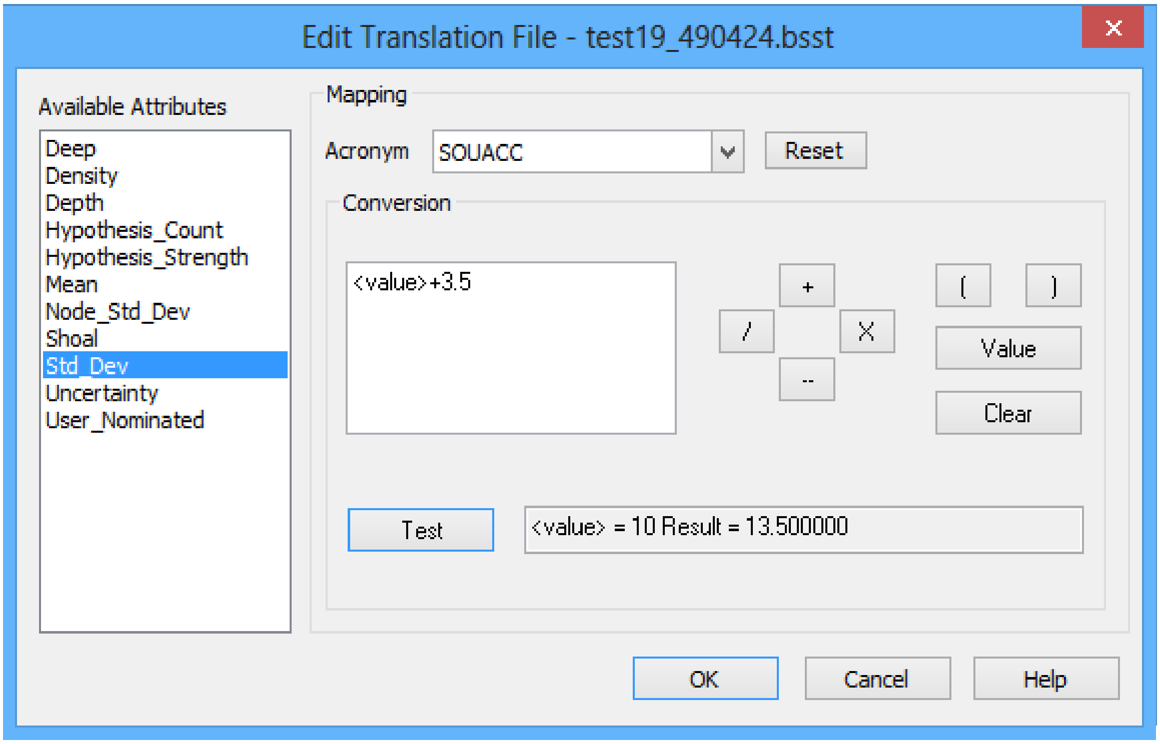


4. Select a surface attribute from the *Available Attributes* list.
5. Select an appropriate S-57 acronym from the drop-down list.

If no mapping is possible between the selected attribute and the chosen S-57 attribute acronym, "No conversion available" is displayed.

If conversion can be applied, the dialog box will display the available values that can be mapped.

HIPS attributes of Integer data type may be appropriate to map to S-57 attributes, using a conversion formula. A conversion formula is created by inserting values and appropriate mathematical operators into the *Conversion* area, as in the example below.



To create a conversion formula, for example, to map Std_Dev to SOUACC, (sounding accuracy attribute):

1. Click **Value** to use the value of the selected surface attribute in the conversion formula.
   - This places the <value> variable in the formula window.
2. Insert mathematical operators using the **+**, **-**, **X**, **/** buttons.
3. Type the numeric value by which to adjust the original value, if needed.
   - In this example, adjust the original value by adding 3.5.
4. Click **Test** to see the formula.
5. If you want to erase the formula, click **Clear**.
6. Click **Reset** to remove the mapping of surface attribute to S-57 attribute.
7. Click **OK** to save the mapping settings and return to the S-57 Options dialog box.
8. Click **Finish.**

The soundings that meet the criteria set in the wizard are selected, and corresponding sounding features are created on the selected feature layer.

You can change the display of the soundings in the Properties for the feature layer. See "Properties Window" on page 518. The sounding layer can be saved to a HOB file, using the File > Save.

Depth Range Files

Depth range files are text files that set a range of depths for tiling and contouring data. These files are automatically generated during the contouring process (see "CONTOURS" ON PAGE 454).

The convention for setting depth values is similar to that of other CARIS files:

- Drying heights above the datum are represented as negative values
- Depths below the datum are represented with positive values.

Below is an example of a range depth file.

```
-50
-30
-20
-10
-5
0
5
10
15
20
30
50
```

In the above file the negative values (-30 to -5) are above the datum, and the positive values (5 to 50) are below the datum.

19

Process Water Column Data

Water column data can be viewed, selected and queried in Subset and Swath Editor. the data can also be imported into Additional Bathymetry.

The display of water column data can be manipulated in the editors, and bathymetry derived from water column imagery can be added to an existing project.

In this chapter...

Water Column Data in HIPS

Water column data can be integrated into existing bathymetry workflows. Water column image (WCI) data can be displayed in both Swath Editor and Subset Editor, and its bathymetry data added to a project. In Swath Editor water column data is displayed as a curtain image. In Subset Editor it is displayed in 3D.

Currently, HIPS and SIPS supports water column data from these sonars:

- Teledyne Reson 7K (.s7k)
- Kongsberg (.all, .wcd)
- Seabeam 30XX series data in XSE format
- R2Sonics WCI data in Hypack format.

"Water Column Data in Swath Editor" on page 490

"WCI Data in Subset Editor" on page 497

Projects

Projects containing water column data are organized in the same Project-Vessel-Day-Line hierarchy as other sonar projects.

When data is converted to HIPS there is the option of copying the original data files to the Processed folders. By default, raw data files are not carried over.

When opening water column data, HIPS will look for WCI data in the line directory. If the data is not found there, HIPS looks in the location of the raw data files as recorded during conversion. If the raw data no longer exists in its original location, you will be prompted to have HIPS search for it, or to search for it yourself.

Processing Workflow for Water Column Data

Open a project to which the water column data will be added.

Open WCI data in Swath Editor (Across and Along track and Stacked views)

- Filter based on intensity
- Overlay Bottom Detection
- Assign Colour Map

- Line editing - Query /Reject/Accept bathymetry
- Select and data to add to project bathymetry
- Re-evaluate intensity filter range

Open data in Subset Editor (2D and 3D views)

- Filter by intensity ranges determined in Swath Editor
- Query/ Reject /Accept bathymetry data
- Designate critical soundings
- Select and add points to additional bathymetry layers.

Working with Additional Bathymetry Layers

Toggle views between WCI, project bathymetry and additional bathymetry layers.

Compute TPU for additional bathymetry.

If required, undo all processing of Additional Bathymetry, with Restart Cleaning command on pop-up menu.

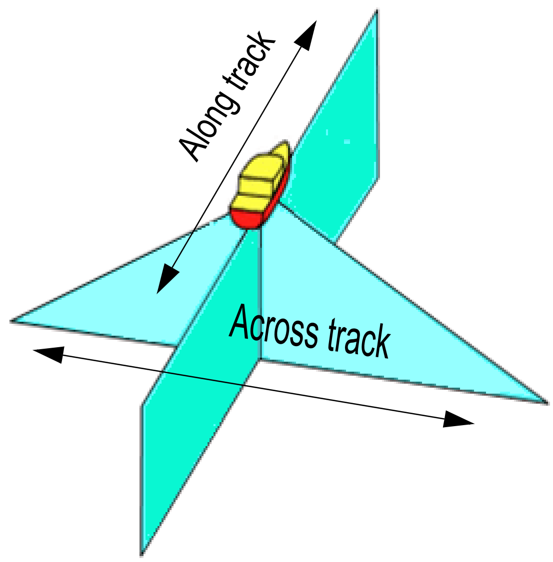
Regenerate additional bathymetry layers if project data has new corrections and new merge applied.

Add additional bathymetry to surfaces.

Water Column Data in Swath Editor

In Swath Editor. water column imagery is displayed in two windows:

- Across track: the water column data as viewed from the stern of the survey vessel. There is an across track image for each selected profile (ping). (Also referred to as a profile view, polar intensity plot or swath profile.)
- Along track: displays the water column data viewed from the starboard side of the vessel. This could also be referred to as a side view or vertical profile. The along track image is made up of the beams which point to nadir for the selected profiles.



Stacked view

WCI can also be displayed in a "stacked" view, to give an idea of what may be in the water column based on multiple profiles and beams. In the across track view all the profiles represented in the Plan view are stacked one on top of another.

In the along track view the beams within a profile are stacked with the highest intensity beams showing through. See "Stacked view" on page 494.

Bottom detections

Bottom detections can be plotted on top of the water column image in the across track view. This can be used to view how the existing bathymetry compares to that which is present in the water column data. This display can be toggled on and off from the toolbar. See "Water Column Toolbar" on page 493.

Basic filtering is also present so the WCI data can be filtered based on intensity values.

All existing Swath Editor options and functionality, where relevant, have been connected within the WCI views, to keep work flows consistent and familiar.

Display water column data

Menu	Tools > Swath Editor > Open
Tool	
Pop-up	Tools > Swath Editor

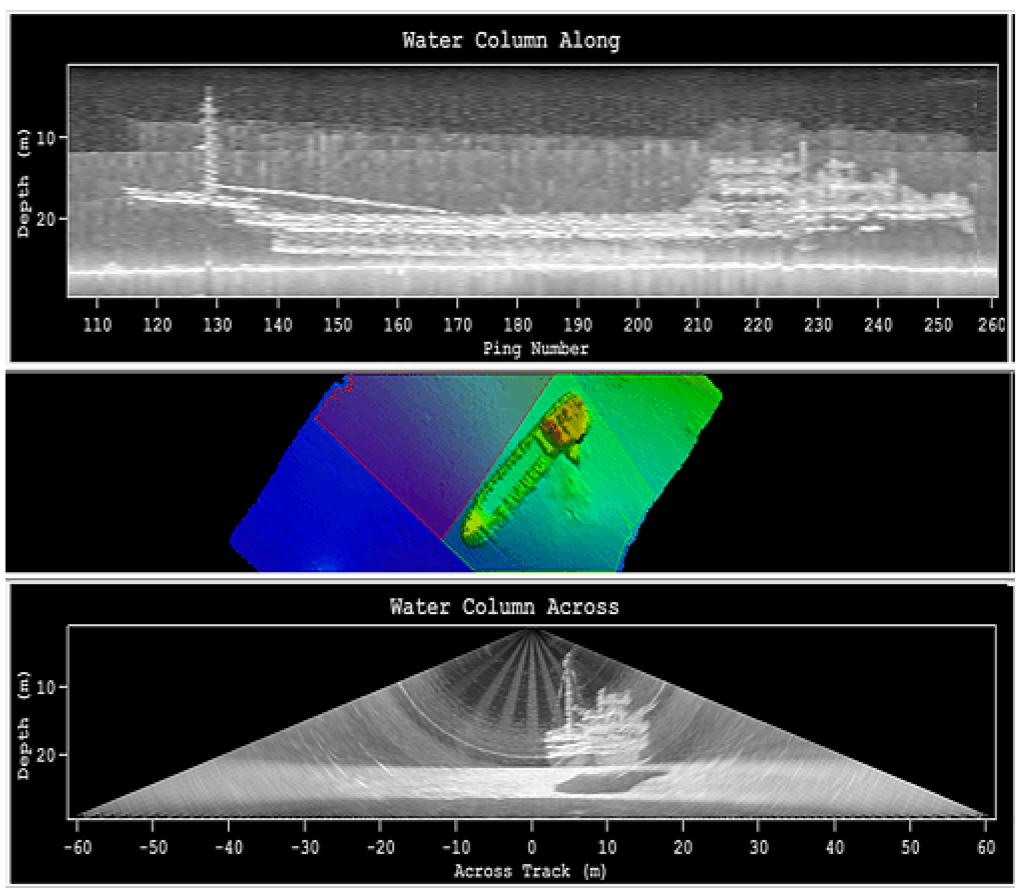
To view WCI data in Swath Editor:

1. Select a survey line in the Display Window.
2. Select the Open Swath Editor command.

If the selected line contains WCI data:

- the data will be loaded into the Along and Across track windows,
- the Water Column Editor toolbar will be displayed,
- and the WCI will be displayed in the Swath Editor Control page.

The example below shows WCI data displayed in the across and along track windows, and bathymetry in the Display window (shown as a surface). [1]



3. Select a profile, for example, in the Plan View.

The related WCI data is displayed in the water column windows

1. Image courtesy of John Hughes Clarke, Ocean Mapping Group, University of New Brunswick.

Scroll through the Plan view using the arrow keys, or scroll buttons, to see the water column image in the across and along track views.

To highlight a WCI profile in the Swath Editor views:

4. Click the middle mouse button in the Along track window.

As each profile is highlighted, the display in the Swath Editor windows is automatically refreshed.

As well, there is a slider bar in the Across track window, similar to that of the Profile window, which you can use to scroll through the data.

Automatic scrolling

As well as viewing WCI data profile by profile, you can view the data in sequence using the playback controls on the Water Column toolbar. (These work in a similar fashion to the playback controls in the Side Scan Editor.)

To scroll automatically through the data:

1. Click on the Plan View window.
2. Click **First Profile** on the Water Column Data toolbar to select the first profile visible in the Plan View.
3. Click **Start playback** to start scrolling the data.

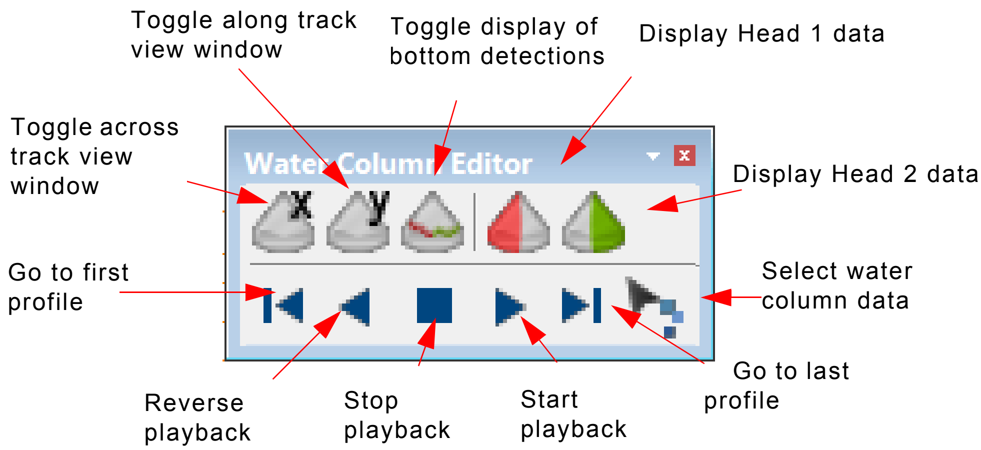
HIPS will move up the track line, profile by profile, changing the view of the water column data, until it reaches the last profile loaded in the Plan View.

4. Click **Reverse playback** to scroll back down the track line.
5. Click **Stop playback** to pause the scrolling.
6. Click **Last Profile** to reset the view to the last profile loaded in the [plan View.

Playback is not available when data is stacked.

Water Column Toolbar

The Water Column toolbar contains the controls to open and close the Across track and Along track windows, to display the bottom detection and to control the play back of the imagery.



The **Display across track** and **Display along track** tool buttons open and close their respective windows.

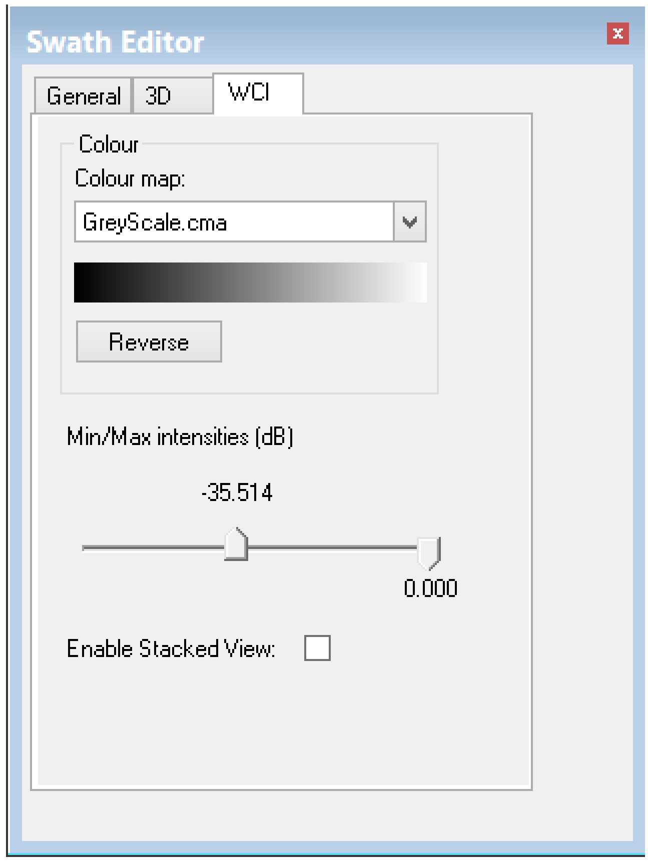
The **Bottom Detection** button toggles the bottom detection on and off. In the across track it will indicates the current swath bathymetry and in the along track, the centre beam bathymetry If activated when Stacked View is enabled, the bottom detections are also stacked.

Head 1 data and **Head 2 data** are used when there is data from both heads of a dual head multibeam configuration. You can toggle between the heads to determine which data should be plotted on top: Head1 (port data) or Head 2 (starboard data).

The **Select** button enables you to select water column data in the across track or along track windows, for example, to add data to the Additional Bathymetry layer.

WCI Controls in Swath Editor

When there is water column data open, the Water Column Editor is available, containing options for displaying water column data.



Use the *Colour* controls set the colour of the data displayed in the Across track and Along track windows.

7. Select a colour from the *Colour map* field.
8. Click **Reverse** to change the current colours to their inverse.

Water column data can be filtered by intensity values, so that only data within the set range is displayed. To filter the data:

9. Use the sliders to set a filter range between -64.000 dB and zero.

These intensity settings are retained so that the same filter range can be applied when the data is loaded into Subset Editor.

Stacked view

Use the *Enable Stacked View* option to see all the across track views layered on top of each other in the Across track window.

1. Make a selection of profiles in the Plan view window.

2. Select the *Enable Stacked View* check box.

The Across track view will display the stacked images.

3. Use the scroll bar on the Plan View, or your middle mouse button, to move the selection to other profiles.

The stacked view will be refreshed with the new data.

You cannot use playback when the data is in Stacked mode.

Add WC Bathymetry to Project

Viewing WCI data in Swath Editor enables you to determine critical data that can then be selected and added as bathymetry in the project. As well, you can then use the editing tools to Reject, Accept and Designate the new additional bathymetry data

To select data in the Across or Along track windows:

1. Click the Select button on the Water Column toolbar.

Tool	

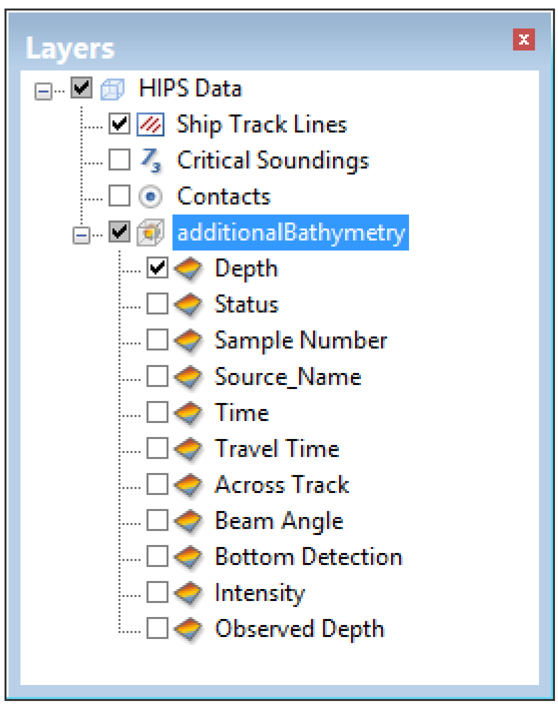
2. Use Select by Range or Select By Lasso to select data.
3. Right-click on the selected data and select Query from the pop-up menu to view in the Selection window.

To add selected data to the project as additional bathymetry:

4. Right-click on the selected data and select Add to Additional Bathymetry.

Pop-up	Add selection to Additional Bathymetry

The data will now appear as "`additionalBathymetry`" layers in the Layers window.



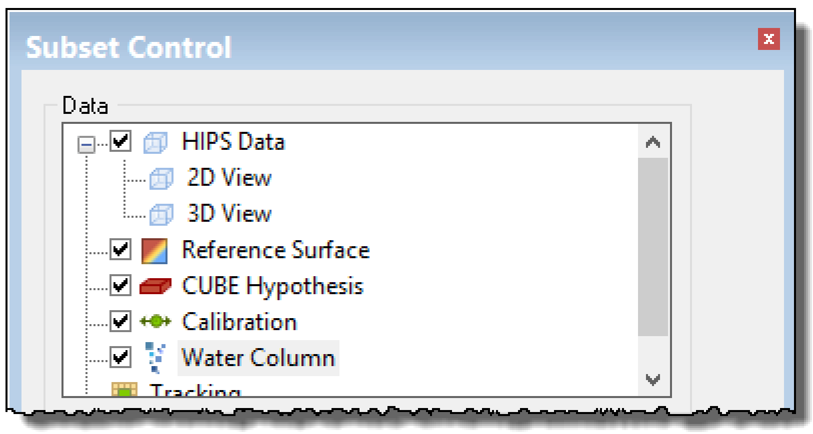
As well, the Output window will report the number of WCI samples exported to the bathymetry layer.

WCI Data in Subset Editor

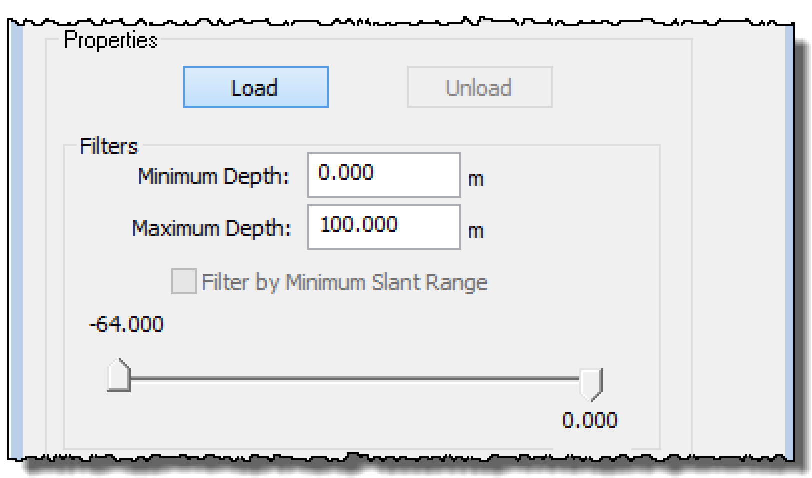
In Subset Editor, WCI data is loaded as a temporary 3D point cloud. Data can be filtered by intensity as well as by a minimum slant range at nadir. WCI data in Subset Editor can also be added to the Additional Bathymetry layer and processed in the standard bathymetry workflow.

To open water column data in Subset Editor:

1. Define and load a subset of the data. (For information on loading a subset see, "Open Subset Editor and Load Data" on page 813.)
2. In the Subset Editor data tree, select the Water Column layer.



The editor is refreshed to display the controls for loading and unloading WCI data, filtering by depth, slant range, and intensity.



3. [Optional] Set a range of depths to load using the Minimum and Maximum Depth fields.

Applying a minimum slant range filter will load only the strong echoes and minimize background noise. Background noise within the minimum slant range has a backscatter strength of lower than -64 dB. Therefore it is logical to load only those data points which have a stronger echo than - 64 dB.

This option is turned off by default.

To set slant range filtering:

4. Select the *Filter by Minimum Slant Range* check box.

Water column data can be filtered by intensity values, so that only data within the set range is displayed. This filter is the same as in Swath Editor. Settings are retained so that the same filter range can be applied in both editors.

To filter the data:

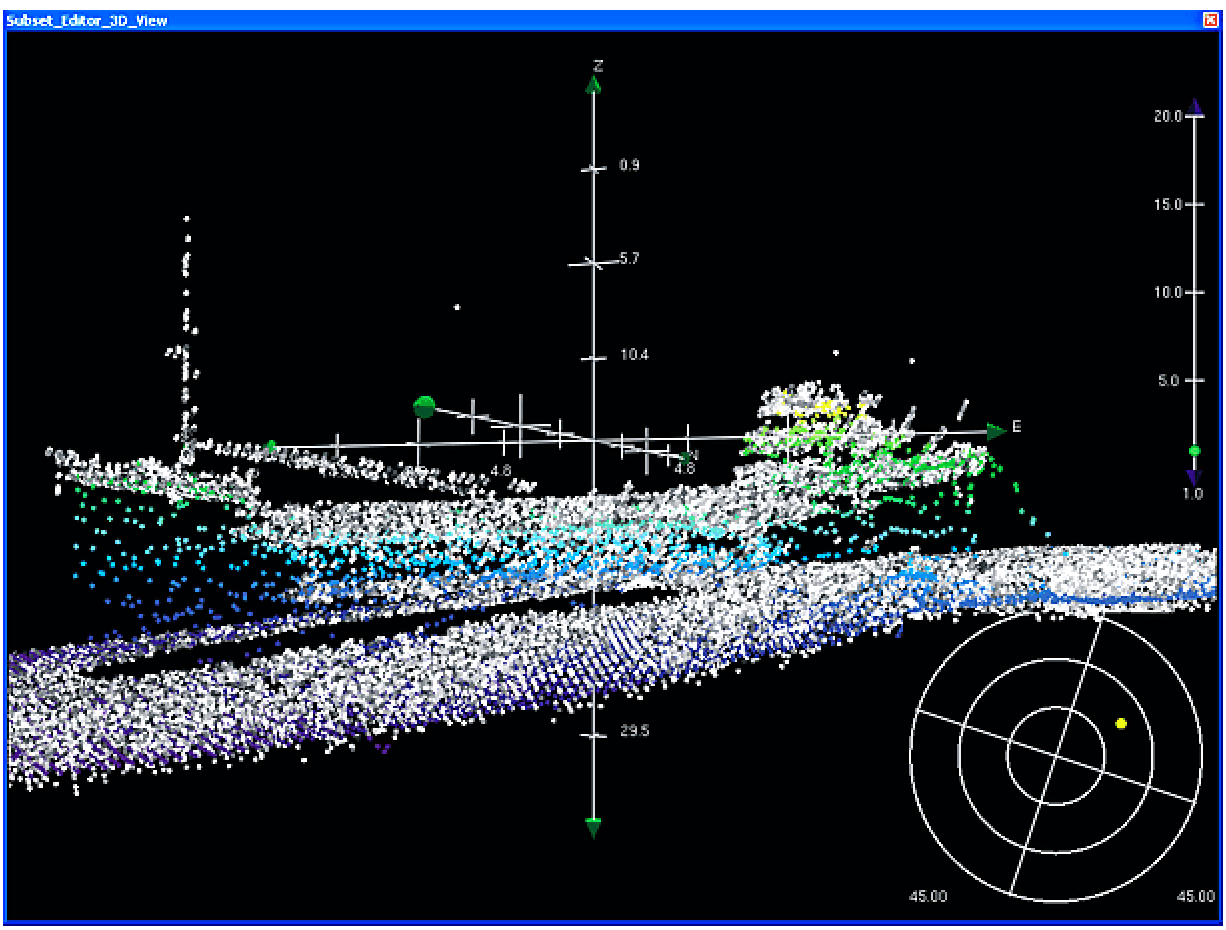
5. Use the sliders to set a filter range between -64.000 dB and zero.
6. Click **Load**.

