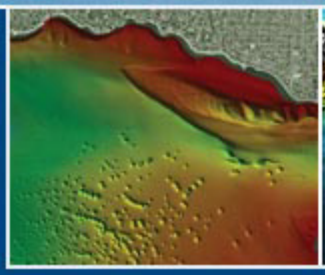
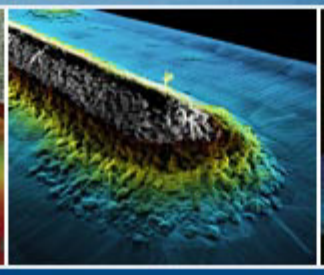
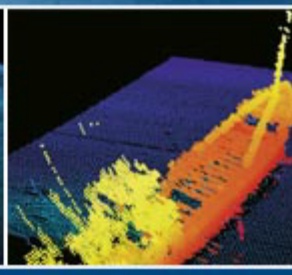
CARIS **HIPS and SIPS 9.1**

Reference







caris THE MARINE GIS EXPERTS

Acknowledgements

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

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

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

Cover images (from left to right):

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

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

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

CARIS 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] interface and how it works. It also describes the options that control the layout and appearance of the interface.

See the User Guide for description of the HIPS and SIPS workflow.

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

The HIPS and SIPS Interface

Describing the interface and its components (menus, toolbars and windows), and how projects and background data can be viewed.

In this chapter...

Opening HIPS and SIPS

To open HIPS and SIPS in Windows 8:

1. Click the Windows key and type HIPS and SIPS.
2. From the HIPS and SIPS versions listed, select HIPS and SIPS 9.0.

In Windows 7 and XP:

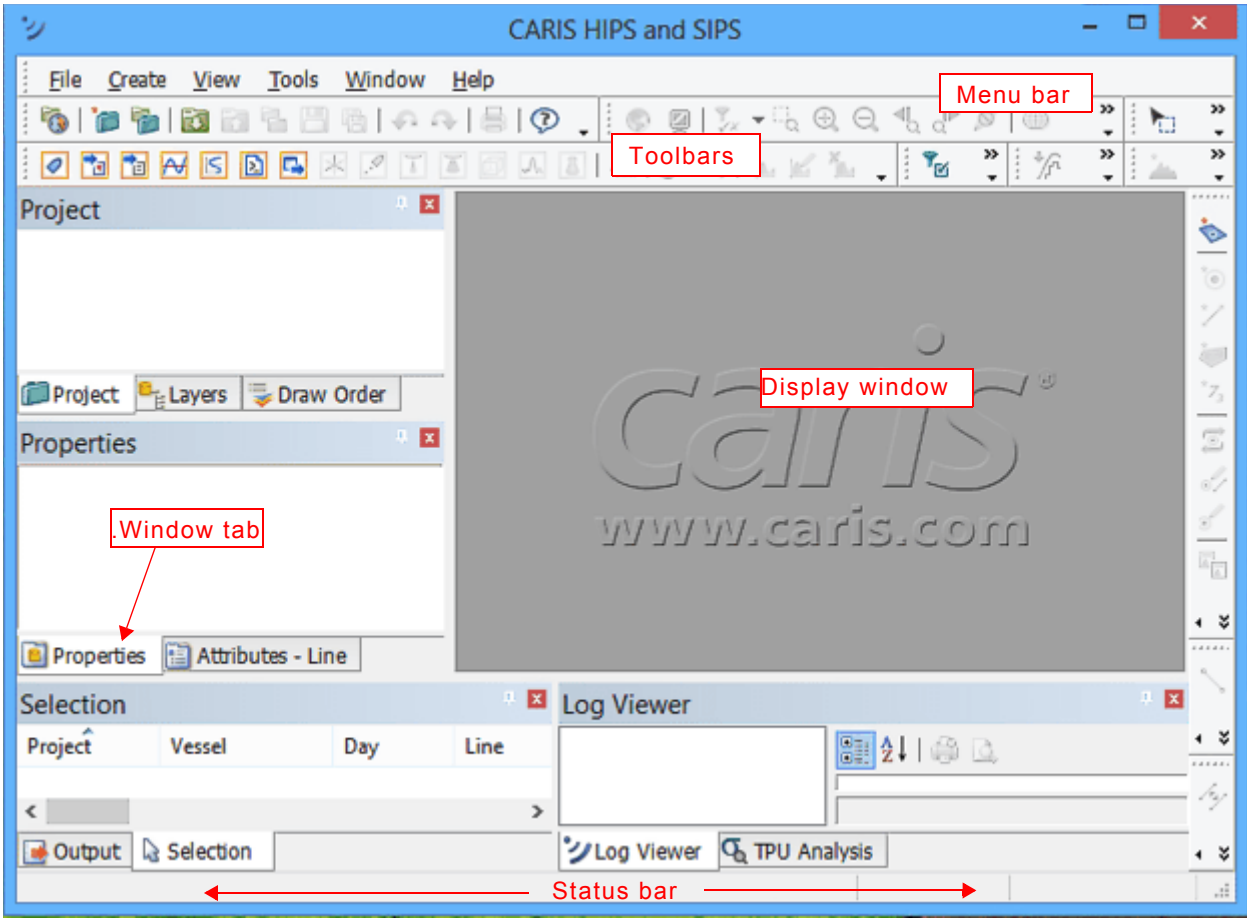
1. Select CARIS HIPS and SIPS from the Start > All Programs / Programs > CARIS > HIPS and SIPS *version number* menu.

If the install prompted you to place an icon on the desktop:

double-click the HIPS and SIPS

Main Interface

The HIPS and SIPS interface is composed of a data display window surrounded by a menu bar, various toolbars, and various dock-able windows.



The main Display window displays 2D and 3D data (see "DISPLAY WINDOW" ON PAGE 16). Other windows display text and graphical information about the open data (see "WINDOWS" ON PAGE 24)

The Menu bar (see "MENUS" ON PAGE 40) contains commands that are also found on toolbars (see "TOOLBARS" ON PAGE 49).

Toolbars and windows can be hidden from display, or moved to other locations on your desktop. See "DISPLAY, REPOSITION AND RESIZE WINDOWS" ON PAGE 38).

Display Window

The main display window shows two-dimensional graphical representations of survey data. In this window, you can view and select data for editing, and create surfaces and layers of features.

"Display multiple windows" on page 16

"Display, Reposition and Resize Windows" on page 38

"Overview Display" on page 19

"Cancel Draw" on page 19

"Refresh Display" on page 18

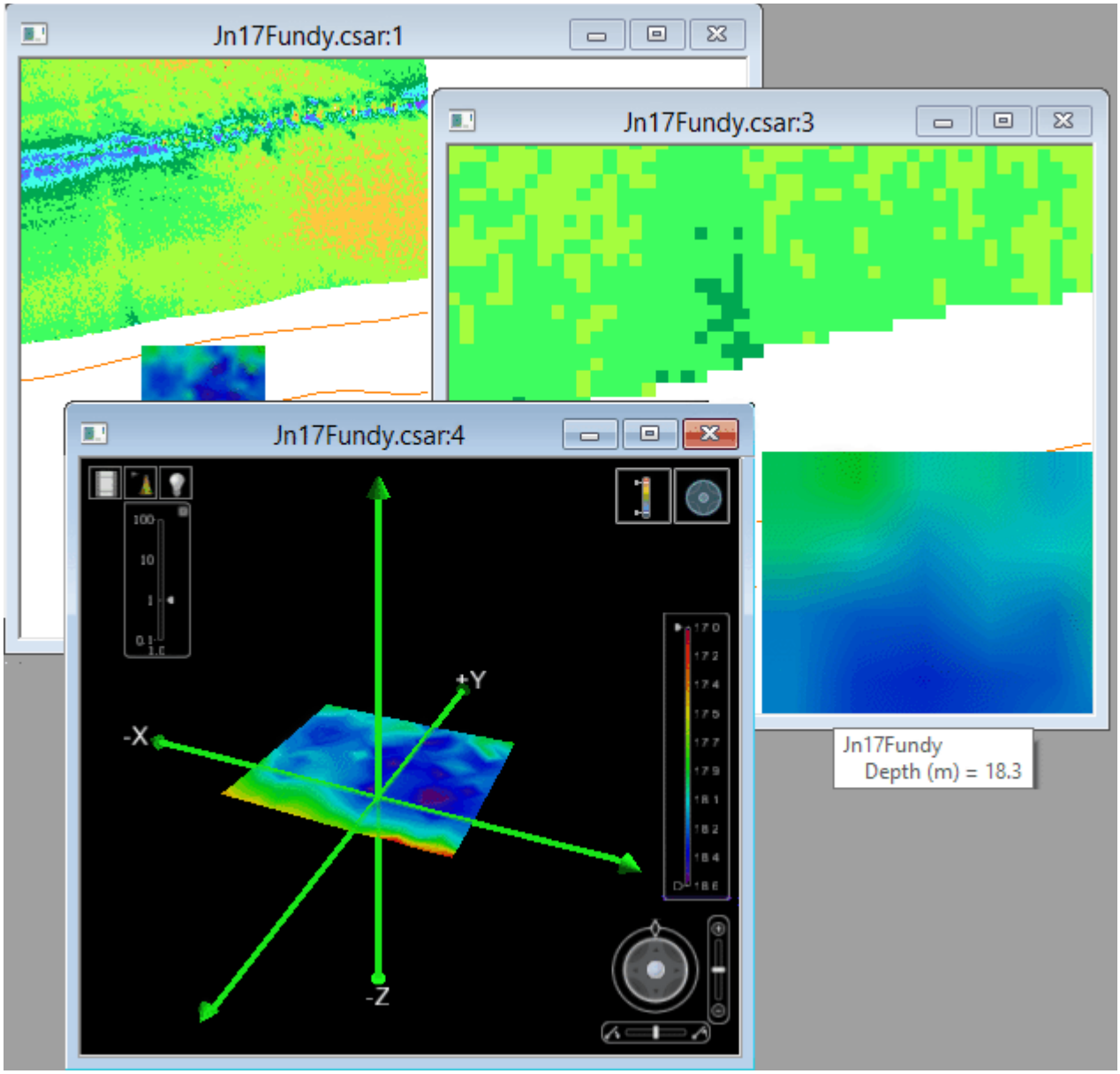
The display also supported by functions:

- zoom and pan data (see "Zoom" on page 21 and "Pan" on page 20).
- view imagery files (see"Open Data" on page 56) and S 57 data (see "Display S-57 Data" on page 75).
- save screen captures of the display in the window (see "Export View - 2D Display Window" on page 573).
- view three-dimensional surfaces in the 3D View in this window space (see "3D Display Window" on page 78).

Display multiple windows

More than one data display window can be viewed at a time using the Cascade and Tile commands. Alternatively you can toggle between views, for example, from 2D to 3D display of the data.

The following is an example of cascaded windows, one being the 3D view of a surface, the other windows are 2D, displaying the surface at different zoom levels.



The 2D and 3D display windows cannot moved outside the main display area.

Layouts

A default layout is used to position the windows and the toolbars in the interface.

New Layout

The New Layout command saves the current layout of the windows and toolbars so that it can be restored at a later time. This enables you to create specific layouts for different tasks, eliminating the need to reposition the interface each time.

To create a new layout:

1. Open and position all windows and toolbars as you want them saved.

Menu	Window > Layouts > New Layout

2. Select the New Layout command.

The New Layout dialog box is displayed.

3. Enter a name for the layout.
4. Click **OK**.

The layout is saved. Saved layouts can be reapplied by selecting them from the list in the Layouts sub-menu, or from the Layouts dialog box.

Reset Default Layout

Return the current layout settings to the default layout setting. Specifically, move the windows and tool bars that are displayed by default back to their default positions, closing all other windows and toolbars. See "WINDOWS" ON PAGE 24 for a list of the default windows.

There is no "undo" for this action.

Menu	Window > Reset Default Layout

1. Select the Reset Default Layout command.

The default layout setting is restored.

Refresh Display

Menu	View > Refresh
Tool	
Key	<F5>

Redraw the view in the Display window.

1. Select the Refresh command
2. Alternatively, click the middle mouse button when the cursor is in the Display window.

The display is redrawn.

Overview Display

Menu	View > Overview
Tool	
Key	<F9>

Redraw the Display window to the full extent of all the data that is currently open.

1. Select the Overview command.

Cancel Draw

You can stop a display from being drawn in the Display window. with the Cancel Draw command. This command is only active while the image is being drawn.

Menu	View > Cancel Draw
Tool	

To cancel the display:

1. Select the Cancel Draw command.

Copy Position

This command on the pop-up menu enables you to copy the coordinates of the current position of your cursor in the Display window to the clipboard.

To copy the position under your cursor in the Display window:

Pop-up	Copy Position

1. Right-click on the position in the Display window.
2. Select Copy Position from the pop-up menu.
3. Open a text editor such as Notepad, and Paste the coordinates into the editor.

Navigating the 2D View

There are two kinds of tools that you can use to manipulate the view of data in a Display window: Pan and Zoom. The Pan function lets you reposition your view within the Display window. The Zoom function magnifies or reduces the viewing area within the Display window.

The Zoom/Pan Reverse and Forward tools let you toggle between views, and you can zoom to centre on a specific set of coordinates using the Zoom To command.

Pan

There are two ways to move the view of data up, down, or to the left or right, within a Display window:

- by using the Pan tool to move by a defined amount, or
- by dragging the display with the mouse.

Pan tool

The Pan tool will let you move the view by a percentage of the visible display. You can set the amount that the view will move in the *Pan Factor* field on the General tab of the Tools > Options dialog box. (See "GENERAL" ON PAGE 173.)

To move the view of data with the Pan tool:

1. Select a Pan command:

Menu	View > Pan > Up/ Down/Left/Right

   - *Pan Up*: Re-centre the display at a position that is above the current display area.
   - *Pan Down*: Re-centre the display at a position that is below the current display area.
   - *Pan Left*: Re-centre the display at a position to the left of the current display area.
   - *Pan Right*: Re-centre the display at a position to the right of the current display area.

The Pan commands on the View menu can be added as buttons to a new or existing Toolbar by using the View > Toolbars > Customize… command.

Pan with the mouse

You can pan the display with your mouse by using a press and drag action.

To pan with the mouse:

1. Press and hold down the scroll wheel or middle button on your mouse. (The cursor changes to a hand.)
2. Use the mouse to drag the view in any direction.
3. Release the mouse button when you have repositioned the view.

Zoom

There are two ways to magnify or reduce the viewing area in the Display window:

- use the scroll wheel on your mouse, if available
- use the Zoom tools

Zoom with the mouse

To zoom in or out using a mouse:

1. Click in the Display window.
2. Scroll back and forth with the scroll wheel to zoom the view up and down.

Zoom tool

Use the Zoom tool to select a specific area of the Display to enlarge:

1. Select the Zoom command.

Menu	View > Zoom
Tool	
Key	<F10>

The cursor changes to a magnifying glass icon.

2. Press and hold the mouse button and drag the cursor to create a rectangle around the area you want to magnify.

The display is enlarged so that the area within the rectangle now fills the Display window.

Once you enlarge the area, the Zoom tool button is no longer active. To keep the Zoom tool active, select the *Enable Constant Zoom* function in the Options dialog box (see "GENERAL" ON PAGE 173).

Zoom In

Menu	View > Zoom In
Tool	
Key	<Ctrl+Down>

The Zoom In tool lets you enlarge the viewing area in the Display window automatically by a set amount.

1. Select the Zoom In command.

The display is automatically enlarged by the percentage set in the *Zoom Factor* field on the General tab of the Options dialog box (see "GENERAL" ON PAGE 173).

Zoom Out

Menu	View > Zoom Out
Tool	
Key	<Ctrl+Up>

Zoom Out automatically reduces the viewing area in the Display window by a set amount.

1. Select the Zoom Out command.

The display is automatically reduced by the percentage set in the *Zoom Factor* field on the General tab of the Options dialog box (see "GENERAL" ON PAGE 173).

Zoom/Pan Previous

Menu	View > Zoom/Pan Previous
Tool	

This tool returns the view to the position or magnification that was displayed before you zoomed or panned the view.

1. Select the Zoom/Pan Previous command.

Zoom/Pan Forward

Menu	View > Zoom/Pan Forward
Tool	

This tool returns the view to the position or magnification that it was in prior to using the Zoom/Pan Previous command.

1. Select the Zoom/Pan Forward command.

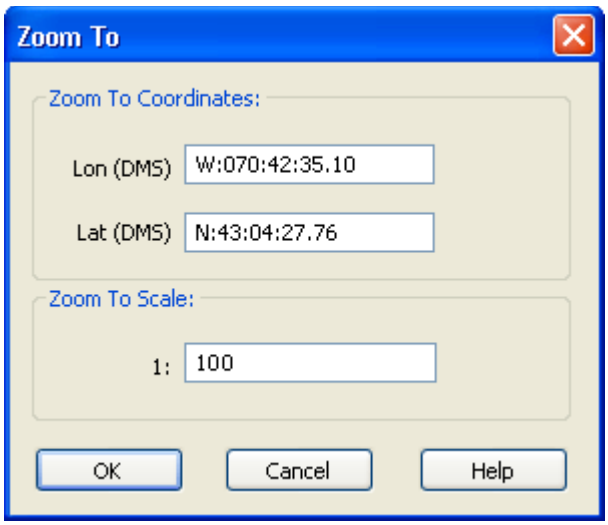
Zoom To

Menu	View > Zoom To

Re-centre the view on specific coordinates in the Display window, or zoom in so that the viewed area is displayed at a specific scale.

1. Select the Zoom To command.

The Zoom To dialog box is displayed.



2. Type the new coordinates in their respective fields to re-centre on that point in of the Display window,

OR

3. Type a new scale to zoom into or out of the display.
4. Click **OK**.

For options for the display of the ship and towfish track lines see "DISPLAY WINDOW" ON PAGE 189.

Windows

When HIPS and SIPS is first opened, only the default windows are displayed.

Additional windows are available for specific tasks.

Attributes Window

Use the Attributes window to view and edit the attributes of features such as contacts and critical soundings.

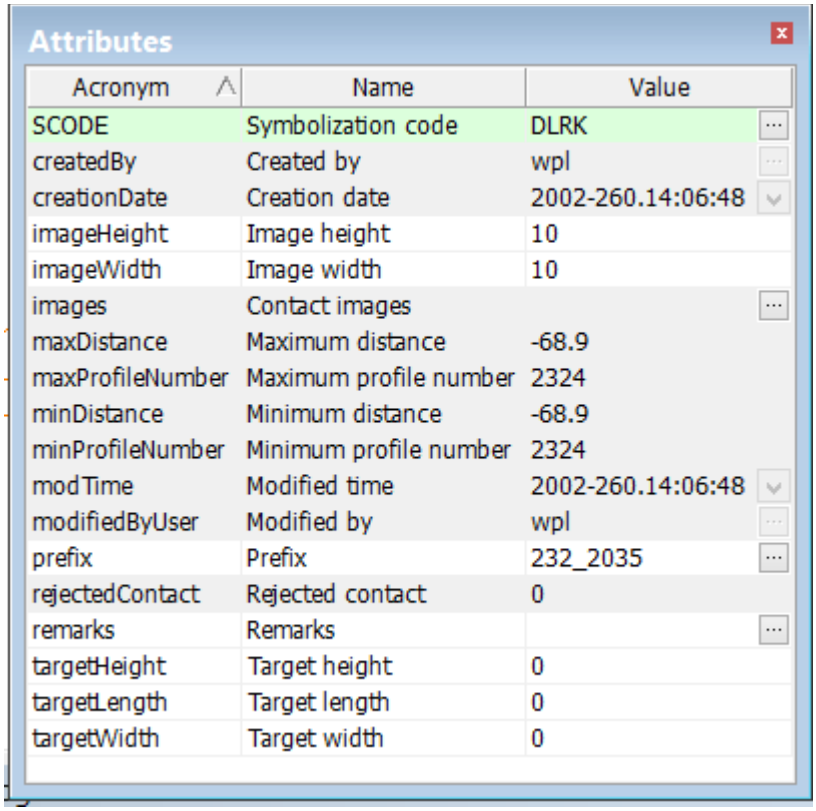
If a number of these features are selected in HIPS, the Attributes window will display the attributes for the feature that is "superselected" in the Selection window. (See "Superselection" on page 34 for more information.)

(The superselected feature is coloured light blue in the Display window, and highlighted in the list in the Selection window. The remainder of the selected features are coloured bright red in the Display and are not highlighted in the Selection window.)

To view attributes in the Attributes window:

1. Select one or more features.
2. Select Attributes from the Window menu to open the window.

The attributes for the superselected feature are displayed. The example below shows attributes of a selected contact.



The window displays the Acronym, Name and Value of those attributes which have been defined in the Catalogue Editor and are present in the selected contact.

- Fields without highlighting can be edited by typing a value or by using the **Browse** button to open the Insert New Value dialog box, and choosing from the options listed there.
- Fields that are “greyed out” are read-only, information fields, and their values cannot be changed.
- Fields highlighted in red must have a value entered. When a value is entered, the field is highlighted green to indicate that this is an attribute which should always have a field value set.
- When the necessary text is entered into the field, it will automatically change to a link. To use the link, press and hold the <Ctrl> key and click the link.

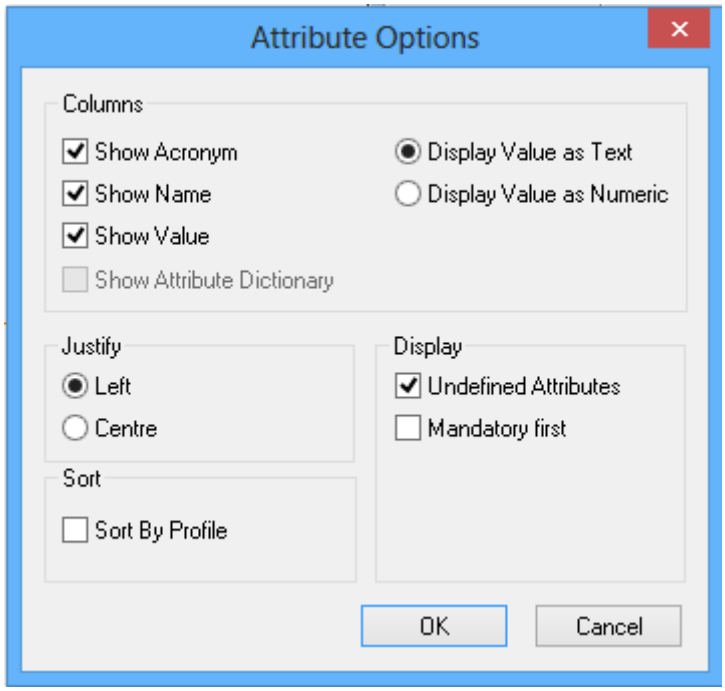
Attributes Window Options

Options for the Attributes window are available on a right-click menu opened within the window. Functions on this menu include setting options, selecting all data in the window, copying selected data, and saving data to a text file.

To view options for the Attributes window:

1. Right-click in the window and select Attribute Options.

The Attribute Options dialog box displayed below shows the default settings for the window.

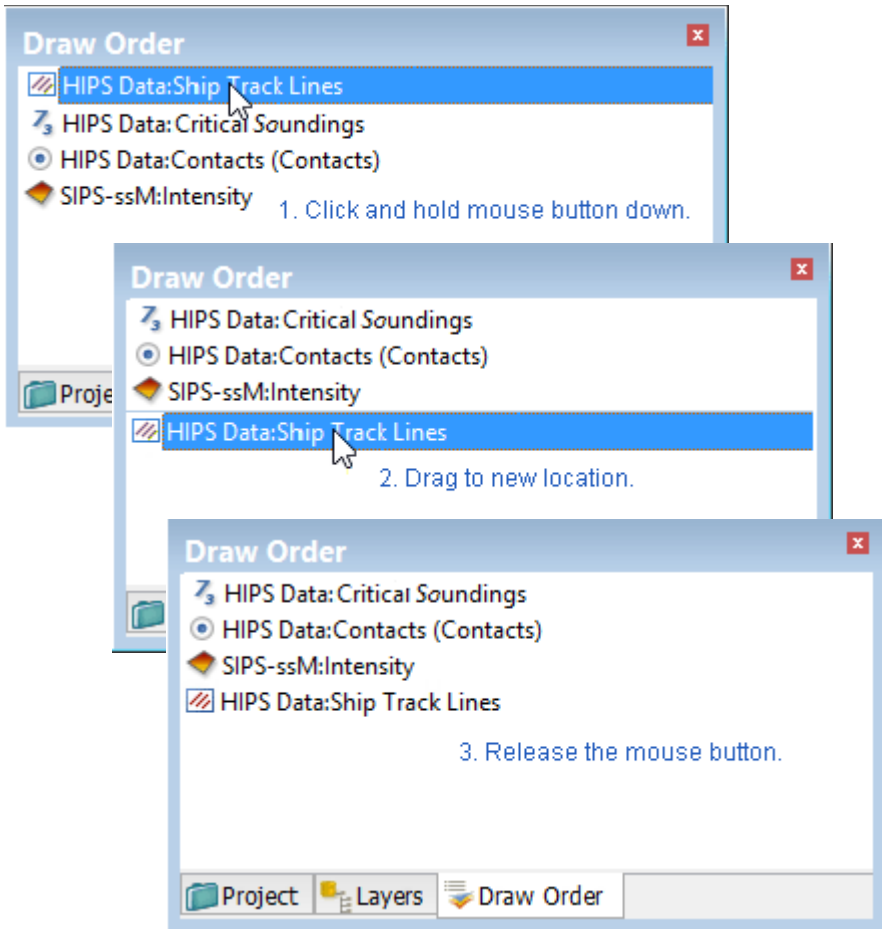


These settings are:

Option	Description
Show Acronym	Displays the Acronym column.
Show Name	Displays the Name column.
Show Value	Displays the Value column.
Display Value as Text / as Numeric	Display attribute values using either the descriptive text or the numeric equivalent as set in the Catalogue for the project
Justify: Left/Centre	Align values in each column to the left or centre the values.
Sort by Profile	List the attributes in the order in which they are used to define the feature. When this option is not selected attributes are listed alphabetically.
Undefined Attributes	By default all attributes are displayed, including those that have no defined values. If not selected, only attributes with values are displayed.
Mandatory first	Display mandatory attributes at the top of the list of attributes.

Draw Order window

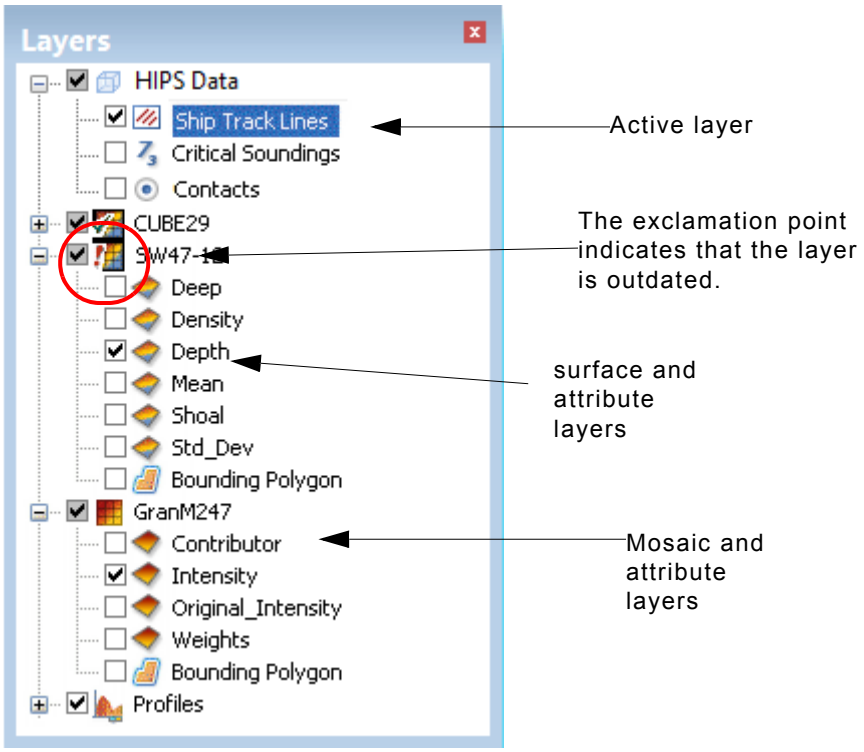
The Draw Order window controls the order that objects are drawn to the Display window. Objects are drawn in order, from top to bottom, as they appear in the Draw Order window. You can change the draw order by dragging an object to another position on the list, as illustrated below.



Layers Window

The Layers window lists the open data layers. Control what is viewed in the Display window by selecting or clearing the check boxes for the various layers.

Besides survey data, a layer can consist of a surface, a surface attribute layer, a CARIS map, an image, or product layers such as contours or selected soundings.



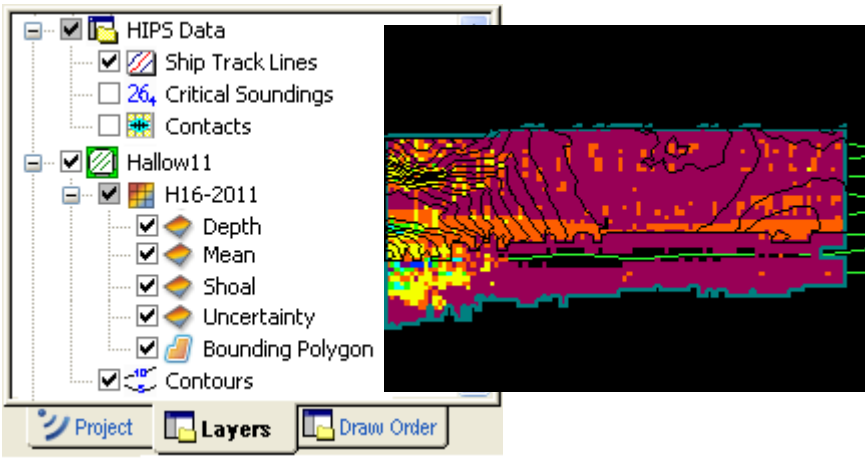
1. Expand the file tree by clicking the + icon beside it.
2. Show/hide the display of a layer (in the Display window) by selecting or clearing the check box beside the layer name.
3. Select a layer (so that it is highlighted) to make it the active layer. The data in that layer can now be selected in the Display window.
4. Select Use Coordinate System from the pop-up menu to apply the coordinate system from a layer to the view of data in the Display window. For an example of changing the coordinate using a background data layer, see "Use Coordinate System" on page 65.
5. To update the Display window, select the Refresh command.
   - Or turn on the Automatic Refresh option in the General Options dialog box (see "General" on page 173).

Menu	View >Refresh
Tools	
Key	F5

Surfaces can be grouped together in the Layers window in order, for example, to apply the same display properties to the attribute layers of multiple surfaces simultaneously. See "Group Multiple Surfaces" on page 244.

- If you select all the child layers, you can toggle them all on and off using the parent layer check box. In the following

example *all* the child layers of H16-2011 have been toggled on.

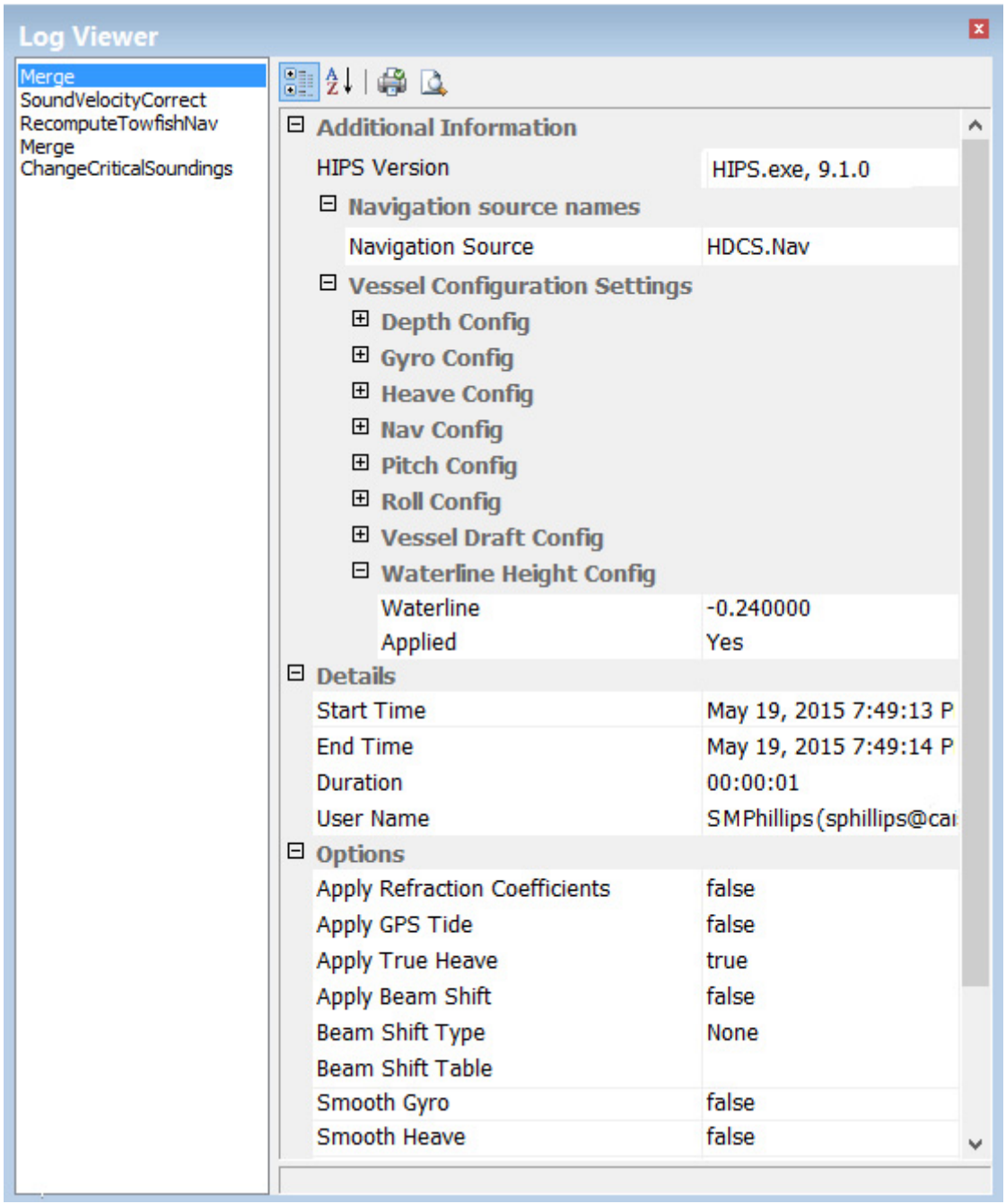


Log Viewer

HIPS automatically creates an XML `LogFile` in the line directory for each line, recording the operations that are applied to the line.

Menu	Window > Log Viewer

The Log Viewer, opened from the Window menu, displays the process log for a superselected line.



Processes are listed in the left hand column in chronological order, with the earliest applied process at the top of the list and the latest process applied at the bottom of the list.

Processes that can be displayed include:

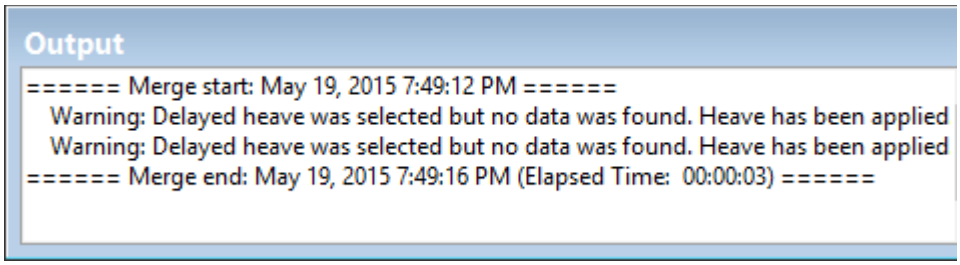
- Conversion
- Load Tide
- Load Delta Draft
- Sound Velocity Correction

- Merge
- Compute TPU
- Compute GPS Tide
- Load Auxiliary Data
- Shift Navigation
- Beam Pattern Correction
- Recompute Towfish Navigation

The Viewer

Output Window

In this window you can view messages about the results of a command or process.



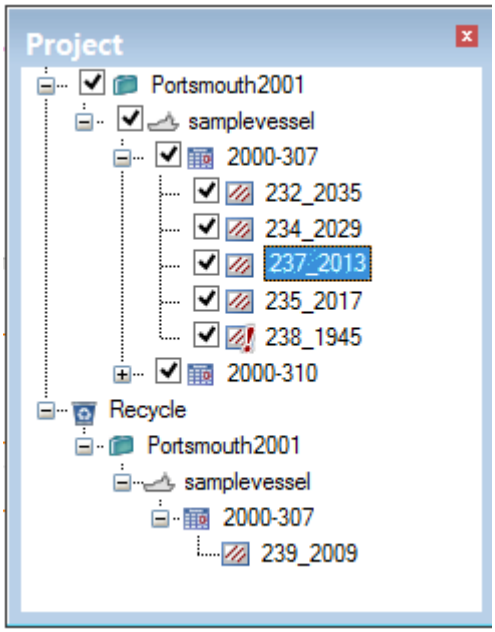
Information in this window is read-only and cannot be modified.

If you want to remove the information in this window, right-click the mouse and select Clear from the pop-up menu.

Project window

The Project window displays the currently open project.

You can expand the tree by clicking the plus sign (**+**) to reveal the layers of the Project/Vessel/Day/Line hierarchy. A check mark in the check box for each line indicates that the line is visible in the Display window. By default all lines are set to visible.



Highlighting on a line indicates it is selected in the Display window.

An empty check box indicates the line is not visible in the Display window.

The red ! icon indicates that the line is Outdated.

A red X icon indicates that the line is Rejected.

Lines displayed in the layers below Recycle are lines that have been removed from the project.

Use the Project window to open and close data and to select track lines.

For information on working with projects, see "CREATE A NEW PROJECT" in the CARIS HIPS and SIPS User Guide.

Properties Window

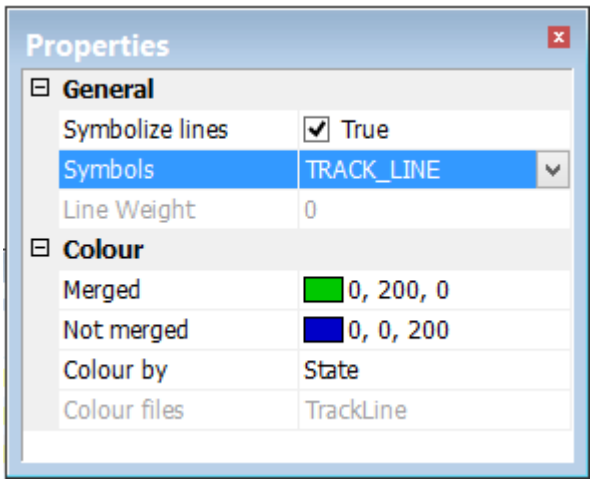
The Properties window displays the properties of lines, surfaces and features that are listed in the Layers window and selected in the Display window. The Properties window is displayed in the default layout.

The properties shown in the window are determined by the type of layer selected, that is, ship track lines and surface layers will display different properties.

Menu	Window > Other Windows > Properties

1. Select the Properties window command.
2. Select a layer in the Layers window.

The properties for that layer are shown in the window. The example shown below shows properties for the Ship Track Line layer.



For more information on viewing and setting properties for specific layers, see the following in the HIPS and SIPS User Guide:

- "Set Line Properties" on page 74
- "Critical Soundings" on page 335
- "View Project Properties" on page 71

Report window

The Reports window contains statistical data generated from the Report tools, such as Detailed Line Query and Line QC on the Process menu.

For details on creating QC reports, see "Quality Control Reports" on page 350.

You can customize display of data in the QC Reports window in the same manner as in the Selection window. See "Selection Window" on page 34 for more information.

Menu	Window > Detailed Line Query

Data is displayed in the window when this is activated from the Detailed Line Query command on the Process menu. (See "Detailed Line Query" on page 314.)

Which data is visible and in what order it is displayed is controlled using the Column Settings dialog box from the right-click menu. (See "Column Settings" on page 35.)

As with the Selection window, records in a column can be sorted by clicking on the column header.

This information can be saved to a text file, using the Save As command on the right-click menu.

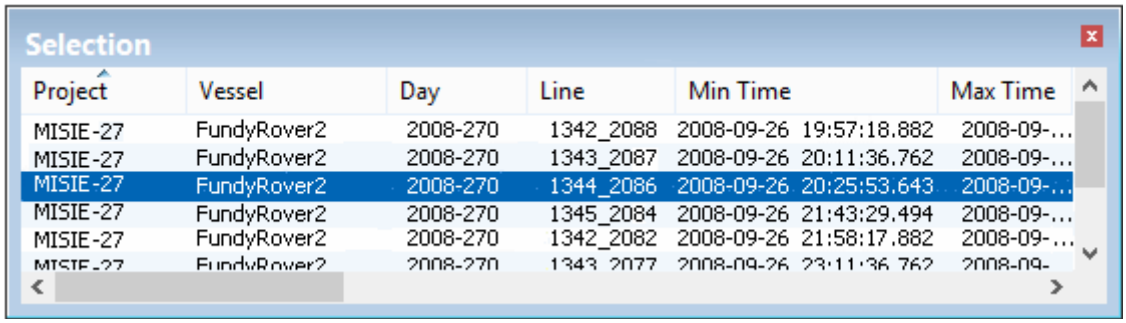
To save a customized selection of the data, see "LINE REPORT" ON PAGE 315.

Selection Window

When you select features such as lines, soundings or contours in the Display window, their data profiles are automatically displayed in the Selection window.

To see this information for data selected in a HIPS and SIPS editor, use a Query command. See "QUERY DATA" ON PAGE 313 in the User Guide.

The example below displays the data profiles for four selected lines. The line that is highlighted in the table is the feature that is superselected.



Superselection

When a number of features are selected, but a function can only be applied to one feature at a time, it is the superselected feature which will be affected. In HIPS and SIPS the superselected feature is coloured light blue in the Display window and other features in the selection are coloured red. In the Selection window, the superselected feature is highlighted.

Data profiles

Data profiles in the Selection window are shown in tabular format, with a column for each data field. Records in a column can be sorted by clicking on the column header. This will sort the records by ascending or descending order of the values in the column.

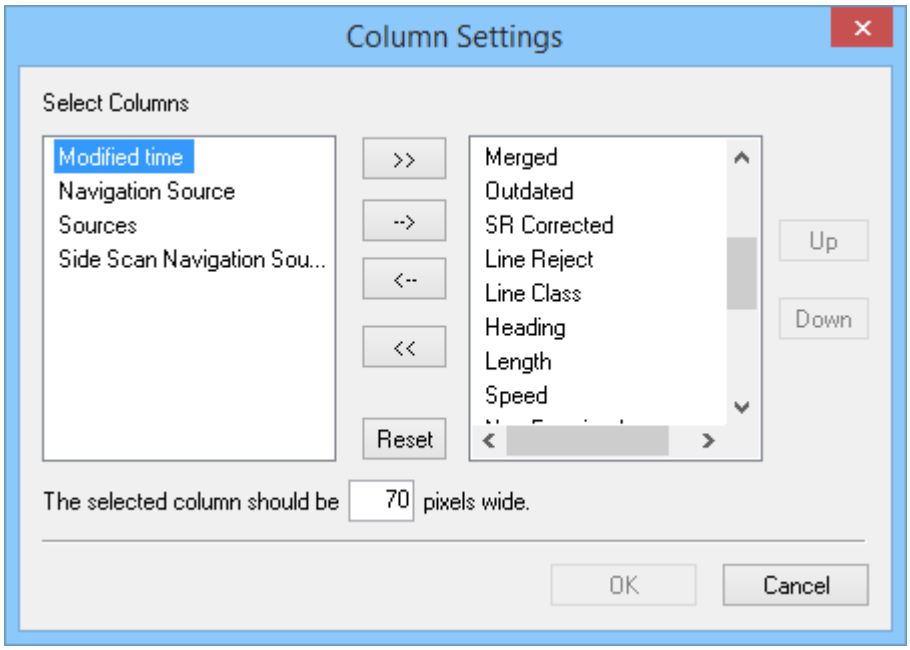
This data is read only. However, it can be saved to a text file using the command on the pop-up menu.

Which columns are displayed, and the order in which they are displayed, is controlled by the pop-up menu. To set which data to display:

1. Right-click on a column header in the Selection window.
2. Select columns by name from the list on the pop-up menu, or select **More** to open the Column Settings dialog box.

Column Settings

Use the Column Settings dialog box to determine the column headings to be displayed. The example below shows the column headings for contacts selected in the Display window.



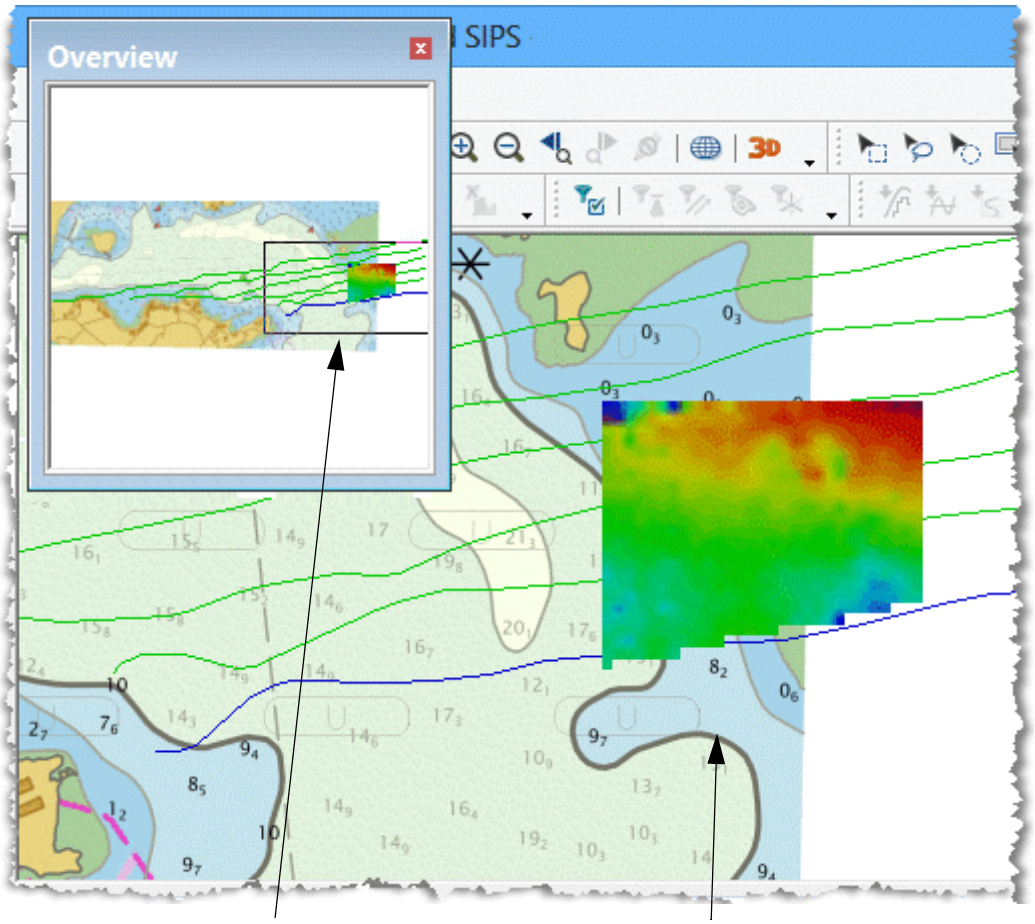
The left side of the dialog box lists headings that are available but not currently visible, and the right side lists the column headings currently displayed in the Selection window

3. Use the ← and → buttons to move the headings from one column to the other.
4. Use the **Up** and **Down** buttons to change the order in which they are shown in the Selection window.
5. To adjust the width of a column, select it in the right-hand list and type the desired width in pixels. (Column width can also be set manually in the Selection window by dragging the right or left edge of a column header.)
6. To restore the default listing of fields and their order, click Reset.
7. Click **OK**.

Overview Window

The Overview window displays an overview of all open data.

Selecting an area in the Overview window will cause the Display window to automatically zoom and re-centre on the area selected in the Overview window, as demonstrated below.



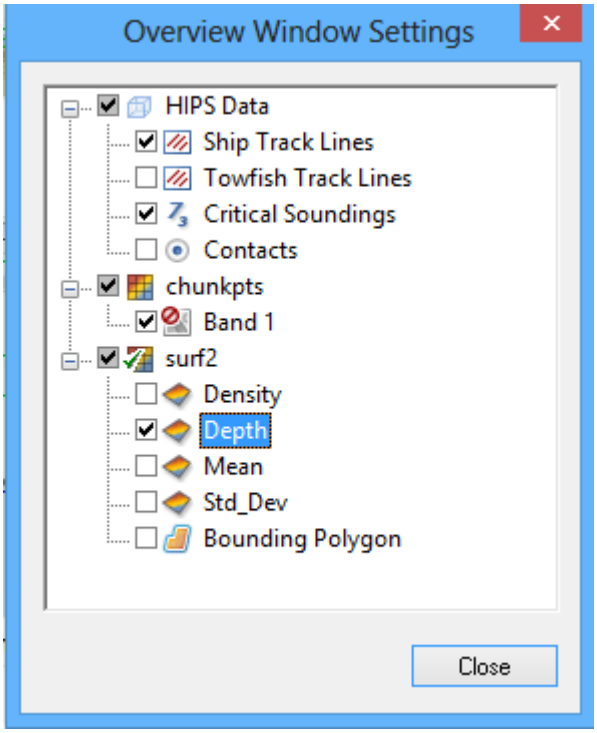
As with the Display window, you can choose which layers to make visible in the Overview window.

Menu	View > Overview Window Settings
Pop-up	Settings (Overview window)

To select which layers to display:

1. Select the **Overview Window Settings** command.

The Overview Window Settings dialog box is displayed.



This dialog box contains a list of layers available for display in the Overview window.

2. Show or hide layers by selecting or de-selecting the check boxes beside the layer names.
3. If necessary, expand a parent layer by clicking its Expand (+) icon.
4. Click **Close**.

The Overview window is redrawn to reflect the changes.

Display, Reposition and Resize Windows

Windows in HIPS and SIPS are “docked” or fixed in their default positions relative to the Display window. Windows can be repositioned within the interface or to positions on your desktop. Closed windows can be reopened from the Windows menu.

You can restore windows to their original positions using the Reset Default Layout command on the Window menu.

Generally, the following repositioning options are available:

- Position a window as a floating window.
- Position a window as a docked window.
- Position a window as a docked window within the area of the Display window.
- Position a window so that it is grouped with other windows.

Prevent the window from automatically docking within the interface, by holding the <**Ctrl**> key while dragging the window.

Docking

Window positioning is controlled through icons that are visible only while a window is being moved. These positioning icons show where a window can be placed.

You can move a group of windows by grabbing the title bar or move a single window by grabbing the window label.

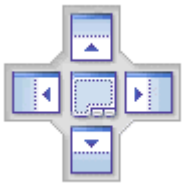
1. Position the cursor over the window’s title bar or label.
2. Press and hold the mouse button while dragging the window into the interface.

As you move the window, the positioning icons are displayed in the interface.

Positioning icons

The following table lists the icons and contains a brief description of each one.

Icon	Description
	The 4 icons clustered here position a window within the area of the Display window. The window can be moved to one of 4 positions (starting clockwise from the top): • top • right • bottom • left
	Position the window as a separate window at the top of the interface (below the toolbars).

Icon	Description
	Position the window as separate window at the right side of the interface.
	Position the window a separate window at the bottom of the interface (above the status bar).
	Position the window as a separate window at the left side of the interface.
	These 5 icons help position the window in relation to other windows. This icon is only displayed over windows that are docked and tabbed together. The centre icon places the floating window with those already docked. The location of the floating window can be one of 5 positions (starting clockwise): • on top of an existing window • to the right of an existing window • below an existing window • to the left of an existing window • as part of a group of tabbed window

3. While holding the mouse button, move the window over one of the icons.

As you move the window over the positioning icon, a blue outline area highlights where the window will be docked in the interface.

Re-size

Windows can also be sized to a new length and width.

1. Position the cursor on any edge or corner of the window.

The cursor becomes a two-headed arrow.

2. Press and hold the mouse button and drag the edge of the window to a new size.

As you drag the cursor, the window expands or contracts, depending on whether you are pulling or pushing the window edge.

3. Release the mouse button when you have resized the window to the desired length and width.

Menus

Most HIPS and SIPS commands can be accessed from menus on the menu bar at the top of the interface. The functions of the various commands are described within the workflow outlined in the HIPS and SIPS Users Guide.

For an alphabetical listing of all HIPS and SIPS commands, see the "Alphabetically by Command Name" list in the HIPS and SIPS Tools guide.

“File menu” on page 41

“Create Menu” on page 42

“Edit menu” on page 43

“View menu” on page 44

“Process menu” on page 45

“Tools menu” on page 46

“Select menu” on page 47

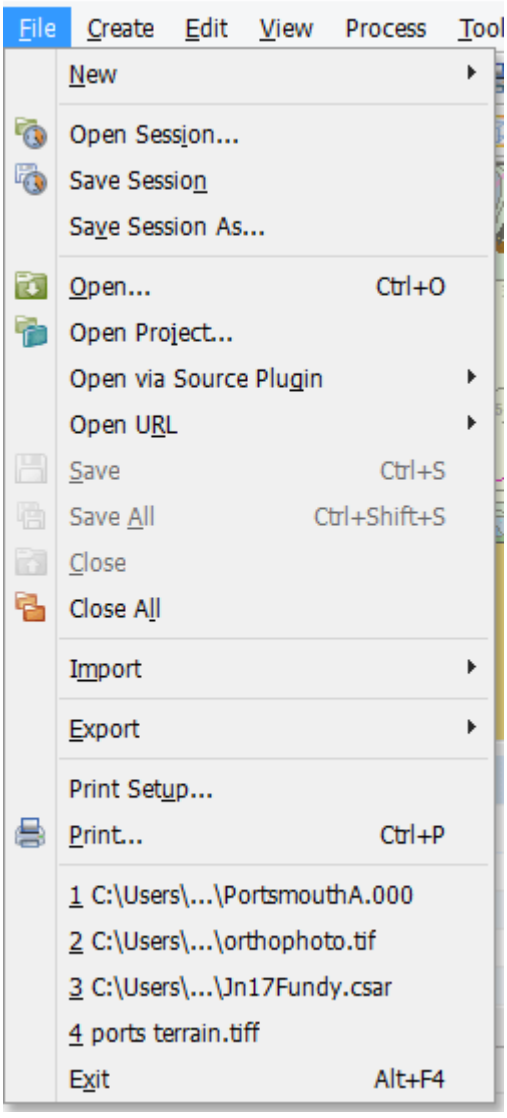
“Window menu” on page 47

“Help menu” on page 48

File menu

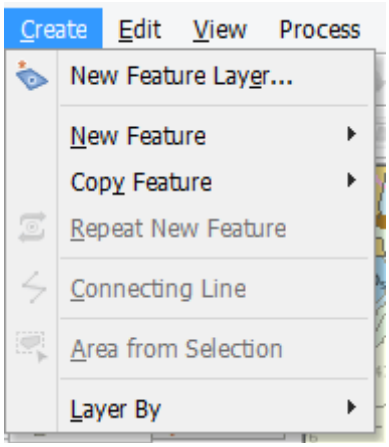
Use the File menu commands to open, save and close projects, sessions and background data.

It also contains the commands for importing and converting data and for exporting processed data to various formats.



Create Menu

Use the Create menu to create feature layers, and to add and edit point, line and area features on those layers.

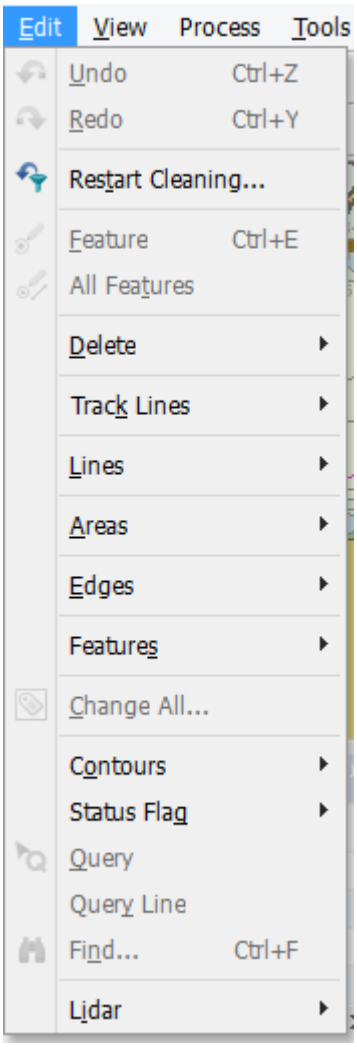


Edit menu

This menu contains editing commands such as Undo, as well as data cleaning commands to reject, accept and query data.

Commands to query tile soundings and to create a tile histogram, as well as viewing and editing commands for LIDAR data are also listed.

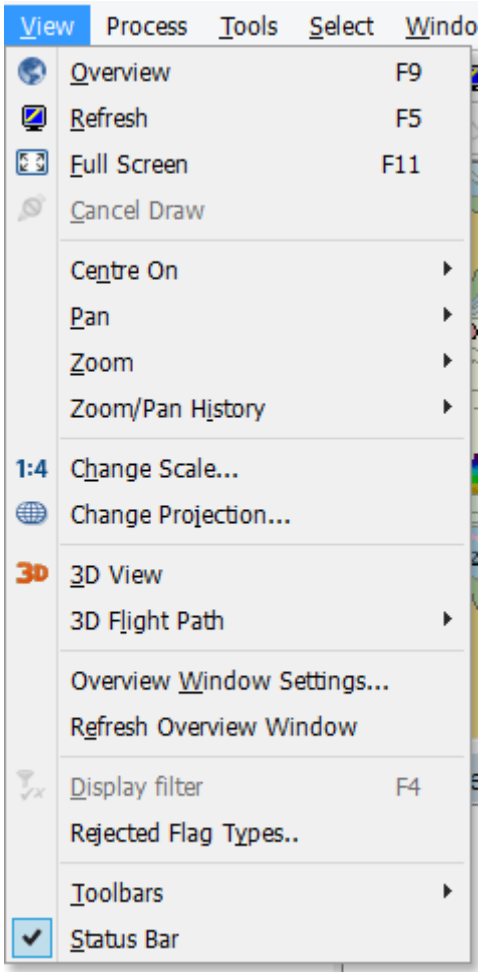
Vessel, SVP and Tide Editors can also be opened from the Edit menu.



