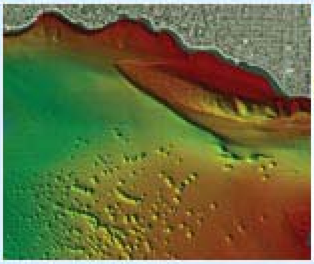
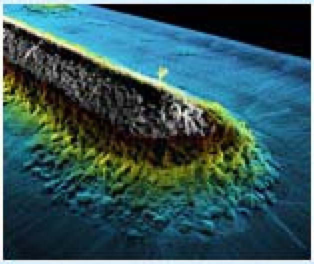
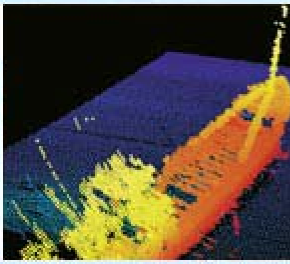
CARIS HIPS and SIPS 8.1

User Guide







THE MARINE GIS EXPERTS

Trademarks

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

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

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

Acknowledgements

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

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

Cover images (from left to right):

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

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

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

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

Phone: 1 (703) 299-9712
Fax: 1 (703) 299-9715
Email: carisusa@caris.com

CARIS bv
Mgr. van Oorschotstraat 13
PO Box 47 5473 ZG
Heeswijk, NETHERLANDS

Phone: +31 (0)413 296 010
Fax: +31 (0)413 296 012
Email: sales@caris.nl
Assistance: support@caris.nl

CARIS Asia Pacific
Level 3 Shell House, 172 North Terrace
Adelaide SA 5000
Australia

Phone: +61.(0).8.8231.9474
Email: info@caris.com

August 2013

Preface

This guide describes the workflow for processing data, using the multibeam to CUBE surface flow.

- "Workflow in HIPS and SIPS" on page 15
- "Create Vessel File" on page 25
- "Create a New Project" on page 57
- " Convert Data" on page 77
- "Sound Velocity Correction" on page 133
- "Correct for Tide" on page 143
- "Compute TPU" on page 153
- "Merge" on page 167
- "Create Field Sheets" on page 173
- "Create BASE Surfaces" on page 187
- "CUBE Processing" on page 247
- "Data QC" on page 269
- "Statistical Surface Cleaning" on page 311
- "Cleaning Swath Data" on page 323
- "Process Data in Subsets" on page 337
- "Create Product Surfaces" on page 355
- "Process Imagery Data" on page 411
- "Process Water Column Data" on page 459
- "Create Publications" on page 475
- "Export Data" on page 479

For processing of single beam data, please see "Process Single Beam Data" on page 89 in the Editors Guide.

For processing of LIDAR data, please see "Process LIDAR Data" on page 27 of the Editors Guide.

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

Navitronics format conversion

Hawkeye waveform viewer

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

Table of Contents

1 **Workflow in HIPS and SIPS 15**

Workflow diagram 16
Overview of workflow stages 17
HIPS Vessel Files 19
Working with Projects 20
Session Files 21
Convert Survey Data 23
Generic Data Parser 24

2 **Create Vessel File 25**

Vessel Files 26
Create a New HVF 27
Open the Vessel Wizard 27
Vessel Information 28
Type of Survey 28
Multibeam 29
Multi-Transducer 30
Motion Sensors 32
Configuration Options 32
Sound Velocity Corrections 33
Create Vessel Shape Outline 35
Sensor Configuration 38
Active sensors 38
Time Stamp 39
Time Correction 40
Sensors 41
Transducer 41
Navigation 42
Gyro 43
Heave 43
Pitch 44
Roll 45
Dynamic Draft 46
SVP 47
Sweep 48
Towed Sensors 49
Waterline Height 50
Total Propagated Uncertainty 51
Vessel Coordinate System 53
Default Ellipsoid 55

3 Create a New Project . . . 57

Define New Project . . . 58
New Project (Step 1) . . . 58
New Project (Step 2) . . . 60
New Project (Step 3) . . . 61
New Project (Step 4) . . . 62
Open Projects . . . 64
Multiple Data Paths . . . 64
Open specific data . . . 66
View Project Properties . . . 67
View Vessel Properties . . . 70
Set Line Properties . . . 71
Rename Day and Line Folders . . . 73
Archive Project . . . 74

4 Convert Data . . . 77

Converting Files to HIPS/SIPS Format . . . 78
Formats that can be converted to HIPS and SIPS: . . . 79
Select Data Format . . . 81
Select Files for Conversion . . . 82
Import Options . . . 83
Select Project, Vessel and Day . . . 86
Set Navigation Coordinate System . . . 88
Select Filters . . . 90
Advanced Filtering . . . 91
Enter Data Parameters . . . 93
Convert Data . . . 94
Options for Specific Formats . . . 96
Atlas . . . 97
Chirpscan3D . . . 98
CMAX . . . 98
Coda . . . 98
EdgeTech . . . 99
EIVA . . . 101
Elac . . . 102
Furuno . . . 103
GeoAcoustics . . . 104
GSF . . . 107
Hawkeye . . . 108
Hypack . . . 109
Imagenex . . . 112
LADS . . . 113
LAS . . . 114
MarineSonics . . . 114
Navitronics . . . 114
Qmips . . . 115
Teledyne Reson PDS . . . 116
PDS data . . . 116

S7K data . . . 118
SAS . . . 119
Scripps . . . 119
SDF . . . 120
Seabeam . . . 121
Seafalcon . . . 121
SEGY . . . 121
SHOALS . . . 122
Simrad . . . 122
Spawar . . . 124
SWATHPlus . . . 124
Teledyne . . . 124
UNB . . . 124
Winfrog . . . 124
XTF . . . 126
Multibeam Filtering . . . 128
Step 8: Convert Side Scan data . . . 129
Step 9: Convert Layback . . . 131

5 **Sound Velocity Correction . . . 133**
Sound Velocity Processing . . . 134
Profile selection method . . . 134
Sound Velocity Correction . . . 136
Query Applied SVC Profiles . . . 140

6 **Correct for Tide . . . 143**
Load Tide . . . 144
Load Tide status . . . 145
Tide Files . . . 146
Tide Zone Files . . . 146
Display tide zones . . . 147
Tide Editor . . . 148
GPS Tide . . . 149

7 **Compute TPU . . . 153**
Total Propagated Uncertainty . . . 154
Compute TPU . . . 156
TPU Filtering . . . 159
Apply Filter . . . 161
Load Error Data . . . 163

8 **Merge . . . 167**
Merge Process . . . 168
Apply Merge . . . 170
Delta Draft . . . 172

9 Create Field Sheets 173
Create a Field Sheet. 174
Open and Close Field Sheets. 179
Set as active field sheet 179
Turn off field sheet 180
Set Field Sheet Properties 181
Field Sheet Projection Grid 182
Annotations 184

10 Create BASE Surfaces. 187
Overview. 188
Four types of BASE surfaces 189
Uncertainty Weight. 189
Swath Angle Weight. 190
Range weighting 190
Shoalest Depth True Position 191
CUBE 191
Create a BASE Surface 193
BASE Surface (Step 1) 193
BASE Surface (Step 2) 195
BASE Surface (Step 3) - Swath Angle Weighting 197
BASE surface (Step 3) - Uncertainty Weighting 199
BASE surface (Step 3) - Shoalest Depth True Position 201
View BASE Surfaces 203
Properties of BASE Surfaces 204
View Surface Properties 204
Attribute Layer Properties 206
Bounding Polygon Properties 213
Vertical Reference System 215
Vertical Reference System Editor 215
Editing the Vertical Reference database 217
Deleting entries 217
BASE Surface Commands 218
Open 218
Delete. 218
Add to BASE Surface. 218
Remove From BASE Surface 219
Add Layer. 219
Recompute surface 221
Interpolate BASE Surface 222
Finalize BASE Surface 224
Combine. 226
Difference Surfaces 229
Compute Surface Statistics 232
Export Histogram to Image 234
Export Statistics to ASCII 237
BASE Surface QC Report 238
Group multiple surfaces 242

Surface Filtering 244

11 **CUBE Processing. 247**
Overview. 248
Generating CUBE Surfaces 251
BASE Surface (Step 1) 251
BASE Surface (Step 2) 251
BASE Surface (Step 3) - CUBE. 253
BASE Surface (Step 4) - CUBE 254
Advanced Options for CUBE surface. 255
Hypothesis Editing 259
Hypotheses views 261
Wire frame Hypotheses 261
Fixed Size Hypotheses 262
Nominate Alternative Hypotheses 263
Hypotheses Cleaning 266
Updating a CUBE Surface 267

12 **Data QC. 269**
Workflow. 270
Tools for examining and editing data 272
Query Data 273
View Line Data. 273
Detailed Line Query 274
Query Data in Editors. 274
Line Report 275
Rejecting and Accepting Data 277
Examine Navigation Data 278
Speed Jump. 279
Time Jump 279
Rejecting Navigation Data 281
Line Interpolation 281
Shift Navigation Data 284
Attitude data 285
Query data 285
Filter Attitude Data 286
Attitude Filter 286
Delayed Heave 289
Load Attitude/Navigation Data 291
Classify Lines 293
Reject Line 294
View Rejected Soundings. 295
Critical Soundings. 297
Designate Critical Soundings from a Surface. 298
Find and Designate Soundings in HIPS Editors 300
View Critical Soundings in HIPS Editors 301
View Critical Soundings in the Display window 302

Protect Critical Soundings 304
Restart Cleaning 305
Quality Control Reports 307
QC Report - Step 1 307
QC Report - Step 2 308
QC Report - Step 3 309
Quality Control Statistics 310

13 **Statistical Surface Cleaning 311**
Process. 312
Polynomial Surface Types 313
Creating Tiles and Cleaning Data. 314
Tile Histogram 319
Query data in tiles. 320
Select and View Tile Data 320
Query Soundings in Tiles. 320
Query Tile Display 321

14 **Cleaning Swath Data 323**
Swath Cleaning 324
Automatic Filtering 325
Apply Filters 327
Manual Cleaning 328
Tools 328
Auto Cursor Mode 328
Find Sounding 328
Rejecting Data 330
Reject Swaths 330
Query Data 331
Update Backscatter 332
Flag Designated Soundings 333
Correction for Refraction Artifacts 334

15 **Process Data in Subsets 337**
Data Cleaning Workflow 338
Subset Cleaning 339
Reject. 339
Accept 340
Outstanding 340
Examined. 340
Query 340
Designate Soundings 342
Track Progress with Subset Tiles 343
Subset tile properties 345
Tracking Layer Options 347
Tracking Cleaning Status 349

Subsets and BASE Surfaces 350
.Open BASE surface in Subset Editor 350
Surface filter. 352

16 **Create Product Surfaces 355**
Product Surfaces 356
Process 356
Defocusing 356
Create a Product Surface 358
Contours 361
Create Contours 361
Contour Labelling. 365
Masking Lines Behind Contour Labels 366
Profiles 367
Digitize a Profile. 367
Create Profile by Superselection 369
Setting Profile Options 370
Import Profile 373
Edit the Profile line. 375
Line Editing Options 376
Remove Profiles. 377
Export Profiles 377
Export Profile to ASCII 377
Export Profile to Image 380
Sounding Selection. 383
Select Height Source 383
Selection Criteria 384
Radius 385
Overplot Removal 387
Output Selection 388
Output to CARIS map 388
Output to HOB file 389
Enable Filter 390
HOB S-57 Options 391
Colour Options for Soundings and Contours 394
Tiling a field sheet. 396
Tiling methods 397
Create Tiles 397
Create Bins 401
Create QTC Tiles. 402
Open and Close Tiles 404
Set Tile Display Properties. 404
View Tile Attributes 406
Tile Display Query 407
Depth Range Files 409

17 **Process Imagery Data 411**
Overview 412
Workflow for Processing Imagery Data 413
Processing Engines 414
Geocoder Engine 414
SIPS Engine 414
Geocoder workflow for multibeam data 415
Open Mosaic Editor 416
Create GeoBars 417
Naming GeoBaRs 418
Create GeoBaR with the Geocoder Engine 419
Create GeoBaRs with the SIPS Engine 422
Beam Pattern Correction 426
Create Beam Pattern file 427
Despeckle 428
Edit GeoBaRs 429
Image Classification 431
Delete GeoBaR 433
Create Mosaic from GeoBaRs 434
Delete Mosaic 437
Edit Mosaics 438
Edit Mosaic Components 440
Interpolate Mosaic 442
Conflict Resolution 444
Ways to Resolve Conflict 445
Sediment Analysis 446
Set Sediment Analysis options 446
Auto Analyze 447
Analyze selected patch 448
Advanced Mode 450
Grain Size table 451
Model Parameters 451
Sediment Analysis Graph 453
View Sediment Patches 454
View Sediment Analysis Overlay 455
Contacts 457

18 **Process Water Column Data 459**
Water Column Data in HIPS 460
Projects 460
Processing Workflow for Water Column Data 461
Water Column Data in Swath Editor 462
Display water column data 463
Automatic scrolling 464
Water Column Toolbar 465
WCI Controls in Swath Editor 466
Stacked view 466
Add WC Bathymetry to Project 468

WCI Data in Subset Editor . . . 469
Display options. . . . 471
Survey lines 472
Additional Bathymetry. . . . 473

19 **Create Publications 475**
Plot Composer 476

20 **Export Data. . . . 479**
Export to S-57. . . . 480
Export Surface Metadata 481
Export Surface to BAG 482
Export Settings. . . . 483
Save and Load Templates. . . . 484
Export Surface to ESRI ASCII Grid 486
Export Selection to DXF 487
Export Selection to Formatted ASCII 489
Export Selection to GML. . . . 490
Formats Exported 492
Export Wizard 494
Step 1: Select Format 494
Step 2: Select Files 495
BAG to ASCII 497
BASE Surface To ASCII. . . . 498
BASE Surface to ASCII (Step 3) 498
Select Export Units 500
Surface To Image 501
Attribute and Image Options 501
Image Output Options 502
Contacts 504
Contacts (Step 3). . . . 504
Contacts (Step 4). . . . 505
GeoBaR to ASCII 507
GeoBaR to ASCII Output Options 507
Select Export Units 508
GeoBaR to Image. . . . 510
GSF to CARIS Map 511
Select GSF Files 511
Export Soundings and Track lines 511
CARIS Map Options. . . . 513
GSF to HOB 514
GSF Files to Export 514
Export Soundings and Track lines 514
GSF to HOB (Step 4). . . . 515
HIPS Tide to ASCII. . . . 516
HIPS Tide to ASCII (Step 3) 516
HIPS To ASCII 517

HIPS to ASCII (Step 3) 517
Select Export Units 519
Coordinate System 520
HIPS to ASCII - Step 6. 521
HIPS To CARIS Map 523
HIPS to CARIS Map (Step 3). 523
HIPS to CARIS Map (Step 4). 525
HIPS to CARIS Map - Options 526
HIPS to FAU 527
HIPS To GSF 528
Output Folder 528
HIPS To HOB 529
HIPS to HOB (Step 3) 529
HIPS to HOB (Step 4) 530
HIPS To HTF 531
Select Input Data 531
HIPS to HTF - Output options 532
HIPS to HTF - Output options page 2 533
HIPS to HTF - Attributes Options. 534
Mosaic to ASCII 535
Mosaic to ASCII Output Options 535
Select Export Units 536
Mosaic To Image 538
Sediment Analysis to ASCII 539
SIPS To UNISIPS. 540
SIPS TO UNISIPS (Step 3) 540
Coordinate System 541
Export 542

1

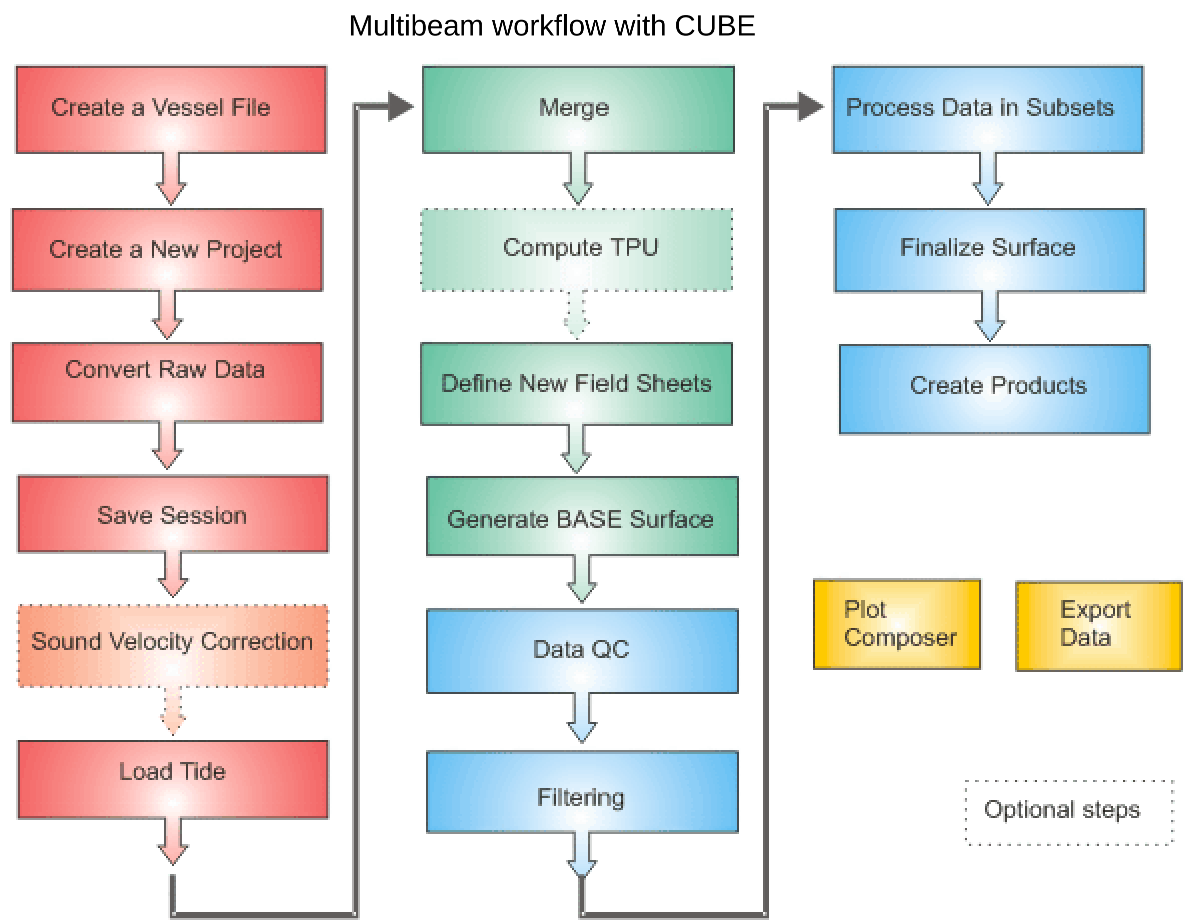
Workflow in HIPS and SIPS

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

In this chapter...