The face of the **Load** button shows the loading progress. Loading can be cancelled by clicking the **Cancel** button.

When the data is loaded, the button label is changed to **Open**.

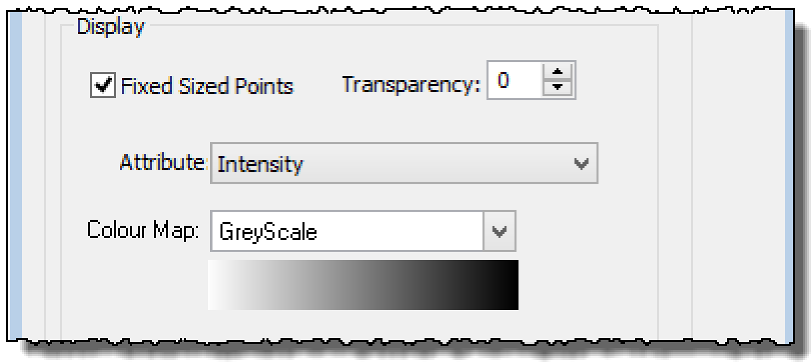
7. Click **Open** to display the data in the Subset Editor windows.

The image below shows WCI data displayed in the Subset Editor 3D view:



Display options

You can set the size of the displayed data points, their transparency and colour them by attribute.



The default setting for the size of points is set in the HIPS Data layer, where the size can be set from 1(smallest) to 10 (largest).

You can override this default setting for the WCI data. To have stronger returns display as larger points, and weaker returns as smaller points, (while other points in the subset are not resized):

1. Select the Water Column layer in the Data tree.
2. Clear the *Fixed Sized Points* check box.

If you want the WCI points to be displayed at a constant size:

3. Select the *Fixed Sized Points* check box.

You can toggle this display option on and off while data is loaded.

Attributes of the WCI data can be displayed one layer at a time. Each attribute (except Bottom Detection) has its own colour map. You can change the assigned colour mapping.

The attributes include:

- Intensity
- Depth
- Line
- Day
- Vessel
- Project

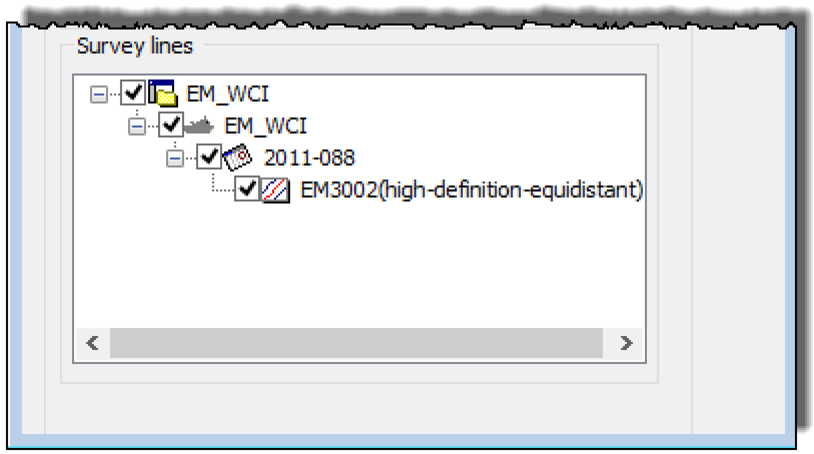
To view the data of a particular attribute:

4. Select the attribute from the drop-down list.
5. Select a colour map from Colour Map drop-down list.

Two colour settings for “Bottom Detection” indicate whether a point is detected bottom or not. You can change the colours of these flag settings also using the colour picker.

Survey lines

The Survey Lines field in the lower part of the Control window shows a data tree in the familiar P/V/D/L layers.



Use the check boxes beside each layer to turn on and off the WCI data associated with individual lines.

Additional Bathymetry

In both Swath Editor and Subset Editor additional bathymetry data can be queried and edited. As well critical soundings can be Designated, and others flagged as Outstanding or Examined.

Restart Cleaning

If you want to reverse the editing of additional bathymetry:

1. Right-click on the additional bathymetry layer, and
2. Select Restart Cleaning from the pop-up menu.

This will restore the status of the data to Accepted.

The `additionalBathymetry` (parent) layer has similar properties to those of a surface: extents, identification, creation date, etc. See "View Surface Properties" on page 220.

As with surface parent layers, there are few editable fields, however, for the additional bathymetry layer there is the option to display any additional bathymetry data that has been rejected in an editor.

1. Set *Show Rejected Points* to True to show rejected data.
2. Set the option to False to hide rejected points.

The attribute (child) layers have the same properties options as do surface attribute layers. See "Attribute Layer Properties" on page 222.

Compute TPU

TPU can be applied to the `additionalBathymetry` layer. This will add two child layers to the additional bathymetry attributes: `DepthTPU` and `PositionTPU`. See "Sources of Uncertainty Data" on page 183.

Regenerate

If corrections such as tide, SVP etc., have been applied to project data during processing, requiring that Merge be re-applied, the additional bathymetry is *not* updated.

To update the additional bathymetry:

1. Right-click on the `additionalBathymetry` layer.
2. Select Regenerate from the pop-up menu.

20

Export Data

Data can be exported by:

- using the HIPS and SIPS Data Export wizard to select data and set export options.
- selecting a surface and a specific export command.
- selecting features and a specific export command.

All export functions are launched from the File > Export menu.

In this chapter...

HIPS and SIPS Data Wizard

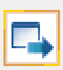
Use this series of dialog boxes to export data to various formats.Depending on the output format, the number of steps will vary.

Exporting to each format begins by selecting the output format (“Step 1: Select Format” on page 505) then selecting the data to be exported (“Step 2: Select Files” on page 506).

Different export formats have specific output options. (“Export Formats” on page 504) All soundings will be transformed to this coordinate system as they are exported.

Many require identifying a coordinate system (“Coordinate System” on page 508).

Once these options are set the last dialog box is used to start and display the progress of the export.

Menu	File > Export > HIPS and SIPS Data
Tool	

To use the HIPS and SIPS Data Export wizard:

1. Select the Export command.

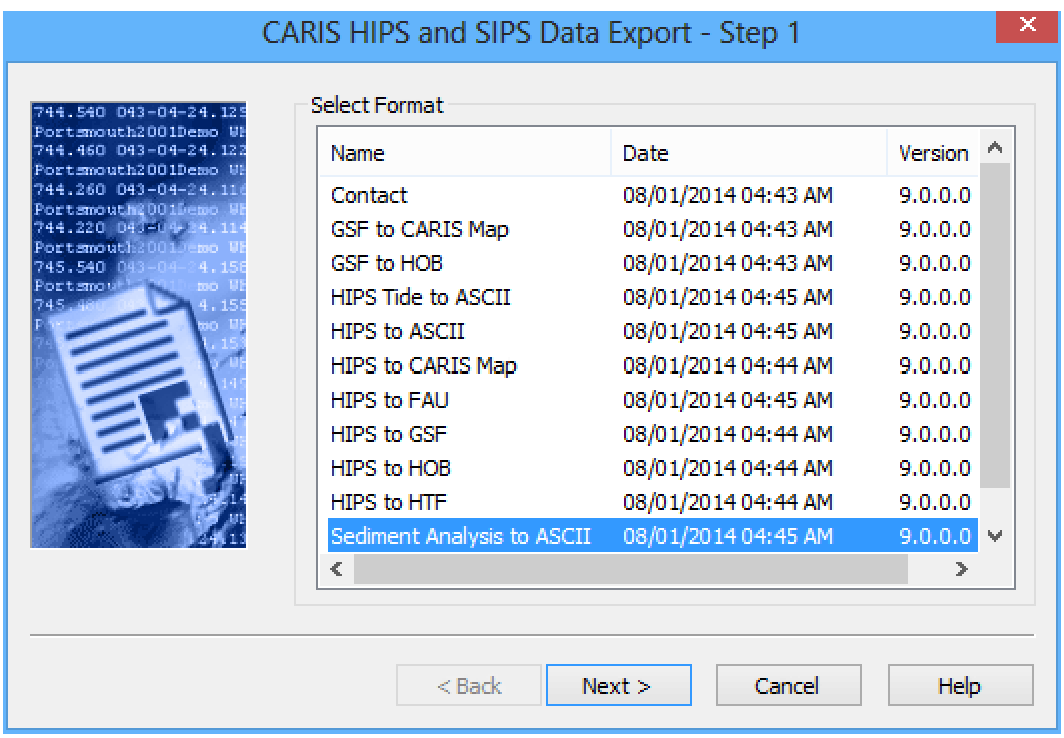
The Data Export wizard is displayed.

Export Formats

Format	Description	For Export settings, see...
Contact	Export contacts to a CARIS map, an ASCII file, and/or a TIFF image.	“Contacts” on page 510
GSF to CARIS map	Export soundings and track lines from one or more Generic Sensor Format (GSF) files to a CARIS map. The GSF data does not have to be converted to HIPS format prior to using this utility.	“GSF to CARIS Map” on page 513
GSF to HOB	Export GSF data to a Hydrographic Object Binary (HOB) file that stores feature objects and associated attribute data. The HOB file can contain links to CARIS spatial objects.	“GSF to HOB” on page 517
HIPS Tide to ASCII	Export tide data from track lines to text format.	“HIPS Tide to ASCII” on page 520
HIPS to ASCII	Export HIPS data to a customized text listing of soundings that you can load into other software systems.	“HIPS To ASCII” on page 521
HIPS to CARIS map	Export HIPS data to a CARIS map. This option contains three modes of exporting data: soundings, track lines and swaths.	“HIPS To CARIS Map” on page 527
HIPS to FAU	Export HIPS data to FAU files.	“HIPS to FAU” on page 532

Format	Description	For Export settings, see...
HIPS to GSF	Export processed HIPS data to GSF format. A new GSF file is created for each track line that is exported from HIPS. Conversely, when HIPS data is created from GSF data, a copy of the original GSF file is maintained inside the HIPS directory structure. The Export to GSF function updates these GSF files.	"HIPS To GSF" on page 533
HIPS to HOB	Export HIPS data to HOB files that store feature objects and associated attribute data. The HOB file can contain links to CARIS spatial objects. This option contains three modes of exporting data: soundings, track lines and swaths.	"HIPS To HOB" on page 534
HIPS to HTF	Export data to a Hydrographic Transfer Format (HTF) file. The HTF is a text file that contains a header section and a sounding record. The Export wizard can save a header section so it can be loaded into multiple HTF files.	"HIPS To HTF" on page 536
Sediment Analysis to ASCII	Export the results of Sediment Analysis.	"Sediment Analysis to ASCII" on page 540
SIPS to UNISIPS	Export SIPS data to a Unified Sonar Imaging Processing System (UNISIPS) format.	"SIPS To UNISIPS" on page 541

Step 1: Select Format



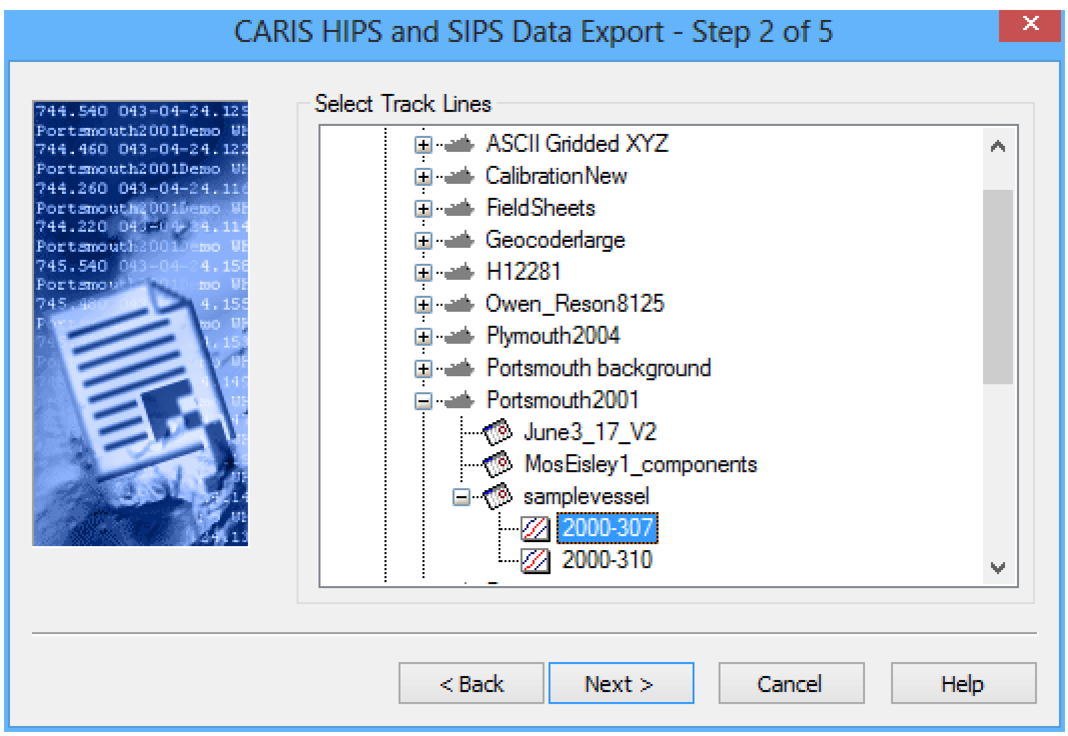
The Select Format dialog box lists all the available export formats. The first step is to select the file format to be exported. The number of steps that follow depends on the export format selected, as there are specific options for some formats.

1. Select an export format.
2. Click **Next** to select data to export.

You may be prompted to select the directory of the data being exported.

Step 2: Select Files

The next step is to select the data to be exported, either by selecting a track line folder from the data tree, or by browsing to a file location.



1. Expand the data tree to select the folder containing the track lines to export.
2. To select more than one line folder, hold down the <`Ctrl`> key and select additional folders.

To export GSF files:

3. Click **Select** and browse to the location of the files to export.

Selected lines can be exported to ASCII, DXF, WKT and GML. See "Export a Selection" on page 564.

Step 3: Export Options

Export options will vary with the data being exported and the formats being exported to. For specific options see:

"Contacts (Step 3)" on page 511

"GSF to CARIS Map" on page 513

"GSF to HOB" on page 517

"HIPS Tide to ASCII" on page 520

"HIPS To ASCII" on page 521

"HIPS To CARIS Map" on page 527

"HIPS to FAU" on page 532

"HIPS To GSF" on page 533

"HIPS To HOB" on page 534

"HIPS To HTF" on page 536

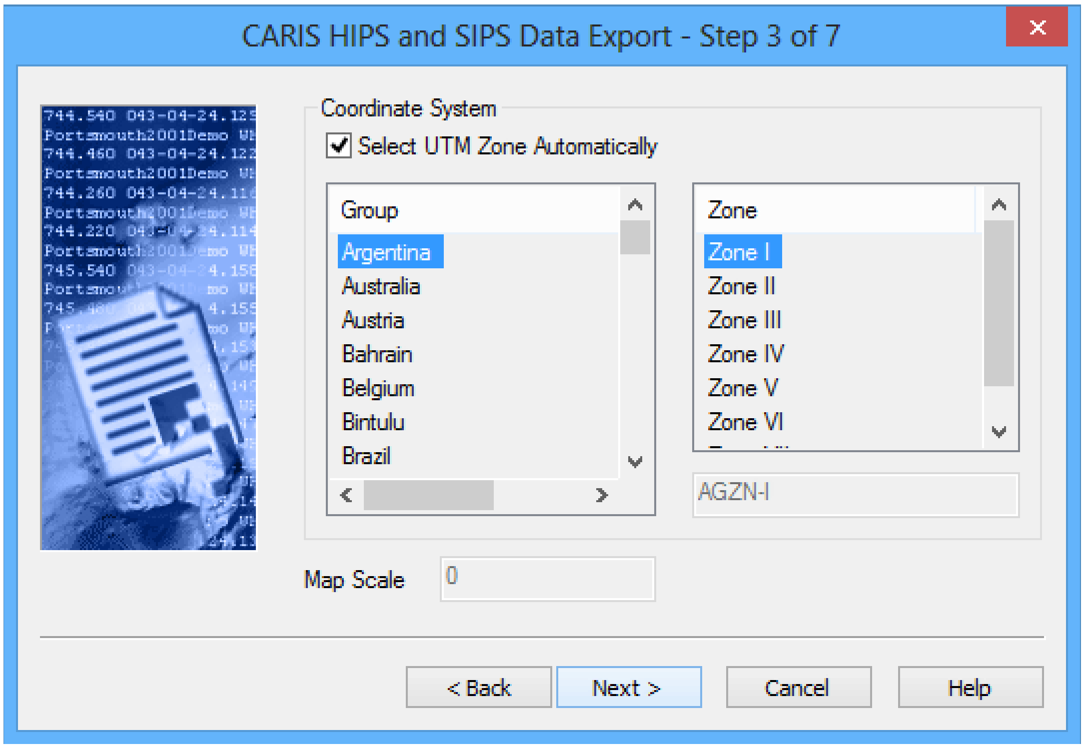
  "HIPS to HTF - Output options" on page 537

"Sediment Analysis to ASCII" on page 540

"SIPS TO UNISIPS (Step 3)" on page 541

Coordinate System

All sounding data is stored in HIPS and SIPS as longitude and latitude. This dialog box sets the appropriate coordinate system to reference soundings. All soundings will be transformed to this coordinate system as they are exported.



By default, the *Select UTM Zone Automatically* option is selected. To set the coordinate systems manually:

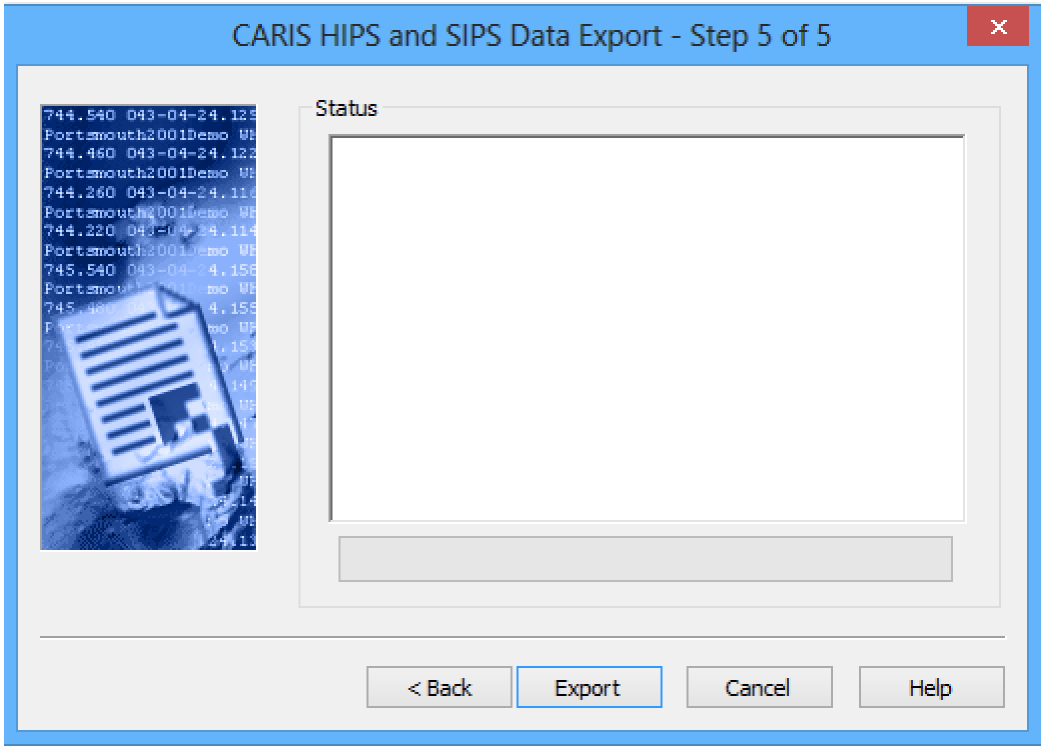
1. Choose a coordinate system by selecting a Group from the list.
2. Select a Zone associated with the Group (the field below it displays the key when the Zone is selected).
3. Click **Next**.

Export

At this step you execute the export of the data with the settings you have chosen.

1. Click **Export** to export the data.

If a large amount of data is being exported, the dialog box will display the percentage of data exported and the status of the export process.



2. Click **Back** to review your settings.
3. Click **Cancel** to call off the process.

A progress indicator shows the percentage of files converted to the selected format.

When the export is completed, the **Restart** button is displayed to enable you to go back and export another file.

Contacts

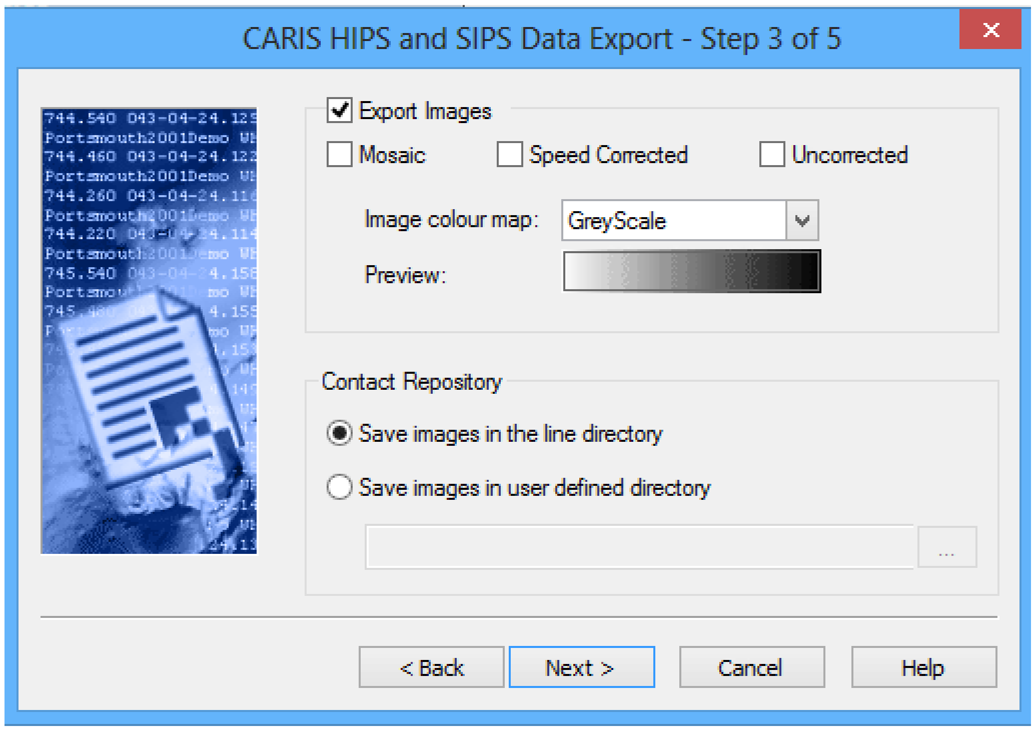
Export contact data for your entire project to a CARIS map and/or an ASCII text file. The ASCII files are structured to be easily imported into a relational database. You can also export contact snapshot images to TIFF files.

If you select ASCII, you will have to browse to a folder location and type a file name. This name is used as the first component of the name of four files that will be created to store the contact information. These files are:

- **filename_ContactLine.txt** contains information identifying the survey line and indexes to the other files.
- **filename_ContactSingle.txt** contains all the information for single point contacts.
- **filename_ContactMulti.txt** contains all information for line contacts.
- **filename_ContactMultiPoints.txt** contains all location information for multipoint contacts.

For information on the structure of exported contact files, see Contact File Formats on page 575 of the HIPS and SIPS Reference Guide

Contacts (Step 3)



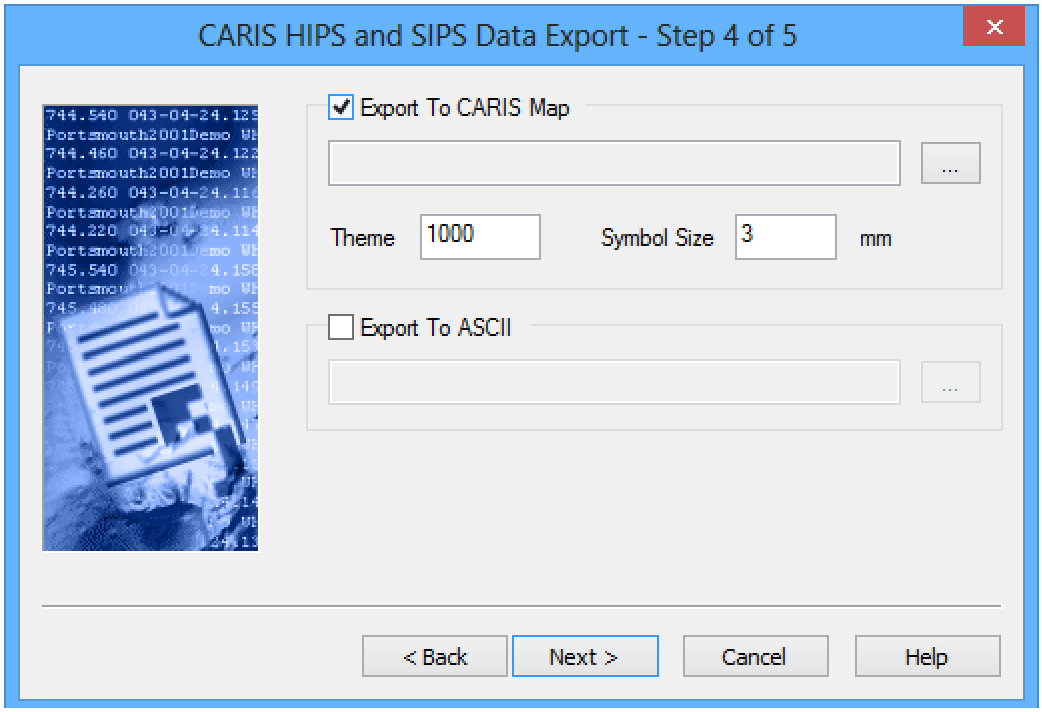
1. Select the *Export Image* check box to export the contact(s) in TIFF image format.
2. If you select *Export Image*, select from the following image formats:
   - *Mosaic*: This option creates a geo-referenced TIFF image from contact data.
   - *Speed Corrected*: This option calculates the distance between profile lines in the contact and corrects for vessel speed.
   - *Uncorrected*: This option does not correct for vessel speed and renders a one-to-one correspondence between the side scan image and the contact.
3. Select an image colour map from the drop-down list.

The selected colour map is displayed in the *Preview* field.

4. Select the directory location for the contact. You can select from two options:
   - *Save Images in the Line Directory*: This option saves the contact data to the line file in the HIPS/SIPS Project/Vessel/Day directory structure that you selected in the second dialog box.
   - *Save Images in User Defined Directory*: This option saves the contact data in another directory. Click **Browse** to select the folder where you want to save the data.
5. Click **Next**.

Contacts (Step 4)

This dialog box contains options for exporting contact data to an ASCII or CARIS file.