View menu

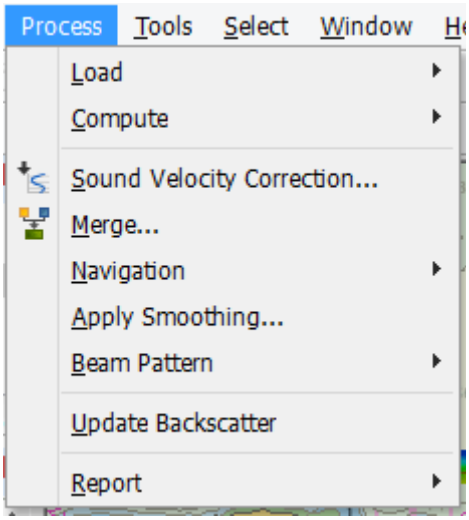
Commands on the View menu enable you to change the display of data with Zoom and Pan, Refresh, Change Projection and Centre on Selection commands.

Commands to display and manipulate data in 3D, and to record 3D “fly-throughs” are also on the View menu as well as commands to display specific data such as rejected data.



Process menu

From the Process menu you can load corrections such as TrueHeave and tide, compute processes such as TPU and SVC and then merge your data. Imagery corrections such as Beam Pattern Correction can be applied from the Process menu.

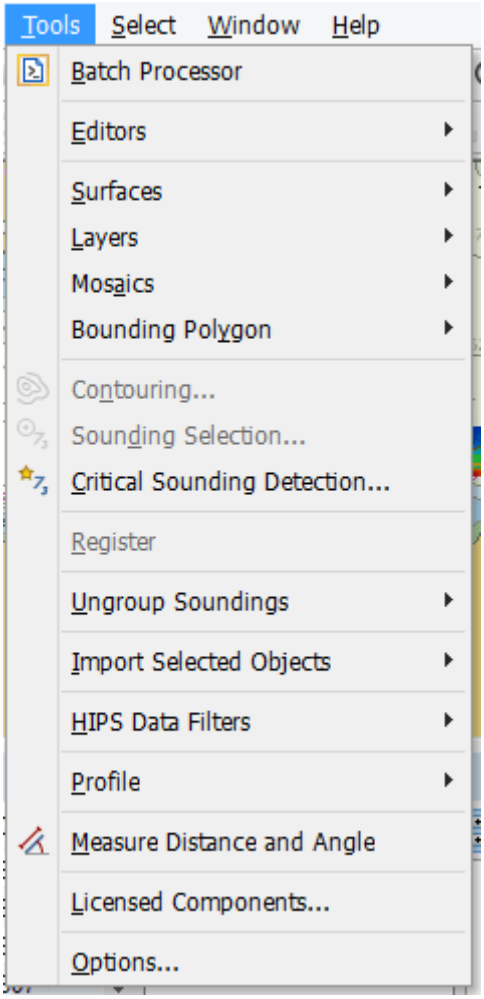


Tools menu

Use the Tools menu commands to run batch processing, interpolation or shifting of navigation data, set and apply filters, measure distances and create beam pattern correction files.

As well, Single Beam, Subset, Swath, Side Scan and Mosaic Editors can be opened from the Tools menu.

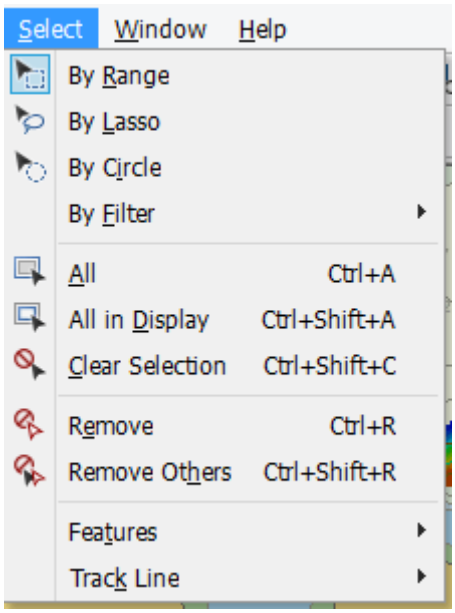
HIPS and SIPS options can also be set from the Tools menu.



Select menu

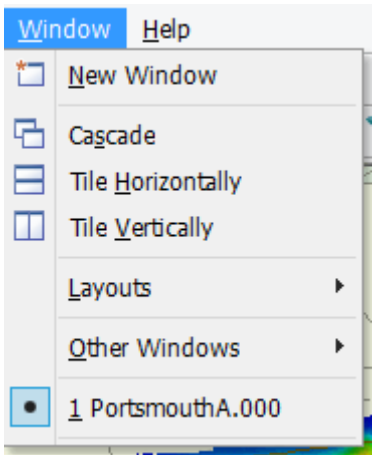
Commands on the Selection menu enable you to select data by range or by lasso, to select all data, or only the data in the display.

There are also commands to enable you to move through data line by line.



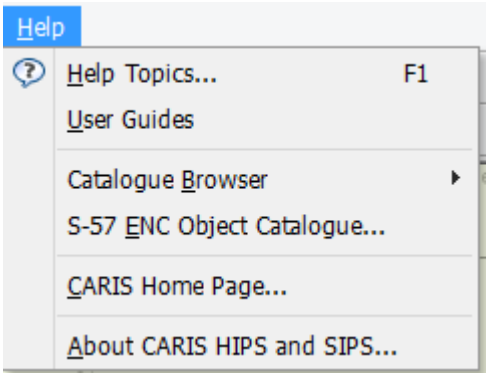
Window menu

From the Windows menu you can display, hide and arrange the main HIPS and SIPS windows, view project properties and restore the arrangement of windows in the HIPS and SIPS interface to that of the initial install.



Help menu

From this menu you can open the Help file and view information about the version of HIPS and SIPS you have installed.



Many of these commands on menus are also accessible from toolbars. For a description of these toolbars, see "HIPS AND SIPS TOOLBARS" in the HIPS and SIPS Tools Guide.

Toolbars

Toolbars enable swift access to commonly used commands. Toolbars can be undocked and moved about the desktop or hidden from view until needed.

The HIPS and SIPS toolbars are:

- **Standard:** activate standard Windows commands such as opening and saving files.
- **View:** activate functions for controlling the view in the Display window.
- **Selection:** activate functions for selecting objects in the Display window.
- **Tools**: activate functions for opening editors and for processing data.
- **Window**: activate commands for toggling windows to on or off.
- **Coordinates**:
- **Filters:** activates functions for defining and executing automatic filters.
- **Feature Creation:** contains tools to create layers, pint line and area features
- **Editing:** activate functions used for editing line, point and areas features.
- **Advanced Editing**: contains such functions to divide lines, merge liens and areas and change line direction
- **Status Edit**: Contains tools such as Auto Cursor, editing, rejecting and designating
- **Process:** activate functions for executing processes such as creating BASE surfaces and merging.
- **HIPS Data Filters**: contains tools to set and apply filters
- **Swath Editor:** activate functions for examining and cleaning pre-merged sounding data.
- **Water Column Editor**: contains tools for viewing WCI data
- **Subset Editor:** activate functions for examining and cleaning sounding data in the subset mode.
- **LIDAR** activate functions for selecting and examining soundings recorded by a LIDAR system.
- **Side Scan Editor:** activate functions for examining data, slant-range correcting raw side data, and creating contacts

- **Beam Pattern:** activate functions for creating and adjusting beam patterns for use in beam pattern correction.
- **Single Beam Editor:** activate functions for examining and cleaning single-beam data.

Display Toolbars

Most toolbars are visible when the application is opened, arrayed below the menu bar. Toolbars can be undocked from this location and positioned within the HIPS and SIPS interface, or on your desktop.

To display toolbars not currently open:

1. Select the View Toolbars command.

Menu	View > Toolbars

2. On the Toolbars sub-menu, click on the toolbar name that you want to display.

The toolbar is displayed the desktop. To hide the toolbar, reverse the process.

Move toolbars

Toolbars can be moved to any location on the desktop.

1. Position the cursor over any area of the toolbar not covered by a button.
2. Press and hold the mouse button while dragging the toolbar to a new location.
3. Release the mouse button to position the toolbar.

Toolbars will automatically dock when they are close to certain areas of the HIPS interface. To stop the toolbar from automatically docking, hold down the <Ctrl> key while moving the toolbar.

To return the toolbar to its previous position in the interface, double-click on the toolbar's title bar.

The Menu bar can be repositioned in the same way as toolbars.

Modify button size

You can alter the appearance and size of toolbar buttons.

1. Select the Customize command.

Menu	View > Toolbars > Customize

The Customize dialog box is displayed.

2. Select any of the following three options by checking a box.
   - *Tool Tips*: Display the name of a command button when the cursor is placed over it.
   - *Cool Look*: Remove the border outline from the buttons.
   - *Large Buttons*: Display buttons with 32 x 32 pixel icons instead of 16 x 16 pixel icons.
3. Click **OK**.

Custom toolbar

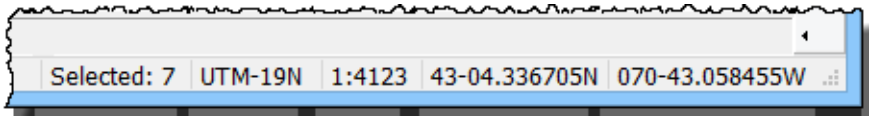
You can also create a custom toolbar (see "CREATE A CUSTOM TOOLBAR" ON PAGE 29 in HIPS and SIPS Tools).

Status Bar

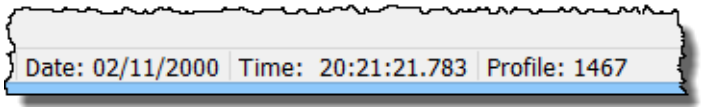
The status bar at the bottom of the HIPS and SIPS application window displays useful information such as:

- the number of track lines currently selected
- the current projection (click the projection name to view details in a pop-up window)
- the scale of the current display in the Display window (when in the 3D Display window this field shows the "height" of the flight above the surface.)
- the coordinates of the current cursor position in the units chosen in Tools > Options (see also "COPY POSITION" ON PAGE 19)

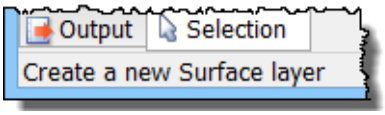
all as illustrated below.



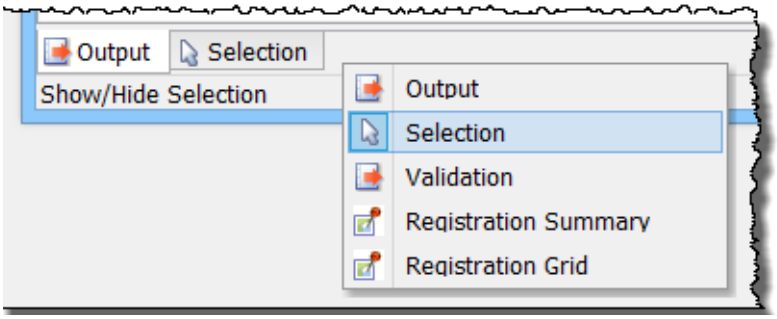
If you have one of the HIPS editors open, the central part of the status bar can also display information such as the survey date for the open project, profile and beam number, as in the example below.



The left end of the status bar will display the function of the command or menu item under your cursor, for example, if the cursor hovers over the command Tools > Surfaces > New, the status bar will display" Create a new surface layer" as in the following illustration.



If you right-click in the tabbed area of windows docked together, for example, the Output and Selection windows, a pop-up menu will show you the list of other windows that can be opened in that space.



Status Bar options

You can hide the status bar:

1. Select the Status Bar command.

Menu	View > Status Bar

The check mark beside the Status Bar option is gone and the status bar is no longer visible in the desktop.

Cursor coordinates

The units for the coordinates of the cursor's current position are set in the Options dialog box of the Tools menu. You can use the Units option to set the type of unit e.g. Geographic or Ground.

The Position Precision option is used to set the number of places to the right of decimal point in the coordinate display.

For further information see "DISPLAY UNITS" ON PAGE 176.

2

Open Data

Geo-referenced TIFF files and other formats can be opened as backdrop to the Display window. S-57 files can also be opened in the HIPS interface.

In this chapter...

Open Data

HIPS and SIPS can open any of the following data types in the Display window:

Format	Extension	Format	Extension
Autodesk AutoCAD DXF/DWG	.dxf and .dwg	GRIdded Binary	.grb, .grib, .grib2
BSB	.kap	[GDAL] ASCII Gridded XYZ[a]	.xyz
Bathymetry Attributed Grid	.bag	Geography Markup Language Files	.gml
Bentley/Intergraph DGN	.dgn	Graphic Interchange Format	.gif
CARIS File	.des	Hydrographic Chart Raster Format	.chr
CARIS HCS	.hcs	Intergraph Raster	.cot, .crl, .rle, .cit
CARIS HIPS & SIPS Project File	.hips, .hpf	Joint Photographic Experts Group 1992	.jpg, .jpeg
CARIS HIPS Weighted Grid	.def	Joint Photographic Experts Group 2000[2]	.jp2, .j2k
CARIS HNS	.hns	Mapinfo	.mif
CARIS HOB	.hob	MrSID Seamless Image Database[b]	.sid
CARIS IGA	.iga	Multi-band GeoTiff[c] (for satellite imagery)	.tif
CARIS Multi-resolution Raster	.bms	NOAA NGS Geoid Height Grids	.bin
CARIS Reference Model	.xml	National Imagery Transmission Format	.ntf
CARIS SAF	.saf	NetCDF Coards Format	.nc
CARIS Shoal	.shl	NetCDF GMT	.gmt
CARIS Spatial ARchive	.csar	Portable Network Graphics	.png
Digital Elevation Map	.dem	S-57 Dataset	.000
ERMapper Enhanced Compression Wavelet[2]	.ecw	Tagged Image Format	.tiff, .tif
ESRI ASCII Grid	.asc	VPF Browse Library	dht
ESRI Binary Grid	.adf	VPF Database	lat
ESRI Floating Point Grid	.flt	VPF Library	lht
ESRI Shapefile	.shp	Windows Bitmap	.bmp

a Details of the GDAL XYZ format specification is available from the GDAL website at http://gdal.org/frmt_xyz.html.
b A plug-in is required to use this file format. The plug-in is available from the CARIS web site, under "Free Downloads".
c This format is typically used to store bathymetry derived from satellite imagery.

Menu	File > Open

To open files in any of these formats:

1. Select the Open command.

The Open dialog box is displayed.

2. Click *All Support Formats* to view the list of formats that can be opened through this dialog box.

Multiple instances of various types of supported files can be opened at the same time if they are located in the same folder.

3. Select one or more files within a folder.
4. [Optional] Use the *Files of type* drop-down list to filter selection choice to files of a specific type.
5. Click **Open**.

The selected files are opened in the Display window and are listed as layers in the Layers window.

Open Datum model file

To open a datum model file:

Menu	File > Open

1. Select the Open command.
2. Select an XYZ file (must have the `.xyz` extension) and click **Open**.

This opens a dialog box so you can select a format description (*.info) file, used to parse the information from the XYZ file. See "INFO FILE" ON PAGE 248 for an example and description of this file.

3. Select the `.info` file that describes the format of the selected.xyz file.
4. Click **Open**.
5. This opens the Select Projection dialog box.
6. Select the Coordinate system that describes the position format in the `*.xyz` file. Click **OK**.

The model file is displayed as a TIN (triangulated terrain model) data in the Display window, using the default *Rainbow* colour map.

Open VPF file

Vector Product Format (VPF) files require additional information before they can be opened. You can open a VPF file by selecting:

- The database header table (DHT). "OPEN A DHT FILE" ON PAGE 58
- The library header table (LHT). "OPEN A LHT FILE" ON PAGE 60
- The library attribute table (LAT). "OPEN A LAT FILE" ON PAGE 62

Open a DHT file

Menu	File > Open
Tool	
Key	<Ctrl+O>

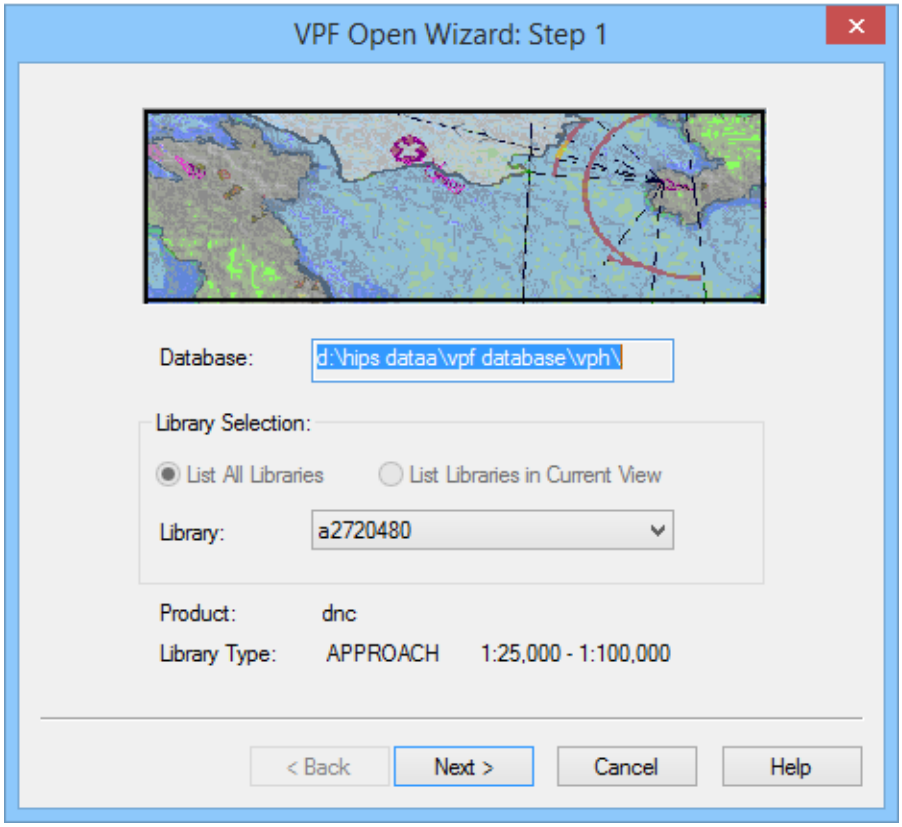
To open a VPF file from a database header table (DHT):

1. Select the **Open** command.

In the Open dialog box:

2. [Optional] Select VPF Database (dht) from the *All Supported Formats* list.
3. Select the DHT file and click **Open**.

Step 1 of the VPF Open Wizard is displayed.



Option	Description
Database	The path and name of the database directory. This field is read-only.
Library Selection	Choose which libraries to list in the *Library* option field: • List All Libraries: All available libraries are listed. This is the default setting. • List Libraries in Current View: Only libraries that contain tiles within the extents of the open dataset are listed.
Library	Select a library from the list.
Product	The VPF product. Displayed once a library is selected. This field is read-only.

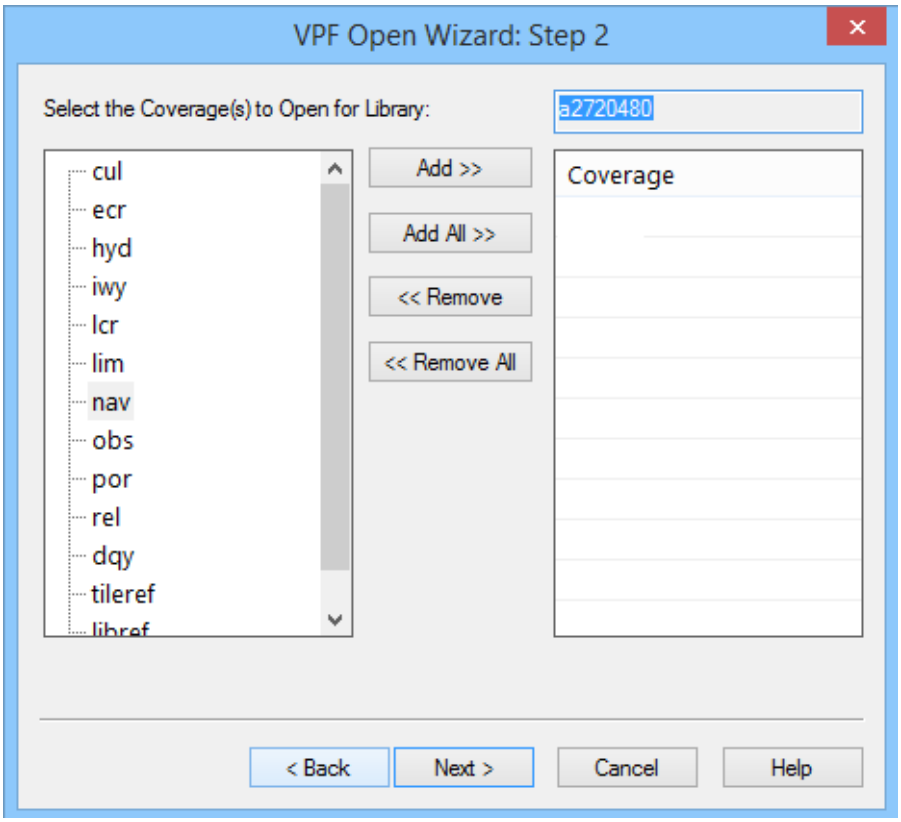
Option	Description
Library Type	The chart type that will be displayed when the VPF is opened. This field is read-only.

1. Choose the libraries to list.
2. Select a library from the list.

The VPF product and chart types are displayed.

3. Click **Next**

Step 2 of the VPF Open Wizard is displayed.



Here you select the coverages to be opened in the editor. The list on the left contains all available coverages. The list on the right contains the coverages that will be opened.

1. Do one of the following:
   - Select one coverage in the list on the left and click **Add.**
   - Click **Add All.**

The selected coverages are added to the list on the right.

To remove coverages:

- Select a coverage on the right and click **Remove.**

This removes the selected coverage.

- Click **Remove All**. This remove all coverages from the list.

Once all appropriate coverages are listed:

2. Click **Next**.

Step 3

Step 3 of the VPF Open Wizard is displayed. Here you select the tiles to be opened.

The list on the left contains all available tiles. The list on the right contains the tiles that will be opened.

1. Do one of the following:
   - Select one tile in the list on the left and click **Add**.
   - Click **Add All** to add all tiles to the Tile Name list.
   - Click **Add Adjacent** to add tiles that are touching the selected tile.
   - Click **Add in View** to add tiles that fall within the extents of the current dataset.

The selected tiles are added to the Tile Name list.

To remove tiles:

- Select a tile on the right and click **Remove**. This removes the selected tile.
- Click **Remove** All. This remove all tiles from the list.

Once all appropriate tiles are listed:

2. Click Finish.

The VPF dataset is opened.

Open a LHT file

Open a VPF dataset using the library attribute table. This method provides a graphical means to select and open VPF datasets.

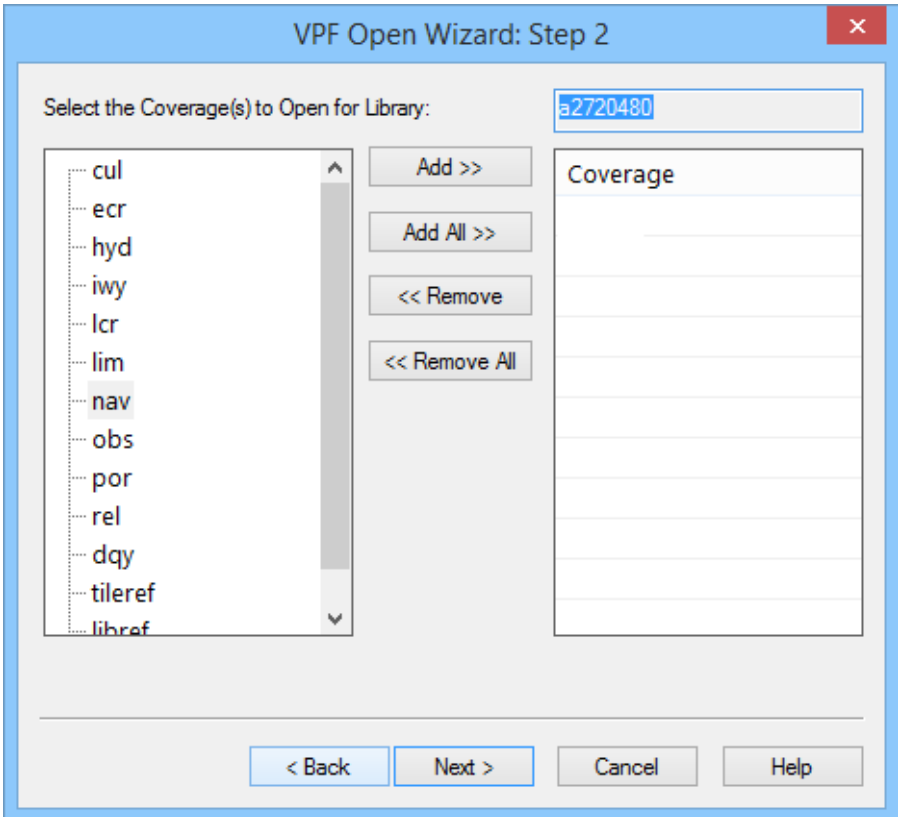
Menu	File > Open
Tool	
Key	<Ctrl+O>

1. Select the **Open** command.

In the Open dialog box:

1. [Optional] Select VPF Browse Library (`lht`) from the *All Supported Formats* list.
2. Select the LHT file and click **Open**.

The VPF Open Wizard Step 2 dialog box is displayed.



Here you select the coverages to be opened in the editor. The list on the left contains all available coverages. The list on the right contains the coverages that will be opened.

1. Do one of the following:
   - Select one coverage in the list on the left and click **Add.**
   - Click **Add All.**

The selected coverages are added to the list on the right.

To remove coverages:

- Select a coverage on the right and click **Remove.**

This removes the selected coverage.

- Click **Remove All**. This remove all coverages from the list.

Once all appropriate coverages are listed:

2. Click **Next**.

Step 3 of the VPF Open Wizard is displayed. Here you select the tiles to be opened in the editor.

The list on the left contains all available tiles. The list on the right contains the tiles that will be opened.

1. Do one of the following:
   - Select one tile in the list on the left and click **Add**.
   - Click **Add All** to add all tiles to the Tile Name list.

- Click **Add Adjacent** to add tiles that are touching the selected tile.
- Click **Add in View** to add tiles that fall within the extents of the current dataset.

The selected tiles are added to the Tile Name list.

To remove tiles:

- Select a tile on the right and click **Remove**. This removes the selected tile.
- Click **Remove** All. This remove all tiles from the list.

Once all appropriate tiles are listed:

2. Click **Finish**.

The VPF dataset is opened.

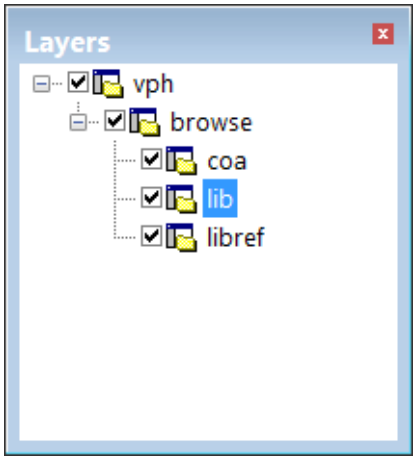
Open a LAT file

Open a VPF dataset using the library attribute table. This method provides a graphical means to select and open VPF datasets.

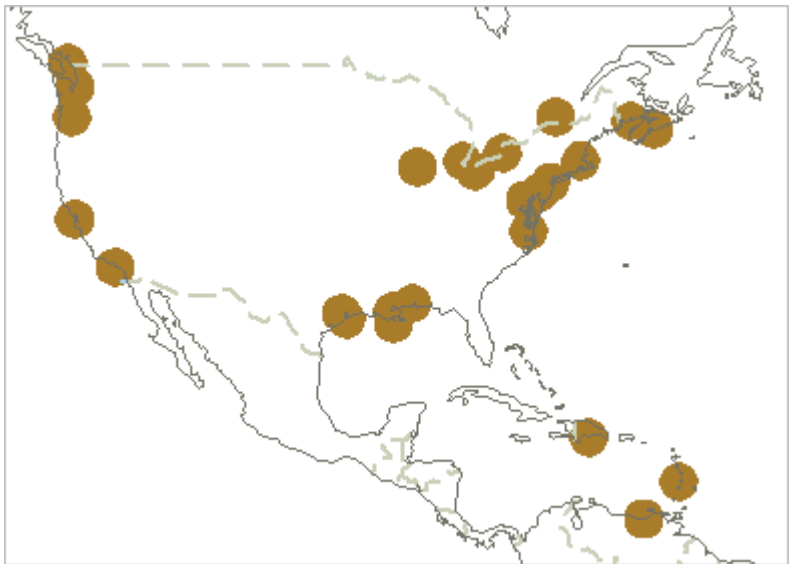
In the Open dialog box:

1. [Optional] Select VPF Browse Library (.lat) from the Files of type list. \
2. Select the LAT file and click **Open**.

The VPF Browse library is loaded to HIPS. Layers associated with the browse library are displayed in the Layers window.



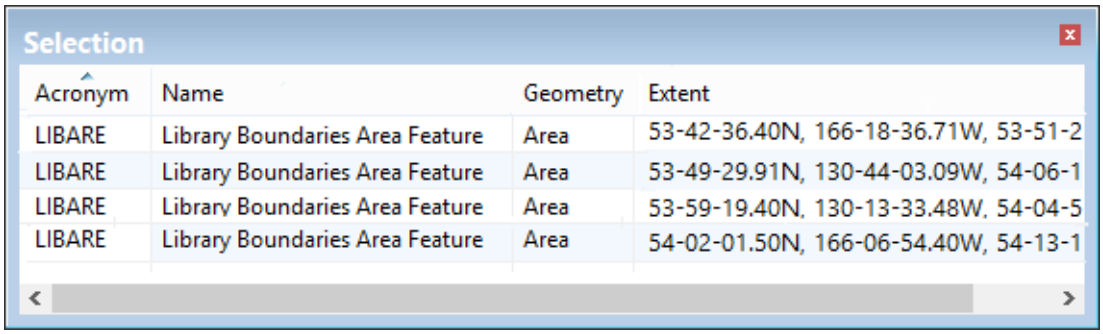
An outline of the area where VPF data is available is displayed in the Display window.



To view data:

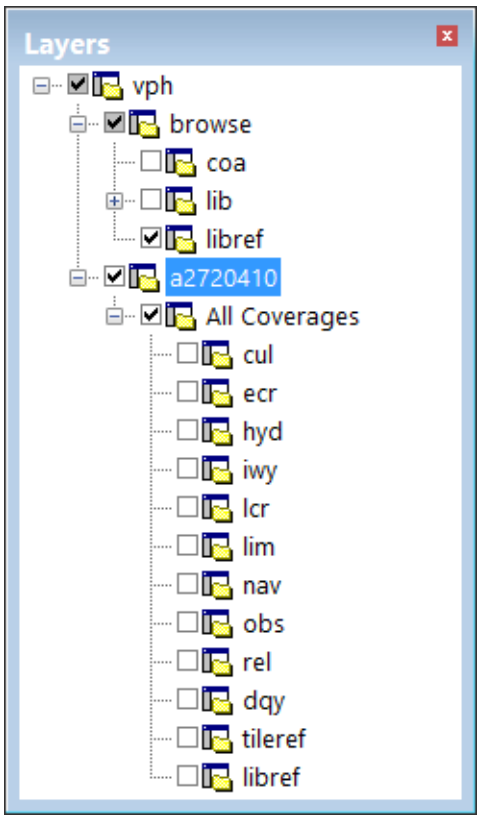
3. Select the *lib* layer in the Layers window.
4. In the Display window, select the area containing the data you want to view.

The selected area is highlighted in the outline, and the LIBARE library boundary areas are displayed in the Selection window.



5. Click on a row to superselect a library boundary area.
6. In the Selection window, right-click in the superselected library row and select Open Selected VPF Libraries from the pop-up menu.

The area in the outline represented by the library boundary is highlighted. The VPF dataset is opened in the outline. The libraries and coverages associated with the LIBARE feature are displayed in the Layers window.



7. Zoom into the dataset to get a better view of the data.

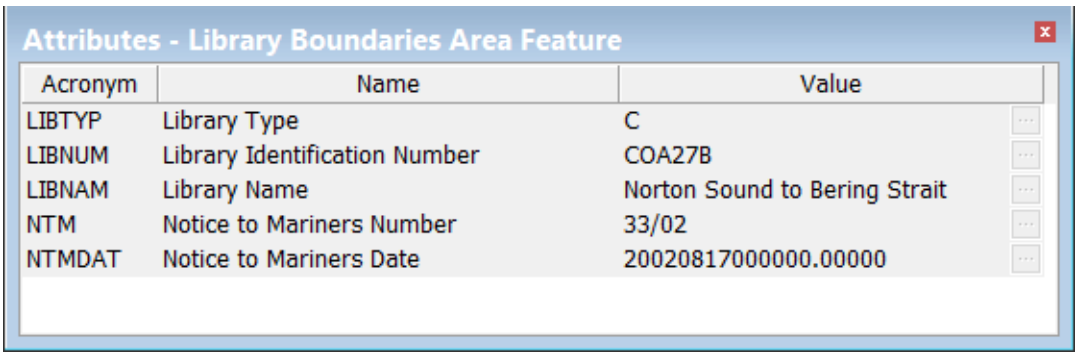
You can control the data display by clearing or selecting the coverage and library check boxes in the Layers window.

To view attribute information for a VPF layer:

select a layer in the Layers window

select a row in the Selection window

The selected row is displayed in the Attributes window. The following example shows the attributes of a selected LIBARE feature.



Use Coordinate System

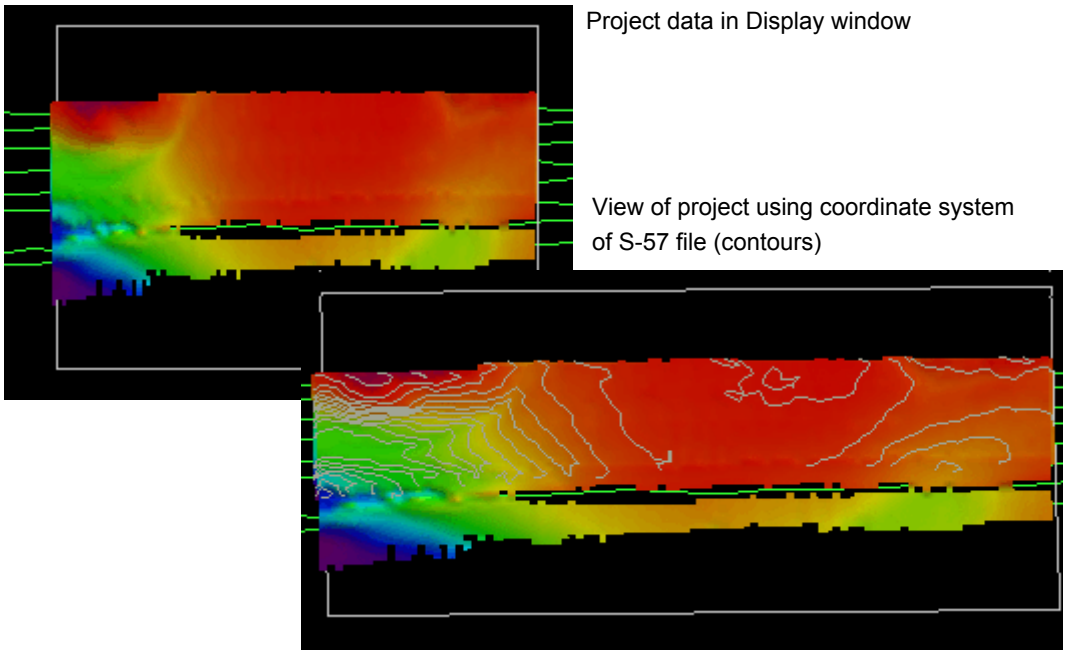
You can apply the coordinate system used by a background layer to the view of the data in the Display window.

Pop-up	Use Coordinate System

1. Right-click on the open background layer.
2. Select the Use Coordinate System command from the pop-up menu.
3. Refresh the display.

The view of the data now reflects the coordinate system of the background layer instead of that defined in the project file.

The following shows the change to the display when the coordinate system of an S-57 file, open as background, is applied to a project that had a different projection.



Project data in Display window

View of project using coordinate system of S-57 file (contours)

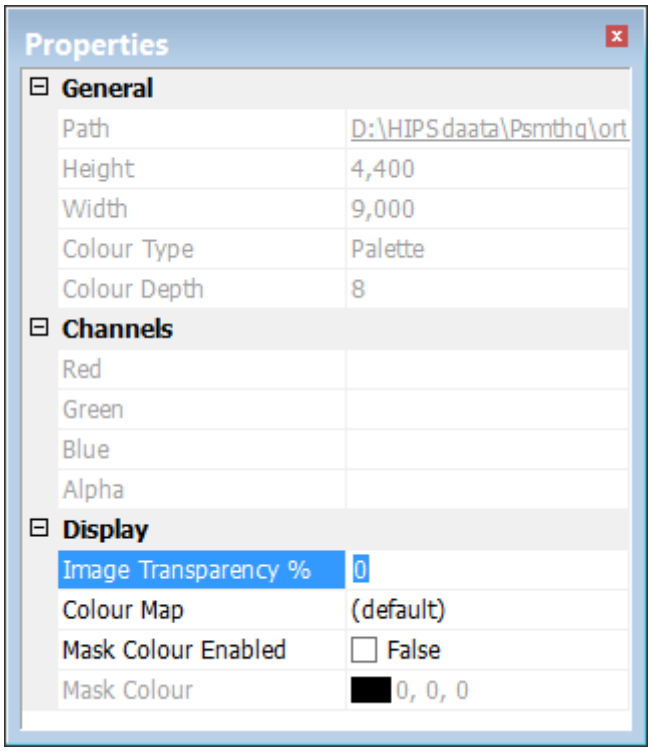
To restore the project coordinate system:

1. Right-click on the project layer.
2. Select the Use Coordinate System command from the pop-up menu.
3. Refresh the display.

Image Properties

Image files which can be opened as background layers (TIFF, TFW, raster images, etc.) display properties in the Properties window. To view the image properties:

1. Select the image layer in the Layers window.
2. Open the Properties window.



The Properties window automatically lists the display options for the selected image layer.

The *General* fields display read-only data such as the location of the image file, and its dimensions.

The *Display* fields adjust transparency and masking for the background image.

The *Image Transparency* field controls the transparency of the entire image: the higher the percentage value, the more transparent the image will appear. The default is 0, (no transparency).

3. To make the entire background image transparent, type a *Transparency* percentage.

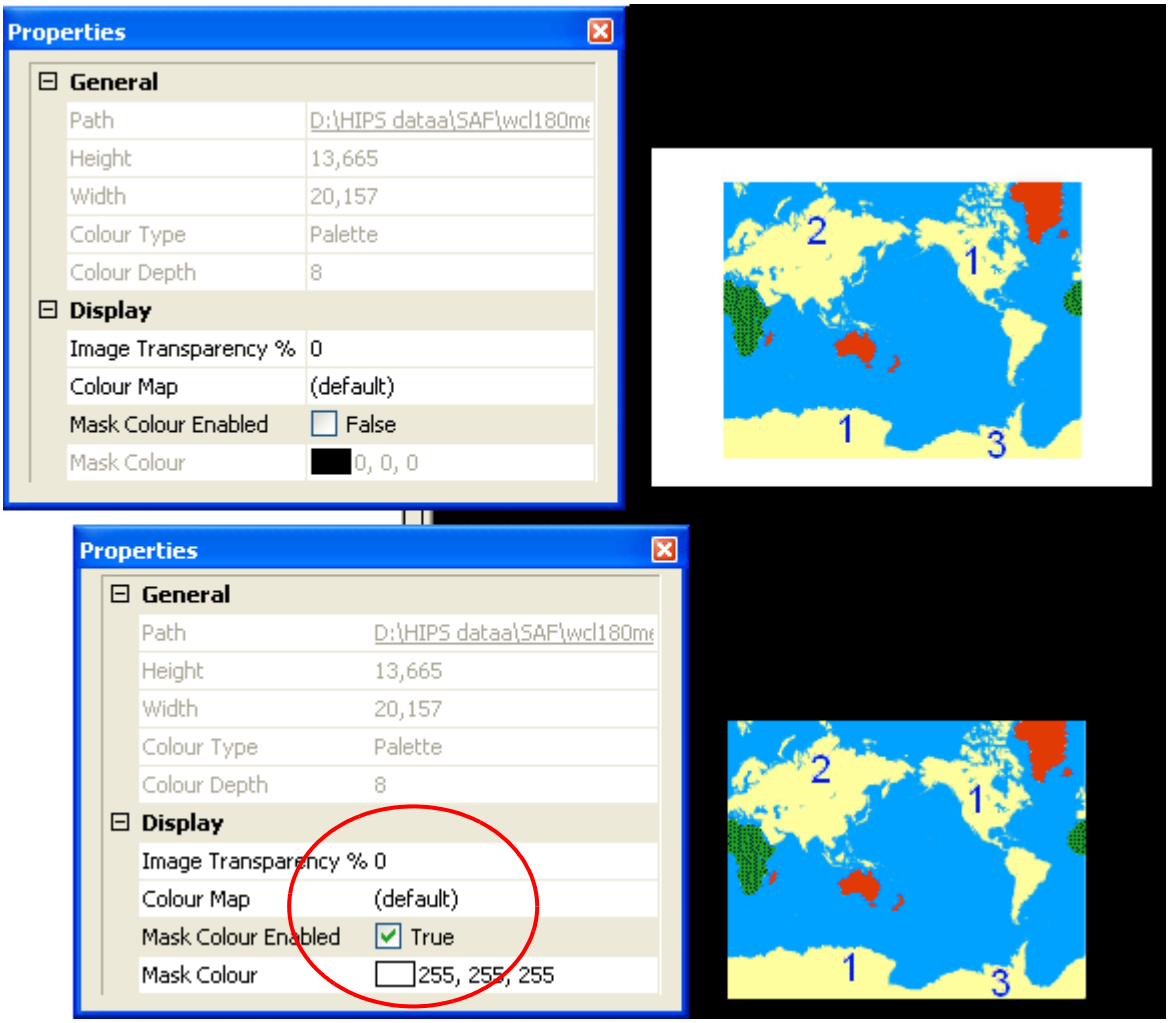
The *Colour Map* field controls the colour display of the image. The value `(default)` indicates that the image is displayed in its original colour mapping. You can apply another colour map by selecting it from the drop down list.

The masking fields make a single colour transparent. This is useful if the image background is different from the colour of the Display window.

To mask a particular colour:

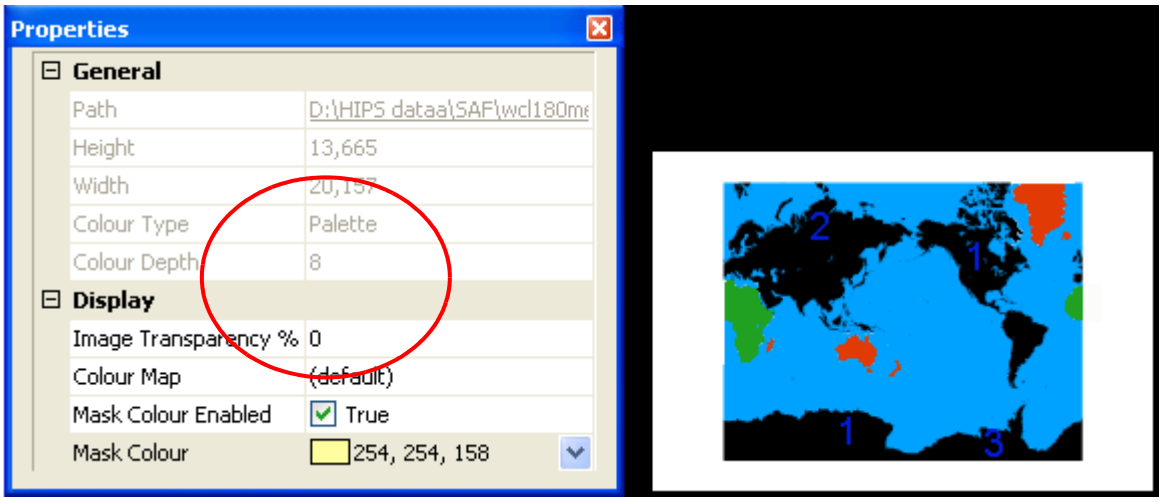
4. Set *Mask Colour Enabled* to True.
5. Select the colour to be made transparent from the *Mask Colour* drop-down list, and refresh the Display.

In the example below, the top image shows a GeoTiff with white background in the Display window, which has its background set to Black. The bottom image shows how masking the white background colour makes it transparent.



The image masks another colour, in this case, the yellow areas.

Only one colour can be masked at a time. In the image below, the white background is not masked while the yellow areas are masked


Properties
General
Path
D:\HIPS dataa\SAF\wcl180m
Height
13,665
Width
20,157
Colour Type
Palette
Colour Depth
8
Display
Image Transparency %
0
Colour Map
(default)
Mask Colour Enabled
True
Mask Colour
254, 254, 158
2
1
1
3

View soundings in HOB file

To display the soundings stored in a HOB file, you must first set the display options in the S-52 tab of the Tools > Options dialog box.

1. Open the Tools > Options dialog box and select the S-52 tab.
2. In the *Display filters* section, select “Base + Other” or “Standard + Other” from the *Category* drop-down list.
3. Select the *Soundings* check box.

Menu	File > Open Background Data

4. Select the Open Background Data command.
5. Browse to the HOB file where the soundings are stored.
1. Click **Open**.

The HOB file layer is displayed in the Layers window, and the soundings are shown in the Display window.

Open URL

This function enables you to open data available through web services. Currently supported services are:

Service	Description
Web Map Service	A Web Map Service (WMS) is an online portrayal service for geospatial data that returns raster images.
Web Map Tile Service	A Web Map Tile Service (WMTS) is a tile-based online portrayal service for geospatial data that returns raster images. Tiles are retrieved at a resolution based on the current view scale, similar to CSAR data. Tiles will be retrieved in their entirety, regardless of the current view extents.
Tile Map Service	A Tile Map Service (TMS) is a tile-based online portrayal service for geospatial data that returns raster images, similar to a WMTS, but with different standard protocols. The default tile size returned from a TMS service is 256 x 256 pixels. A well-known provider of a TMS is OpenStreetMap, however, users should be aware of their Tile Usage Policy before using this service. Information about this policy can be found at: http://wiki.openstreetmap.org/wiki/Tile_usage_policy
Web Coverage Service	A Web Coverage Service (WCS) allows for opening coverages from a server. Coverages are geospatial representations of map data retrieved as a raster. These may be image rasters or may contain depth information.
Enhanced Compression Wavelet	The Enhanced Compression Wavelet (ECW) Service allows remotely opening ECW and JPG2000 rasters available from a server.
Seafloor Information System	The Kongsberg Seafloor Information System (SIS) supports managing surveys across a network. This allows for remote access of raster data.
Onboard™ Service	The Onboard Service enables viewing of a product such as a mosaic or gridded surface as it is being updated in real time during data processing in Onboard.

Some services provide different sources in different versions, but not all versions are supported in HIPS and SIPS. Currently supported versions are:

- WMS 1.0.0, 1.1.0, 1.3.0
- WMTS 1.0.0
- WCS 1.0.0, 1.1.0, 1.1.1

Contact CARIS Customer Service if you have issues with the version used by the server that you are accessing.

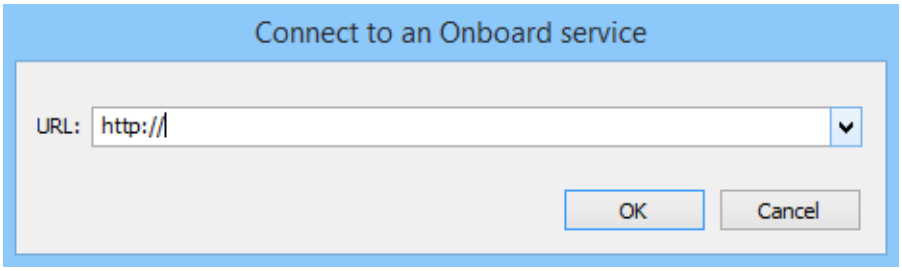
If you normally connect to a proxy server to access your network or Internet browser, you may need to define the *Connection* settings in *Options* to be allowed to connect to the proxy server. Contact your system administrator for assistance.

To access data via the web services:

Menu	File > Open URL > [*service name*]

1. Select the command for the desired service.

The dialog box for the selected service is displayed.

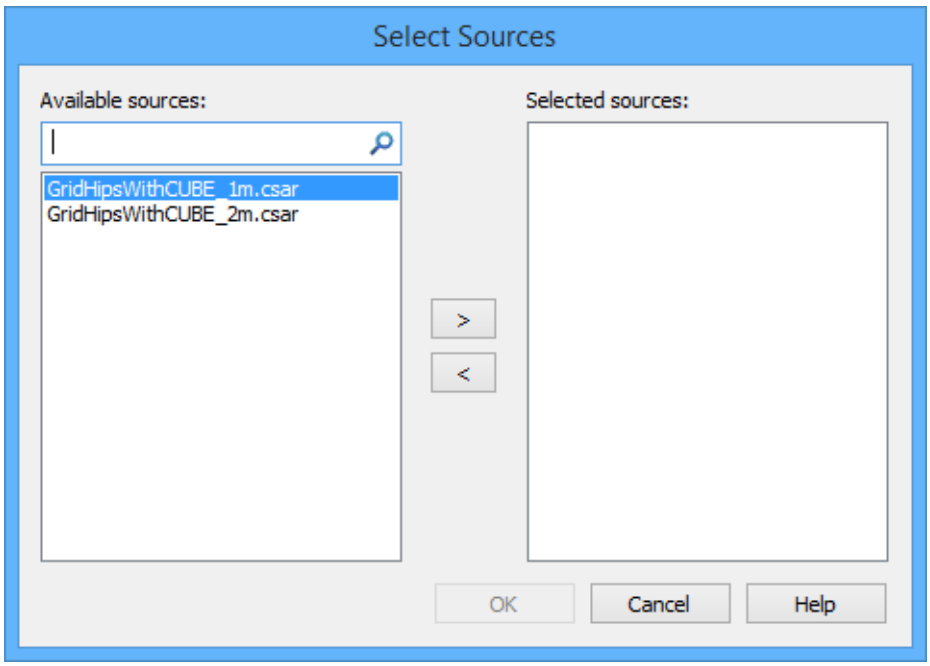


The URL field is used to specify the address of the service to which you want to connect. Each time that an address is entered and run, it will be saved in the drop-down list of that service for future use. Each service can remember up to 10 addresses.

If you wish to remove a saved address, right-click the address in the drop-down list and choose *Delete*.

2. Type a valid address in the *URL* field or select an address from the drop-down list.
3. Click **OK**.

For the WMS, WMTS, WCS, SIS and Onboard service types, the Select Sources dialog box will be displayed if more than one data source is available from the service. If only one source is available, this step is skipped.



In this dialog box, you can choose to open multiple sources or just one. All available sources for the specified URL are displayed.

Each source available from a service can have unique data extents. Data will display according to the extents of the selected source and/or the current extents of the application, depending on whether data was open in the application prior to connecting to the service.

If data was open, the service will only display data within the extents of the current view. If no data was open, the selected source will be displayed using the full extents available for that source. For a TMS service, this means the full world map. The view can be changed to access all of the data provided for a service using the Zoom, Pan and Overview tools. The data is updated dynamically anytime the view is changed or refreshed.

For the other service types, data will be returned for the view extents specific to the URL entered.

If you experience problems displaying data from a service, you may need to select the correct coordinate system for the service layer.

The *Search* field can be used to filter the list of sources. As you type in the field, the list will be filtered to show only the sources that contain the value entered.

4. [Optional] Type an entry in the *Search* field to filter the list of map layers.
5. In *Available Sources*, select the layer that you want to open, then click the **right-arrow** button.

This will move the layer into the *Selected Sources* list.

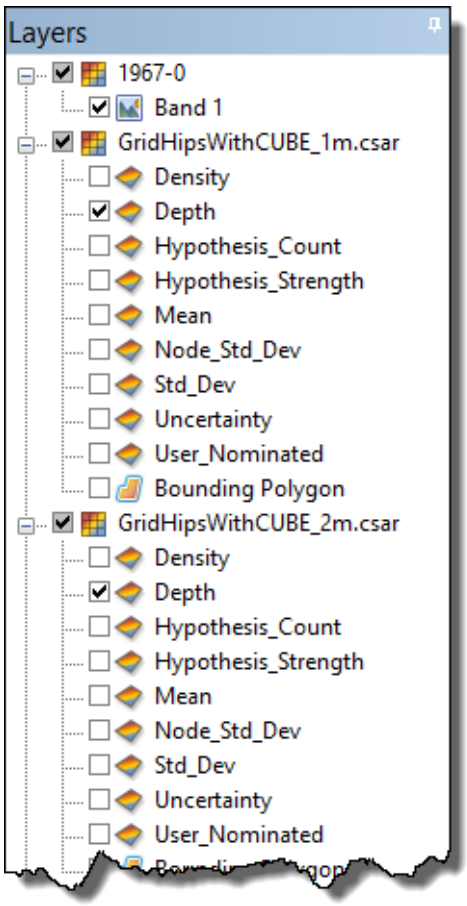
6. Repeat step 5 for each layer you want to open.
7. Click **OK**.

The selected data is opened via the URL. Each source will be opened as a raster image. The format of the image will default to the first available format from the service that is also supported in HIPS and SIPS, however, certain services attempt to use specific formats. For example, WCS will attempt to retrieve a GeoTIFF if GeoTIFFs are available from the server. If this format is not available, it will then look for PNG, then others. WMS will preferentially return PNG as it is an image raster, not data.

If you chose to open multiple sources from a service at once, the images are automatically grouped in the Layers window with the group layer named according to the service being used.

WMS, WMTS and WCS data will be opened in the first coordinate system available from the server that is also supported in the application. TMS is always opened in EPSG:3857 (Google Mercator).

If you selected more than one source, each source will be displayed in the layer tree. You can select the layer or layers of interest and refresh the screen to see the desired data. The example below shows gridded surface products generated from Onboard.


Layers
1967-0
Band 1
GridHipsWithCUBE_1m.csar
Density
Depth
Hypothesis_Count
Hypothesis_Strength
Mean
Node_Std_Dev
Std_Dev
Uncertainty
User_Nominated
Bounding Polygon
GridHipsWithCUBE_2m.csar
Density
Depth
Hypothesis_Count
Hypothesis_Strength
Mean
Node_Std_Dev
Std_Dev
Uncertainty
User_Nominated

Display S-57 Data

HIPS and SIPS is capable of displaying S-57 Ed 3.1 (ENC) and HOB (Hydrographic Object Binary) files. (For information on other data types that can be opened in HIPS and SIPS, see "OPEN DATA" ON PAGE 56.)

To open S-57 Ed 3.1 (ENC) data files:

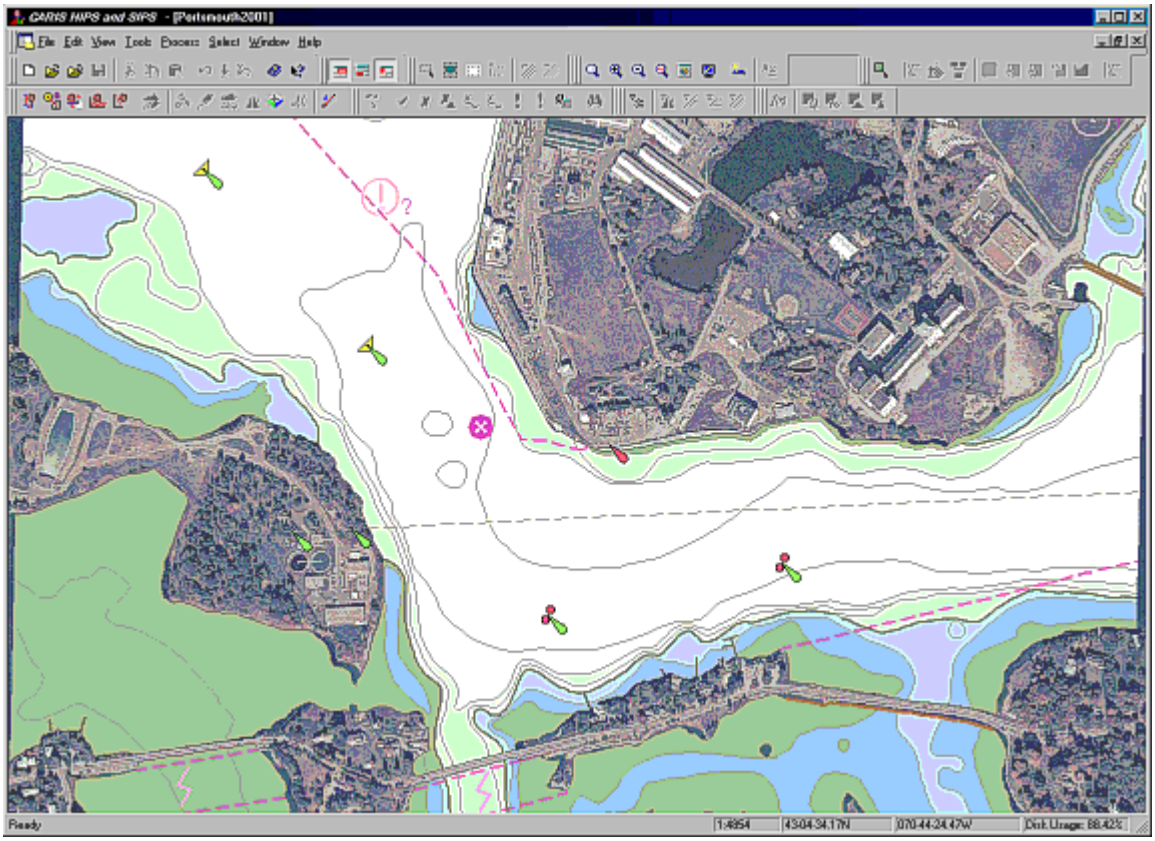
Menu	File > Open Background Data

1. Select the Open Background Data command.

The Open dialog box is displayed.