Workflow diagram 16
HIPS Vessel Files 19
Working with Projects 20
Session Files 21

Workflow diagram



"Overview of workflow stages" on page 17

Other workflows (in the HIPS and SIPS Editors guide) :

• "Side Scan Editor workflow" on page 50

• "Single Beam Workflow" on page 90

• "LIDAR Data in HIPS" on page 28

In this User Guide:

• "Process Imagery Data" on page 411 (backscatter and sidescan imagery workflow)

• "Processing Workflow for Water Column Data" on page 461

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

The order in which certain functions are performed is important, however, many of the functions can be automated with the Batch Processor (see "Batch Processing" on page 95).

Overview of workflow stages

- Create a Vessel File: Create or edit vessel information on sensor locations and uncertainties. See "Create a New HVF" on page 27
- Create a new project: Set up the Project - Vessel - Day data structure. See "Define New Project" on page 58
- Convert raw data: Convert data into HIPS data format using automated process. See " Convert Data" on page 77
- Save session: Save the current workspace (data and current view). See "Session Files" on page 21
- [Optional] Sound Velocity Correction (SVC): Load and edit sound velocity profiles and apply the correction. See "Sound Velocity Correction" on page 136
- Load tide: Load tide data from one or more tide stations to correct for tide variance. See "Load Tide" on page 144
- Merge: Combine vertical and horizontal information to produce geo-referenced data. If changes are later made to the navigation and/or motion data during the QC process, the Merge process is re-applied.
- Compute Total Propagated Uncertainty (TPU): Use uncertainty values entered in the HVF to compute the total propagated error of each individual sounding. See "Compute TPU" on page 156
- Define new field sheets: Organize dataset into manageable areas with defined locations and map projection. See "Create a Field Sheet" on page 174
- Generate a Bathymetry Associated with Statistical Error (BASE) Surface: Either a Swath Angle, Uncertainty or CUBE surface from data contained in field sheet. See "Create a BASE Surface" on page 193
- Generate a Combined Uncertainty and Bathymetry Estimator (CUBE) Surface: Use merged data to produce a CUBE surface.
- Data QC: Examine and edit sensor data, such as navigation, gyro, heave, if problems have been identified in the BASE (CUBE) surface.

- Filtering: Use automated or manual tools to filter soundings using swath geometry and/or according to IHO survey order accuracies. Filter water column data for editing in Subset Editor.
- Process Data in Subsets: Using hypothesis editing, validate the CUBE surface and edit geo-referenced soundings. Add bathymetry from edited water column data.
- Finalize BASE Surface: Update the validated CUBE Surface after data has been edited, and ensure designated soundings are carried through to bathymetric products.
- [Optional] Create Product Surface: Produce a generalized product surface from the BASE Surface, and add:
  - Contours: Use automated functions with either the BASE Surface or product surface to output contours. (Contours can also be generated manually from tiles.)
  - Sounding Selection: Use a height source for selection of a representative sounding set.
- Plot Composer: publish HIPS and SIPS data and product surfaces.
- Export Data: Soundings and surfaces can be exported to various formats for data transfer (e.g., S-57).
- [Optional] Process Imagery Data: backscatter or sidescan data can be processed in mosaics using Mosaic Editor.

HIPS Vessel Files

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

The HVF can be created or modified through the HIPS Vessel Editor. The Vessel Editor is a separate application that is launched from the HIPS interface. See "Create a New HVF" on page 27.

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

Working with Projects

Every project in HIPS and SIPS is organized in a Project/Vessel/Day/Line folder hierarchy. New projects are created with a wizard, which generates a HIPS Project File (*.hips).

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

A project can be created with new Project/Vessel/Day folders, or new Vessel/Day folders can be added to an existing project. Day and Line folders can be renamed, or taken out of the active project or deleted outright. (See "Define New Project" on page 58.)

Projects can also be archived. (See "Archive Project" on page 74.)

All projects must contain a vessel file. (See "Create a New HVF" on page 27). If your raw data files are set up in a Project/Vessel/Day/Line structure and a HIPS Vessel File already exists for that data, you can use the Conversion wizard to create the HIPS directory structure. (See "Converting Files to HIPS/SIPS Format" on page 78 and "Rename Day and Line Folders" on page 73 for more information.)

You can be connected to more than one project repository at one time, using the commands from the right-click menu in either the New Project or Open Project dialog box. See "Multiple Data Paths" on page 64.

Session Files

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

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

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

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

Save a session

Menu	File > Save Session/ Save Session As

To save a session

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

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

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

The Session file is saved.

Open a session

Menu	File > Open Session

Open one or more saved sessions.

1. Select the Open Session command.

The Open Session File dialog box is displayed.

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

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

Close a session

Menu	File > Close Session

1. Select the Close Session command.

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

2. Click **OK**.

Convert Survey Data

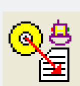
HIPS and SIPS files are created from survey data using the Conversion Wizard. The Conversion Wizard is a separate application that is launched from the HIPS and SIPS interface.

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

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

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

> You must organize your raw data files in a HIPS and SIPS Project/Vessel/Day structure to convert complete Project, Vessel and/or Day data files.

Menu	Import > Conversion Wizard
Tool	

1. Select a Conversion Wizard command.

The Conversion Wizard is displayed.

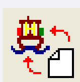
For more information on how to use the Conversion Wizard, see “Converting Files to HIPS/SIPS Format” on page 78.

Generic Data Parser

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

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

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

Menu	Import > Generic Data Parser
Tool	

1. Select the Generic Data Parser command.

The Generic Data Parser is displayed.

For more information see GENERIC DATA PARSER, in the Tools guide.

2

Create Vessel File

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

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

In this chapter...

Vessel Files 26
Create a New HVF 27
Create Vessel Shape Outline 35
Sensor Configuration 38
Sensors 41
Vessel Coordinate System 53

Vessel Files

The Vessel file (HVF) in HIPS and SIPS describes the location and calibration of sensor equipment installed on the survey vessel. Written in Extensible Mark-up Language (XML), it defines the offsets and any associated error estimates for each of the sensors.

For a description of the coordinate system upon which the vessel configuration is based, see "Vessel Coordinate System" on page 53.

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

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

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

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

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

Create a New HVF

A vessel file is created in HIPS Vessel Editor, a separate application launched from the HIPS and SIPS interface. (For a description of the Vessel Editor and its properties, see "Vessel Editor Interface" on page 172.)

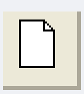
There are four main steps to creating and configuring a HIPS Vessel file in Vessel Editor:

- Open the HIPS and SIPS Vessel wizard to enter the parameters for the vessel file, based on the type of data being surveyed. (See "Open the Vessel Wizard" on page 27.)
- Enter sensor position data using a 3D outline for the vessel. (See "Create Vessel Shape Outline" on page 35.)
- Enter additional information such as the ellipsoid used in the survey. (See "Default Ellipsoid" on page 55.)
- Enter and edit sensor configuration data. (See "Sensor Configuration" on page 38.)

Open the Vessel Wizard

1. Open Vessel Editor from the HIPS and SIPS interface.

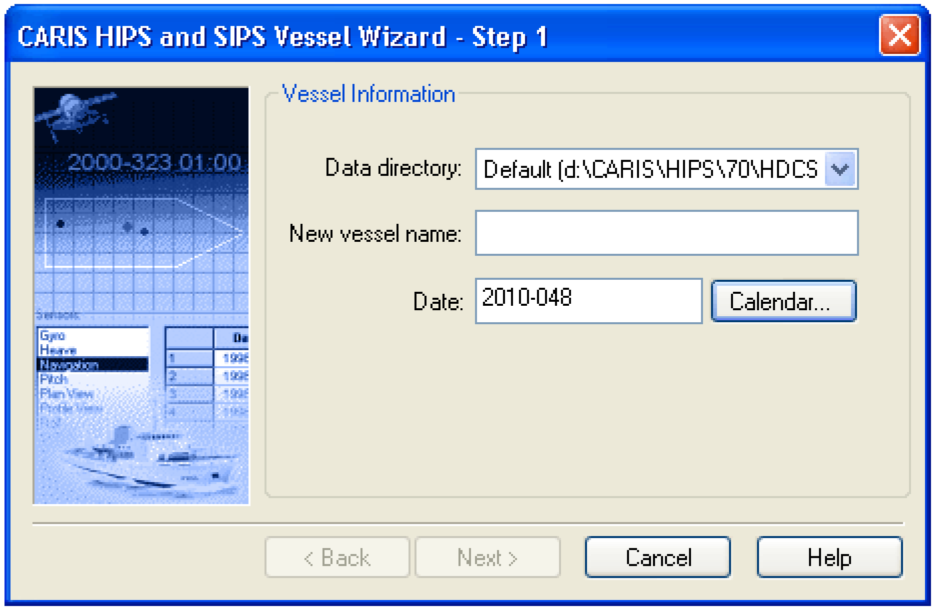
In Vessel Editor:

Menu	File > New
Tool	

2. Select the New Vessel File command.

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

Vessel Information

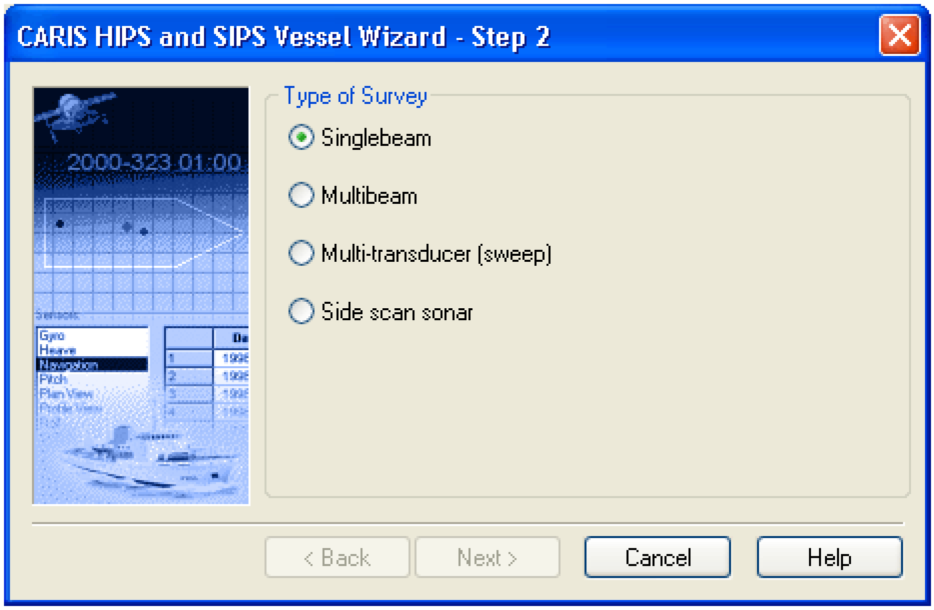


The HVF is saved to the \VesselConfig folder in your data directory.

1. Select the directory holding your data, from the drop-down list.
2. Type a name to identify the vessel used in the survey. (The name must not include any spaces.)
3. Click the **Calendar** button to insert the year and day the survey began. The Vessel Date must be before the start of your data collection.
4. Click **Next** to open the next dialog box:

Type of Survey

This dialog box prompts you to enter the sonar type used in the survey.



There are four sonar types:

- *Singlebeam*
- *Multibeam*
- *Multi-transducer*
- *Side Scan Sonar*

1. Select a sonar type by clicking the appropriate check box.
2. Click **Next** to go to the next step in the Vessel Wizard,

    or

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

(One or more dialog boxes will follow depending on the type of sonar selected in this dialog box.)

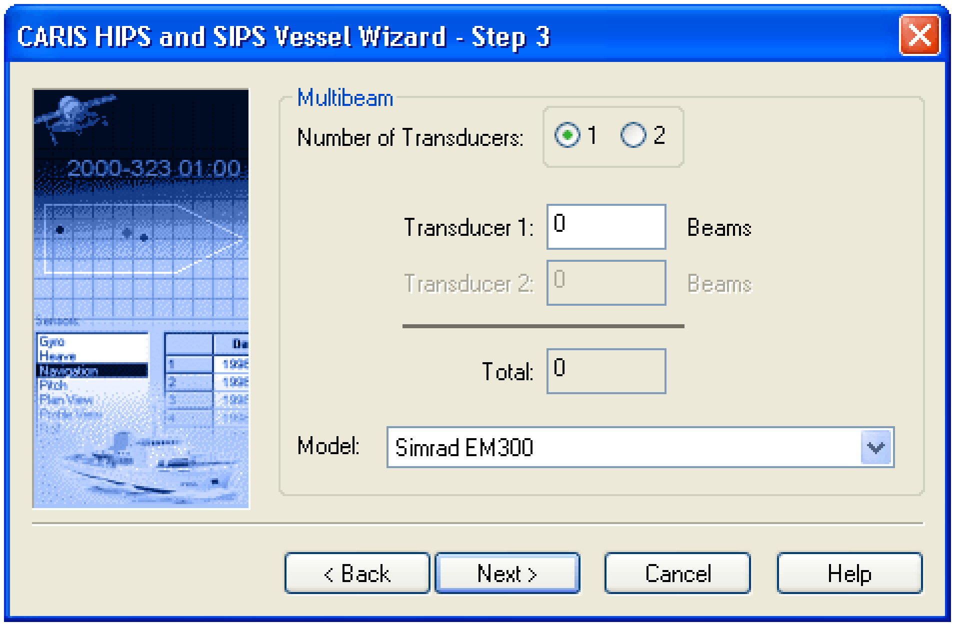
If you selected Singlebeam, go to: "Motion Sensors" on page 32

If you selected Multibeam, go to: "Multibeam" on page 29

If you selected Multi-transducer, go to: "Multi-Transducer" on page 30

Multibeam

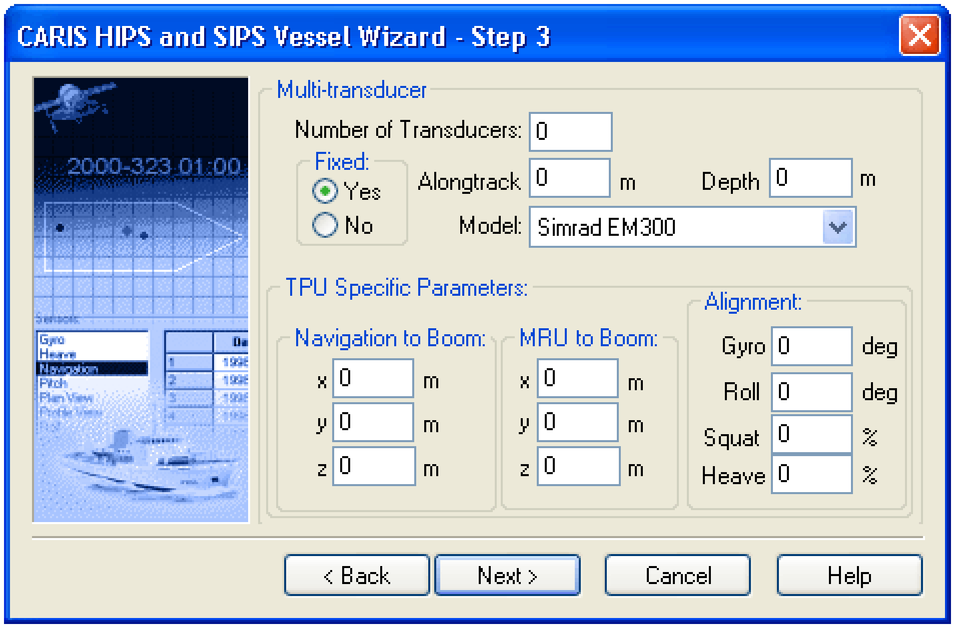
If you selected Multibeam in the Step 2 dialog box, Step 3 will ask you for information on the sonar model and number of transducers.