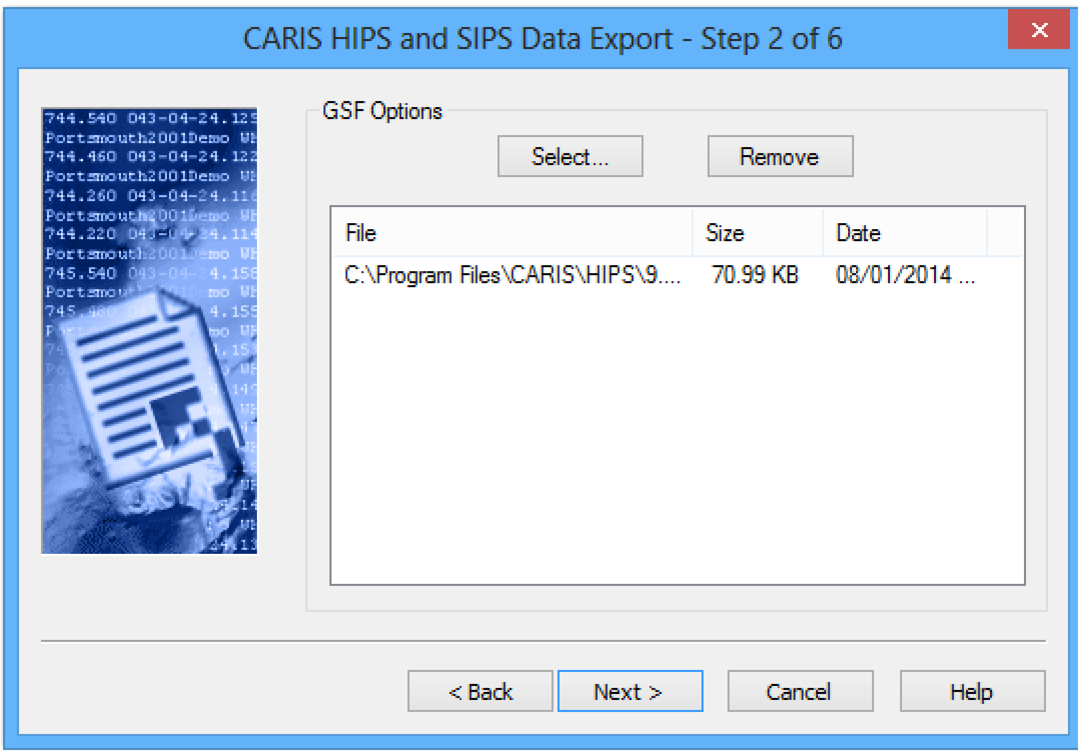
1. Select either the *Export to CARIS Map* or *Export to ASCII* check boxes. Both can be selected at the same time.
2. Type a file name and path to store the contact data after export, or click **Browse** to select the file path (both CARIS data and ASCII text files are typically stored in `..\Hips\CARIS_Files`).
3. Type a layer number for the contact data in the *Theme Number* field.
4. Type a symbol size for the contact data in the *Symbol Size* field.
5. Click **Next**.

“Export” on page 508

GSF to CARIS Map

The Generic Sensor Format (GSF) to CARIS function loads soundings and track lines from one or more GSF files into a CARIS map. GSF data does not have to be converted to HIPS data prior to using this utility. For exporting to HOB, see "GSF to HOB" on page 517.

Select GSF Files

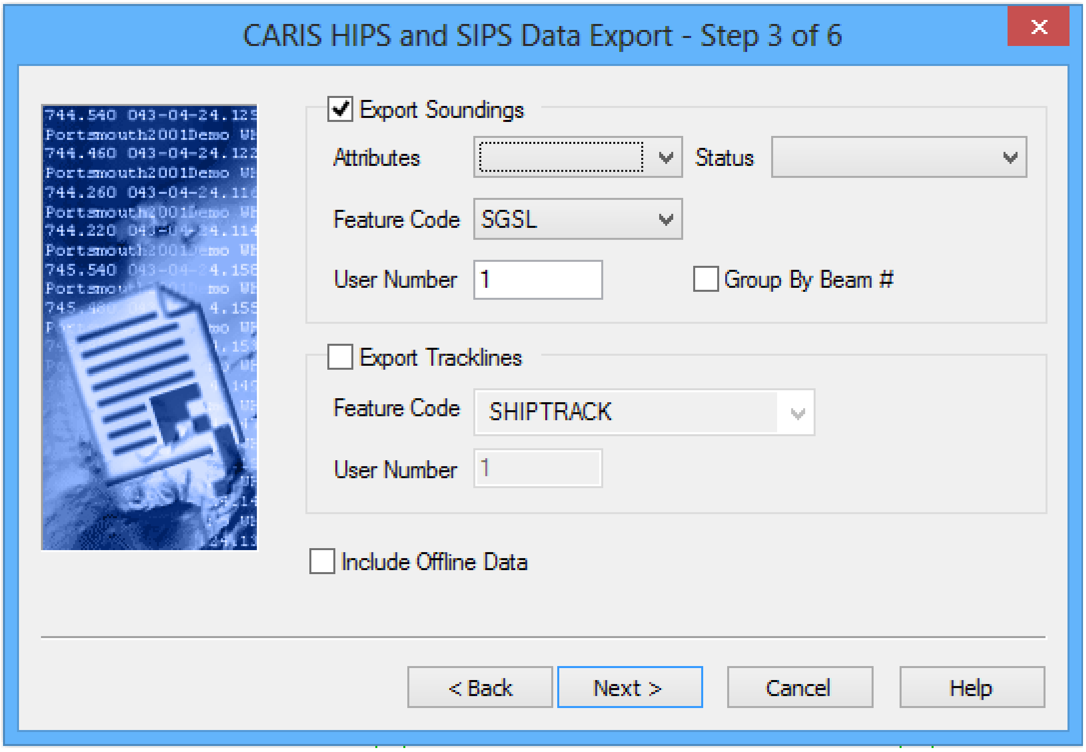


In this dialog box you can list the file path to the GSF file(s) you want to export.

1. Click **Select** to choose the GSF files to be exported.
2. Hold the **<Ctrl>** key while selecting to add more than one file to the list.
3. To remove a file from the list, highlight the file and click **Remove**.
4. Click **Next**.

Export Soundings and Track lines

Data can be exported either in the form of soundings or track lines:



- **Soundings**. Export all soundings that fit within the boundaries of the work file (subject to the HIPS Sounding Status and Data Thinning settings).
- **Track Lines**. Export a line that connects the centre of each swath for each survey line, thus showing the track of the vessel along each survey line. It is not necessary to export soundings to use this option.

1. Select either *Export Soundings* or *Export track lines* or both.
2. If you selected *Export Soundings*, complete any or all of the following fields:
   - *Attributes*: Select *Basic* or *Advanced* from the drop-down list.

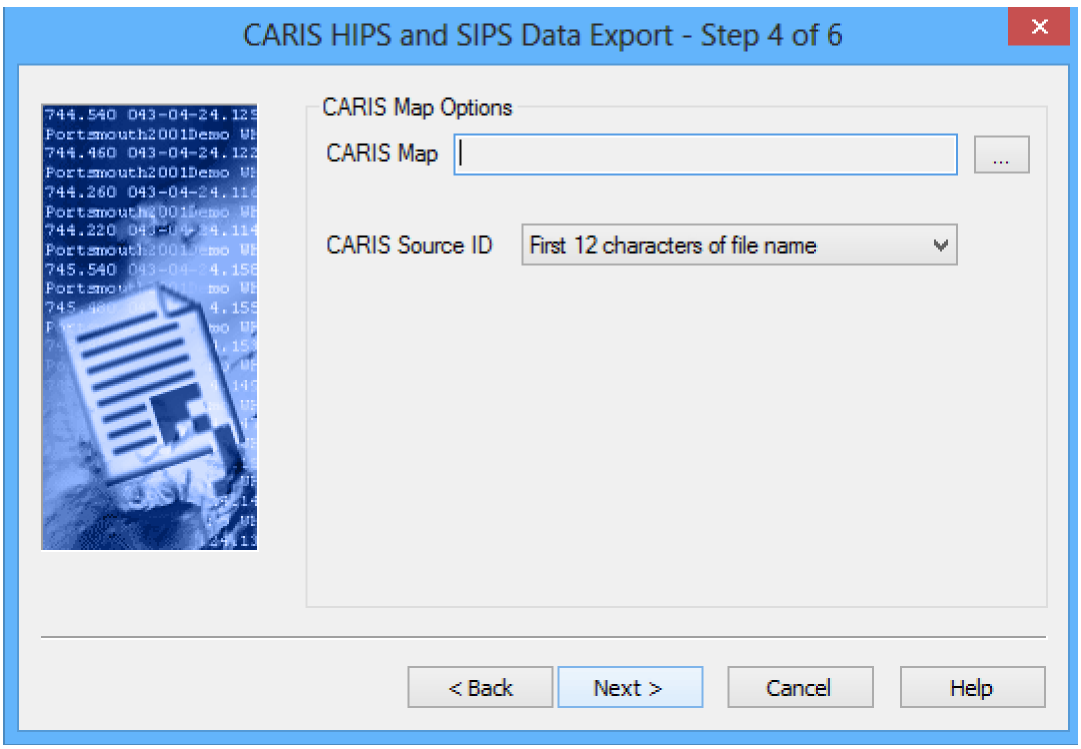
     The *Basic Attributes* option creates mandatory CARIS feature attributes for the map file, such as feature code, user number and source ID.

     The *Advanced Attributes* option creates additional CARIS feature attributes for soundings such as time stamps, launch identifier and tidal correction.
   - *Status*: Select soundings flagged as Selected, Accepted or Accepted and Rejected.
   - *Feature Code*: Select a feature code for the soundings from the drop-down list.
   - *User Number*: Type a number value for layering data in the CARIS map or HOB file.

- *Group By Beam*: Load soundings into user numbers according to sonar beam number. For example, all soundings from beam 37 are loaded to user number 37 in the CARIS map file or HOB file

3. If you selected *Export Tracklines*, complete any or all of the following fields.
    - *Feature Code*: Select a feature code for the track lines object from the drop-down list.
    - *User Number*: Type a number value for layering data in the CARIS map.
4. Select the *Include Offline Data* check box to include data recorded during turns between track lines.
5. Click **Next**.

CARIS Map Options



Use this dialog box to set the destination of the exported data.

1. Click the **Browse** button [**...**] to select CARIS map (*.des file) for data export, or to create a new map file.

The map file path and name are displayed in the *CARIS Map* field.

The *CARIS Source ID* is a 12-character alphanumeric attribute in CARIS maps. The ID can be the first or last 12 characters in a GSF project name.

2. Choose either the first 12 characters or the last 12 characters for the *CARIS Source ID* from the drop-down list.

3. Click **Next**.

GSF to HOB

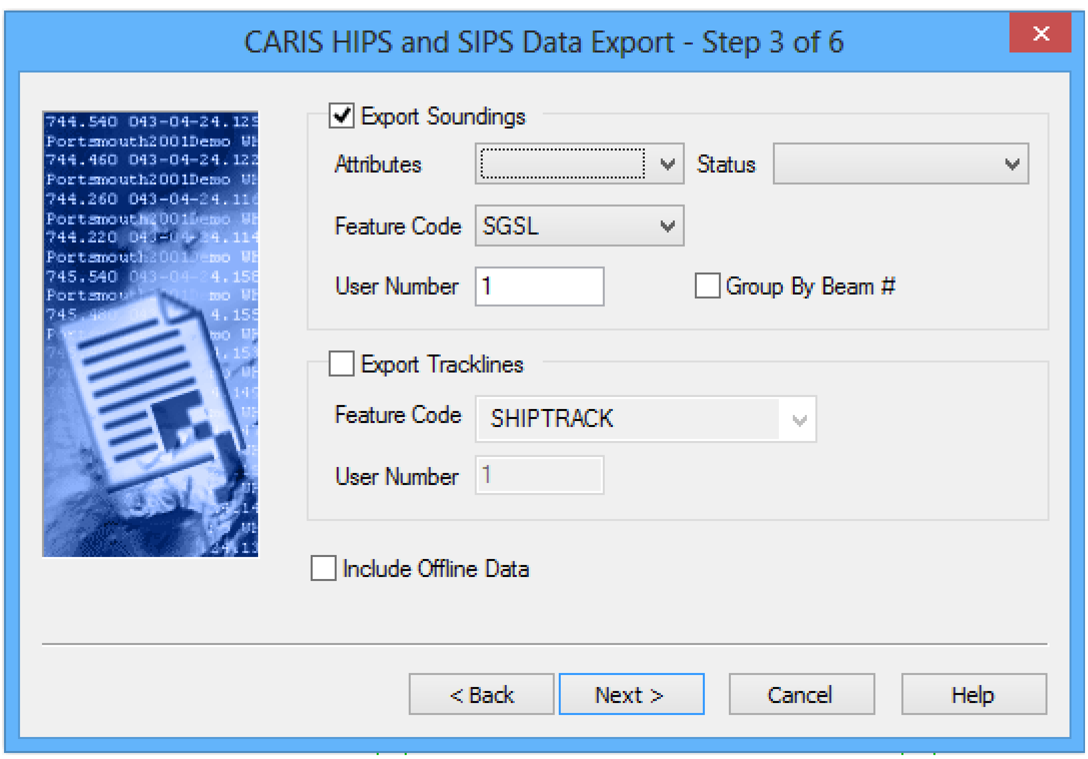
The Generic Sensor Format (GSF) to HOB export loads soundings and track lines from one or more GSF files to a HOB file. GSF data does not have to be converted to HIPS data prior to using this utility. For information on converting GSF to CARIS map, see "GSF to CARIS Map" on page 513.

GSF Files to Export

In this dialog box you can list the file path to the GSF file(s) you want to export.

1. Click **Select** to choose the GSF files to be exported.
2. Hold the <**Ctrl**> key while selecting to add more than one file to the list.
3. To remove a file from the list, highlight the file and click **Remove**.
4. Click **Next**.

Export Soundings and Track lines



Data can be exported either in the form of soundings or track lines:

- **Soundings**. Export all soundings that fit within the boundaries of the work file (subject to the HIPS Sounding Status and Data Thinning settings).
- **Track Lines**. Export a line that connects the centre of each swath for each survey line, thus showing the track of the vessel along each survey line. It is not necessary to export soundings to use this option.

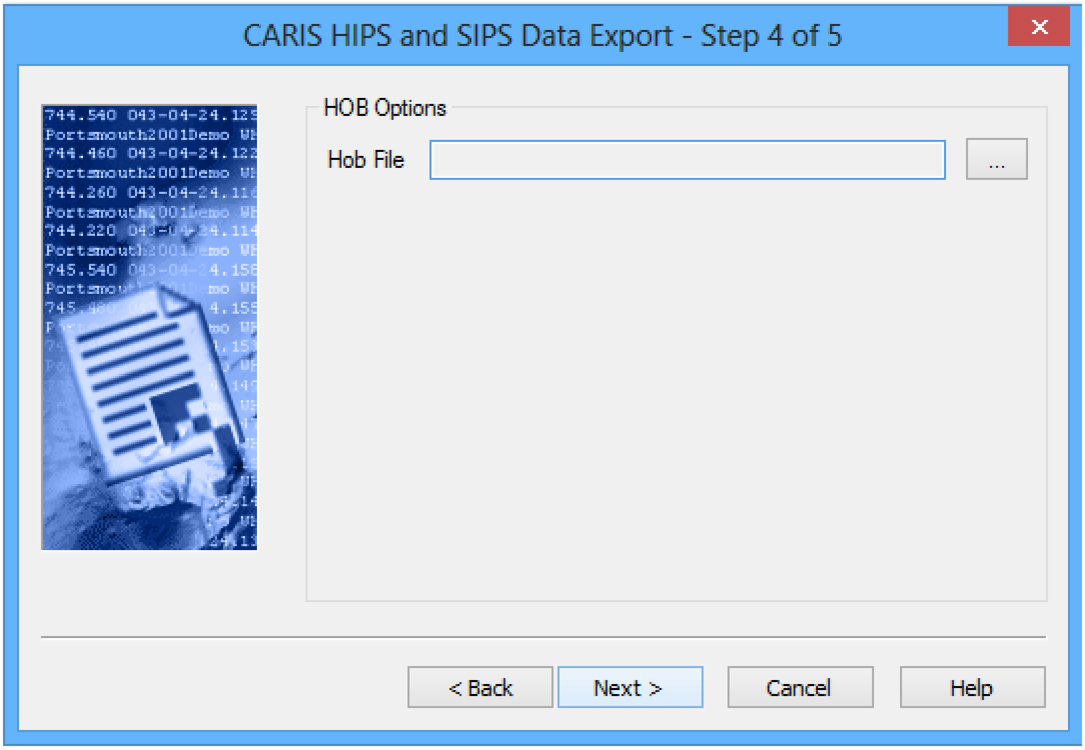
1. Select either *Export Soundings* or *Export track lines* or both.
2. If you selected *Export Soundings*, complete any or all of the following fields:
   - *Attributes*: Select *Basic* or *Advanced* from the drop-down list.

     The *Basic Attributes* option creates mandatory CARIS feature attributes for the map file, such as feature code, user number and source ID.

     The *Advanced Attributes* option creates additional CARIS feature attributes for soundings such as time stamps, launch identifier and tidal correction.
   - *Status*: Select soundings flagged as Selected, Accepted or Accepted and Rejected.
   - *Feature Code*: Select a feature code for the soundings from the drop-down list.
   - *User Number*: Type a number value for layering data in the CARIS map or HOB file.
   - *Group By Beam*: Load soundings into user numbers according to sonar beam number. For example, all soundings from beam 37 are loaded to user number 37 in the CARIS map file or HOB file
3. If you selected *Export Tracklines*, complete any or all of the following fields.
   - *Feature Code*: Select a feature code for the track lines object from the drop-down list.
   - *User Number*: Type a number value for layering data in the CARIS map.
4. Select the *Include Offline Data* check box to include data recorded during turns between track lines.
5. Click **Next**.

GSF to HOB (Step 4)

Use this dialog box to set the location of the HOB file that will contain the exported GSF data.



1. Click the Browse button [...] to select the HOB file for data export.

The Save As dialog box is displayed.

2. Browse to the location where you want to save the HOB file, and click **Save**.

The file path and name will be displayed in the *HOB File* field.

3. Click **Next**.

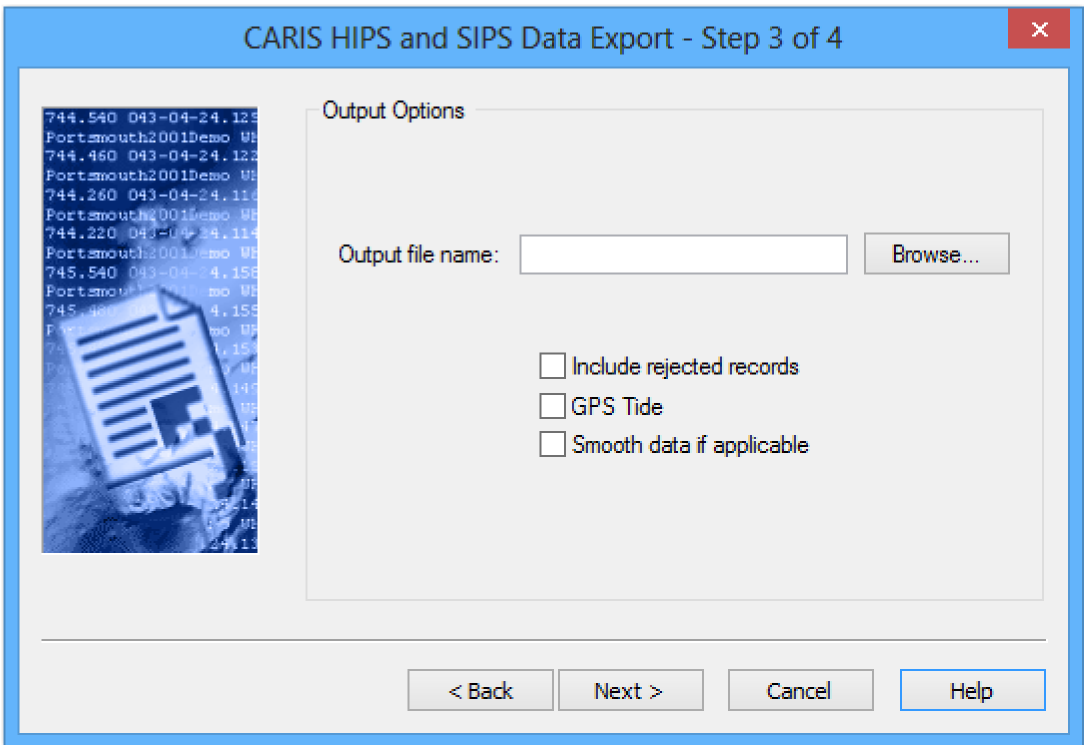
“Export” on page 508

HIPS Tide to ASCII

Export HIPS tide data from selected track lines to a TID file. You can create TID files from single or multiple track lines in a project. TID files can be viewed in Tide Editor or in a text editor.

HIPS Tide to ASCII (Step 3)

The Output Options dialog box creates the output tide file.



1. Click **Browse** and use the standard Windows Open dialog box to select a path and name for the output TID file.

The path and name of the tide file is displayed in the *Output File Name* field.

2. Select the *Include Rejected Records* check box to include tide records that have been rejected.
3. Select *GPS Tide* to use tides calculated from a GPS ellipsoid height.
4. Select *Smooth data* to use tide data that has been smoothed for localized variability.

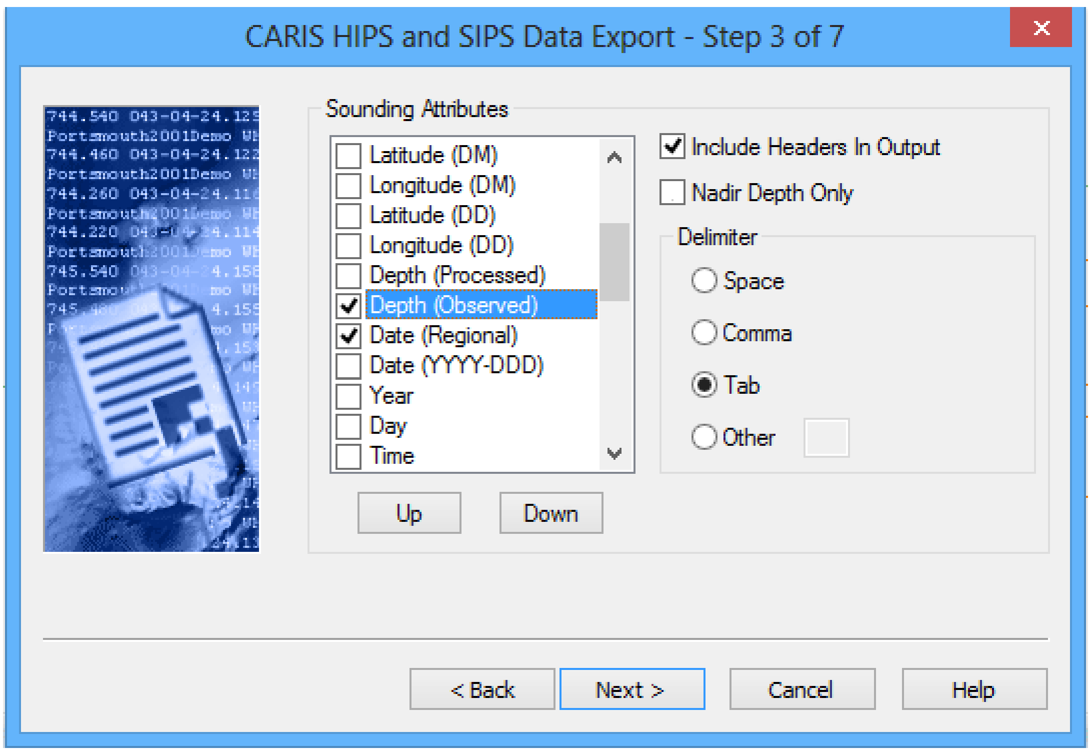
"EXPORT" ON PAGE 508

HIPS To ASCII

The HIPS to ASCII function creates a customized text listing of soundings that you can load into other software systems.

HIPS to ASCII (Step 3)

This dialog box lists the attributes that can be exported to the ASCII file and the delimiters used to separate data in the file.



The attributes that are available to export will vary with the type of data being exported, for example, if single beam data is being exported, Primary Depth and Secondary Depth are listed with other attributes in the Available list.

1. Select the *Sounding Attributes* to be included in the ASCII file.
   - To add an attribute, select the attribute in the *Available* list and click **Add** to move it to the *Active* list.
   - To remove an attribute, select the attribute in the *Active* list and click **Remove** to move it to the *Available* list.

If you select attributes which have values expressed in units of measure, for example, Depth, when you click **Next** the Select Units dialog box will be displayed. See “Select Units” on page 522.

2. To change the order that attributes are exported, select the attribute in the *Active* list and click the up or down arrow buttons.
3. Select the *Headers* check box to create print attribute names at the top of the attribute rows.
4. Select either the *Space*, *Comma*, or *Tab* check boxes to use these characters as delimiters, or select the *Other* check box and type a character to use as a delimiter.

You can define the boundaries of the ASCII map data from the boundaries of an existing CARIS map. If you do not select this option, this is defined from the ASCII data.

5. Select the *Clip to CARIS Map* check box to use a CARIS map to define the boundaries of the ASCII map data.
6. Click **Browse** to select an existing CARIS map.

To use the coordinate system from the selected CARIS map for the ASCII data:

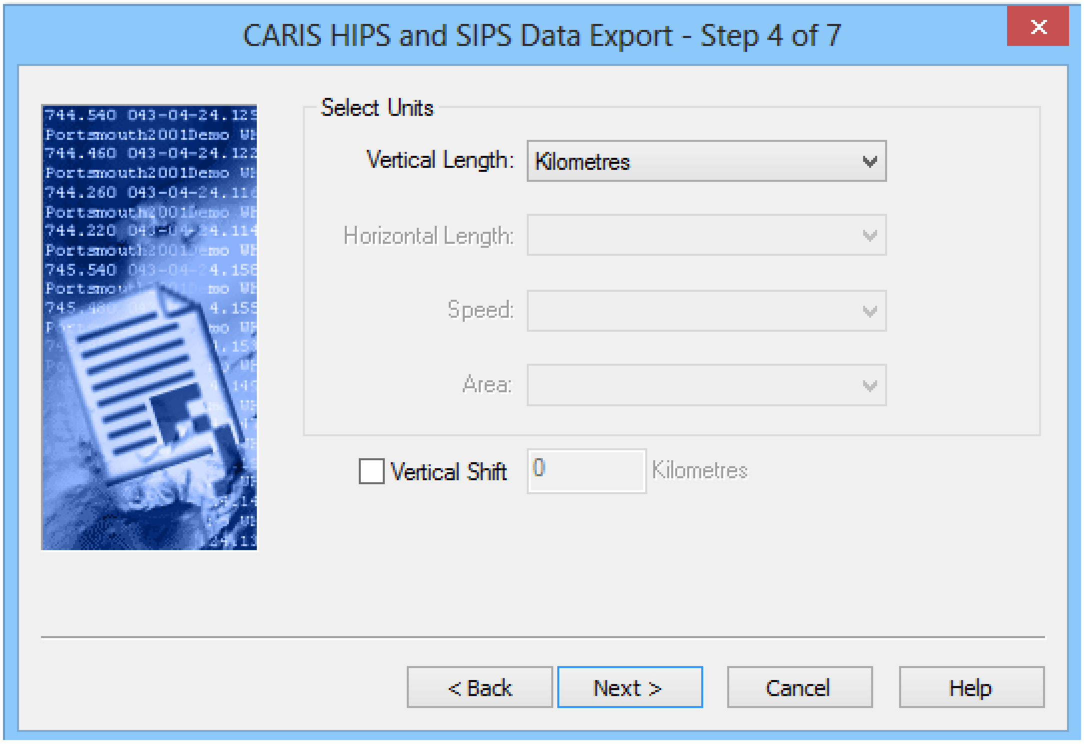
7. Enable the *Use Coordinate System From CARIS Map* check box.
8. Click **Next** to export the data.