2. Select S-57Files (*.000) from the Files of type drop-down menu.
3. Browse to the folder containing the S-57 dataset and select the file.
4. Click **Open.**
5. From the Select Catalogue dialog box, select either S-57 ENC 3.1 or S-57 USACE ENC IENC 4.0 from the Name list, whichever is appropriate to the ENC being opened. (The other fields on the dialog box are automatically filled.)
6. Click **OK**.
7. In the S-57 Update Options dialog box, click **OK**.

This will display the selected S-57 dataset in the Display window and list the file in the data tree in the Layers window.



Create Layers of S-57 objects

You can create a layer of feature objects from S-57 data using the filtering function of the Create Layer command.

You can create a layer of all objects with the same attribute value, acronym, ID or type, or use the Rule Wizard to create customized layers. You can also automatically create layers for objects by their unique feature acronyms.

Menu	Create > Layer By

1. Select the S-57 file in the Layers window.
2. Select the Create Layer command.
3. Select an option to filter by:

- "Create Layer by Attribute Value" on page 147
- "Create Layer by Feature Acronym" on page 148
- "Create Layer by Feature Object ID" on page 149
- "Create Layer by Feature Type" on page 149
- "Create Layers by Unique Feature Acronyms" on page 150
- "Create Layer using the Rule Wizard" on page 152
- "Create Layer using a Rule File" on page 165

3

3D Display

3D Display Window

The 3D Display window shows project-level information in three dimensions. Any data layers that are viewable in the HIPS and SIPS Display window can be viewed in 3D. This includes attribute layers for surfaces, vector data and background images such as GeoTiffs.

The 3D Display is displayed in the same location as the data Display. You can view both data and 3D display windows at the same time by using the Cascade or Tiling commands from the Window menu or toolbar.

The 3D Display is enhanced with a “fly-through” effect that gives the effect of gliding through the display. With practice manipulating the fly-through controls, you will be able to zoom and pan in this 3D window. This fly-through can also be recorded to video.(See “3D FLIGHT PATH” ON PAGE 98.)

Menu	View > 3D Display
Tool	

To activate the 3D Display window:

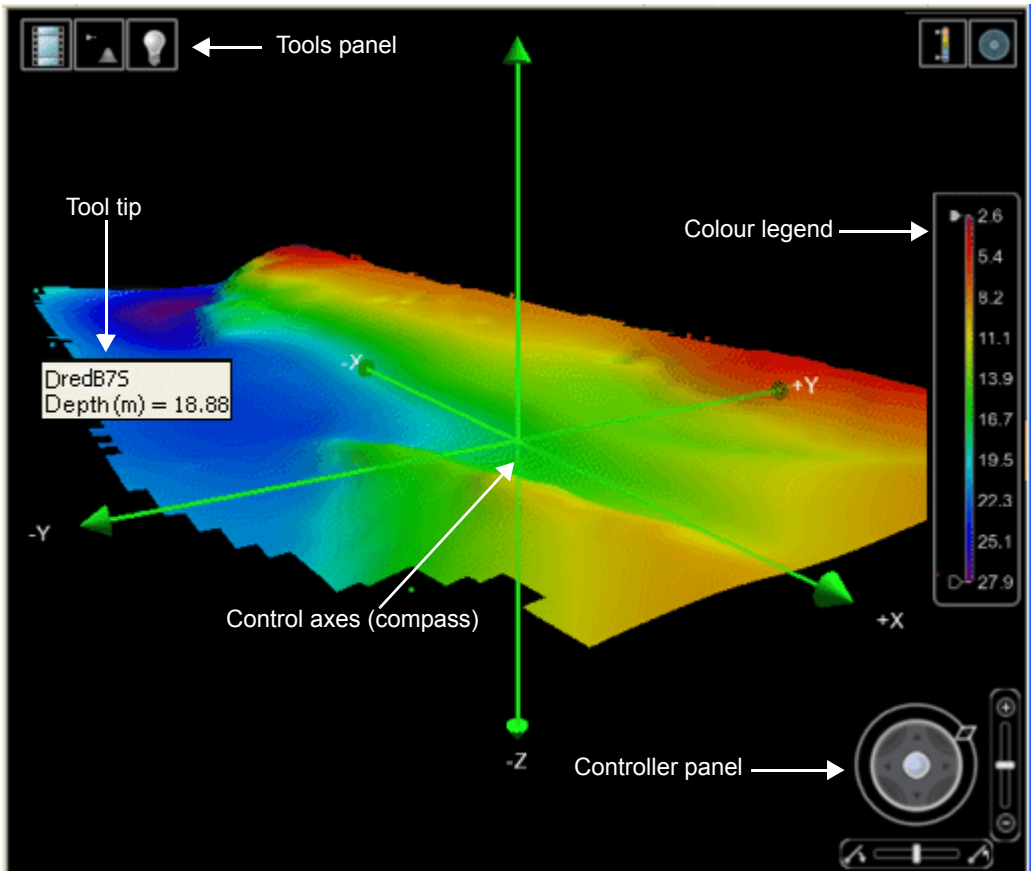
1. Select the 3D Display command or tool button.

When first opened, the 3D window is empty of data. Only the default black background is displayed. Data is displayed in by turning layers on and off in the Layers window. Only those layers which can be viewed in 3D are visible when the 3D window is open.

To display data:

2. Select a layer in the Layers window.

The following image shows a Depth layer opened in the 3D window.



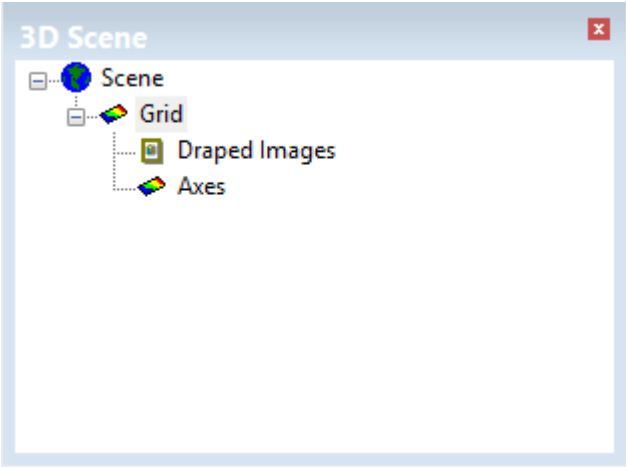
Hover your mouse over the data to display a tool tip giving the name of the data source and the value at the current position of the cursor.

The value is expressed in the units used by the selected layer. In the image above of a Depth layer, the tool tip displays “DredB7S” (the surface name) and “Depth(m)=X” (the depth under the cursor).

The coordinates of the current cursor location are displayed in the HIPS Status bar, on the right. Also displayed in the status bar is the “height” of the flight above the surface.

3D Scene window

The 3D Scene window in the Control window lists the various types of data that are currently open in the 3D window.



Each type of data displayed in the 3D Display has its own layer in the 3D Scene window.

- The Grid layer contains data for the surface being displayed.
- The Draped Images layer contains the data for all draped images.
- The Axes layer contains the Terrain Axes for the surface. The Terrain Axes (displayed as an XYZ grid) are only visible when turned on in Tools > Options.(See "3D DISPLAY" ON PAGE 186.)
- The Vector Data layer contains any vector data.
- The Image layer contains data for open raster images.

Display Options

Display options are set in the Display window of the Tools > Options dialog box.

1. Select Options from the Tools menu.
2. Select 3D from the Display window.

Menu	Tools > Options > Display > 3D

Use the options to turn on and off the display of:

- the *Terrain Axes* (x,y,z) grid for the selected surface (see "TERRAIN AXES" ON PAGE 90.)
- the *Colour Legend* to indicate depths of the surface coloured areas
- the *Controller Panel*

- the 3D View *Toolbar* (these commands can also be activated from the View > 3D Flight Path sub-menu)
- the colour to use as the *Background Colour* of the display
- the *View Style* of the 3D window, for example, the default is Full View (the 3D window occupies the entire Display window). The other options include Vertical or Horizontal Tile, and Cascade.

If the 3D View is open when these options are changed, the new settings do not take effect until the 3D view is closed and reopened.

3D Display Window Properties

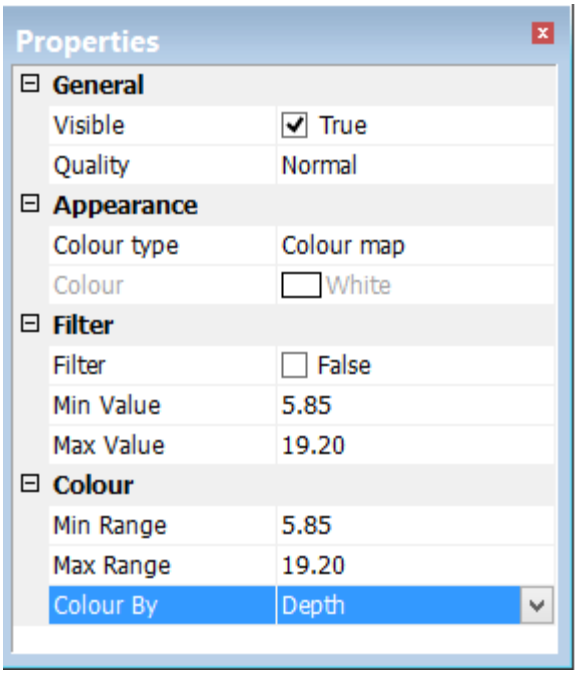
The display in the 3D View can be adjusted using properties.

If you have data visible in both the 3D Display and the Display window, switching back and forth from one window to the other will display different properties. A layer must be activated (its check box selected) and selected (highlighted) to enable properties to be set in 3D.

The following sections provide information on the 3D properties:

Surface Data Properties

1. Select an attribute layer, for example, Depth, in the Layers window to view the available properties of that layer in the Properties window.



General

The *Visible* property enables you to have the data open and included in calculations, without actually having it displayed. This frees up resources to speed up processing time. By default this option is set to *True*.

1. [Optional] Set the *Visible* property to False to hide the selected image.

The *Quality* property enables you to display the surface using a finer or coarser resolution. The default setting is *Normal*.

2. Select the *Quality* property and select an option from the drop-down list

Appearance

The *Appearance* property controls the colour of your 3D surface. You can display the surface using a colour map or using a solid colour. By default the display is set to *Colour Map*.

3. [Optional] Change the display by selecting Solid Colour from the drop-down list and selecting the colour from the colour picker.

Filter properties

The Filter properties can be used to filter data displayed in 3D. To use the filter:

4. Enable the *FIlter* property by clicking the check box.

The value will change to True.

Menu	View > Refresh (F5)
Tool	

5. Enter the *Min* and *Max Value* of the data you want displayed.
6. **Refresh** the display.

Only data that falls within the range of the *Min* and *Max* values will be displayed.

Colour properties

Colour properties enable you to adjust the colour range used to colour the displayed data. (See also "COLOUR RANGE" ON PAGE 88.)

7. Enter the *Min* and *Max Values* to be covered by the colour range.

Using the *Colour By* property, you can choose which attribute will be used as the source data for colouring the display.

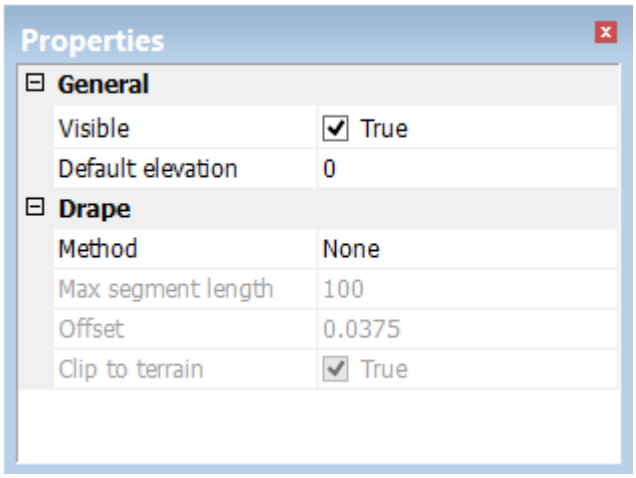
8. Select an option from the *Colour By* drop-down list.

The display is updated to match the selected properties.

Vector Data Properties

To display the 3D display properties for vector data:

1. Select a vector data layer in the Layers window while the 3D Display is active.
2. Open the Properties window.



The *Visible* property enables you to have the data open and included in calculations, without using resources to actually have it displayed. By default this option is set to True.

3. Click the *Visible* check box to display the selected image, if necessary.

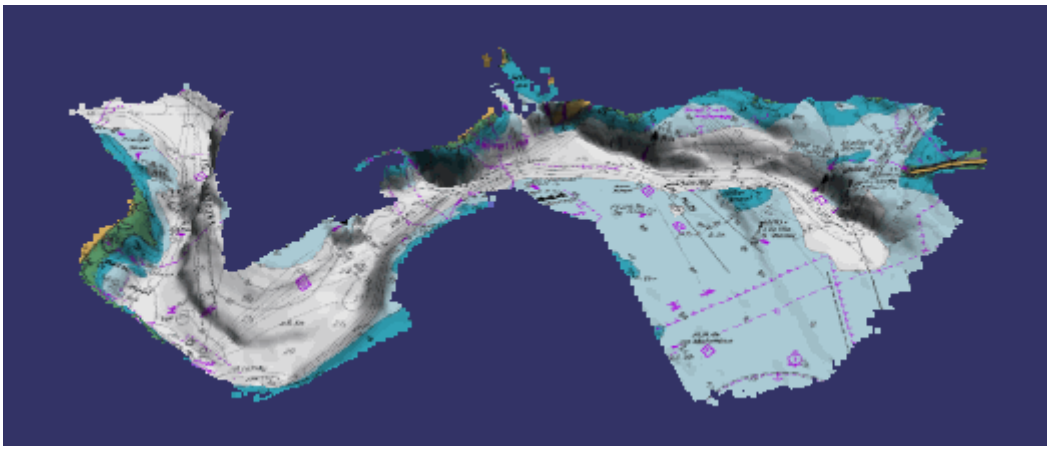
When visible, you have the option of draping data over the surface, so you can accurately view contours and other vector data.

4. Expand the Drape property and in the *Method* field, select one of the following three options:

- None: Data is displayed according to the *Default Elevation* property. With this option, all vector data will be on a level plane unless the data contains its own Z value. The Elevation defines the height at which the data sits above the surface. (The other Draping properties are disabled when None is selected as the method.)Drape: Data is draped using the bathymetry of the surface and the *Offset* property. The bathymetry will apply the proper Z value to the data and the *Offset* will define the height at which it sits above the surface.
- Drape *(use line splitting)*: Data is draped using the same values as the *Drape* method, but line segments are adjusted in length based on the *Max segment length* property. A larger segment length means less vertices and faster processing times. A smaller segment length means more vertices and slower processing times, but a more accurate representation of the elevations. (The *Max segment length* property is only enabled when the *Drape (use line splitting)* method is selected.)

5. Choose an option from the drop-down list.
6. Depending on the selected method, enter any necessary values in the other properties.

The image is draped over the surface.The example below shows a Geo-Referenced TIFF draped over data.



7. [Optional] If the image extends beyond the extents of the data, select the *Clip to terrain* check box to only show the section of the image within the surface extents.

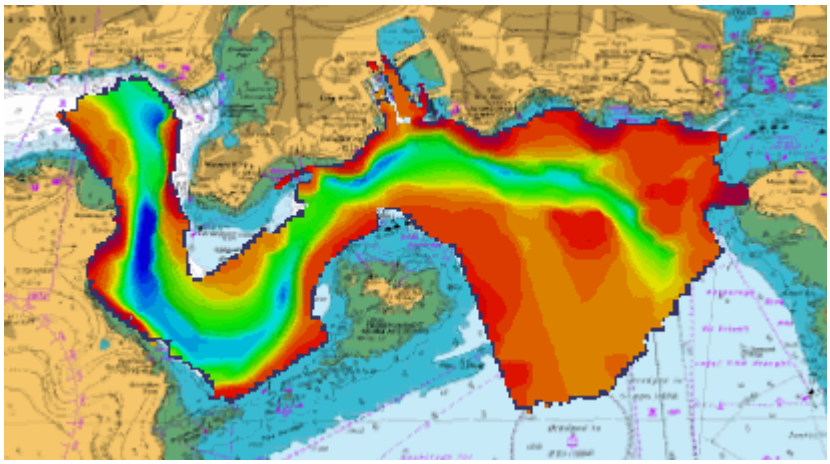
You can also perform the opposite of the drape function where the image is cut away according to the bounding polygon of the surface. This enables you to view a background chart image with the actual surface visible within the chart.

Data that extends beyond the extents of the surface will be displayed according the *Default Elevation* value, regardless of the selected method.

To cut away the image:

8. In the Layers window, right-click the image layer, select Cut Terrain, then select the name of the surface with which you want to cut the image.

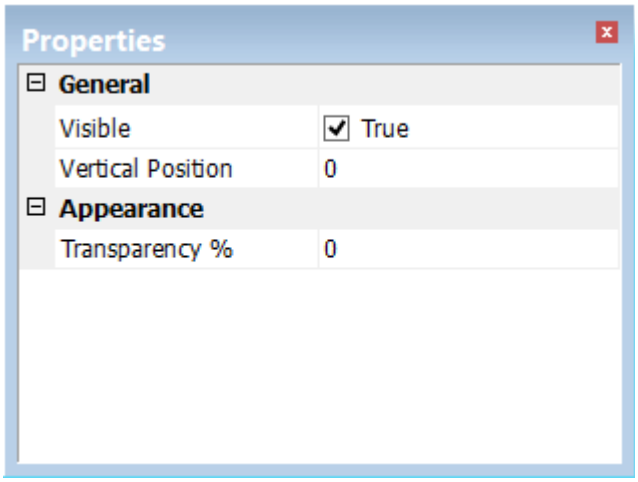
The image is cut and the gap is filled with the surface, as in the example below.



Raster Data Properties

To display the 3D display properties of raster data:

1. Select a raster data layer in the Layers window while the 3D Display is active.
2. Open the Properties window.



The *Visible* property enables you to have the data open and included in calculations, without using resources to actually have it displayed. This option is set to "True" by default.

3. Click the *Visible* check box to display the selected image.

The *Vertical Position* property enables you to adjust the elevation of the image in the display so that it lines up with the surface properly.

4. [Optional] Enter a value in the *Vertical Position* field.

The *Transparency* property enables you to make the image transparent so the surface can be seen in relation to the image.

5. [Optional] Enter a value in the *Transparency* field.

The display is updated to match the selected properties.

Draping Raster Data

Raster images can also be draped over a surface using the bathymetry of the surface.

To drape a raster image:

1. Open a raster image.
2. Select the attribute layer in the Layers window that you want to use to define the draping elevation.
3. Right-click the attribute layer, then select Drape Image and then the name of the raster image to be draped.

The image is draped over the surface and a sub-layer is added to the Layers window for the draped image.

Adjusting the Display

In addition to defining properties, there are various tools within the 3D Display that can also be used to adjust the display of data, such as, lighting, vertical exaggeration, colour range and control Axes that rotate the data.

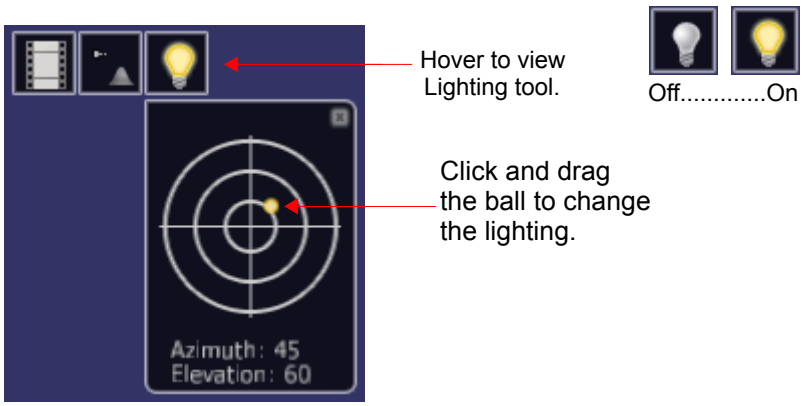
Lighting

The lighting control handles shading effects by controlling the position and angle of the light. The elevation value (displayed when the tool is selected) varies from 0º to 90º starting from the middle of a sundial. The azimuth value (also displayed when selected) varies from 0º to 360º in a clockwise direction.

To turn on the Lighting Control:

1. Pass your cursor over the Light Control button in the Tools panel located in the top-left corner of the 3D Display.

The Lighting tool displays the current azimuth and elevation values.



2. Click and drag the ball to change the lighting.

As you drag, the values in the tool will be updated and you can see the affect on the data.

3. Release the mouse when the display has the proper lighting.
4. To close the tool, click the X in the top-right corner of the box.
5. To re-open the lighting tool, click the light bulb.

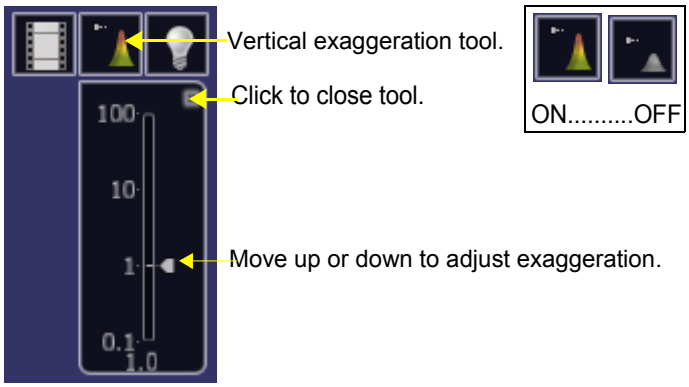
Vertical Exaggeration

Use the Vertical Exaggeration tool to increase or decrease the vertical exaggeration of the data to display seabed features more clearly.

To access the Vertical Exaggeration tool,

1. Hover the cursor over the middle button in the Tools panel (located in the top-left corner of the 3D Display).

The tool activates and a slider bar is displayed below the panel.



2. Click the arrow on the side of the slider and move it up or down the bar.

As you move the arrow, note the change in the display.

3. Release the arrow when the data has the appropriate vertical exaggeration.
4. To close the tool, click the X in the top-right corner of the box.

Colour Range

The colour legend slider bar located along the right side of the 3D Display adjusts the colour range. The arrows on the side of the legend represent the *Min* and *Max Values* of the colour range.

To display the colour legend:

1. Click the Colour Range control.

To change the colour range using the colour legend:

1. Select the control arrow at the top or bottom of the legend.
2. Drag the control to the desired Min or Max value.

The colours in the display will change as you drag the slider.

3. To reverse the colours, drag one control past the other.


Off.....On

The legend can be turned on or off using the **Colour Legend** button in the top-right corner of the 3D View.

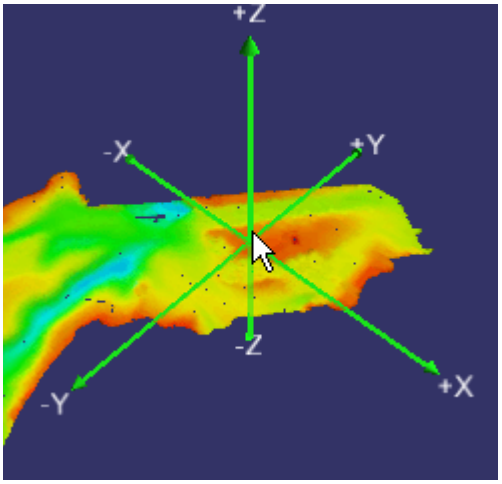
When the button is coloured, the legend is active.

Control Axes

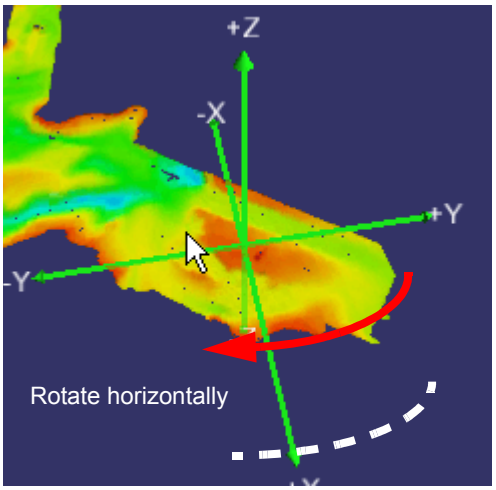
You can manipulate the view along the XYZ axes with the Control axes. Sometimes called compass controls, these are displayed as an overlay to the surface.

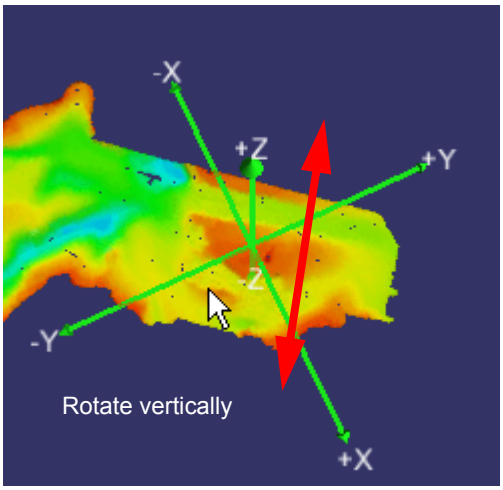
1. While pressing the middle mouse button, click on the surface in the Display.

The control axes appear. They are centred on the point at which you clicked the surface.



If you continue to press the middle mouse button you can tilt the display in three dimensions.





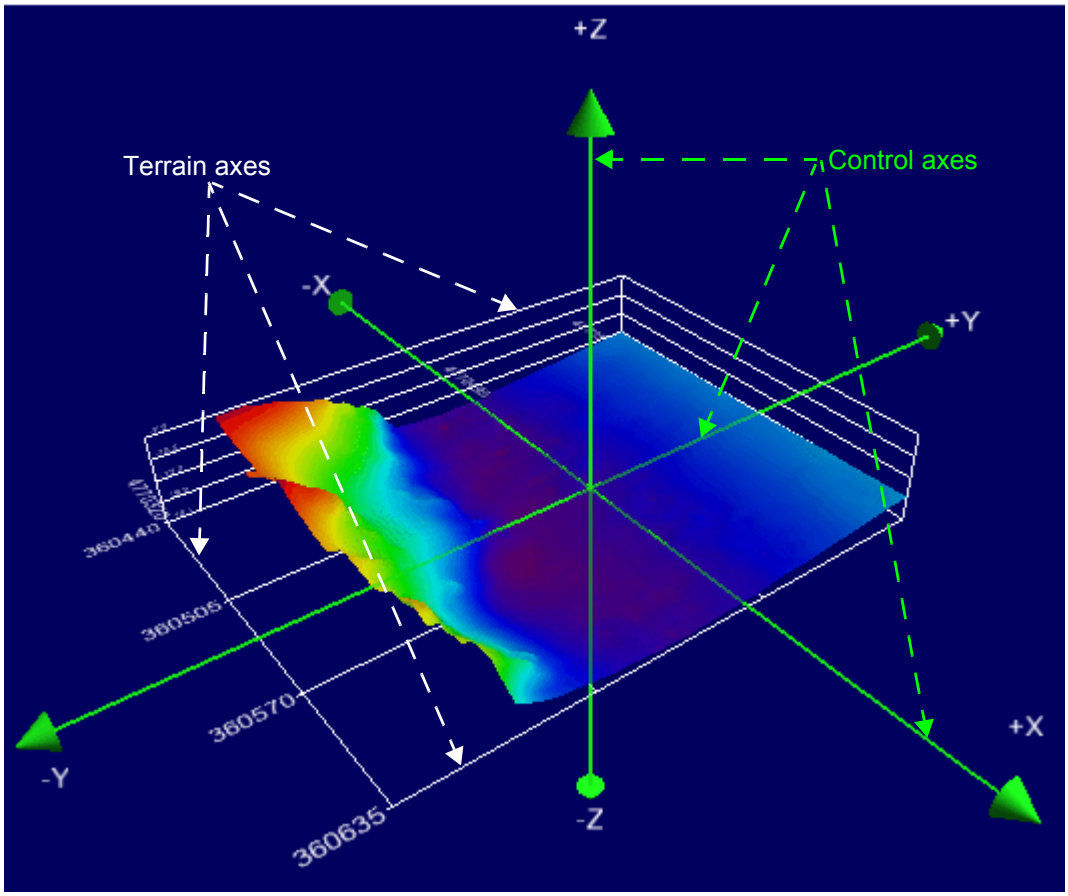
Terrain Axes

The terrain axes make up an XYZ grid that can be turned on in the 3D Display window. This grid runs par all ell to the control axes. Whereas the control axes are only visible while being used to rotate data, the terrain axes remain visible until turned off.

To turn on the terrain axes:

Menu	Tools > Options > Display > 3D

1. Select Options from the Tools menu.
2. Select 3D from the Display window.
3. Select the Terrain Axes check box to make them visible.



Navigate the 3D Display

You can use Zoom commands to zoom in and out, or you can zoom to the extent of the selected layer.

To zoom to the extent:

1. Right-click on a layer in the Layers window.
2. Select Zoom to Map from the pop-up menu.

You can also use the zoom and pan functions of the Controller panel, the mouse or the keyboard to create the effect of flying over the display.

- To "fly" using the Controller, manipulate the controls in the CONTROLLERpanel.
- To fly using the mouse, press and hold one or both of the mouse buttons while simultaneously moving the cursor. See "MOUSE AND KEYBOARD CONTROLS" ON PAGE 92
- To fly using the keyboard, use the <Arrow> keys or keyboard shortcuts. The speed you travel through the display or pan the view varies with the cursor's distance from the centre (stationary) point. See "MOUSE AND KEYBOARD CONTROLS" ON PAGE 92
- You can also zoom into the display using the "USING THE NAVIGATION ICON TOOL" ON PAGE 95.

These controls can be used while recording the fly-through.

Controller

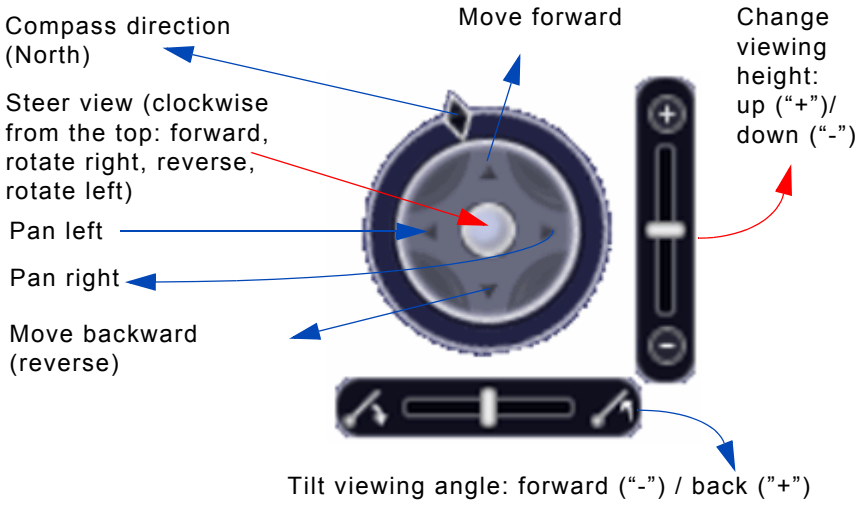
The Controller panel consists of a directional pad and slider controls used to modify the view and perform a fly-through.

Tool

Off ..... On

The Controller panel can be turned on or off using the Controller Panel button in the top-right corner of the 3D View. When the button is coloured, the panel is turned on and available for use. When turned on but not in use, the panel becomes transparent so as not to obstruct the view of the data.

The Controller panel contains the following controls:



Mouse and keyboard controls

When navigating with a mouse you can use the left and right buttons as well as the scroll wheel. Certain keys control these same movements.

With both mouse and key controls, the position of the cursor in the display determines the direction in which the image moves. When you click in the window, the data moves away from the cursor.

For example, when you click with the left mouse button, the data will move to the left or the right, depending on where the cursor is in relation to the data. When you click with the right mouse button, the data moves up or down.

Note that the mouse controls differ according to the *Controller Type* set in Tools > Options > 3D:

- If the *Controller Type is* set to Terrain Flyer, the controls enable you to navigate from the perspective of the height source. See “Navigating as Terrain Flyer” on page 93.
- If set to First Person, the controls navigate from a camera view of the current 3D scene. See “Navigating as First Person Control” on page 93.

See “Mouse/Keyboard Controller Options” on page 96.

Navigation tools

The following tables list the keyboard and mouse controls for viewing and navigating the display.

Navigating as Terrain flyer

Action	Mouse Control		Key
	Move cursor	Press button	
Pan backward in a straight line, horizontal to the TERRAIN AXES (PAGE 90)	Down	Right	Shift +S
Pan forward in a straight line, horizontal to the TERRAIN AXES (PAGE 90)	Up	Left	Shift +W
Pan left in a straight line, horizontal to the TERRAIN AXES (PAGE 90)	Left	Left + right	Shift + A
Pan right in a straight line, horizontal to the TERRAIN AXES (PAGE 90)	Right	Left + right	Shift + D
Pan up in a straight line, vertical to the TERRAIN AXES (PAGE 90)	Up	Left + right	Shift + R
Pan down in a straight line, vertical to the TERRAIN AXES (PAGE 90)	Down	Left + right	Shift + F
Rotate left	Left	Left	←
Rotate right	Right	Left	→
Tilt back, viewing angle goes up	Up	Right	↓
Tilt forward, viewing angle goes down	Down	Left	↑

Navigating as First Person Control

Action	Mouse Control		Key
	Move cursor	Press button	
Pan backward in a straight line, horizontal to the Terrain Axes.	Down	Right	Shift + S
Pan forward in a straight line, horizontal to the TERRAIN AXES (PAGE 90).	Up	Right	Shift + W
Pan left in a straight line, horizontal to the TERRAIN AXES (PAGE 90)	Left	Right	Shift + A
Pan right in a straight line, horizontal to the TERRAIN AXES (PAGE 90)	Right	Right	Shift + D
Pan up in a straight line, vertical to the TERRAIN AXES (PAGE 90).	Up	Left + Right	Shift + R
Pan down in a straight line, vertical to the TERRAIN AXES (PAGE 90)	Down	Left + Right	Shift + F
Rotate left.	Left	Left	←

Action	Mouse Control		Key
	Move cursor	Press button	
Rotate right.	Right	Left	→
Tilt back, viewing angle goes up	Up	Left	↓
Tilt forward, viewing angle goes down	Down	Left	↑

Common controls

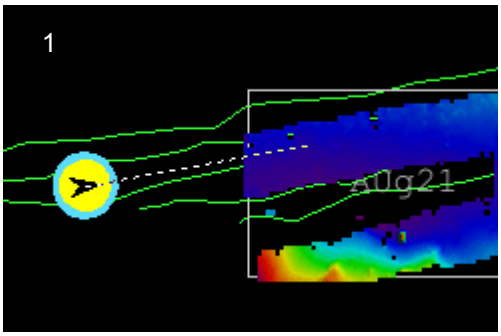
Action	Mouse Control		Key
	Move cursor	Press button	
Rotate and tilt around the x,y,z Control axes	Any direction	Middle	--
Zoom in	Wheel: Scroll forward		Ctrl+ ↓
Zoom out	Wheel: Scroll backward		Ctrl+ ↑

Using the Navigation Icon Tool

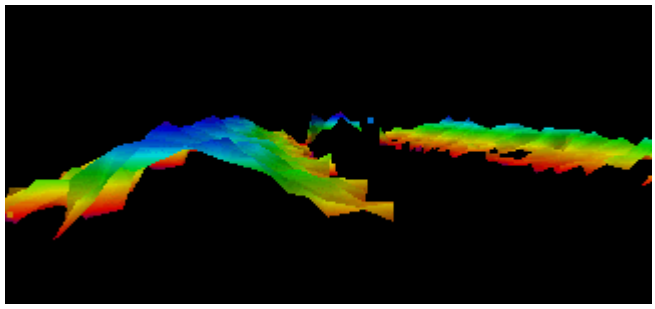
You can also change the view by manipulating the navigation icon around the selected surface in the Display window. Movement of the icon in the Display will be reflected in the 3D View.

To use this tool,

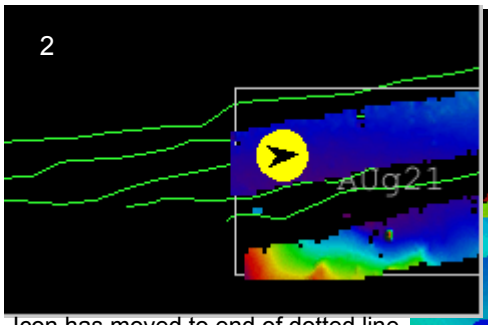
3. Click on the arrow icon and drag your mouse in the direction you want to view. This drags out a dashed line.
   - When you release the mouse click, the icon will move in the direction of the arrow to the spot where you released it.
   - In the 3D view the surface zoom in to follow path made by the dotted line in the Display.



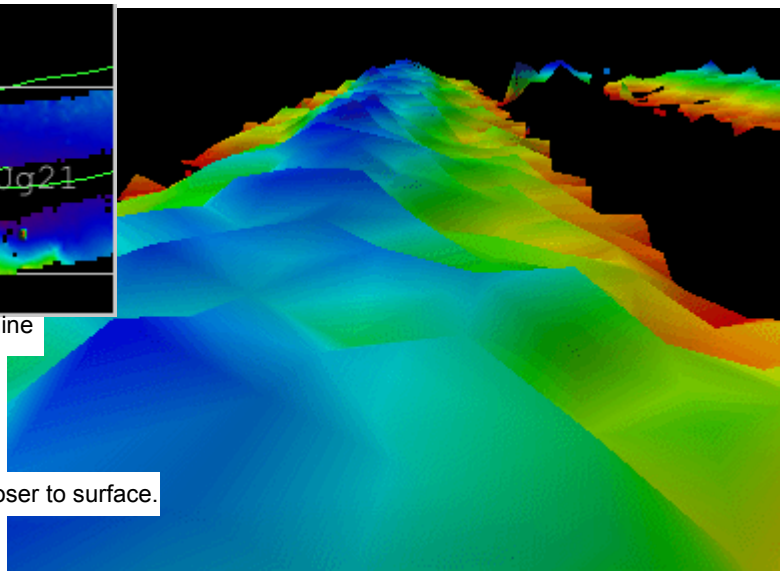
Position of icon in Display window



View of surface in 3 D window



Icon has moved to end of dotted line



View is now zoomed closer to surface.

Mouse/Keyboard Controller Options

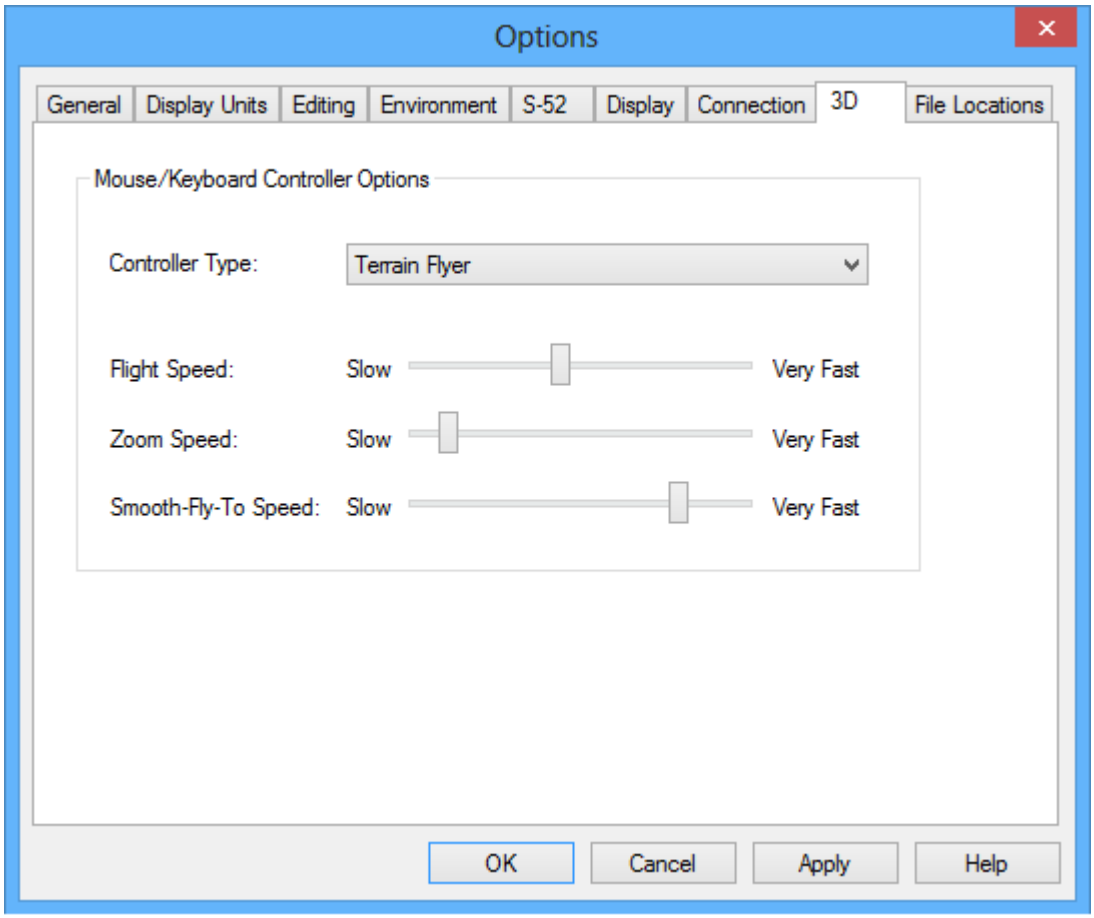
In the Options dialog box, you can adjust the settings of the keyboard and mouse controllers used to navigate the 3D Display.

Menu	Tools > Options
Tool	

To change controller options:

1. Select the Options command.

The Mouse/Keyboard Controller options are displayed.



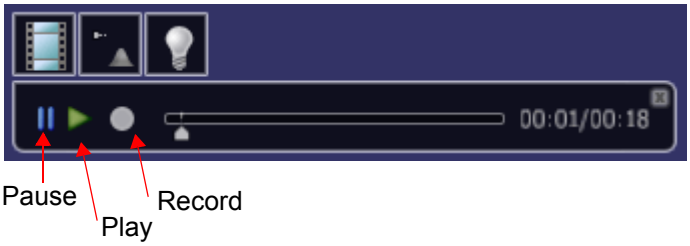
The following options are available:

- **Controller Type**: Select one of two types of controls to use when navigating the 3D Display:
  - Terrain Flyer: enables you to navigate from the perspective of the height source.
  - First Person: enables you to navigate from a camera view of the current 3D scene.
- **Flight Speed**: Controls the speed at which you move through the display during a fly-through. Move the slider bar left or right to increase or decrease flying speed.

- **Zoom Speed**: Controls how fast the view is changed when using the zoom controls. Move the slider bar left or right to increase or decrease zooming speed.
- **Smooth-Fly-To Speed**: Controls how fast the view is changed when you double-click an area of the 3D View. Move the slider bar left or right to increase or decrease zooming speed.

3D Flight Path

You can make and save movies of a 3D fly-through for playback at a later time. The controls for creation and playback are located in the 3D Display Tools panel and the 3D Flight Path submenu.



Instructions for using these tools are provided in the following sections.

- "Record a Flight" on page 98
- "Play a Flight" on page 99
- "Load an Existing Flight" on page 99
- "Save a Flight" on page 99

"Export a Flight to Video" on page 100

Record a Flight

To record a fly-through:

1. Select a layer in the Layers window to display in the 3D Display.
2. Adjust the view to display the starting point for your fly-through.
3. Select the **Record** command.

Menu	View > 3D Flight Path > Record
Tool	

The Record button will change to red to indicate that you are now recording.

4. Use the controls to navigate your flight path through the 3D Display. See "Navigate the 3D Display" on page 91 for information on navigating.

All movements in the Display window are recorded.

Menu	View > 3D Flight Path > Stop

5. To stop recording, select the **Record** command again or select the **Stop** command.

The **Record** button will return to the inactive state and your fly-through is recorded and ready for playback.

Play a Flight

To playback a recorded fly-through:

1. Select the **Play** command.

Menu	View > 3D Flight Path > Play
Tool	

The fly-through will begin to play. As it does so, the arrow on the slider bar will move from left to right and the counter at the end of the slider bar will count down, indicating the progress of the fly-through. The slider bar can also be used to go forward or backward in the recording.

2. [Optional] Click the arrow on the slider bar and drag it to the left or right to go forward or backward in the recording.

You can pause the playback if necessary without stopping it altogether.

3. [Optional] Select the **Pause** command to pause the playback.

Menu	View > 3D Flight Path > Pause
Tool	

The Pause button will be coloured to indicate that playback has been paused. Both the Pause and Play commands can be used to resume playback after the Pause command has been selected.

4. To stop the playback, either select the **Play** command again, or select the **Stop** command.

Menu	View > 3D Flight Path > Stop

Otherwise, playback stops when you reach the end of the recording.

Load an Existing Flight

If you have saved a fly-through to a file, that file can be loaded and played in the 3D View at anytime.

To load an existing flight:

1. Open the 3D Display.
2. Select the **Load from File** command.

Menu	View > 3D Flight Path > Load from File

A standard Windows Open dialog box is displayed.

3. Navigate to the flight file and click **Open**.

The fly-through is loaded into the 3D Display and the flight path controls are available.

Save a Flight

Once you have finished recording a fly-through, you can save it to an XML file that can be viewed in HIPS and SIPS at any time.

To save a flight to an XML file:

1. Record the fly-through.

Menu	View > 3D Flight Path > Save to File

2. Select the **Save to File** command.

A standard Windows Save As dialog box is displayed.

3. Select a location for the file and enter a **File name**.
4. Click **Save**.

The flight is saved as an XML file in the selected location with the specified name.

Export a Flight to Video

Use the Export to Video command to save the flight path so it can be opened in a standard media player. If you do not export your recording, it can only be re-opened in HIPS and SIPS.

To export a record of your fly-through:

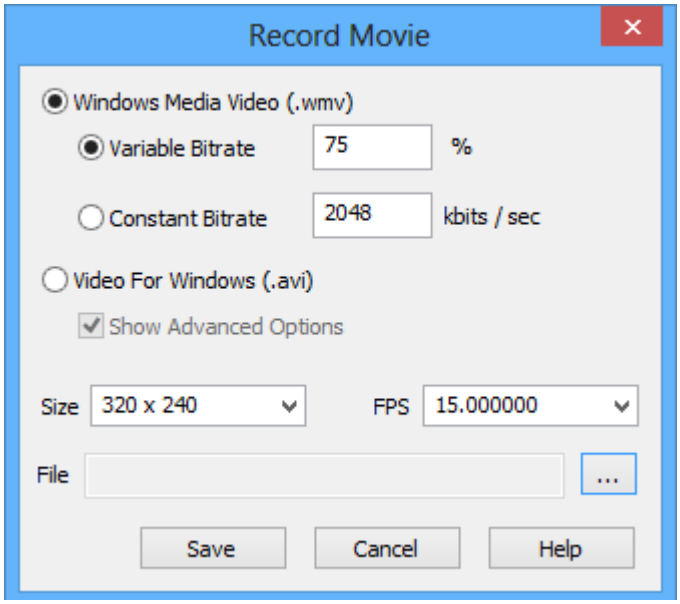
1. Create a fly-through and save the file.

Menu	View > 3D Flight Path > Load from File

2. Alternatively, load a saved file.
3. Select the Export to Video command.

Menu	View > 3D Flight Path > Export to Video

The Record Movie dialog box is displayed.



4. Choose the format of the video file. If you selected AVI format, go to step 6. If you selected WMV format, continue with the next step.

You may select a bitrate for the recorded data as a variable or a constant value.

- Variable Bitrate records the video using a fluctuating bitrate, which is based on a user-defined average quality level. During encoding, the bitrate will fluctuate to ensure the video is

created with the highest possible quality while maintaining the specified average.

- Constant Bitrate records the video using a user-defined number of kilo-bits per second. This option ensures that a constant bitrate is used during encoding, but may result in a lower quality video.

5. Select a bitrate option.
6. If you selected *Variable Bitrate*, enter the average quality level for the resulting video and go to step 7.

   OR

   If you selected *Constant Bitrate*, enter the number of kilo-bits per second to use to record the video and go to step 7.
7. [Optional] To see the advanced playback options for your AVI movie before it is saved, select the *Show Advanced Options* option.

You must select image size and FPS (Frames-Per-Second) options for the playback of your movie.

8. Select a *Size* option.
9. Select an *FPS* option.
10. To select the save name and location for your movie file, click the Browse button (**...**).

A Save As dialog box is displayed.

11. Type a save name.
12. Select a save folder.
13. Click **Save**.

The Save As dialog box is closed, returning you to the Record Movie dialog box.

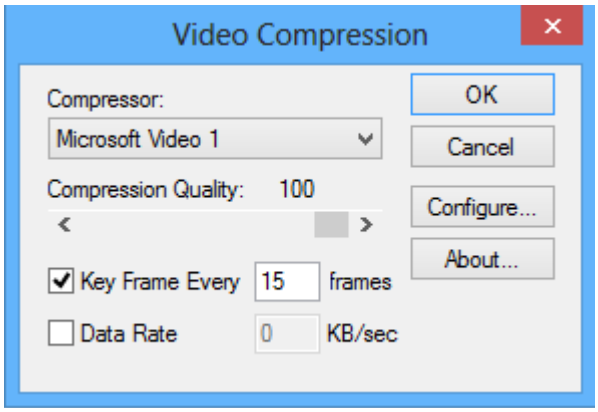
14. If you clicked the WMV format option, click **Save** and go to step 19;

    OR,

    if you clicked the AVI format option, click **Save** and continue with the next step.

Video Compression

The Video Compression dialog box is displayed.



15. Select a *Compressor*.

Some video compressors do not allow you to set the following options (in steps 15 to 17) for your movie files.

16. Use the slider control to select the compression quality (displayed as a value in the range of 0 and 100 percent).
17. [Optional] To insert key frames, select the *Key Frame* option and type a *frames* value.
18. [Optional] To set the data rate, select the *Data Rate* option and type a *KB/sec* (kilobits per second) value.

With some compressors, you can also choose the configuration settings to use for your AVI movie.

19. [Optional] To configure the selected compressor, click **Configure** and select from the available configuration options.
20. Click **OK**.

A message is displayed stating that the file was saved.

21. To finish, click **OK**.

The process is finished.

4

Creating and Editing Features

Create features such as contours and profiles. Features are digitized on layers and can be edited.

In this chapter...

Overview

Creating a feature involves assigning an acronym and attribute values then digitizing on a feature layer. The procedure is basically the same for each feature type.

Feature types are:

- Point
- Line
- Area
- Sounding

The steps in feature create:

1. Create or select a feature layer.
2. Select a New Feature command.
3. Select an object acronym.
4. Define any mandatory attributes.
5. Define any other attributes required by your system.
6. Digitize (draw the spatial representation of) the feature.

Once a feature has been created, you can edit it as needed.

Create a Feature Layer

Create a layer to which you can add features, such as contours, soundings, or digitized point, line, or area features.

The following are some of the operations that require a feature layer:

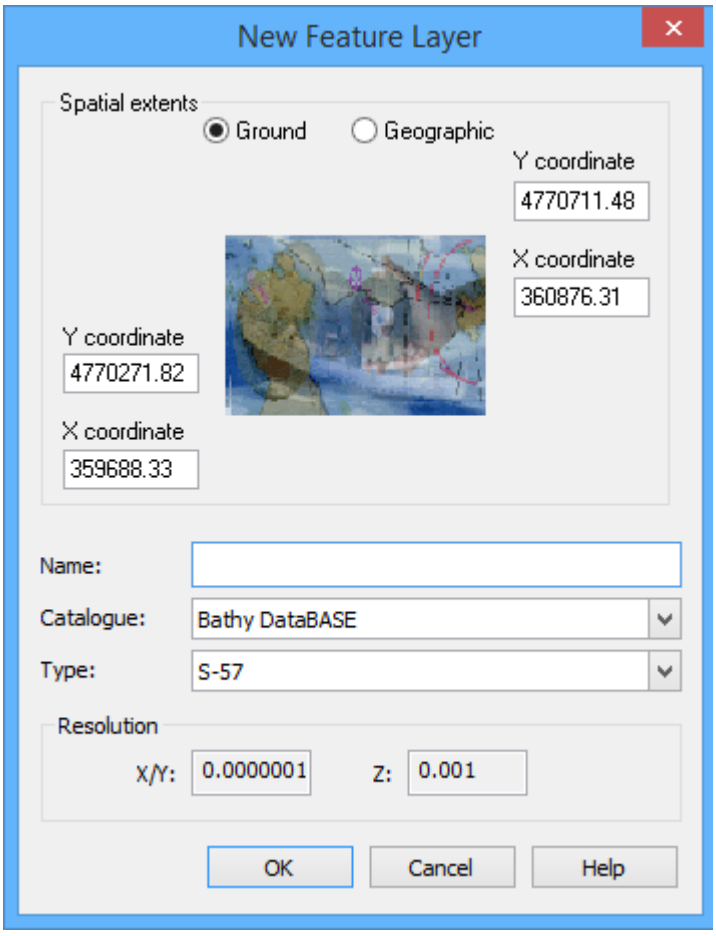
- create a soundings layer (see "SOUNDING SELECTION" ON PAGE 472)
- create a contours layer (see "CONTOURS" ON PAGE 450)
- manually digitize features (see "CREATE FEATURES" ON PAGE 107)
- create filtered child layers (see "CREATE FILTERED LAYERS" ON PAGE 146)

Feature layers are saved in HOB format. You can reopen a feature layer using the File > Open command, then continue to add objects to it.

Menu	Create > New Feature Layer
Tool	

1. Select the New Feature Layer command.

The New Feature Layer dialog box is displayed.



If you have a dataset or background map open, the *Spatial Extents* fields are populated with the extents of the open data, but can be

edited if necessary. If you do not have data open, you must use these fields to specify the extents to be covered by the new layer. To enter or edit extents, continue with the next step. To use the extents of open data, skip to step 4.

2. Select *Ground* or *Geographic*.
3. Enter coordinates in the *Latitude* (Y) and *Longitude* (X) fields.
4. Type a name for the feature layer in the *Name* field.
5. Select a *Catalogue*.
6. Select a layer *Type*, e.g. S57.

The *Resolution* settings for the layer type you selected are displayed in the *X/Y* and *Z* fields. These settings cannot be changed in this dialog box. Layer resolution is defined in the Resolution.xml file, which defines a resolution value for each feature layer type. This file can be found in `C:\Program Files\CARIS\HIPS\<version>\system`.

You can modify this file to control the feature types displayed in the list and to change the resolution values for the layer types.

7. To create the layer, click **OK**.

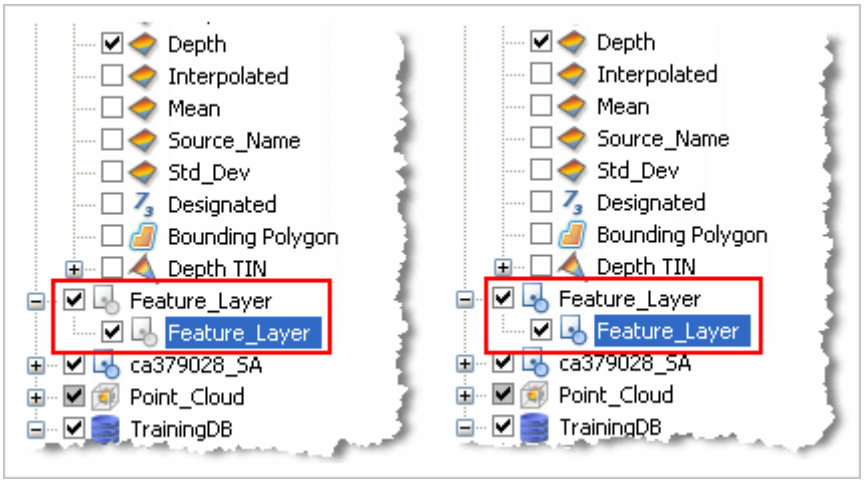
A new feature layer is displayed in the Layers window.

At least one feature must be added to the layer before the layer can be saved.

8. Add a feature to the layer.
9. Select a **Save** command.

Menu	File > Save
Tool	
Pop-up	Save

Once the layer is saved, the icon representing the layer changes to the one shown below.



Create Features

The following is a description of the common procedures for creating a feature. For information on creating specific types of features see:

"NEW POINT FEATURE" ON PAGE 137

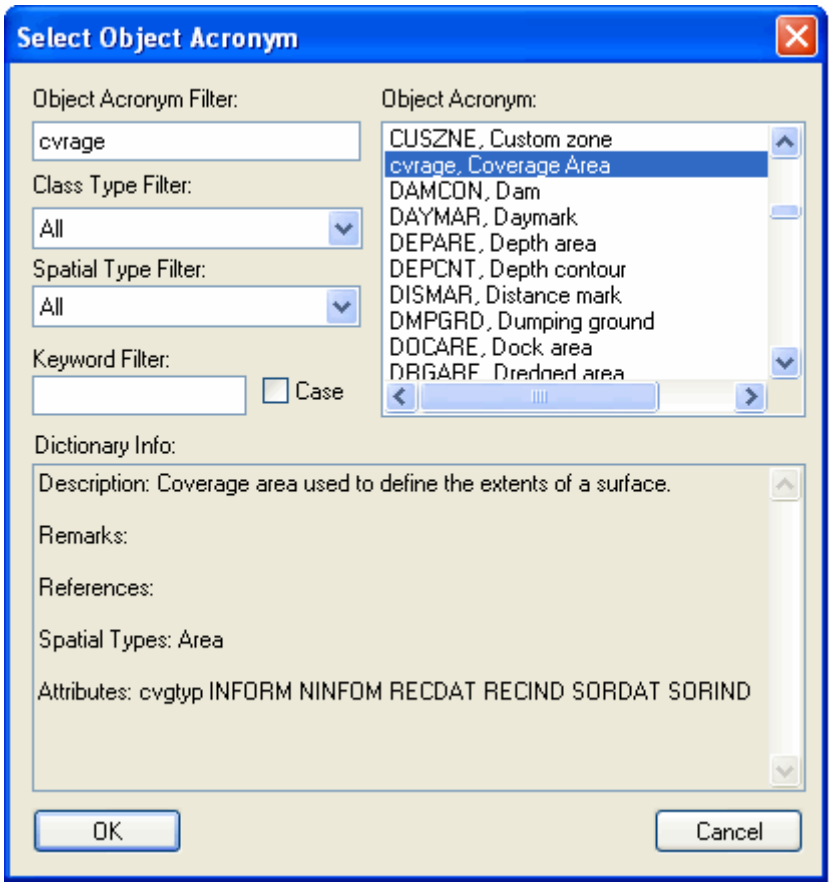
"NEW LINE OR AREA FEATURE" ON PAGE 138

"NEW SOUNDING FEATURE" ON PAGE 138

To create a new feature:

1. Select a New Feature tool (Point, Line, Area or Sounding).

The Select Object Acronym dialog box is displayed.