1. Select the appropriate check box to indicate that one or two transducers were used in the survey.
2. Enter the number of beams in each transducer.
3. Select the sonar model from the drop-down list.
4. Click **Next** to continue to "Motion Sensors" on page 32

Multi-Transducer

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



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

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

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

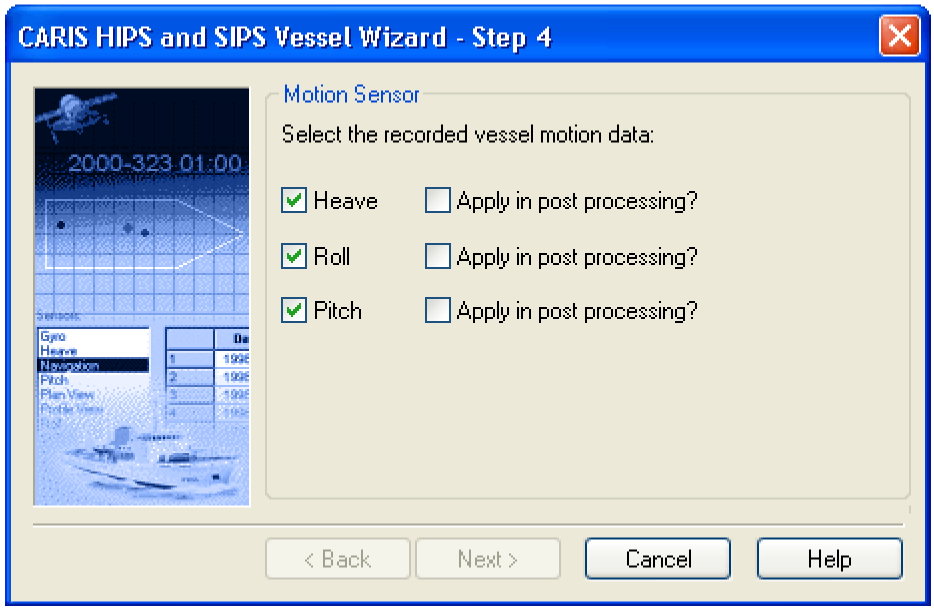
The *Roll* and *Gyro* misalignment values are needed for fixed-boom system. However, for a decoupled boom, only a *Gyro* misalignment value is needed.

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

8. Type a degree offset in the *Roll* and *Gyro* fields.
9. Type a percentage value in the *Squat* and *Heave* fields.
10. Click **Next** to open the dialog box for: “Configuration Options” on page 32.

Motion Sensors

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



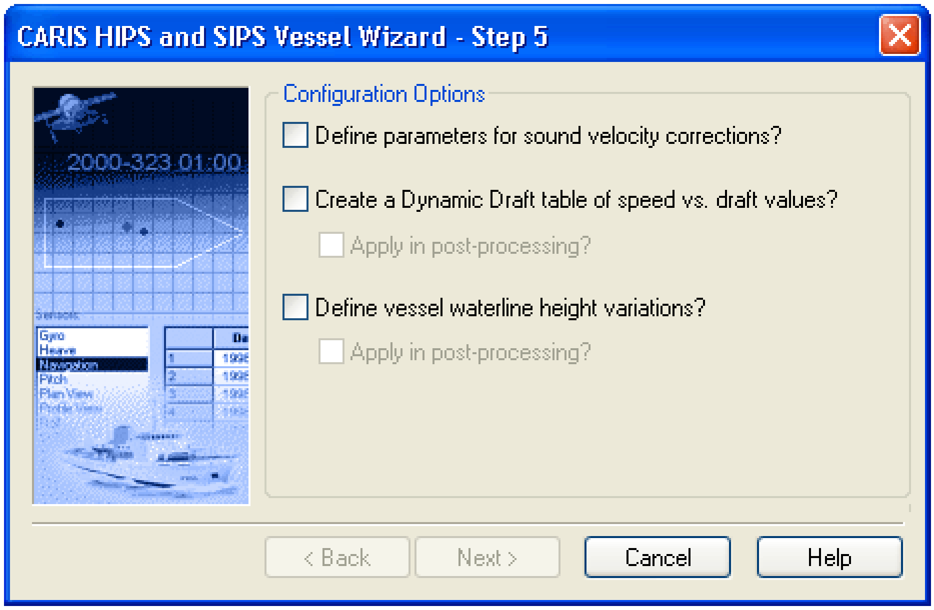
1. Create entries for any (or all) of the following sensors by clicking the appropriate box:
    - *heave*
    - *pitch*
    - *roll*

The *Apply in Post Processing* option means that the data is applied during the Merge process or during Sound Velocity Correction.

2. Click any of the check boxes if you want to apply the attitude sensor data in post processing.
3. Click **Next** to open the dialog box for: "Configuration Options" on page 32

Configuration Options

The Configuration Options dialog box is displayed



Use this dialog box to enable any or all of three options:

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

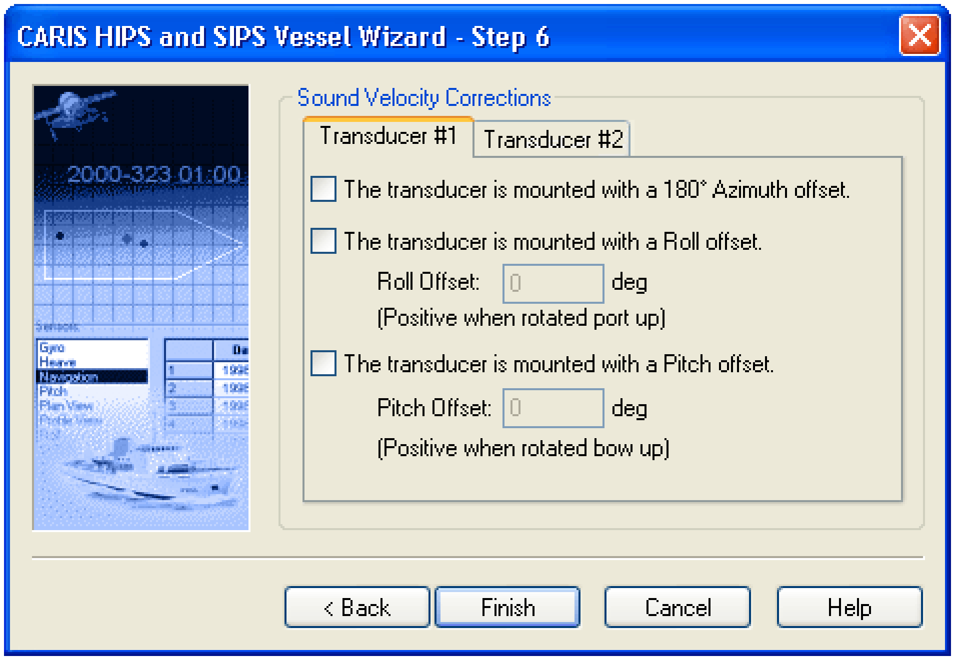
1. To include sound velocity corrections, click the *Define Parameters for Sound Velocity Corrections* check box.
2. To apply dynamic draft values to vessel speeds, click the *Apply Dynamic Draft* check box.

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

3. If you want to apply long period variations in the waterline height in processing data, click the *Define Vessel Waterline Height Variation* check box.
4. Click the appropriate *Apply in Post Processing* check boxes, as needed.
5. If you selected the *Define Parameters for Sound Velocity Corrections* check box, click **Next**, or click **Finish** if you did not select this option.

Sound Velocity Corrections

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



This dialog box records any transducer pole offsets.

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

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

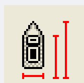
A new HIPS Vessel File is created. You can now create an outline for the vessel (see "Create Vessel Shape Outline" on page 35).

Create Vessel Shape Outline

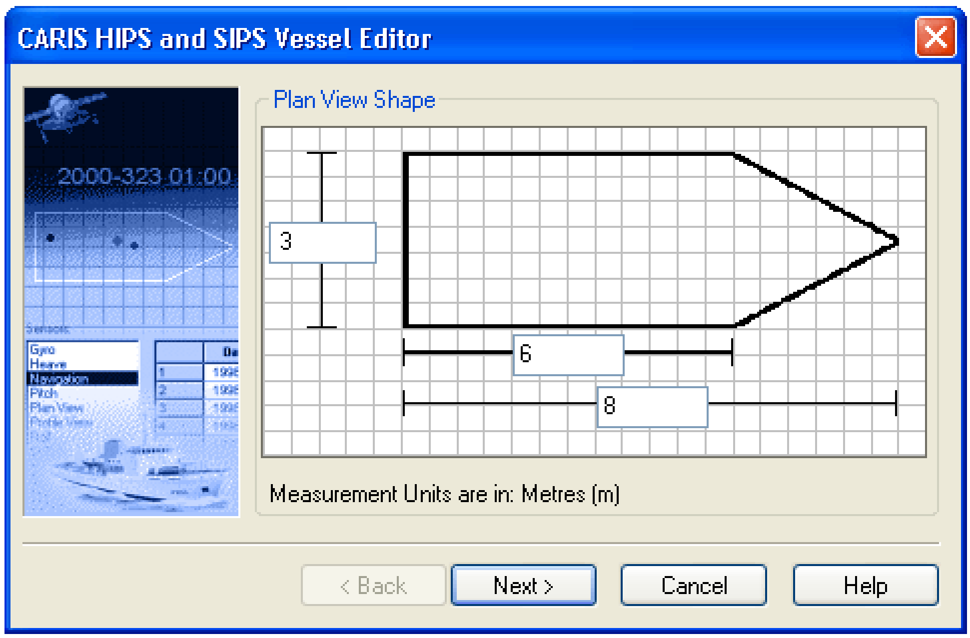
Use the Vessel Shape command to create a 3-D outline of the vessel, by entering measurements to define the length, width and height of the vessel, as well as the position of the reference point in the vessel.

You can also use the Vessel Shape command to revise an existing vessel outline.

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

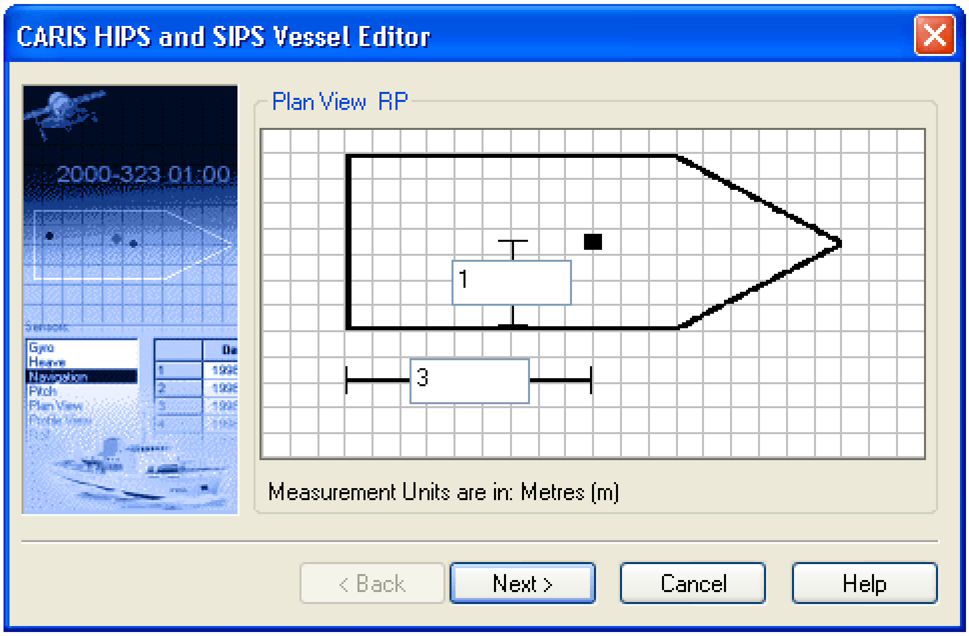
Menu	Edit > Vessel Shape
Tool	

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



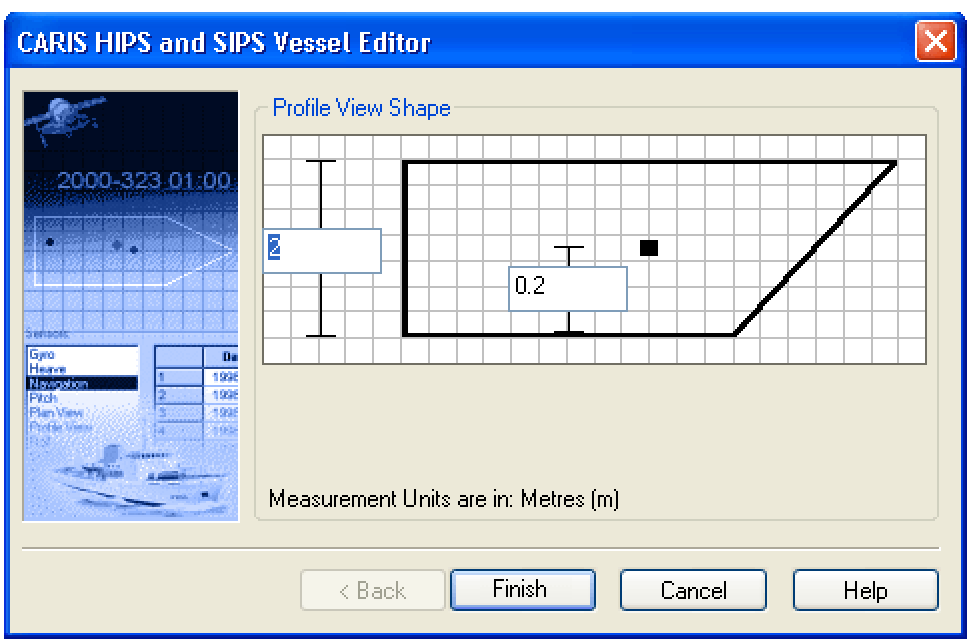
1. Enter a width for the vessel (port to starboard).
2. Enter a length for the vessel (stern to the base of the bow).
3. Enter a length for the vessel (stern to the tip of the bow).
4. Click **Next**.

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



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

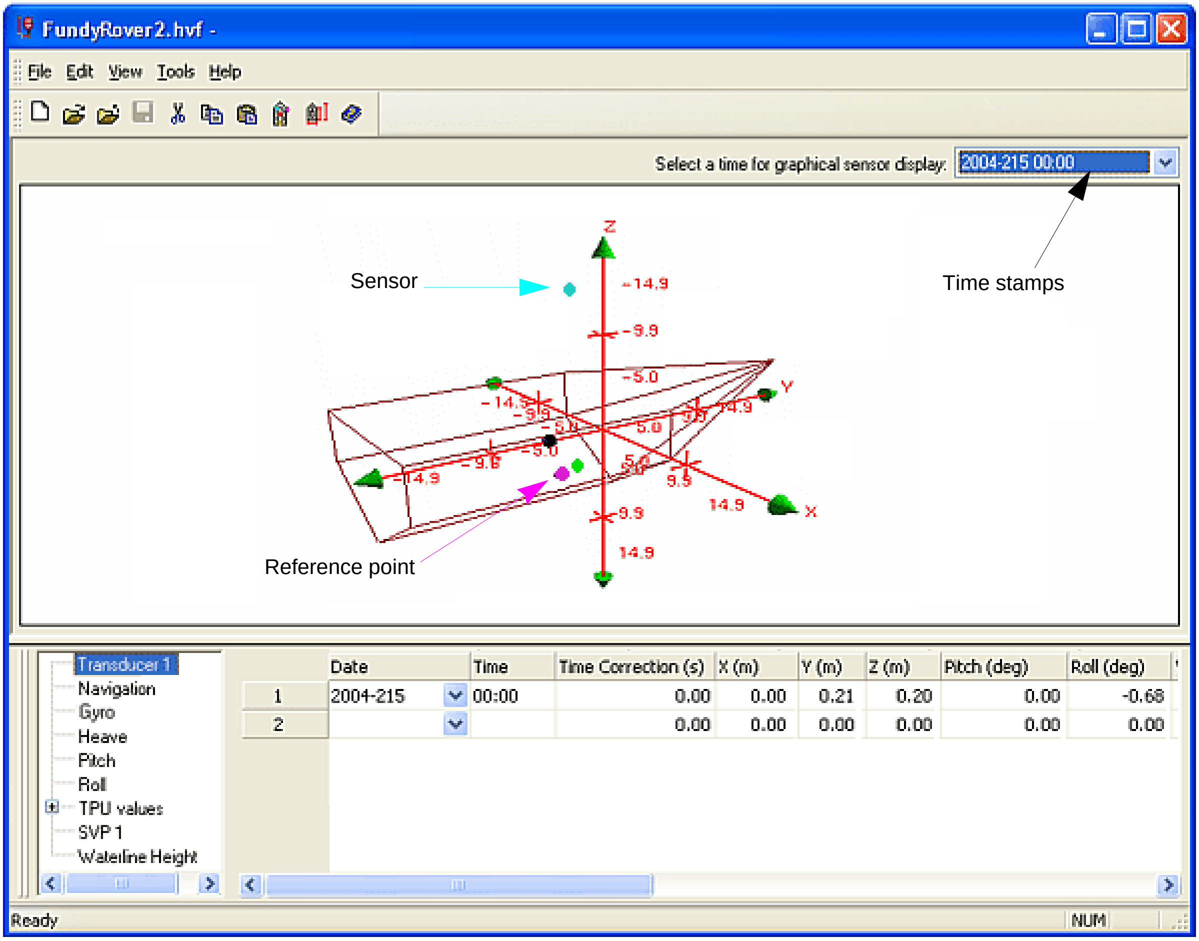
The Profile View Shape dialog box is displayed.