If you do not select *Use Coordinate System From CARIS Map*, the next dialog box prompts you to select a coordinate system.

9. Click **Next**.

Select Units

If you selected attributes which have values expressed in units of measure, the Select Units dialog box will be displayed. The unit files are activated by the type of attribute you selected. Units which are not available to the selected attributes will be greyed out.



Vertical length units are required for these attributes:

- Depth TPU
- Tide
- Depth (Observed)
- Depth (Processed)

Horizontal length units are required for these attributes:

- Position TPU
- Northing
- Easting

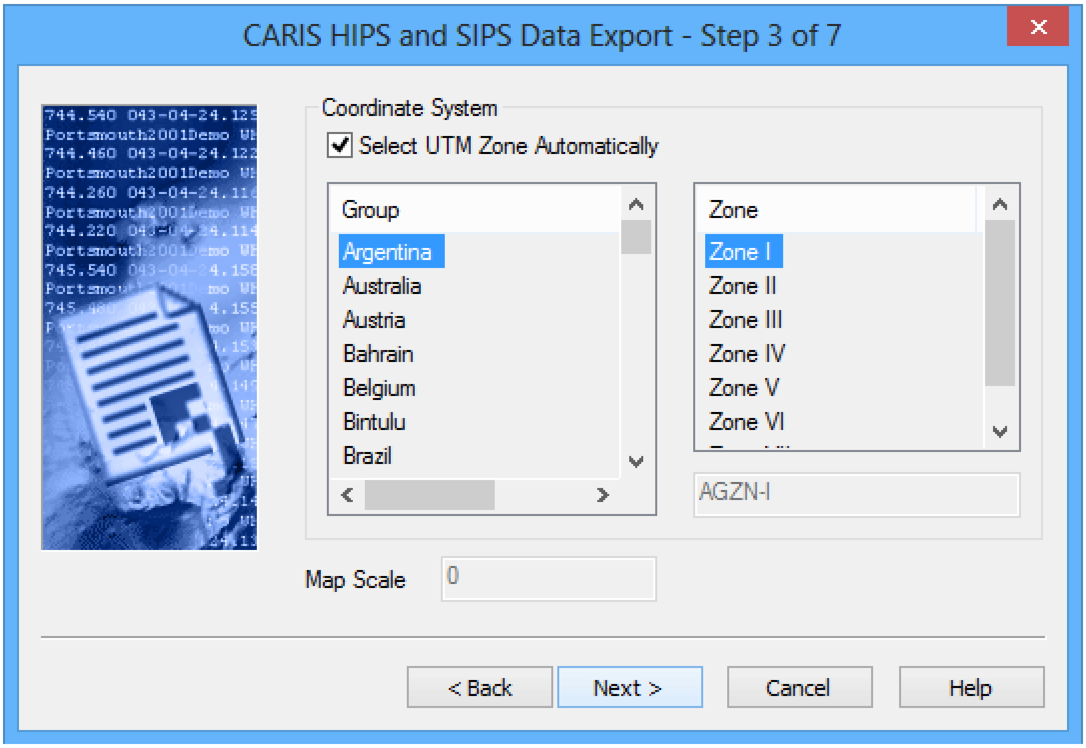
1. Select a unit of length from the *Vertical Length* or *Horizontal Length* lists.

To apply a static vertical shift to depths:

2. Click the Vertical Shift check box to activate the value field.
3. Type the amount of shift to be applied.
4. Click **Next.**

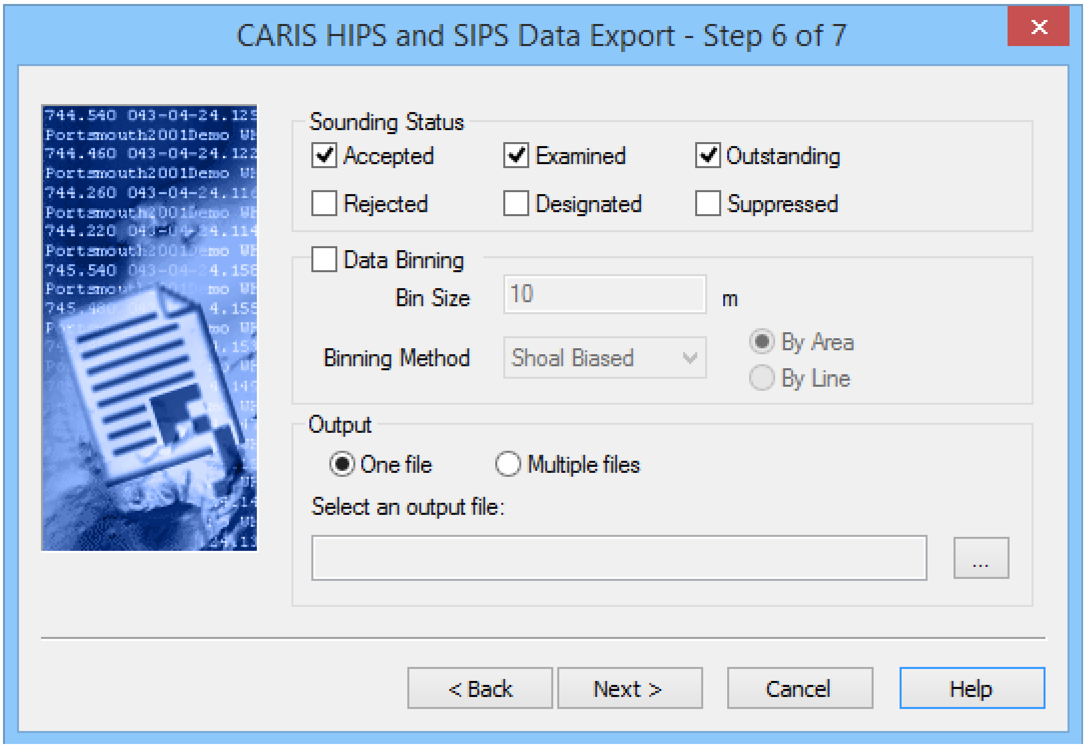
Coordinate System

All sounding data is stored in HIPS and SIPS as longitude and latitude. This dialog selects the appropriate coordinate system to reference the soundings. All soundings will be transformed to this coordinate system as they are exported.



1. Choose a coordinate system by selecting a Group from the list.
2. Select a Zone associated with the Group (the field below it displays the key when the Zone is selected).
3. Click **Next**.

Sounding Status and Output File



Each sounding in a cleaned HIPS data set are stored with bits that determine the status of that sounding. You can choose to restrict the export process to soundings with specific status flags.

1. To export soundings with a specific status flag, select the appropriate *Sounding Status* check boxes.
2. To apply a data thinning function, select the *Data Binning* check box.

The *Data Binning* option divides the survey area into a grid and selects either the shoal- or deep-biased soundings from each cell for export. The *Bin Size* option sets the size of the grid cells.

3. Type the size for the grid cell in the *Bin Size* field. The units used here can be changed in the Tools > Options > Display >Units dialog box.
4. Select either the *Shoal-Biased* or *Deep-Biased* options to export only the shoalest or deepest soundings, respectively, to the ASCII file.
5. Select the *By Area* option to export one sounding per bin.
6. Select the *By Line* option to process each line, and export a sounding for each line per bin. Where line data overlaps this method can give multiple soundings per bin.
7. Select an output location for the data. Select *One File* to export all data to a single file or select *Multiple Files* to export data to separate files for each survey line.

- If you select *One File*, type a name and file path for the ASCII file, or click **Browse** to select a location and name for the file.
- If you select *Multiple Files*, click **Browse** to select a folder to save the files. The files are automatically named according to Project_Vessel_Date_SurveyLine.

8. Click **Next**.

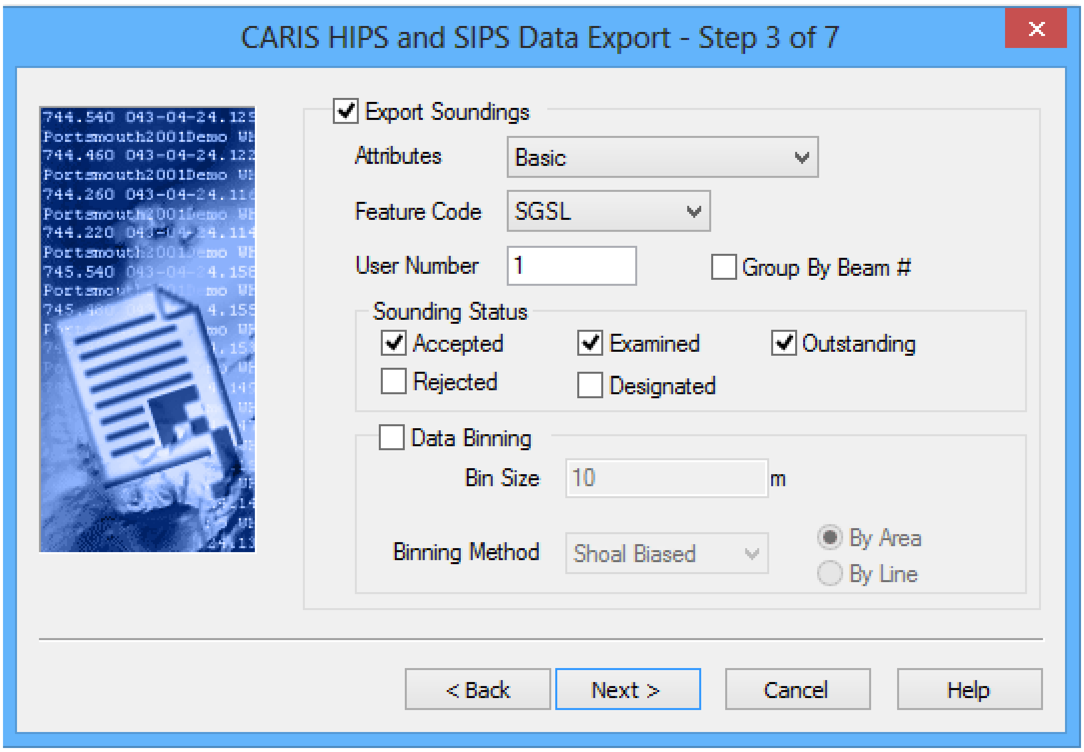
"EXPORT" ON PAGE 508

HIPS To CARIS Map

Export data to a CARIS map.

HIPS to CARIS Map (Step 3)

You can select which soundings are to be exported and how they will be stored in the CARIS map.



The *Export Soundings* option exports all soundings (subject to the sounding status and data binning settings).

1. Select the *Export Soundings* check box if you want to export soundings to a CARIS map. (See the next dialog box for the *Export Tracklines* and *Export Swaths* options).

There are four methods for exporting sounding attributes to a CARIS map.

- *Basic*: The basic sounding attributes in the CARIS map. These include user number, feature code, and source ID. This information is adequate to generate products such as sounding plots, contour plots, and DTMs.
- *Extended*. Basic attributes plus time stamps, launch identifier (first four letters of the vessel name), and tidal correction.
- *Extended with Key*: The extended attributes set, plus the profile number and beam number as the CARIS Key. The profile

number and the beam number, in addition to the other attributes, allow each sounding to be identified uniquely with respect to the original data set.

- *Extended with Unique Key*: The extended attributes set, plus every object is given a unique key.

This additional information allows more flexibility for visualizing and manipulating CARIS files. For example, if time stamps are associated with each sounding, you can limit the display of soundings based on a time range.

However, each extra attribute adds to the storage requirements. More disk space is required to store the work file, and any HIPS program that processes the file must do more work and use more time and CPU resources.

2. Select from the four options in the *Attributes* drop-down list.
3. Select a new *Feature Code* for the soundings, if needed.

The *Group by Beam Number* option layers soundings according to a beam number in the CARIS file. Soundings belonging to a beam are assigned to the same user number in the map. User numbers in CARIS file are similar to layering. This option is used when exporting soundings from check lines into a separate check line map so that a Quality Control Report can be generated. If you do not select this option, then a single user number will be used to hold all the exported soundings.

4. Select the *Group by Beam Number* check box to layer soundings according to beam numbers in the CARIS map.

The *User Number* option is available only if you do not select the *Group by Beam Number* option.

5. [Optional] Type a *User Number* to which all soundings are to be assigned.

The *Sounding Status* option enables you to export only soundings with a specific status flag.

6. To export soundings with a specific status flag, select any of the *Sounding Status* check boxes.

The *Data Binning* option divides the survey area into a grid for sounding selection. The *Bin Size* sets the size of the grid cells.

7. To apply a data binning function on export, select the *Data Binning* check box.
8. Type the size for the grid cell in the *Bin Size* field and select either *Metres* or *Feet*.
9. Select either *Shoal-Biased* or *Deep-Biased* to export only the shoalest or deepest soundings from each cell, respectively, to the CARIS file.

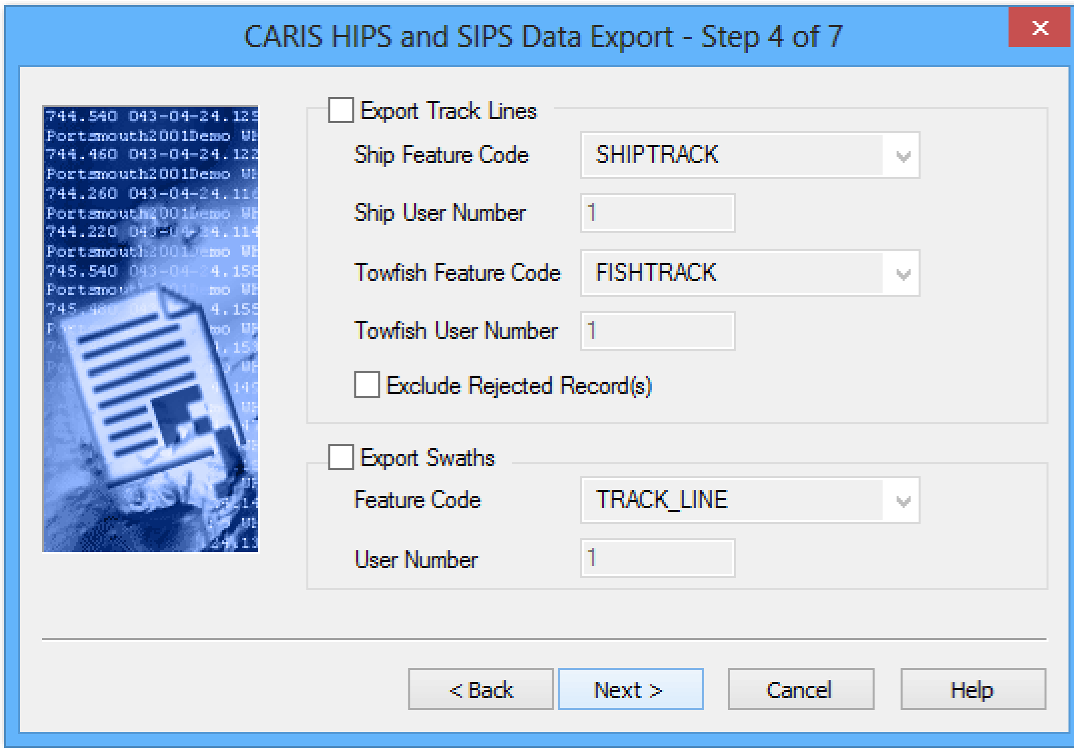
Soundings can be selected for export from the total survey area or by processing each track line in sequence.

10. Select either the *By Area* or *By Line* options.

11. Click **Next**.

HIPS to CARIS Map (Step 4)

This dialog box sets the track line and swath export options.



The *Export Track Lines* option connects a line along the centre of each swath, thus showing the track of the vessel along each survey line. It is not necessary to import soundings to use this option.

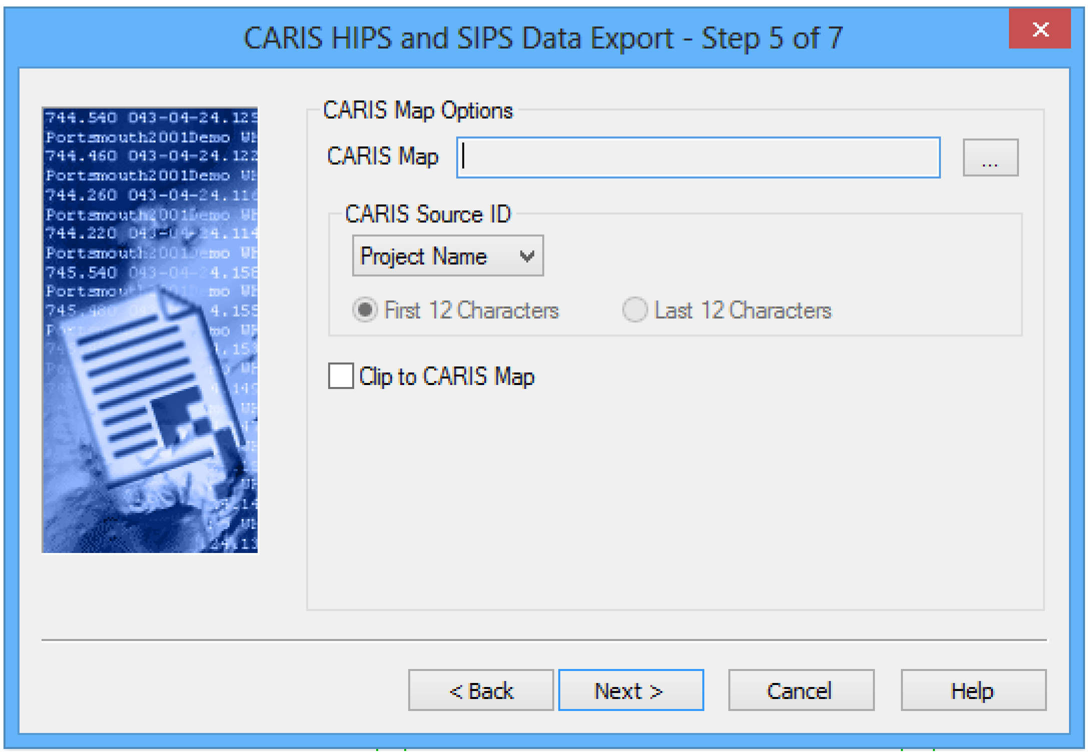
The *Export Swaths* command connects a set of lines (one per swath) along the outermost port and starboard beams, thus showing the coverage of each ping. It is not necessary to import soundings to use this option.

1. Select the *Export Track Lines* check box to export a line representing the track of the vessel along the survey area.
2. Type a new feature code for the vessel, if needed.
3. Type a *User Number* for the vessel so it is distinguished from other vessels in the same CARIS map.
4. Select a new *Towfish Feature Code*, if needed.
5. Type a *User Number* for the towfish so it is distinguished from other vessels in the same CARIS map.
6. Select the *Exclude Rejected Record(s)* check box to not include this data.

7. Select the *Export Swaths* check box to export a set of lines that represents the swaths along the survey.
8. Type a *User Number* for the swath so it is distinguished from other swaths in the same CARIS map.
9. Click **Next**.

HIPS to CARIS Map - Options

1. Click **Browse** to select a CARIS map for data export, or type the name of a new map.



The map file path and name are displayed in the *CARIS Map* field.

The Source ID is a 12-character alphanumeric attribute in CARIS maps. The ID can be the first or last 12 characters in a project name or line name.

2. Select either *Project Name* or *Line Name* to use as a Source ID.
3. Choose either the first 12 characters or the last 12 characters for the Source ID.

The *Clip to CARIS Map* option cuts the HIPS data at the boundaries of an existing CARIS map. If this option is not selected, the geographic extent of the map are extended to accommodate the HIPS data.

4. Select the *Clip to CARIS Map* check box to implement this option.

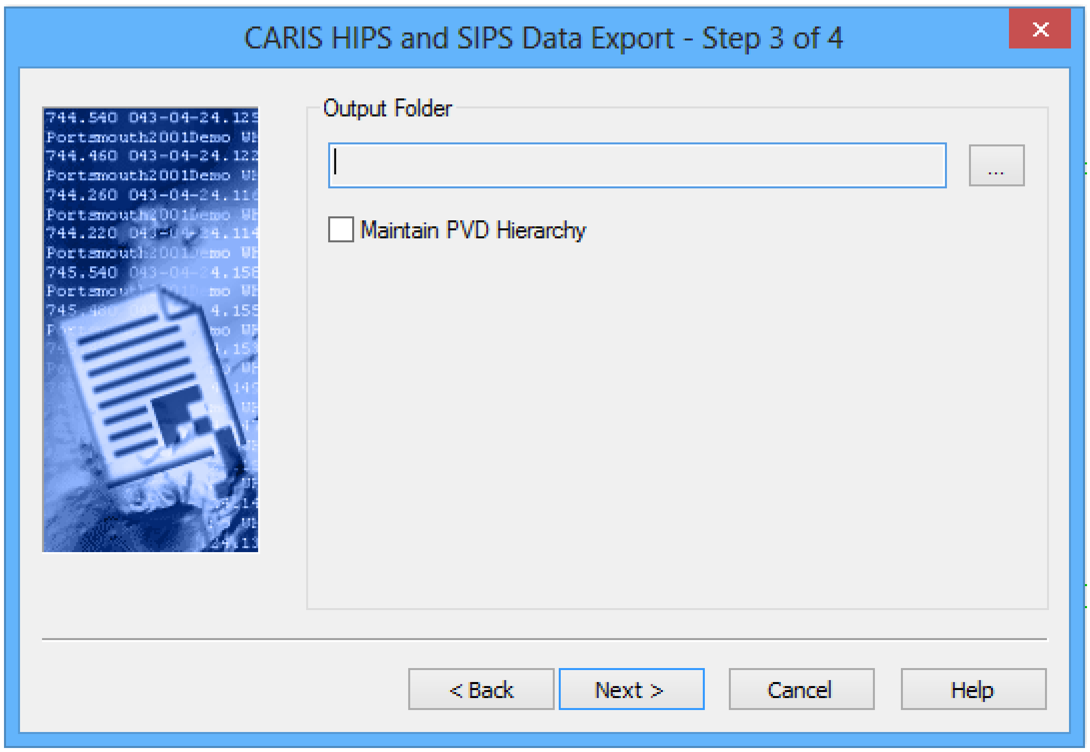
5. Click **Next**.

If you created a new CARIS map in Step 1, a dialog box for selecting a coordinate system is displayed. Otherwise you can begin export of the data in the next dialog box.

HIPS to FAU

Export processed HIPS data, with optional heave/pitch/roll/quality/amplitude attributes, to the FAU format. The process will create multiple files with the extension *.`fauTC` designating the data as “tide corrected”. The names of the output files will match the HIPS line names.

After you have selected the track lines to export, the next step is to designate an output folder.



1. Type a path and name for the output file, or click the Browse button (**...**) to select a folder and then type the file name.
2. Select *Maintain PVD Hierarchy* to create output directories that correspond to the HIPS data being exported.
3. Click **Next** to set the coordinate system.

“Coordinate System” on page 508

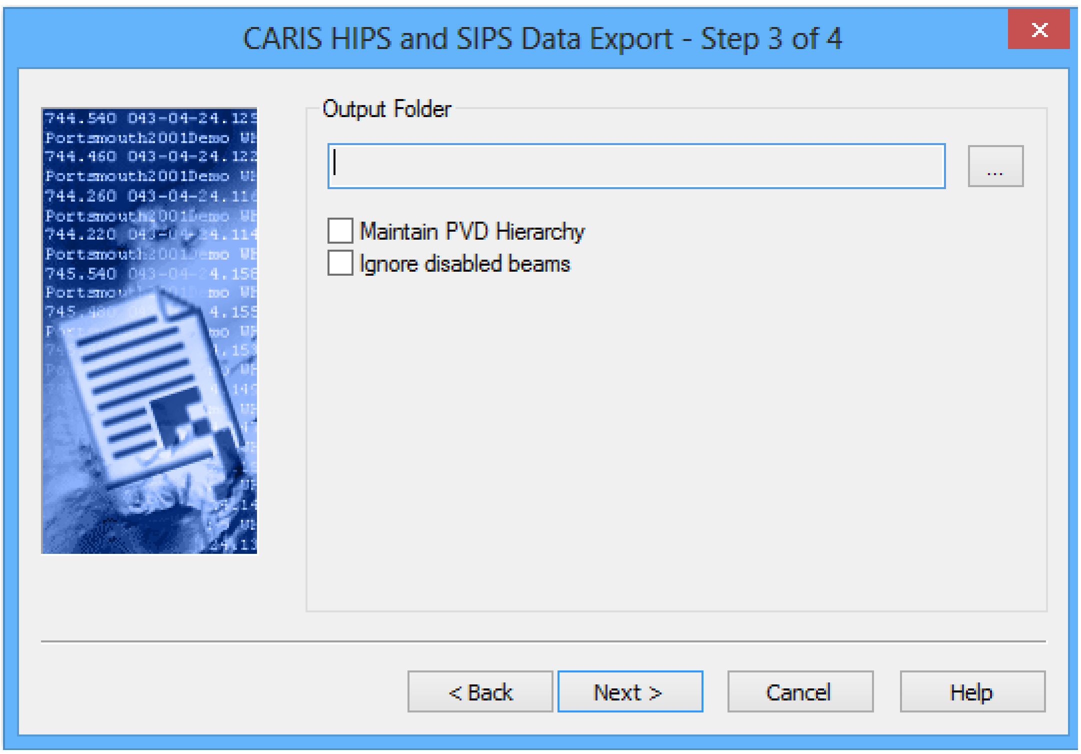
“Export” on page 508

HIPS To GSF

The HIPS to GSF function exports processed HIPS data to a GSF format. A new GSF file is created for each track line that is exported from HIPS.

Conversely, when HIPS data is created from GSF data, a copy of the original GSF file is maintained inside the HIPS directory structure. The Export to GSF function creates a new GSF file using the copy and updates these GSF files.

Output Folder



1. Click the **Browse** button (**...**) to select a location for exported GSF files.

The Browse for Folder dialog box is displayed. The stored file will have the same name as the track line.