Select Object Acronym

In this dialog box, you select an object type for the new feature. The list of object acronyms is populated based on the new feature tool you selected. For example, if you selected the New Point Feature tool, the list would include point feature type acronyms.

If you know the acronym, you can type it in the *Object Acronym Filter* field. This will select the acronym in the list. If you do not know the acronym, you can search for it using the other fields in the dialog box or by scrolling through the list.

Below is an example, of how you would search for the lateral buoy acronym (BOYLAT) using the *Keyword Filter* field.

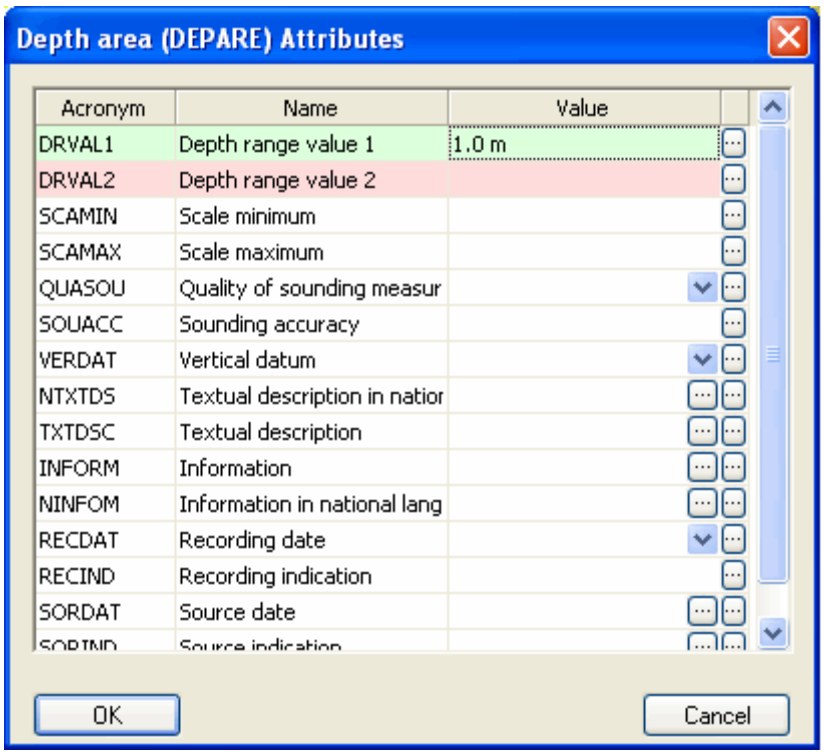
1. Begin by typing "B" in the *Keyword Filter* field.

The list of acronyms is filtered to include only acronyms with a "B" in them. Each time you add a letter, the list gets filtered further.

2. Continue adding letters to the field until you see the correct acronym.
3. Select the acronym from the list.
4. Click **OK**.

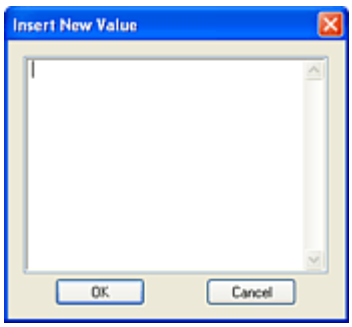
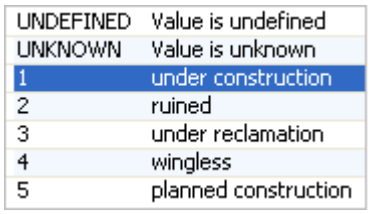
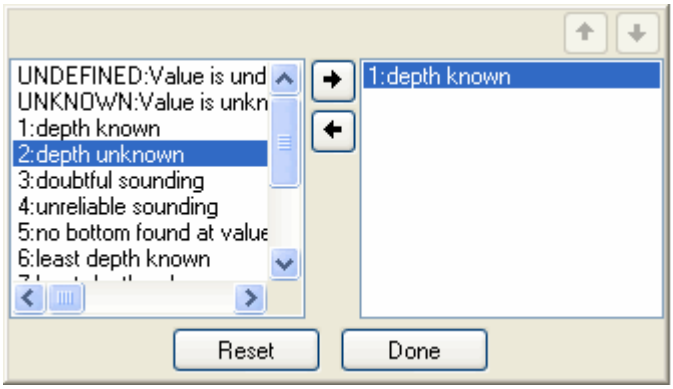
Set Attributes

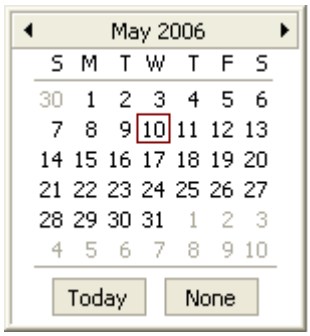
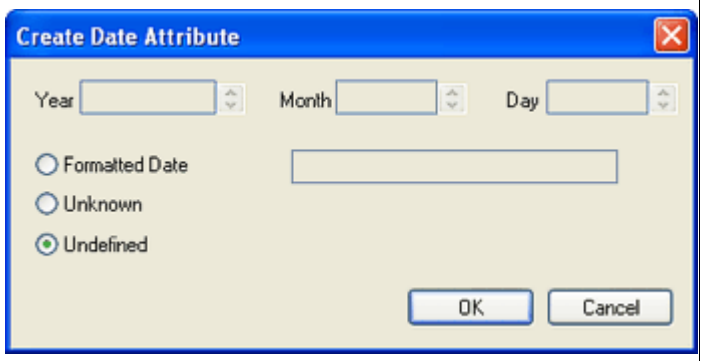
The Attributes dialog box is displayed.



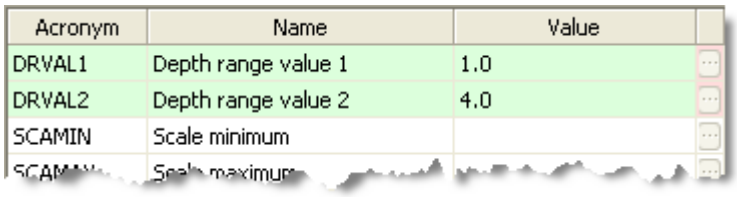
Attribute acronyms highlighted in red (DRVAL2 in the above image) are mandatory and have not yet had values assigned to them. Attribute acronyms highlighted in green (DRVAL1) are also mandatory, but they have had values assigned to them.

Values may be added to both mandatory and optional attribute acronyms using one of the methods described in the table below.

Field type	Data entry method
Text entry	Type text in the text entry field, or click Browse (**...**) to display the Insert New Value dialog box. Type any details you wish to include and click **OK**. 
Single value entry	Select a value from the list. 
Multiple value entry	Select a value from the list on the left and double-click it or click **Add >** to add it to the list on the right. To remove a value, select a value from the list on the right and click **Remove**. To reposition the items in the list on the right, select a value and click the up and down arrow buttons. To clear this list, click **Reset**. 

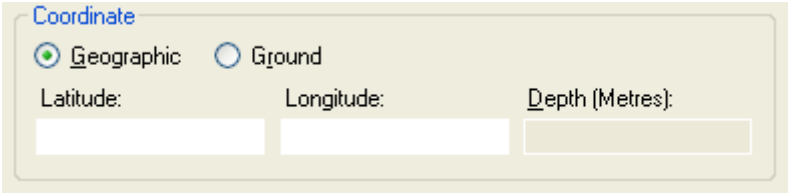
Field type	Data entry method
Date: calendar	Select a date from the calendar. Use the arrow buttons to scroll back ward (<) and forward (>) by one month. 
Date: Create attribute date	Select Undefined or Unknown as the value if a date is not available. If adding a date, use the arrow buttons to select the value for each part of the date value. The date value will be displayed in the Formatted Date field. 
Files	Click Browse (...) to open a standard Windows Open dialog box, which can be used to locate and select a file. When you select a file, the filename (not the file itself) is included as an attribute and added to the database.

Once values are assigned to the mandatory attribute acronyms, the highlight colour changes to green.



5. Set the other values as appropriate.

6. For point or sounding features, specify coordinate values, as necessary.



7. Click **OK**.

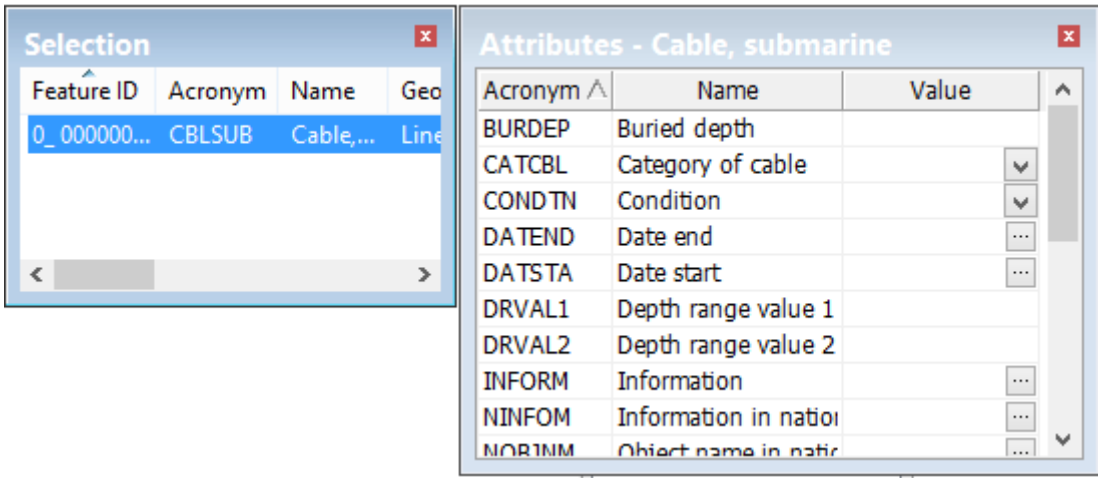
Digitize the feature

Next, digitize the spatial representation of the feature (draw and position the a representation of the feature. The cursor changes to a shape based on the type of feature being digitized. This also indicates that digitizing mode is active.

8. Digitize the feature.

See "DIGITIZING FEATURES" ON PAGE 112 for full instructions on digitizing specific features.

After a feature is digitized, it is displayed in the Selection window, and its attributes are displayed in the Attributes window. You can edit attribute information here using the data entry method listed in the table above.



Digitizing Features

The digitizing tools are accessed from a pop-up menu in the Display window. Digitize mode is enabled when any of the following functions are active.

- Create new point feature
- Create new line feature (Line Digitizer)
- Create new area feature (Area Digitizer)
- Create new sounding feature

Digitizing Lines

You can digitize the following types of lines:

- POINT-TO-POINT LINE
- STREAM LINE
- LOXODROME
- GEODESIC
- SQUARED LINE
- FITTED CURVE
- SPLINE

The general procedure for digitizing a line is as follows:

1. Right-click and select the appropriate line digitizing command.
2. Add points to define the line.
3. End the line using one of the following methods:
   - Press <Home> to close the line (connect the first point to the last point)
   - Press <End> to complete the line as it is.
   - Right-click, select the Digitizer submenu, then select End Line.
4. Exit from the digitize command by selecting the line digitizing command a second time.

You can change digitizing types "on the fly" while you are digitizing. Before ending the line, select a different digitizing type and continue to add points.

Point-to-Point Line

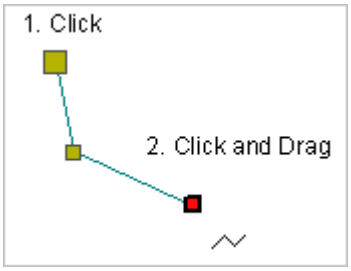
A point-to-point line contains two or more digitized points joined by straight line sections.

To create a point-to-point line:

1. Select the **Point-to-Point Line** command.

Pop-up: Line Digitizer > Point-to-Point Line

2. Click once at the location of the first point (1).



3. Continue clicking for each additional point.

If you keep the mouse button pressed, you can drag a point to a desired location and then release the button.

You are not limited to using only the line type selected when you began digitizing, you can switch line types any time "on the fly" while digitizing. When you change line types, the line continues automatically from the last point digitized.

4. [Optional] Switch line types by selecting a different line digitizer tool.

Pop-up: Line Digitizer > *Line types*

5. When you are finished digitizing, there are three possible ways to end the line:
   - Deactivate the new feature (**New Line** or **New Area**) button
   - Select Edit Line > End line from the pop-up menu.
   - Press <End>.

To create an enclosed feature, press <Home> and the line will join with the first point created.

Stream Line

A stream line is a curved line in which points are automatically inserted at intervals along the line.

1. Choose a **Stream Line** command.

Pop-up: Line Digitizer > Stream Line

2. Press and hold the left mouse button and drag the cursor to draw the line.

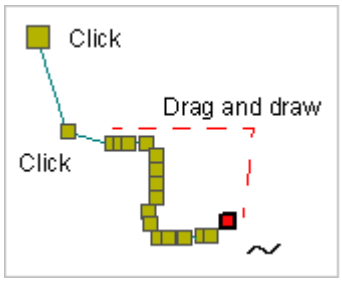
The line follows the path of the mouse.

3. Release the mouse button.

This ends the current "stream" but does not end the line.

4. [Optional] Click on another point.

This adds a straight section of line between the existing stream and the new point.



You are not limited to using only the line type selected when you began digitizing, you can switch line types any time "on the fly" while digitizing. When you change line types, the line continues automatically from the last point digitized.

5. If necessary, digitize further streams or choose another line command and continue adding more spatials.

Pop-up: Line Digitizer > *Line types*

6. When you are finished digitizing, there are three possible ways to end the line:
   - Deactivate the new feature (**New Line** or **New Area**) button.
   - Select Edit Line > End line from the pop-up menu.
   - Press <End>.

To create an enclosed feature, press <Home> and the line will join with the first point created.

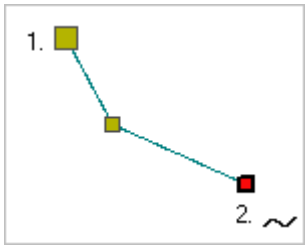
Loxodrome

A loxodrome is a line that cuts all meridians (lines of longitude) at the same angle. This is a more accurate line for navigation than normal line types because the angles are calculated as if on a real-world or spherical surface.

1. Select the Loxodrome Line command to start drawing mode.

Pop-up: Line Digitizer > Loxodrome

2. Click once at the location of the first point (1).



3. Continue clicking to add points as needed.

When placing additional points, if you keep the mouse button pressed after clicking, you can drag the new point to a desired location and then release the button.

You are not limited to using only the line type selected when you began digitizing, you can switch line types any time "on the fly"

while digitizing. When you change line types, the line continues automatically from the last point digitized.

4. [Optional] Switch line types by selecting a different line digitizer tool from the Editing toolbar or from the pop-up menu.

Pop-up: Line Digitizer > *Line types*

5. When you are finished digitizing, there are three possible ways to end the line:
   - Deactivate the new feature (**New Line** or **New Area**) button.
   - Select Edit Line > End line from the pop-up menu.
   - Press <End>.

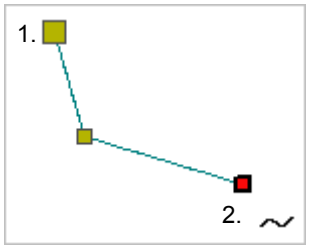
To create an enclosed feature, press <Home> and the line ending will join with the first point created.

Geodesic

A geodesic line the shortest distance between two points on the ellipsoid. For example, with a global map view, a geodesic line with two points, one in Whitehorse, Canada and one in Oslo, Norway would be drawn as a straight line crossing the Arctic circle as opposed to being drawn parallel to a line of latitude.

1. Select the Geodesic Line command.
2. Click once at the location of the first point (1).

Pop-up: Line Digitizer > Geodesic



3. Continue clicking to add points as needed.

When placing additional points, if you keep the mouse button pressed after clicking, you can drag the new point to a desired location and then release the button.

You are not limited to using only the line type selected when you began digitizing, you can switch line types any time “on the fly” while digitizing. When you change line types, the line continues automatically from the last point digitized.

4. [Optional] Switch line types by selecting a different line digitizer tool.

Pop-up	Line Digitizer > *Line types*

5. When you are finished digitizing, there are three possible ways to end the line:
   - Deactivate the new feature (**New Line** or **New Area**) button.
   - Select Edit Line > End line from the pop-up menu.
   - Press <End>.

To create an enclosed feature, press <Home> and the line will join with the first point created.

Squared Line

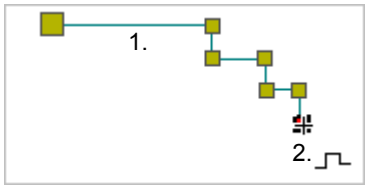
A squared line is a point-to-point line in which the adjacent segments can be at increments of 90 degrees.

Pop-up	Line Digitizer > Squared Line

1. Choose the **Squared Line** command.
2. Click to place the first two points to create a line segment (1).
3. To create additional points, keep the mouse button pressed after clicking, then drag the new point to the desired location (2).

The line will snap into position when the two segments approach multiples of 90 degrees.

4. Release the mouse button when the line snaps into place.



5. Continue clicking to add points as needed.

When placing additional points, if you keep the mouse button pressed after clicking, you can drag the new point to a desired location and then release the button.

You are not limited to using only the line type selected when you began digitizing, you can switch line types any time “on the fly” while digitizing. When you change line types, the line continues automatically from the last point digitized.

6. [Optional] Switch line types by selecting a different line digitizer tool.

Pop-up	Line Digitizer > *Line types*

7. When you are finished digitizing, there are three possible ways to end the line:
   - Deactivate the new feature (**New Line** or **New Area**) button.
   - Select Edit Line > End line from the pop-up menu.
   - Press <End>.

To create an enclosed feature, press <Home> and the line will join with the first point created.

Fitted Curve

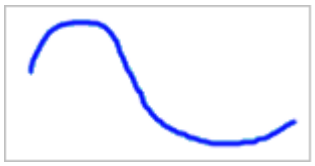
A fitted curve is similar to a point-to-point line, except that the sections between the points that you define are curved instead of straight. The curves are defined using a mathematical algorithm based on polynomials.

Tool	
Pop-up	Line Digitizer > Fitted Curve

1. Choose a **Fitted Curve** command.
2. Click to place the first point (1).
3. To create additional points, keep the mouse button pressed after clicking, then drag the new point to the desired location (2).

The curve of the line segment will be adjusted as you drag the cursor. The line segment of the last point added will also be adjusted to line-up with the curve of the new segment.

4. Release the mouse button when the line segment has the correct shape.



Spline

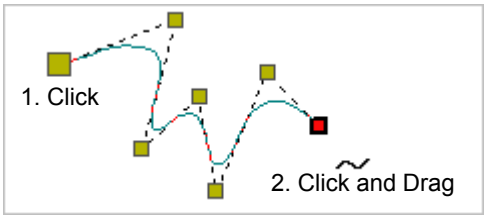
A spline is a line that uses a mathematical algorithm to calculate curves based on reference points along the line. The points that you digitize are the reference points.

As you digitize points, the application automatically inserts points for the line to follow, based on the algorithm. You will not see these points, but you will see an outline of the line as it is being drawn. Rarely will the line actually pass through the reference points that you digitized.

Tool	
Pop-up	Line Digitizer > Spline

1. Select the **Spline** command.
2. Click to place the first point (1).
3. Click a second point.
4. Continue clicking to add points as needed.

When placing additional points, if you keep the mouse button pressed after clicking, you can drag the new point to a desired location and then release the button. The curve of the line segment will be adjusted as you drag the cursor.



You must digitize at least three points to create a spline.

Digitizing Arcs

Digitizing an arc follows the same general procedure as digitizing a line. See "DIGITIZING LINES" ON PAGE 112 for the general procedure.

You can digitize an arc using any of the following methods:

- Arc Using Centre and Two Points
- Arc Using Three Points
- Arc Using Two Points and Direction
- Geodetic Arcs

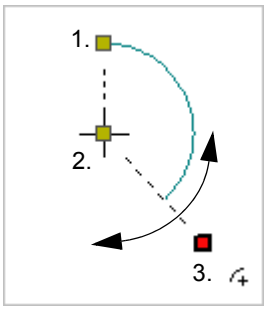
Arc Using Centre and Two Points

Digitize an arc by specifying a centre and two radius points.

Pop-up	Line Digitizer > Arc [centre + 2 points]

1. Choose the **Arc [centre + 2 points]** command.
2. Click to place the starting point of the arc (1).
3. Click to place a centre point (2).

The distance between the first two points defines the radius of the circle on which the arc is based.



4. To place the third point, press and hold the left mouse button, then drag the cursor until the arc reaches the desired length (3).
5. To finish, release the mouse button.

Arc Using Three Points

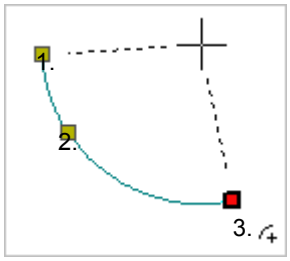
Digitize an arc by specifying three points. The first and third points determine the start and end nodes of the line, while the second point determines the direction of the arc.

1. Choose the **Arc [3 points]** command.
2. Click to place the starting point of the arc.

Pop-up	Line Digitizer > Arc [3 points]

The second point should be placed in relation to the first point in the direction you want the arc to go. For example, if the second point were placed to the lower-right of the first point, and the third point were place to the lower right of the second point, the arc would be formed as in the image below. The arc must start and end with the first and third points, and also cross through the second point.

3. Click to place the second point of the arc.
4. For the third point (3), press and hold the left mouse button, then drag the cursor until the arc reaches the desired shape and size.



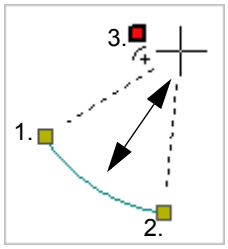
5. To finish, release the mouse button.

Arc Using Two Points and Direction

Digitize an arc by specifying the start and end nodes of the arc and a direction.

1. Choose the **Arc [2 points + direction]** command.
2. Click two points to define the start and end points of the arc (1 and 2).

Pop-up	Line Digitizer > Arc [2 points + direction]

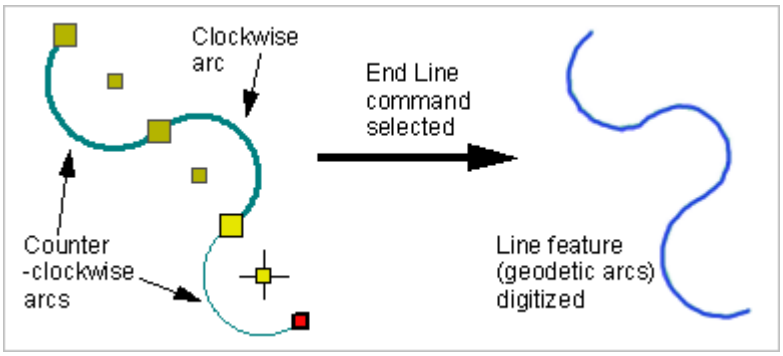


3. For the third point, press and hold the left mouse button and drag the cursor until the arc reaches the desired shape and size.
4. To finish, release the mouse button.

Geodetic Arcs

You can digitize a geodetic arc in either counter-clockwise (CCW) or clockwise (CW) direction.

When creating geodetic arcs as part of a line feature, the radius point of the current geodetic arc is taken from the final point of the previous section of line.



Geodetic Arc: Counter-clockwise (CCW)

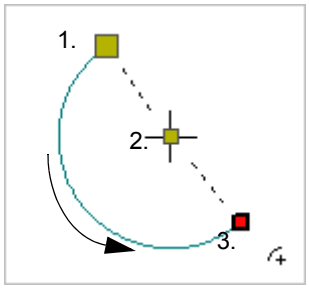
Pop-up	Line Digitizer > Geodetic Arc [CCW]

Create a digitized counter-clockwise geodetic arc using a scaling and rotating centre point.

1. Choose the **Geodetic Arc (CCW)** command.
2. Click to place a radius point (1).
3. Click to place the centre point (2).

The third point defines the arc.

4. Press and hold the left mouse button, then drag the cursor until the arc is the desired shape and size.



5. Release the mouse button.
6. Repeat for each arc required.

You are not limited to using only the line type selected when you began digitizing, you can switch line types any time "on the fly" while digitizing. When you change line types, the line continues automatically from the last point digitized.

7. [Optional] Switch line types by selecting a different line digitizer tool.
8. When you are finished digitizing, there are three possible ways to end the line:
   - Deactivate the new feature (**New Line** or **New Area**) button.
   - Select Edit Line > End line from the pop-up menu.
   - Press <End>.

To create an enclosed feature, press <Home> and the line will join with the first point created.

Geodetic Arc: Clockwise (CW)

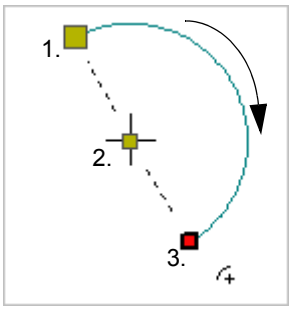
Create a digitized clockwise geodetic arc using a scaling and rotating centre point.

Pop-up	Line Digitizer > Geodetic Arc [CW]

1. Choose the **Geodetic Arc (CW)** command.
2. Click to place a radius point (1).
3. Click to place the centre point (2).

The third point defines the arc.

4. Press and hold the left mouse button, dragging the cursor until the arc is the desired shape and size.

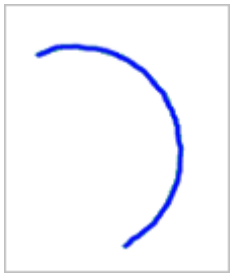


5. Release the mouse button.
6. [Optional] Repeat steps 3 to 5 for each arc required.

You are not limited to using only the line type selected when you began digitizing, you can switch line types any time "on the fly" while digitizing. When you change line types, the line continues automatically from the last point digitized.

7. [Optional] Switch line types by selecting a different line digitizer tool.
8. When you are finished digitizing, there are three possible ways to end the line:
   - Deactivate the new feature (**New Line** or **New Area**) button.
   - Select Edit Line > End line from the pop-up menu.
   - Press <End>.

To create an enclosed feature, press <Home> and the line will join with the first point created.



Digitizing Circles

Digitizing a circle follows the same general procedure as digitizing a line, except that closing the line is done automatically. See "DIGITIZING LINES" ON PAGE 112 for the general procedure.

You can digitize standard circles or geodetic circles using the following methods:

- Circle with Centre and Radius

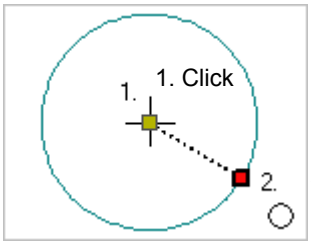
- Circle with Three Points
- Circle with Two Point Diameter
- Circle with Two Points and a Radius

Circle with Centre and Radius

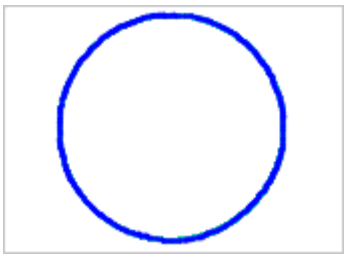
Digitize a circle by specifying the centre point and a radius.

Pop-up	Line Digitizer > Circle [centre and radius]
Pop-up	Line Digitizer > Geodetic Circle [centre and radius]

1. Choose the **Circle [centre and radius]** or the **Geodetic Circle [centre and radius]** command.
2. Click to place the centre point (1) of the circle.



3. To place the radius point (2), press and hold the left mouse button and drag the cursor until the circle reaches the desired size.
4. To finish, release the mouse button.

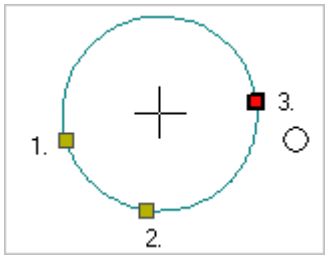


Circle with Three Points

Digitize a circle by specifying three points on the circumference.

Pop-up	Line Digitizer > Circle [3 points]

1. Choose the **Circle [3 points]** command.
2. Click to place the first points of the circle.
3. Click again to place the second point of the circle, defining the direction.
4. To place the third point (3), press and hold the left mouse button and drag the cursor until the circle reaches the desired size.



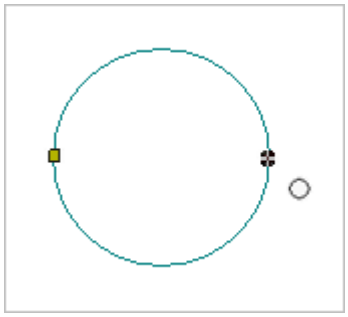
5. To finish, release the mouse button.

Circle with Two Point Diameter

Digitize a circle by defining its diameter.

Pop-up	Line Digitizer > Circle [2 diameter points]
Pop-up	Line Digitizer > Geodetic Circle [2 diameter points]

1. Select the **Circle [2 diameter points]** or the **Geodetic Circle [2 diameter points]** command.
2. Click to place the first point that defines the diameter.



3. To place the second point, press and hold the left mouse button and drag the cursor until the circle reaches the desired size.
4. To finish, release the mouse.

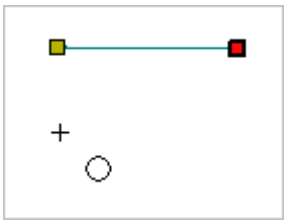
Circle with Two Points and a Radius

Digitize a circle by specifying two points on the circumference and the radius. The distance of the radius from the two points determines the size of the circle.

Pop-up	Line Digitizer > Circle [2 points + radius]
Pop-up	Line Digitizer > Geodetic Circle [2 points + radius]

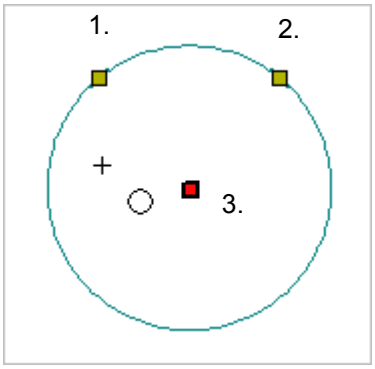
1. Select the **Circle [2 points + radius]** or the **Geodetic Circle [2 points + radius]** command.
2. Click to place the first two points.

The points are connected by a line.



For the third point, it does not matter where you click, the point will automatically position itself between the first two points.

3. Press and hold the left mouse button and drag the cursor until the circle reaches the desired size.



4. Release the mouse button when finished.

A circle is drawn using the distance between the third point and the first or second point as the radius.

Digitizing Rectangles

Digitizing a rectangle follows the same general procedure as digitizing a line, except that closing the line is done automatically. See "DIGITIZING LINES" ON PAGE 112 for the general procedure.

Digitize a rectangle using one of the following methods:

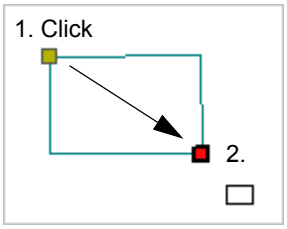
- Rectangle Using Two Points
- Rectangle Using Three Points

Rectangle Using Two Points

Digitize a rectangle by specifying two points.

Pop-up	Line Digitizer > Rectangle [2 points]

1. Choose a **Rectangle [2 points]** command.
2. Click to place the first point.
3. To place the second point (2), press and hold the left mouse button and drag the cursor until the rectangle reaches the desired size.



4. To finish, release the mouse button.

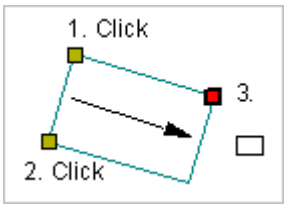
Rectangle Using Three Points

Digitize a rectangle by specifying three points.

Pop-up	Line Digitizer > Rectangle [3 points]

1. Choose the **Rectangle [3 points]** command.
2. Click to place the first point.
3. Click in another location to place the second point.

The third point defines the depth of the rectangle. Note that it is not necessarily on a corner.



4. To place the third point (3), press and hold the left mouse button and drag the cursor until the rectangle reaches the desired size.
5. To finish, release the mouse button.

Additional Digitizing Methods

In addition to the standard digitizing methods, there are also digitizing tools that enable you to take advantage of existing features. These tools make digitizing quicker and more accurate. Refer to each of the sections in this topic for information on the additional tools.

"CLICK WITHIN EDGES" ON PAGE 127

"SNAP AND SNAP-INTERSECT" ON PAGE 129

"GRAB" ON PAGE 131

"VECTORIZE FROM RASTER" ON PAGE 133

"DIGITIZING USING THE COORDINATES WINDOW" ON PAGE 133

Click Within Edges

Create new or modify existing area objects by re-using existing polygon edges.

You can use multiple polygons to create a single feature if those polygons are contained within a single outer boundary polygon. For example, you can select a lake polygon, then all of its islands to create an area using all of the selected edges.

Menu	Create > New Feature > Area
Tool	

Tool	
Pop-up	Edit Line > Polygon [Click Within]

1. Select the **New Area Feature** tool.
2. Select an object acronym and assign attribute values.
3. Select the **Click Within Edges** tool.
4. Click inside one or more polygons.

If the selected area is a subset of the area you want, click again in a neighbouring part of that area.

5. When all polygons are selected, end digitizing mode by deactivating the New feature tool.

Create cut-out areas in a (polygon) feature

Menu	Create > New Feature > Area
Tool	

You may also use the tool to cut out areas in existing polygon features.

1. Select the **New Area Feature** tool.
2. Select an object acronym and assign attribute values.

Now, create a new cut-out area feature. The boundaries of the new feature must be completely inside the boundaries of the previously existing (original) feature.

3. Create the new area feature by clicking points inside the original feature.
4. To finalize the new feature, deselect the New Area Feature tool.

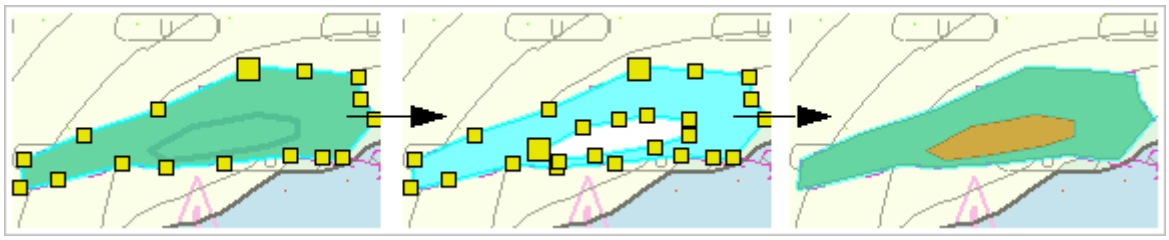
Menu	Edit > Feature
Tool	

Tool	
Pop-up	Edit Line > Polygon [Click Within]

Now you can use the new feature to cut the original feature.

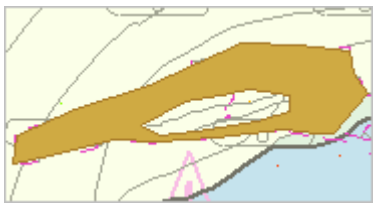
5. Select the **Edit** command.
6. Select the original feature.
7. Select the **Click Within Edges** tool.
8. To cut the original feature, click inside the new feature.
9. To finish cutting, click the Edit tool again.

The selected polygon is cut.



10. [Optional] To remove the cut-out feature from the current feature layer, select it and select the **Delete** command.

Menu	Edit > Delete > Superselection/Selection



Snap and Snap-Intersect

Snap and snap-intersect can assist in digitizing features accurately by letting you snap the vertices of new features to those of existing features.

- Snapping causes the cursor to automatically jump to the closest vertex on an existing feature, based on the settings in Options. To snap to a feature, press and hold the <S> key while clicking with the mouse.
- Snap-intersect also causes the cursor to automatically jump to the closest vertex on a feature, but it splits the selected edge into two edges at the snapped location. To snap-intersect, press and hold the <I> key while clicking with the mouse.

In order to snap to existing vertices, you must have a layer designated as the target layer containing the vertices to which you will snap. The target layer is selected using the *Set As Snap/Grab Target* command. This command is located in the toolbar as a drop-down list and in a pop-up menu in the Layers window. Only layers that can be snapped to will be included in the list or provided in the pop-up menu. By default, the currently selected layer is set as the snap/grab target and the drop-down list will read “Use current layer”. If you select a layer that cannot be snapped to, the drop-down list is disabled. When the target layer is changed from the default to a specific layer, the selected layer is remembered until a different target is selected, regardless of the active layer in the Layers window.

There are several snapping icons to help you visually identify where to snap or intersect. A specific icon is used for each type of vertex and a specific cursor is displayed when the cursor is moved over each type of vertex while in snapping mode.

- A large yellow square represents the beginning node of the feature being digitized.
- A red square drawn in the centre of the snapping symbol represents the current vertex being digitized.
- A small yellow square represents an intermediate vertex on the feature being digitized.
- A blue diamond with a red outline represents nodes that have been added using snap-intersect.

Example: Snapping to a starting node of an edge draws a large cross-hatched hexagon with a red square. It is replaced with a large yellow square when the next vertex is added.

The standard snap and snap-intersect cursors are:

Type of Vertex	Snapping Cursor
Starting node of an edge	
End node of an edge	
Intermediate edge vertex	
Nearest edge vertices	
Vertex	

To cycle through multiple vertices,
- click to place the cursor in snapping mode,
- release the <S> key, and
- press the <SPACEBAR> several times.

Each time you press the Spacebar, a different vertex is selected and the cursor changes based on the type of vertex.

Snapping

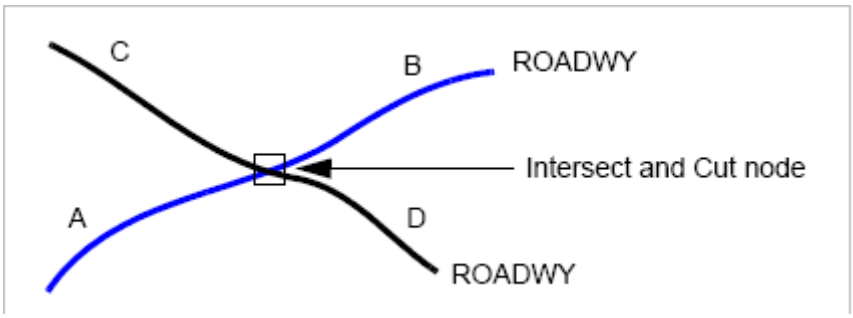
Snapping is used when you want to snap to a single vertex of an edge. To use snapping:

Tool	Use current layer
Pop-up	Set as Snap/Grab Target (Layers window)

1. Use the Set as Snap/Grab Target command to set the target layer.
2. Press and hold the mouse button.
3. Move the cursor to the vicinity of where you want the vertex snapped.
4. Press and hold <S> to trigger a search for the nearest snap vertex.
5. If necessary, use the <Spacebar> to toggle between several available vertices within the tolerance.
6. When the desired vertex is indicated, release the mouse button.

Intersect

Snap-intersect is used to digitize a new feature that will intersect with an existing edge. The new vertex will be digitized at the point where the edges intersect, splitting the existing edge into two edges. In the following example the blue line (AB) is digitized after being split into two edges at the point where it intersects with the black line (CD).



1. While digitizing a line, position the cursor over an existing edge.
2. Press the <I> key to intersect the lines.

The cursor changes shape to show that an intersect vertex has been inserted.

3. Finish digitizing the line.

Grab

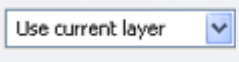
When digitizing or editing line and area features, including bounding polygons, you can use this command to grab existing features. The grabbed feature is duplicated and the duplicate is used to form the feature you are currently editing or digitizing.

Digitize Using Grab

To digitize a feature using Grab:

1. In the Layers window, select the feature layer on which you want to digitize the feature.

The *Snap/Grab Target* list in the toolbar controls the layer that will be used as the source from which to grab feature geometry and snap to existing vertices. Only layers that can be snapped to will be included in the list or provided in the pop-up menu. If a non-snappable layer is selected, the list will be disabled. By default, the currently selected layer is set as the snap/grab target and the drop-down list will read "Use current layer". If this setting is changed from the default to a specific layer, the selected layer is remembered until a different target is selected, regardless of the active layer in the Layers window.

Tool	
Pop-up	Set as Snap/Grab Target (Layers window)

2. If necessary, use the **Set as Snap/Grab Target** command to change the target layer.

Menu	Create > New Feature > Line/Area
Tool	

3. Select the **New Line Feature** or **New Area Feature** command.
4. Select an object acronym and define any necessary attributes.

The cursor switches to digitizing mode.

5. Select the **Grab** command.

The appearance of the pointer changes to indicate that the Grab method is enabled.()

Tool	
Pop-up	Line/Area Digitizer > Grab

6. Select the edge in the target layer to use for the new feature.

All vertices on the selected edge become active.

7. If you require more than one edge from the grabbed object, continue clicking along the edge until all necessary sections are selected.
8. To finish digitizing, right-click and select one of the following commands from the pop-up menu:
   - Line Digitizer > End Line
   - Area Digitizer > Close Area

The feature is created using the geometry of the grabbed object.

Edit Using Grab

Menu	Edit > Feature
Tool	

Pop-up	Edit Line > Redigitize

Tool	
Pop-up	Edit Line > Grab

To edit a feature using Grab:

1. In the Layers window, select a feature layer.
2. Select the feature to be modified (1).
3. Select the **Edit Feature** command.
4. Select a point on the feature (2).

If the selected point is not a start or end point, it is removed when you use the Redigitize command. The line segment to be modified is indicated by a dotted line. This segment will be replaced with the grabbed feature plus any necessary connecting lines.

5. Right-click and select the **Redigitize** command from the pop-up menu.
6. Select the **Grab** tool.

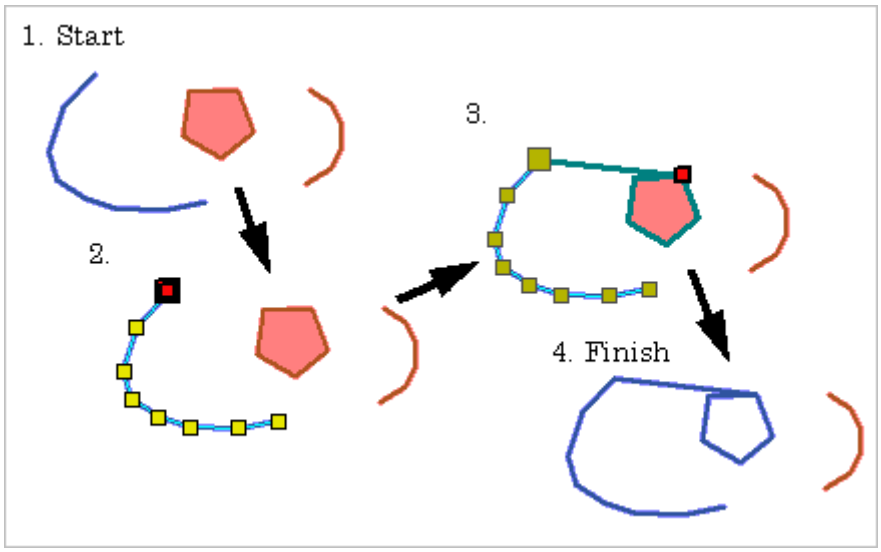
If you grab a line feature, a connecting line is drawn from the highlighted segment of the selected feature to the closest end point of the line grabbed. If you grab an area feature, a connecting line(s) is drawn to the first created, or origin, point of the selected edge for that feature.

7. Click an edge of the feature you want to grab (3).

If the selected feature is an area feature, the new area is included only if the updated feature is valid (the connecting lines to the grabbed feature do not cross). Otherwise, when the Edit tool is deselected, the grab update to the selected feature is cancelled.

8. To finish, deselect the **Edit** tool.

The grabbed feature is now included as part of your selected feature (4).



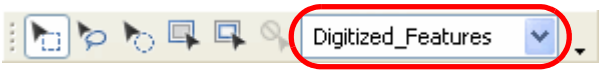
Vectorize from Raster

Grab edges from an image backdrop file to form a vector object.

The vectorizing tool is used to convert lines from a raster image backdrop file into vector objects on a feature layer. Conversion is performed by selecting a line from the raster image and tracing it. This creates a new vector line in the feature layer. The vectorizing tool detects the last pixel that you clicked and follows the same pixel colour as you click to create vertices for the line in the feature layer. The resulting vector line will be traced along the centre of the raster line.

Menu	Create > New Feature > Line / Area
Tool	

1. Open a raster image.
2. Select the layer where you want the digitized feature.
3. Select a **New Line or Area Feature** command.
4. Select the object acronym and assign attribute values.
5. Select the raster layer from the *Snap/Grab Target* list in the toolbar.



Pop-up	Edit Line > Vectorize from Raster

6. Right-click in the Display window and select the **Vectorize from Raster** command.
7. In the raster image, click the line you want to trace.

A vector line follows the raster image until it reaches a decision point. A decision point can be a break in the raster line or a junction.

8. Continue clicking along the line or area of the raster image that you want to capture.
9. When finished, deselect the digitizing tool.

When you view the workspace layer, you will see the digitized image in vector format.

Digitizing using the Coordinates Window

The Coordinates window can be used to digitize vertices of an edge or area at specific geographic locations. With this functionality you can:

- insert new line vertices,
- delete line vertices,
- specify azimuth and distance between vertices, and
- specify coordinates.

When an edge is selected for editing, the nodes of the edge are displayed in the Coordinates window. If you select a node in the

Coordinates window, it will be highlighted in the Display window.

Digitizing line vertices using the Coordinates window

Menu	Create > New Feature > Line / Area
Tool	

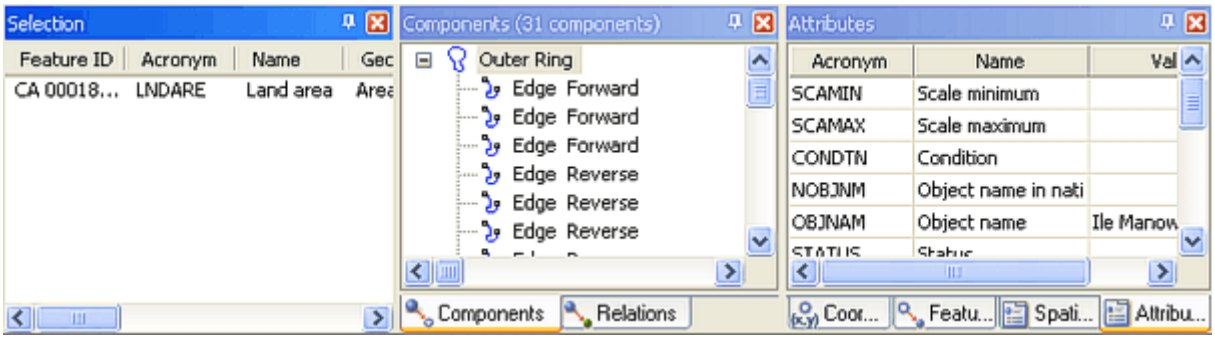
To digitize vertices using the Coordinates window:

1. Select the **New Line** or **New Area Feature** tool.
2. Select the object acronym and attributes of the object to be digitized.
3. Double-click the first row in the Coordinates window.
4. Type the Latitude for the first vertex then press <Tab> to move to the Longitude field.
5. Type the Longitude of the first vertex.
6. Do one of the following:
   - Press <Enter> to complete the first row of the table.
   - Press <Tab> and define the remaining fields in the row, such as the distance to the next vertex.
7. Continue to specify vertices for the new feature.

Menu	View > Refresh
Tool	
Key	<F5>

8. To end digitizing mode, right-click in the Display window and select Digitizing > End Line or Close Line from the pop-up menu.
9. Select the Refresh command.

The following is an example of the information displayed in the tabbed windows after you have selected a feature.

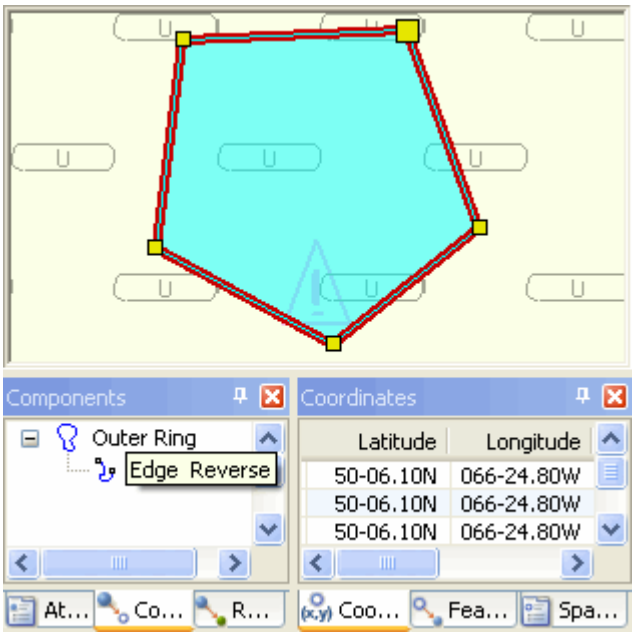


You can zoom and re-centre the Display window on a selected feature by double-clicking the feature in the Selection or Components window.

1. To display the coordinates of a component, such as the edge of the outer ring in the figure above, click **Edge** (highlighted in grey).
2. To activate edit mode for the spatial information in the Coordinates window, select the Edit Feature tool.

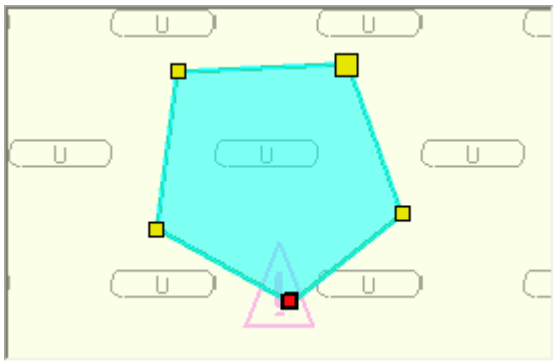
Menu	Edit > Feature
Tool	

The display of feature is refreshed to show its vertices for editing.

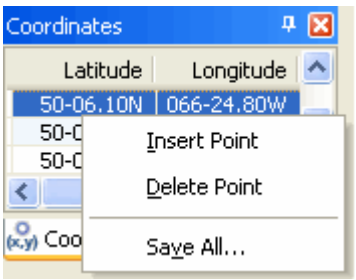


3. In the Coordinates window, scroll left or right to view all the available fields.

When you click an entry in the Coordinates window, that node is highlighted red in the Display window. In the following example, the bottom-most node of the selected feature is selected.



4. Type new values in any of the fields as necessary, or manually move, delete or add nodes as described in "EDITING FEATURES" ON PAGE 140.
5. Right-click in the Coordinates window to access a pop-up menu.



From this menu you can insert or delete a node, or save the coordinates list to a file.

- To insert a point, select a node in the Coordinates window. The new point will be added above this. Right-click and select Insert Point from the pop-up menu. Type the appropriate values in the latitude and longitude fields.
- To delete a point, select the node to be removed, right-click and select Delete Point.
- To save a list of the feature coordinates to a text file, right-click and select Save All.

Menu	Edit > Feature
Tool	

6. To exit edit mode, deselect the Edit Feature tool.

New features

You can create four types of features:

Feature Type	Toolbar Button	Menu Command
Point		Create > New Feature > Point
Line		Create > New Feature > Line
Area		Create > New Feature > Area
Sounding		Create > New Feature > Sounding

For specifics on selecting object acronyms and setting attributes, see "CREATE FEATURES" ON PAGE 107.

For information on digitizing and editing features, see:

- "DIGITIZING FEATURES" ON PAGE 112
- "EDITING FEATURES" ON PAGE 140

Create new features on a selected feature layer

New Point Feature

To create a new point feature in a feature layer. For more information about creating new features, see"CREATE FEATURES" ON PAGE 107.

Menu	Create > New Feature > Point
Tool	

1. Select a layer containing point features.
2. Select the (New Feature) Point command.
3. In the Select Object Acronym dialog box, select the type of feature to be created and click **OK**.
4. In the Attributes dialog box, assign values to at least the mandatory attributes and click **OK**.
5. Specify coordinate information or click at the location in the Display window where the point feature is to be placed.

A new point feature is displayed. The appearance of point features are determined by the object acronym selected.

For more information about creating new features, see "CREATE FEATURES" ON PAGE 107

New Line or Area Feature

Create new line and area features in a feature layer. For more information about creating new line features, see "CREATE FEATURES" ON PAGE 107

Menu	Create > New Feature > Line
Tool	
Menu	Create > New Feature > Area
Tool	

1. Select a (New Feature) Line or Area command.
2. In the Select Object Acronym dialog box, select the feature type to be created and click **OK**.
3. In the Attributes dialog box, assign values to at least the mandatory attributes and click **OK**.
4. Digitize the line or area using any of the following digitizing options:

- "STREAM LINE" ON PAGE 113
- "FITTED CURVE" ON PAGE 117
- "SPLINE" ON PAGE 117
- "DIGITIZING ARCS" ON PAGE 118
- "DIGITIZING CIRCLES" ON PAGE 122
- "DIGITIZING RECTANGLES" ON PAGE 125

You are not limited to using only the line type you first select. You can select a different type "on the fly" to continue digitizing the line from the last point digitized.

5. When you are finished digitizing:
   - De-activate the new feature (New Line or New Area) button, or
   - Select Edit Line > End line from the pop-up menu, or
   - Press <**End**>.

The new feature is digitized on the selected layer.

To create an enclosed feature from a line or arc type, press <Home> and the end of the line will join with the first point created.

New Sounding Feature

Create a new sounding feature on a feature layer.

Menu	Create > New Feature > Sounding
Tool	

1. Select a feature layer in the Layers window.
2. Select the (New Feature) Sounding command.

The Attributes dialog box is displayed with attributes for the default SOUNDG feature.

3. Set any attributes required.
4. Specify coordinate information for the sounding feature and click OK, or
5. Click in the Display window where the sounding feature is to be placed.

A new sounding feature is created on the selected feature layer.

Editing Features

Tools that can be used to edit features include:

Copy Feature

Copy a superselected feature using one of the following methods:

- Create a new feature using the object class and attributes of a superselected feature.
- Create a new feature using the same geometry or class as an existing selected feature.

Class

Create a new feature by copying the object class and attributes of an existing, superselected feature.

1. Select a feature layer in the Layers window.
2. Superselect the feature from which the object class and attributes are to be used.
3. Select the (Copy Feature) Class command.

Menu	Create > Copy Feature > Class
Tool	

The appropriate digitize mode for the feature type is activated. See "DIGITIZING FEATURES" ON PAGE 112 for information on the available digitizing methods.

4. Digitize the new feature.

The new feature is created.

Geometry

Create a new feature by copying the geometry of an existing, superselected feature.

1. Superselect the feature from which the geometry is to be used.
2. Select the (Copy Feature) Geometry command.

Menu	Create > Copy Feature > Geometry
Tool	

The Select Object Acronym dialog box is displayed.

3. Select the acronym for the new feature.

The Attributes dialog box is displayed.

4. Define the required attributes.

5. Click **OK**.

The new feature is created.

Repeat New Feature

Create a new feature with identical acronym and attribute values to the one last created.

Menu	Create > Repeat New Feature
Tool	

1. Select the Repeat New Feature command.
2. Perform the same steps described in "CREATE FEATURES" ON PAGE 107.

After digitizing the feature the cursor does not return to normal, but remains in digitizing mode.

3. Continue clicking in the Display window to add more new features.
4. To stop digitizing, either:
   - Select the Repeat New Feature command a second time BEFORE digitizing the last feature.
   - Select the New Feature command a second time AFTER digitizing the last feature.

The cursor returns to normal and the new features are available for editing.

Connecting Line

Close small gaps between two lines by adding a short point-to-point connecting line (a new line feature), which snaps to vertices on existing lines, intersecting them.

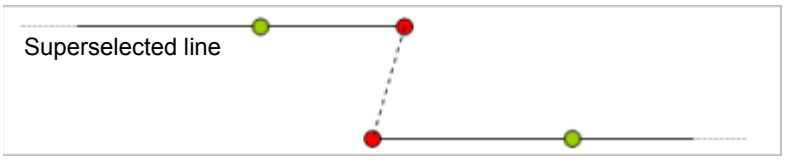
A new line feature is created with a new Feature Object ID. The new line inherits the object class acronym and attributes from the superselected (starting) line.

Afterward, the line can be drawn, selected, and further edited as necessary.

If any dangling lines are created by snapping/intersecting, you must select and delete these lines manually as a separate step afterwards.

Connect line end points

In this example, you join the end point of a superselected line to the end point of a second line.



1. Superselect the line from which the new connecting line will be added.

Menu	Create > Connecting Line

2. Select the Connecting Line command.
3. Press and hold the left mouse button, and move the cursor over the superselected end point.
4. Press <I> for snap-intersect, and click the end point of the superselected line.

The nearest point is highlighted by a diamond-shaped snap marker.

5. If the correct point is not highlighted:
   - Press <Spacebar> to find the next closest point (repeat as necessary).
   - If no vertex is found, move the cursor along the superselected line and repeat steps 3 and 4.
6. [Optional] Use the Grab Line command to use existing line segments.

You are now ready to complete the connection.

Menu	Edit > Digitize > Grab Line
Tool	
Pop-up	Line > Digitizer > Grab Line

7. Press and hold the left mouse button, press <I>, and click on the end point of the line to be joined to the superselected line.