To set the height measurements for the vessel,

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

The Vessel Editor displays the outline of the vessel.



Negative values are used for these measurements:

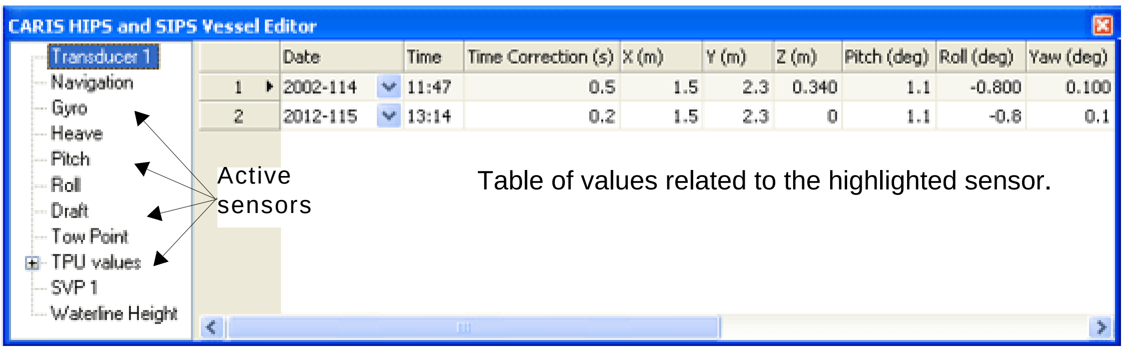
- from the origin (crosspoint of the axes) to port on the X axis,
- from the origin aft towards the stern on the Y axis,
- from the origin point up, on the Z axis.

Negative values on the axes are displayed with a “-” minus sign.

Sensor Configuration

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

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



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

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

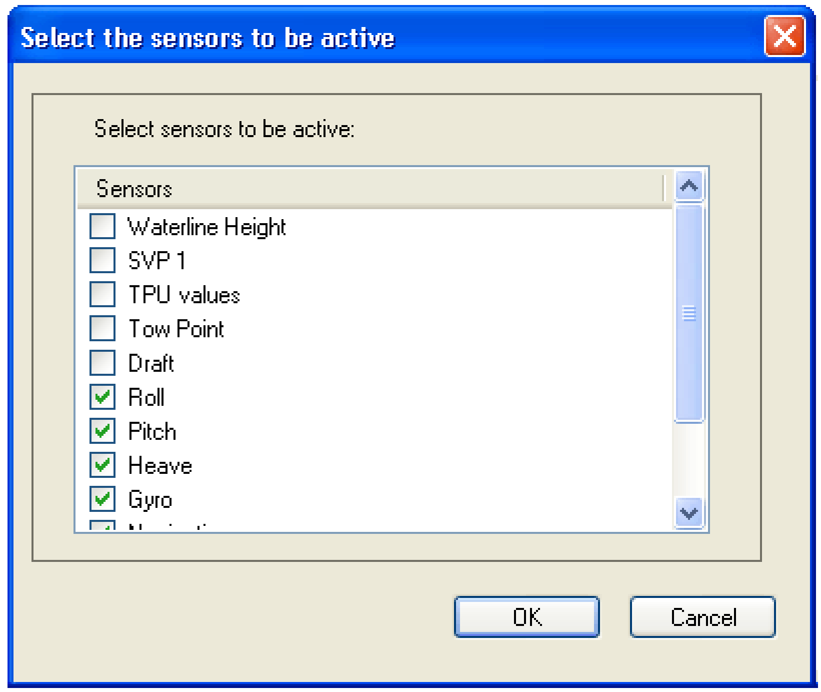
Active sensors

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

Menu	Edit > Active Sensors
Tool	

1. Select the Active Sensors command.

The Active Sensors dialog box is displayed.



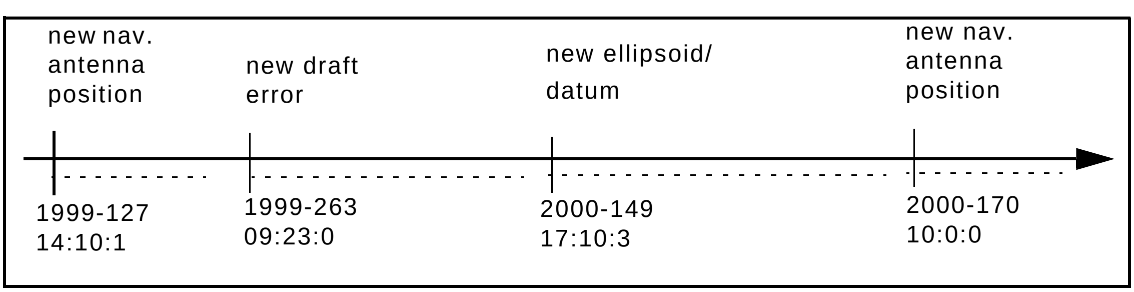
If a box is checked, the sensor is included in the vessel file and displayed in the Editor table.

2. Make sure a sensor box is checked to enable the sensor, or cleared to remove the sensor.
3. Click **OK**.

The list is refreshed to show the selected active sensors.

Time Stamp

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



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

Time Correction

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

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

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

Sensors

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

Transducer

The swath sonar, also known as multibeam, typically has beams that are formed in a fan shape, radiating from the centre of the transducer.

1. Click *Transducer1* in the Sensors list box so the selection is highlighted and the swath data fields are displayed.
2. Type data (as needed) in the following fields:
   - Date: The year and Julian day of current swath time stamp.
   - Time: The hour and minute of the current swath time stamp.
   - Time Correction: The time correction value.

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

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

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

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

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

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

Navigation

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

1. Click *Navigation* in the Sensors list box so the selection is highlighted and the navigation data fields are visible.
2. Type data (as needed) in the following fields:
   - *Date*: The year and Julian day of the current navigation time stamp.
   - *Time*: The hour and minute of the current navigation time stamp.
   - *Time Correction*: The time correction value.

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

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

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

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

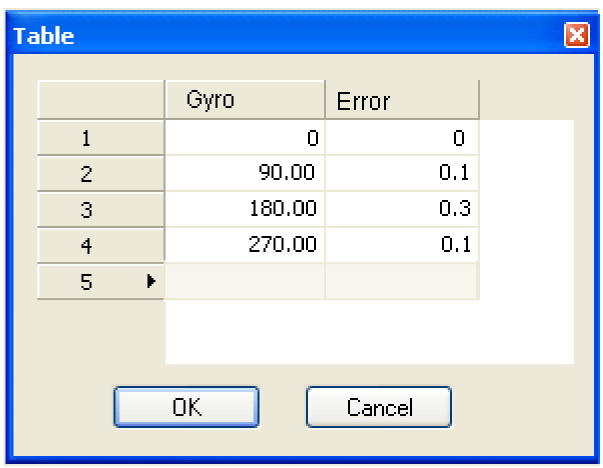
Gyro

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

1. Click *Gyro* in the Sensors list box so the selection is highlighted and the gyro data fields are displayed.
2. Type data (as needed) in the following fields:
   - *Date*: The year and Julian day of the current gyro sensor time stamp.
   - *Time*: The hour and minute of the gyro sensor time stamp.
   - *Time Correction*: The time correction value.

The Gyro Error is measured in degrees and is the difference between the recorded sensor value and the applied heading of the vessel: `Applied Heading = Recorded value - Gyro Error`.

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



4. Type new values in the *Gyro* and *Error* fields and click **OK** to close the dialog box.
5. Type data (as needed) in the following fields.
   - *Manufacturer*: The maker of the sensor equipment.
   - *Model*: The particular make of sensor equipment.
   - *Serial Number*: The serial number of the sensor equipment.

Heave

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

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

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

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

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

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

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

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

Pitch

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

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

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

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

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

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

Roll

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

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

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

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

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

Dynamic Draft

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

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

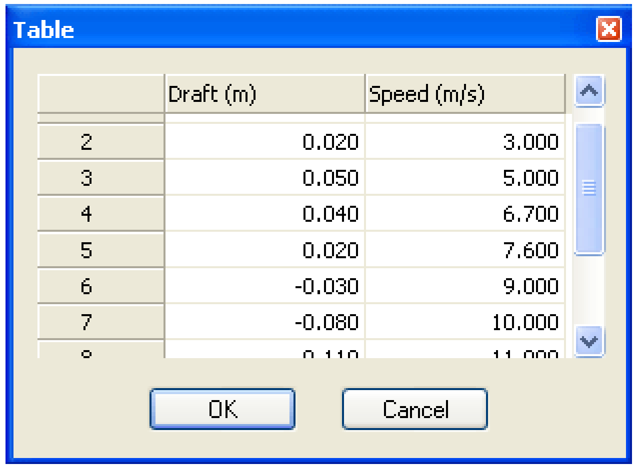
The following formula is used to correct soundings for dynamic draft:

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

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

You can also create your own dynamic draft model and load Delta Draft information directly which will override the Dynamic Draft table in the HVF. (See "Delta Draft" on page 172.)

1. Click *Dynamic Draft* in the sensors list box so the selection is highlighted and the dynamic draft data fields are displayed.
2. Type data (as needed) in the following fields:
   - *Date*: The year and Julian day of the current time stamp for the draft table.
   - *Time*: The hour and minute of the current time stamp for the draft table.
   - *Apply*: Select Yes to apply the table during merge (or No to not apply the table).
3. Click **Edit** in the *Error* field. A dialog box containing a table with up to 10 speed-draft pairs is displayed.



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

The speed-draft table is saved in the HVF.

SVP

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

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

1. Click *SVP 1* in the Sensors list box so the selection is highlighted and the SVP transducer data fields are visible. If there is a second transducer then click *SVP 2* and complete the same procedures as listed below.
2. Type data (as needed) in the following fields:
   - *Date*: The year and Julian day of the current SVP pole time stamp.
   - *Time*: The hour and minute of the current SVP pole time stamp.
3. Select Yes if there are dual transducers (or No if there is only a single transducer).

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

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

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

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

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

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

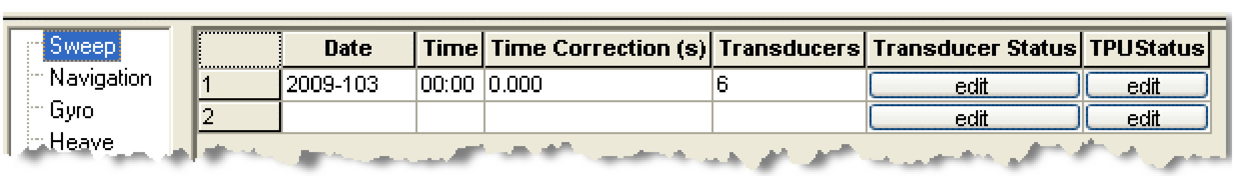
See also “Sound Velocity Corrections” on page 33.

To apply SVP in processing see “Sound Velocity Correction” on page 133.

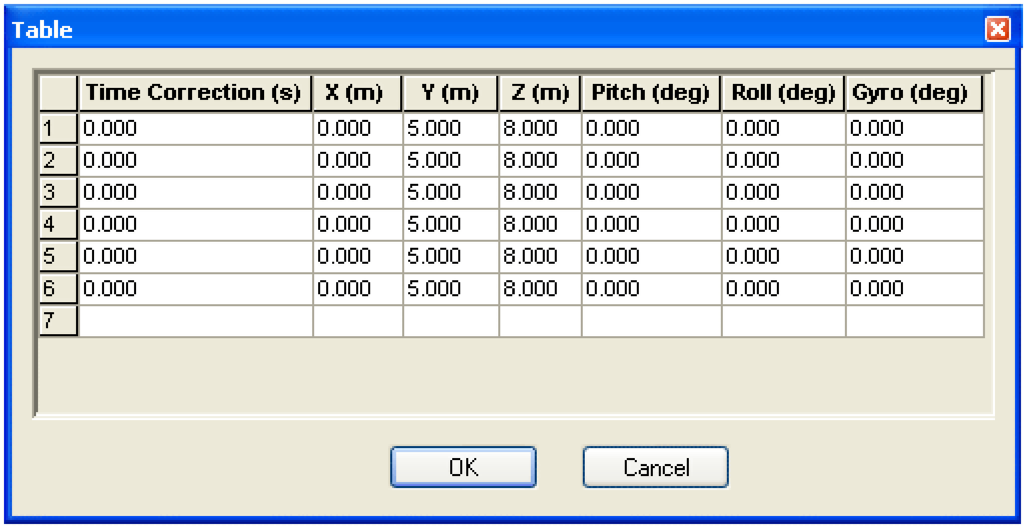
Sweep

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

1. Click *Sweep* in the Sensors list box so the selection is highlighted and the data fields are visible.

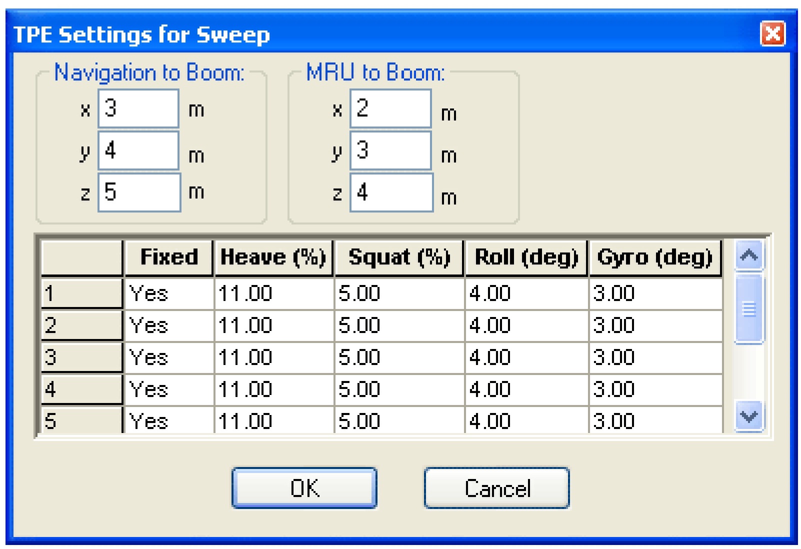


2. Type data (as needed) in the following fields:
   - *Date*: The year and Julian day of the current sweep time stamp.
   - *Time*: The hour and minute of the current sweep time stamp.
   - *Time Correction*: The time correction value.
   - *Transducers*: Number of beams mounted on the boom.
3. Click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



	Time Correction (s)	X (m)	Y (m)	Z (m)	Pitch (deg)	Roll (deg)	Gyro (deg)
1	0.000	0.000	5.000	8.000	0.000	0.000	0.000
2	0.000	0.000	5.000	8.000	0.000	0.000	0.000
3	0.000	0.000	5.000	8.000	0.000	0.000	0.000
4	0.000	0.000	5.000	8.000	0.000	0.000	0.000
5	0.000	0.000	5.000	8.000	0.000	0.000	0.000
6	0.000	0.000	5.000	8.000	0.000	0.000	0.000
7							

4. Type values in the fields and click **OK**.
5. Click **Edit** in the *TPU Status* field to enter TPU values for each beam on the boom and to edit values entered in the new vessel wizard.



	Fixed	Heave (%)	Squat (%)	Roll (deg)	Gyro (deg)
1	Yes	11.00	5.00	4.00	3.00
2	Yes	11.00	5.00	4.00	3.00
3	Yes	11.00	5.00	4.00	3.00
4	Yes	11.00	5.00	4.00	3.00
5	Yes	11.00	5.00	4.00	3.00

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

Towed Sensors

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

1. Click *Towed* in the Sensors list box so the selection is highlighted and the Towed data fields are displayed.
2. Type data (as needed) in the following fields:

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

Waterline Height

The purpose of this section of the vessel file is to define long-period changes in the vessel's draft due to fuel burn or other loading changes.

Specifically, you can enter the height of the waterline below the Reference Point.

If the Waterline Height section of the vessel configuration is not defined here, then HIPS uses a default of zero as the height of the waterline below the RP.