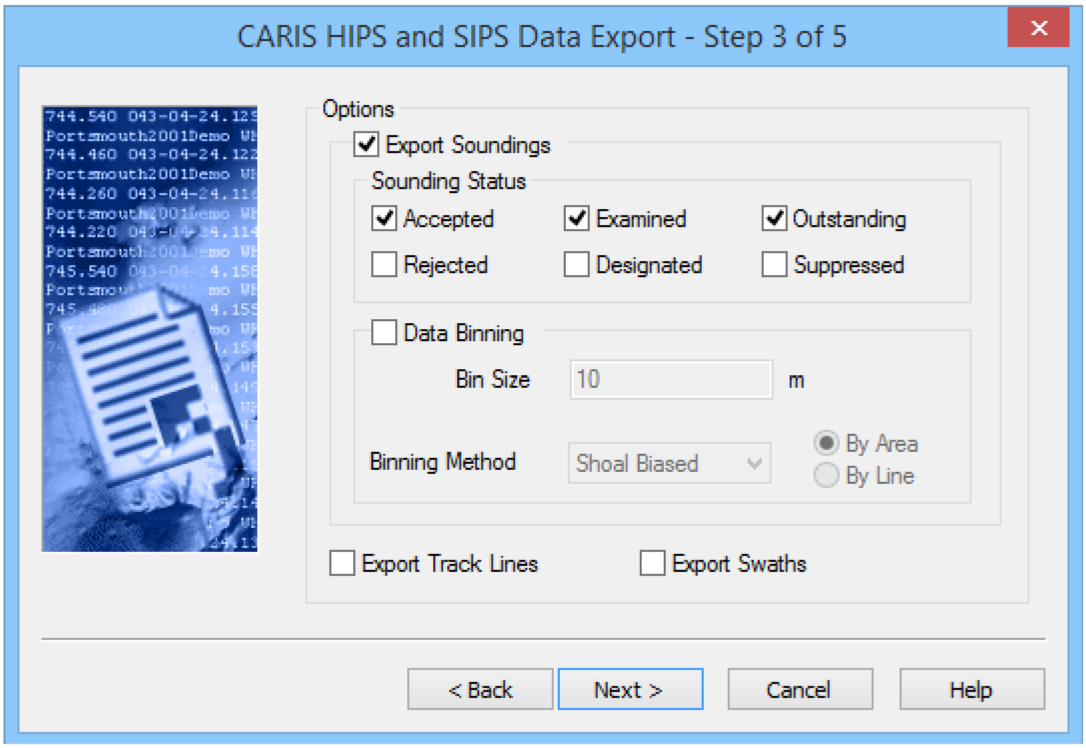
2. To export entire projects and maintain the HIPS Project/Vessel/Day/Line structure in the output, click the *Maintain PVD Hierarchy* check box.
3. Select the *Ignore Disabled Beams* option to exclude any soundings which have been rejected (because of disabled beams) from your export file.
4. Click **Next**.

"Export" on page 508

HIPS To HOB

The Hydrographic Object Binary (HOB) file stores feature objects and associated attribute data.

HIPS to HOB (Step 3)



Use the *Export Soundings* option to define which soundings are exported using the sounding status and data binning settings.

1. Select the *Export Soundings* check box to export soundings to a HOB file.
2. To export soundings with a specific status flag, select any of the *Sounding Status* check boxes.

The *Data Binning* option divides the survey area into a grid for sounding selection. The *Bin Size* sets the size of the grid cells. To apply data binning:

3. Select the *Data Binning* check box.
4. Type the size for the grid cell in the *Bin Size* field and select either Metres or Feet.
5. In the *Binning Method* field, select either Shoal Biased or Deep Biased to export only the shoalest or deepest soundings, respectively.

Soundings can be selected for export by processing each track line in sequence or from the total survey area.

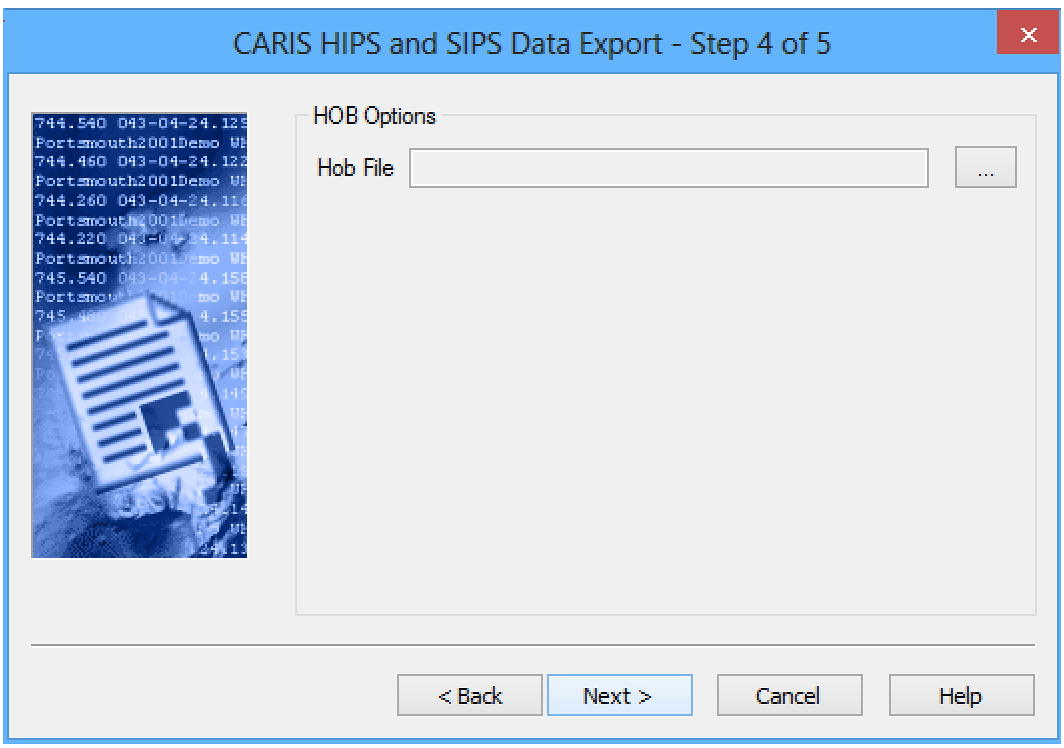
6. Select either the *Export Track Lines* or *Export Swaths* options.

The *Export Track Lines* option connects a line along the centre of each swath, thus showing the track of the vessel along each survey line. It is not necessary to export soundings to use this option.

The *Export Swaths* command connects a set of lines (one per swath) along outermost port and starboard beams, thus showing the coverage of each ping. It is not necessary to export soundings to use this option.

7. Select the *Export Track Lines* and *Export Swaths* options.
8. Click **Next**.

HIPS to HOB (Step 4)



1. Click the **Browse** button to select a location for the HOB files.

The Browse for Folder dialog box is displayed.

2. Select a directory to store the HOB data and click **OK** to close the dialog box.

The file path is displayed in the *Output File* field.

3. Click **Next**.

"Export" on page 508

HIPS To HTF

The HIPS and SIPS Export Wizard converts data and soundings to a Hydrographic Transfer Format (HTF) file.

Hydrographic Transfer Format

The HTF file is an ASCII file that can be opened in any text editor (such as Notepad). It consists of three sections:

- HTF Header, which contains information about the data, such as coordinate system, survey title, date, data density, number of positioning systems etc.
- Sounding Header, which describes the fields that are in the sounding data section.
- The sounding data record contains the exported data on depths and other relevant information.

The total number of fields generated in the sounding data record depends on the survey. At the start of the sounding data record is a Field Population Key that gives each field a 1 bit to show that a field is included, or a 0 bit to show if a field is not included.

For example: `[00001111100000000000]`

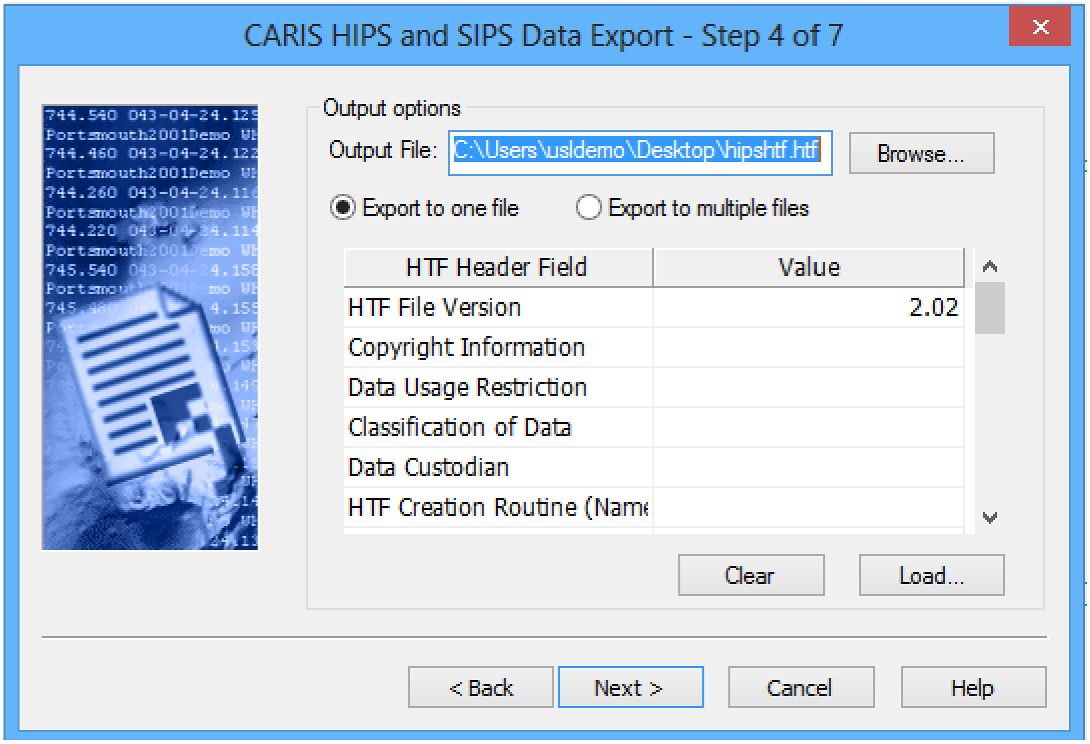
In the above Field Population Key, fields 5 to 9 contain data (hence the 1 bits), while fields 1 to 4 and 10 to 20 contain no data (hence the 0 bits). Fields without data are not written to the HTF file when it is generated. There are a total of 20 HTF fields in the sounding data record. Two additional fields are added by the Export Wizard for beam and profile number. Extra fields can be added, if needed. The HTF export automatically exports TPU depth and TPU position values if these are included in the track line.

The HTF file contains an ordered list of all possible fields and a description of each field.

Hydrographic Transfer Format Template

Sometimes the same header information can be used for multiple HTF files. The export wizard gives you the option of saving a header for use in another file. Headers are saved as HTFT files (Hydrographic Transfer Format Template). The file is in XML format that can be viewed in a text editor or an XML-compliant Internet browser.

HIPS to HTF - Output options

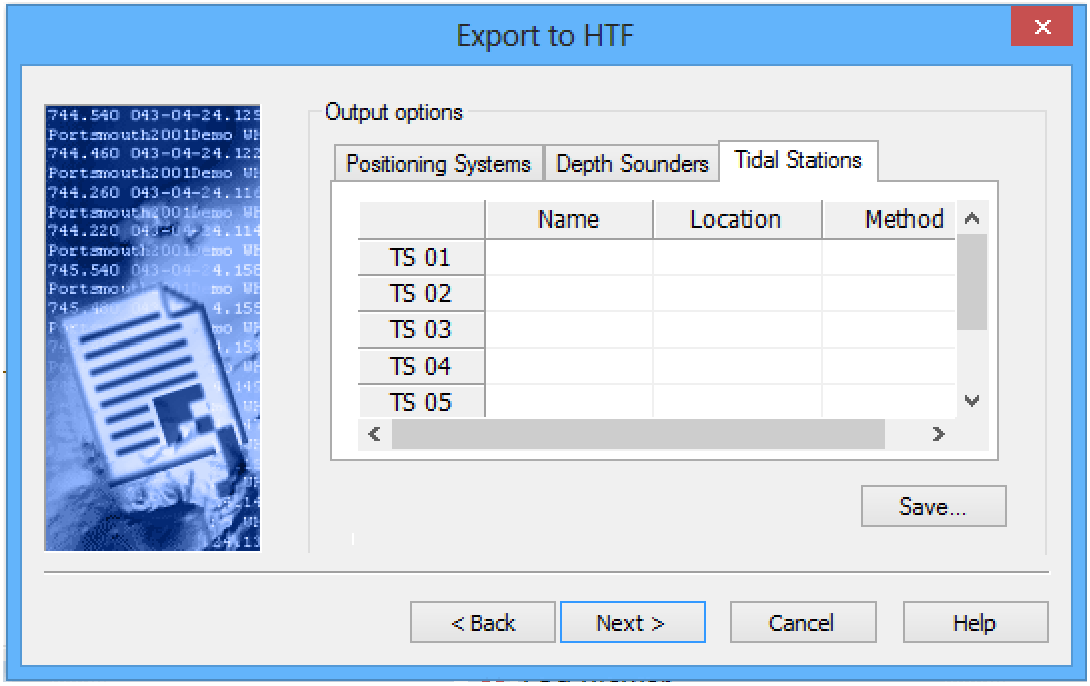


Enter the necessary header information. You can directly enter the data or load it from an HTFT (Hydrographic Transfer Format Template) file.

If you are exporting soundings, you can export all data to a single file or export data from each survey line to its own file.

1. Select an output location for the HTF file(s).
   - If you choose *Export to One File*, type a name and file path for the HTF file, or click **Browse** to select a location and name for the file.
   - If you choose *Export to Multiple Files*, click **Browse** to select a folder to save the HTF files. The files are automatically named according to Project_Vessel_Date_SurveyLine.
2. To enter new information, for example, the number of depth sounders or tidal stations, click in the *Value* field next to the appropriate *HTF Header Field* and type the data, or
3. [Optional] Click **Load** to open data from an existing HTFT file into the *Value* fields.
4. Click **Clear** to remove data from all fields, if needed.
5. Click **Next**.

HIPS to HTF - Output options page 2



1. Select the *Include Survey Line Name* check box to include this information in the HTF file.
2. Select the *Include Beam and Profile Information* check box to include the beam and profile numbers in the HTF file.

If you entered values in the depth sounder, positioning system, and tidal station fields in the previous dialog box then the respective fields in these tabs are enabled. If you did not enter values, click **Back** to fill in these fields.

The Positioning Systems tab is for entering navigation systems information.

3. Type the name and model (and/or any additional information) of the system in the *Name* field.
4. Type the system used on the survey (GPS, DGPS, etc.).

The Depth Sounders tab is for entering sonar information.

5. Type the name and model of the sonar system used on the survey.
6. Type the mode of sonar (multibeam, single beam, multi transducer).

The Tidal Stations tab is for entering tide information.

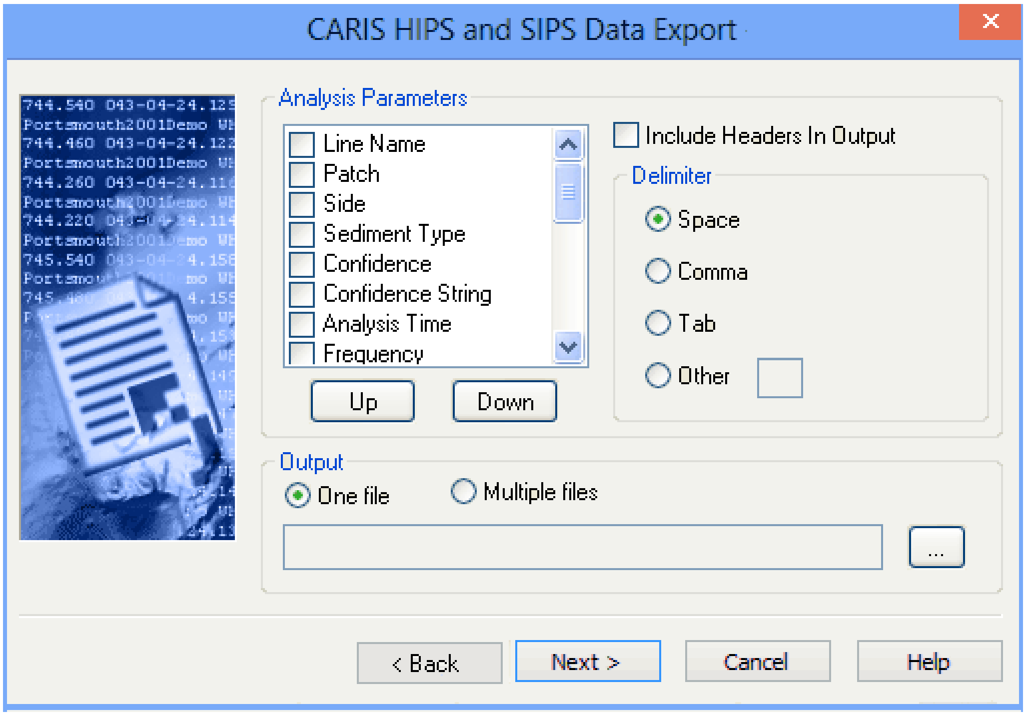
7. Type the name of a tidal station.
8. Type the location.
9. Type the method used to acquire data (predicted, observed, etc.)
10. Click **Save** to save information to a new or existing HTFT file.

11. Click **Next**.

"Export" on page 508

Sediment Analysis to ASCII

Export the results of sediment analysis. You can select which parameters to export, and the order they will be displayed, as well as the type of delimiter to be used in the ASCII file. Results can be exported from one line or multiple lines.



1. To include a parameter in the ASCII output, select its check box in the *Analysis Parameters* list. You can select more than one parameter at a time.
2. Use the **Move Up** and **Move Down** buttons to change the order in which the data will be shown in the output file.
3. If you want to include the parameter names as column headers in your output file, select the *Include Headers in Output* check box.
4. Select the type of delimiter you want to use to space the data in your output file. If you select *Other*, type the character you want to use.

If you are exporting the sediment analysis data from more than one line you can export the data to a single file or to a file for each line.

5. Select *One file* or *Multiple files*.
6. Click **Browse** [**...**] and select the destination folder for the output file(s).
7. Type the name of the file if output is going to a single file.
8. Click **Next**.

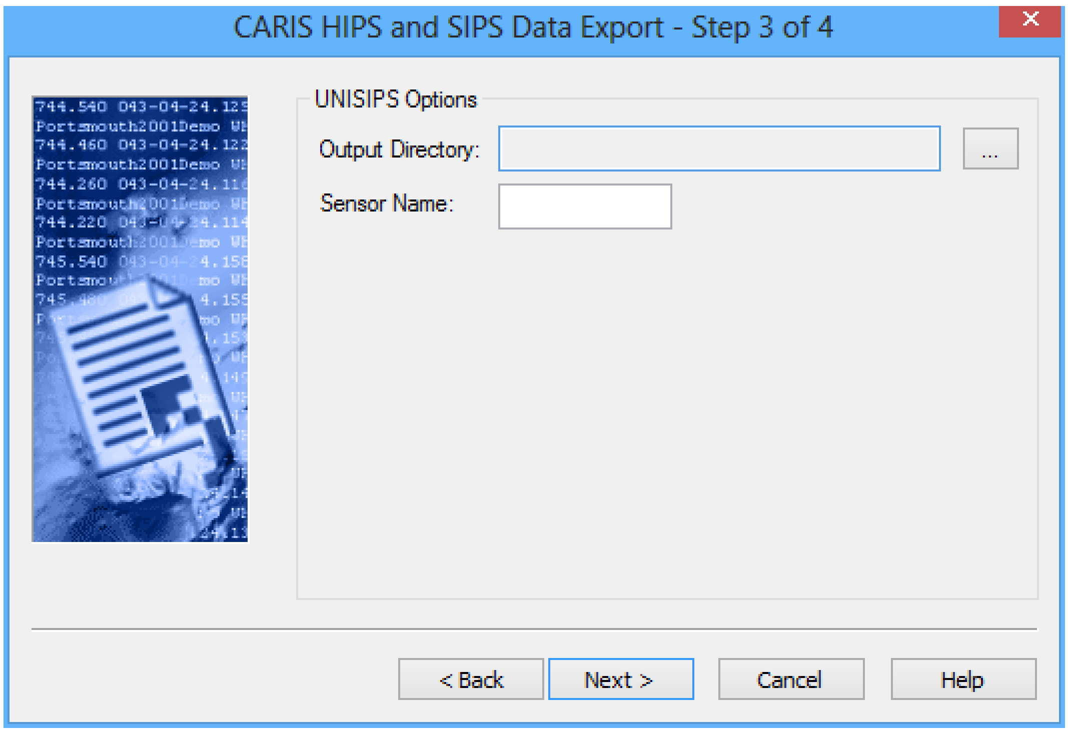
"EXPORT" ON PAGE 508

SIPS To UNISIPS

Export SIPS data to a Unified Sonar Imaging Processing System (UNISIPS) format.

SIPS TO UNISIPS (Step 3)

Select the location to store the output data.



1. Click **Browse** to select a location for the UNISIPS files.

The Browse for Folder dialog box is displayed.

2. Select a directory to store the UNISIPS data and click **OK** to close the dialog box. (The UNISIPS file will have the same name as the track line with an `.u` file extension.)
3. Type a *Sensor Name* in the field (up to 10 characters).

The file path is displayed in the *Output File* field.

4. Click **Next.**

“EXPORT” ON PAGE 508

Export a Surface

A surface in HIPS and SIPS can be exported to:

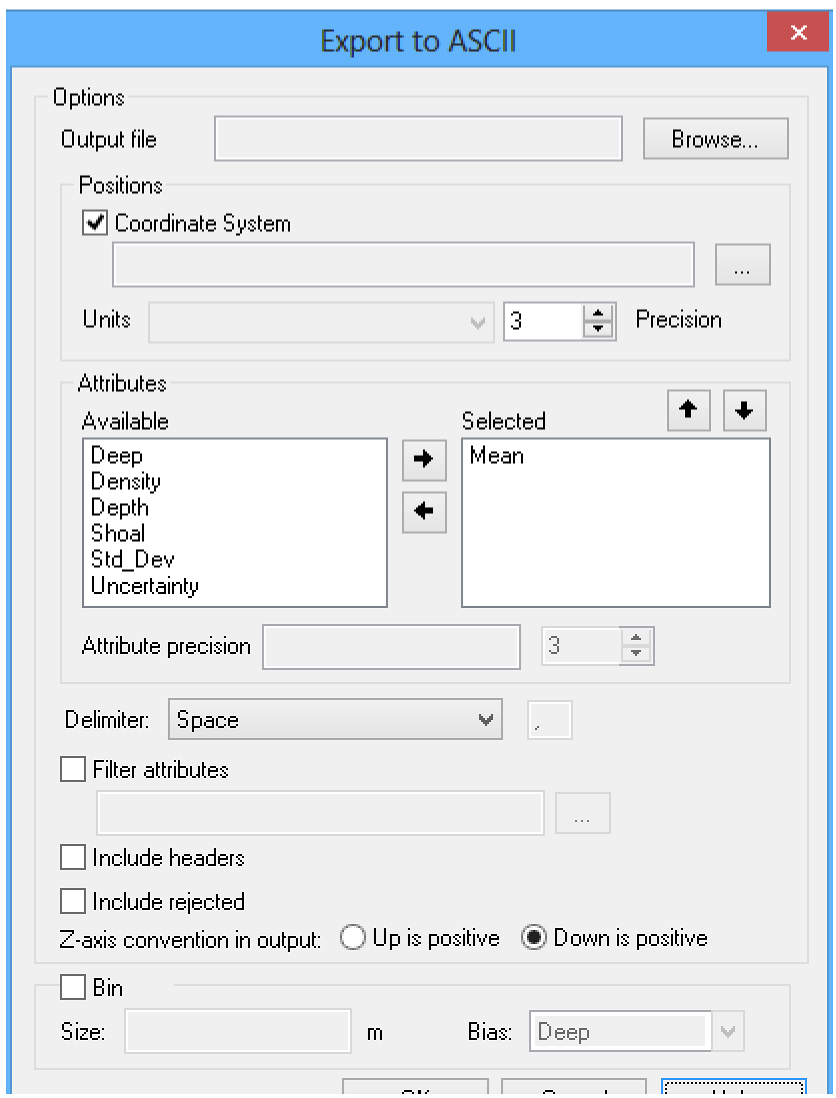
- ASCII text file, see “Surface to ASCII” on page 542
- point cloud, see “Surface to Point Cloud” on page 546
- HTF, see “Surface to HTF” on page 548
- XML (surface metadata),“Export Surface Metadata” on page 551
- raster product, see “Export Surface to Raster Product” on page 552
- STL (3D) format, see “Export Surface to STL” on page 562

Surface to ASCII

Menu: File > Export > Surface to ASCII

1. Select the surface in the Layers window.
2. Select the Export Surface to ASCII command.

The Export to ASCII dialog box is displayed.



Values entered in this dialog box that are not dependant on the current dataset (for example, Precision or Delimiter) will be remembered for future use.

1. Click **Browse** to go the destination folder for your export file.
2. Type the file name in the Save As dialog box, and click **Save**.

You can export using the coordinate system of the source data, or you can specify a different coordinate system. To use the system of the selected surface, leave the *Coordinate System* check box empty.

3. [Optional] Enable the Coordinate System check box, and **Browse** to select an alternate system from the Output Coordinate System dialog box.
4. Set the horizontal position units for data from the *Units* drop-down list:
    - Ground units (East, North): position displayed as Easting, Northing to the precision defined (for example:`360425.000, 4770355.000`)
    - Geographic (DMS): position displayed as Latitude, Longitude in degrees, minutes, seconds to the precision defined (for example: `43-04-23.03N, 070-42-51.89W`
    - Geographic (Unformatted DMS): displays the position without using hyphens or direction indicators, for example, `43 04 23.03, -070 42 51.89.`
    - Geographic (Decimal Degrees): position displayed as Latitude, Longitude, in decimal degrees to the precision set, for example, 43.073 -070.714.
    - Geographic (Degree Minutes): position displayed as Latitude, Longitude, in degrees and minutes, to the precision set, for example, 34-23.610N,119-52.585W.

For positions in the northern hemisphere, longitude West values are negative. In the southern hemisphere, the South latitude values are negative. For example, a position near Portsmouth NH, USA is `43 04 23.03, -070 42 51.89.` A position near Sydney Australia is `-33 51 00.84 151 11 43.7`

5. Specify the precision for the display of position units, by using the *Precision* up or down arrow button to set the number of digits to be displayed to the right of the decimal point. The maximum number is 9.

The surface layers that can be exported are listed in the Available attributes column. Any or all of these attributes can be exported at the same time. (See "Export Bounding Polygon to ASCII" on page 546 for information on exporting that layer.)

6. From the *Available* list, select each attribute you want to export.
7. Click the right arrow ➔ to move the attribute to the *Selected* list.
8. Click **Up** and **Down** arrows to move the attributes into the order you wish them to appear in the file.

All attributes in the *Selected* list will be included in the ASCII output.

When you highlight an attribute in the *Selected* list, the attribute name is displayed in the *Attribute Precision* field,

9. Set the *Attribute Precision* value (number of digits to the right of decimal point) for each attribute in the *Selected* list.

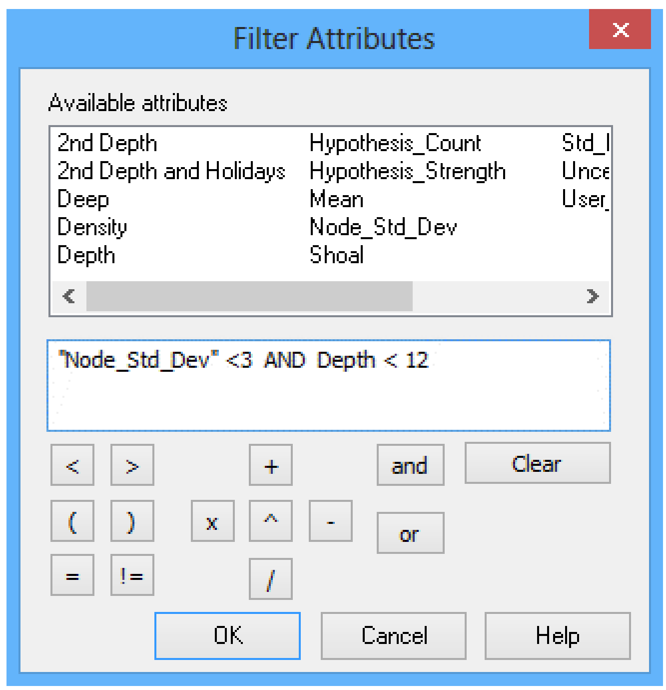
The values in the exported file will be separated by whichever character you set as a delimiter.