The nearest point is highlighted by a diamond-shaped snap marker.

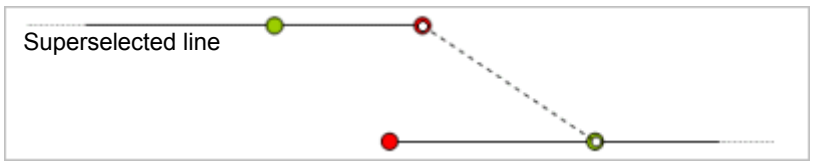
8. If the correct point is not highlighted:
   - Press <Spacebar> to find the next closest point (repeat as necessary).

- If no vertex is found, move the cursor along the second line and repeat step 7.

The lines are now joined by the new segment.

Connect end point to vertex of second line

In this example, you join the end point of a superselected line to a vertex on the second line other than the end point.

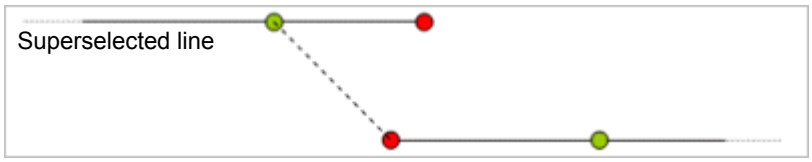


1. Repeat the procedure above, but in step 6 select a non-end point to snap-intersect.

Intersecting the second line splits that line into two new line features.

Connect vertex to end point of second line

In this example, you join a vertex other than the end point of the superselected line to the end point of the second line.



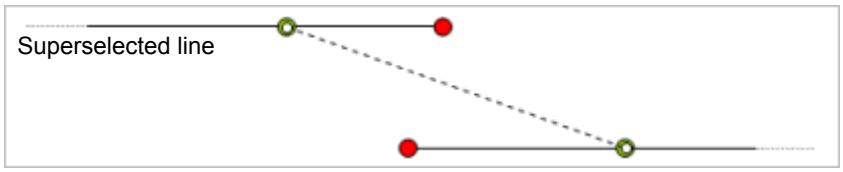
1. Repeat the procedure for joining two end points (above), but select a vertex on the superselected line other than the end point.

In this instance, the superselected line is split into two new line features.

2. Select the end point of the second line to create the join line.

Join lines at vertices

In this example, you join a vertex on the superselected line to a vertex on the second line. Both lines are split.



1. Select a vertex other than the end point on both the superselected line and the second line.

Area from Selection

Create a new feature using the exact geometry of an existing feature or multiple adjacent features (e.g., features that have a common boundary). This command can apply to a selection of lines or a selection of areas.

The original features remain intact, while the new feature has only the duplicate geometry.

To create an area from a selected feature object:

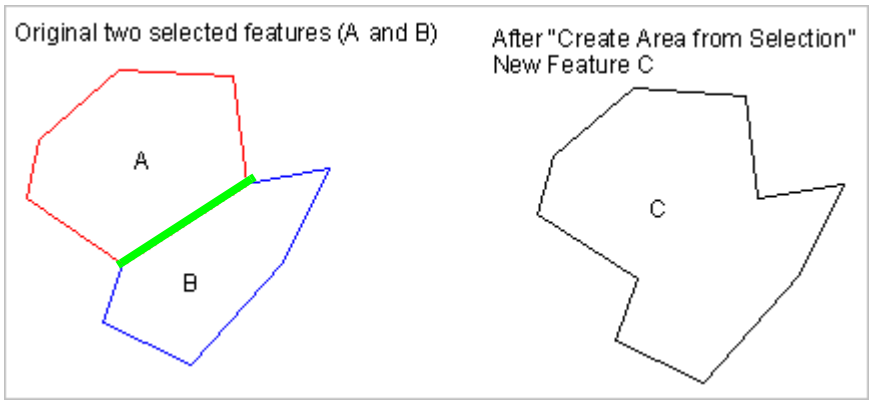
1. Select one or more features to make up the geometry of the new feature (based on the conditions described above).
2. Select the (Create) Area from Selection command.
3. Assign attributes to the new feature.

Menu	Create > Area from Selection

From examples describe how the command works.

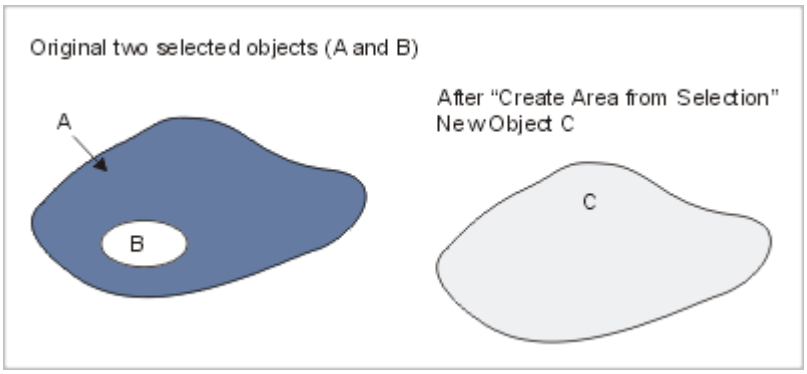
Example 1:

For example two areas have a common boundary (highlighted in green). Selecting these two features and running the Create Area from Selection command results in a new area C.



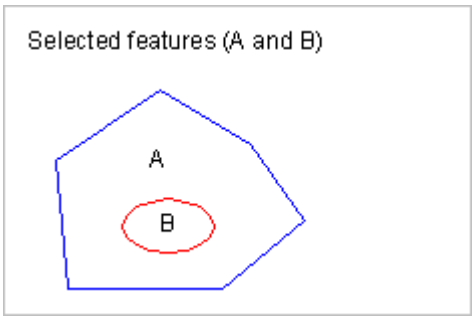
Example 2:

In this example, assume you have two distinct areas such as two DEPARE features. Depth area A has a hole, which is filled by depth area B. You can create a new feature from these two features because they share a common boundary. The result is Feature C — without a hole.



Example 3:

Assume you have a BUAARE (B) on top of a LNDARE (A). Trying to create a new area feature by selecting these two features would generate an error because the polygons do not share a common boundary.



Create Filtered Layers

Create layers for specific types of data by applying a filter to any of the following layer types:

- S-57 data
- HOB data
- Feature layer
- Database connection

The layers can be based on attribute value, feature acronym, feature object ID (FOID) or feature type. You can also use filter rules, which essentially provide a combination of multiple filters.

Layers are listed in the Layers window when they are created. Layers can be shown or hidden by selecting or clearing the check box beside the layer name.

For more information on creating layers, see the following sections:

- "Create Layer by Attribute Value" on page 147
- "Create Layer by Feature Acronym" on page 148
- "Create Layer by Feature Object ID" on page 149
- "Create Layer by Feature Type" on page 149
- "Create Layers by Unique Feature Acronyms" on page 150
- "Create Layer using the Rule Wizard" on page 152
- "Create Layer using a Rule File" on page 165

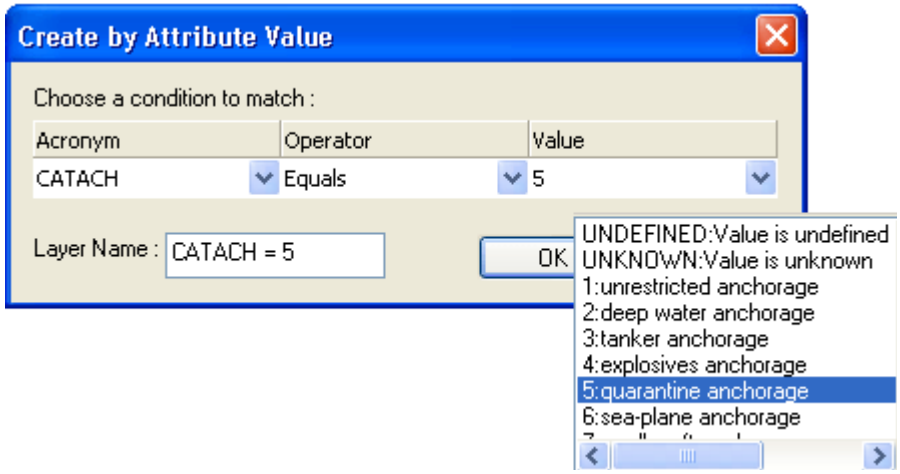
Create Layer by Attribute Value

This filter enables you to create a layer based on attribute values. All features using the value specified for a particular attribute will be added to a new child layer. This is a useful filter if you need to apply a change to all features with a specific attribute value.

Menu	Create > Layer By > Attribute Value
Tool	
Pop-up	Create Layer > Attribute Value

1. Select the (Layer By) Attribute Value command.

The Create Layer by Attribute Value dialog box is displayed.



2. Select an attribute acronym from the list.

The acronyms available will differ depending on the data being filtered.

3. Select an operator from the list:
   - Equals
   - Does not equal
   - Less than
   - Less than or equal
   - Greater than
   - Greater than or equal
4. Type a value to be matched.
5. Click in the *Layer Name* box to automatically generate a name based on the condition, or type your own name for the layer.
6. Click **OK**.

A layer containing the selected features is added to the Layers window and displayed in the Display window.

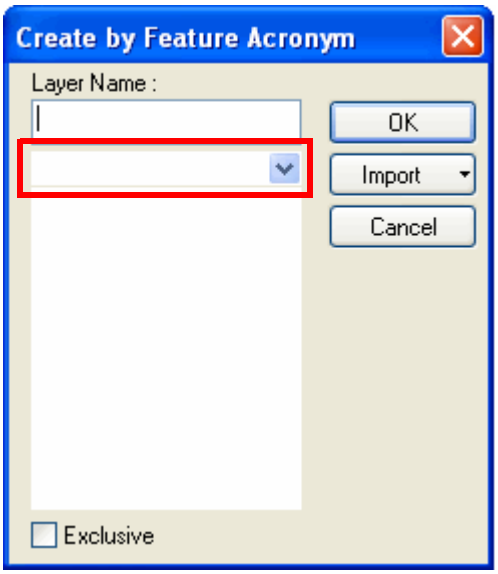
Create Layer by Feature Acronym

This function enables you to create a feature layer that filters for specific acronyms. This will enable you to locate all objects in the map using the specific acronyms. For example, filtering for all of the buoy feature acronyms would locate all buoys in an area.

Menu	Create > Layer By > Feature Acronym
Tool	
Pop-up	Create Layer > Feature Acronym

1. Select the HOB or S-57 layer in the Layers window.
2. Choose the (Layer By) **Feature Acronym** command.

The Create by Feature Acronym dialog box is displayed.



3. Select data using either of the following methods:
   - Click a *Feature Acronym* cell (see image) and select an object acronym from the list.
   - Add object acronyms to the clipboard (using the cut or copy command) from another application, such as Notepad, and select *From Clipboard* from the Import list.

The acronyms available in the list will differ depending on the data being filtered. If you add acronyms from the clipboard, make sure they are uppercase.

4. To include all features except those that meet the specified criteria, select the *Exclusive* option.
5. Click **OK**.

All objects with the selected acronym(s) are included in the new layer. If *Exclusive* was selected, the new layer contains all features except those with the selected acronym(s).

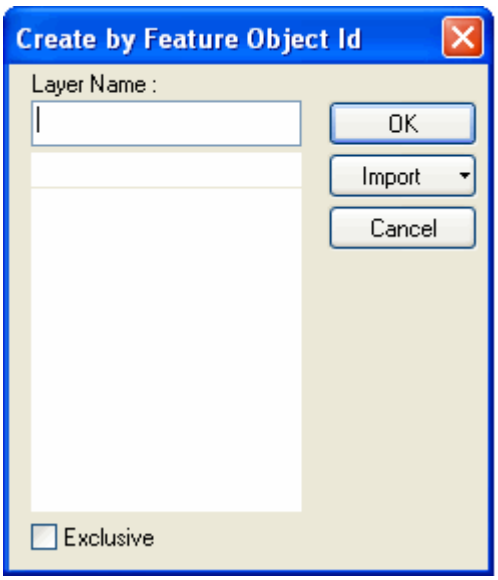
Create Layer by Feature Object ID

Create layers using Feature Object IDs. Because all features have unique IDs, you can use this filter to search for specific objects in a map using the object IDs.

1. Select the HOB or S-57 layer in the Layers window.
2. Choose the (Layer By) Feature Object ID command.

The Create by Feature Object ID dialog box is displayed.

Menu	Create > Layer By > Feature Object ID
Tool	
Pop-up	Create Layer > Feature Object ID



3. Select objects by any of the following methods:
   - Click in a *FOID* cell and type an ID. You can include more than one.
   - Add object feature IDs to the clipboard (using the cut or copy command) from another application, such as Notepad, and select From Clipboard from the Import list
4. To exclude the chosen features from the selection, select the *Exclusive* option.
5. Click **OK**.

A layer containing the features with the selected IDs are displayed (or not displayed, if you selected the *Exclusive* option).

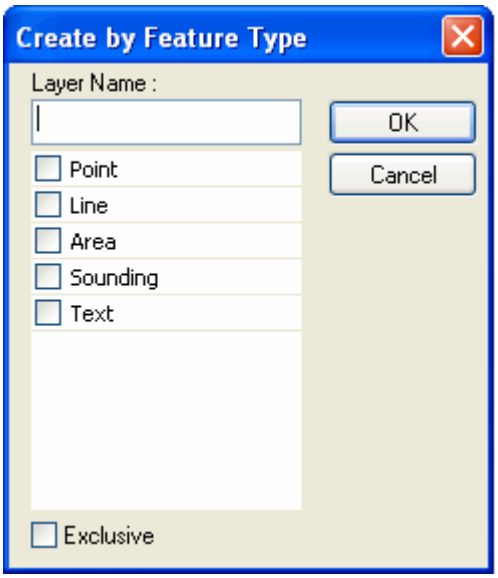
Create Layer by Feature Type

Create layers according to feature type. When this filter is applied, only features of the specified type will be accessible through the layer. This would be useful if you needed to select or edit all features of a specific type. For example, if you wanted to import all of the point features from a map into a feature layer, you could filter for point features and then select the *Select All* command. All point features would be selected.

Menu	Create > Layer By > Feature Type
Tool	
Pop-up	Create Layer > Feature Type

1. Select the HOB or S-57 layer in the Layers window.
2. Choose the (Layer By) Feature Type command.

The Create by Feature Type dialog box is displayed.



3. Select one or more feature object types.
4. Select the *Exclusive* option to create a layer that includes all features except the selected feature type.
5. Click **OK**.

A layer is created that includes the selected feature types.

Create Layers by Unique Feature Acronyms

This filter is similar to the Feature Acronym filter, except that it does not create just one layer for a specific acronym.

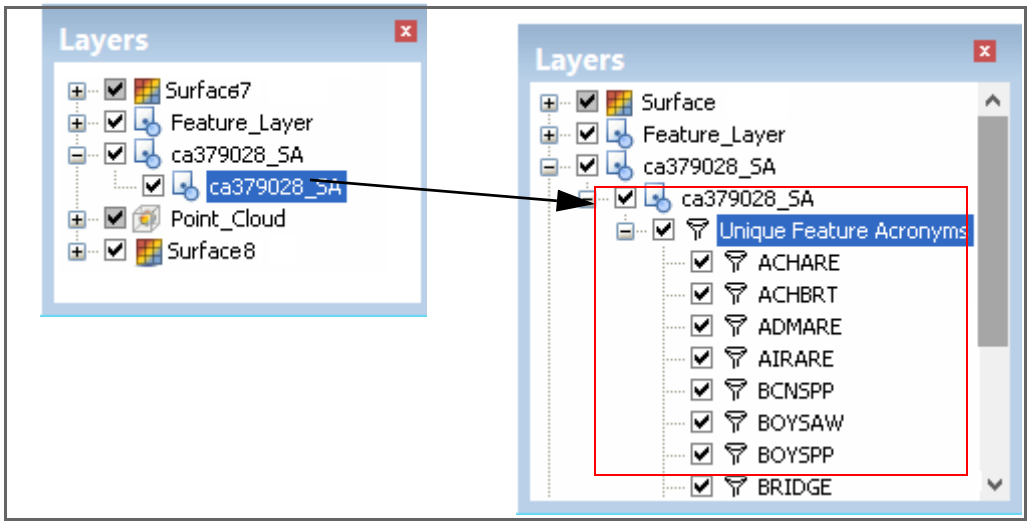
This filter creates a separate layer for every object acronym in the selected layer. Each child layer is named according to the object acronym it is filtering for. All objects with that acronym can be identified through the layer.

For example, if you were to select one of the layers then click Select All, all objects with that acronym would be selected.

Menu	Create > Layer By > Unique Feature Acronyms
Tool	
Pop-up	Create Layer > Unique Feature Acronyms

1. Select a feature layer, from the Layers window.
2. Select the (Layer By) Unique Feature Acronyms command.

A new feature layer is created called Unique Feature Acronyms. This layer contains a new “child” layer for each feature acronym in the selected map.


Layers
Surface7
Feature_Layer
ca379028_SA
ca379028_SA
Point_Cloud
Surface8
Layers
Surface
Feature_Layer
ca379028_SA
ca379028_SA
Unique Feature Acronyms
ACHARE
ACHBRT
ADMARE
AIRARE
BCNSPP
BOYSAW
BOYSPP
BRIDGE

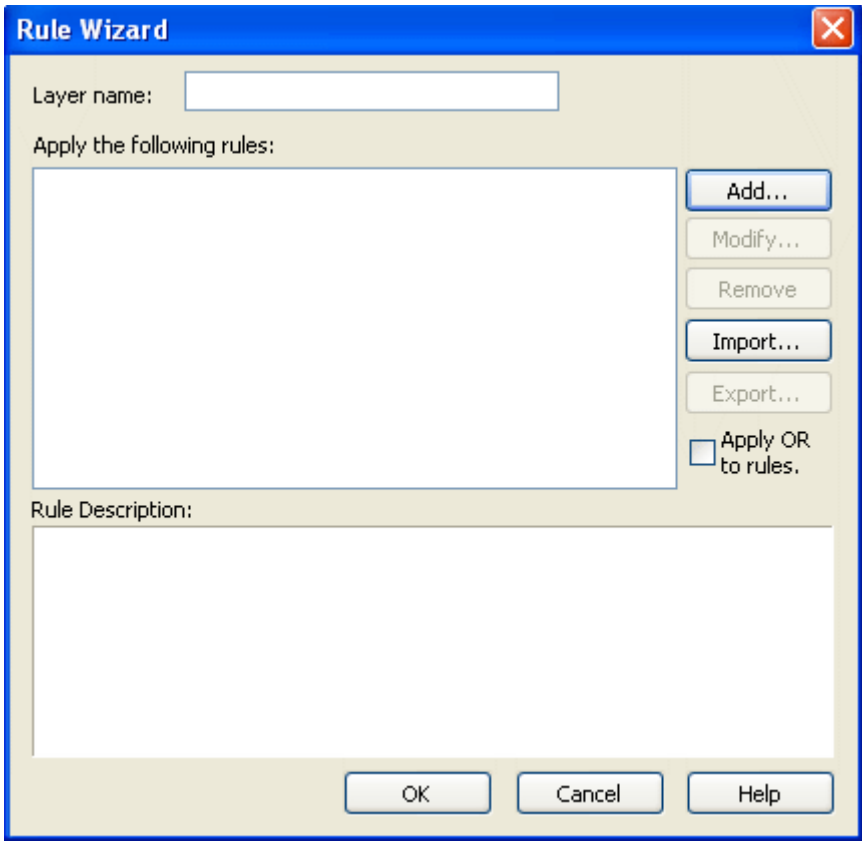
Create Layer using the Rule Wizard

Menu	Create > Layer By > Rule Wizard
Tool	
Pop-up	Create Layer > Rule Wizard

The Rule Wizard creates custom layers using filtering rules. Each rule is defined by one or more conditions, which would be like using a combination of different filtering tools. You can apply any number of rules to a layer and save the rules to re-open in another dataset. You can also modify a saved rule.

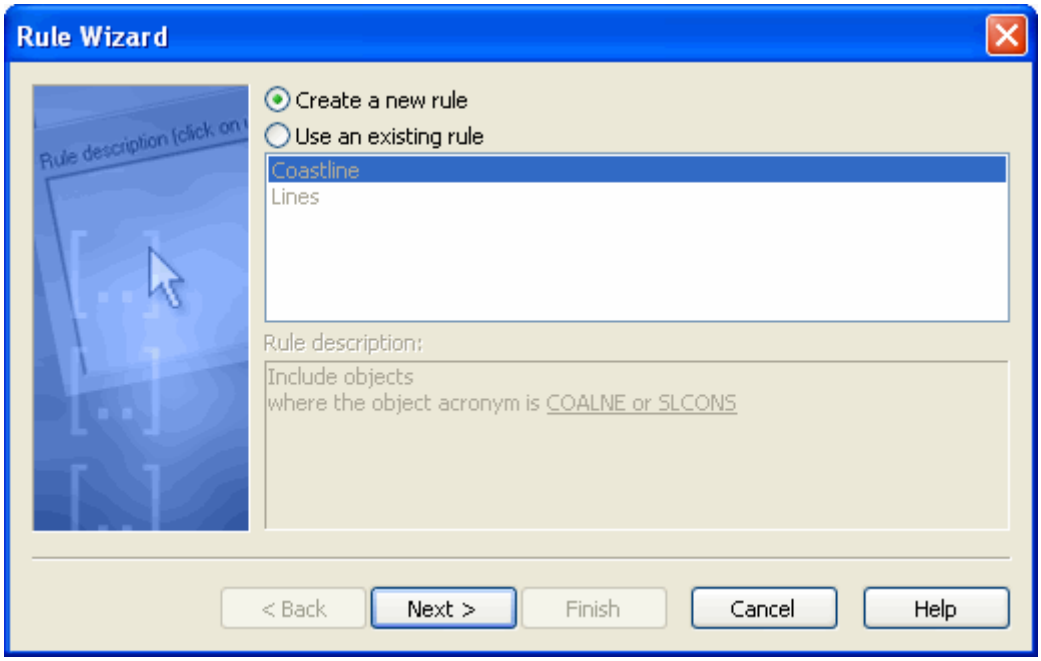
1. Select the Rule Wizard command.

The Rule Wizard dialog box is displayed.



2. Type a name for the layer in the *Layer name* field.
3. To create a new set of rules, click **Add**.

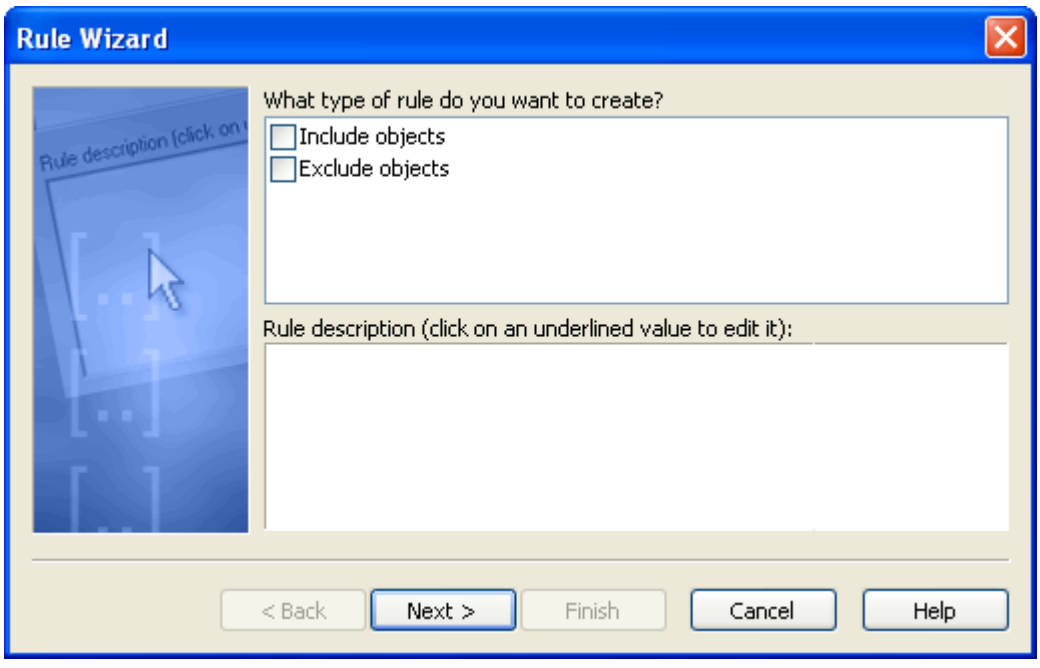
The Rule Wizard dialog box is displayed.



In this step you have the option of creating a new rule, or editing an existing rule.

4. Select *Create a new rule* (selected by default) or *Use an existing rule*.
5. If you chose to create a new rule, click **Next** and continue with the next step. If you chose to use an existing rule, select a rule from the list and click **Next**.

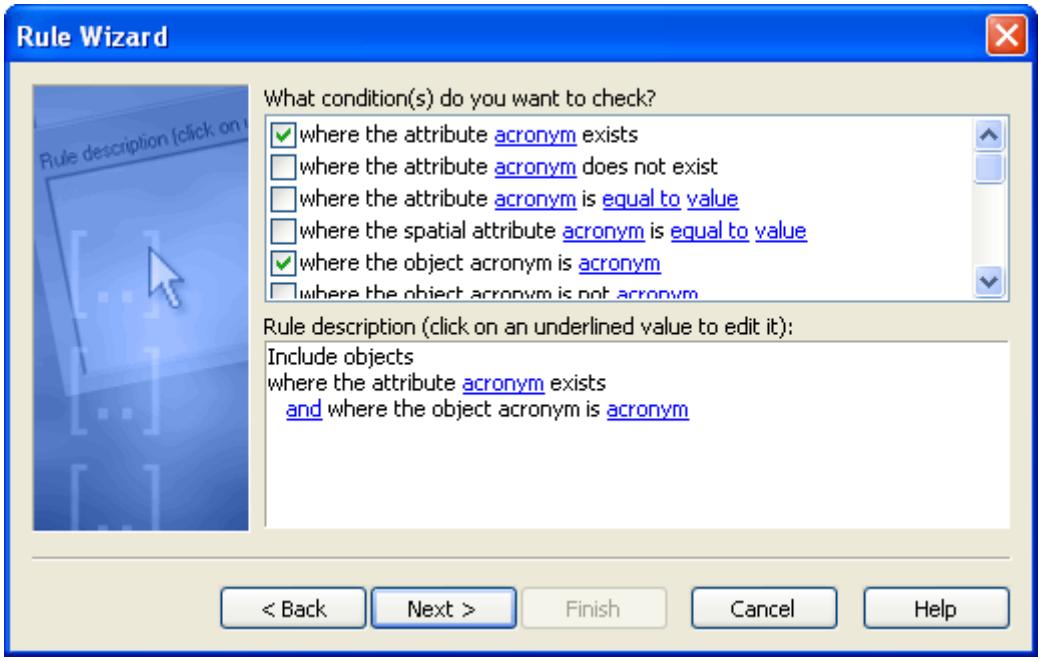
The rule type options are displayed.



These options enable you to define that the new layer will contain *only* objects that meet the specified criteria, or that the layer will include all objects *except* those that meet the specified criteria.

6. Select a rule type option (*Include objects* or *Exclude objects*).
7. Click **Next**.

The conditions available for the new rule are listed.

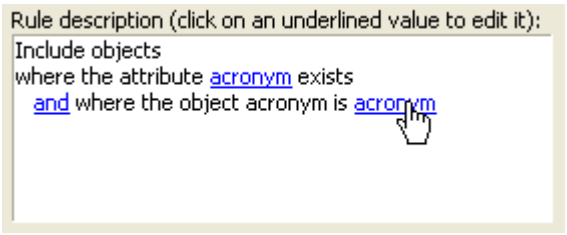


The top list contains the conditions that can be applied to the rule.

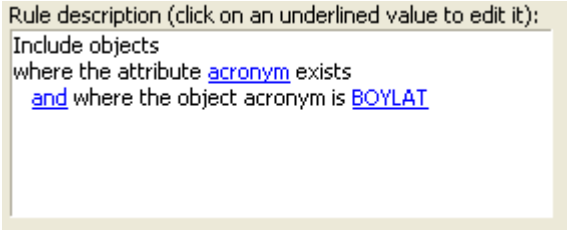
8. Select the conditions that you want to use for creating the filter.

When a condition is selected, it is displayed in the *Rule description* pane of the dialog box.

9. Click an underlined link in the *Rule description* pane to select the appropriate acronym, feature type, or value needed to define the condition.



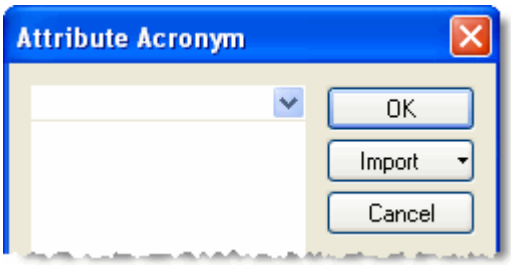
The selections are displayed in the description as in the example below.

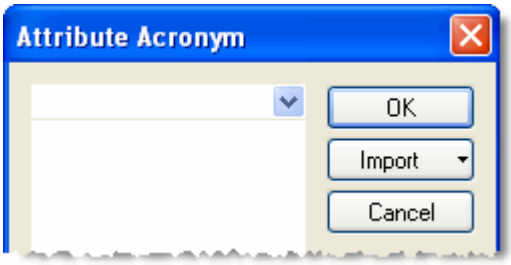
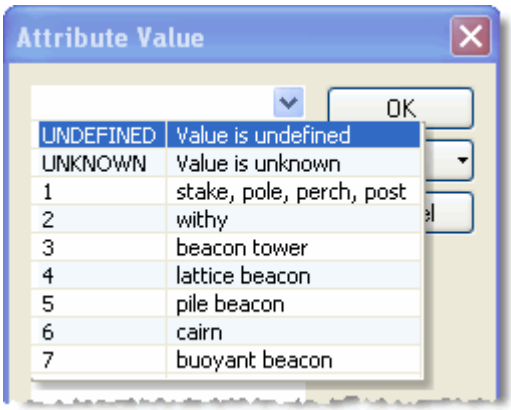


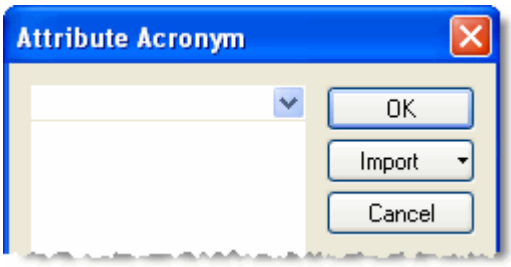
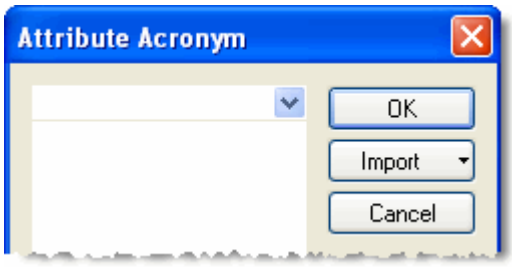
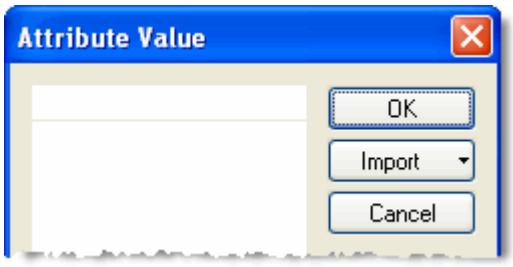
Rules that include a unit of measure will use the units and Measurements Reference settings selected in the Display Units tab in Options.

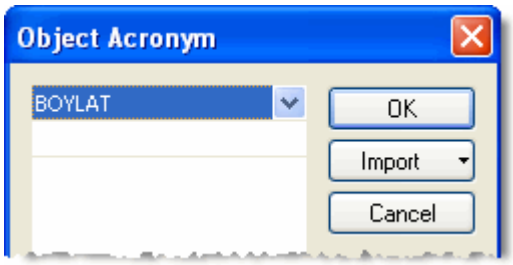
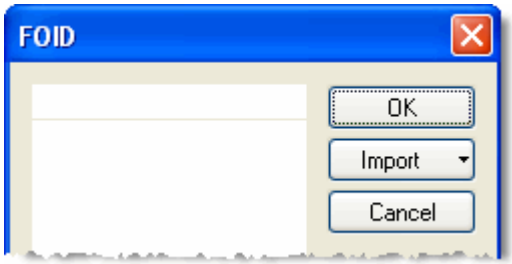
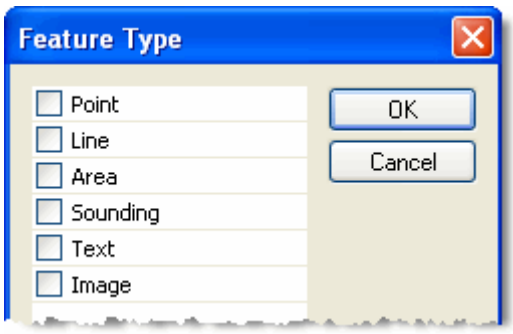
If the *Measurements Reference* option is set to Projected, but there is no projection since the view is in a Geographic coordinate system, the filter will use Geodetic measurements.

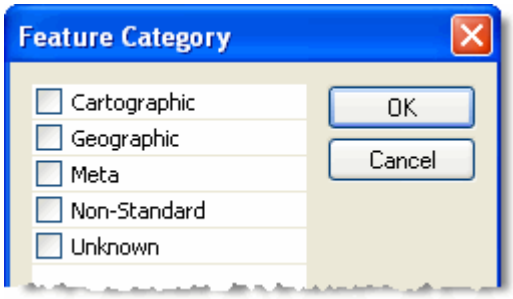
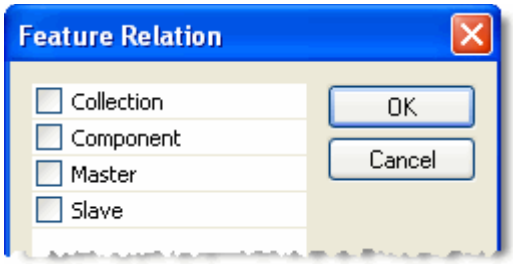
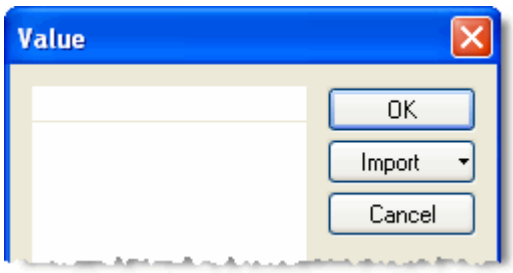
The following table contains a list of conditions and brief descriptions on how to select values for each condition.

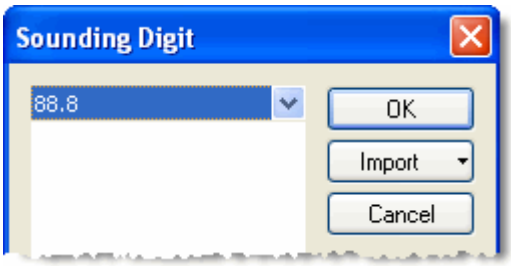
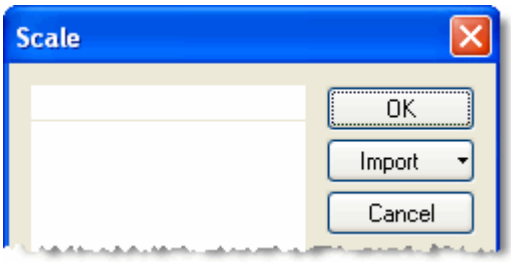
Condition	Procedure
Where attribute acronym exists. Where attribute acronym does not exist.	1. Click the attribute *acronym* link. The Attribute Acronym dialog box is displayed.  2. Select an attribute acronym from the list. 3. Click **OK**.

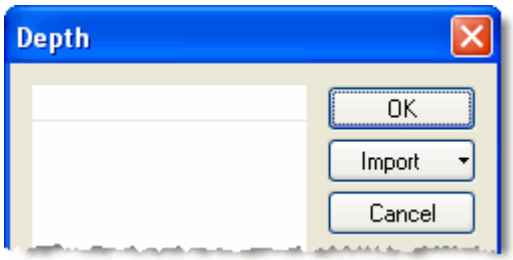
Condition	Procedure
Where the attribute acronym is equal to value. Where the attribute acronym is not equal to value. Where the attribute acronym is less than value. Where the attribute acronym is less than or equal to value. Where the attribute acronym is greater than value. Where the attribute acronym is greater than or equal to value.	1. Click the attribute *acronym* link. The Attribute Acronym dialog box is displayed.  2. Select an attribute acronym from the list. 3. Click **OK**. 4. Click the *value* link. The Attribute Value dialog box is displayed.  5. Select a value from the list (values shown in the dialog box are determined by the selected attribute acronym). 6. Click **OK**.

Condition	Procedure
Where the attribute acronym contains value. Where the attribute acronym does not contain value.	1. Click the *acronym* link. The Attribute Acronym dialog box is displayed.  2. Select either SoftwareVersion or SoftwareName from the list. 3. Click **OK**. 4. Click the *contains/does not contain* link to toggle from one to the other. 5. Click the value link. 6. Enter the name or version number of the software that was used to create the data that you wish to select. 7. Click **OK**.
Where the spatial attribute acronym is equal to value. Where the spatial attribute acronym is not equal to value. Where the spatial attribute acronym is less than value. Where the spatial attribute acronym is less than or equal to value. Where the spatial attribute acronym is greater than value. Where the spatial attribute acronym is greater than or equal to value.	1. Click the *acronym* link. The Attribute Acronym dialog box is displayed.  2. Select an attribute acronym from the list. 3. Click **OK**. 4. Click the *equal to* link to select the appropriate operator. 5. Click the value link. The Attribute Value dialog box is displayed.  6. Enter a value in the text entry field. 7. Click **OK**.

Condition	Procedure
Where object acronym is acronym. Where object acronym is not acronym.	1. Click the *acronym* link. The Object Acronym dialog box is displayed.  2. Select an object acronym from the list. 3. Click **OK**.
Where the feature ID is value.	1. Click the *value* link. The FOID dialog box is displayed.  2. Type a value in the field. 3. Click **OK**.
Where feature type is type. Where feature type is not type.	1. Click the *type* link. The Feature Type dialog box is displayed.  2. Click a Feature Type check box so it is selected. 3. Click **OK**.

Condition	Procedure
Where feature category is type. Where feature category is not type.	1. Click the *type* link. The Feature Category dialog box is displayed.  2. Click a Feature Category check box so it is selected. 3. Click **OK**.
Where feature relation is relation. Where feature relation is not relation.	1. Click the *relation* link. The Feature Relation dialog box is displayed.  2. Click a Feature Relation check box so it is selected. 3. Click **OK**.
Where the area is smaller than value *square metres*. Where the area is larger than value *square metres*.	1. Click the *smaller than*/*larger than* link to toggle from one to the other. 2. Click the *value* link. The Value dialog box is displayed.  3. Type an area value. 4. Click **OK**.

Condition	Procedure
Where the width is thinner than value *metres*. Where the width is wider than value *metres.*	1. Click the *thinner than/wider than* link to toggle from one to the other. 2. Click the *value* link. The Value dialog box is displayed. 3. Type a width value. 4. Click **OK**.
Where the perimeter is shorter than value *metres*. Where the perimeter is longer than value *metres*.	1. Click the *shorter than/longer than* link to toggle from one to the other. 2. Click the value link. The Value dialog box is displayed. 3. Type a perimeter value. 4. Click **OK**.
Where the object is too small to enclose a sounding digit at a scale of value.	1. Click the *sounding digit* link. The Sounding Digit dialog box is displayed.  2. Select a digit option. 3. Click **OK**. 4. Click the *scale* link. The Scale dialog box is displayed.  5. Type a scale value. 6. Click **OK**.

Condition	Procedure
Where the depth is equal to value. Where the depth is not equal to value. Where the depth is less than value. Where the depth is less than or equal to value. Where the depth is greater than value. Where the depth is greater than or equal to value.	1. Click the *equal to* link to toggle between the operators. 2. Click the *value* link. The Depth dialog box is displayed.  3. Enter a value in the text entry field. 4. Click **OK**.

When using the width condition with line features, wizard performs the following tasks to calculate a width value for lines:

a) The points of each feature are transformed into the coordinate system of the view, if not already the same.

b) A convex hull is built from the points.

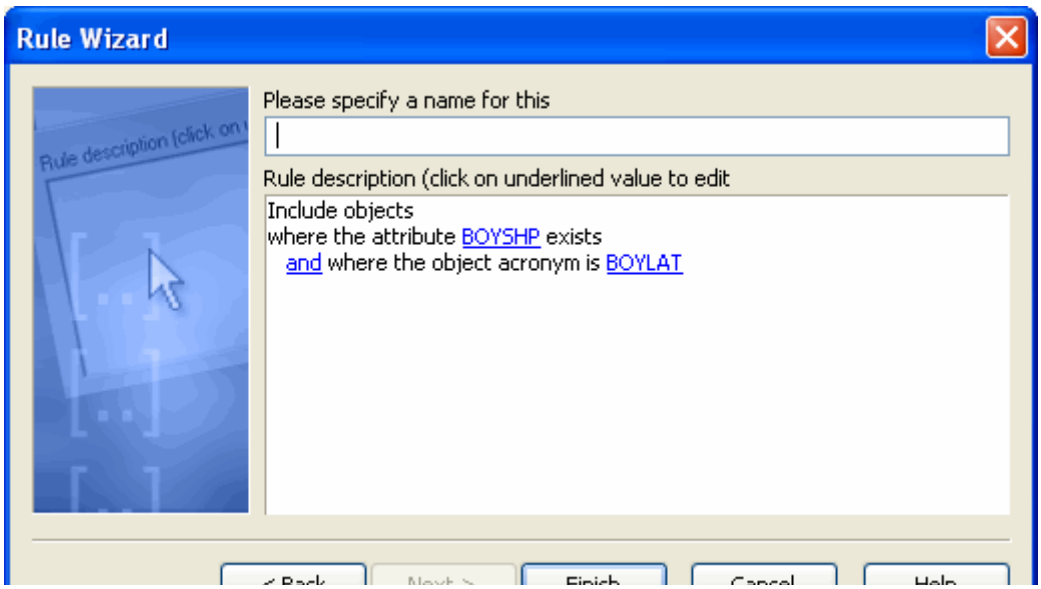
c) The width and height of the hull are calculated.

The smallest value (width or height) is used to compare against the specified 'width' value during conflict resolution.

You can use the area, width, perimeter and sounding digit conditions to create layers for tiny contour features.

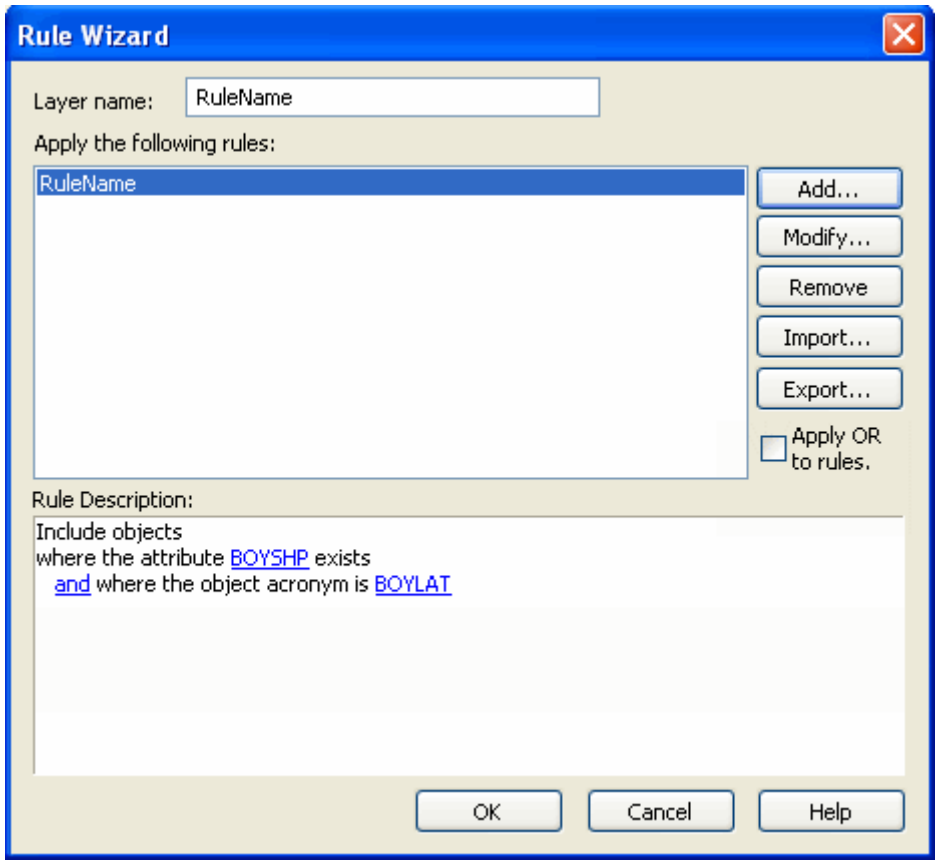
10. Repeat Steps 8 and 9 as necessary.
11. Click **Next** to continue.

The Rule Wizard: Rule name dialog box is displayed.



12. Type a name for the new rule. If you are modifying an existing rule, give it a new name,
13. Click **Finish**.

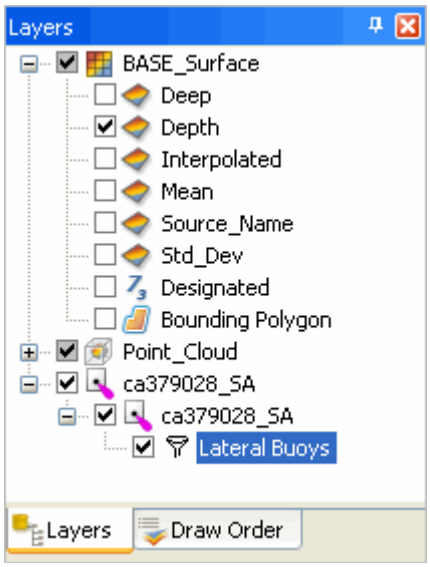
The first Rule Wizard dialog box opens and the new rule is displayed.



Rules can be saved in XML format and reopened to create layers in other datasets. Rules are saved as CRFX files and can be viewed in a text editor or XML-viewer.

14. [Optional] To change the order in which rules are applied to the layer, click a rule and while pressing the mouse button drag it to a new position in the list, then release the mouse button.
15. [Optional] To export a rule to a CRFX file, select a rule, select a file path and file name for the rule and click **Export**.
16. [Optional] To apply any of the rules from the list that are relevant to the data selected, select the *Apply OR to rules* option.
17. Click **OK** to create a layer using the filter.

The layer is displayed in the Layers window.



Open or Close Layers

Close custom layers in the Layers window.

1. In the Layers window, select a layer.
2. Right-click over the layer and, from the pop-up menu, select the **Close** command.

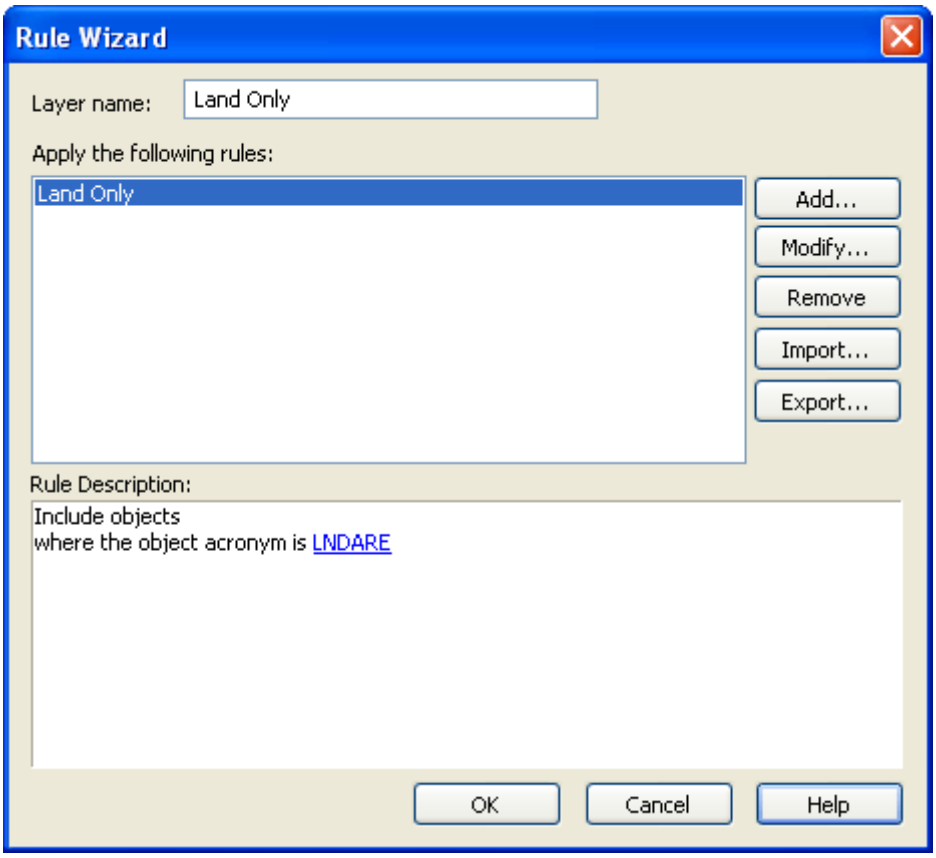
The layer is removed from the file tree.

Modify

Change layer properties (such as editing, adding or removing rules, changing rule conditions, etc.).

1. Select the custom layer to change, right-click and, from the pop-up menu, select **Show/Edit Filter** command.

The Rule Wizard dialog box is displayed.



2. Click **Import**.
3. Select a rule filter (CRFX) file.
4. Click the following buttons to modify the filter.

Button	Description
Add	Create a new layer.
Modify	Open the Rule Wizard dialog box to modify the existing filter.
Remove	Permanently remove a filter rule from the layer.
Import	Open an existing filer CRFX file.
Export	Save filters to a CRFX file using a name you specify.

5. Click **OK** to apply the changes.

Create Layer using a Rule File

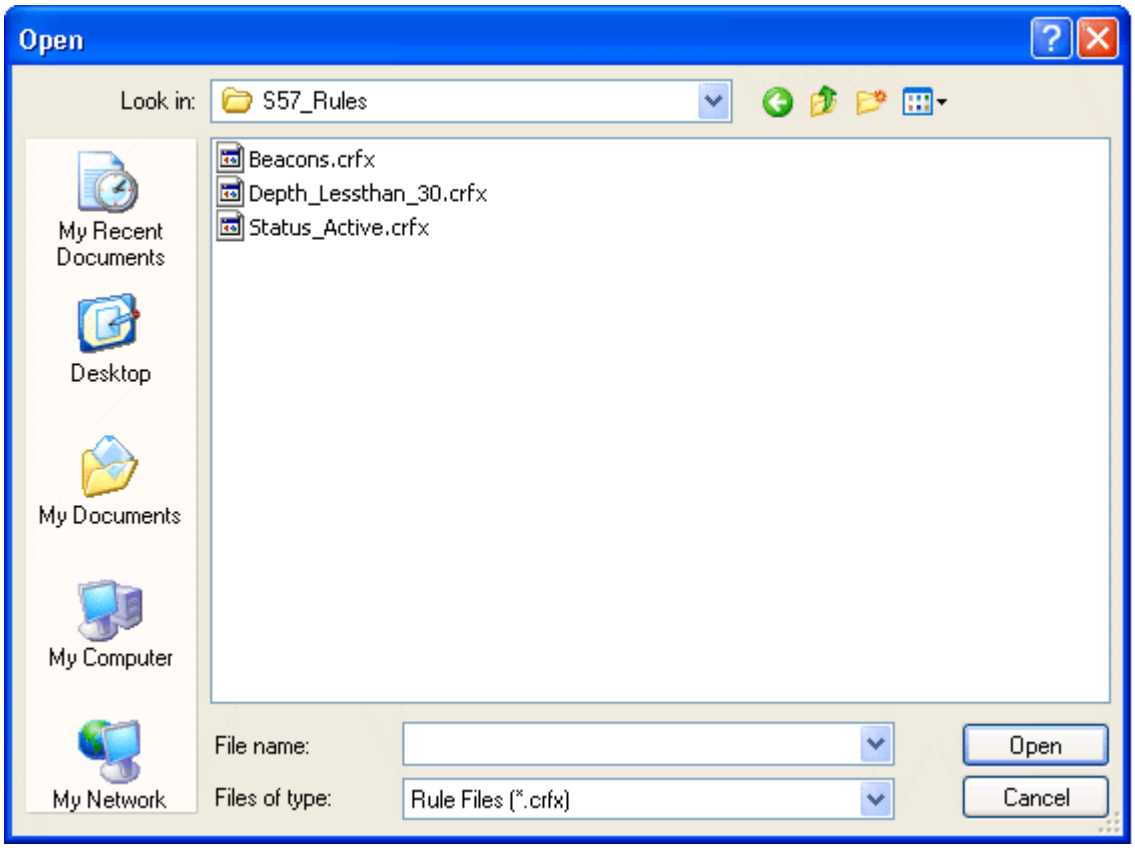
The Rule File filter creates custom layers using filtering rules. Each rule can contain a number of conditions that determine the rule. This would be similar to using a combination of different filtering tools. Rule files are created using the Rule Wizard and have a *.crfx file extension. See "CREATE LAYER USING THE RULE WIZARD" ON PAGE 152 for information on creating a rule file.

Menu	Create > Layer By > Rule File
Tool	
Pop-up	Create Layer > Rule File

To create a layer using a rule file:

1. Select a feature layer in the Layers window.
2. Select the (Layer By) **Rule File** command.

The standard Open file dialog box is displayed.



3. Navigate to the location of the appropriate rule file.
4. Select the file and click **Open**.

A child layer is created under the selected feature layer, containing only features that meet the filter criteria. To view only the features in the filtered layer, turn off all other layers.

Right-clicking on the filtered layer will provide an option to edit the filter being applied to the layer. Selecting this option will open the Rule Wizard.

Tips for Digitizing and Editing

The following are common aspects of digitizing that will help to make digitizing simpler:

Function	Description
Save changes	Save your changes frequently.
Snapping	Accurately digitize objects by using (snapping to) existing vertices or nodes when creating new features. See "SNAP AND SNAP-INTERSECT" ON PAGE 129.
Coordinate window	Use the Coordinates window to specify coordinates, distances, etc. for vertices when digitizing or editing features. See "DIGITIZING USING THE COORDINATES WINDOW" ON PAGE 133.
Cursors	Various cursors are used to indicate the digitizing method that is enabled. See "DIGITIZING CURSORS" ON PAGE 169.
Keyboard shortcuts	A variety of keyboard shortcuts are available for use while digitizing. See "KEYBOARD DIGITIZING SHORTCUTS" ON PAGE 166.
Error correction	Use <Backspace> when digitizing to erase the last vertex. Use <Esc> to remove the current edge from an edge or area.
Edit feature attributes	Convert features (change acronym) and/or change feature attributes values.
End Edit Mode	To escape from spatial edit mode, click the Edit Feature tool to put it in the inactive state. The feature remains selected and its information is displayed in the Selection and Attribute windows.
Remove last Vertex	Erase the last vertex digitized by selecting Line Digitizer > Remove Last from the pop-up menu in the Display window.
End line	End a line by selecting Line Digitizer > End Line from the pop-up menu in the Display window.
Undo	Use Undo to remove the last feature added or revert the last feature modification.

Keyboard Digitizing Shortcuts

When you are digitizing or editing objects, you can use the following keyboard shortcuts.

Key	Action	Mode
Backspace	Delete the most recent vertex when digitizing lines.	D
Ctrl+Click	Add a vertex or an edge if the cursor is over the edge segment.	E
Ctrl+Z	Remove the last feature added or revert the last feature modification using undo.	E
End	Terminate digitizing of a line or area feature.	D

Key	Action	Mode
Enter	Process a change to a line (for example, when re-digitizing a line segment).	E
Esc	Cancel digitizing in progress and discard the object. Pressing this key does not exit digitizing mode. Terminate measure distance and azimuth.	D, E
Delete	Delete a vertex	D, E
C	Close an edge.	D, E
I	Snap/Intersect while digitizing a line/edge.	D, E
F	Filter selected vertices.	E
G	Grab an existing edge to digitize using the existing vertices.	D
M	Merge an edge at a selected vertex.	E
O	Smooth an edge.	E
P	Spatial split at a selected vertex.	E
R	Redigitize selected vertices.	E
S	Snap to the nearest node if manual snapping is enabled. Press the <SPACEBAR> to cycle through multiple points.	D, E
Home	Close a line or area feature.	D
Shift + Click	Split an edge by adding a node.	E
Spacebar	Toggle between snap vertices.	E
~	Invert the selection of vertices.	E

* D=Digitize, E=Edit

Keyboard Shortcuts for Vertex Selection

The following table lists keyboard shortcuts that can be used to select existing vertices.

Key(s)	Action
<End>	Select the last vertex in a line.
<Home>.	Select the first vertex in a line
<Shift + Home>	Select all vertices from the selected vertex to the start of the line.
<Shift + End>	Select all vertices from the selected vertex to the last vertex.
<Right arrow key>	Move the selection forward one vertex.
<Left arrow key>	Move the selection back one vertex.
<Up arrow key>	Same as the right arrow key.
<Down arrow key>	Same as left arrow key.
<Shift + right arrow>	Move the selection forward one vertex adding to the selection.
<Shift + left arrow>	Move the selection back one vertex adding to the selection.
<Shift + up arrow>	Same as <Shift + right arrow>
<Shift + down arrow>	Same as <Shift + left arrow>

Digitizing Cursors

Once you select the digitizing tool, the cursor changes to one of the following to indicate you are in “digitize mode”.

Cursor	Description
	Selection/query. This is the default mode.
	Digitize point-to-point line
	Digitize squared line
	Digitize stream line, fitted curve
	Digitize point
	Digitize sounding
	Digitize rectangle
	Digitize arc
	Digitize circle
	Grab line
	Click within edges

5

Options

HIPS and SIPS options control the layout and appearance of the HIPS and SIPS interface, as well as the access to data repositories and files on a network.