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

> No interpolation of waterline height is done during Sound Velocity Correction.

1. Click *Waterline Height* in the Sensors list box so the selection is highlighted and the waterline data fields are displayed.
2. Type data (as needed) in the following fields:

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

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

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

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

   Comments: A text field for your use.

For Simrad data, the Waterline value must be set to the same value recorded as **WLZ** in the Simrad Installation Datagram.

This Waterline value will only be used during Sound Velocity Correction.

Set the Apply switch to **"**No**"**. If it is set to "Yes", it will be applied twice, once in SVC and again in Merge.

Total Propagated Uncertainty

This section defines the values used in the calculation of Total Propagated Uncertainty (TPU). TPU is derived from a combination of estimates of the accuracy of each individual sensor, estimates such as.

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

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

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

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

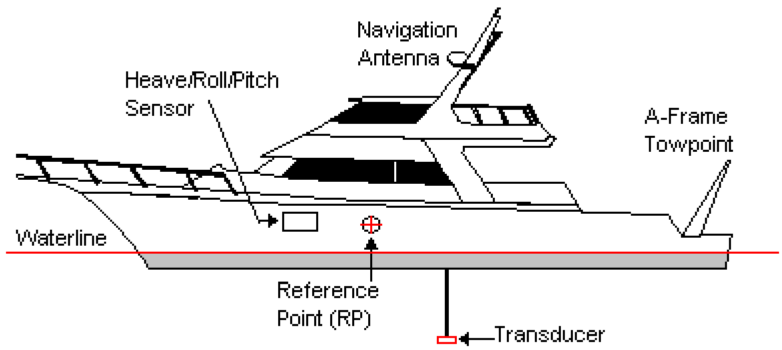
1. Expand the TPU section of the HVF by clicking the + icon.
2. Type data (as needed) in the Offsets section:
   - *MRU to Transducer*: The physical offset in three dimensions from the motion recording unit to transducer 1 on the vessel.
   - *MRU to Transducer2*: The physical offset in three dimensions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

- *Transducer Roll 2*: The mounting roll offset for transducer 2. The offset is positive when rotating the transducer away from starboard (starboard down).

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

Vessel Coordinate System

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



Reference Point

Sensor positions are described using X-Y-Z axis coordinates relative to a Reference Point (RP). The RP is a location from which all other positions are derived. (Imagine standing at the location of the Reference Position and describing how far and in what direction the sensor is away from you.)

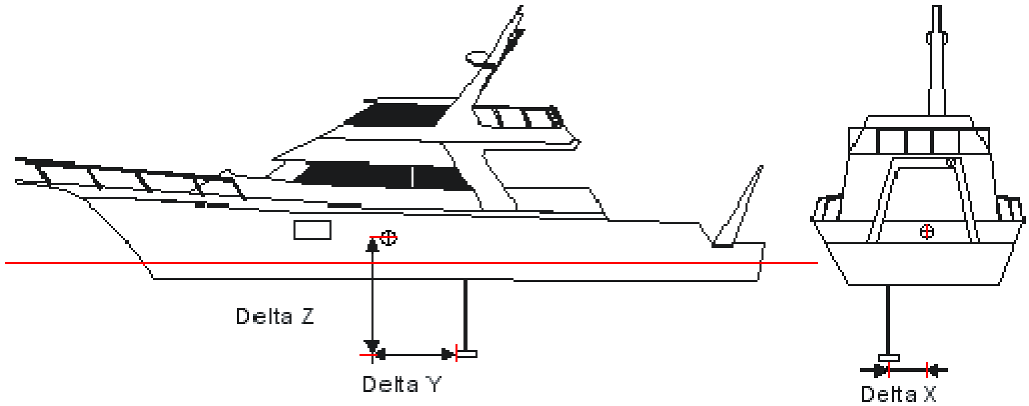
The location of the RP depends upon the type of sonar system used in the survey. Its location is usually at the centre of the ship's gravity. This position is used as the centre of the ship's rotation when applying the pitch/heave/roll parameters, and must be applied as accurately as possible.

Coordinate definitions

The axes are defined as follows:

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

This is illustrated in the following image.



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

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

Default Ellipsoid

The HVF must contain the ellipsoid and datum used by the navigation system in the survey vessel, regardless of whether projection coordinates or geographic coordinates are stored in the data.

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

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

Menu	Tools > Options

1. Select the Options command.

The Options dialog is displayed.

2. Select the Options - General tab.
3. Select an ellipsoid from the pull-down menu.
4. Click **OK** to close the dialog box.

The name of the ellipsoid is displayed in the Navigation section of the editor.

3

Create a New Project

All project information in HIPS and SIPS is maintained in a project data base. New projects are created with a wizard, which generates a HIPS Project File (.hpf).

In this chapter...

Define New Project 58
Open Projects 64
View Project Properties 67
Set Line Properties.. 71
Rename Day and Line Folders 73
Archive Project 74

Define New Project

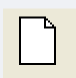
In HIPS and SIPS, survey data is organized into projects. The default directory for project files is *drive*\Program Files\CARIS\HIPS\8.1\Data.

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

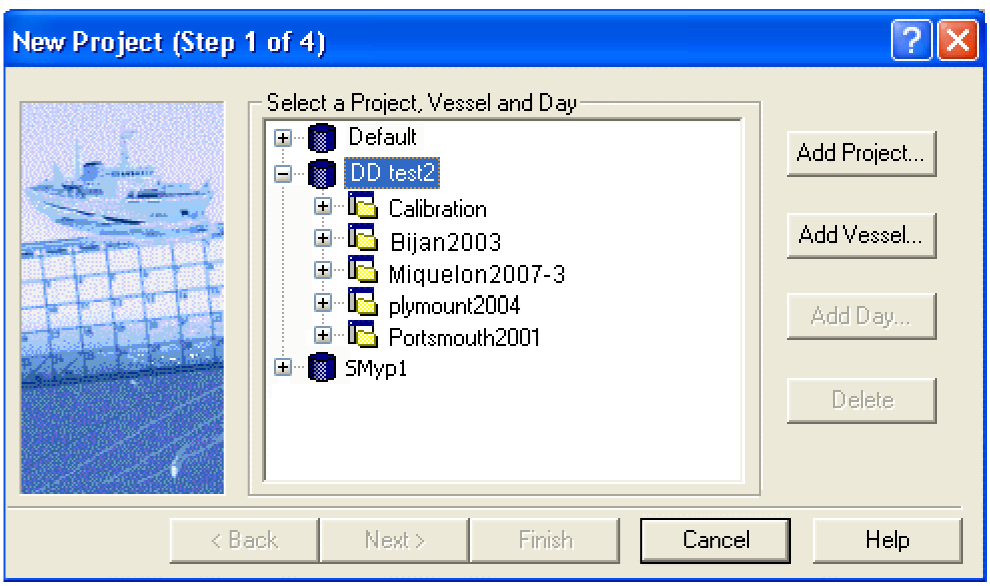
If no HVF is available, it must be defined before the new project can be created. (See “Create a New HVF” on page 27.)

To define a new project to organize your survey data in P/V/D/L/ hierarchy:

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

Menu	File > New > Project
Tool	

New Project (Step 1)



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

Add a new Project

1. Select an existing data folder, or,
1. Right-click in the default listing and select *Connect to* to browse to another folder location.
1. To create a new project folder, click **Add Project**.

The New Project Name dialog box is displayed.

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

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

Change or add data repository

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

To create a new project in another project repository:

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

Add a new Vessel folder

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

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

The Available Vessels dialog box is displayed.

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

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

Add a new Day folder

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

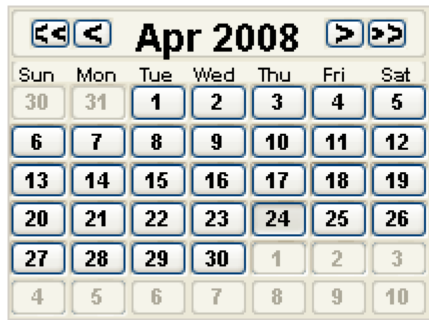
Day folders are shown in Julian Day format, for example, April 24, 2009 is shown as 2009-114.

To add a Day folder to a Vessel folder:

9. Select the Vessel folder.

10. Click **Add Day**.

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



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

Add to an existing project

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

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

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

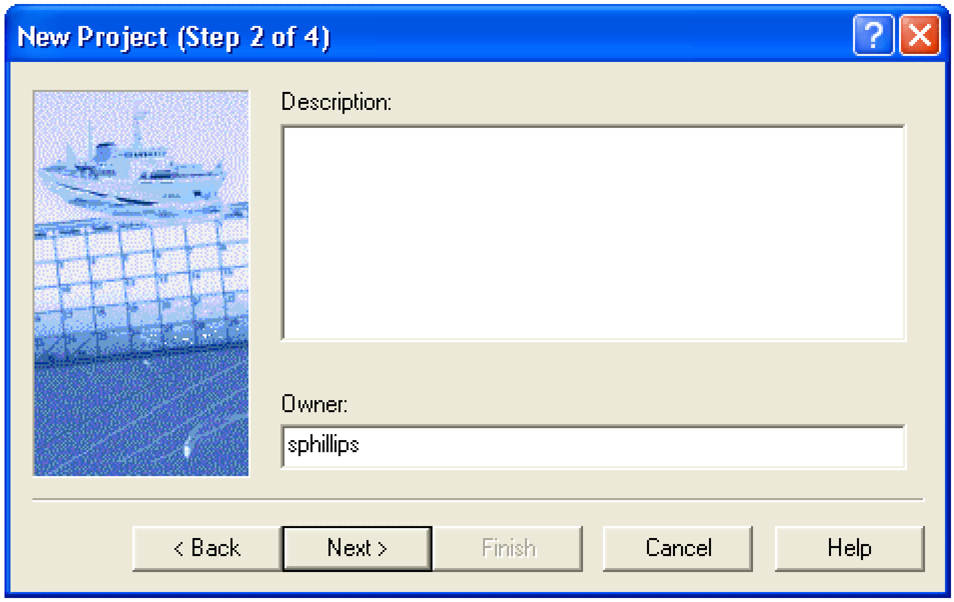
Delete empty folders

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

New Project (Step 2)

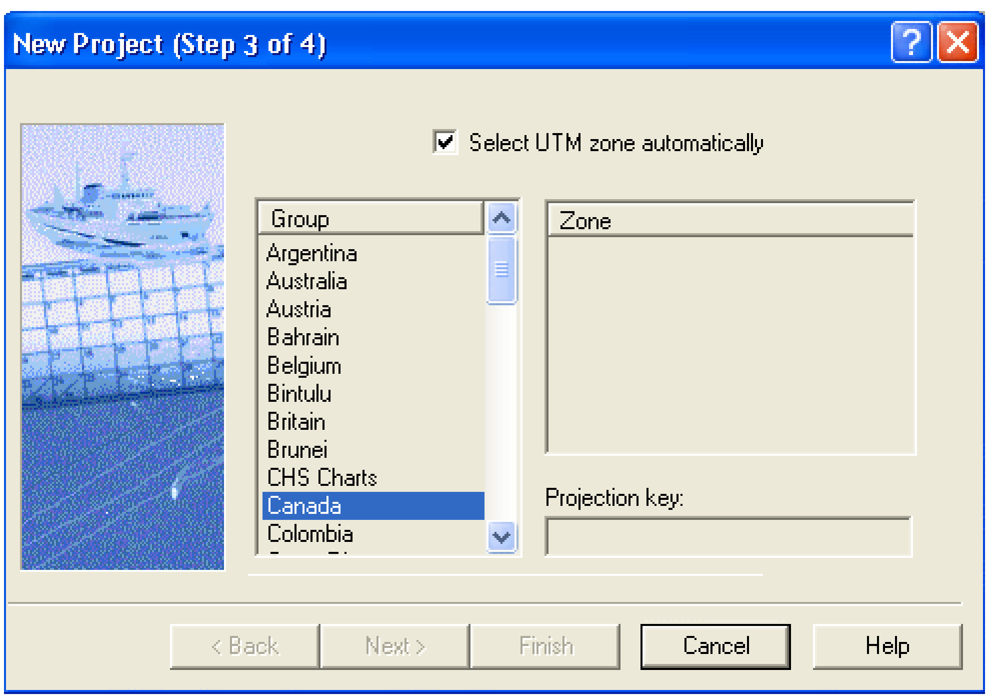
Use the New Project (Step 2) dialog box to describe the project..

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



New Project (Step 3)

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



To set a different projection for your project:

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

3. Select a zone from the *Zone* list box.

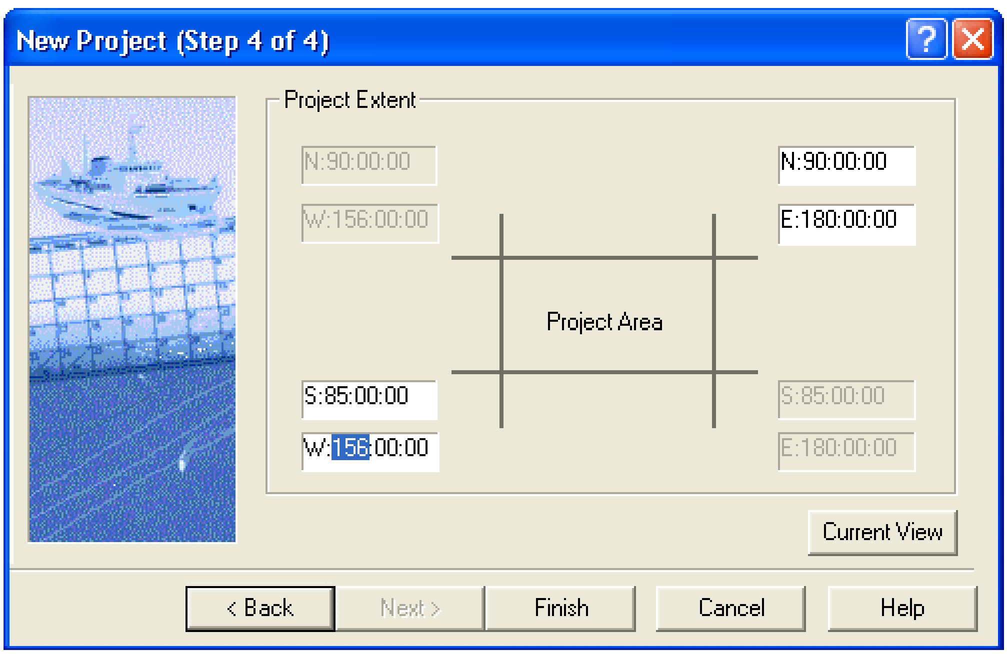
The Projection key name is automatically displayed once the country and zone have been selected.

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

4. Click **Next**.

New Project (Step 4)

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



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

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

1. Click **Current View**.

To manually enter the geographic coordinates for the project area:

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

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

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

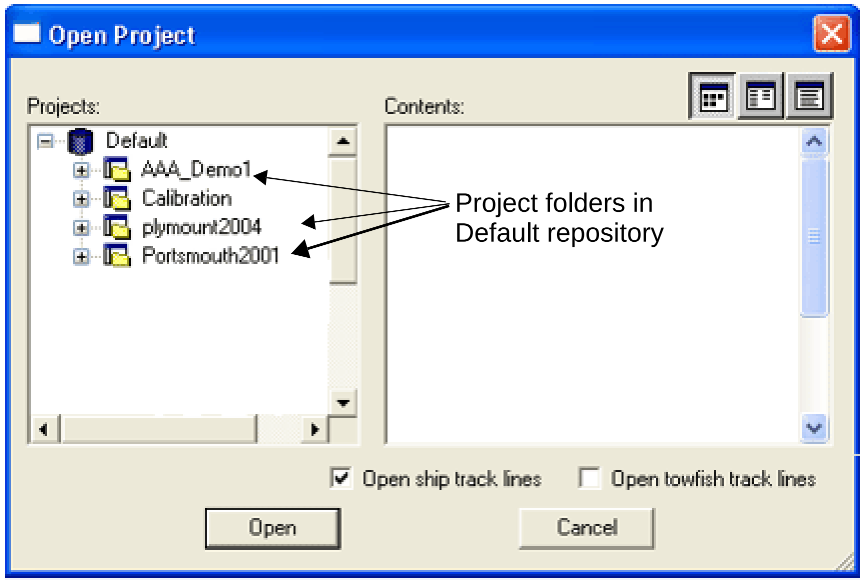
The new project has been created with Project-Vessel-Day folders according to the options selected in the wizard.

Open Projects

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

1. Select the Open Project command.

The Open Project dialog box is displayed.



2. Select a Project folder so it is highlighted.

If the project contains both bathymetric and side scan data, you can open either the vessel or the towfish navigation, or open both. Your choice is saved in the registry and the session file for future use.

3. Select either *Open ship track lines* or *Open towfish track lines*, or both options, to open the appropriate track lines in the Display window.
4. Click **Open**.

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

When you open a project created in a version of HIPS and SIPS prior to 8.0, the project will automatically be updated use the new *.hips database file.