10. In the *Delimiter* field, select Space, Comma, or Tab from the list to use to separate fields in the output file. If you select Other, type an alphanumeric character to use.

The *Filter attributes* option enables you to create an equation that will filter out specific data from being included in the output file.

11. [Optional] Enable the Filter attributes check box.

12. Click Browse to open the Filter Attributes dialog box.



The *Available Attributes* field lists the attribute layers currently present in the selected surface.

Use a combination of attribute, operator and values to create a filtering equation:

13. .Double-click on an attribute in the *Available attributes* list to add it to the equation field. (If you type the name instead, be sure to copy exactly in order for the equation to be applied successfully.) .

If there are spaces in the name of the selected attribute, place quotation marks around the attribute in the conditions field, (e.g., “attribute name”).

14.Click an operator to add it to the equation:.

Operator	Function	Example
< >	Less than / greater than	Depth < 1
( )	bracketed operation	(Depth+4.5) +
= !=	equals, does not equal	Depth != 45
+	addition	Depth + 4.7
x	multiplication	Depth * 4
–	subtraction	Depth - 4
\	division	Depth / 4
^	exponentiation (Depth^{X})	Depth ^ 4
and	logical operator	AND
or	logical operator	OR

15.Click **OK** to close the dialog box and display the equation in the Export to ASCII dialog box.

16.[Optional] Enable the *Include Headers* check box to have the appropriate column headers included in the ASCII file.

When exporting a CSAR point cloud, by default soundings that are flagged as rejected are not included in the exported file to ensure that a clean surface can be created from the exported data.

17.[Optional] Enable the *Include Rejected* check box to include rejected soundings in the exported file.

The Z-axis convention controls the direction of vertical values in data. The HIPS and SIPS default setting is *Down is positive*. To change the convention setting for exported Z values:

18.[Optional] Select *Up is positive*.

The Bin option enables you to define the range of each grouping of data that will be displayed as a row in the file. Binning divides the surface into squares or “bins” based on the value set in the Size field. Within each bin, the shallowest or deepest value will be exported, depending on which bias you select. By default the bias is set to Deep.

To use the Bin option on the exported data:

19.. Enable the *Bin* check box.

20.In the *Size* field, type a value in metres for the size of a bin.

21.[Optional] Set the *Bias* to Shoal.

22.Click **OK**.

The example below shows a comma-delimited ASCII file for the Depth attribute of a surface with the positions in decimal degrees, and column headers turned on.

```
//Lat (DD) Long (DD)Depth  Mean Shoal
43.073, -070.714,  5.989, 5.99, 5.895
43.073, -070.714,  6.459, 6.46, 6.045
43.073, -070.714,  6.906, 6.93, 6.490
43.073, -070.714,  7.443, 7.44, 6.936
43.073, -070.714,  8.002, 8.00, 7.491
43.073, -070.714,  6.628, 6.63, 6.130
```

Export Bounding Polygon to ASCII

The coordinates of the bounding polygon layer of a surface can be exported to an ASCII text file.

To export bounding polygon to ASCII:

1. Select the Bounding polygon layer in the Layers window.
2. In the Display window, range-select polygon.

The coordinates of the polygon will be displayed in the Coordinates window.

3. Right-click in the Coordinates window, and select Save All from the pop-up menu.
4. In the Save As dialog box, enter a name and location for the file.
5. Click **Save**.

The coordinates will be saved to the ASCII file at the selected location.

Surface to Point Cloud

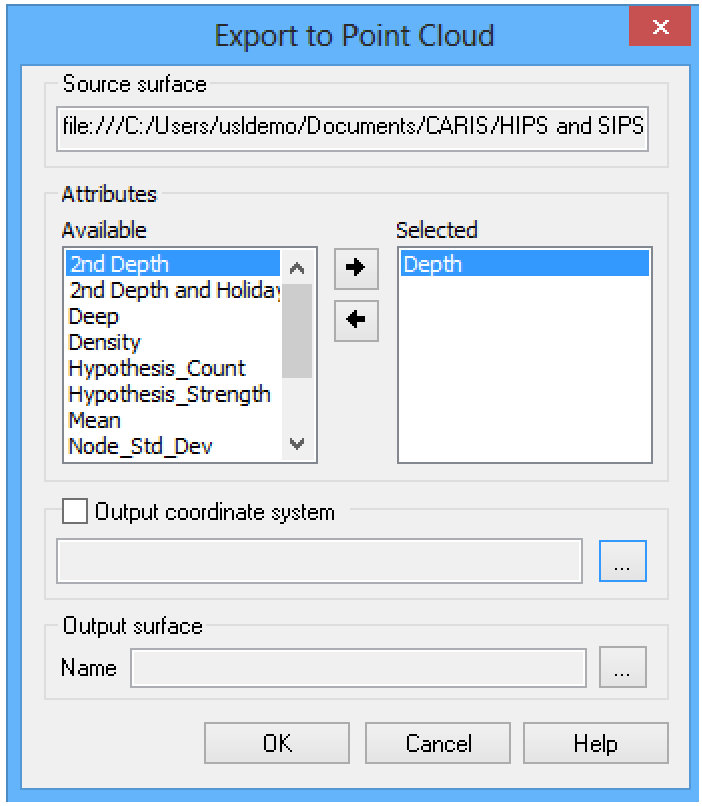
A point cloud is a set of vertices in a three-dimensional coordinate system. In HIPS and SIPS these vertices are usually defined by X, Y, and Z coordinates, and represent data in 3-D display

To export a surface to a point cloud:

1. Select the surface in the Layers window.
2. Select the Surface to Point Cloud command.

Menu	File > Export > Surface to Point Cloud

The Export to Point Cloud dialog box is displayed.



The *Source surface* field displays the path to and name of the selected surface. This cannot be changed.

The surface attributes are listed in the *Attributes* fields. The primary Z attribute layer is displayed in the *Selected* field. This cannot be removed from the list.

3. From the *Available* list, select each attribute you want to export.
4. Click the right arrow to move attributes to the *Selected* list.

The coordinate system used by default in the export is that of the source data. You can specify a different coordinate system. To use the system of the selected surface, leave the *Coordinate System* check box empty.

5. [Optional] Enable the Coordinate System check box, and **Browse** to select an alternate system from the Output Coordinate System dialog box.
6. Click **Browse** to go the destination folder for your export file.
7. Type the file name in the Save As dialog box, and click **Save**.
8. Click **OK** to export the surface.

Surface to HTF

See "HIPS To HTF" on page 536 for description of HTF format.

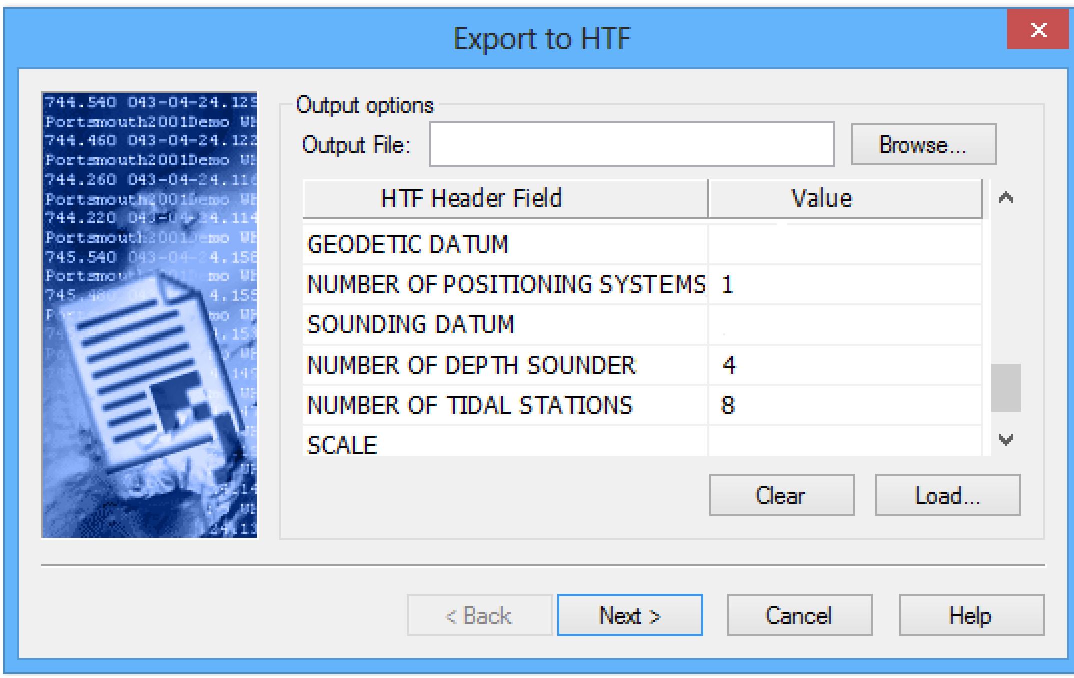
Export a surface to Hydrographic Transfer Format (HTF).

To export to HTF:

Menu: File > Export > Surface to HTF

1. Select a surface in the Layers window.
2. Select the Surface to HTF command.

The first Export to HTF dialog box is displayed.



3. Type a name in the Output field, or click Browse and enter a name and location.

If you do not select a directory, the HTF file will be saved to the last one used.

The HTF Header Field and Value fields are used to define values for each of the lines in the header of the exported HTF file. You can directly enter values for these header fields or load them from a Hydrographic Transfer Format Template (HTFT) file.

Any value fields left blank will be populated with an asterisk in the output file.

4. Type a value for the fields you want to include in the exported file.

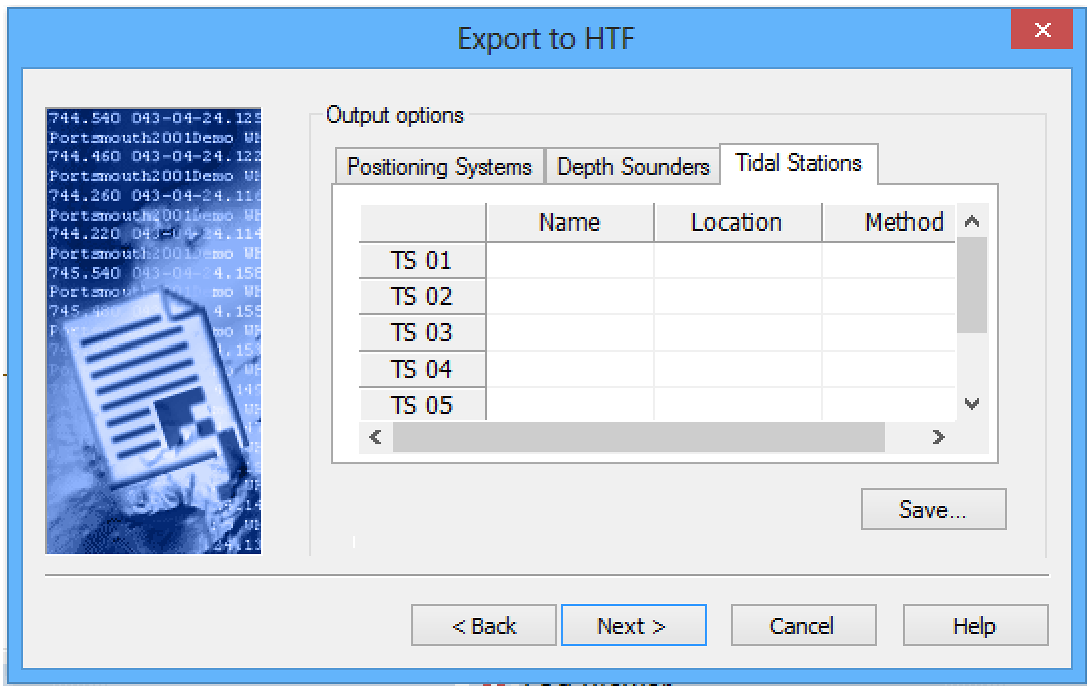
   OR

5. Click **Load** and select an HTF template file from which to load values for the header settings.
6. Click **Clear** to remove all entered values.

7. Click **Next**. "Output Options - Step 2" on page 549

Output Options - Step 2

This Output Options dialog box is used to define positioning systems, depth sounders and tidal stations.



The fields in this dialog box display the values entered in the *Number of Positioning Systems*, *Number of Depth Sounders* or *Number of Tidal Stations* fields in the previous dialog box.

If you did not enter values, click **Back** to return to the previous step to fill in this data.

The Positioning Systems tab contains navigation system information.

8. Select the Positioning Systems tab.
9. Enter the name and model (and/or any additional information) of the system in the *Name* field.
10. Enter the system used on the survey (GPS, DGPS, etc.).

The Depth Sounders tab contains sonar information.

11. Select the Depth Sounders tab.
12. In the Name field, type the name and model of the sonar system used on the survey.
13. In the Mode field, type the mode of sonar, for example multibeam, single beam or multi-transducer.

The Tidal Stations tab lists tide information.

14. In the *Name* field, type the name of each tidal station.

15. .In the Location field, type the location of the station.

16. In the *Method* field, type the method used to acquire the tidal data, such as predicted or observed.

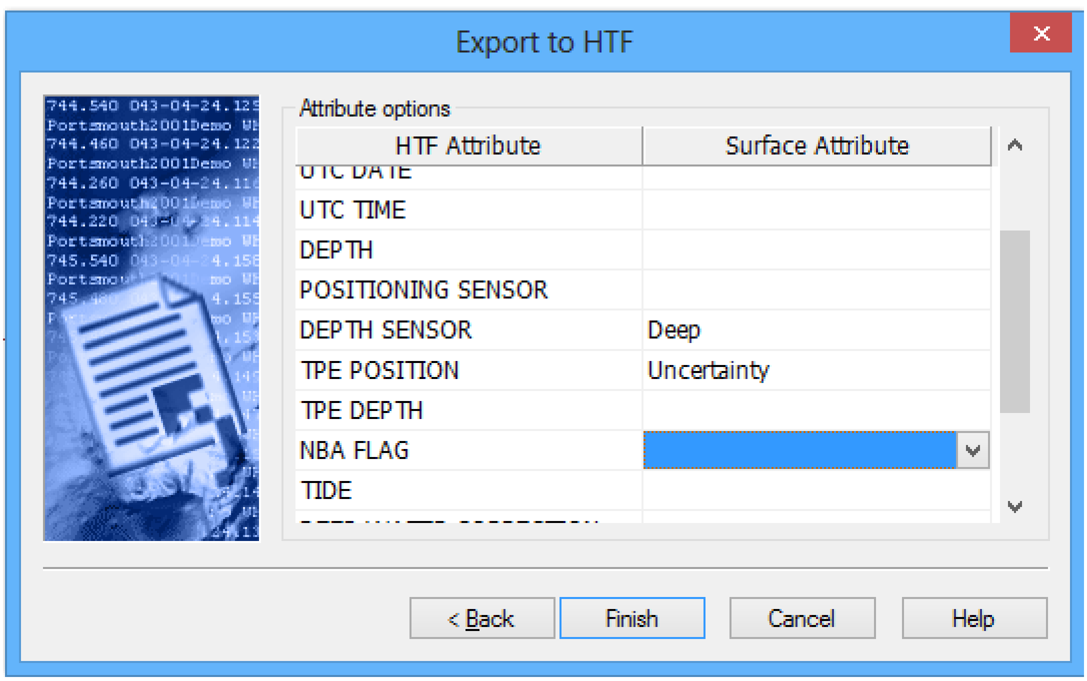
If the same header information will be used for multiple HTF files, you can save the header for future use. Headers are saved as HTFT files (Hydrographic Transfer Format Template). The file is in XML format that can be viewed in a text editor or an XML-compliant Internet browser.

17. [Optional] To save your changes to a template file, click **Save**.

18. Click **Next**. "Export To HTF: Attribute Options" on page 550

Export To HTF: Attribute Options

Use the dialog box to map surface attributes to each of the HTF attributes.



1. Select an HTF attribute from the list.
2. Click in the corresponding Surface Attribute field and select a surface attribute by name from the list.
3. Repeat as necessary for all the HTF attributes.
4. Click *Finish* to export the surface to an HTF file.

Export Surface Metadata

Save the metadata from a CSAR surface to an ISO 19115 / 19139 compliant metadata file. Metadata can be viewed in the Properties for the selected surface.

See "VIEW SURFACE PROPERTIES" ON PAGE 196.

The metadata file is saved in XML format.

1. Select the surface in the Layers window.

Menu	File > Export > Surface Metadata

2. Select File> Export > Surface Metadata command.

The Save As dialog box is displayed.

A default file name is displayed, created from the name of the selected surface with the word "metadata" appended to it, as in: `FUNDY421_metadata.xml`.

3. [Optional] Type a name for the metadata file.

The default location for the exported file is `\CARIS\ HIPS\ version \Session`.

4. Click **Save**.

Export Surface to Raster Product

Use the Raster Product Export tool to create image products from surfaces. The Raster Product Export tool is both an export tool and a template management tool.

It is an export tool in that you can export a raster surface and its metadata to a number of different formats. The output can be a raster image in 24-bit GeoTIFF format that records the geo-referencing information within the file, or any of the following raster DEM formats:

- BAG
- ESRI ASCII
- GeoTIFF
- USGS DEM

Metadata files can also be exported as part of a raster surface product.

It is a template management tool in that you can create product templates for exporting raster products and then perform exports using the templates. Products include any combination of the available output formats, as well as metadata and settings for clipping and tiling.

To export a surface layer to a raster product format, see “Export a product” on page 553

To create a template that can be used to export a surface to multiple products, see “Create a product template” on page 558.

To edit an existing product template, see “Edit a Template” on page 560

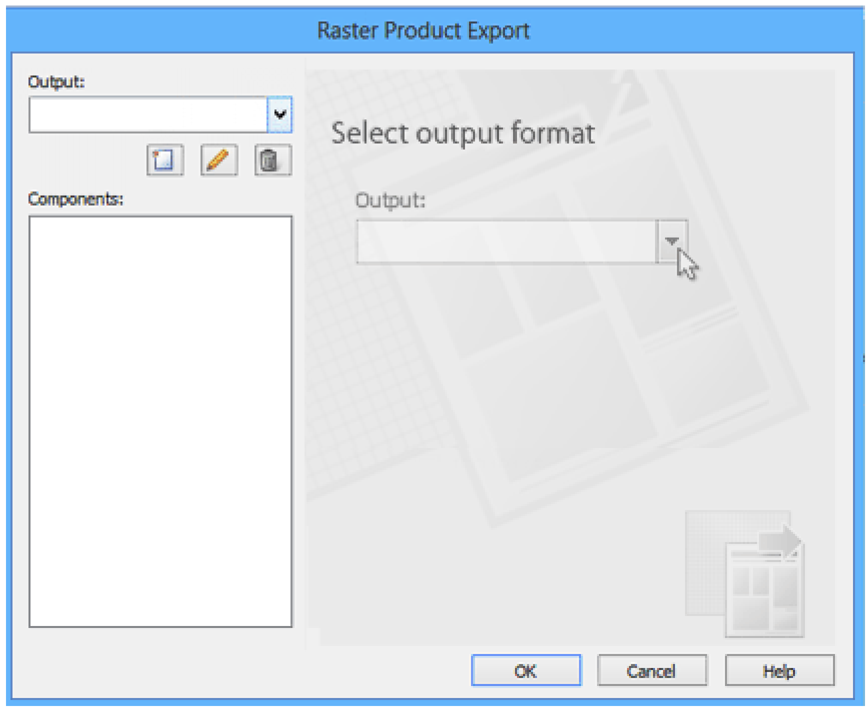
Export a product

Menu | File > Export > Raster Product

To export a surface to a raster product:

1. Select a surface in the Layers window.
2. Select the Raster Product command.

The Raster Product Export dialog box is displayed.

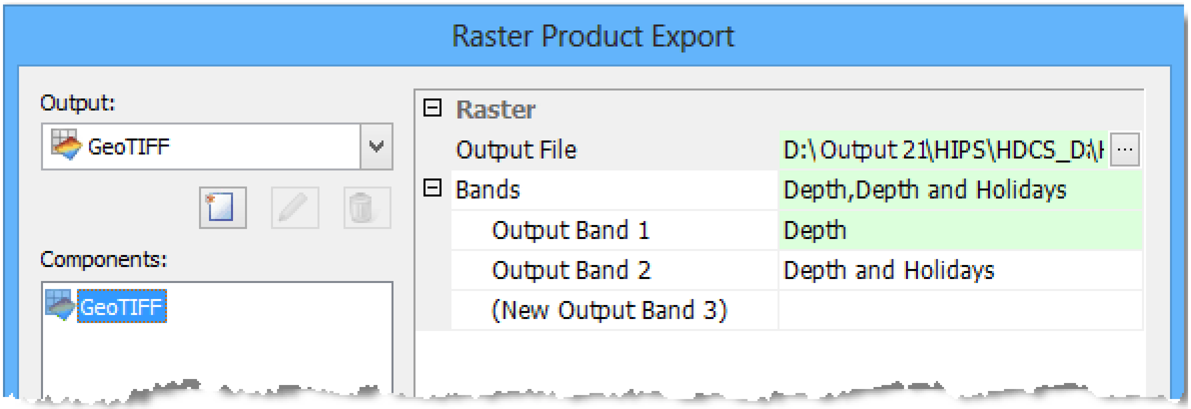


3. Select a format from the Output drop-down list.

The dialog box now displays the properties fields for the selected format. The available properties will differ based on the selected component.

Properties highlighted in red are mandatory. The highlighting will change to green once a value is entered. Attempting to complete the export without defining these properties will cause a warning message to be displayed.

The image below shows the fields for GeoTIFF output.



4. Define the properties for the output file. See “Format Properties” on page 554.
5. Click OK to export the surface.

The Output window will report start and end times for each format exported. If any selected format cannot be exported, that too will be reported in the Output window

The product output will be saved to the location specified in the properties.

To export a surface to more than one product format simultaneously, create a product template. See “Create a product template” on page 558.

Format Properties

The table below describes the properties for each format.

Component	Property		Description
BAG	Output File		The file name to assign to the file(s) output for this format.
	Bands	Elevation	The attribute band in the source surface containing the elevation values (Depth, Shoal, etc.) to be exported.
		Uncertainty	The attribute band in the source surface that contains uncertainty data to be included in the export.
	Options	Compression Level	The level of compression to apply to the resulting BAG file(s). The valid range is 0 to 9, with 0 being no compression and 9 being highly compressed.
ESRI ASCII	Output File		The file name to assign to the file(s) output for this format.
	Bands	Output	The attribute band to export from the source surface.
	Decimal Precision		The number of decimal places to include in elevation values in the output.

Component	Property		Description
GeoTIFF DEM	Output File		The file name to assign to the file(s) output for this format.
	Bands	*Output #*	The attribute band to export from the source surface. Multiple attributes can be exported if desired. As each attribute is selected, another field will be added to specify additional attributes.
USGS DEM[a]	Output File		The file name to assign to the file(s) output for this format.
	Bands	Output	The attribute band to export from the source surface.
	Decimal Precision		The number of decimal places to include in elevation values in the output.
GeoTIFF Image	Output File		The file name to assign to the file(s) output for this format.
	Bands	Output	The attribute band to export from the source surface.
	Colour Map		The colour map file to apply to the resulting image. The data will be coloured according to depth values.

Component	Property		Description
BAG Metadata	Bands	Uncertainty Type	The type of uncertainty data being exported. Select an option from the drop-down list. If you do not have uncertainty data, select *Unknown*.
	General	Abstract	A brief description of the data being exported. Enter any necessary information in the field.
		Status	The status of the data at the time of export. The options include: • completed • historicalArchive • obsolete • onGoing • planned • required • underDevelopment Select an option from the drop-down list.
		Vertical Datum	The vertical datum to which the data is referenced. Enter a value in the field.
	Responsible Party	Name	The name of the person responsible for the data in the resulting file. Enter a name in the field.
		Position	The position of the person responsible for the data in the resulting file. Enter a position in the field.
		Organization	The organization responsible for the data in the resulting file. Enter a name in the field.
		Role	The role of the person responsible for the data in the resulting file. This field should be defined according to tasks that the user will be performing with the file. Select an option from the drop-down list.
	Constraints	Legal	The legal constraints to apply to the file to define whether it is legally restricted from being copied, altered, or referenced without permission from the responsible party. Select an option from the drop-down list.
		Other	Additional restrictions that are not listed under the *Legal* field. This field is only enabled if *otherRestrictions* is selected in the *Legal* list. Enter information in the field.
		Security	The security classification level to apply to the file for access restriction. Select an option from the drop-down list.
		Notes	Additional information about the constraints on the data. Enter any necessary information in the field.
ISO19115	Output File		The name to assign to the output file for the ISO19115 metadata for the source surface.

<table>
<tr><th>Component</th><th colspan="2">Property</th><th>Description</th></tr>
<tr><td rowspan="5">Clipping</td><td colspan="2">Unit Type</td><td>The unit type in which values are to be entered when defining dimensions for clipping.

The format and precision of the clipping values are controlled by the settings in the Display Units tab in Options.
• For the Origin values, the Coordinate options will be applied based on the selected Unit type.
• For the Width and Height values:
  • the Geographic Format and Geographic Precision options will be used for the geographic unit type.
  • the Ground Units and Ground Precision options will be used for the ground unit type.

See “Display Units” on page 630 for more information.

A different unit of measure can be specified when entering values, however, it will automatically be converted to the settings in Options. The units specified must match the selected unit type. For example, you cannot enter geographic values if the Unit type is set to Ground. If the Unit type is set to Geographic, the width and height must be specified in degrees.</td></tr>
<tr><td rowspan="4">Clipping Window</td><td>X Origin</td><td>The X coordinate of the lower-left of the data to be exported.</td></tr>
<tr><td>Y Origin</td><td>The Y coordinate of the lower-left of the data to be exported.</td></tr>
<tr><td>Width</td><td>The horizontal distance from the X Origin point to include in the area of the surface to be exported.</td></tr>
<tr><td>Height</td><td>The vertical distance from the Y Origin point to include in the area of the surface to be exported.</td></tr>
<tr><td rowspan="3">Tiling</td><td colspan="2">Unit Type</td><td>The unit type in which values are to be entered when defining dimensions for tiling.

The format and precision of the values when exported will be controlled by the Distance settings in the Display Units tab in Options.
• For Geographic, the Geographic Format and Geographic Precision settings will be applied.
• For Ground, the Ground Units and Ground Precision options will be applied.

See “Display Units Tab” on page 730 for more information.

The same restrictions and conditions that apply to clipping values also apply to tiling values.</td></tr>
<tr><td rowspan="2">Tile</td><td>Width</td><td>The width to include in each tile if exporting to tiles.</td></tr>
<tr><td>Height</td><td>The height to include in each tile if exporting to tiles.</td></tr>
</table>

a The USGS format supports only Geographic and UTM coordinate systems and a limited number of horizontal datums (NAD27, NAD83, WGS72, and WGS84). If the selected surface references an unsupported coordinate system or datum, the export will fail. UTM data must be in the northern hemisphere to be valid and export successfully to USGS DEM format.

Create a product template

You can create a new product template using the settings of an existing product template, or start from a blank template and customize your own.

All templates and their settings are saved to the rasterproducttemplate.xml found in: `C:\Program Data\CARIS\HIPS\<version>\System`.