In this chapter...

Introduction

The Options dialog box has a number of parameters to help you control the appearance of the interface, the behaviour of some commands, and the system environment.

Menu	Tools > Options

1. Select the Options command.

The Options dialog box is displayed.

The dialog box contains the following tabs:

"GENERAL" ON PAGE 173

"DISPLAY UNITS" ON PAGE 176

"EDITING" ON PAGE 179

"ENVIRONMENT" ON PAGE 181

"S-52 DISPLAY" ON PAGE 182

"DISPLAY" ON PAGE 185

"CONNECTION" ON PAGE 211

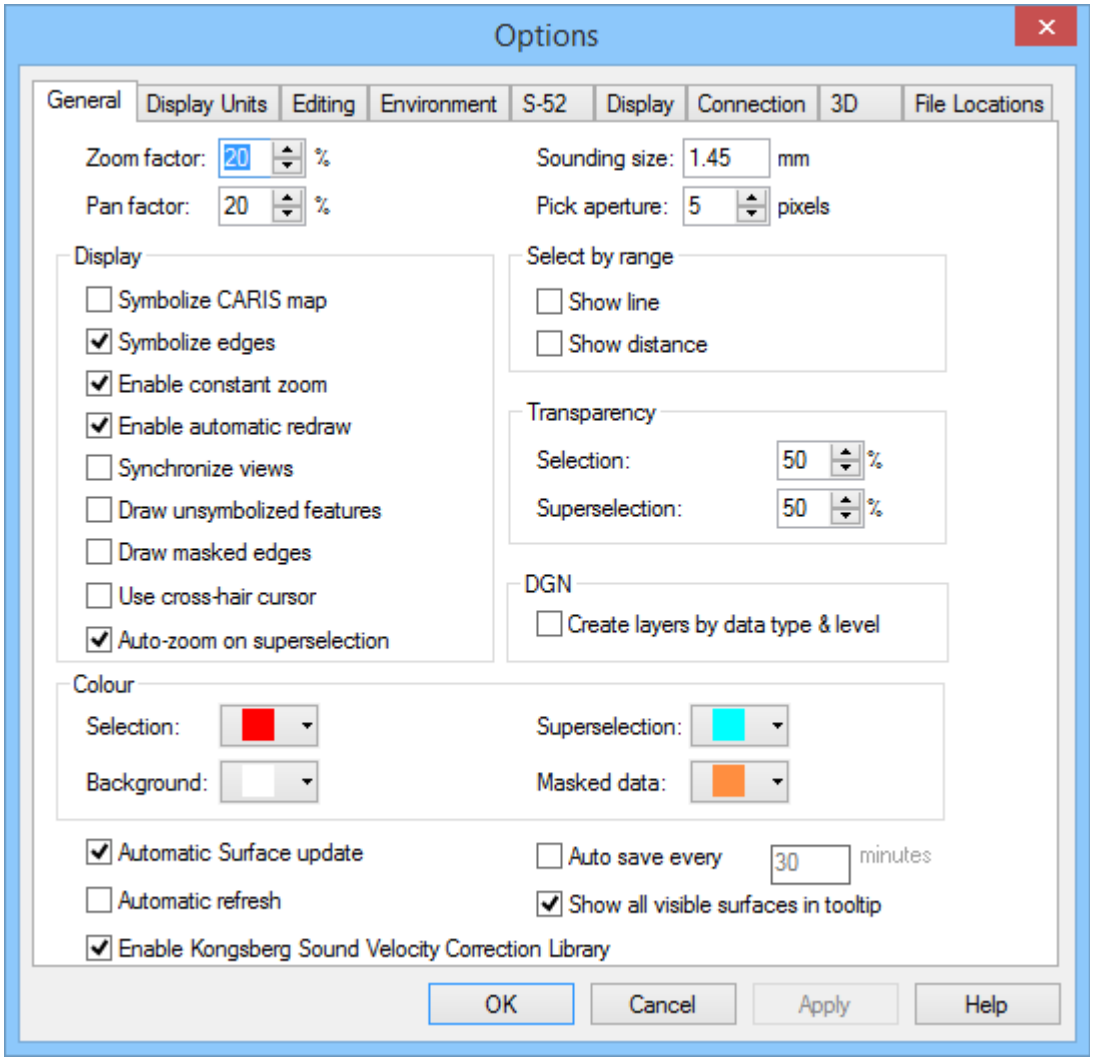
"3D" ON PAGE 213

"FILE LOCATIONS" ON PAGE 215

2. Set one or more options.
3. Click **Apply** to implement the changes without closing the dialog box.
4. Click **OK** to implement the changes and close the dialog box.

General

On the Options - General tab you can set parameters that determine how you want to work with data.



1. Set options as described in the table below.

Field	Description	To assign a value...
Zoom factor	Set the percentage used to increase or decrease the display scale when using the Zoom functions.	Select a percentage rate for the zoom tool using the up or down arrows in the *Zoom factor* box.
Pan factor	Set the percentage used to move up, down, left, or right while using the Pan tools.	Select a percentage by clicking the up or down arrow keys beside the *Pan factor* box.
Sounding size	Display sounding digits at a selected size (millimetres at map scale).	Enter a sounding size value in the *Sounding Size* field.

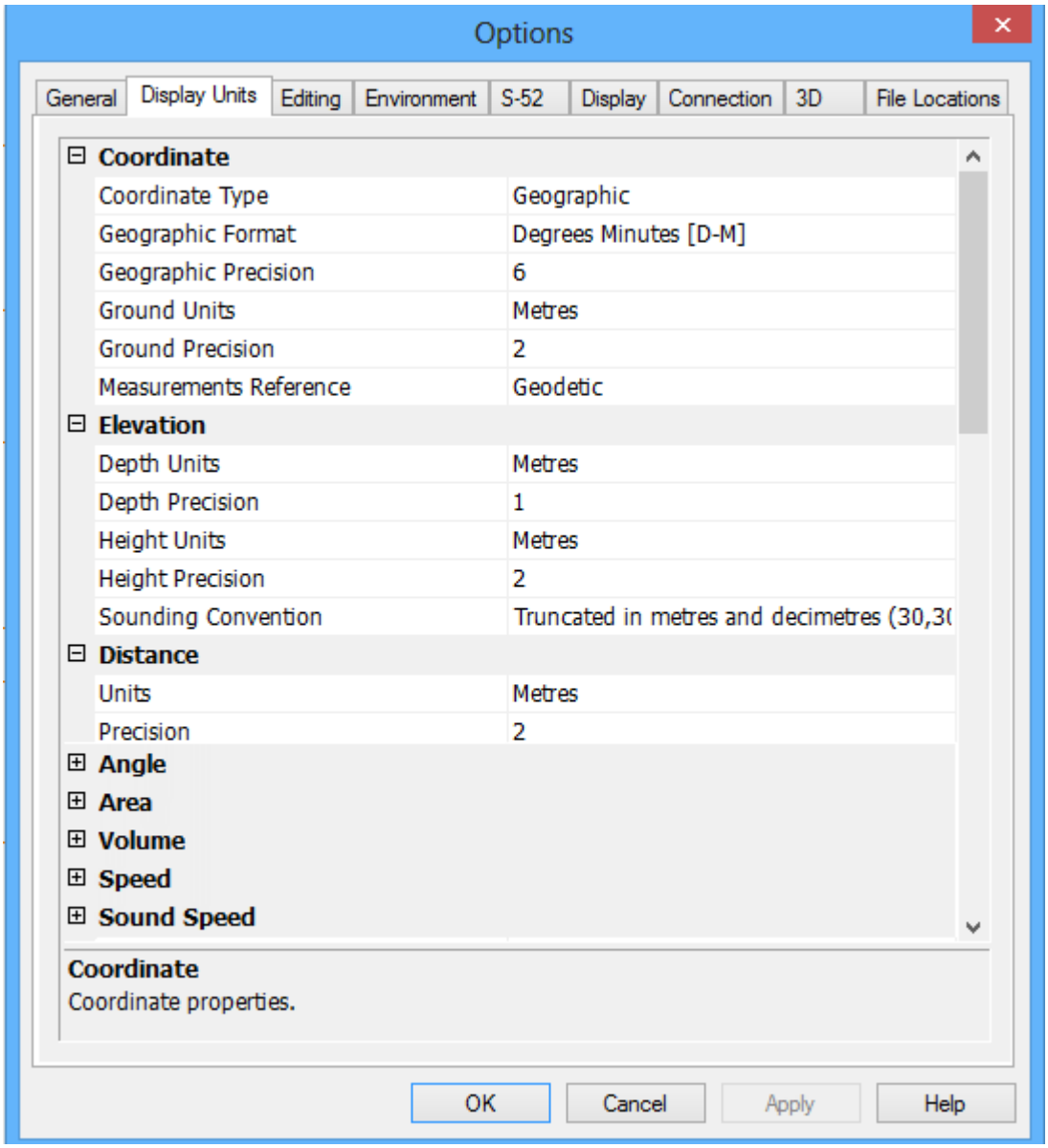
Field	Description	To assign a value...
Pick aperture	The Pick Aperture determines the maximum distance (in pixels) that a cursor can be from an object to select it. For example, if the Pick Aperture is set to five, the cursor must be a distance of five pixels (or less) to select an object.	Select the pixel area for the Pick aperture by clicking the up and down arrow buttons.
Display		
Symbolize CARIS map	When enabled, line weights and symbology map are applied based on CARIS master and symbol files. When turned off, the lines are drawn as hair lines with no symbology.	Check the *Symbolize CARIS map* box to implement this option for a background CARIS map.
Symbolize edges	Display edges using the presentation library rules (for example, S-52).	Select the check box to apply this option. Enabled by default.
Enable constant Zoom	Set the Zoom function to allow repeated Zoom commands until the function is turned off. (Default is to switch to Selection function after one Zoom.)	Select the *Enable constant Zoom* check box to implement this option. Enabled by default.
Enable automatic redraw	Refresh the Display window automatically when changes are made to the data.	Select the check box to apply this option. Enabled by default.
Synchronize views	Apply the zoom and pan operations across all windows open in the display.	Select the *Synchronize views* check box to apply this option.
Draw un-symbolized views	Display features that are normally masked or without symbology.	Select the *Draw unsymbolized views* check box to apply this option.
Draw masked edges	Use the masked data colour to display edges that have been set as masked.	Select the *Draw masked edges* check box to apply this option.
Use cross-hair cursor	Expand the cursor cross hairs to edges of the Display window	Select the *Use cross-hair cursor* check box to apply this option.
Auto-zoom on superselection	Zoom in and centre on a feature when the feature is double-clicked. If disabled, the view will centre on the feature, but not zoom in.	Select the check box to apply this option. Enabled by default.
Select by range		
Show line	Display a diagonal line inside the bounding box that is drawn when selecting data.	Enable the *Show line* check box.
Show distance	Display the distance that the bounding box covers when it is drawn on the screen during selection.	Enable the *Show distance* check box.

Field	Description	To assign a value...
Transparency		
Selection, Superselection	Control the transparency levels of fills that indicated selection or superselection in the Display window.	Select a percentage value for transparency (0 equals no transparency, 100 equals total transparency) by clicking the up or down arrow buttons.
DGN		
Create layers by data type and level	Display DGN files by data type and levels..	Enable the check box to apply this option.
Colour		
Selection Background Superselection Masked data	Select a colour for the named display component.	Select a colour from the colour palette or create a custom colour.
Other		
Automatic Surface update	Automatically update surfaces in Subset Editor changed due to cleaning sounding data or after cleaning CUBE data.	Enable the check box to activate the *Automatic surface update*.
Automatic refresh	Refresh the Display window automatically when layers are turned on or off.	Make sure the *Automatic refresh* box is checked to implement this option.
Auto save every	Automatically save all status flag changes at set intervals.	Select the *Auto Save* check box to implement this function. Enter the interval (in minutes) that work is to be saved.
Show all visible surfaces in tooltip	Control which surface information is displayed in a tooltip. When enabled (the default setting) the data value under the cursor position is displayed in the tooltip. When not enabled, the values for the *selected* layer or surface are displayed. If the parent layer is selected, values for all child layers are displayed. See "LAYERS WINDOW" ON PAGE 27 for tooltip function.	Accept default or de-select the check box.
Enable Kongsberg Sound Velocity Correction library	Set which algorithm is used in sound velocity correction. (This option is only visible if the user has obtained a license for the Kongsberg Sound Velocity Correction library.)	ON: The Kongsberg library is used to perform sound velocity correction of compatible data. OFF: (Default) The CARIS sound velocity correction algorithm is used.

2. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Display Units

Display Units options control the units displayed for any field in HIPS and SIPS that uses units.



1. Click in each field to select a value from the drop-down list.

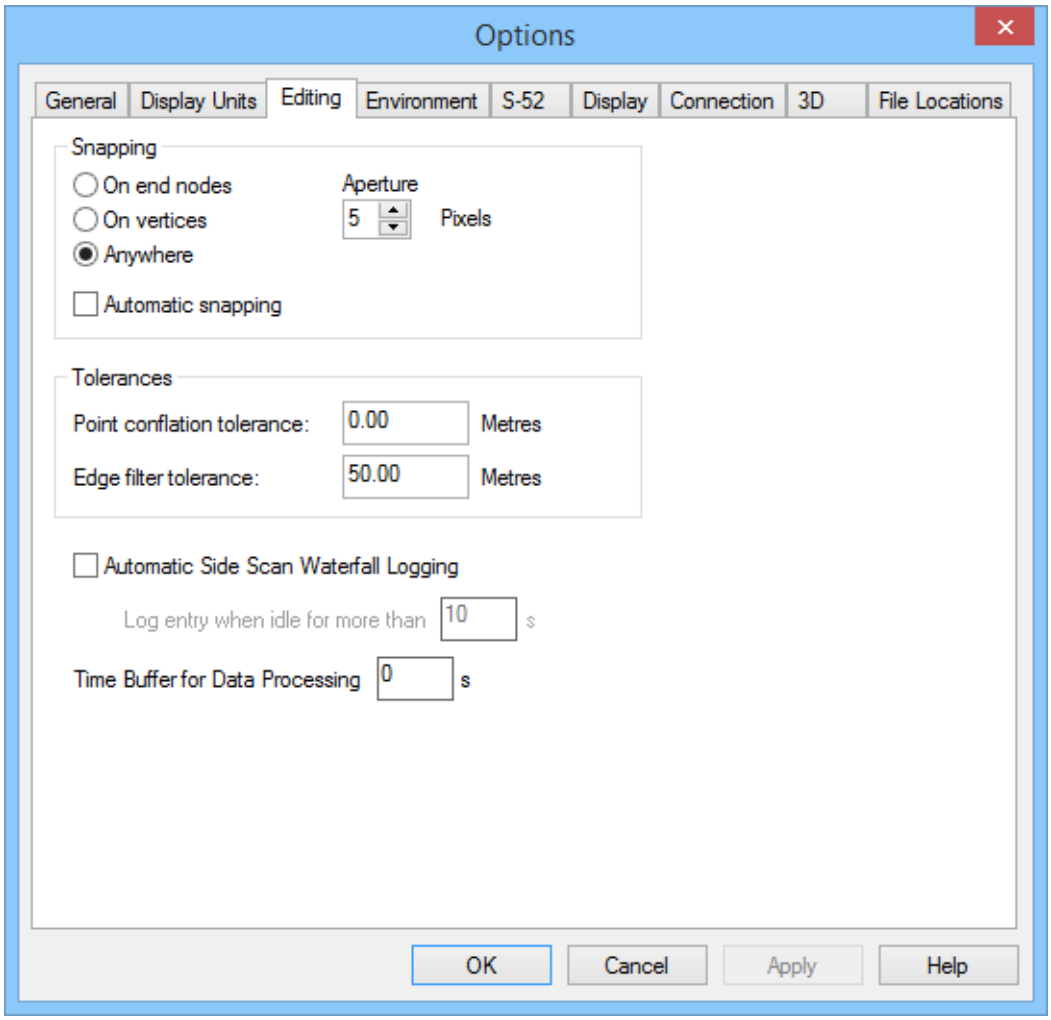
Option	Description
Coordinate	
Coordinate Type	Type of coordinates used to display positions. Geographic coordinates are latitudes and longitudes. Ground coordinates are northings and eastings.

Option	Description
Geographic Format	Format for latitudes and longitudes. This is only used when displaying geographic coordinates.
Geographic Precision	Number of decimal places for geographic coordinates.
Ground Units	Units for northings and eastings. This is only used when displaying ground coordinates. The following ground unit types are available: • Metres • Kilometres • Centimetres • Millimetres • Int. Nautical Miles* • Imperial Feet • Imperial Yards • Int. Miles* • Inches • US Survey Feet • US Survey Yards • US Survey Miles • US Chains • Cables * Int. - abbreviation for “International”
Ground Precision	Number of decimal places for ground coordinates.
Measurements Reference	The coordinate system that the units refer to. • Projected measurements are in reference to a mapped model of the earth. • Geodetic relates measurements to the defined vertical datum.
Elevation	
Depth Units	Units for soundings or depths values.
Depth Precision	Number of decimal places for depth values.
Height Units	Units for height values.
Height Precision	Number of decimal places for height values.
Sounding Convention	Convention for displaying soundings. These are read from the file called soundingrounding.xml. See “Sounding Rounding” on page 169 of the Tools guide for a description of the available rules.
Distance	
Units	Units for distance values
Precision	The number of decimal places for distance values.
Angle	
Reference	The convention for the location of the reference and which orientation is positive. • North Azimuth: North is 0. Increases clockwise. • South Azimuth: South is 0. Increases clockwise. • Bearing: Measured from north or south as indicated. For example, N20E indicates 20 degrees East of North. • Mathematical: East is 0. Increases counter-clockwise.
Format	The format for the display of angles.
Precision	The number of decimal places for angle values.

Option	Description
Area	
Units	Units for area values. Default is square metres.
Precision	The number of decimal places for area values.
Volume	
Units	Units for volume values.
Precision	The number of decimal places for volume values.
Speed	
Units	Units for speed values.
Precision	The number of decimal places for speed values.
Sound Speed	
Units	Units for sound speed values.
Precision	The number of decimal places for sound speed values.

Editing

Controls the editing of features when in Edit mode.



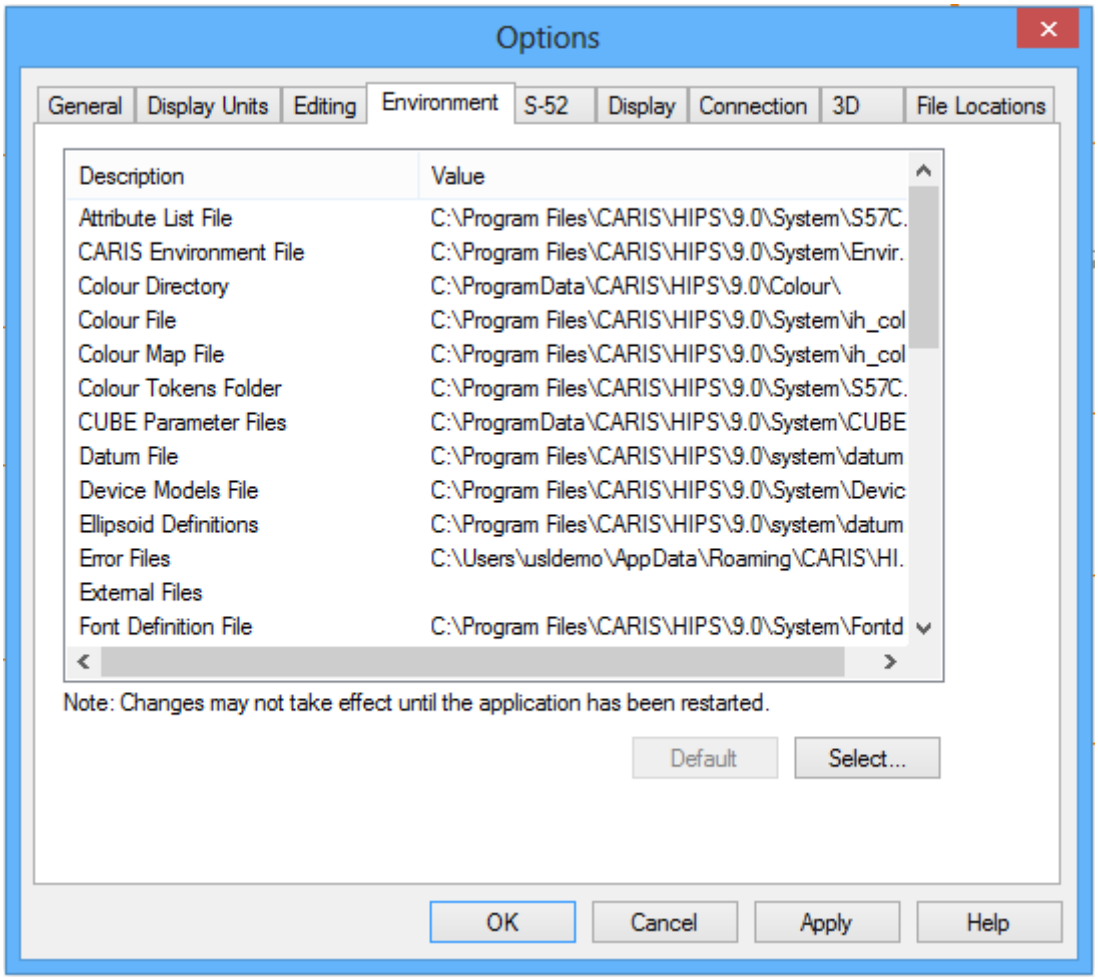
1. Set options as described in the table below.

Option	Description
Snapping	
	Select one: • *On end nodes:* Snap to the end point of an existing line. • *On vertices:* Snap a line end to the closest vertex. • *Anywhere*: Snap the starting point of a new line to the closest point of an existing line, regardless of whether the point is a vertex.
Aperture	The size of the snap aperture in pixels. Features being added or edited can be snapped to other features if the cursor is within this distance. Type the number of pixels in the box or use the arrows to set the value.

Option	Description
Automatic snapping	**On**: Snapping is automatic when digitizing. The selected snapping option is applied. **Off**: Snapping must be applied manually, by pressing and hold <**S**> while clicking to snap.
Tolerances	
Point conflation tolerance	The maximum distance for applying conflation between selected and surrounding points. Type a value in metres.
Edge filter tolerance	The maximum distance used by the Douglas-Peucker algorithm when filtering a line in Edit Feature mode. Type a value in metres.
Side Scan Logging	
Automatic side scan waterfall logging	**On**: A log file is automatically created for each line viewed. The log tracks when the session begins and ends, and logs if an area of data is viewed longer than a specified time. **Off**: No session log is created.
Log entry when idle for more than	The number of seconds that an area of data is viewed before the time is logged.
Time buffer for data processing	Time buffer is the amount of overlap in seconds to apply to beginning and end of each line to avoid data loss. If default buffer size of 0 seconds is applied, data will be processed *without* using time overlap. Type a value between 1 and 120 seconds.

Environment

The Environment tab sets the locations for the files which control environment variables used by HIPS and SIPS, such as colour definitions and mapping, device definitions, licence files and feature codes.



To set or change default environment settings:

1. Double-click on an item in the Description column, or select it and click **Select**.

The Open dialog box is displayed, if you are setting the location for a file. The Browse for Folder dialog box is displayed to set a folder destination.

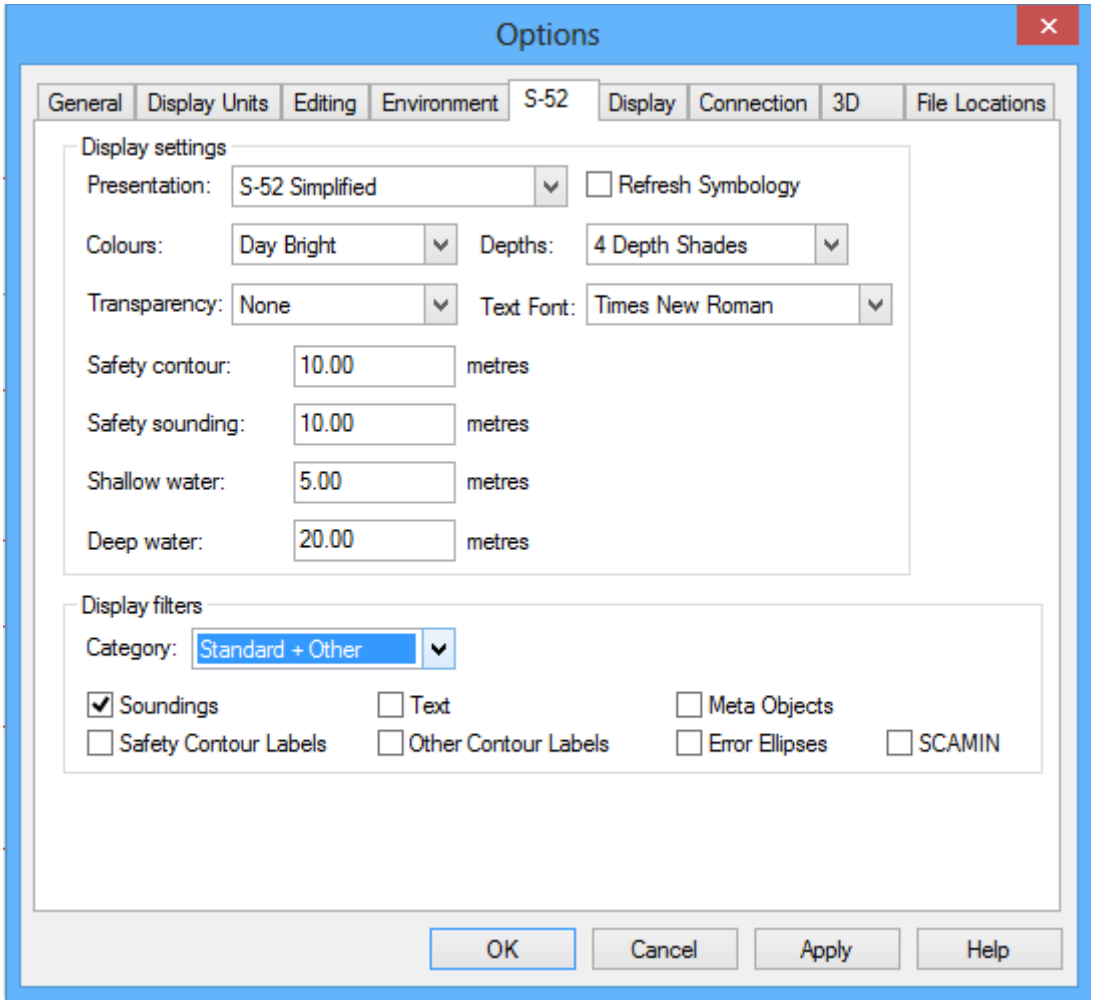
2. Navigate to the desired location, and select the file or folder.
3. Click **OK** to close the dialog box.

The new path is displayed in the *Value* column of the Options dialog box.

4. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

S-52 Display

The S-52 Display tab controls the appearance of S-57 Ed. 3.1 data shown as background data in the Display window.



Option	Description	To assign a value...
Display settings		
Presentation	Set the display mode for S-57 and HOB data to one of six different formats, including S-52, IENC, INT1 and VPF.	Select a presentation format from the drop-down list.
Refresh Symbology	**On**: Automatically refresh the display to show the symbology of the selected presentation library. **Off**: Refresh only with the Refresh command.	Select the check box to make this option active.

Option	Description	To assign a value...
Colours	Select the time-of-day colour scheme in which to display S-57 data: Day Bright, Day White, Day Dark, Dusk, Night.	Select the appropriate item from the drop-down list.
Depths	Select the number of depth shades to use: 2 or 4 (default). This value works in conjunction with *Safety Contour, Shallow Water* and *Deep Water* depths (see below).	
Transparency	Select a transparency setting percentage for area colour fills. This is useful when comparing layers. Values are None, 25, 50, 75 or 100 percent.	
Text Font	Select a font to use in the Display window. The default font is Times New Roman.	
Safety Contour	Define a sufficient depth for safe navigation. The safety contour is symbolized as a thick solid line. It also defines the boundary of shallow and deep-water areas for the purpose of area symbolization.	Type a value in the appropriate field.
Shallow Water	Define the contour boundary between the two darker colours used for water shallower than the Safety Contour.	
Safety Sounding	The safety depth is used in the symbolization of soundings. Soundings whose depth is less than the safety depth are displayed in bold text.	
Deep Water	Define the contour boundary between the two lighter colours used for water deeper than the Safety Contour.	
Display filters		
Category	The category of features to be displayed. Use Standard & Other to draw all features. except soundings and text, which must be enabled separately.	Select a category from the drop-down menu.
Soundings	**On**: Display soundings. **Off**: Do not display soundings	
Text	**On**: Display text. **Off**: Do not display text.	
Meta Objects	**On**: Display meta objects. Meta object data represents data quality, accuracy, etc. **Off**: Do not display meta objects.	
Safety Contour Labels / Other Contour Labels	**On**: Display depth labels on the safety contours. **Off**: Do not display depth labels on the safety contours.	
Error Ellipses	**On**: Display ellipses that represent error margins of all positioning equipment (GPD, navigation, etc.). **Off**: Do not display error ellipses.	

Option	Description	To assign a value...
SCAMIN	**On**: Filter features from the Display window that have SCAMIN values less than the display scale. These features are still drawn in the selection/superselection colours if they are selected or superselected. **Off**: Do not filter features based on SCAMIN values.	

1. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Display

The Display tab lists various display controls for such items as fonts, visibility and position of labels, and so forth.

Many of the colour options set in this tab indicate the status of displayed elements such as track lines (merged or not,) and critical soundings (designated, rejected, outstanding, etc.).

These colours can also be changed in the Properties for a layer, for example, for example, the colour of designated soundings can be changed in the Critical Soundings layer. However, these colour settings are not retained unless saved to a session file.

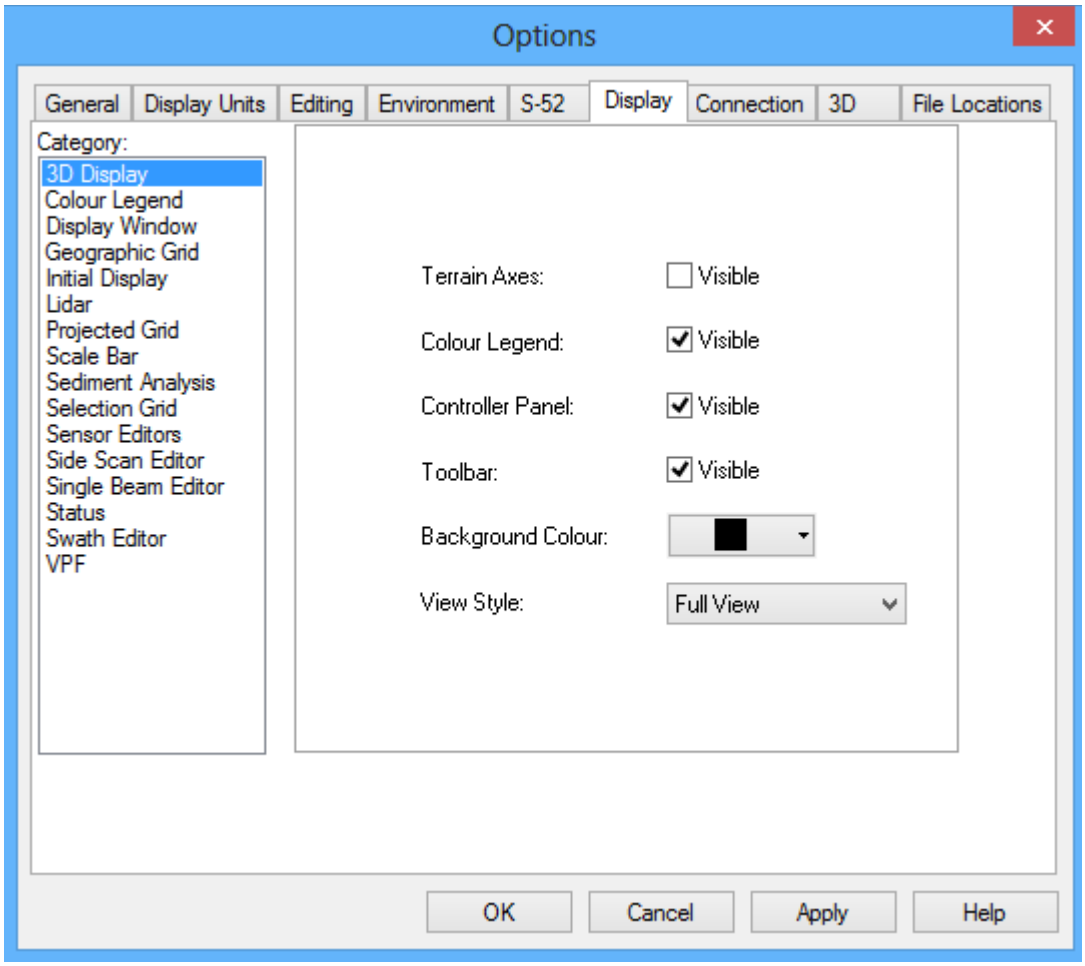
1. Select the Display tab.

The Options - Display dialog box is displayed.

2. Select an item from the Category list.

The controls for the selected editor or window are displayed on the right side of the window. These controls are described below.

3D Display



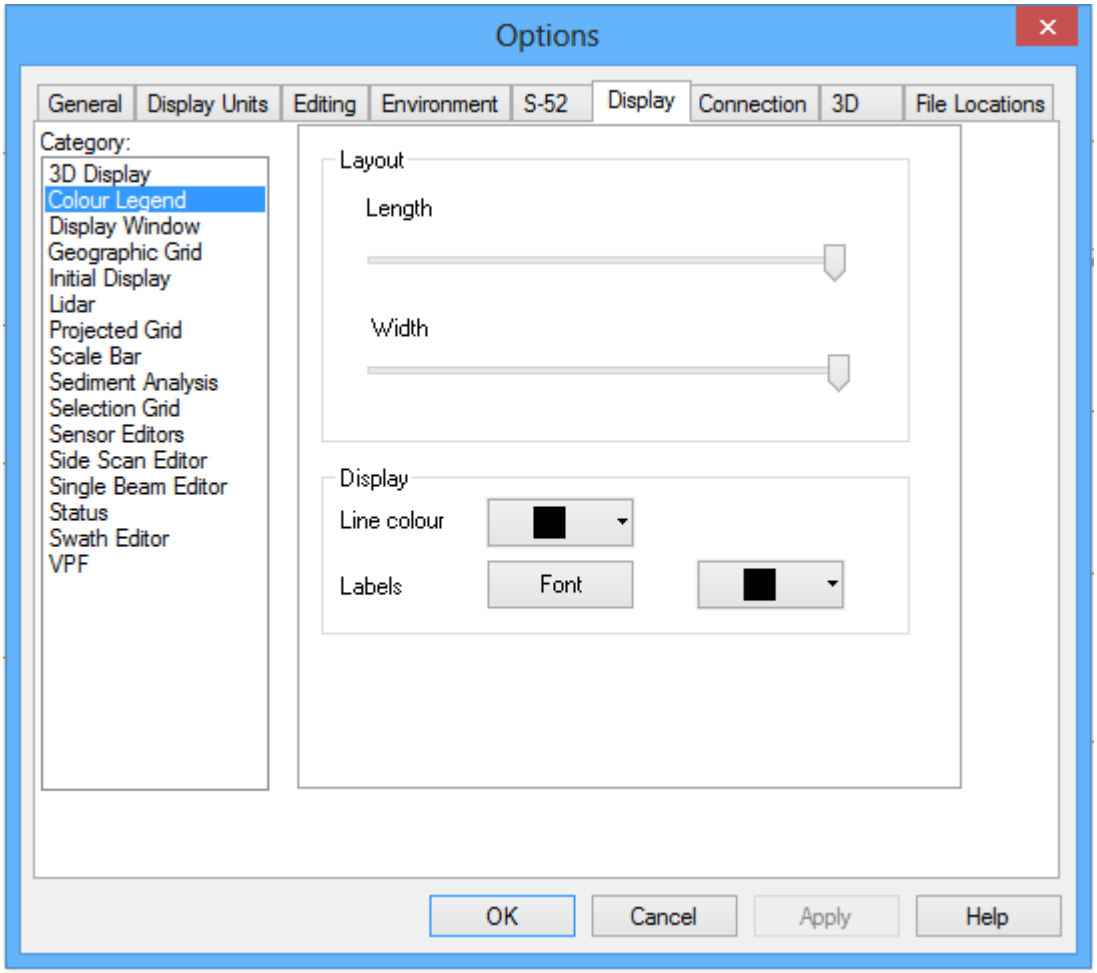
Use settings in Tools > Options > Display > 3D to:

- display *Terrain Axes* (x,y,z) grid for the selected surface.
- turn on or off the display of the *Colour Legend* which indicates depths of the coloured areas in the display.
- turn on or off the display of the *Controller Panel*. When turned off, the controller panel's on/off button within the 3D View is also removed.
- display the 3D View *Toolbar*. If this option is not enabled, the commands can be activated from the View > 3D Flight Path submenu.
- select the colour to use as the *Background Colour* of the 3D View.
- *View Style* sets the manner in which the 3D window is displayed within the Display window area.
  - In Full View the 3D View occupies the entire Display window.

- Vertical Tile (and Horizontal Tile) the 3D View is tiled vertically (or horizontally) with the Display window and any other open windows.
- Cascade arranges all the open windows within the Display window so that they overlap one another, with their title bars visible.

Colour Legend

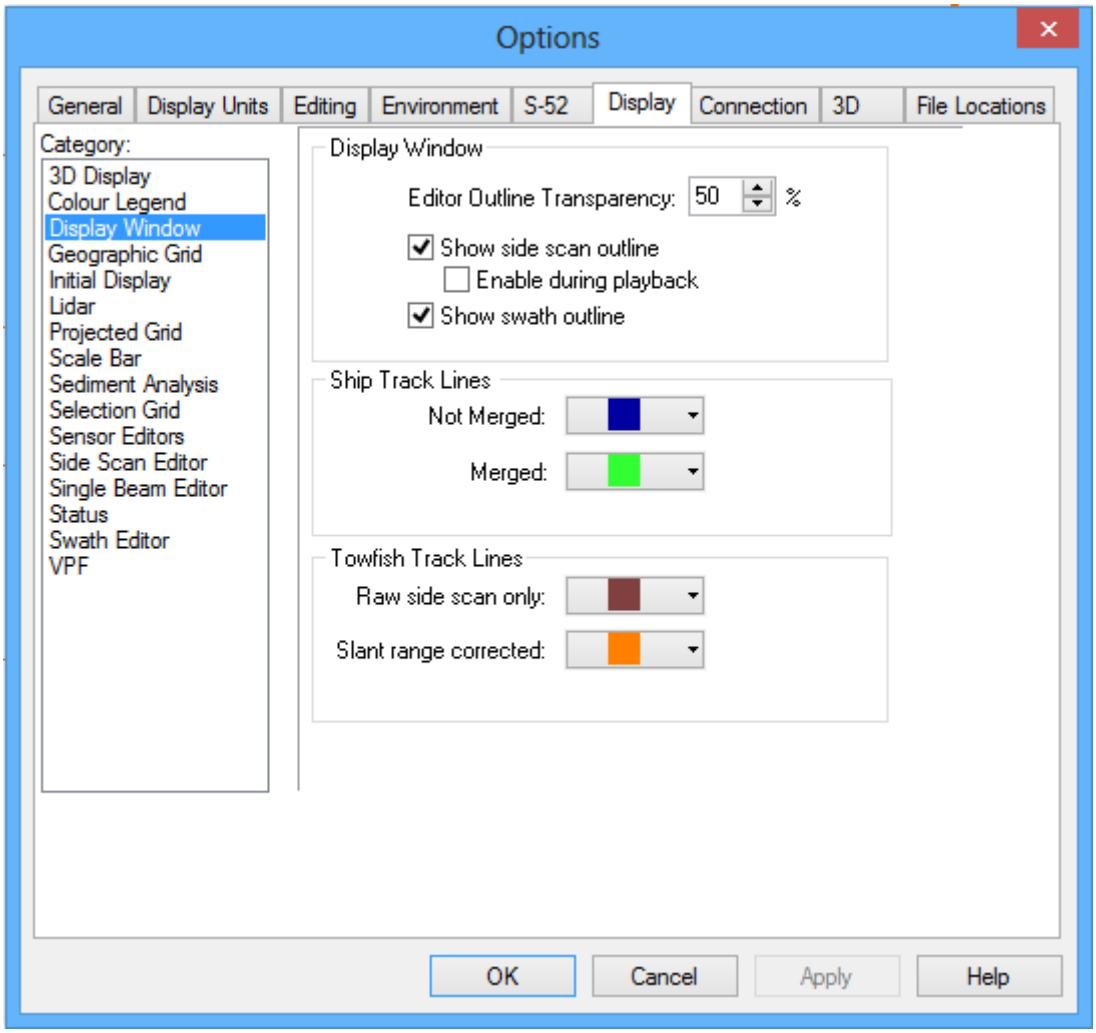
Define the appearance of the colour legend that can be displayed in the 2D view.



Option		Description	To assign a value...
Layout	Length	Define the length of the legend.	Click and drag the slider bar to the right to increase the length.
	Width	Define the width of the legend .	Click and drag the slider bar to the right to increase the width.
Display	Line colour	Select the colour of the border and tick marks on the legend.	Select a colour from the colour picker, or create a custom colour from the standard Windows colour palette.
	Labels	Select the font type, style and size for the colour legend labels.	Click **Font** to open the Font dialog box and select the type, style and size.
	Label colour	Select the colour for the colour legend labels.	Select a colour from the colour picker, or create a custom colour from the standard Windows colour palette.

Display window

Set the display and colour features for the ship and towfish track lines. The image below shows the default settings.



1. Set the parameters described in the table below.

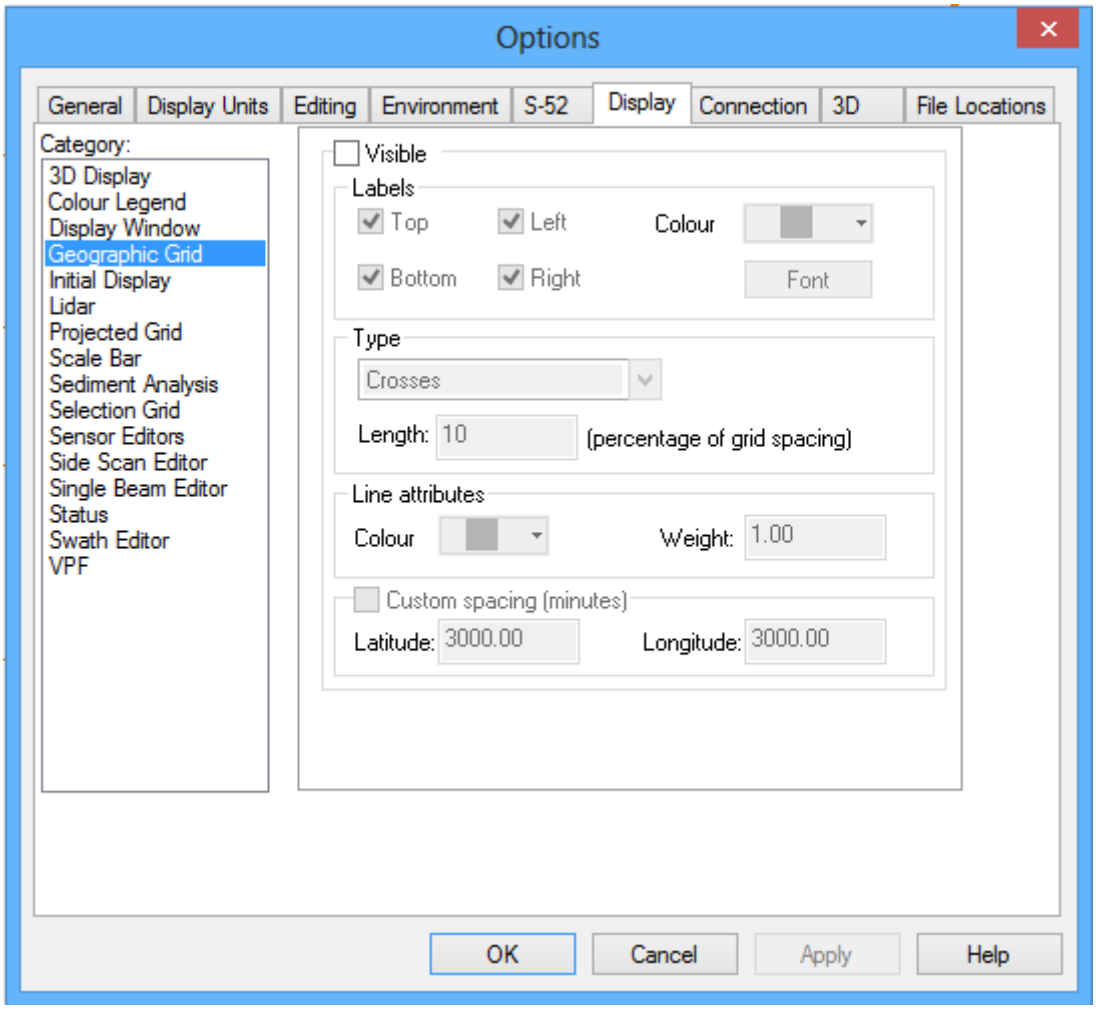
Field	Description	To assign a value...
Display Window fields:		
Editor Outline Transparency	Control the transparency of the sounding coverage outline in the Display window. Applies to Swath Editor and Side Scan Editor.	Select a percentage value for transparency (0 equals no transparency, 100 equals total transparency) by clicking the up or down arrow buttons.
Show Side Scan Outline	On: Display an outline of the sounding data currently visible in Side Scan Editor. Off: Sounding coverage outline is not displayed.	Select the *Show Side Scan Outline* check box to display the marker in the Display window while the Side Scan Editor is open.

Field	Description	To assign a value...
Enable during playback	On: Move the sounding coverage outline along the track line as playback automatically scrolls through data in waterfall view. Off: The coverage outline does not track data being viewed in playback	Select the *Enable During Playback* check box to make this option active.
Show Swath Outline	On: Display an outline of the sounding data currently visible in Swath Editor.Off: Do not display sounding coverage outline. Off: Sounding coverage outline is not displayed	Select the *Show Swath Outline* check box to display the marker in the Display window while the Swath Editor is open.
Measure Distance	Select a font to display measurements when the Measure Distance command is used.	Click **Font** and select a type face, font size and colour from the dialog box.
Ship Track Lines fields:		
Not Merged	The colour for track lines that have not been merged into a position/depth file	Select a colour from the colour palette, or create a custom colour from the standard Windows colour picker. Default colours are
Merged	The colour of track lines that have been merged into a position/depth file.	
Towfish Track Lines fields:		
Raw Side Scan Only	The colour used to display track lines when side scan data is still in raw mode.	
Slant Range Corrected	The colour used to display track lines when side scan data has been slant range corrected.	

2. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Geographic Grid

Options for the Geographic Grid control the display of a temporary grid based on geographic units, drawn over the extent of your project.



1. Select the *Visible* check box to display the grid in the Display window.
2. Set other display options as described below.

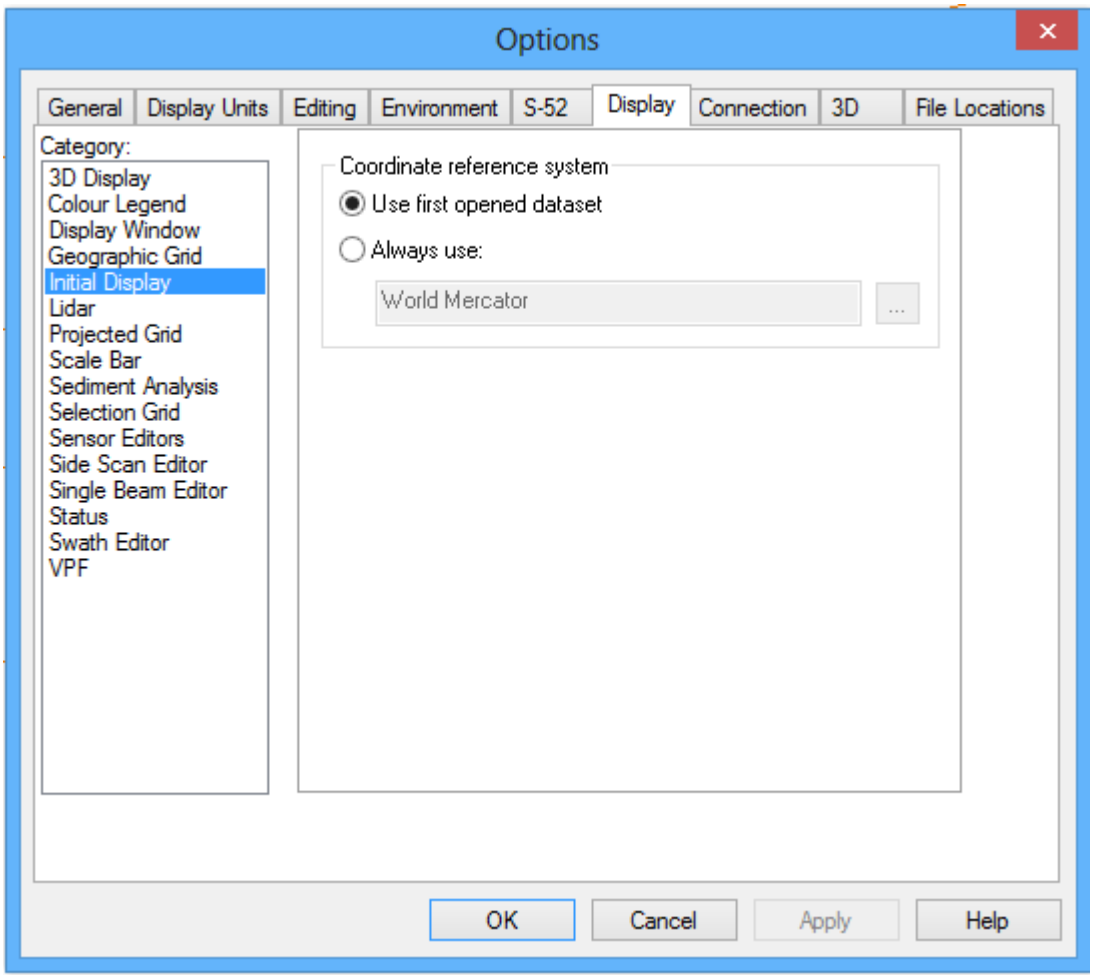
Field	Description	To assign a value...
Labels: Top/Left/ Bottom/Right	Display geographic coordinates at selected locations in the Display window	Select the check boxes.
Colour	Set a colour for the grid labels.	Select a colour from the colour picker, or create a custom colour from the standard Windows colour palette.

Field	Description	To assign a value...
Font	Typeface and point size for grid labels	Click the Font button to open the standard Windows Font dialog box to select a font type and size.
Type	The type of markings to make up the geographic grid.	Select a type from the drop-down list.
Length	The length of the markings (mm at map scale).	Type the length.
Line Attribute Colour	Colour for the geographic grid markings.	Select a colour from the colour picker, or create a custom colour from the standard Windows colour palette.
Weight	Thickness of the markings	Type a value into the field.
Custom Spacing	Set the distance between grid markings in the display.	Select the check box.
Latitude/Longitude	Spacing of grid markers in minutes.	Type a value in the provided field.

3. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Initial Display

Define the default coordinate reference system that will be used for the initial display of data each time HIPS and SIPS is opened.



1. Select either option:
   - *Use first opened dataset:* This option will cause the application to automatically use the coordinate reference system of the first dataset opened.

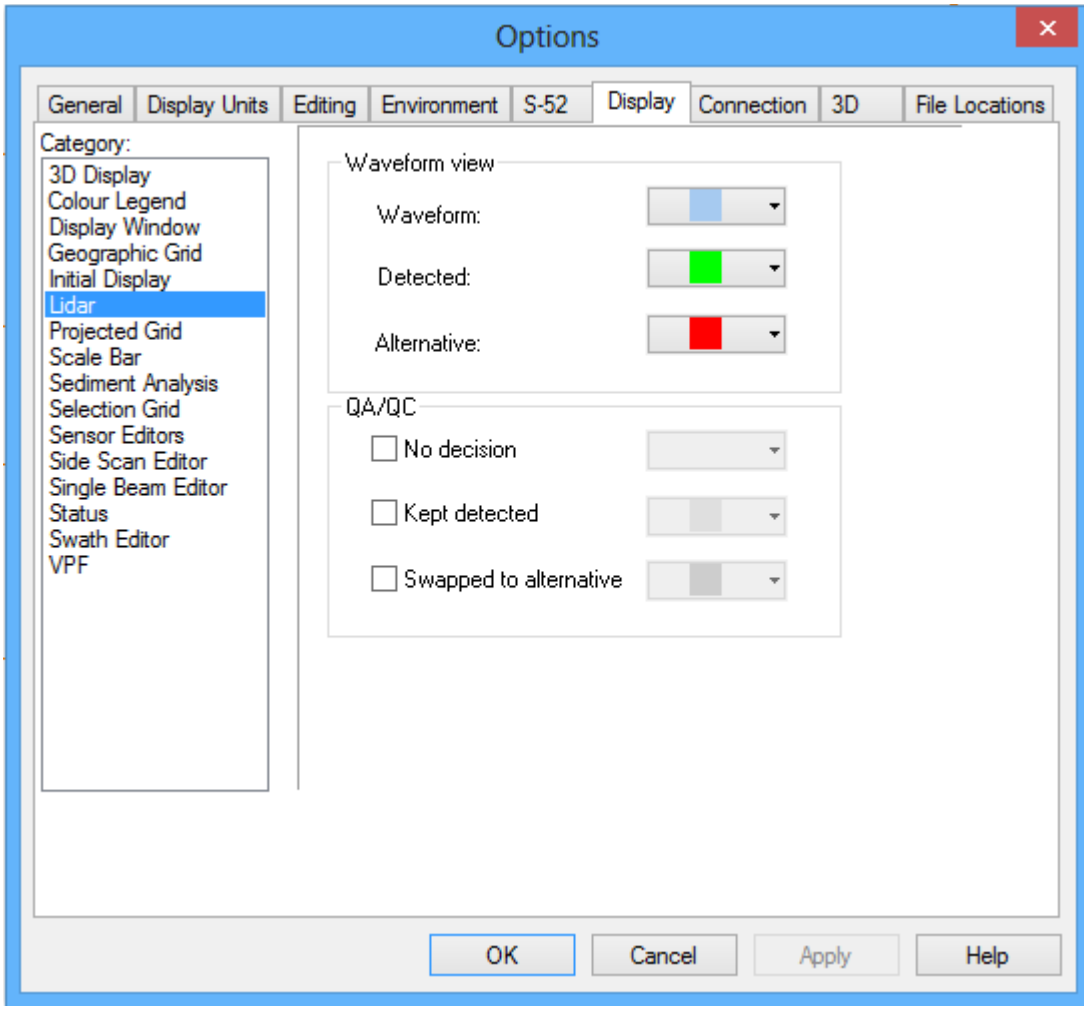
     • If the first dataset opened does not have a coordinate reference system, the user will be prompted to select one.The selected coordinate reference system then becomes the default for all other datasets opened in the session.
   - *Always use*: This option enables you to select the coordinate reference system that will be used to display all datasets, regardless of the projections of the datasets opened. Click the Browse button (...) to launch the Select Projection dialog box and select a projection.

Regardless of the option selected here, if a dataset is opened at anytime that does not have a coordinate reference system, you

will be prompted to identify the projection of the data. This provides the application with the information needed to project the data again using the coordinate reference system defined by this setting.

LIDAR

LIDAR options control the colour options for LIDAR data in the Swath and Subset Editors.



Waveform view

Select a waveform colour from the colour palette, or create a custom colour from the colour picker.

- Waveform: colour for displaying waveform data.
- Detected: colour for displaying detected soundings.
- Alternative: colour for displaying soundings with an alternative depth.

QA/QC

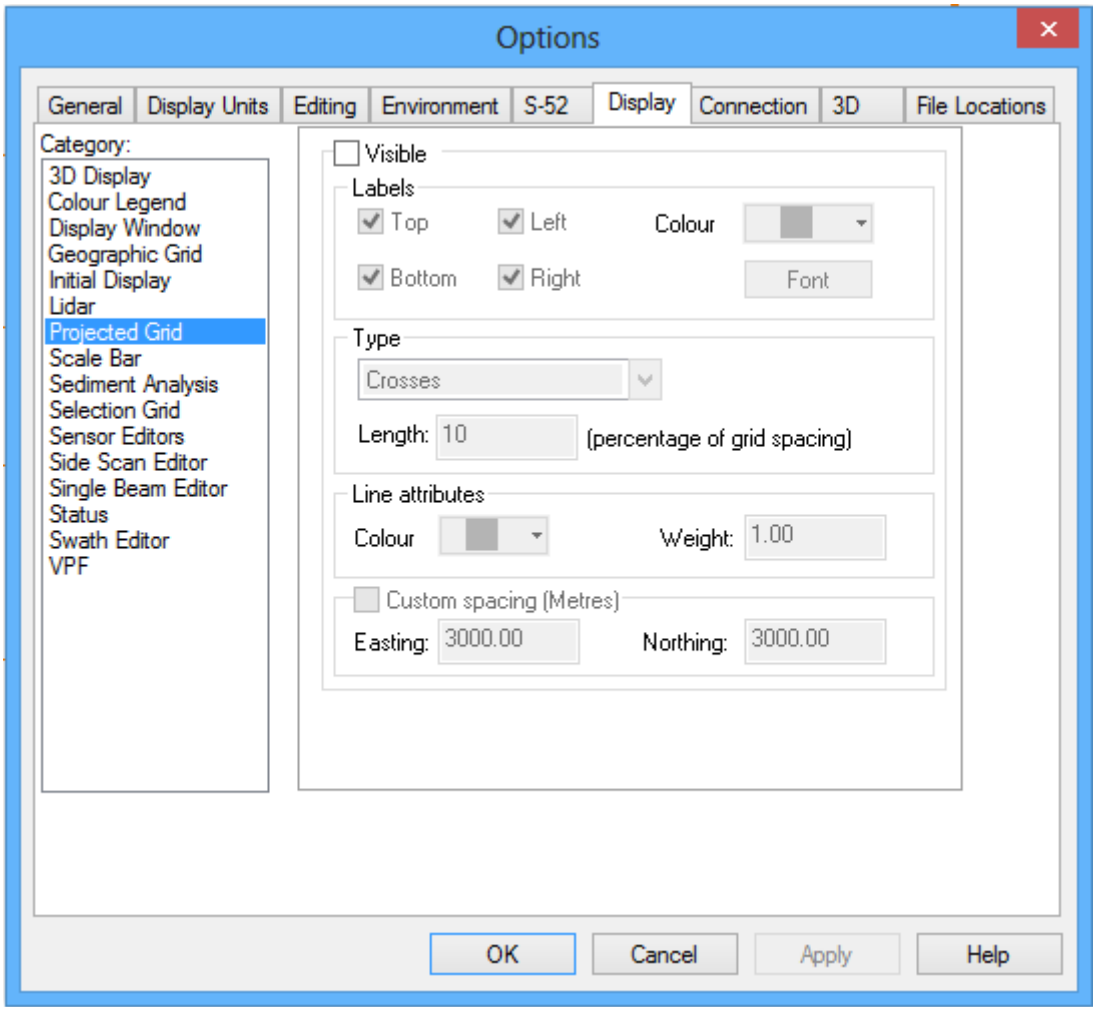
Display or hide soundings with specific quality flag by selecting or clearing any of the quality flag check boxes.