Multiple Data Paths

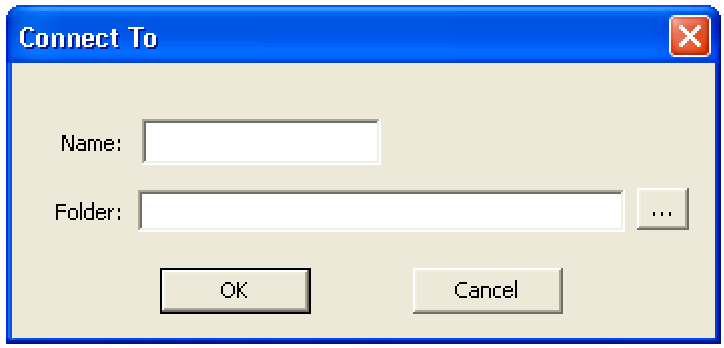
The default repository for HIPS data is *drive*\Program Files\CARIS\HIPS\8.1\Data or *drive*\Program Data \CARIS\HIPS\8.1\Data. The projects stored here are shown in the Connect To dialog box under “Default”.

You are not limited to a single repository for project data. You can add other locations to the project tree in both the New Project and Open Project dialog boxes.

To add a new project location:

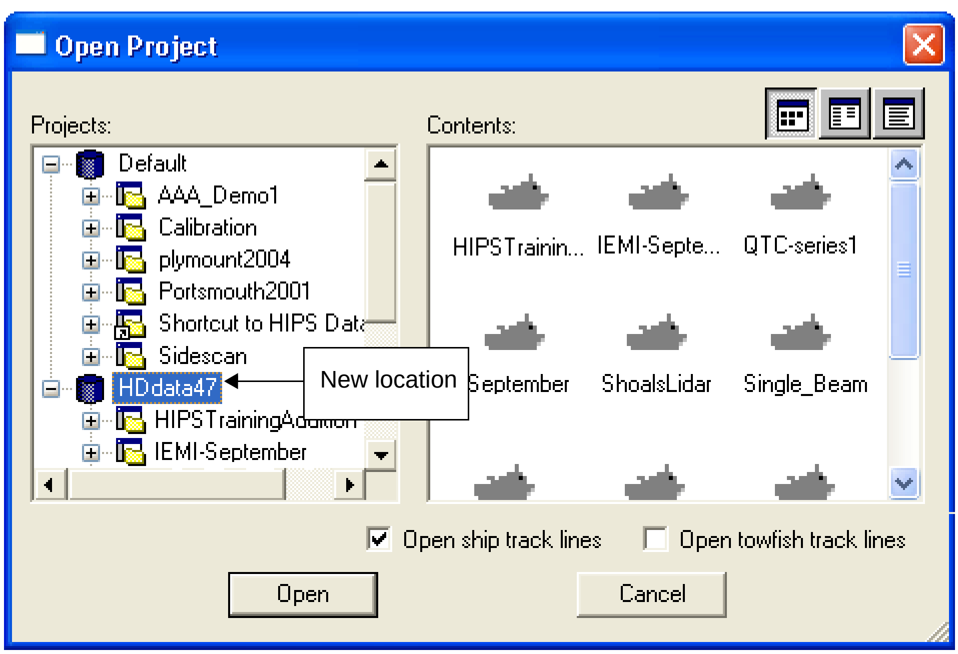
1. Select Connect To from the right-click menu in the Projects listing.

The Connect To dialog box is displayed.



2. Browse to the location of project folders that are on a different path from your Default repository.
3. Type a name for the new location
4. Click **OK**.

The new project location is listed in project tree in the Open Project dialog box.



Open specific data

You can also open the data for a specific vessel (if the project contains more than one), or for a specific survey day, or for an individual track line. As you expand each folder, the contents are displayed in the Contents pane of the Open Projects dialog box.

1. Choose the Open Project command.
2. In the Open Project dialog box, expand the folder tree by clicking the + icon so that the Vessel and Day folders are visible.
3. Select a vessel folder name so it is highlighted, and click **Open**, or

   Expand the Vessel folder to see the Day folders.
4. Select a Day folder and click **Open**, or
5. Expand the Day folder to see the Line folders.
6. Select a Line folder and click **Open**.

Combining project data using Windows shortcuts

You can use a Windows shortcut to link to HDCS Project/Vessel/Day/Line data on different drives or network locations. The project, plus the shortcut to the other data, is displayed in the Project dialog box. The data folders must contain a Vessel Configuration File.

1. Create a Windows shortcut in any P/V/D/L directory.
2. Copy the shortcut to the project where you want to place the shortcut. Remember that the shortcut must be copied to the appropriate folder. A Project folder shortcut must be copied to a HDCS data folder, a Vessel folder shortcut must be copied to a Project folder, etc.
3. Open the data using the Open Project command.

View Project Properties

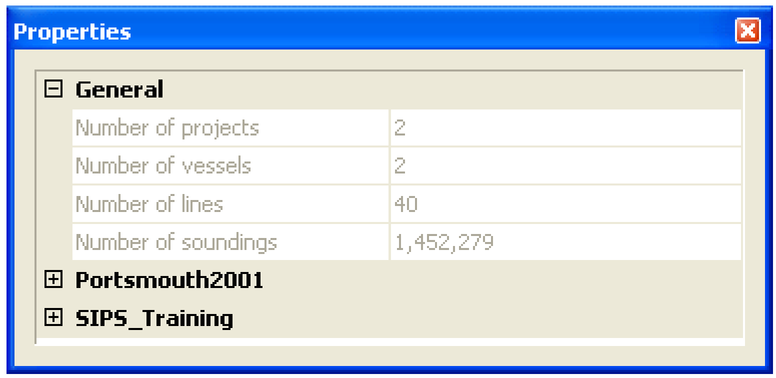
Essential information for open projects can be viewed in the Properties window.

1. Select the HIPS Data layer in the Layers tab of Control window.

Menu	Window > Properties
Pop-up Menu	Properties

2. Open the Properties window.

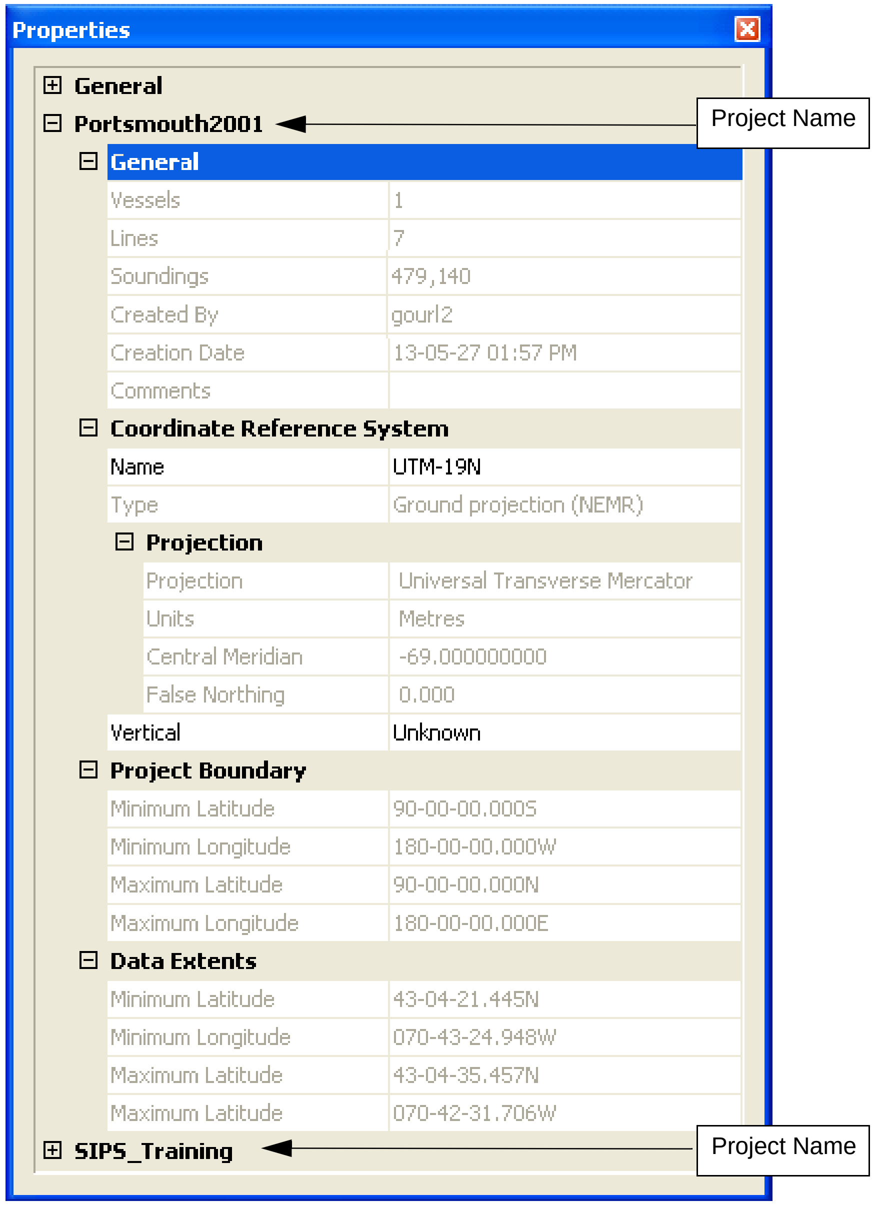
The Properties for all open projects are displayed, as in the examples below.



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

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

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



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

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

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

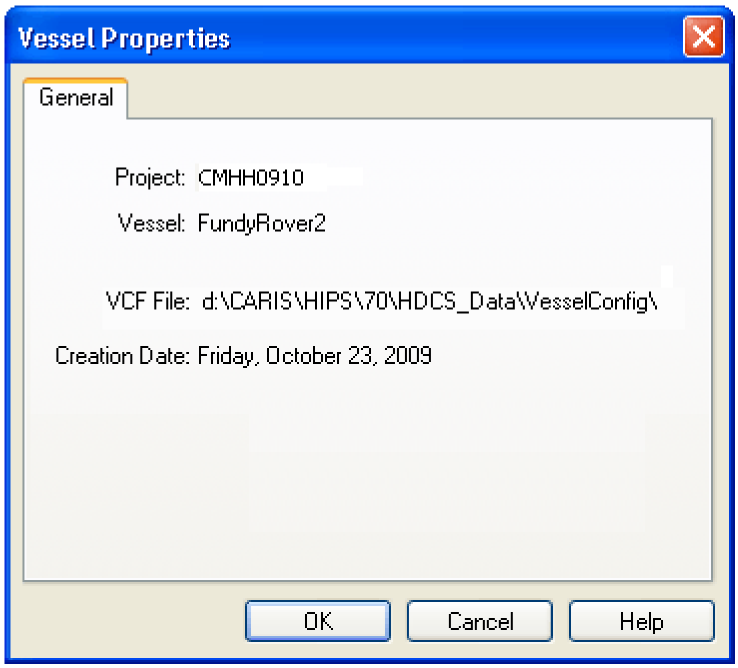
View Vessel Properties

Vessel information for the currently open project can be viewed in the Vessel Properties dialog box.

Menu	View > Properties
Pop-up	Properties

1. Select a Vessel folder from the Project tab in the Control window.
2. Select the Properties command.

The Vessel Properties dialog box displays the location and creation date of the Vessel file. This information is read-only.



Set Line Properties

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

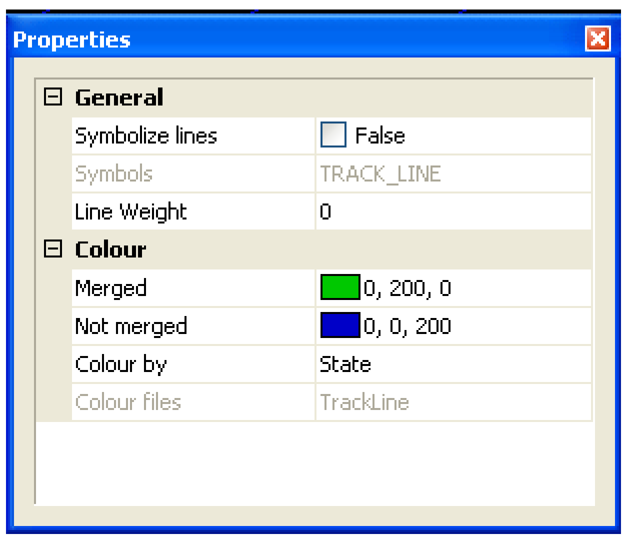
These default colours are set in the Display tab of the Tools > Options dialog box. See "Display window" on page 100 of the Reference guide.

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

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

1. Select the Ship Track Lines or Towfish Track Lines layer in the Layers tab of the Control window.
2. Open the Properties window. (Shown below are the properties of the Ship Track Lines layer.)

Menu	Window > Properties
Tools	
Pop-up	Properties



General

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

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

   Select a feature code from the *Symbols* drop-down list. (The default symbolization is TRACK_LINE.
2. Type a Line Weight value to set the size of a selected line in the Display. (A line weight can only be set if *Symbolize lines* is set to "False").

Line Colour

Ship track lines and Towfish track lines have colour properties based on these different states:

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

To change these identifying colours:

3. Highlight the field and select a new colour from the drop-down list.

Line colour can indicate State, but as well lines can also be coloured by SV Profile or by Tide applied. You can also have a different colour per line.

To change the determinant of colour for either kind of line:

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

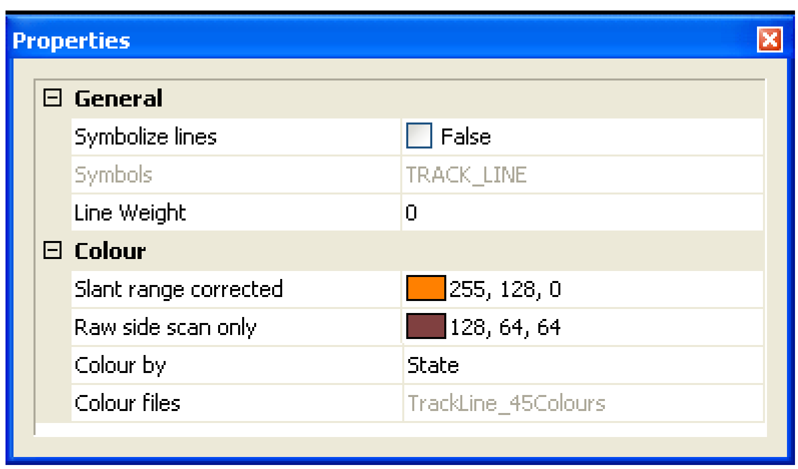
   State is the default setting, e.g., Merged/Not merged.

   SV Profile colours the lines based on the sound velocity profile applied. (Ship track line only)

   Tide colours them based on the tide zone file applied. (Ship track line only)

   Multiple colours uses a set of 10 distinct colours to colour the lines. This can be useful if data was logged as multiple files along the same runline.

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



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

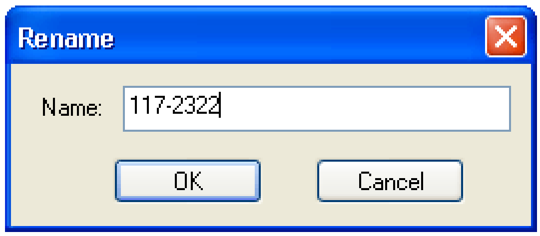
Rename Day and Line Folders

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

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

Menu	Edit > Rename
Pop-up	Rename

2. Select the Rename command.



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

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

The changed file name is displayed in the Control window.