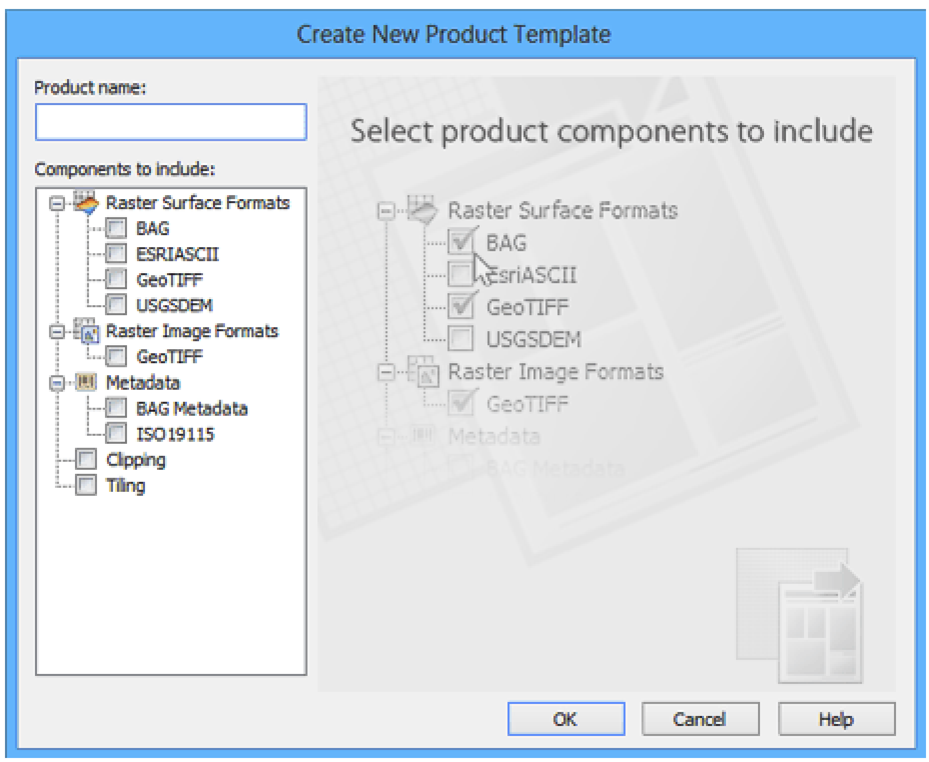
When creating a new product template, HIPS and SIPS will check this file to determine if there are existing product templates. If templates are found, the Reference Template dialog box is displayed which lists the names of the existing templates.

To create a product template in the Raster Product Export tool:

Tool 

1. Click the New Product Template tool button.

The Create New Product Template dialog box is displayed.



2. Enter a Product name for the template.

Any or all of the listed output components can be included in a product template. The available components are:

- **Raster Surface** and **Raster Image** formats
- **BAG Metadata** and **ISO19115** which define what metadata is exported to a file.

- **Clipping**, which exports only a portion of the source data.
- **Tiling**, which exports the source data as multiple tiles of a user-specified size, rather than a single large file. See "Clipping and Tiling" on page 559

> At least one raster format component must be included in the template when Clipping and Tiling are selected.

3. Select a component so it is highlighted in the *Components to include* list.

The available properties for that component will be displayed on the right side of the dialog box.

4. Click the check box so that the component will be included in the template.
5. Define the properties for the output file. See "Format Properties" on page 554.

Values defined here will be saved as default values for the product template.

6. Select Ground or Geographic from the *Unit type* list for the Clipping and/ or Tiling components.
7. Click **OK** to create the product template.

The template is added to the rasterproducttemplate.xml file in the system folder. The Raster Product Export dialog box now displays the name of the new template in the *Output* drop-down list.

Clipping and Tiling

If Clipping and Tiling components are included in a template, they will be applied to all output formats generated by an export of the product template. For example, if you export both a raster surface and a raster image, the output for each format will be clipped and/or tiled if these components are included in the product template.

You must specify a *Unit Type* for the Clipping and Tiling components. This controls the format that can be used when entering values for these properties. The same units should be used for both components. If data being exported with the product template does not use the same units, the export will not complete successfully

Clipping

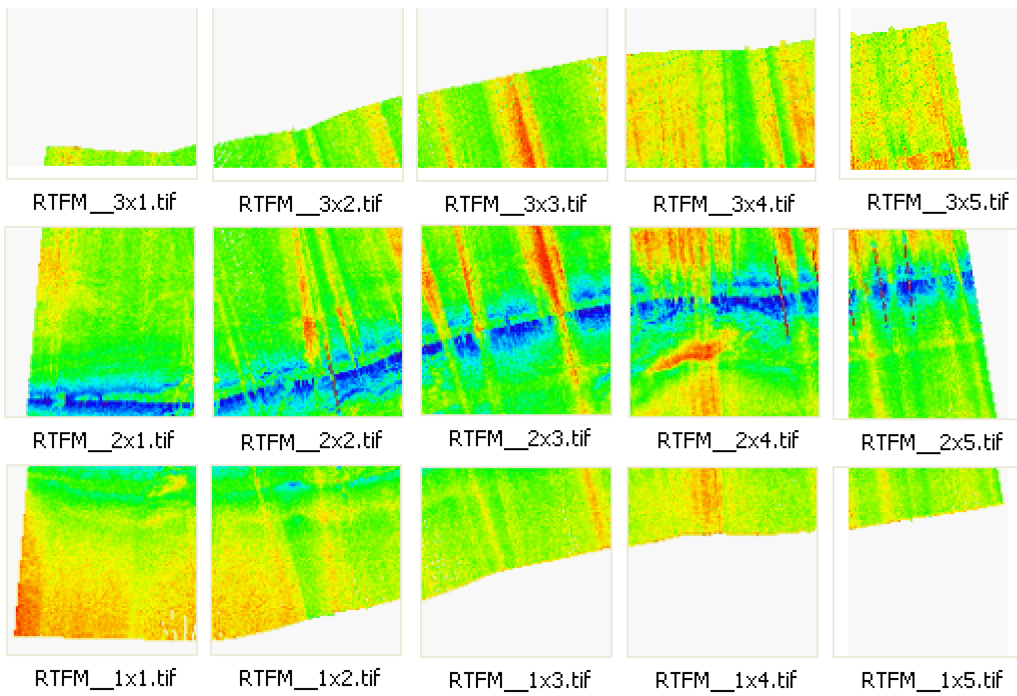
The Clipping export portion is defined by specifying coordinates for the X and Y origin points in the surface and the width and height distances to include from those points. The units are controlled by the Distance settings in the Display Units Options (see "Display Units" on page 160).

Tiling

The Tiling option creates a separate raster for each tile in the surface. The tiles are determined based on user-specified width and height dimensions. These units are also controlled by the Distance settings in the Display Units Options. See "Display Units" on page 160.

When creating a tile set, each file will be numbered according to the location of the tile in the surface. The tile number is in the format `"RxC"`, where R and C are the row and column index of the tile. The tile number will be included in the file name for each tile, along with the Output name assigned for the export format.

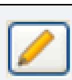
For example, If you were exporting a GeoTIFF image to tiles with `RTFM` set as the *Output* name, the resulting files would be numbered as follows:



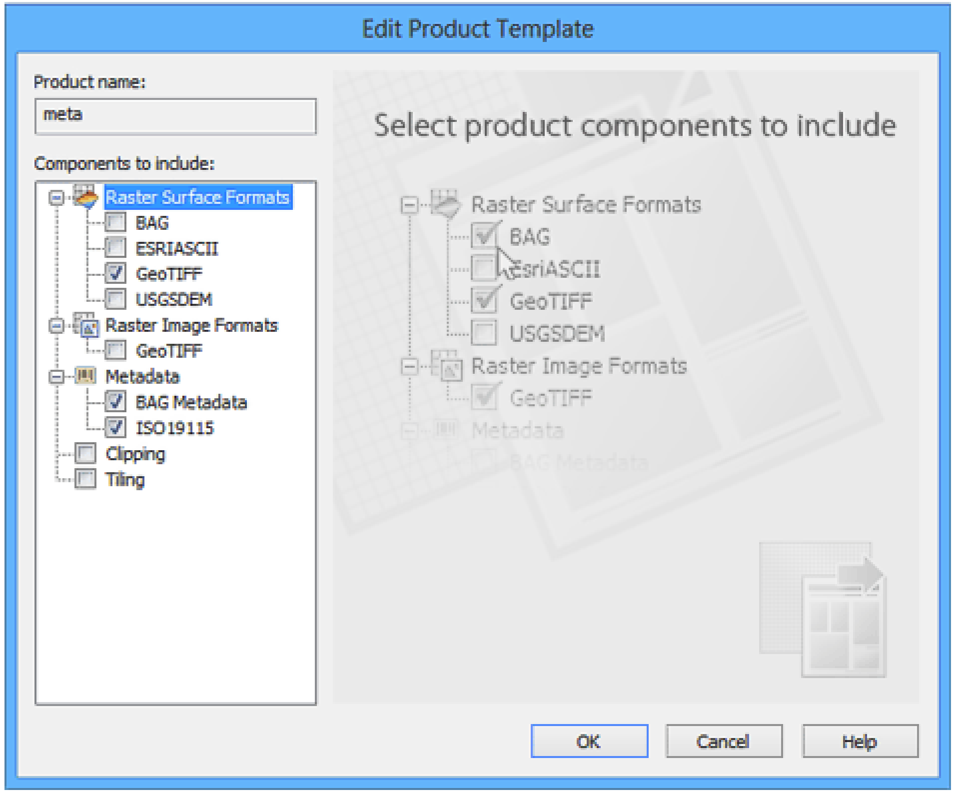
Edit a Template

Product templates can be edited. To edit a product template:

1. Select the product template from the Products group in the *Output* drop-down list.
2. Click the Edit Product Template tool button.

Tool 

The Edit Product Template dialog box is displayed. The right side of the dialog box will not display any fields until a component is selected in the *Components to include* list.



The name of the product template cannot be changed, but components can be added or removed and properties can be changed, added or removed.

3. [Optional] Change the components included in the template.
4. [Optional] Select each checked component and change its property values as needed.
5. Click **OK**.

The changes are saved to the rasterproducttemplate.xml file.

Delete a Template

If a product template is no longer needed, it can be deleted.

1. Select the desired product template from the *Products* group in the *Output* drop-down list.

Tool 

2. Click the **Delete Product Template** button.

A confirmation message is displayed to ensure you want to permanently delete the product template.

3. Click **Yes** to complete the deletion.

The product template is removed from the rasterproducttemplate.xml file and is no longer displayed as an option in the *Output* list.

Export Surface to STL

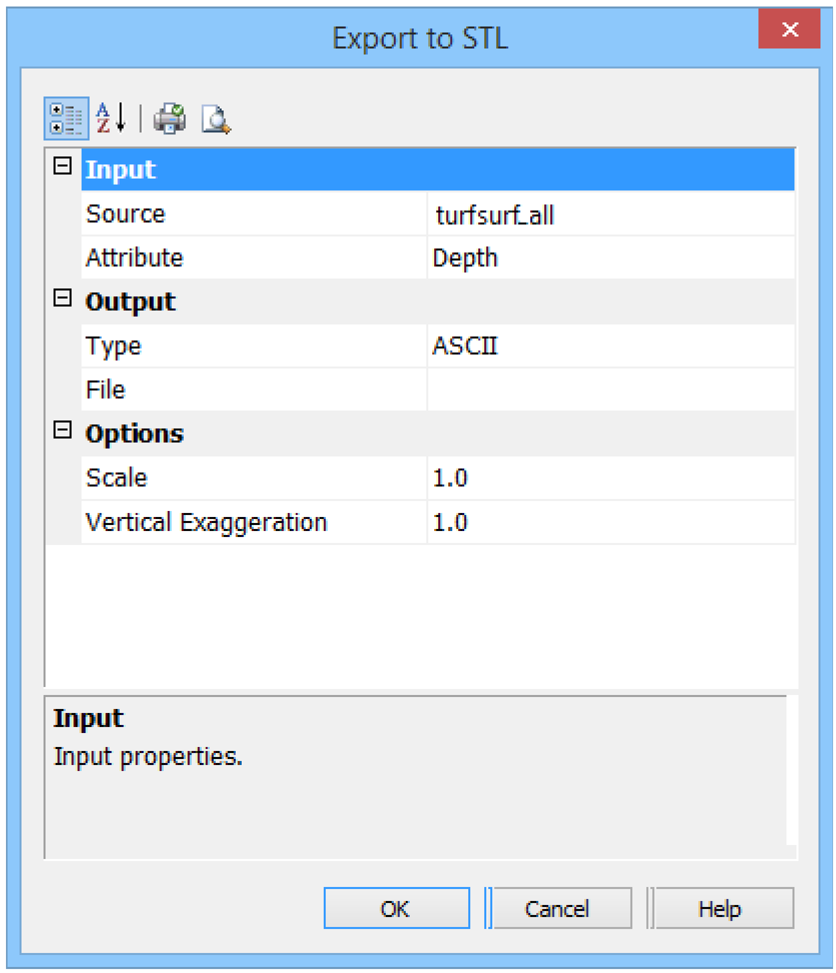
Export a surface to STL, an interchange format for 3D printers that can be used to create a three-dimensional model in the real world.

To export to STL:

Menu	File > Export > Surface to STL

1. Select a surface in the Layers window.
2. Select the Surface to STL command.

The Export to STL dialog box is displayed.



1. Enter appropriate values as follows:

Field name	Function
Input	
Source	Select the surface by name from the list, if it is not already displayed in the field.
Attribute	Select the attribute layer to export.

Field name	Function
Output	
Type	Choose either ASCII or Binary as the output format for the STL files.
File	Browse to set the file name and location for the output STL file.
Options	
Scale	Set a factor to apply to the input surface to scale the coordinates to printable 3D scale. Increasing this value will change the size of the printed model.
Vertical Exaggeration	Exaggeration factor to apply to the scaled vertical height.

Export a Selection

Selected features such as track lines, soundings, and contacts can be exported to these formats:

- ASCII text file
- CARIS Map
- CARIS Well-Known Text (WKT)
- DXF file
- GML file
- KML file
- HOB file
- S-57 file
- Shape file

Export Selection to ASCII text file

Export information about selected features such as track lines, soundings, and contacts to an ASCII text file. If the selected features are points or soundings, their coordinates are exported. If the selected features are lines or areas, their coordinates are not exported.

Data is exported as rows and columns. You can define these using delimited data, or tabular data.

Delimited Data

In delimited data, columns of data are separated by a single delimiting character. The default delimiter is a semi-colon. In addition to the semi-colon, you can use underscores, spaces, commas, colons and any alpha-numeric character as a delimiter.

> Punctuation marks are usually the better choice because letters and numbers can be difficult to distinguish from the data.

If a selected feature does not have a value for a particular attribute column, delimiting characters will still be present for the column, but there will not be a value between the characters.

The following is an example of an ASCII file with delimited data, using a semi-colon as the separator.

crit2asciiEX.txt - Notepad

File Edit Format View Help

```
X;Y;Project;Vessel;Day;Line;Profile;Beam;Depth (m);Status
360507.177;4770509.958;Portsmouth2001;samplevessel;2000-307;235_2017;968;100;10.375;Designated
360507.138;4770510.366;Portsmouth2001;samplevessel;2000-307;235_2017;968;101;9.136;Designated
360503.475;4770517.895;Portsmouth2001;samplevessel;2000-307;235_2017;973;100;12.757;Designated
360503.413;4770518.310;Portsmouth2001;samplevessel;2000-307;235_2017;973;101;11.295;Designated
360502.460;4770508.622;Portsmouth2001;samplevessel;2000-307;235_2017;975;99;11.556;Designated
360502.406;4770508.961;Portsmouth2001;samplevessel;2000-307;235_2017;975;100;10.318;Designated
360502.361;4770509.076;Portsmouth2001;samplevessel;2000-307;235_2017;975;101;9.032;Designated
360494.518;4770525.099;Portsmouth2001;samplevessel;2000-307;235_2017;985;101;13.017;Designated
360494.327;4770515.784;Portsmouth2001;samplevessel;2000-307;235_2017;987;101;10.657;Designated
```

Tabular Data

Data exported in tabular format is in columns that are aligned by padding the header and value fields with a filler character. The default filler is a space.

The width of the columns is determined by comparing the number of digits in the header and its value field. For example, the “Type” column may list “Points” as a value. In this case, two filler characters would be added to the column heading to make it as long as the value. The use of filler characters to align the columns can be seen in the illustration below.

The columns are aligned using the # character as filler to both headers and values.

#defaul.txt - Notepad

File Edit Format View Help

```
Project######Vessel####Max Time###############Merged#SR Corrected#Outdated#
Ptsmouth2001#samplevel#2000-11-02 20:42:48.030#Yes####Yes##########No#######
Ptsmouth2001#samplevel#2000-11-02 20:34:49.382#Yes####Yes##########No#######
Ptsmouth2001#samplevel#2000-11-05 18:48:26.103#Yes####No###########No#######
```

Indicates the beginning of a column

If there is no value for a particular attribute column, filler characters will be used as a placeholder for the required number of characters in the column.

Export Process

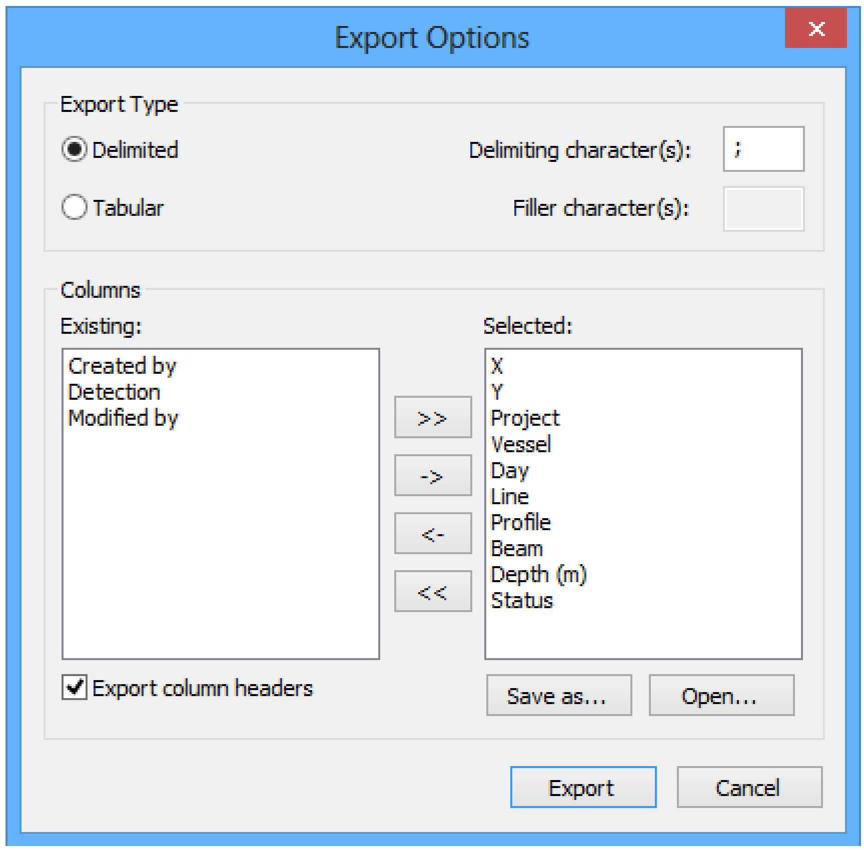
To export a selection of features to a delimited ASCII file:

1. Select a feature layer in the Layers window, for example, Critical Soundings.
2. Select features.

Menu	File > Export > Selection to ASCII File

3. Select the (Export) Selection to ASCII File command.

The Export Options dialog box is displayed.



You can export the selection as delimited data or as tabular data.

4. Select an *Export Type*.
5. Depending on the export type, you must specify either a *Delimiting character* or a *Filler character*.

6. Select the columns that will be exported so that they are listed in the Selected list. Use the buttons to move attributes between columns, as follows:

To export a column:

1. Select the column in the list on the left.
2. Click [->].

The column is added to the *Selected* list.

To remove a column from the *Selected* list:

1. Select the column in the list.
2. Click [<-].

The column is added to the *Existing* list.

To export all columns, click [>>].

To remove all columns from the Selected list, click [<<].

By default, column headers will be included in the output file. The headers use the names of the exported attributes listed in the Selection column.

7. [Optional] Disable the *Export column headers* check box so that no headers are included in the output file.

The Selected list of attributes can be saved to an `.xml` file and loaded into the dialog box for future exports. This saves having to re-select the attributes if the same attributes are to be used for multiple exports.

8. [Optional] To save the attributes list, click **Save As** then assign a name and location to the file.
9. [Optional] To load a previously saved list, click **Open** and navigate to the file.
10. To generate the file, click **Export**.

A Save As dialog box is displayed.

11. Specify a path and file name for the file.
12. Click **Save**.

The *.txt file is created in the specified location. The settings used for the export are saved to the applications settings and will be displayed the next time the Export Options dialog box is opened.

Export Selection to CARIS Map

Save selected features as CARIS map (.des) file.

1. Select a feature layer in the Layers window.
2. Select one or more features to export.

Menu	File > Export > Selection to CARIS Map

3. Select the Export Selection to CARIS map command.

A Save As dialog box is displayed.

4. Type a name and select a location for the exported file.
5. Click **Save**.

A CARIS Map file is created in the selected location containing the selected features.

Export Selection to CARIS Well-Known Text (WKT)

Export selected soundings and lines to formatted ASCII. The exported files will have the extension *.WKT (well-known text).

To export a selection, for example, of lines to a formatted ASCII file:

1. Select the lines.

Menu	File > Export > Selection to CARIS Well-Known Text

2. Select Export > Selection to CARIS Well-Known Text command.
3. In the *Save As* dialog box, select the folder to export to, and type a file name.
4. Click **Save**.

The selected data is exported to the named WKT file.

Export Selection to DXF

Export a selection of data to a DXF (AutoCAD) file. Contours, soundings, lines and bounding polygons can be exported to DXF format.

Data is saved in two files:

- The DXF file contains the exported data. This type of file can be opened in CAD applications or viewed in HIPS and SIPS as background data.
- The DXF_RXL file contains the coordinate system, unit type and unit scale of the data. This file can only be opened in CARIS applications.

Export contours, soundings or bounding polygon

To export from HIPS to DXF:

1. Open the project containing the contours, soundings or bounding polygon to be exported.
2. In the Layers window, highlight the specific layer, for example, Contours or Depth.
3. In the Display window, select the data to be exported.

Menu	File > Export > Selection to DXF

4. Select the Export to DXF command.

A Save As dialog box is displayed.

5. Enter a file name and browse to the folder where you will save the exported file and click **Save**.

The Output window will report the progress of the export and the number of objects exported. The data is saved to the specified folder.

Export Lines or Critical Soundings

To export selected lines or selected critical soundings to DXF:

1. In the Layers window, highlight the Ship Track Lines or Critical Soundings layer.
2. In the Display window, select the lines or soundings to be exported.

Menu	File > Export > Selection to DXF

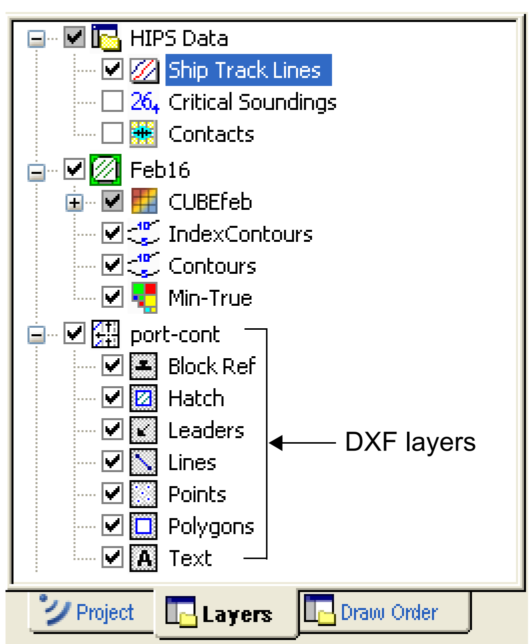
3. Select the Export to DXF command.

The Save As dialog box is displayed.

4. Enter a file name and browse to the folder where you will save the exported file and click **Save**.

The data is saved to the specified folder.

DXF files can be opened in HIPS and SIPS as background data. The opened file is listed in the Layers window, as illustrated below.



Export Selection to Formatted ASCII File

Export selected features such as contours to a formatted `*.ascii` file.

To export to formatted ASCII:

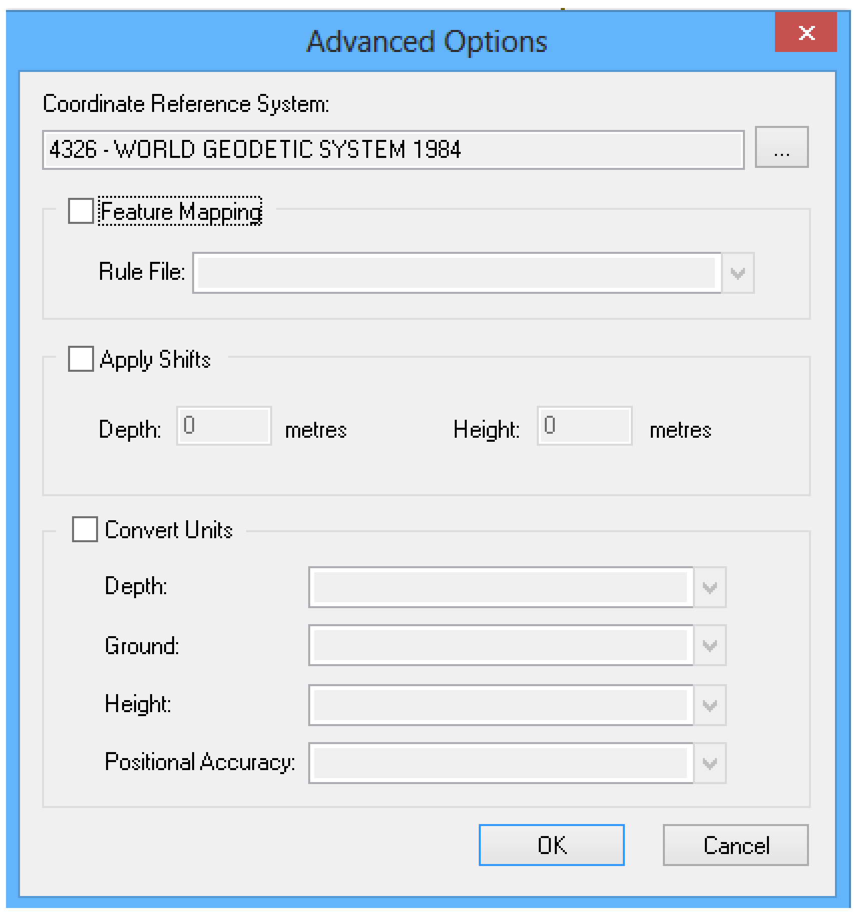
Menu	File > Export > Selection to Formatted ASCII File

1. Select the feature(s) to be exported.
2. Select the Export Selection to Formatted ASCII command.

The Save As dialog box is displayed.

3. Enter a file name and browse to the folder where you will save the exported file.
4. Click **Save**.

The Advanced Options dialog box is displayed



Coordinate Reference System

5. Click **Browse**, and select the project that applies to the selected features you are exporting.

Feature mapping

There are no default mapping files in HIPS. If you want to use a mapping file you must create it before using the Feature Mapping option.

To map the selected HIPS features:

6. Enable the *Feature Mapping* option.
7. From the *Rule File* drop-down list, select a file to use for mapping features.