- No Decision: soundings that have a detected and alternative depth. (Quality 1 flag).
- Kept Detected: soundings where the detected depth is retained over the alternative depth (Quality 2 flag).
- Swap to Alternative: soundings where the detected depth has been replaced by the alternative depth (Quality 3 flag).

1. Select a colour for each type of quality flag.
2. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Projected Grid

Projected Grid display options control visibility, labelling and appearance of a grid based on ground coordinates, projected over your entire project.



1. Select the *Visible* check box to display the grid in the Display window, and to activate the display options.

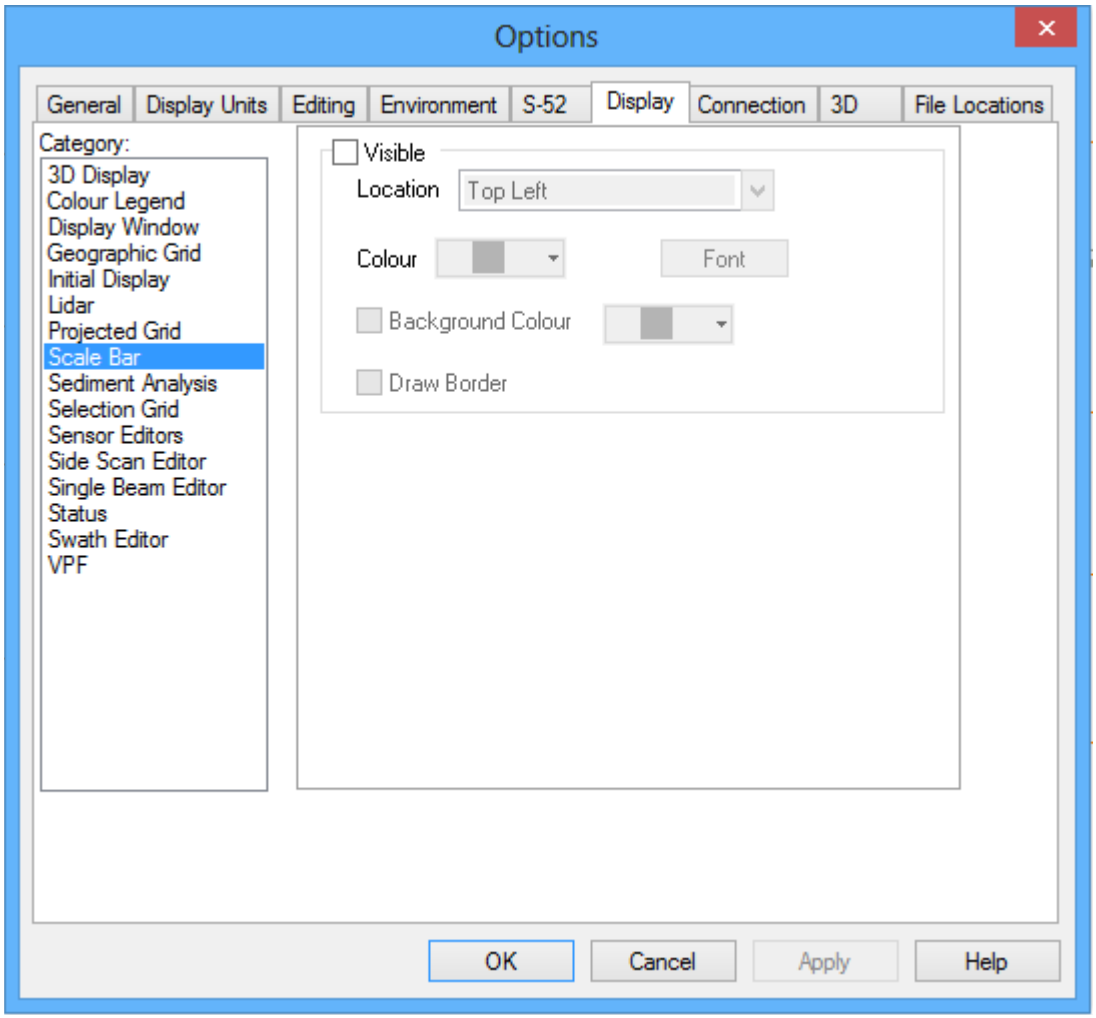
To change from default values:

2. Select any of the *Labels* check boxes to display coordinates at those selected locations in the Display window: Top/Left/Bottom/Right.
3. Click **Colour** to select a colour for the grid labels, from the colour picker, or create a custom colour.
4. Click **Font** to open the Windows Font dialog box to select a typeface and point size for grid labels.
5. From the *Type* drop-down list, select the type of markings to make up the projected grid, e.g., full lines, ticks and crosses.
6. Set the *Length* for the markings in mm at map scale.

7. Select a colour for the grid lines.
8. Type a *Weight* value for the thickness of the grid lines.
9. Set the distance between grid markings in the display, by selecting the *Custom Spacing* check box and typing a value into the *Easting* and *Northing* fields. (Spacing of markers in mm at map scale.)
10. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Scale Bar

This tab controls the options for the display of the scale bar.



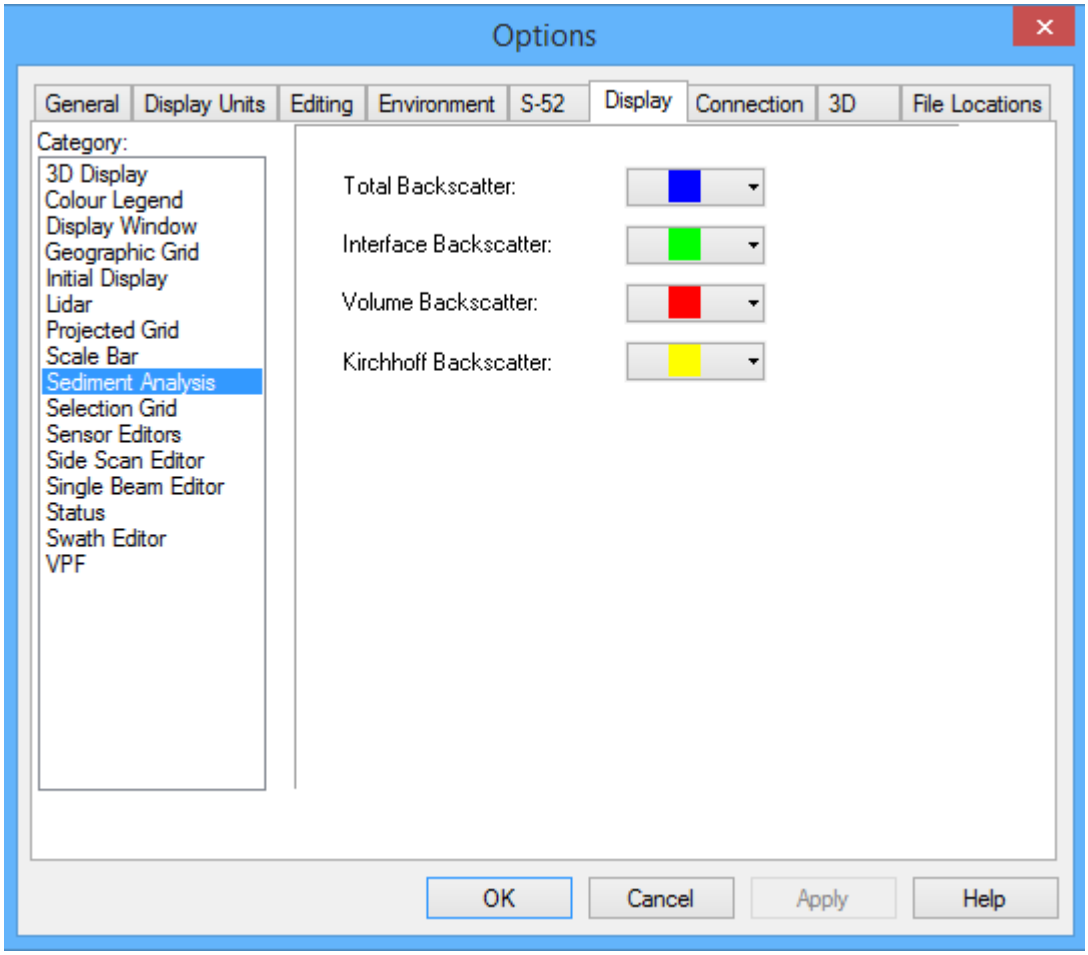
1. To show a scale bar in the Display window, select the *Visible* check box.

To change from the default settings:

2. Select the position of the scale bar in the window from the *Location* drop-down list.
3. Select a *Colour* for the scale bar from the colour picker, or create a custom colour from the Windows colour palette.
4. Click **Font** to select the font type, style and size from the Font dialog box.
5. Select the *Background Colour* check box to turn on a colour for the scale bar. Change the colour by selecting from the colour picker.
6. To draw a border line around the scale bar, select the *Draw Border* check box.

Sediment Analysis

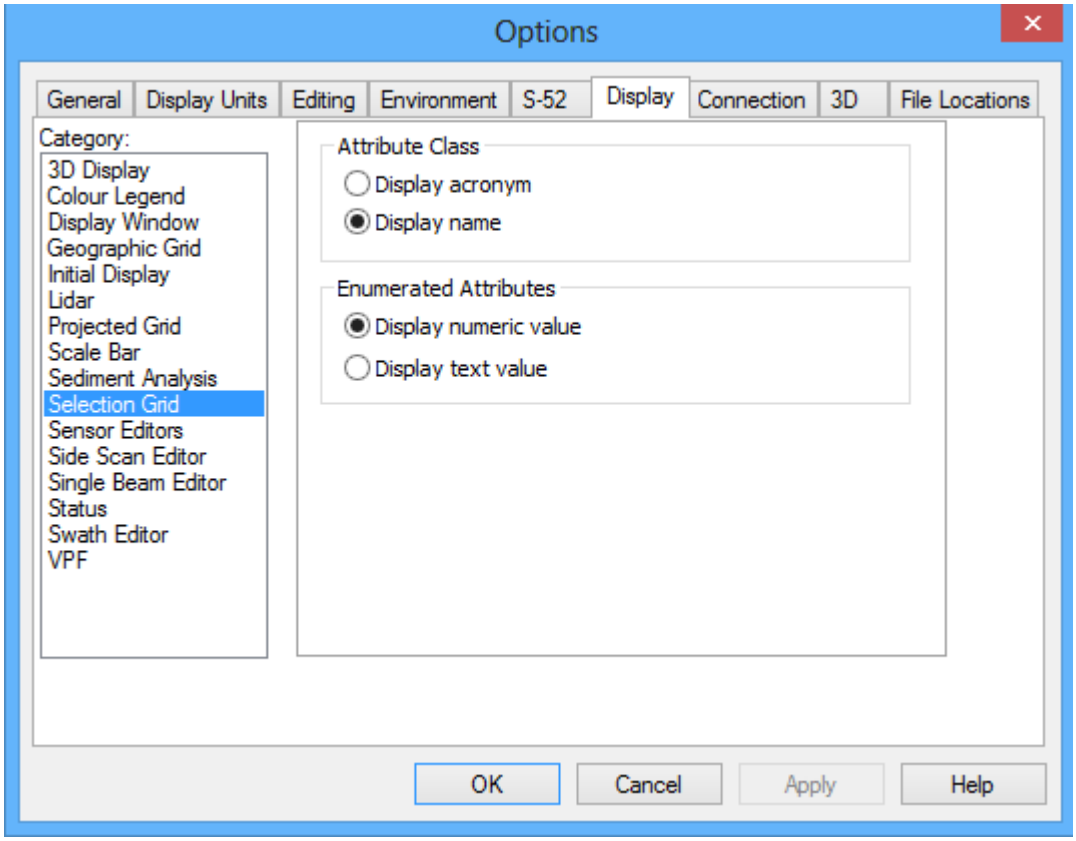
Set the colours for the display of backscatter trace in a Sediment Analysis graph.



- Total Backscatter (default colour blue) - combined backscatter from the three sources
- Interface Backscatter (default colour light green) - from the sea floor/water interface
- Volume Backscatter (default colour red) - from within the sediment
- Kirchhoff Backscatter (default colour yellow) - models the sea floor roughness.

Selection Grid

Control the display of column headings and attribute values in the Selection window and thereby in data exported to ASCII files.



1. Select an Attribute Class option. The default is Display name.
2. Select an Enumerated Attributes option. The default is Display numeric value.

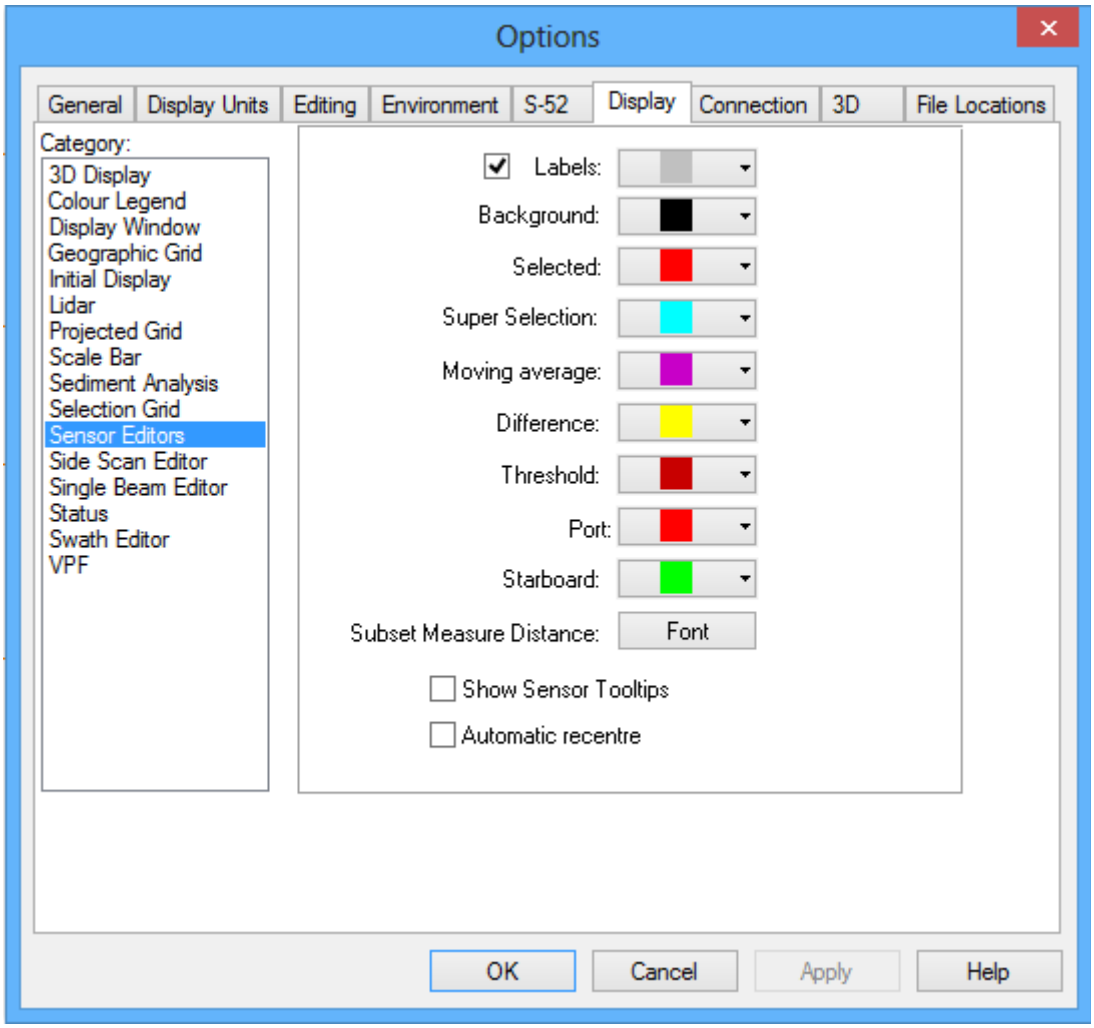
The Attribute Class option controls whether column headings are displayed as acronyms or full textual names. For example, if the CATCBL attribute is a column heading, you can either display it as “CATCBL” or as “Category of cable”.

The Enumerated Attributes option controls whether attribute values are displayed as numeric representations, or as textual representations. For example, the values available for the COLOUR attribute include a list of colours with corresponding numbers. You can choose to display/ the selected value as a number or as a name.

3. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Sensor Editors

The Sensor Editors tab controls some of the colour settings for editors such as Attitude Editor and Navigation Editor.



1. Set the parameters described in the table below.

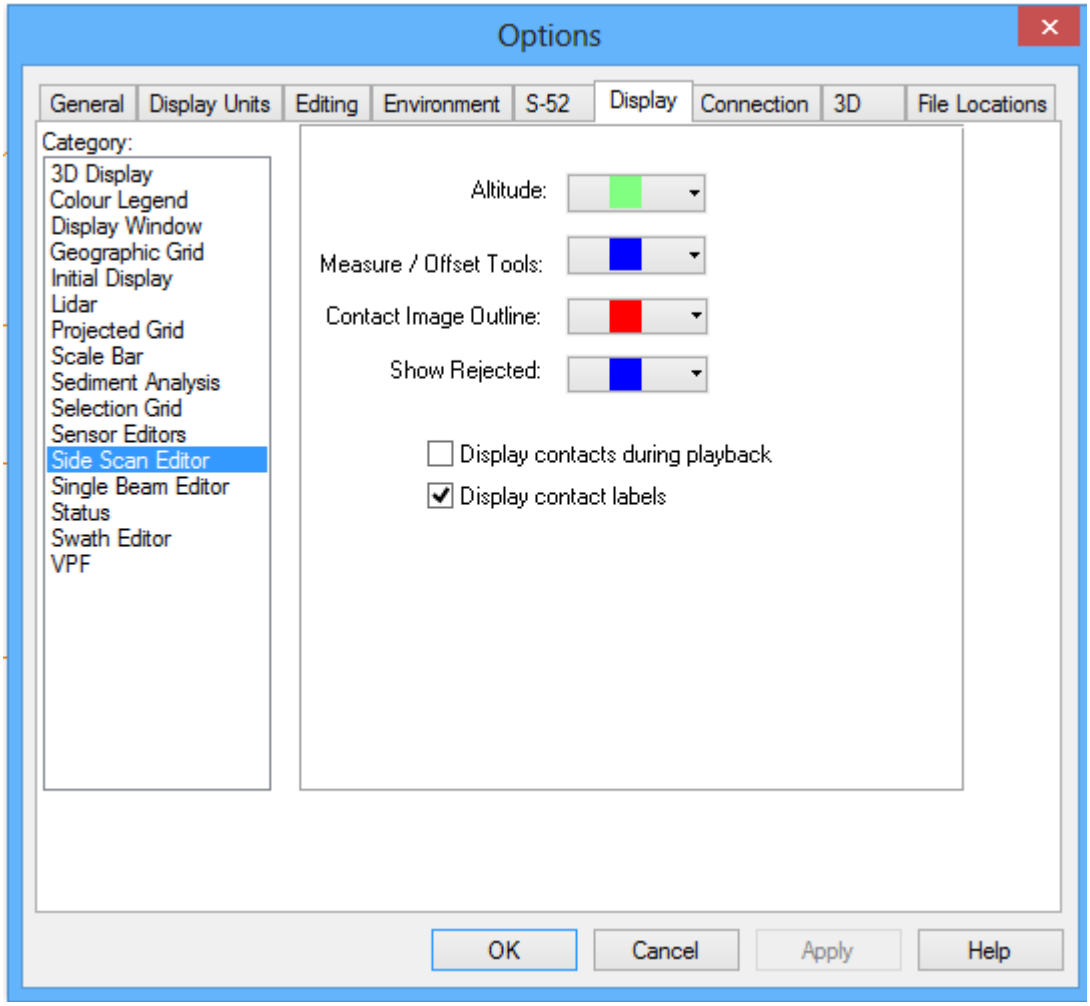
Field	Description	To assign a value...
Labels	Labels display the horizontal and vertical scale lines and annotations in the editors.	Show or hide labels (the labels are toggled to on when the Label box is checked). Select a colour for the labels from the colour palette, or create a custom colour from the standard Windows colour selector.

Field	Description	To assign a value...
Background	Use this control to set the background colour in the editors.	Select an alternate colour from the colour palette, or create a custom colour from the Windows colour selector.
Selected	The colour for data selected and queried in the editor windows. Default colour is red.	
Superselection	The colour for superselected data in the editor windows. Default colour for superselection is cyan blue.	
Moving Average	Sets the colour for the Moving Average line in the Attitude and Single Beam Editors.	Select a colour from the colour palette, or create a custom colour from the standard Windows colour selector.
Difference	Sets the colour for Difference line in the Attitude and Single Beam Editors.	
Threshold	Sets the colour for the Threshold line in the Attitude and Single Beam Editors.	
Port	Sets the colour for the port side of the side scan coverage outline in the Side Scan display, or the port soundings in the Plan View of Swath Editor.	Select a colour from the colour palette, or create a custom colour from the standard Windows colour selector.
Starboard	Sets the colour for the starboard side of the side scan coverage outline in the Side Scan display, or the starboard soundings in the Plan View of Swath Editor.	
Subset Measure Distance	Sets the font to be used when displaying distance measure in Subset Editor.	Click **Font** and select a name, style and size for the displayed font.
Show Sensor Tooltips	Tooltips are used in the Attitude and Navigation Editors for displaying data values as the cursor moves over the graphs.	Show or hide tools tips by clicking the *Tool Tips* check box (checked means this feature is visible).
Automatic Recentre	Redraw the Display window so that the sounding coverage outline is always displayed in the centre when scrolling along the line.	Select the *Automatic recentre* check box.

2. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Side Scan Editor

This tab controls the colours used for the Side Scan Editor display. (The *Port* and *Starboard* colour palettes are set in Tools > Options > Display > "SENSOR EDITORS" ON PAGE 201.)

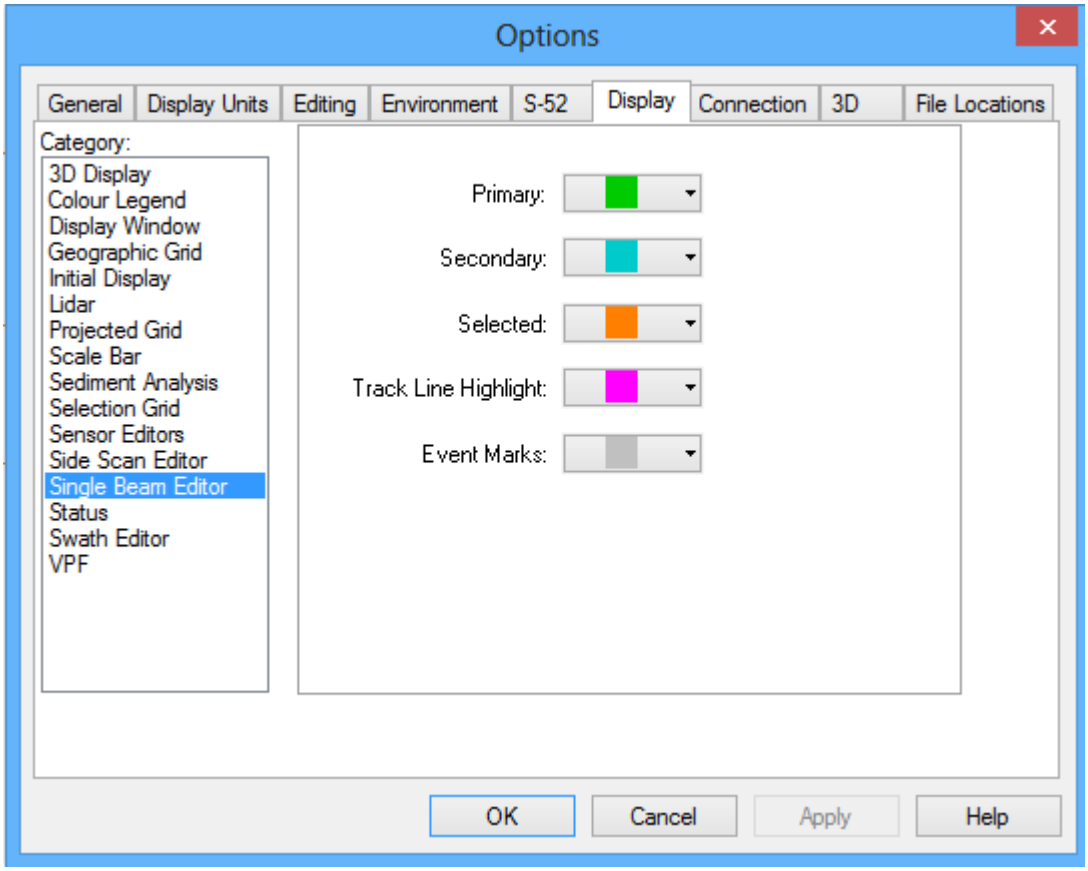


1. Select a colour from the appropriate colour palette, or create a custom colour from the standard Windows colour picker:
   - *Altitude*: Sets the colour of the digitized line along the seabed that is used determine the height of the towfish from nadir when slant-range correction is performed.
   - *Measure /Offset Tools*: Sets the colour for displaying the values when using the Measure Distance and Measure Shadow tools, and the colour of the range display.
   - *Contact Image Outline*: Sets the colour for displaying the outline of a contact.
   - *Show Rejected:* Sets the colour of rejected pings (which are visible when the Show Rejected button on the Toolbar is activated)
2. Select the *Display contacts during play back* check box to make contacts visible in waterfall window during scrolling or playback.

3. Select the *Display contact labels* check box to display the labels for the contacts in the waterfall view. Contacts are displayed in the colour set for the contact image outline in Tools > Options > Display > Side Scan Editor.
4. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Single Beam Editor

This tab controls the colours for the Single Beam Editor display.



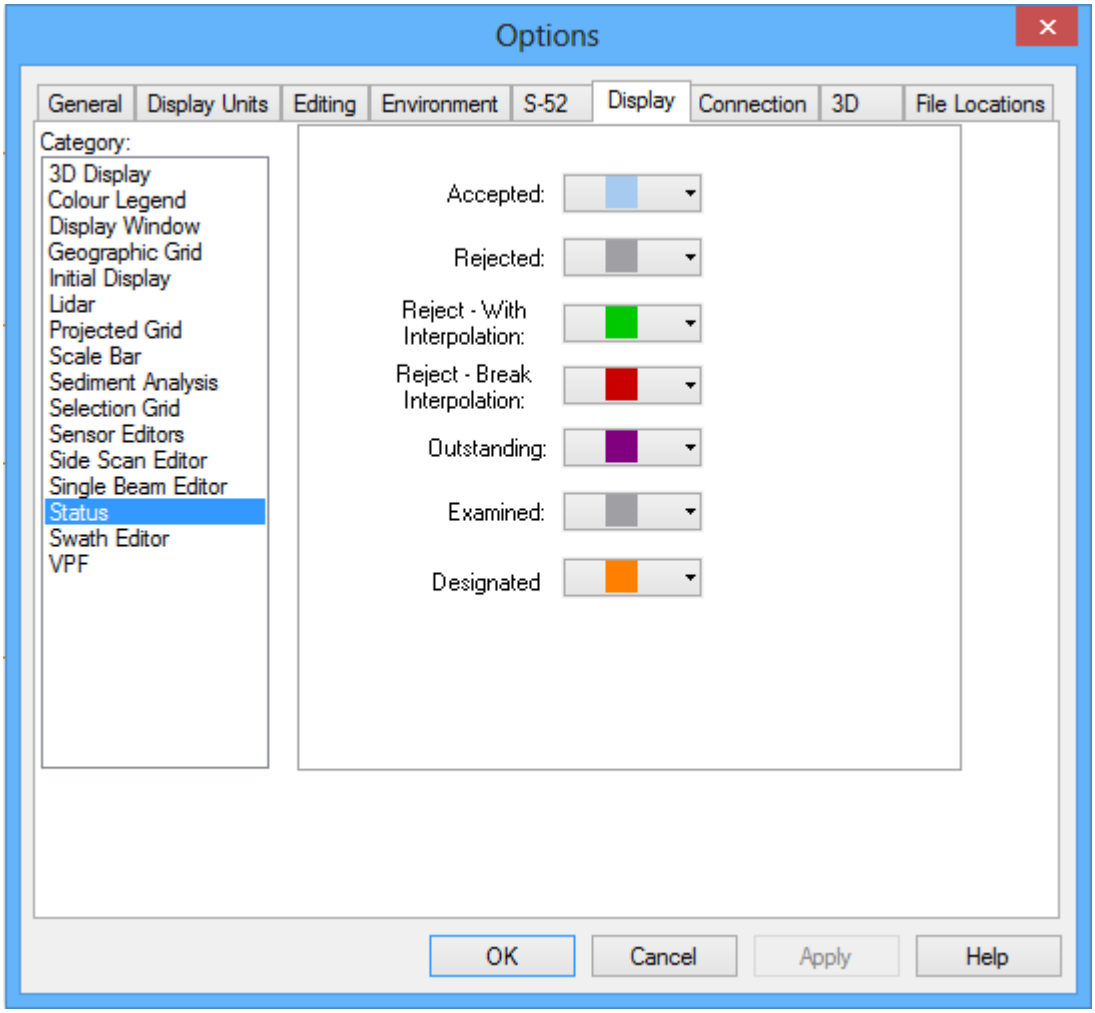
1. Set the colours for data by selecting from the appropriate colour palette, or create a custom colour from the Windows colour picker.
   - *Primary*: Colour for primary frequency soundings in a dual-frequency system.
   - *Secondary*: Colour for secondary frequency soundings in a dual-frequency system.
   - *Selected*: Colour for soundings that are selected in the Single Beam Editor window.
   - *Track Line Highlight*: The colour in the Display window for the section of track line selected in the Display window and visible in the Single Beam Editor window.

- *Event Marks*: A unique marker placed at regular intervals in the survey.

2. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Status

The Status tab controls the status colours for soundings.



1. Select a colour from the colour palette, or create a custom colour from the colour selector within the palette.

Status	Description
Accepted	Colour for data that is flagged as ready for processing.
Rejected	Colour for data that is flagged as rejected and excluded from further processing.
Reject - With Interpolation	Colour for attitude and navigation data that has been flagged as rejected but where associated soundings can still be processed.
Reject - Break Interpolation	Colour for attitude and navigation data that has been flagged as rejected and where associated soundings are also rejected.
Outstanding	Colour for soundings in the Subset Editor that have been flagged as needing further examination.
Examined	Colour for soundings in the Subset Editor that have been flagged as examined and verified.

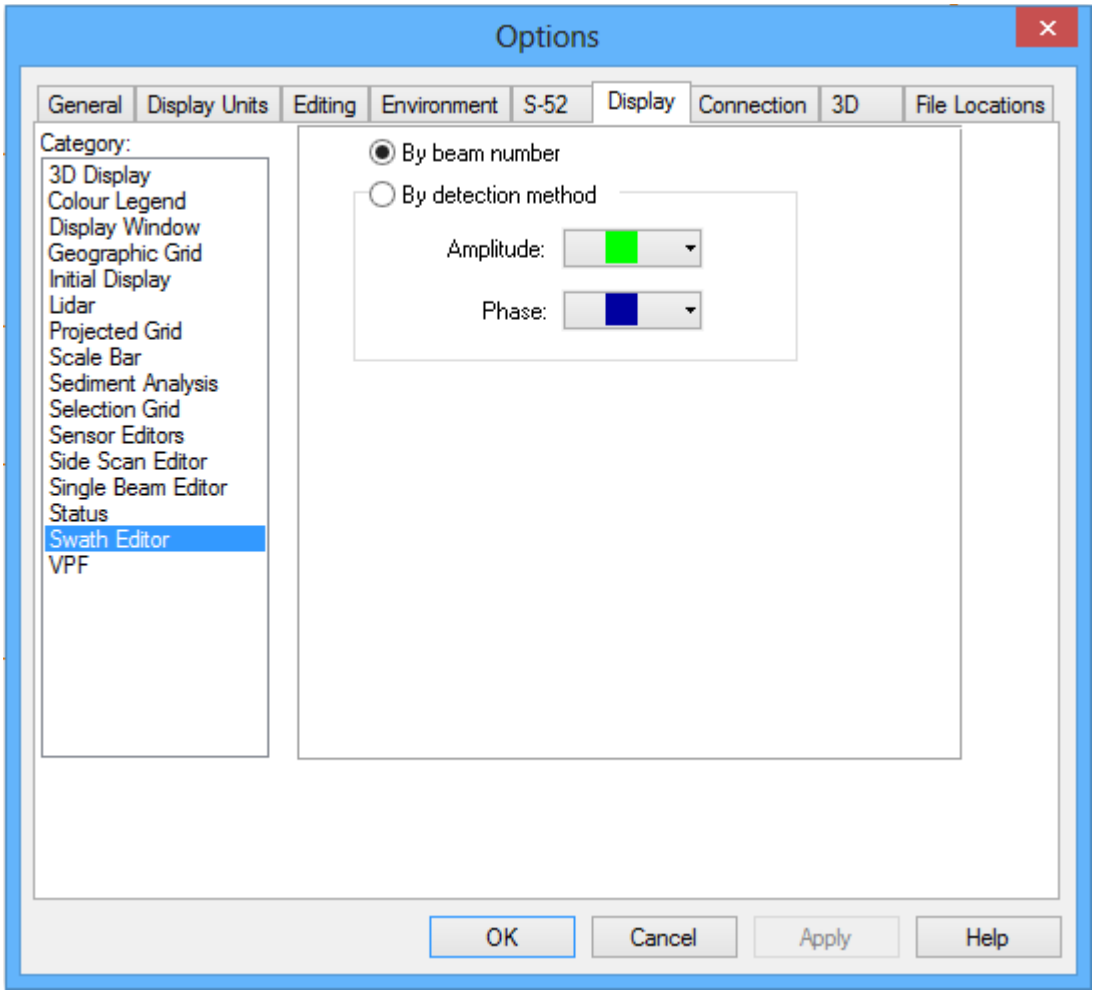
Status	Description
Designated	Colour for the sounding that has been flagged in the Swath or Subset Editors as being the shoalest sounding on a feature.

2. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Swath Editor

Use the Swath Editor options to set colours for the Plan View display. Soundings can be coloured by location in the swath (port or starboard), or by detection method.

Soundings in the Rear, Side, and Profile Views are always coloured by their location in the swath.



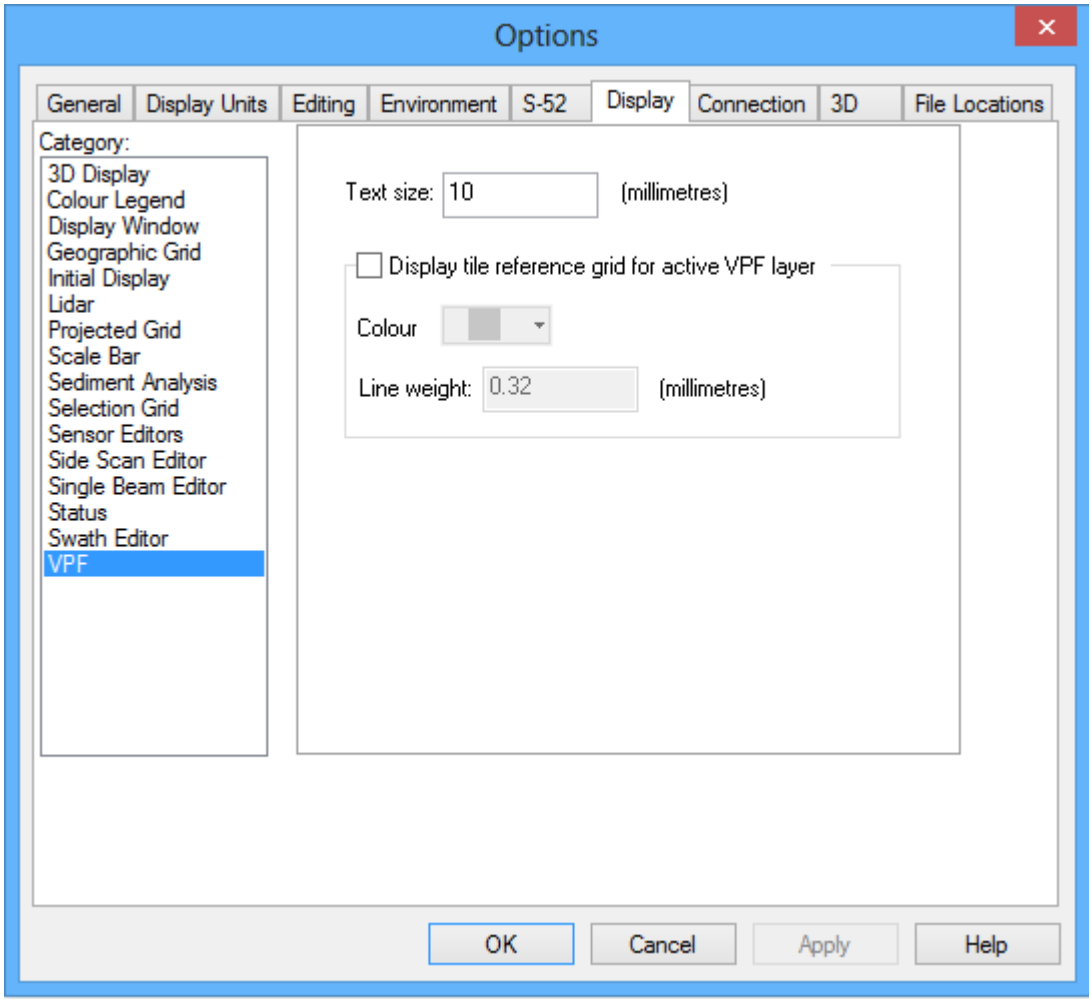
1. Click the *By Beam Number* box to enable colour options for port and starboard.

Colour options for Port and Starboard are set in the Sensor Editors page. See "SENSOR EDITORS" ON PAGE 201

2. Click the *By Detection Method* box so the amplitude and phase colour options are enabled.
3. Use the *Phase* and *Amplitude* colour palettes to select a colour, or create a custom colour from the standard Windows colour selector.
4. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

VPF

Use this dialog box to define display settings for VPF data.



The *Text Size* option controls the size which all text will be displayed in a VPF file. This setting will be loaded each time a VPF file is opened. The default size is 10 millimetres at map scale. Only whole positive values can be entered in this field.

To set *Text Size*:

1. Type a value in millimetres at which to display all text in a VPF file.

You can display a reference grid that identifies the cell tiles of the currently active VPF layer. Changing the active layer will either remove the grid if it is not a VPF layer, or change the grid to reflect the newly selected VPF layer.

To display a tile reference grid for active VPF layer:

2. Enable the check box to display a reference grid for the tiles of the currently active VPF layer.

To display a colour for lines in the tile reference grid:

3. Select a colour from the colour picker, or create a custom colour from the standard Windows colour palette

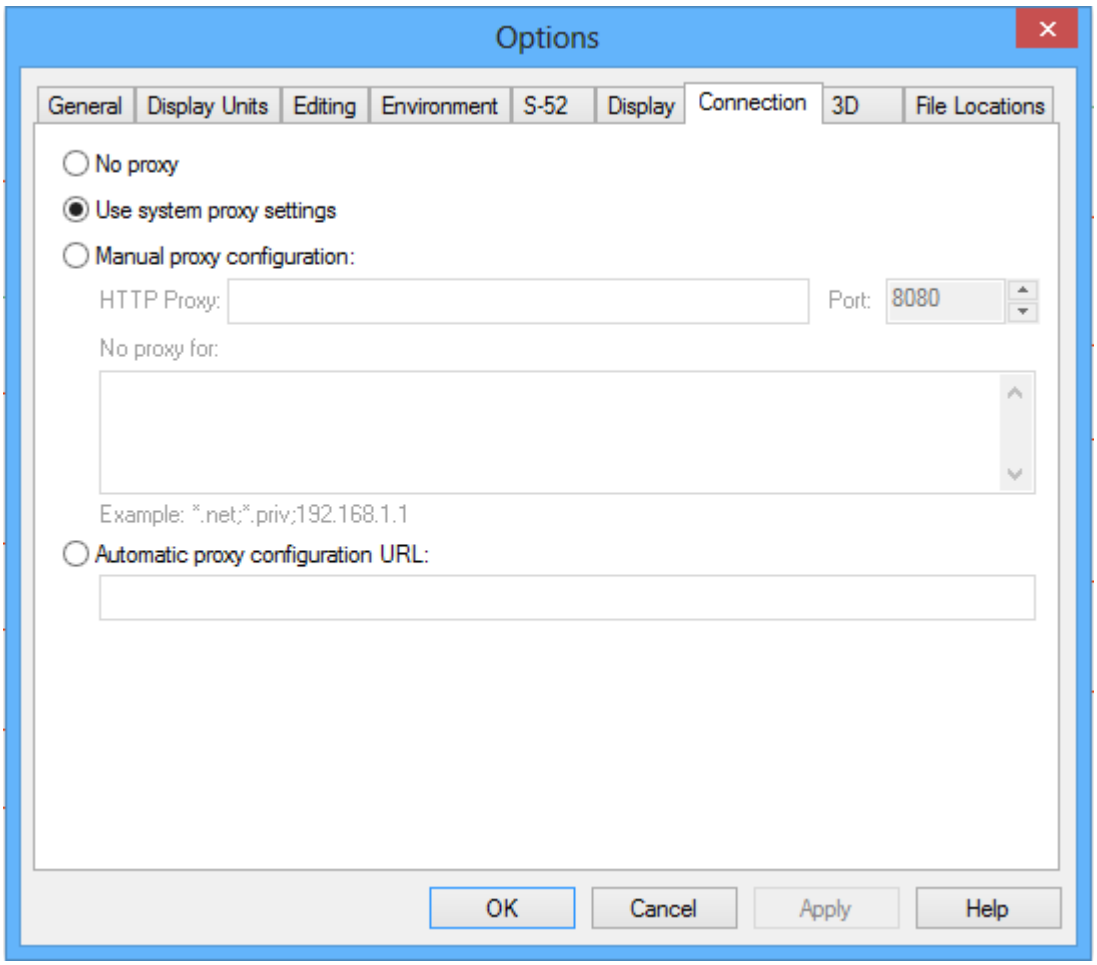
To set Line Weight:

4. Type a value in mm for the thickness of the lines in the tile reference grid.
5. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Connection

The Connection tab is used to define proxy settings so that external URLs can be reached even if HIPS and SIPS is behind a firewall.

The default setting is to use the settings defined for the operating system.



Option	Description
No proxy	No proxy setting is required because the application is not on a network computer behind a firewall.
Use system proxy settings	Use the proxy server settings defined for the operating system. These are defined in the Control Panel under the Local Area Network (LAN) settings This is the default setting.

Option	Description
Manual proxy configuration	Manually define proxy settings. • *HTTP Proxy*: The name of the network computer acting as the proxy server. • *Port*: The port number on the server computer being used to send and receive information. Select a value using the arrows or type it manually into the field. • *No proxy for*: Specify connections that do not require a proxy server but might trigger the firewall. List these connections, separated by semicolons. For example: `*.priv; 192.168.1.1;*.net`
Automatic proxy configuration URL	Specify the location of a Proxy Automatic Configuration (PAC) file on the network. A PAC file is a JavaScript function which contains network IP addresses and predefined settings for the proxy server and port.

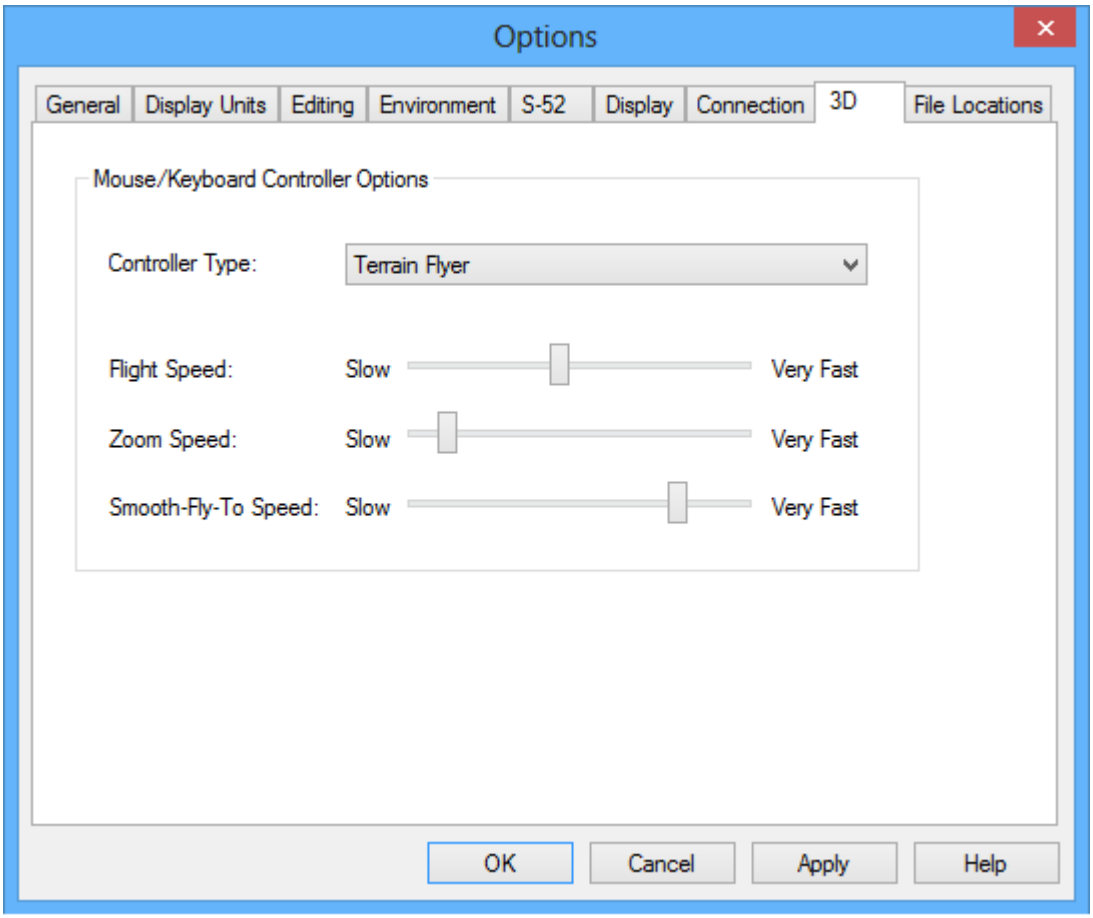
Example of a PAC file.

```
function FindProxyForURL(url, host)
{
    if (host == "127.0.0.1" ||
        isPlainHostName(host) ||
        dnsDomainIs(host, ".company_name.priv") ||
        dnsDomainIs(host, ".company_name.com") ||
        isInNet(host, "1.1.1.1", "255.255.255.255") ||
        isInNet(host, "192.168.0.0", "255.255.0.0") ||
        isInNet(host, "201.160.52.0", "255.255.255.192")
        )
        return "DIRECT";
    else
        return "PROXY server_name.company_name.priv:port#; ";
}
```

3D

See also "NAVIGATE THE 3D DISPLAY" ON PAGE 91 and "3D FLIGHT PATH" ON PAGE 98.

The 3D tab provides options for adjusting the settings of the keyboard and mouse controllers used to navigate through data in the 3D Display window.



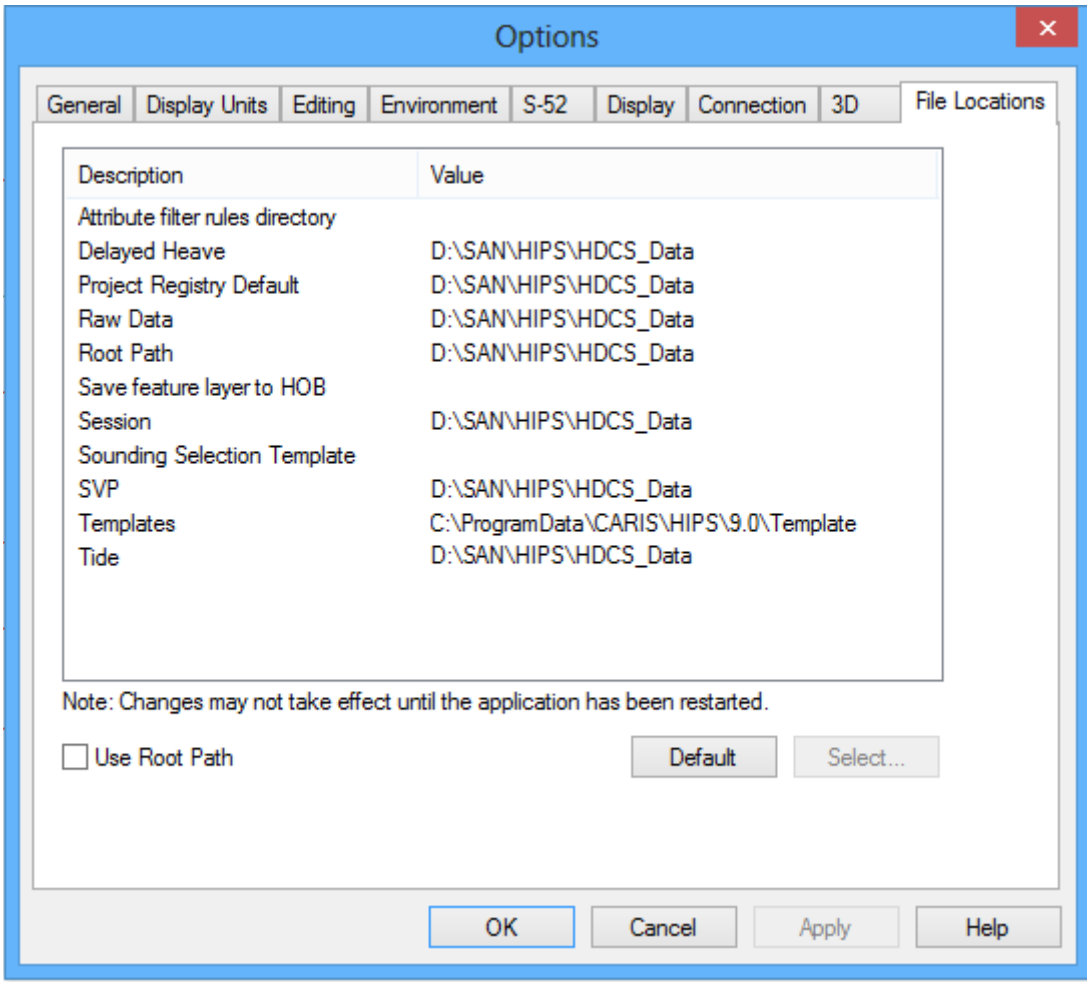
The following options are available:

- **Controller Type**: Select one of two types of controls to use when navigating the 3D View:
  - Terrain Flyer: enables you to navigate from the perspective of the height source.
  - First Person: enables you to navigate from a camera view of the current 3D scene.
- **Flight Speed**: Controls the speed at which you move through the display during a fly-through. Move the slider left or right to increase or decrease flying speed.

- **Zoom Speed**: Controls how fast the view is changed when using the zoom controls. Move the slider bar left or right to increase or decrease zooming speed.
- **Smooth-Fly-To Speed**: Controls how fast you zoom in when you double-click on the surface or terrain in the 3D View. Move the slider left or right to increase or decrease zooming speed.

File Locations

The File Locations tab sets the locations of the folders that contain data such as raw data, tide data, projects and saved sessions.



:

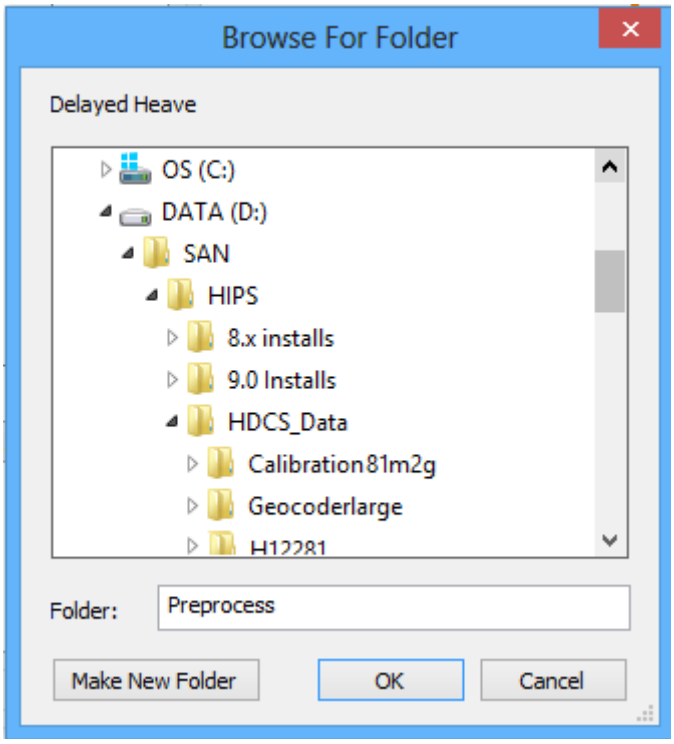
Option	Description
Attribute filter rules directory	The location in which rule files are saved when exported from the Rule Wizard. All rules in this folder are proposed as templates for newly created rules.
Save feature layer to HOB	The location in which HOB files will be created by default if a feature layer is saved to a new file.
Sounding selection template	The location in which sounding selection templates are stored. The application accesses this location during sounding selection if the user opts to create a template of the sounding selection criteria, or to define selection criteria using a template.

The *Root Path* field displays the location of the HIPS and SIPS data directories. This location must contain folders with the same names as these default data directories.

To set the root path:

1. Select the *Use Root Path* check box.
2. Click **Select**.

The Browse for Folder dialog box is displayed.



3. Expand the directory tree to select the folder you want to set as root.
4. Click **OK**.

Change paths

You can change the path to an individual directory listed. To change a default file location:

5. Double-click on a directory in the list, or select it and click **Select**.

The Browse for Folder dialog box is displayed. The name of the selected file is displayed above the directory tree.

6. Expand the directory tree to view the directory you want to change to.
7. Select the folder.
8. Click **OK** to close the Browse for Folder dialog box.

The new path is displayed in the *Value* column of the Options dialog box.

9. Click **Apply** to implement the changes to File Locations without closing the dialog box, or **OK** to implement the changes and close the Options dialog box.

6

Data Directories

This section describes the Project/Vessel/Day/Line structure of the data directories in HIPS and SIPS.

In this chapter...

Introduction

The structure of the HIPS and SIPS data directories is designed to maximize data access efficiency during processing. By definition, these directories only store raw or processed data and products, not the HIPS and SIPS applications nor their associated system files and libraries.

The data directories are saved by default to the My Documents folder, as `..\Documents\CARIS\HIPS and SIPS\ 9.0`.

For consistency with older code contained within HIPS and SIPS, directory names and paths should not contain any spaces, or any special Windows characters such as `(\, /, :, *,?,",<,>, or |)`.

Projects

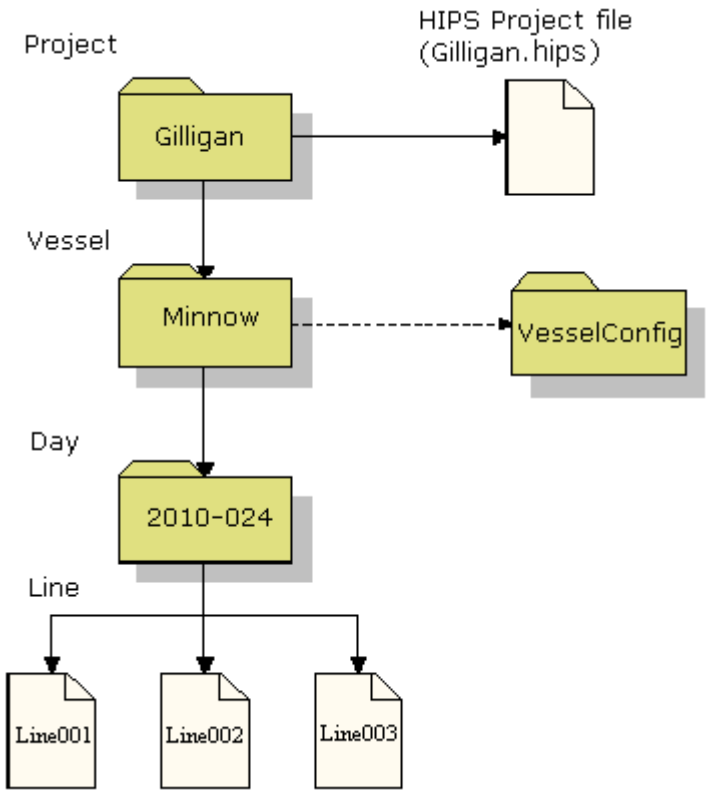
The Project directory holds all of the converted data. The Project directory contains two main components:

- HIPS and SIPS data
- Vessel files

HIPS and SIPS Data

Converted data in HIPS and SIPS is stored in a Project/Vessel/Day/Line (P/V/D/L) structure. Each survey project is divided into one or several vessels (for example, survey launches), contributing to the same project. The data can be from different generations of surveys, or different hardware collection platforms. Each vessel folder is divided into separate survey days and each survey day contains one or more survey lines.