Archive Project

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

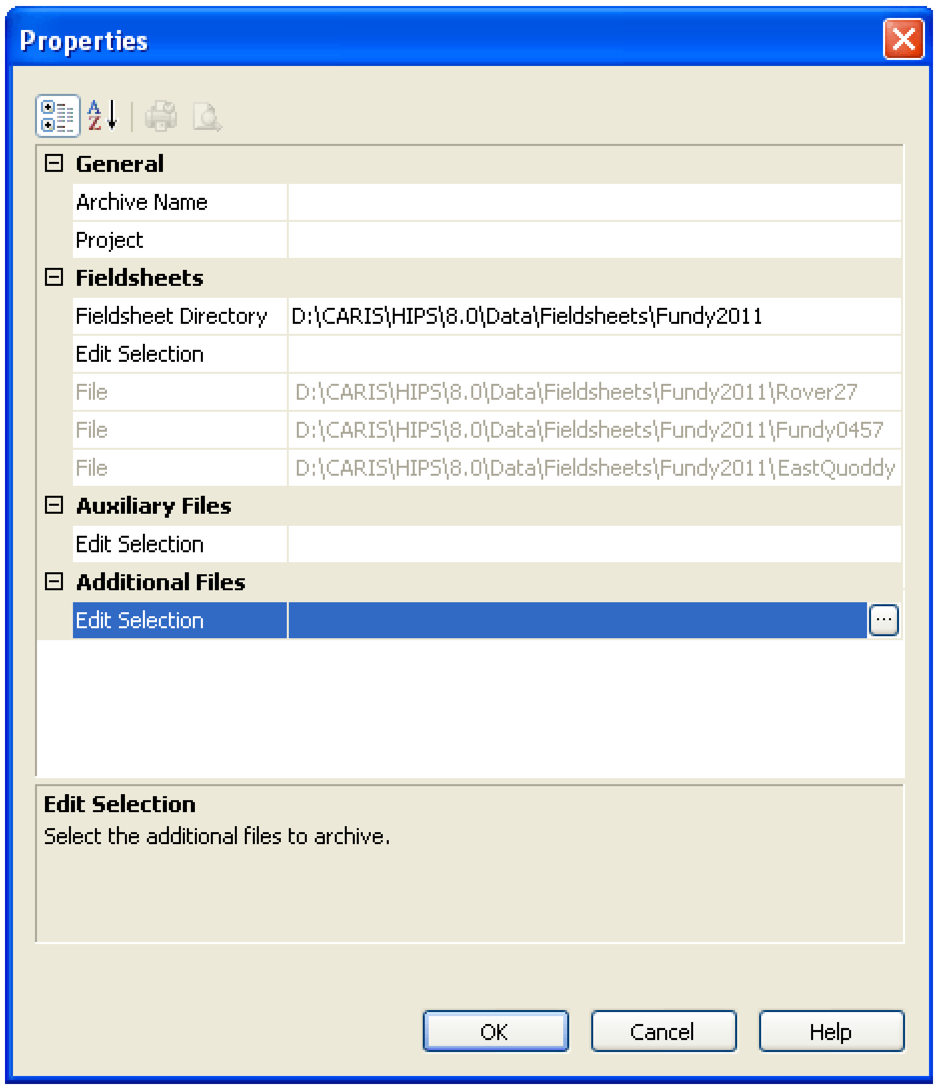
To archive a project:

Menu	File > New > Archive

1. Select the New Archive command.

The archive Properties dialog box is displayed.

.



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

2. Set paths and select files, as follows:

Field name	
General:	
Archive Name	1. Click **Browse** to open the Save As dialog box. 2. Type a name and set a destination directory for the archive file and click **OK**.
Project	1. Click **Browse** to open the Select Project dialog box. 2. Double-click on Default to open the project tree. 3. Select the project and click **OK.**
Fieldsheets:	
Fieldsheet Directory	1. Click **Browse** to select the field sheet directory from the Browse for Folder dialog box. 2. [Optional] Make a new folder. 3. Click **OK** to enter the path in the field.
Edit Selection	1. Click **Browse** to view the Open Field Sheets dialog box. 2. Click on the project name to expand the list of available field sheets for the project. 3. Select the field sheets to be included in the archived file. • Select a single fieldsheet by clicking on it. • Use <Ctrl> to select more than one field sheet from the list. 4. Click Open to list the selected field sheets.
Auxiliary Files:	(Related files necessary for the successful processing of data, for example, tide, sound velocity, delta draft, delayed heave files, etc.)
Edit Selection	1. Click **Browse** to open the Select Files dialog box. 2. Click **Add** to open the standard Select Files dialog box and locate the auxiliary files to be included. 3. Repeat until list of files is complete. 4. Click **OK**. Use **Remove** and **Clear** to adjust your selection.
Additional Files:	(Data that is associated with the project, but is not necessary for processing data, for example, background images and associated charts.)
Edit Selection	1. Click **Browse** to open the Select Files dialog box. 2. Click **Add** to open the standard Select Files dialog box and locate the additional files. 3. Repeat until list of files is complete. 4. Click **OK**. Use **Remove** and **Clear** to adjust your selection.

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

The ZIP file is created and contains these folders:

- VesselConfig
- Project
- PreProcess
- Fieldsheets
- Background.

4

Convert Data

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

In this chapter...

Converting Files to HIPS/SIPS Format 78
Formats that can be converted to HIPS and SIPS: .. 79
Options for Specific Formats 96

Converting Files to HIPS/SIPS Format

HIPS and SIPS files are created from survey data using the Conversion Wizard.(Survey data in ASCII format can also be converted using the Generic Data Parser. See "Generic Data Parser" on page 105 of the Tools guide.)

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

All HIPS and SIPS data is organized in a Project/Vessel/Day/Line directory structure. If you organize the raw data files according to this structure, then you can convert line data from entire Project or from multiple Vessel and Day folders contained in a common Project folder. Use the File Selection Type option in Step 2 of the wizard to do this (see "Select Files for Conversion" on page 82).

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

> You must organize your raw data files in a HIPS/SIPS Project-Vessel-Day structure to convert complete Project, Vessel and/or Day data files.

Formats that can be converted to HIPS and SIPS:

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

Convert Data: Formats that can be converted to HIPS and SIPS:

File Format	File extensions	Data types processed in HIPS and SIPS	Geocoder Support
Simrad	*.all, *.out, *.raw, *.depth, *.mb57	multibeam, single beam, side scan	**Yes** BA/TS (for *.all data only)
Spawar	*.dat	swath	No
Swathplus	Submetrix SXP files *.sxp, *.sxr, *.sxi	multibeam, side scan	No
Teledyne	*.tdy, *.*	swath, side scan	No
UNB	swathed files (*.merged) Reson *.*	swath	No
Winfrog	*.*	single beam	No
XTF	*.xtf	multibeam, single beam, side scan, sweep	**Yes** (Reson, R2Sonic, various side scan)

BA = Beam Average

TS = Time Series

Select Data Format

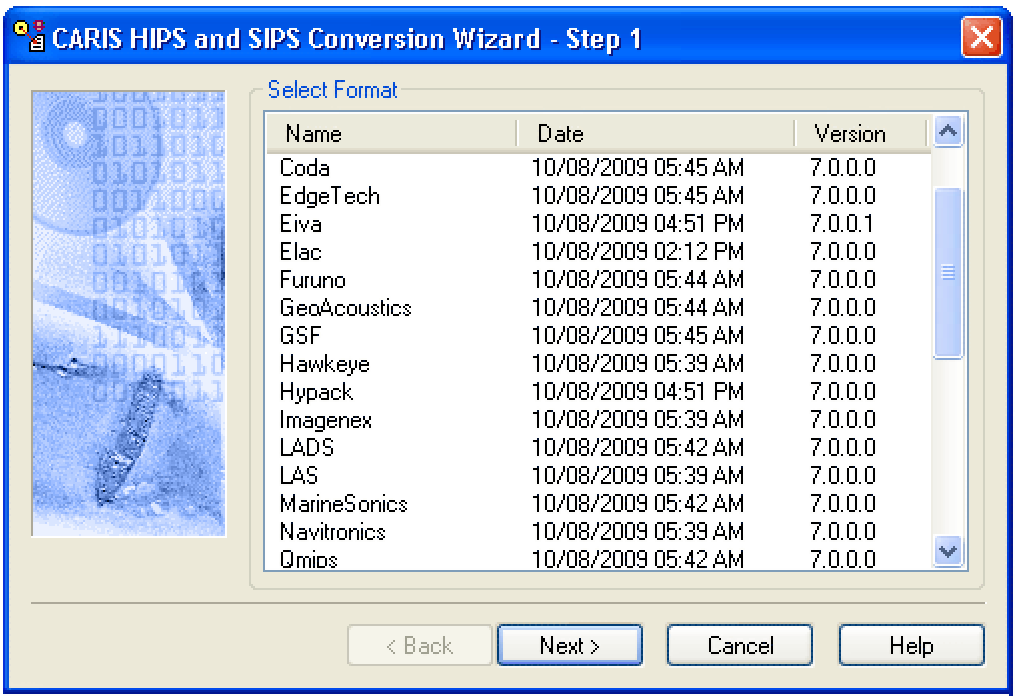
Menu	File >Import > Conversion Wizard
Tool	

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

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

The first Conversion dialog box is displayed.

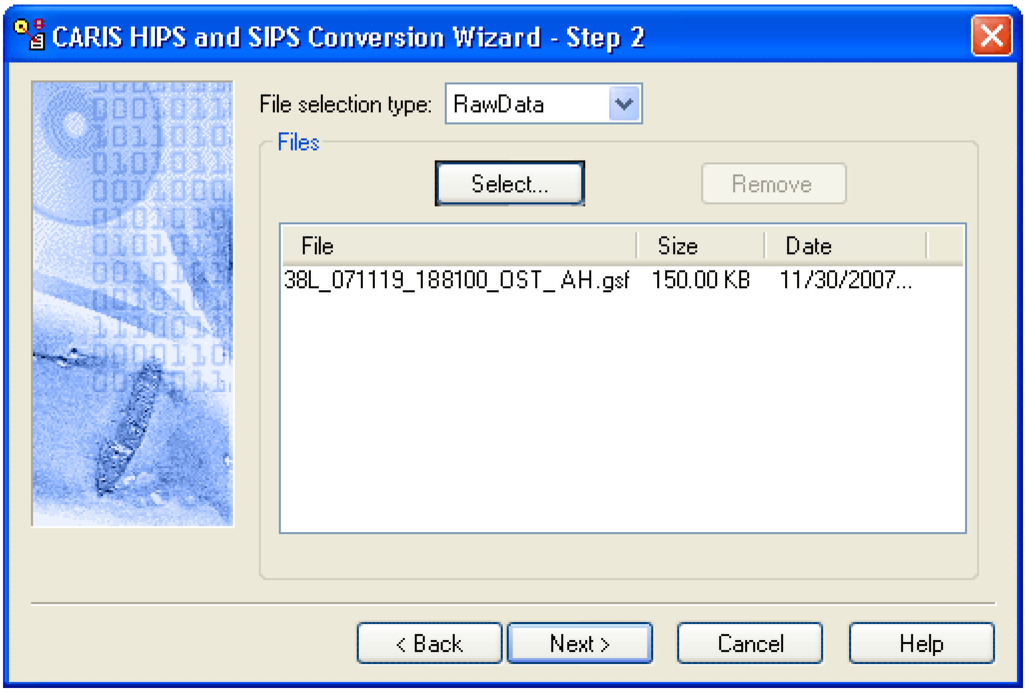
The Step 1 dialog box lists the data acquisition systems for which data can be converted to HIPS and SIPS files. (This list is created dynamically from all format libraries with the name `convert_*.dll` in the Hips\Bin directory.)



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

Select Files for Conversion

In most converters this dialog box will appear so you can select the files to be converted. Some formats have other import options to set before selecting data files. (See “Import Options” on page 83.)



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

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

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

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

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

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

Import Options

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

Create or Update survey lines

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

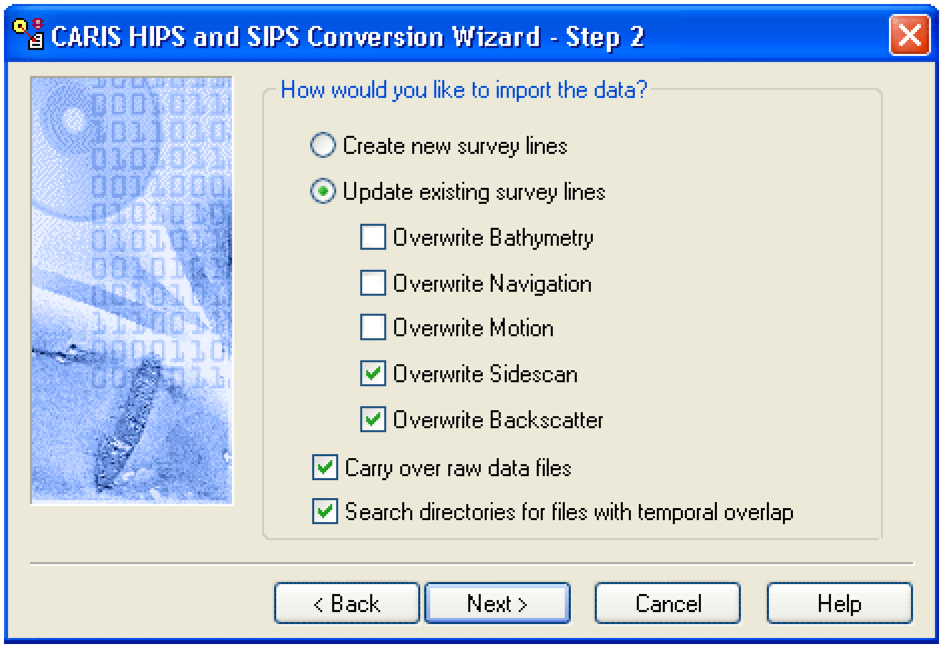
Instead of overwriting all existing data, you can choose to update only certain data. For example, you can opt to update only the navigation information on your processed survey lines, or to add backscatter imagery data to a project, without overwriting any of the rest of your processed data.

At present this option is available for these formats:

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

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

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



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

Carry Over Raw Data Files

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

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

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

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

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

Search for Files with Temporal Overlap

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

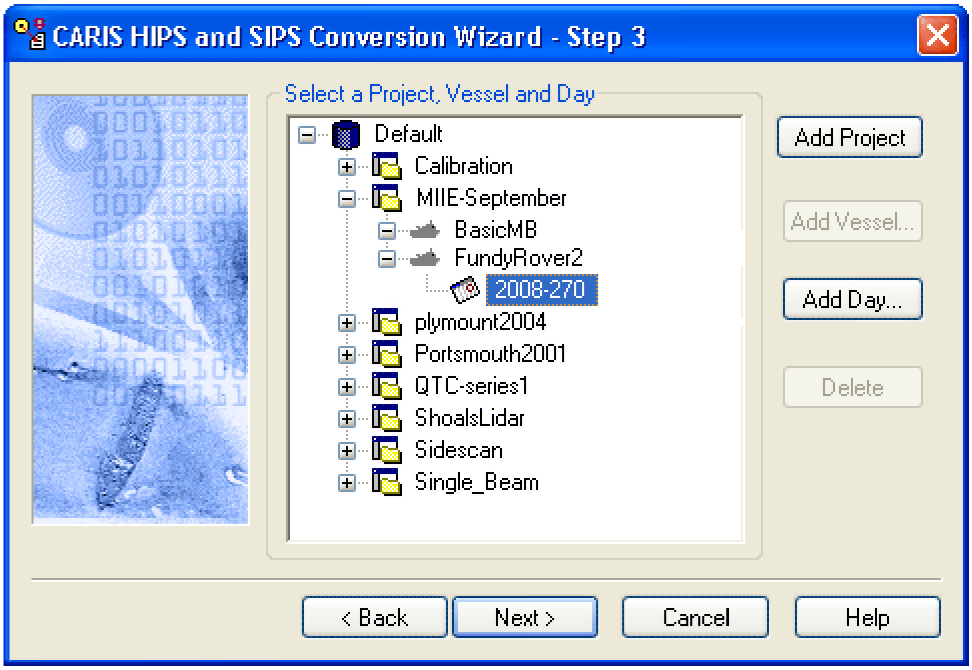
This scanning can be time-consuming, particularly with large data sets. However, if this synchronizing is not done when a line has been split, the data in the converted line segments may not overlap correctly.

To turn off this synchronizing function:

4. Select the *Search directories for files with temporal overlap* check box.
5. Click **Next** to select the specific files to import.

Select Project, Vessel and Day

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



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

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

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

To create a new project:

1. Click **Add Project** and follow the steps to "DEFINE NEW PROJECT" ON PAGE 58.

Add Vessel or Day

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

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

The Available Vessels dialog box is displayed.

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

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

To add a Day folder to the project:

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

The Calendar dialog box is displayed.