Apply Shifts

To apply a custom shift to the depths or height:

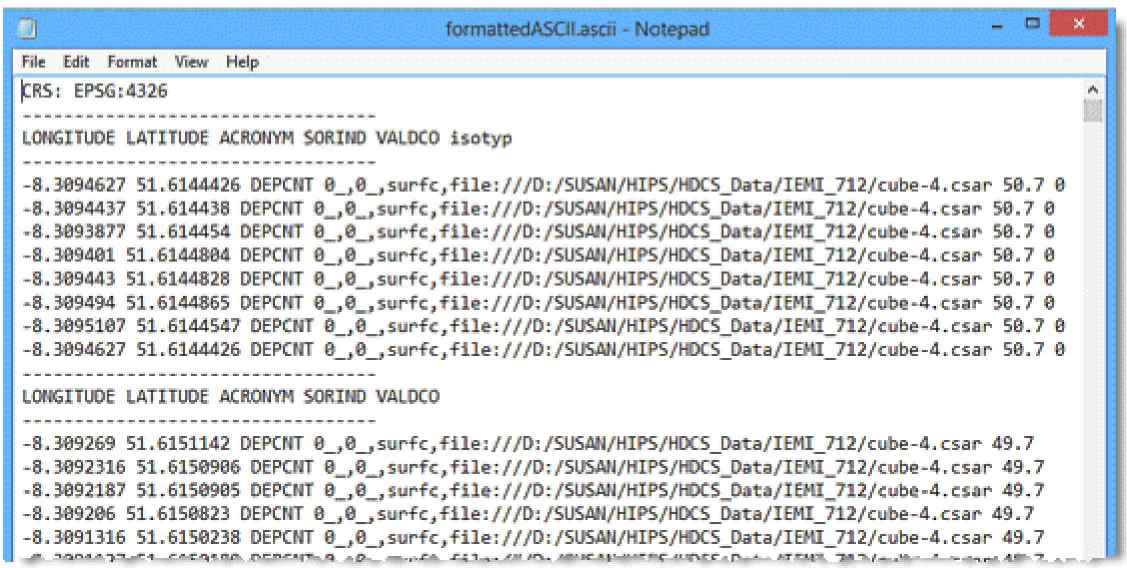
8. Click the check box to enable the Apply Shifts option.
9. Enter values in Depth and Height fields.

Convert Units

To convert Depth, Ground, Height or Positional Accuracy units to another unit of measure:

10. Click the Convert Units check box to enable the option.
11. Select a new unit of measure from the drop-down list for each type you want to change.
12. Click **OK**.

The *.`ascii` file is save to the selected location. The image below shows a sample file.



formattedASCII.ascii - Notepad

File Edit Format View Help

```
CRS: EPSG:4326
----------------------------------
LONGITUDE LATITUDE ACRONYM SORIND VALDCO isotyp
----------------------------------
-8.3094627 51.6144426 DEPCNT 0_,0_,surfc,file:///D:/SUSAN/HIPS/HDCS_Data/IEMI_712/cube-4.csar 50.7 0
-8.3094437 51.614438 DEPCNT 0_,0_,surfc,file:///D:/SUSAN/HIPS/HDCS_Data/IEMI_712/cube-4.csar 50.7 0
-8.3093877 51.614454 DEPCNT 0_,0_,surfc,file:///D:/SUSAN/HIPS/HDCS_Data/IEMI_712/cube-4.csar 50.7 0
-8.309401 51.6144804 DEPCNT 0_,0_,surfc,file:///D:/SUSAN/HIPS/HDCS_Data/IEMI_712/cube-4.csar 50.7 0
-8.309443 51.6144828 DEPCNT 0_,0_,surfc,file:///D:/SUSAN/HIPS/HDCS_Data/IEMI_712/cube-4.csar 50.7 0
-8.309494 51.6144865 DEPCNT 0_,0_,surfc,file:///D:/SUSAN/HIPS/HDCS_Data/IEMI_712/cube-4.csar 50.7 0
-8.3095107 51.6144547 DEPCNT 0_,0_,surfc,file:///D:/SUSAN/HIPS/HDCS_Data/IEMI_712/cube-4.csar 50.7 0
-8.3094627 51.6144426 DEPCNT 0_,0_,surfc,file:///D:/SUSAN/HIPS/HDCS_Data/IEMI_712/cube-4.csar 50.7 0
----------------------------------
LONGITUDE LATITUDE ACRONYM SORIND VALDCO
----------------------------------
-8.309269 51.6151142 DEPCNT 0_,0_,surfc,file:///D:/SUSAN/HIPS/HDCS_Data/IEMI_712/cube-4.csar 49.7
-8.3092316 51.6150906 DEPCNT 0_,0_,surfc,file:///D:/SUSAN/HIPS/HDCS_Data/IEMI_712/cube-4.csar 49.7
-8.3092187 51.6150905 DEPCNT 0_,0_,surfc,file:///D:/SUSAN/HIPS/HDCS_Data/IEMI_712/cube-4.csar 49.7
-8.309206 51.6150823 DEPCNT 0_,0_,surfc,file:///D:/SUSAN/HIPS/HDCS_Data/IEMI_712/cube-4.csar 49.7
-8.3091316 51.6150238 DEPCNT 0_,0_,surfc,file:///D:/SUSAN/HIPS/HDCS_Data/IEMI_712/cube-4.csar 49.7
```

Export Selection to GML

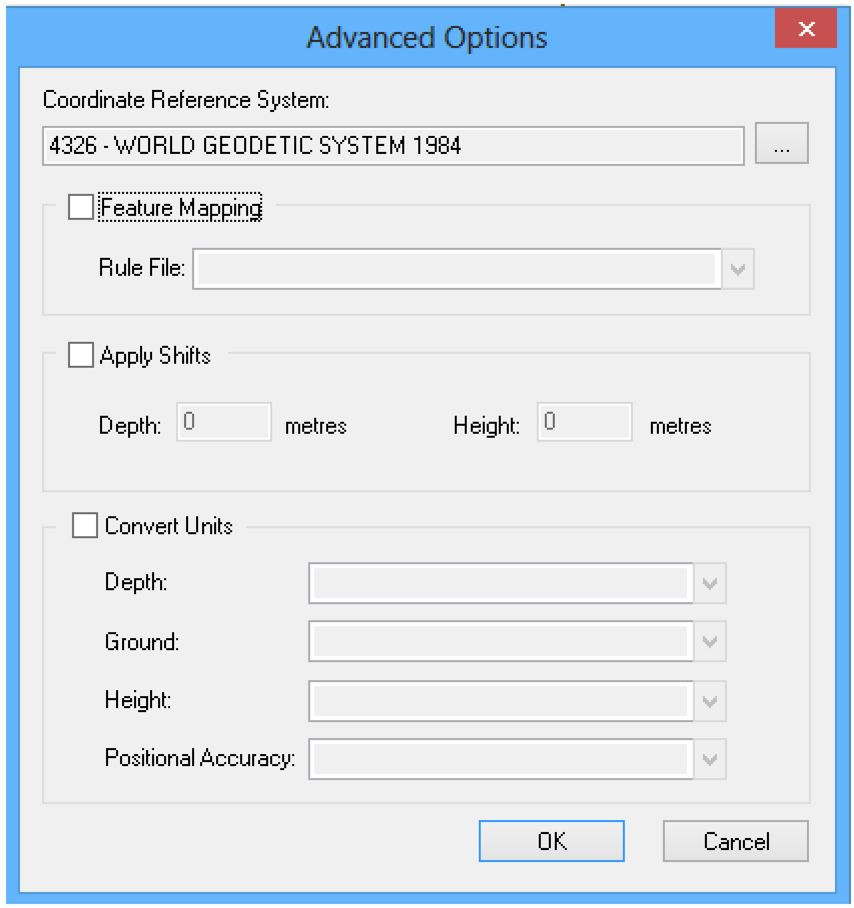
Export information about selected HIPS and SIPS features such as lines, soundings and contacts to a Geography Markup Language (GML) file.

To export to a GML:

Menu	File > Export > Selection to GML

1. Select the feature(s) to be exported.
2. Select the Export Selection to GML command.
3. Click **Save**.

The Advanced Options dialog box is displayed



Coordinate Reference System

4. Click **Browse**, and select the project that applies to the selected features you are exporting.

Feature mapping

There are no default GML mapping files in HIPS. If you want to use a mapping file you must create it before using the Feature Mapping option.

To map the selected HIPS features to GML:

5. Enable the *Feature Mapping* option.
6. From the *Rule File* drop-down list, select a file to use for mapping features.

Apply Shifts

To apply a custom shift to the depths or height:

7. Click the check box to enable the Apply Shifts option.
8. Enter values in Depth and Height fields.

Convert Units

To convert Depth, Ground, Height or Positional Accuracy units to another unit of measure:

9. Click the Convert Units check box to enable the option.
10. Select a new unit of measure from the drop-down list for each type you want to change.
11. Click **OK**.

KML file

S-57 features created in HIPS and SIPS and from background layers can be exported to a Keyhole Markup Language (KML) file.

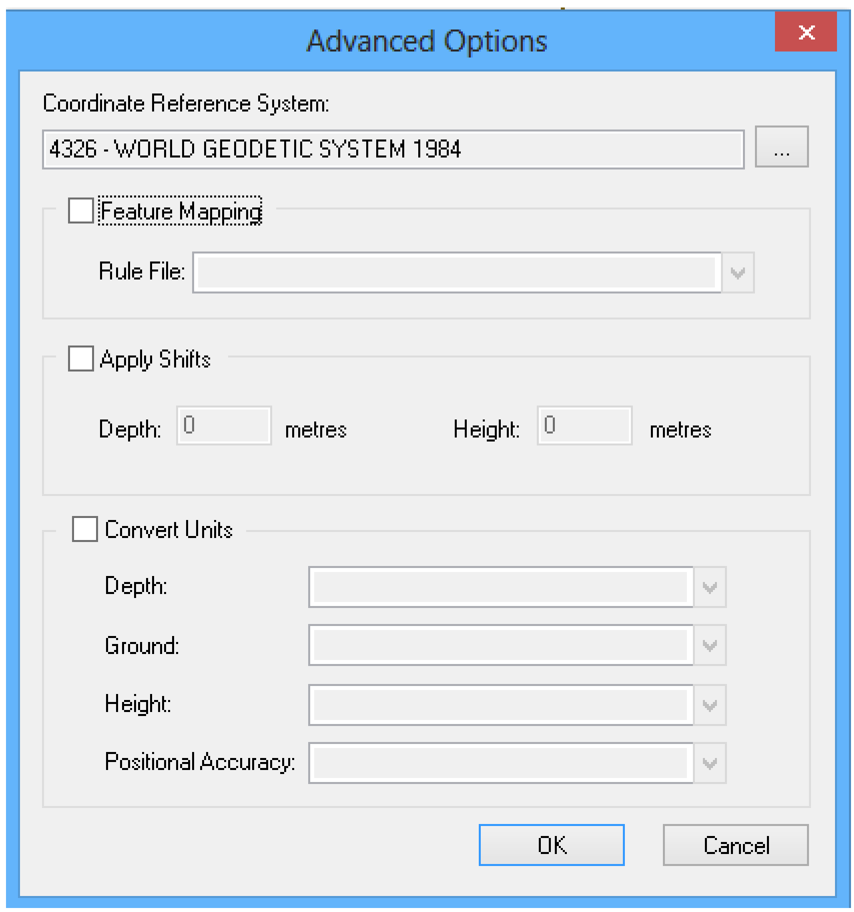
To export to KML:

1. Select the feature(s) to be exported.

Menu	File > Export > Selection to KML File

2. the Export to KML command

The Advanced Options dialog box is displayed



Coordinate Reference System

3. Click **Browse**, and select the project that applies to the selected features you are exporting.

Feature mapping

There are no default mapping files in HIPS. If you want to use a mapping file you must create it before using the Feature Mapping option.

To map the selected HIPS features to KML:

4. Enable the *Feature Mapping* option.
5. From the *Rule File* drop-down list, select a file to use for mapping features.

Apply Shifts

To apply a custom shift to the depths or height:

6. Click the check box to enable the Apply Shifts option.
7. Enter values in Depth and Height fields.

Convert Units

To convert Depth, Ground, Height or Positional Accuracy units to another unit of measure:

8. Click the Convert Units check box to enable the option.
9. Select a new unit of measure from the drop-down list for each type you want to change.
10. Click **OK**.

HOB file

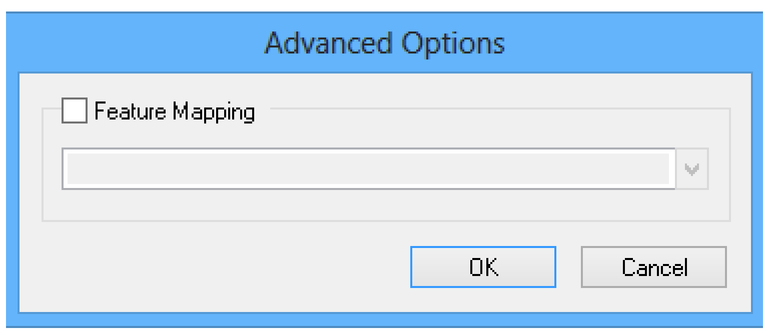
S-57 features created in HIPS and SIPS and from background layers can be exported to a Hydrographic Object Binary (HOB) file.

To export to HOB:

1. Select features from a feature layer.
2. Select the Export to HOB command.
3. In the Save As dialog box, type a name for the exported file.
4. Click **Save**.

Menu	File > Export > Selection to HOB file

The Advanced Options dialog box is displayed.



5. [Optional] Enable the Feature Mapping option and select a mapping file from the drop-down list.
6. Click **OK**.

Export to S-57

S-57 features created in HIPS and SIPS and from background layers can be exported to an S-57 file.

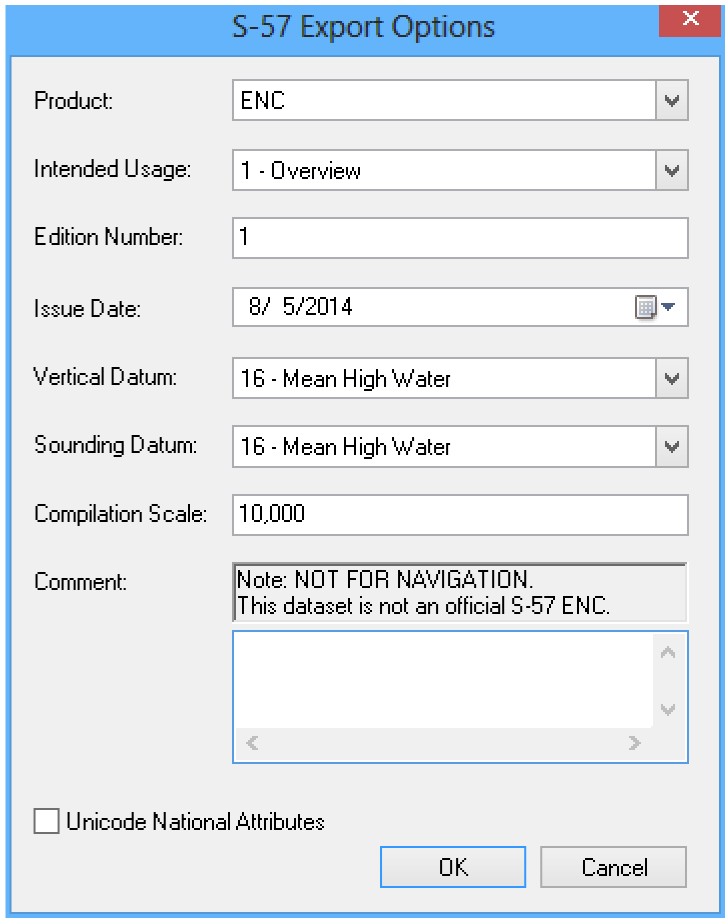
To export to S-57:

1. Select features from a feature layer.

Menu	File > Export > Selection to S-57 file

2. Select Export to S-57 from the File menu.
3. In the Save As dialog box, type a name for the exported `.000` file.
4. Click **Save**.

The S-57 Export Options dialog box is displayed.



5. Select the type of S-57 product you want to save to, from the *Product* list.
6. Choose a navigational purpose layer from the *Intended Usage* list.
7. [Optional] Type a positive number to replace the default number 1 in the *Edition Number* field.
8. [Optional] Change the default *Issue Date* from the current date to another using the calendar tool.
9. Select a Vertical Datum and a Sounding Datum from the lists.
10. Type a new Compilation Scale for the file.
11. [Optional] Type comments in the *Comments* field.
12. Enable *Unicode National Attributes* to export string-type attributes in a non-Roman character set.
13. Click **OK**.

Export to Shapefile

S-57 features created in HIPS and SIPS and from background layers can be exported to ESRI.SHP format.

To export to Shapefile:

1. Select features from a feature layer.

Menu	File > Export > Selection to Shape File

1. Select the Export to Shape File command from the File menu.
2. In the Save As dialog box, type a name for the exported file.
3. Click **Save**.

The features are saved to the named SHP file.

Export View

Export the current view in the Display window to various formats. If more than one 2D display window is open, the view in the currently active window is exported.

Export View - 2D Display Window

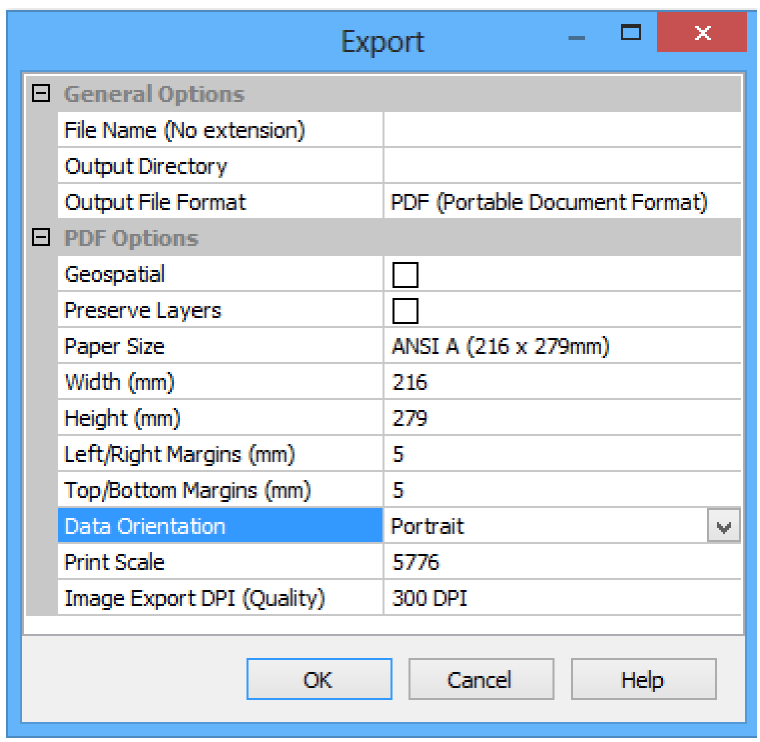
Export the current view of the contents of the Display window to various file formats.

1. Select the 2D Display window view.
2. Make sure data is drawn in the Display window.
3. Zoom / Pan to centre the view.

Menu	File > Export > View

4. Select the (Export) **View** command.

The Export dialog box is displayed.



5. Type a *File Name* for the output file.
6. Select the *Output Directory* field and click the Browse (**...**) button.

The Browse For Folder dialog box is displayed.

7. Select a location for the exported file and click **OK**.

The dialog box closes and the Export dialog box is displayed. The location you selected is displayed in the *Output Directory* field.

8. From the *Output File Format* field, select a format from the drop-down list.

Once you select a format, specific options for that format are displayed in the lower section of the dialog box. Default values are also displayed. The available options are described in the table below.

Options marked with an asterisk* are tied together. Changing one value will affect the value of the related option.

File Formats	Option	Description
• EPS (Encapsulated PostScript) • SVG (Scalable Vector Graphics)	Width /Height (mm)	The width and height of the image in the resulting file. Use the arrow buttons to increase or decrease the size.
	Image Export DPI (Quality)	Set the resolution used to export the view. The higher the resolution, the better the quality of the image when zoomed in, but the larger the size of the output file. If the output file is going to be used in digital form with the image being zoomed in tightly, a higher resolution is recommended. If the file will simply be printed, a smaller resolution may be adequate.
• GeoTIFF (Georeferenced TIFF) • TIFF (Tagged Image File Format) • PNG (Portable Network Graphics)	* Ground Resolution (metres)	Set the size of each pixel in ground units. For example, a pixel represents 100 metres on the ground. As the DPI increases, the ground resolution value decreases.
	* Image Resolution (DPI)	Set the resolution used to export the view. The higher the resolution, the better the quality of the image when zoomed in, but the larger the size of the output file. As the DPI increases, the ground resolution value decreases.
	* Width/Height (Pixels)	The width and height of the resulting image in pixels. Changing the image resolution will adjust both these values.
	Image Depth	Set the number of values applied to each pixel in the image (one value for each colour, plus one for transparency if using 32-bit). If you want a transparency setting applied to the background colour, the 32 Bit (RGBA) option must be selected.
	Background Colour	The colour displayed in the background of the exported image. The default is black.
	Background Alpha	(This option is only available when the Image Depth is set to 32-bit RGBA). Set the degree of transparency applied to the background colour. The default setting is 0, or fully transparent.
	Compression (GeoTIFF and TIFF only)	Compress the resulting image file during export to decrease the file size. This option is only available when using a TIFF format. There are three “lossless” compression methods available. By default, *LZW* is used. To turn off any image compression, select *None*.

File Formats	Option	Description
• PDF (Portable Document Format) • PS (Postscript)	Geospatial (PDF only)	Select the check box to export the view to a geospatial PDF.
	Preserve Layers (PDF only)	Select the check box to print visible information to separate layers.
	* Paper size	Select the page size for the exported content. The values will be displayed in the Width and Height fields.
	* Width /Height (mm)	The width and height of the resulting image. The default values are determined by the Paper Size setting. Changing the default values will reset the Paper Size option to Custom.
	Left/Right Top/ Bottom Margins	Set the size of the area between the captured view and the left and right and top and bottom edges of the output image. The default is 5mm. Use the arrow buttons to adjust the size.
	Data Orientation	Set the orientation of the image in the resulting file. Portrait is a vertical position; Landscape is a horizontal position. Landscape is selected by default.
	* Print Scale	The scale at which the image will be printed (not the scale at which the view is captured). The default value is determined by the Paper Size setting, but can be changed manually. A higher number produces an image more zoomed out, a lower value an image more zoomed in.
	Image Export DPI (Quality)	Set the resolution used to export the view. The higher the resolution, the better the quality of the image when zoomed in, but the larger the size of the output file. If the output file is going to be used in digital form with the image being zoomed in tightly, a higher resolution is recommended. If the file will simply be printed, a smaller resolution may be adequate.
• KMZ (zipped KML files)	Width/Height (Pixels)	Set the width and height of the resulting image in pixels. Use the arrow buttons to adjust the size, or type a value. Changing the width will not change the height value and vice versa.

To change the default values, either select from a list or use the arrow buttons to adjust the values.

Values for related options are automatically adjusted when one value is changed. For example, if the image resolution in dpi is increased, the width and height in pixels is also increased, while the ground resolution in metres is decreased.

You can also override any of these linked values.

9. Enter or select a value for the options as appropriate.

10. Click **OK**.

The file is saved to the specified location in the selected format.

Export View - 3D Display Window

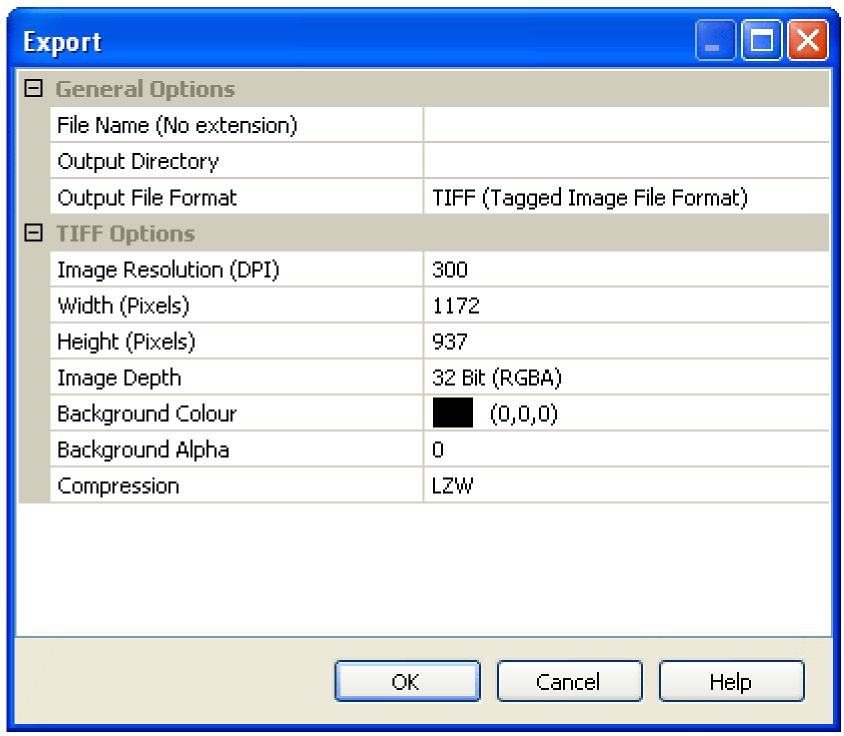
Menu	View > 3D View
Tool	

Menu	File > Export > View

Export the current view of the contents of the Display window to various file formats.

1. Select the 3D Display window view.
2. Make sure data is drawn in the Display window.
3. Zoom / Pan to centre the view.
4. Select the (Export) **View** command.

The Export dialog box is displayed.



5. Type a *File Name* for the output file.
6. Select the *Output Directory* field and click the Browse (**...**) button.

The Browse For Folder dialog box is displayed.

7. Select a location for the exported file and click **OK**.

The dialog box closes and the Export dialog box is displayed. The location you selected is displayed in the *Output Directory* field.

In the *Output File Format* field, TIFF (Tagged Image File Format) is selected. This is the default file type for exporting from the 3D Display view.

Specific options for the format selected are displayed in the lower section of the dialog box. Default values also are displayed. The following options are available for the TIFF format:

Option	Function
* Image Resolution (DPI)	Set the resolution used to export the view. The higher the resolution, the better the quality of the image when zoomed in, but the larger the size of the output file. As the DPI increases, the ground resolution value decreases.
* Width/Height (Pixels)	The width and height of the resulting image in pixels. These fields are updated by changes to the image resolution.
Image Depth	Set the number of values applied to each pixel in the image (one value for each colour, plus one for transparency if using 32-bit). If you want a transparency setting applied to the background colour, the 32 Bit (RGBA) option must be selected.
Background Colour	The colour displayed in the background of the exported image. The default is black.
Background Alpha	(This option is only available when the Image Depth is set to 32-bit RGBA). Set the degree of transparency applied to the background colour. The default setting is 0, or fully transparent.
Compression	Compress the resulting image file during export to decrease the file size. There are three lossless compression methods available. By default, LZW is used. You can also choose not to apply any image compression by selecting *None*.

* Settings marked with an asterisk are related.

To change the default values, either select an option from a list or use the arrow buttons to change the value.

Values for related options are automatically adjusted when one value is changed. For example, if the image resolution in dpi is increased, the width and height in pixels is also increased, while the ground resolution in metres is decreased.

8. Enter or select a value for the options as appropriate.
9. Click **OK**.

The image file is saved to the specified location.

Index

Symbols

Numerics

A

B

C

D

E

K

L

M

N

O

P

Q

R

S

W

X

Z