Below is an outline of the P/V/D/L structure for a project called Gilligan with a vessel called Minnow. The project contains three line files.



In the Project directory, in addition to the folder(s) containing the vessel(s), there is also a HIPS file (*.hips) that contains the projection information used by the HIPS and SIPS interface to define the default projection for that project. When HIPS and

SIPS accesses the navigation data (which is stored in un-projected latitude and longitude), the data is projected using the projection stored in this file. The HIPS file is given the same name as the project.

For information on creating a new project see "CREATE A NEW PROJECT" ON PAGE 61.

The name of the vessel in the P/V/D/L path provides the link to the actual HIPS Vessel File and therefore must be spelled the same as the HVF in the `VesselConfig` directory. The data cannot be processed if the HVF is not present or if the vessel name in the P/V/D/L path is misspelled.

The survey day in the P/V/D/L structure is composed of two components separated by a hyphen—the four-digit year and the three-digit Julian date. The date used in the Day directory name does not actually play a role in data processing—it is used as a convenient mechanism for organizing survey data.

When a new project is created, this P/V/D/L directory structure is built and the HIPS file is generated. The vessel file must exist in the `VesselConfig` directory at the time the project is defined, however it can be modified at any time (providing the name is not changed).

VesselConfig Directory

The `VesselConfig` directory contains the HIPS Vessel File (HVF) used in the vessel component of the P/V/D/L path. The name of this directory must not be changed and it always has to be located inside the Raw Data directory on the same level as the project directories.

For more information about the HVF see "VESSEL FILES" ON PAGE 30.

Raw Data

The raw data directory is used to store un-processed files that are used as input into HIPS via the Conversion Wizard. Un-processed files can be stored in any order and in any location on your system. However it is recommended that you store these files in a folder structure that mimics the P/V/D/L structure of the Projects folder.

Not only does the P/V/D/L structure provide a consistent path for data retrieval. You can also take advantage of a feature in the Conversion Wizard that lets you open for conversion at any level of the P/V/D/L folder structure and maintain that structure through conversion.

For example, a new project called “Gilligan” is created by the New Project wizard without adding a vessel or survey days to the project definition.

Raw data is then converted to HIPS and SIPS format through the Conversion Wizard. The File Selection Type is set to “Project” in Step 2 of wizard. The converter then knows that data is stored in the P/V/D/L structure and that you are loading from the Project level. All folders on the next level are interpreted as vessel names and the level after that as survey days.

Each vessel name must exist in the `VesselConfig` directory, otherwise the converter will report errors. In this example, there is only one vessel folder, called “Minnow,” and it contains several survey days. There happens to be a HVF in the `VesselConfig` directory called “Minnow”, so the converter creates the vessel folder beneath the project name “Gilligan” in the Project directory and also all of the survey days that are necessary. If the survey days or the vessel are already present, the converter uses them.

Finally, all of the lines found are converted.

To convert raw data that is not stored in the P/V/D/L structure, simply set the file selection type to “Raw” and select the files individually.

For more information on converting data to HIPS/SIPS format see “Rename Day and Line Folders” on page 76.

SVP and Tide

The SVP folder contains all of the sound velocity profile files. They can be stored in subdirectories of this folder if desired.

The Tide Folder contains all of the tide files. They can be stored in subdirectories if desired.

7

HIPS Status Flags

Within HIPS and SIPS data structure each sensor data value has a series of status flags. Each profile in the depth and side scan formats, and each survey line also has status flags.

The status flag settings are determined during data cleaning and processing. They are used to control the generation of new data layers such as BASE surfaces, tiles, and selected soundings.

Status Flags and Values

Status Flag	Value
Accepted	The default status of sounding.
Rejected	True if the sounding is rejected for any reason.
Examined	Indicates that the sounding was verified.
Designated	Indicates that the sounding is the shoalest in a cluster of soundings.
Outstanding	Indicates that the sounding needs further examination.
Rejected by Swath Editor or Single Beam Editor	True if sounding was rejected in the Swath Editor or Single Beam Editor, or by the swath or single beam filters.
Rejected by Hydrographer	True for soundings rejected manually by the hydrographer within the Subset Editor.
Rejected by Surface Cleaning	True if sounding was rejected during surface cleaning
Rejected by Depth Gate	True for soundings rejected by the depth gate during conversion.
Rejected by Disabled Beam	True for soundings rejected at conversion time because of a disabled beam flag in the HVF or due to a pre-flagged bad sounding in the original raw data format.
Rejected by TPU	True for soundings rejected during Total Propagated Uncertainty filtering.
Rejected by surface filter.	True for soundings rejected during surface filtering.
Rejected by Auto Classification	True if the sounding fails the auto classification tests filter contained in the HIPS v4.3.3 subset mode. This flag is not used in v5.x software.
Rejected by Bad Navigation	True if the profile was rejected for bad navigation.
Rejected by Bad Gyro	True if the profile was rejected for bad gyro.
Rejected by Bad Heave	True if the profile was rejected for bad heave.
Rejected by Bad Pitch	True if the profile was rejected for bad pitch.
Rejected by Bad Roll	True if the profile was rejected for bad roll.
Rejected by Bad Tide	True if the profile was rejected for bad tide.

8

Contact File Formats

Contacts are representations of features on the sea bottom that are visible in the sonar data. They are added to the data using the Side Scan Editor. Information about the contacts in a project can be exported into the file formats described here.

In this chapter...

Introduction

Contacts are created in the Side Scan Editor waterfall view, and saved in the *.hips file. For information on creating and editing contacts, see CONTACTS ON PAGE 98 OF THE HIPS AND SIPS EDITORS GUIDE.

When you export contacts, a set of output files are produced that represent all contacts from all of the survey lines in your project.

The Contact export function from SIPS produces four files:

- **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 single point contacts.
- **filename_ContactMultiPoints.txt** contains all location information for multipoint contacts.
- **filename_ContactTiff.gat** contains all image files created during export.

These files can be saved in the line folder, or to a folder you specify, and will use the file name you set during export. See "CONTACTS" ON PAGE 506 of the User Guide for information on exporting contact data for the project.

ContactLine

The filename_ContactLine.txt file contains the names of the HIPS track lines and the corresponding index number. Each line in the file is a comma- delimited string terminated by the newline character (**\n**). Each line has the following format:

Field Name	Field Description	Description
Line Index	4-digit integer	The index number for the line.
Project Name	256 characters (maximum)	The name of the HIPS project that the line belongs to.
Vessel Name	256 characters (maximum)	The name of the HIPS vessel that the line belongs to.
Day Name	256 characters (maximum)	The name of the HIPS day that the line belongs to.
Line Name	256 characters (maximum)	The name of the HIPS line that the contact refers to.

ContactSingle

The filename_ContactSingle.txt file contains the list of single point contact features from the export. Each line in the file is a comma (',')delimited string terminated by a new line character ('\n'). Each line in the file has the following format:

Field Name	Field Description	Description
Contact Index	4-digit integer	The index number for the contact. Index numbers are assigned sequentially from 0 when the export is performed. Guaranteed to be unique for all exported single contacts only.
Key	12 characters	The key for the contact. This is created by concatenating the line name and the contact number together. The first 8 bytes are the first 8 bytes in the line name, and the last 4 bytes are the system generated number assigned to the contact.
Line Index	4 digit integer	The line record number from the corresponding Filename_ContactLine.txt file.
Contact Number	4 digit integer	The system generated contact number assigned to the contact when it was created in the SIPS software (same as the last 4 bytes of the 'key' field).
Creation Date	10 characters	The date the contact was created. Has the mm/dd/yyyy format
Creation Time	10 characters	The time the contact was created. Has the following format: HH:MM:SS.S
Created User ID	3 characters	The user ID entered in the SIPS software at the time of contact creation (usually user initials)
Modified Date	10 characters	The last date the contact was modified. It may be the same as the Creation Date if the contact has not been modified since creation. It has the following format: mm/dd/yyyy
Modified Time	10 characters	The time of the last modification done to the contact. The time may be the same as the Creation Time if the contact has not been modified since it was created. It has the following format: HH:MM:SS.S
Modified User ID	3 characters	The user ID of the user that last modified the contact. The ID may be the same as the Created User ID if the contact has not be modified since it was created.
Contact Prefix	8 characters	The prefix assigned by the SIPS software when the user created the contact. This is normally used to generate the key for the contact.
Feature Code	12 characters	The CARIS Feature code used to represent the point contact.
Contact Type	1 character	This identifies the contact record as either a single point or multi point contact. Single point contacts are represented by a '1' and multipoint contacts are represented by a '2'.
Profile Number	4 digit integer	The profile number that the contact was picked from in the SIPS software.
Across Distance	6.2 Floating point	The across-track distance recorded by the SIPS software that the point contact was picked from.
Latitude	13.7 Floating point	The latitude of the point contact.
Longitude	13.7 Floating point	The longitude of the point contact.
Target Height	6.2 Floating point	The height of the target (in metres).

Field Name	Field Description	Description
Target Width	6.2 Floating point	The width of the target (in metres).
Target Length	6.2 Floating point	The length of the target (in metres).
Image Height	4 digit integer	The height of the contact image as defined by the user in the SIPS software.
Image Width	4 digit integer	The width of the contact image as defined by the user in the SIPS software.
Status	1 digit integer	The status of the contact. 0 if the contact is Accepted, 1 if the contact is Rejected.
Remarks	85 (maximum) characters	Remarks entered by the hydrographer about the created contact.

ContactMulti

The filename_ContactMulti.txt file contains the list of multipoint contact features from the export. Each line in the file is a comma (',') delimited string terminated by a newline character ('\n'). Each line in the file has the following format:

Field Name	Field Description	Description
Contact Index	4-digit integer	The index number for the contact. Index numbers are assigned sequentially from 0 when the export is performed. It is guaranteed to be unique for all exported multi point contacts only.
Key	12 characters	The key for the contact. It is created by concatenating the line name and the contact number together. The first 8 bytes are the first 8 bytes in the line name, and the last 4 bytes are the system generated number assigned to the contact.
Line Index	4 digit integer	The line record number from the corresponding Filename_ContactLine.txt file.
Contact Number	4 digit integer	The system generated contact number assigned to the contact when it was created in the SIPS software (same as the last 4 bytes of the ‘key’ field).
Creation Date	10 characters	The date the contact was created. It has the mm/dd/yyyy format
Creation Time	10 characters	The time the contact was created. It has the following format: HH:MM:SS.S
Created User ID	3 characters	The user ID entered in the SIPS software at the time of contact creation (usually user initials)
Modified Date	10 characters	The last date the contact was modified. The data maybe the same as the Creation Date if the contact has not been modified since creation. It has the following format: mm/dd/yyyy
Modified Time	10 characters	The time of the last modification done to the contact. The time maybe the same as the Creation Time if the contact has not been modified since it was created. It has the following format: HH:MM:SS.S
Modified User ID	3 characters	The user ID of the user that last modified the contact. The ID maybe the same as the Created User ID if the contact has not be modified since it was created.
Contact Prefix	8 characters	The prefix assigned by the SIPS software when the user created the contact. This is normally used to generate the key for the contact.
Feature Code	12 characters	The CARIS Feature code used to represent the point contact.
Contact Type	1 character	This identifies the contact record as either a single point or multi point contact. Single point contacts are represented by a ‘1’ and multipoint contacts are represented by a ‘2’.
Target Height	6.2 Floating point	The height of the target (in metres).
Target Width	6.2 Floating point	The width of the target (in metres).
Target Length	6.2 Floating point	The length of the target (in metres).
Image Height	4 digit integer	The height of the contact image as defined by the user in the SIPS software.

Field Name	Field Description	Description
Image Width	4 digit integer	The width of the contact image as defined by the user in the SIPS software.
Status	1 digit integer	The status of the contact—0 if the contact is Accepted, 1 if the contact is Rejected.
Remarks	85 (maximum) characters	Remarks entered by the hydrographer about the created contact.

ContactMultiPoints

The filename_ContactMultiPoints file contains the list of points used to construct the geometry of the multi point contacts listed in the Filename_ContactMulti.txt from the export. Each line in the file is a comma (',')delimited string terminated by a newline character ('\n'). Each line in the file has the following format:

Field Name	Field Description	Description
Record number	4-digit integer	The record number. It is unique throughout the file.
Contact Record Number	4-digit integer	The contact record number that this point belongs to. This corresponds to the Contact Index in the corresponding Filename_ContactMulti.txt file.
Profile Number	4-digit integer	The profile number that the point was computed from in the SIPS software.
Across Distance	6.2 floating point	The across-track distance recorded by the SIPS software that the point contact was picked from.
Accumulated Distance	6.2 floating point	The accumulated distance. This is the distance from the first point in the set, to this point.
Latitude	13.7 floating point	The latitude of the point.
Longitude	13.7 floating point	The longitude of the point.

9

Tide File Formats

HIPS supports a standard Canadian Hydrographic Service tide format called COWLIS as well as two NOS/NOAA formats. HIPS also supports a basic format consisting of just the required date, time, and tide values.

HIPS does not use predicted tide tables. The time zone of the tidal observation data must match the time zone of the bathymetry to which it will be applied.

As an alternative, a Computed GPS Tide calculation can be used with data in HIPS

In this chapter...

Tide files (*.tid)

TID files are ASCII files consisting of date, time, and tide values collected from a single tide station. These files can be created or edited in a text editor (such as Notepad), or in the Tide Editor.

The following tide data file formats are supported by HIPS:

- CHS Cowlis
- CHS TMS 50
- CHS TMS 1000
- NOS Preliminary Tide Data
- NOS Final Tide Data
- NHS Tide file

Basic Tide Format

The basic tide format used by HIPS begins with a single line containing at least 8 dash characters, followed by tidal observation records consisting of date, time, and tide. Here is a simple example:

```
--------
2000/11/01 00:00   0.522
2000/11/01 01:00   0.317
2000/11/01 02:00   0.395
2000/11/01 03:00   0.751
2000/11/01 04:00   1.296
2000/11/01 05:00   1.848
2000/11/01 06:00   2.339
2000/11/01 07:00   2.589
```

To apply tide data, see "CORRECT FOR TIDE" ON PAGE 169 in the User Guide. See also "TIDE EDITOR" ON PAGE 177 in the Editors guide.

COWLIS

```
---------------------- LIBEX file dump -----------------------
----------------------- STN?????.OBS -----------------------
------------------ Time Zone:  GMT ( 0.0) ------------------

--------------------- Invariant Fields ---------------------
Name             Type Size    Units Value
------------------------------------------------------------
station_id       CHAR    5
station_name     CHAR   16          Rimouski, Quebec
data_product     CHAR    3          TMS1000
start_time       INTG    4  seconds 93/06/29 18:45:00
end_time         INTG    4  seconds 91/11/16 20:45:00
file_date        INTG    4  seconds 95/12/20 15:24:24
max_water_level  REAL    4   metres    1.640
min_water_level  REAL    4   metres    0.050
------------------------------------------------------------

---------------------- Variant Fields ----------------------
Name             Type Size    Units
------------------------------------------------------------
time             INTG    4  seconds
water_level      REAL    4   metres
std_dev          REAL    4   metres
------------------------------------------------------------
1993/06/29 18:45:00   1.95  0.030
1993/06/29 19:00:00   1.84  0.030
1993/06/29 19:15:00   1.74  0.030
1993/06/29 19:30:00   1.63  0.030
1993/06/29 19:45:00   1.55  0.030
1993/06/29 20:00:00   1.48  0.030
1993/06/29 20:15:00   1.41  0.030
1993/06/29 20:30:00   1.37  0.030
1993/06/29 20:45:00   1.34  0.030
-------------------- End Of LIBEX File ---------------------
```

NOS Preliminary Tide Data

```
************************ PRELIMINARY DATA ***************************

     THIS RAW DATA HAS NOT BEEN SUBJECT TO THE NATIONAL OCEAN SERVICE'S
     QUALITY CONTROL OR QUALITY ASSURANCE PROCEDURES AND DOES NOT MEET
     THE CRITERIA AND STANDARDS OF OFFICIAL NATIONAL OCEAN SERVICE
     DATA.  IT IS RELEASED FOR LIMITED PUBLIC USE AS PRELIMINARY DATA TO
     BE USED ONLY WITH APPROPRIATE CAUTION.

************************ PRELIMINARY DATA ***************************

      Water Level -  Acoustic (A1)
      Station    --   Unique seven character identifier for the station
      DCP        --   A one character identifier for the data collection
                      platform at a station
      SE         --   A two character identifier for the data sensor
      Date Time  --   Date and time the data were collected by the DCP
      WL_Value   --   Water level height
      Sigma      --   Standard deviation of 1 second samples used to
                      compute the water level height
      Out        --   Count of number of samples that fall outside a
                      3-sigma band about the mean
      Flat       --   A flag that when set to 1 indicates that the flat
                      tolerance limit was exceeded
      ROFC       --   A flag that when set to 1 indicates that the rate
                      of change tolerance limit was exceeded
      Temp       --   A flag that when set to 1 indicates that the
                      temperature difference tolerance limit was exceeded
      Limit      --   A flag that when set to 1 indicates that either the
                      maximum or minimum expected water level height limit
                      was exceeded

      Data are in meters above MLLW
      Times are on UTC (GMT)

9453220 1 Yakutat, AK  from  19990614 to 19990618
  ------------------------------------------------------------------------
Station  DCP  SE  Date       Time   WL_Value   Sigma  Out Flat RofC Temp Limit

9453220   1   A1  1999/06/14 00:00     1.881   0.014   0    0    0    0    0
9453220   1   A1  1999/06/14 00:06     1.814   0.025   0    0    0    0    0
9453220   1   A1  1999/06/14 00:12     1.746   0.018   0    0    0    0    0
9453220   1   A1  1999/06/14 00:18     1.692   0.014   0    0    0    0    0
9453220   1   A1  1999/06/14 00:24     1.632   0.013   0    0    0    0    0
9453220   1   A1  1999/06/14 00:30     1.560   0.015   0    0    0    0    0
  ------------------------------------------------------------------------
```

NOS Final Tide

```
 NOS SIX MINUTE WATER LEVEL HEIGHTS

      Data are verified

      Station   --   Unique seven character identifier for the station
      Date Time --   Date and time the data were collected by the DCP
      WL_Value  --   Water level height
      Sigma     --   Standard deviation of 1 second samples used to
                     compute the water level height
      Infer     --   A flag that indicates that the water level value
                     has been inferred.
      Flat      --   A flag that when set to 1 indicates that the flat
                     tolerance limit was exceeded
      RofC      --   A flag that when set to 1 indicates that the rate
                     of change tolerance limit was exceeded
      T_Flag    --   A flag that when set to 1 indicates that the
                     temperature difference tolerance limit was exceeded

      Data are in meters above MLLW
      Times are on UTC (GMT)

9452210 Juneau, Gastineau Channel, Stephens Pass, AK, USA  from  19980525 to
19980530
  ------------------------------------------------------------------------
Station Date       Time    WL_Value   Sigma  Infer Flat RofC T_Flag

9452210 1998/05/25 00:00     2.473     0.019     0    0    0     0
9452210 1998/05/25 00:06     2.330     0.022     0    0    0     0
9452210 1998/05/25 00:12     2.200     0.019     0    0    0     0
9452210 1998/05/25 00:18     2.064     0.020     0    0    0     0
9452210 1998/05/25 00:24     1.938     0.019     0    0    0     0
  ------------------------------------------------------------------------
```

NHS (Norwegian Hydrographic Service)

This is an example of an NHS tide file.

```
#Site name: Stavanger
#Provider: Norwegian Hydrographic Service
#Latitude: 58.9700
#Longitude: 5.7300
#Datum: EUREF89
#Time interval: 600 seconds
#Reference level: CD (Chart Datum)
#Series1: Water level observations
#Series2: Predicted tide
#Series3: Residuals (observed-predictions)
#Unit: cm

2011-12-24T00:00+01                              97.0     82.0      15.0
2011-12-24T00:10+01                              97.0     81.0      16.0
2011-12-24T00:20+01                              96.0     80.0      16.0
2011-12-24T00:30+01                              94.0     79.0      15.0
```

Tide Zone Definition Files

The following is an example of a Zone Definition File (.zdf).

```
[ZONE_DEF_VERSION_3]

[ZONE]
PIS1,10
43.069551, -70.687251
43.061704, -70.717396
43.067896, -70.717241
43.071497, -70.712564
43.072329, -70.710498
43.075112, -70.710772
43.078966, -70.71016
43.081576, -70.708724
43.082372, -70.690817
43.069551, -70.687251

[ZONE]
PIS2,11
43.08703, -70.720529
43.07693, -70.724628
43.072765, -70.724696
43.071497, -70.712564
43.072329, -70.710498
43.075112, -70.710772
43.078966, -70.71016
43.081576, -70.708724
43.083356, -70.715428
43.086439, -70.715696
43.08703, -70.720529

[TIDE_ZONE]
PIS1,419870,PRIM,-12,1.06,0.0,0.2
PIS2,419871,PRIM,-12,1.05,0.0,0.2

[TIDE_STATION]
419870,43.067896,-70.717241,2.0,0.1,D:\HIPS\Tide\419870.tid
419871,43.072329,-70.710498,1.0,0.15,D:\HIPS\Tide\419870.tid

[TIDE_AVERAGE]
PIS1,419870,419871
PIS2,419870,419871

[OPTIONS]
Outage, 600
Interval, 10
```

Tide Zone Data

The Zone Definition File has the following components:

- The mandatory header section must consist of this text string: [ZONE_DEF_VERSION_3]. This specifies the version of the Zone Definition File.
- The section labelled [ZONE] contains the name of the zone(s) and the coordinates that define the boundary of the zone(s). All tide zone files must contain this section.

  There are two fields in the first line:

  - <Zone Label> contains name of the zone (the name must not be longer than 256 characters).

- number of points that define the boundary of the zone. (These are listed starting in the next line of the section.)

The remaining lines in this section list the geographic coordinates of these points, (latitude, longitude) in decimal degrees.

The last line in the list must contain the same coordinates as the first line to close the polygon.

If there is more than one zone being defined, this section is repeated with the definition information for each zone.

Tide zones should not overlap each other.

- The section labelled [TIDE_STATION] contains the definition of the tide stations collecting the tide observations. There are six fields in each record of this section:
  - *station label* (the name of the tidal station) - The name must not be longer than 256 characters.
  - *latitude* in decimal degrees - Valid ranges are from -90.0 to +90.0. The negative range represents the Southern Hemisphere.
  - *longitude* in decimal degrees - Valid ranges are from -180.0 to +180.0. The negative range represents the Western Hemisphere.
  - *max amplitude* (in metres) expected from the tidal station - This value will be used to compute a scaling factor during the Tidal TPU computation.
  - *uncertainty* error value associated with the data from the station (in metres)
  - [optional] full file path and name, or just the name of the tide file associated with the station ID. If this is omitted, HIPS will look for an observation file in the same directory as the *.zdf, with the file name <Station_Label>.tid.
- The section labelled [TIDE_ZONE] contains specific attributes that define a zone's tidal parameters. There are seven fields in this section:
  - tide *zone label* - this must match the name of the zone(s) defined in the [ZONE] section.
  - tide *station label* - name of the tidal station associated with this entry.
  - *priority* designation of the tidal station entry: either the primary (PRIM), secondary (SEC), tertiary (TER) or preliminary (PRELIM) station for this zone.

  - *time correction* - the tide offset for this zone (in minutes) (This is not used when tide is loaded with the weighted average option enabled.)
  - *range correction* - a multiplier (for example, 1.06) used to scale the tidal value read from the observation file (This is not used when tide is loaded with the weighted average option enabled.)
  - *tidal shift* - a number in metres added to the values after range correction has been applied.
  - *uncertainty* value for the tidal zone (in metres) used to compute the Tidal Zone Error.
- The section labelled [TIDE_AVERAGE] assigns tide stations to zones for tide-weighted averaging. Each record has at least two station fields because each zone can be assigned one or more tide stations for the weighted average calculation.

  The section has the following fields:
  - tide *zone label* - the zone (as defined in the [ZONE] section) to which the averaging should be applied
  - tide *station label*, (as defined in the [TIDE_STATION] section). Enter as many as needed for averaging.
- The last section labelled [OPTIONS] contains options used during tide zoning or tide averaging.
  - The limit value field contains the outage time in minutes before switching from the current station to the next priority station
  - final interval in seconds of tide data loaded into the track line.

For information on applying the data in a ZDF to survey lines see "CORRECT FOR TIDE" ON PAGE 169 in the User Guide.

GPS Tide format

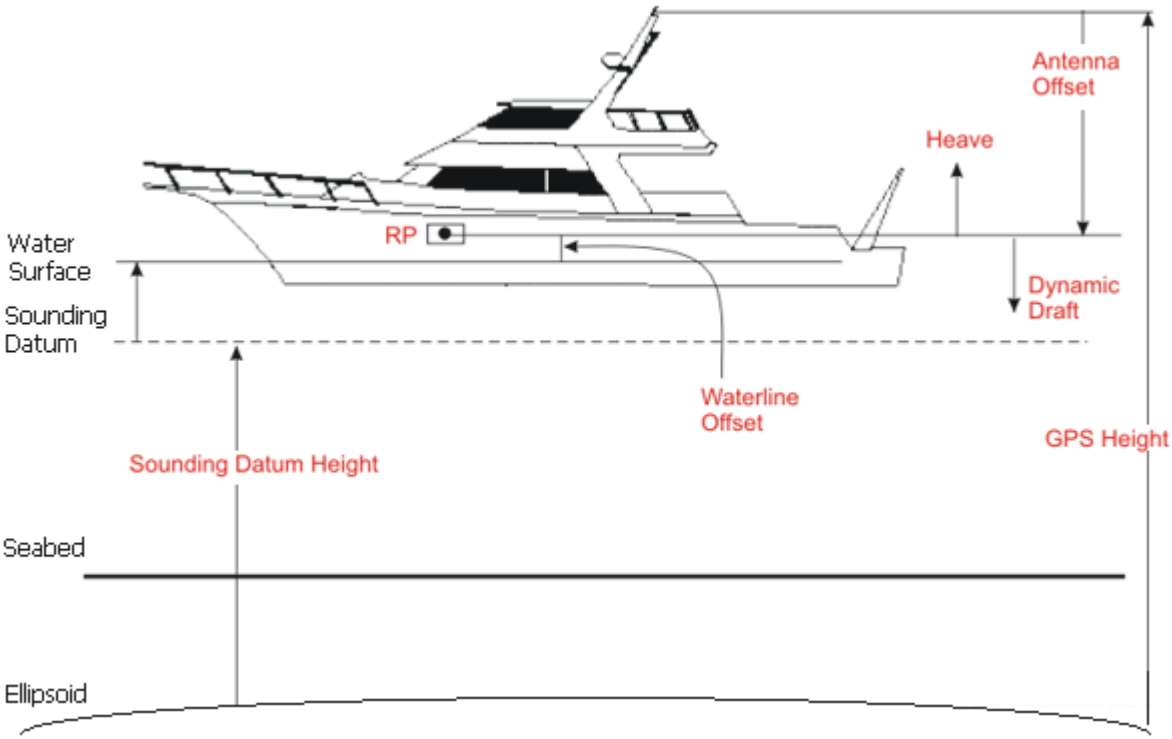
The Compute GPS (Global Positioning System) Tide function provides an alternative to normal tidal observation for reducing soundings to the sounding datum

It is based on the recorded GPS ellipsoid height of the navigation antenna and uses the recorded vessel motion, if available, to reduce the ellipsoid height measurements to water level.

The formula for calculating GPS tide is:

```
GPS Tide = GPS Height - Datum Height + Antenna Offset - Heave
+ Dynamic Draft + Height Offset - Waterline Offset
```

The image below demonstrates the values in the GPS tide formula. The arrows indicate the positive sense of the values.



This diagram and equation show the formula for GPS tide where GPS Height needs to be motion corrected. However, in some systems, GPS Height may already be motion corrected. Applying sensor values (such as dynamic heave) again will only distort the final GPS Tide. The Compute GPS Tide command lets you include or omit selected sensor values.

To compute GPS tide, see "COMPUTE GPS TIDE" ON PAGE 176 of the User Guide.

Sounding datum models

When applying the sounding datum height(s) (distance from the ellipsoid to the sounding datum) to a track line, these methods may be used:

- apply a single height to a track line, or,
- apply a selected surface layer, with a set elevation attribute, or,
- apply a gridded binary (.bin) file, or,
- apply a sounding datum model file in ASCII format, that contains a grid of sounding datum heights (with either Ground or Geographic coordinates).

Sounding datum heights can be applied for Computing GPS tide.

ASCII datum model files do not need to follow a specific format or have a specific extension. However, when an ASCII datum file is selected as the datum model, a `*.info` must also be opened to control the parsing of the ASCII data. (See "INFO FILE" ON PAGE 248.)

As well, a coordinate system must be selected to identify the position information in the ASCII file.

XYZ and BIN files used in the Compute GPS Tide function can be opened as background data. (See "OPEN DATA" ON PAGE 56.)

When an XYZ is opened in HIPS, the selection of *.info and coordinate system is saved in a file with the same name of the `.xyz` file, but with an "`_rxl`" suffix.

For example, if you select "`mydata.xyz`," then "`mydata.xyz_rxl`" will be created in the same directory as the original XYZ file. If you open the same `.xyz file` again, the information from this file is automatically used.

10

Support Files

In this chapter...

INFO file

Files with a INFO extension, such as the sample file below, contain the locations of the positional (Lat/Long) and other attributes in the ASCII format files.

Info files are used by HIPS to parse the information from the XYZ file when opening as background data (see "Open Datum model file" on page 57) or for such processes as Compute GPS Tide (see "Sounding datum models" on page 245).

"File structure" on page 248

"Position components" on page 250

"Additional Attributes" on page 252

File structure

Files in ASCII format can have their data arranged in fixed columns or in free format. An info file defines which parts of the ASCII file contain the data that is needed by HIPS and SIPS.

In XYZ files the data includes coordinates as well as a depth or height value. These numeric values, which can be called fields, or attributes, can be separated in each line by a space or other delimiter.

The first part of the file defines what character is used in the ASCII file as a delimiter or spacer between the fields of data in each line. Then it specifies the number of spaces or lines to be skipped between the beginning of the file and the start of the essential data, for example, if some lines contain a file header that is not needed for the HIPS process.

Next, the*.info file locates the essential position information used by HIPS when reading ASCII sounding datum files: the X, Y,Z coordinates. For fixed-width formatted files, this is done by defining the starting position and length of each column containing coordinate data. (This function is optional for delimited files.)

There are options to define a multiplier or offset if it is needed to transform a value when it is used by HIPS. For example, the value may be stored with an implied decimal point, and the multiplier will need to be applied to convert the ASCII value.

The sample file `xyzformatinformation.info` (displayed below) is included in the installed Template directory.

See "POSITION COMPONENTS" ON PAGE 250 for a description of its contents.

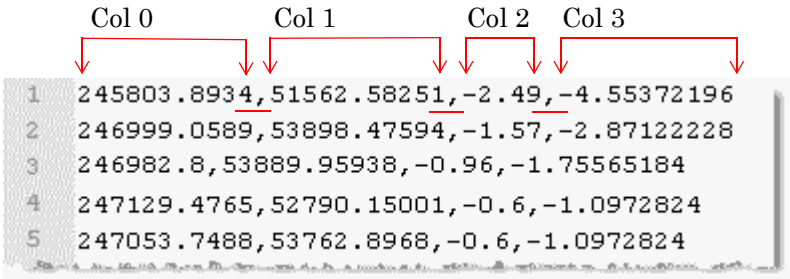
```
<?xml version="1.0" encoding="UTF-8"?>
<Format_Information version="1.2">
    <!-- The Delimiter value if this file has one -->
    <!-- Available Delimiter values: TAB or any letter/number/symbol -->
    <Delimiter value=" "/>
    <!-- The number of lines to skip at the head of the file -->
    <Skip_Lines value="0"/>
    <!-- The information required to decode the x, y, and z coordinates -->
    <Position>
        <!-- The multiplication value for the x coordinate -->
        <XMultiplier value="1.0"/>
        <!-- The multiplication value for the y coordinate -->
        <YMultiplier value="1.0"/>
        <!-- The multiplication value for the z coordinate -->
        <ZMultiplier value="1.0"/>
        <!-- The information required to decode the x coordinate -->
        <X_Coord>
            <!-- The zero based column index of the x coordinate -->
            <Col_Index value="0"/>
            <!-- The starting position of the x coordinate. -->
            <Start_Pos value="0"/>
            <!-- The width of the x coordinate. -->
            <Width value="0"/>
        </X_Coord>
        <!-- The information required to decode the y coordinate -->
        <Y_Coord>
            <!-- The zero based column index of the y coordinate -->
            <Col_Index value="1"/>
            <!-- The starting position of the y coordinate. -->
            <Start_Pos value="0"/>
            <!-- The width of the y coordinate. -->
            <Width value="0"/>
        </Y_Coord>
        <!-- The information required to decode the z coordinate -->
        <Z_Coord>
            <!-- The zero based column index of the z coordinate -->
            <Col_Index value="2"/>
            <!-- The starting position of the z coordinate. -->
            <Start_Pos value="0"/>
            <!-- The width of the z coordinate. -->
            <Width value="0"/>
        </Z_Coord>
    </Position>
```

```
    </Position>
    <!-- The information required to decode the attributes -->
    <Attributes>
        <!-- Available Data Types                          -->
          <!-- 4 byte real value (FLOAT)                  -->
          <!-- 8 byte real value (DOUBLE)                 -->
          <!-- 2 byte unsigned integer value (USHORT)     -->
          <!-- 2 byte signed integer value (SHORT)        -->
          <!-- 4 byte signed integer value (INT)          -->
          <!-- 4 byte unsigned integer value (UINT)       -->
          <!-- 1 byte unsigned char value (UCHAR)         -->
          <!-- Hour time value (TIME_HOUR)                -->
          <!-- Minute time value (TIME_MINUTE)            -->
          <!-- Second time value (TIME_SECOND)            -->
          <!-- Year date value (DATE_YEAR)                -->
          <!-- Month date value (DATE_MONTH)              -->
          <!-- Day date value (DATE_DAY)                  -->
          <!-- String value (STRING)                      -->
          <!-- Red portion of a colour (COLOUR_RED)       -->
          <!-- Green portion of a colour (COLOUR_GREEN)   -->
          <!-- Blue portion of a colour (COLOUR_BLUE)     -->
          <!-- Alpha portion of a colour (COLOUR_ALPHA)   -->

        <!-- Available Attribute Types  -->
          <!-- Depth TPE - Expecting values to be scaled to 95% CI (DEPTH_TPE)-->
    </Attributes>
</Format_Information>
```

Position components

In info files the delimiter can be a TAB character or any letter, number or symbol. In the XYZ example below, the values are separated by commas, so, in the info file the *Delimiter* would be expressed as ",".



Since this comma-delimited file does not have a heading, in the info file the number of *Skip Lines* is set to "0".

The Position information section first defines a multiplier for each coordinate. If no adjustments need to be made to the coordinate information then this value is set to "1".

Then the location of the values of the X, Y and Z coordinates are given.

- If the ASCII file is free form format, with fields of varying lengths separated by delimiters, a *Col_Index* value and a delimiter value must be specified.
- If the file is in fixed column format, the Start position and the width of the value field are defined.

Column Index

The column index value, *Col_Index*, assigns a column number to each field or attribute in the file. This number corresponds to the columns in the ASCII file. The columns are numbered from left to right, starting with 0. So X (Longitude) would be column 0, Y (Latitude) would be column 1 and Z (Primary Elevation) would be column 2.

In the example above, commas are the delimiters, and four columns (0-3) are identified (as highlighted in the example).

Start and Width

Alternatively, you can identify the data boundaries for each attribute column by using the *Start_Pos* and *Width* fields. These fields are also used for XYZ files in fixed column format.

Using the start and width values, each character in a line is numbered and the number of characters in each attribute value is used to identify a column. The characters in the line are numbered from left to right, starting with 0 and continued until the end of the line. For example a line with 6 attribute columns might have 70 characters.

In the example above, the first attribute column would have a start value of 0 with a width value of 11 because there are 11 characters in the X (Longitude) value. The second column (Y or Latitude) would start with the next consecutive number, making the start value 12. It has a width value of 11, taking up characters 12 to 23 in the array of numbers. The start value of the third column (elevation) would then be 24 and the width value would be 5.

- **x values**: Start_Pos = 0; Width = 11
- **y values**: Start_Pos = 12; Width = 11
- **z values**: Start_Pos = 24; Width = 5

The start position and width values can also be used to omit character columns of information from the attribute.

For example, to omit coordinate quadrant information where:

- the quadrant is the last character:

```
/-=-=-=-=-=|=-=-=-=-=-=-|-=-=-=
51-24-18.34N003-34-23.91E 10.23
```

  x values: Start_Pos = 0; Width = 11; **y values**: Start_Pos = 12; Width = 12

- the quadrant is the first character:

```
/-=-=-=-=-=|=-=-=-=-=-=-|-=-=-=
N51-24-18.33E003-34-23.91  9.81
```

  x values: Start_Pos = 1; Width = 11; **y values**: Start_Pos = 12; Width = 12

If data is in a delimited file where the coordinates have a fixed precision, including quadrant information, this can be extracted using the Start_Pos and Width relative to the delimiter character. For example, the following data,

```
/-=-=-=-=-|=-=-=-=-=-|-=-=-
51.394526N  3.555878E  0.00
```

could be read with an INFO file set with a delimiter of a single space character (' ') and the following values for the coordinates:

- **x values**: Col_Index = 1, Start_Pos = 0; Width = 9;
- **y values**: Col_Index = 0, Start_Pos = 0; Width = 9;
- **z values**: Col_Index = 2, Start_Pos = 0; Width = 0;

The width value does not include the quadrant letter, so only the non-letter characters of the coordinates will be read.

Additional Attributes

Additional attributes (that is, attributes available in the source file that are not XY position or Depth) can also be defined in the info file. The options for these attributes can be set in the additional attributes section.

The file named *XYZ++ Format Information File.info* in the installed Template directory contains an example of additional attributes.

The following additional attribute types are supported:

- String
- Number
- Date
- Time
- Uncertainty

- Colour

The following data types are supported, as can be seen in the sample file provided

```
<!-- The information required to decode the attributes -->
<Attributes>

 <!-- Available Data Types                       -->
 <!-- 4 byte real value (FLOAT)                  -->
 <!-- 8 byte real value (DOUBLE)                 -->
 <!-- 2 byte unsigned integer value (USHORT)     -->
 <!-- 2 byte signed integer value (SHORT)        -->
 <!-- 4 byte signed integer value (INT)          -->
 <!-- 4 byte unsigned integer value (UINT)       -->
 <!-- 1 byte unsigned char value (UCHAR)         -->
 <!-- Hour time value (TIME_HOUR)                -->
 <!-- Minute time value (TIME_MINUTE)            -->
 <!-- Second time value (TIME_SECOND)            -->
 <!-- Year date value (DATE_YEAR)                -->
 <!-- Month date value (DATE_MONTH)              -->
 <!-- Day date value (DATE_DAY)                  -->
 <!-- String value (STRING)                      -->
 <!-- Red portion of a colour (COLOUR_RED)       -->
 <!-- Green portion of a colour (COLOUR_GREEN)   -->
 <!-- Blue portion of a colour (COLOUR_BLUE)     -->
 <!-- Aplha portion of a colour (COLOUR_ALPHA)   -->
```

The positional (XYZ) attributes in an INFO file are predefined. However, when using an additional attribute with multiple data types, each data type must be defined by the user.

For example, the TIME and DATE attributes each have multiple pieces of data that make up their over-all values (hour/minute/second, day/month/year). Each of these pieces of data is a different data type and must be identified individually.

Similarly, to create a single attribute layer called “Colour” in the resulting CSAR file, each of the Red, Green, and Blue portions in the XYZ file are defined. The Alpha portion, which is the transparency value, is optional.

When defining each data type, you have the same options as defining a position attribute: column index value, or start and width values for a character array. The difference is that you must complete all fields for each data type as if it were an attribute itself.

For user defined time and date type attributes:
- if a delimiter value is defined, the Start_Pos is relative to the beginning of each column respectively; otherwise,
- if no delimiter is defined, the Start_Pos is relative to the beginning of the line.

The width is always the number of characters from the Start_Pos.

The following example shows TIME_HOUR and TIME_MINUTE, defined using Col_Index and a delimiter.

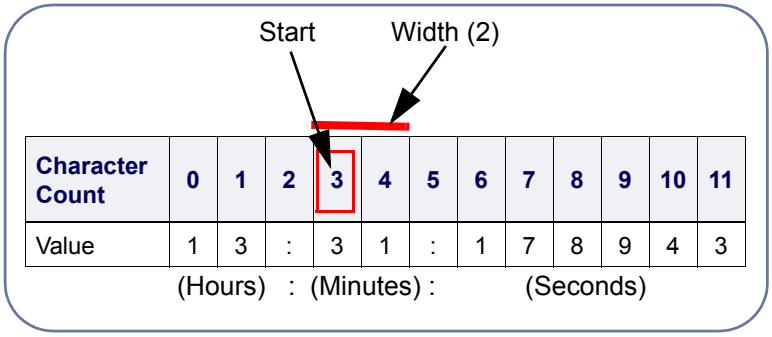
```
<Attribute>
    <!-- The name of the attribute. Spaces are not valid -->
    <Name value="Time"/>
    <!-- The zero based column index of the attribute -->
    <Col_Index value="6"/>
    <!-- The starting position of the attribute. -->
    <Start_Pos value="0"/>
    <!-- The width of the attribute. -->
    <Width value="2"/>
    <!-- The data type of the attribute -->
    <Data_Type value="TIME_HOUR"/>
</Attribute>
<Attribute>
    <!-- The name of the attribute. Spaces are not valid -->
    <Name value="Time"/>
    <!-- The zero based column index of the attribute -->
    <Col_Index value="6"/>
    <!-- The starting position of the attribute. -->
    <Start_Pos value="3"/>
    <!-- The width of the attribute. -->
    <Width value="2"/>
    <!-- The data type of the attribute -->
    <Data_Type value="TIME_MINUTE"/>
</Attribute>
```

In order for the Import tool to recognize which attribute the data type belongs with, you must identify the *Name value* of the attribute (“Time”) and the *Data_Type value* of the data (“TIME_HOUR”).

The same column number is referenced for each piece of data, but a start and width value have been specified for the characters of each piece of data within the column.

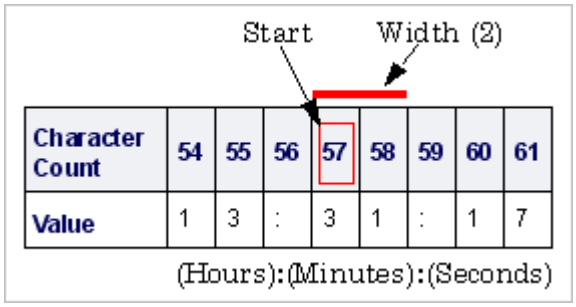
When specifying the number of characters this way, the numbering starts over at the beginning of each column.

- TIME_HOUR = Column 6, Start 0, Width 2
- TIME_MINUTE = Column 6, Start 3, Width 2
- TIME_SECOND = Column 6, Start 6, Width 6



Note that a character is used as a delimiter between each value within the column and included in the character count; however, it is not included in the value in the surface.

If there were no delimiter value specified in the INFO file, each piece of data would need to be identified using the character count of the entire line. For example, TIME_MINUTE may start at character 57 and have a width of 2. TIME_SECOND would then start at 60 with a width of 2.



See the *XYZ++ Format Information File.info* file for examples of how the other additional attributes are entered.

Invalid Attribute Values

If the data being imported contains invalid values (for example, points/nodes without elevation values), the info file can be used to catch these values and exclude them from the import. When the importer encounters an invalid value, it will skip that point/node and move on to the next one.

To use this function, you must list the values for each attribute that you want the importer to recognize as invalid. The following lines are used to list invalid values:

```
<Invalid_Values>
<Value value="-9999"/>
</Invalid_Values>
```

These lines would be added to the group of lines for each attribute. Multiple values can be listed within the opening and closing lines for the parameter. Below is an example of an invalid value specified for the Z attribute.

```
<!-- The information required to decode the z coordinate -->
  <Z_Coord>
     <!-- The zero based column index of the z coordinate -->
       <Col_Index value="2"/>
     <!-- The starting position of the z coordinate. -->
       <Start_Pos value="0"/>
     <!-- The width of the z coordinate. -->
       <Width value="0"/>
       <Invalid_Values>
           <Value value="-9999"/>
       </Invalid_Values>
  </Z_Coord>
```

If you use invalid values that have multiple representations (such as 0, 000 and 0.0), each representation must be reflected in the INFO file as the values are treated as strings.

Supported sonars

The file devicemodels.xml lists the sonar models supported by HIPS and SIPS. The sonar information page on the CARIS web site, (http://www.caris.com/tpu/idx_sonar.cfm) lists the sonar specifics that are included in the `devicemodels` file. The file is located in the system directory.

This file can be modified, but users are cautioned that because of the important role of the devicemodels.xml file particularly in computing TPU, this should only be done by advanced users. (Contact CARIS Customer Support for assistance.)

Illustrated below is a section of the devicemodels.xml file showing some of the data it contains.

Glossary

altitude

The height of the side scan sonar transducer above the sea floor. The transducer is typically mounted on a towed body or the surface vessel, but can also be mounted on an ROV or AUV.

Angle Varying Gain (AVG)

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

Attitude Data

Commonly used to refer to the orientation of the vessel in three dimensions as represented by the Gyro, Heave, Pitch, and Roll sensors. The Attitude Editor displays this data as well as other sensors that consists of simple “time vs. sensor value” data.

background data

Georeferenced vector and raster data providing visual context to the bathymetry and side scan data being processed. Examples include CARIS files, BSB raster charts, GeoTIFF orthophotos, S-57 electronic chart data.

backscatter

Intensity of sounding returns collected by swath multibeam sounding systems. Backscatter data creates an image of the sea floor that can indicate the bottom type.

BAG

The Bathymetric Attributed Grid (BAG) is an open-source binary file format for transferring gridded bathymetry and uncertainty values between software applications and agencies. The file contains five elements: metadata (in XML format), elevation, uncertainty, tracking, and certification. For more information on BAG, go to http://www.opennavsurf.org/.

beam

A single depth measurement is produced from a beam. Many beams make up a profile or a ping.

Beam Averaged

A single backscatter return value (intensity, dB) is logged with each beam, taken as an average intensity centred on the bottom detect of the entire time series to produce an image of the sea floor. The intensity for each beam can be directly georeferenced using the positioned beam footprint, but spatial resolution is lost by reducing the full time series to a single value.

Beam Pattern Correction

Beam pattern correction uniformly removes along-track banding inherent to the sonar beam. This effect is produced by each transducer uniquely.

Beam pattern correction relies on a user-generated beam pattern file to identify and remove this effect

cable out Length of cable (tow wire) deployed from a towpoint to the towfish. Used in the calculation of lay back.

CARIS file Now called CARIS map format, this records vector map data features such as lines, symbols, text, soundings, etc. It consists of a set of files.

CATZOC The zone of confidence (CATZOC) attribute. The CATZOC attribute is part of the S-57 Quality of Data (M_QUAL) object class. It indicates that data meets the minimum criteria for sea floor coverage, and position and depth accuracy as defined by the attribute classification structure. There are five CATZOC classifications: A1, A2, B, C, and D.

contacts Point and line features identified in side scan sonar data. Attributes describing the location, size, and description or the target are recorded. Image snapshots are also generated.

CSAR CARIS Spatial ARchive: a means of storing high-volume grid and point bathymetric data. Surfaces created in HIPS use CSAR format.

CUBE surface A Combined Uncertainty and Bathymetry Estimator (CUBE) surface contains multiple hypotheses representing potential depth variances on the sea floor. As soundings are propagated to nodes, based on distance and uncertainty, a hypothesis is developed. If the there is a significant variation of depth at a node, a new hypotheses is created. A node can contain more than one hypothesis.

d value Shows the confidence in the grain size assessment and sediment analysis completed by GeoCoder. Smaller values indicate more confidence in the assessment, with values below 0.5 being very good and values above 1.0 being potentially inconclusive. Appears when using Advanced Sediment Analysis.

day A day for which there is survey data. Expressed as a Julian day.

delta draft A “time vs. delta-draft” data structure that can be loaded into HIPS and used to model the dynamic squat and lift of the vessel. Also used to store the recorded depth of towfish and ROV/AUV mounted multibeam sonars.

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

disambiguation The process in CUBE of selecting one hypothesis over other hypotheses to represent the surface. Hypotheses can be selected or “nominated” by density, locale, locale and density, or by the nearest value to a previous CUBE Surface.

DpTPU Depth Total Propagated Uncertainty. The difference between the observed or computed depth value of a sounding and its true depth value (at a 95% confidence level).

ellipsoid The ellipsoid and datum to which logged positions in the survey refer must be stored in the HIPS vessel file, regardless of whether projection coordinates or geographic coordinates are stored in the logged file. Any ellipsoid may be used, so long as it is defined in the file referenced by the environment variable `uslXdatum`.

ENC Electronic Navigational Chart

field sheet A collection of data products, within a defined geographic area, derived from the processed bathymetry and side scan data. Field sheets were used in version of HIPS and SIPS prior to 9.0.

Example products included BASE surfaces, mosaics, tiles, selected soundings, and contours.

filtering The process of detecting outliers in the data and setting the status flags to Rejected. Data points are not removed from the HIPS/SIPS format.

frequency Number of acoustic waves produced per second (Hz).

gain A measure of increase in amplitude of a signal.

Gain Correction Gain Correction adjusts signal intensities using independent port and starboard Gain factor settings. Gain correction can also be applied uniformly to both sides.

gamma An advanced GeoCoder parameter indicating the spectral exponent of bottom relief.

Generic Data Parser A program for converting recorded ASCII data sources into the HIPS/SIPS format. New survey lines can be generated for single beam bathymetry, or the program can update a sensor within an existing HIPS or SIPS project.

GPS Tide Instantaneous water level height above the sounding datum derived from the observed GPS ellipsoid heights of the navigation antenna. Reduced for

	sensor offsets, vessel motion, known sounding datum height above the ellipsoid, and other parameters.
grain size	An advanced GeoCoder parameter indicating the size of a particle in phi.
GSF	Generic Sensor Format - a file format used for storing bathymetry data.
holiday	Formed when enough significant data gaps are clustered together within a specified radius.
HOB	The Hydrographic Object Binary (HOB) file is a spatially referenced dataset that is used to support the internal structure of S-57 data in CARIS products. The HOB file does not require a CARIS vector map and contains the point, line, and area geometry for hydrographic objects.
HTF	Hydrographic Transfer Format
HVF	HIPS Vessel File. A vessel configuration file in XML format that consists of a list of sensors with their physical and calibration measured offsets, plus any error values. These are applied to the observed data during processing. The HVF supersedes the Vessel Configuration File (VCF).
hypothesis	A representation of depth in a surface. The hypotheses that are displayed in the surface are selected or “nominated” through disambiguation. Hypotheses that were not selected remain as alternative hypotheses and can be nominated in the Subset Editor.
HzTPU	Horizontal Total Propagated Uncertainty. The difference between the observed or computed position value of a sounding and its true position value (at a 95% confidence level).
index contour	A contour line accentuated by a heavier line weight to distinguish it from intermediate contour lines. Index contours are usually shown as every fifth contour with their assigned values, to facilitate reading elevations.
interlaced backscatter	Scattering of the acoustic energy (or acoustic impedance) that occurs at the sediment-water interface where the sea floor reflects and scatters the incident acoustic energy.
Kirchhoff backscatter	Scattering of acoustic energy due to sea floor roughness.

LADS Laser Airborne Depth Sounder (LADS) is a LIDAR system developed by Tenix Corporation.

layback If a towed sensor is used during a survey, the position of that sensor can be calculated as a horizontal “layback” from the position of the towing vehicle. The sensor layback is computed from the tow wire length and sensor depth.

LIDAR Light Detection And Ranging. LIDAR uses laser technology to calculate bathymetry.

line A single pass of the survey vessel over some or all of the area being surveyed, during which time referenced sensor data is continuously collected. Often the line is approximately straight.

loss An advanced GeoCoder parameter measuring the ratio of the imagined wave number to the real wave number for the sediment.

Merge The process of calculating final positions and depths for soundings, based on all relevant inputs such as observed depths, navigation information, vessel dynamics such as gyro, heave, pitch and roll, and tide. Merging is carried out after these inputs are loaded and checked.

mosaic A geo-referenced raster image that is a composite of imagery on one or more survey lines.

object A point, line or area feature including its spatial information (geometry), feature attributes, and symbology. Also referred to as feature, or feature object.

outliers Unwanted data resulting, for example, from incorrect sea floor detection.

permeability An advanced GeoCoder parameter measuring the ability of a material (e.g. rock or sediment) to transmit fluids. A factor in the Biot model.

phi A unit-less number used to represent particle grain size in sediment analysis.

ping A single output pulse of a sonar system.

point cloud 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.

porosity An advanced GeoCoder parameter measuring the fraction of voids in a medium (e.g. rock or sediment), or the water space between grains of sediment. A percentage that is a factor in the Biot model.

profile / swath A set of soundings, approximately perpendicular to the ship's track, which is produced by the swath or sweep sonar over an instantaneous or very short time period.

product surface A shoal-biased surface. Data is down-sampled so that finer details of the sea floor are not visible, but the shoals are still maintained. Product surfaces can be used to create navigation products, such as ENCs.

project A survey area with data collected by one or more survey vessels over one or more days.

raw side scan Time indexed imagery profiles where the across-track scale is time in milliseconds.

roughness An advanced GeoCoder parameter measuring the spectral strength (cm^4) of bottom relief at wave number 1. Stronger returns would indicate a rougher bottom.

S-44 S-44 is the international standard developed and maintained by the International Hydrographic Organization (IHO) that sets the minimum standard for hydrographic surveys. The standard classifies surveys into survey orders (Special Order, Orders 1a, 1b, 2) based on an area's importance for safety of surface navigation.

S-57 S-57 is the international standard developed and maintained by the International Maritime Organization (IMO). It is used by hydrographic data producers to describe how real-world entities are stored digitally for transfer among different computer platforms between the various hydrographic offices.

session A record of all the data (survey line names, background raster images, surfaces and their properties) that are open in the HIPS and SIPS interface when the session is saved. This enables you to re-open an integrated data set from a previous working session.

SHOALS	Scanning Hydrographic Airborne LIDAR Survey (SHOALS) is a LIDAR system owned and operated by the Joint Airborne LADAR Bathymetry Center of Expertise.
side scan	The emission of sonar pulses in a wide angle perpendicular to the sea floor to port and starboard that are received and logged as a time series of intensities in order to form an image. This data is logged separately from the bathymetry data. A swath multibeam system forms two additional receive beams in addition to the bathymetry profile.
slant range	Raw travel time used to compute sounding depth (accompanied by receive angle).
Slant Range Corrected Side Scan	Time-indexed imagery profiles generated from raw side scan data using the measured altitude of the transducer. The across-track scale is distance in metres.
sounding	A measured depth of water.
Sounding datum height	Distance from the ellipsoid to the sounding datum.
Sound Velocity Correction	The process of applying rigorous refraction corrections to the raw travel time/angle bathymetry data using recorded sound velocity profiles.
Status Flag	The indicator of the acceptance or rejection of a data point during cleaning.
subset	A rectangular area which encloses some or all of a survey project shown in the Display window. It is used during sounding cleaning to limit the number and extent of soundings being processed during one session.
superselected	A superselected feature is a subset of a selection of multiple features. When a number of features are selected, but the attributes or properties of only one feature can be viewed or acted upon, it is the superselected feature which will be affected. In HIPS and SIPS the superselected feature is coloured light blue in the Display window and other features in the selection are coloured red. In the Selection window, the superselected feature is highlighted.
surface cleaning	A statistical process of detecting bathymetric outliers within an area. A least square polynomial regression algorithm is used.

tiles

A representation of the sea floor consisting of interlocking tiles of varying sizes. Tiles can make large data sets more manageable by subdividing them by user-defined criteria.

Tile positions are defined by a Morton Number scheme. Tiles can carry a variety of attributes generated from the data within the area of the tile.

time series

A series of intensity values (dB) logged with each beam that is added based on the time and range of detection point to assemble a trace similar to a side scan image. Each individual time series is associated with a portion of the bathymetric profile.

This detection method allows for a higher across-track spatial resolution of sea floor features (with spatial frequencies higher than the beam spacing) since a portion of the time series is preserved for each beam.

Time-Varying Gain (TVG)

The attenuation of acoustic energy that occurs due to absorption and spreading as the sonar beam travels back to the receiver, resulting in a non-uniform gain across the swath. Since signal returns are received over a predictable and constant time period, a correction can be used to adjust the signal intensity by applying a non-uniform, time dependent gain.

TIN

A triangulated irregular network (TIN) is a digital terrain model which consists of a series of triangles with the data points as vertices. A TIN uses the actual elevation for each data point.

tortuosity

A unit-less advanced GeoCoder parameter indicating the characteristic of flow through porous media, or twisting of the pore space. This is a factor in the Biot model, and smaller numbers (1 is the minimum value) indicate a straighter path of potential flow, while higher numbers would have a more twisted path.

total backscatter

Combination of backscatter that occurs due to the interface change (Interface Backscatter), sediment heterogeneities (Volume Backscatter), and roughness of the sediment (Kirchhoff Backscatter).

TPU

Total Propagated Uncertainty. TPU is derived from a combination of all individual error sources and is used to calculate horizontal and vertical uncertainties for soundings.

vessel

A survey platform, whether it be a ship, a towfish, an ROV/AUV, or an aircraft.

velocity

A unit-less advanced GeoCoder parameter measuring the ratio of sediment sound speed to water sound speed.

volume backscatter

Scattering of the transmitted acoustic energy into the sea floor by heterogeneities in the sediment structure.

WKT Well Known Text - a text mark-up language for representing vector geometry objects on a map, spatial reference systems of spatial objects and transformations between spatial reference systems.

Index

Symbols

Numerics

A

B

C

D

E

F

G

H

I

K

L

M

N

O

P

Q

R

S

T

U

V

W

X

Z