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

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

Remove folder

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

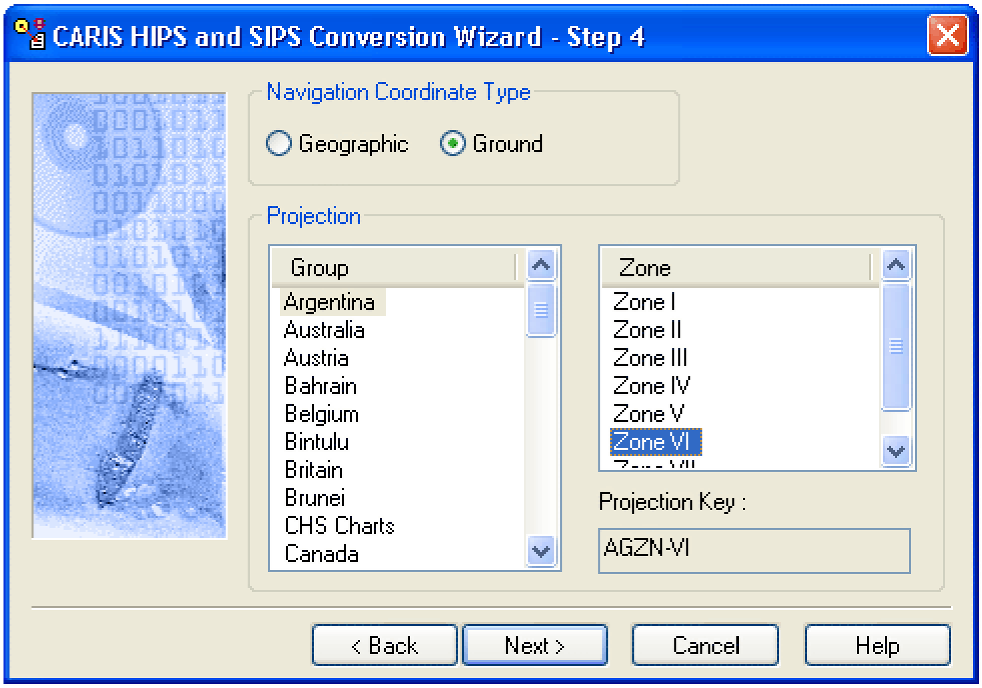
To add a Project to the root directory:

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

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

Set Navigation Coordinate System

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



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

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

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

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

Setting Geographic as Navigation Coordinate Type for Hypack data:

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

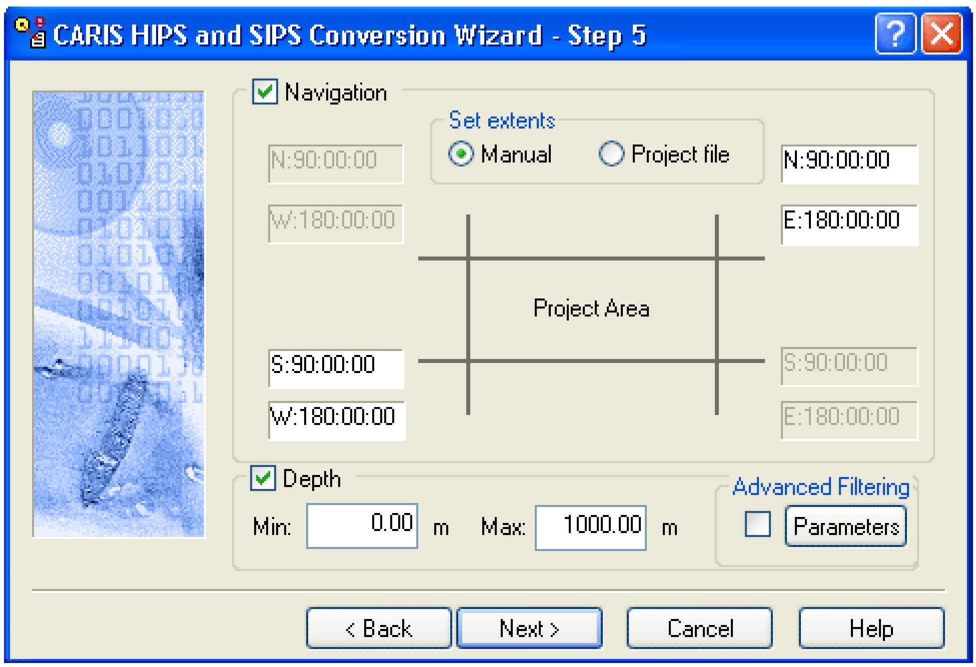
When both the Geographic option and Multibeam options are set during conversion, the converter will search for *.raw files in the same directory with the same line name to get this information.

Select Filters

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

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

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



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

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

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

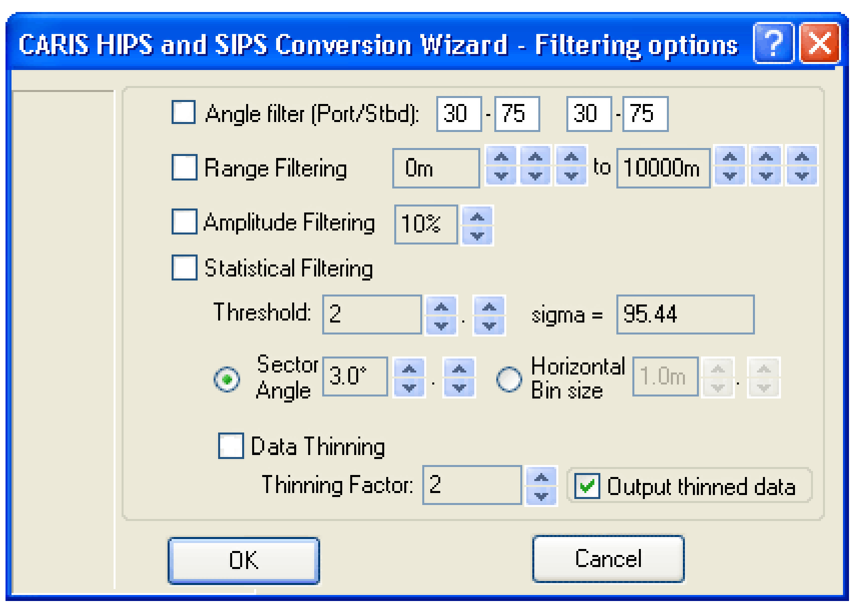
The HIPS converter for reading Klein SDF data files has been enhanced to read the bathymetry data from the Klein data format 5000 V2. As a result of this enhancement, the filtering capabilities in step 5 of the Conversion Wizard are now enabled for this conversion. These settings are functional only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

4. Select the *Depth* check box to enable the depth filter.
5. Enter the minimum acceptable depth.
6. Enter a maximum acceptable depth.
7. For advanced filtering for multibeam or multi-transducer data select the *Advanced Filtering* check box and click **Parameters** to set advanced filtering options.

Advanced Filtering

Currently, advanced filtering is only available for these multibeam and multi-transducer formats: GeoAcoustics, Klein SDF, EIVA and Hypack.

If filtering is not available for the type of data you are converting, these options will be greyed-out.



Certain data can be filtered so that only the best quality data is imported to HIPS. The filter is applied to port and starboard beam angles.

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

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

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

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

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

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

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

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

The equivalent confidence value is displayed as a percentage.

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

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

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

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

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

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

6. Click **Next**.

Enter Data Parameters

Depending on the data acquisition format you selected in the Step 1 of the wizard, one or more dialog boxes, appropriate to the format, are now displayed. See "Options for Specific Formats" on page 96 for description of options for different raw data formats.

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

Convert Data

The final step is to launch the conversion process.

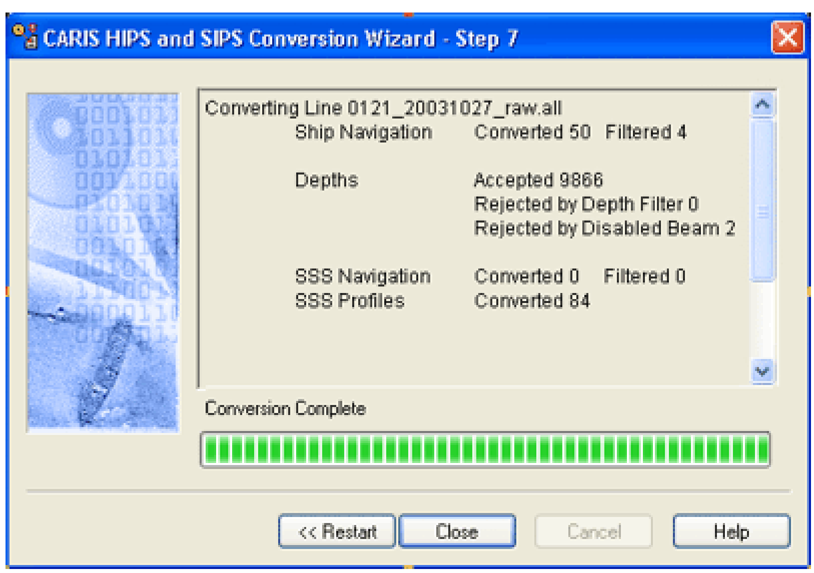
The settings and parameters you have selected are displayed in the dialog box. This information can be copied to a text file, using the Copy command on the right-click menu.

To convert the data using the settings:

1. Click **Convert**.

Conversion progress

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



This information can also be copied to a text file.

The converted files are saved in the directory you selected in Step 2 of the conversion process (see “SELECT FILES FOR CONVERSION” ON PAGE 82).

2. Click **Close** to close the wizard.

[Optional] to return to the initial step of the conversion process click **Restart.**

Options for Specific Formats

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

These are the current formats which can be converted:

"Atlas" on page 97
"ChirpScan3D" on page 98
"CMAX" on page 98
"Coda" on page 98
"EdgeTech" on page 99
"EIVA" on page 101
"Elac" on page 102
"Furuno" on page 103
"GeoAcoustics" on page 104
"GSF" on page 107
"Hawkeye" on page 108
"Hypack" on page 109
"Imagenex" on page 112
"LADS" on page 113
"LAS" on page 114
"MarineSonics" on page 114
"Navitronics" on page 114
"Qmips" on page 115
"Teledyne Reson PDS" on page 116
"SAS" on page 119
"Scripps" on page 119
"SDF" on page 120
"Seabeam" on page 121
"Seafalcon" on page 121
"SEGY" on page 121
"SHOALS" on page 122
"Simrad" on page 122
"Spawar" on page 124
"SWATHPlus" on page 124
"Teledyne" on page 124
"UNB" on page 124
"Winfrog" on page 124
"XTF" on page 126

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

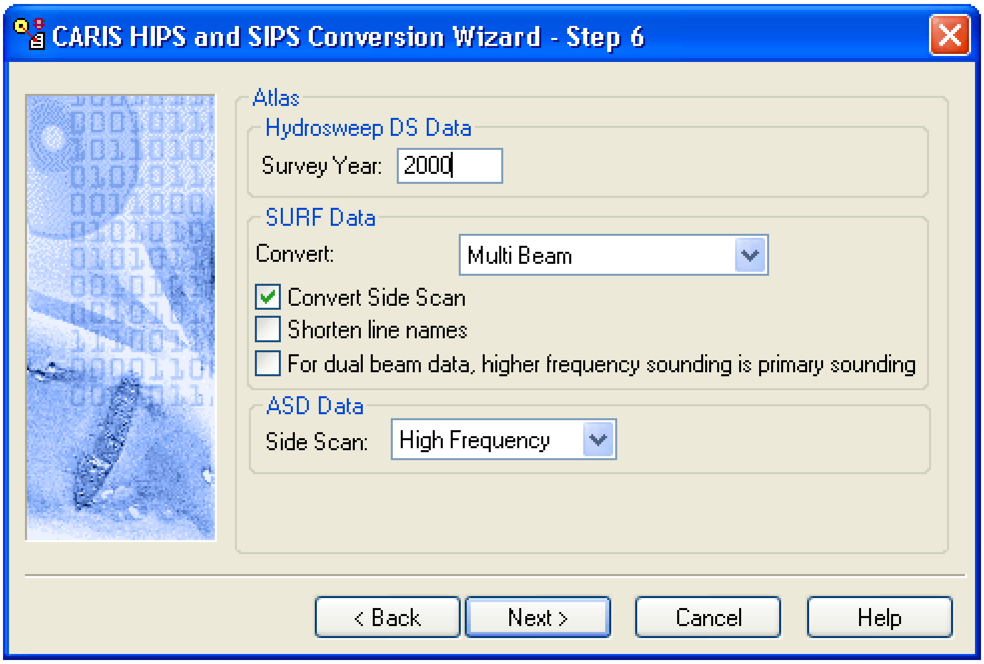
- Navitronics format conversion
- Hawkeye waveform viewer

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

Atlas

The ASCII versions of Hydrosweep DS files are now supported.

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



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

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

4. Select the *Shorten line names* check box to modify the file name to a 12-character name (YYDDD_HHMMDD).
5. Select the higher frequency data as the primary soundings (if data format is dual-frequency).

6. Select High Frequency or Low Frequency from the drop-down list to convert Atlas ASD side scan data (if available).

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

Chirpscan3D

There are no options for the Chirpscan3D format.

CMAX

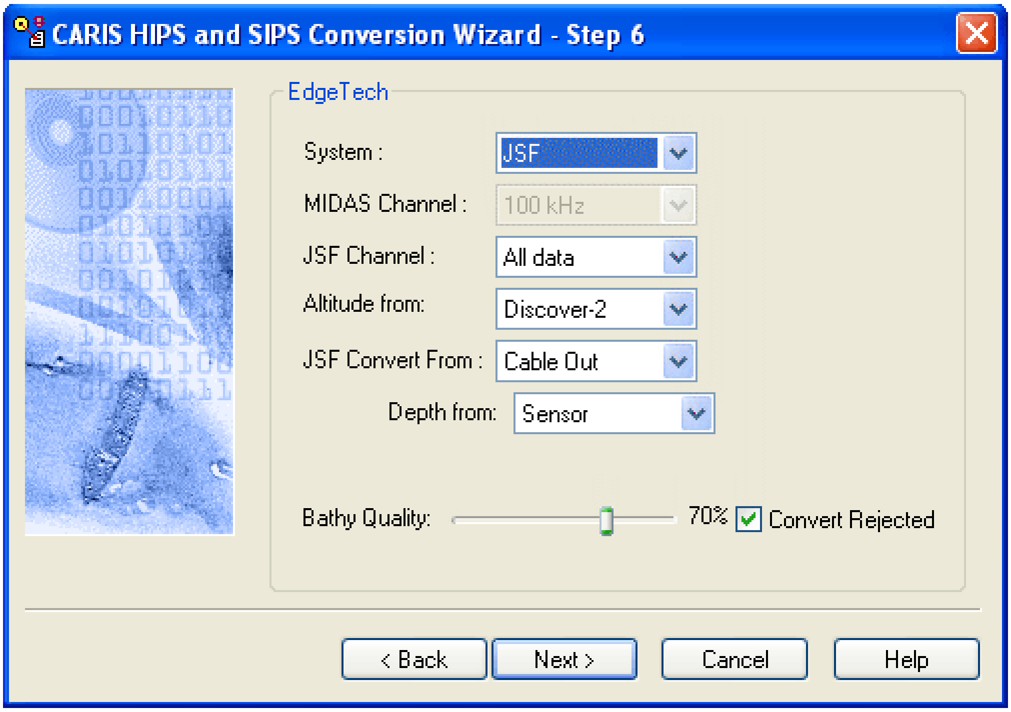
There are no options for the CMAX format.

Coda

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

EdgeTech

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



1. Select a side scan system:
    - Midas
    - 260
    - JSF
2. If Midas is enabled, select a sonar frequency channel:
    - 100 KHz
    - 500 KHz
3. If JSF is enabled, select a sonar frequency channel:
    - All data
    - Low Frequency
    - High Frequency
4. If altitude data is present in the JSF data, select the data source:
    - Sensor
    - Discover-2
5. Select either Layback or Cable Out from which to convert the JSF data.
    - If you select Cable Out, select the source for Depth, either Sensor or Discover-2.

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

Use the Bathy Quality slider to set the quality value below which soundings will be automatically rejected during conversion.

6. Set a value between 50% and 100% using the slider.

Since each ping can contain thousands of samples, you can reduce the volume of data that's converted by turning off the *Convert rejected* option.

7. [Optional] Clear the *Convert rejected* check box.

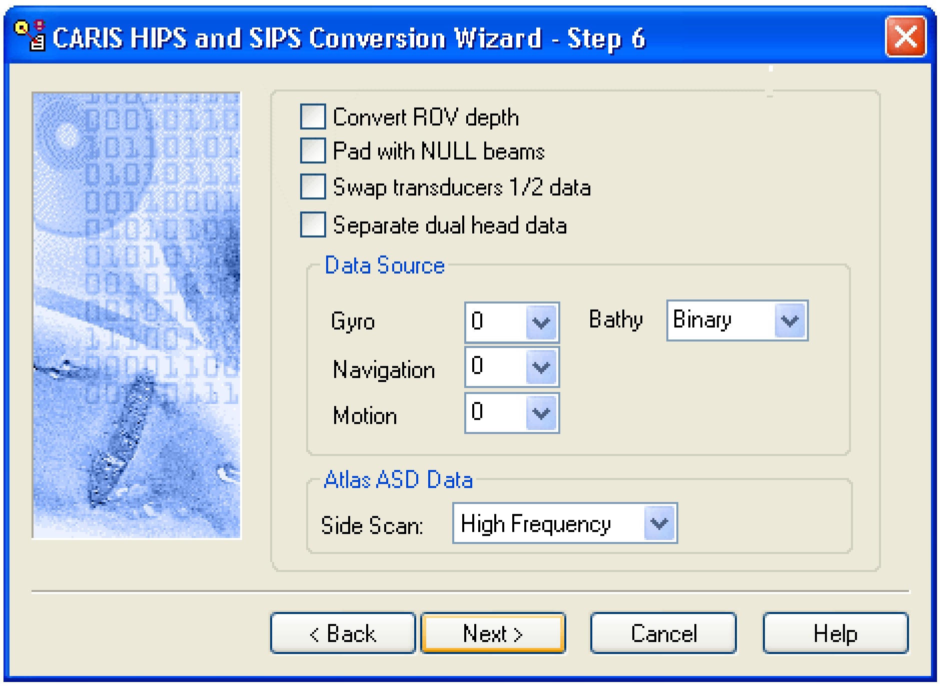
If you leave this check box selected, all data will be converted. However, data which does not meet the quality value will be converted with “Rejected” status.

EIVA

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

See "ADVANCED FILTERING" ON PAGE 91.

Step 6 offers further conversion options specific to EIVA data.



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

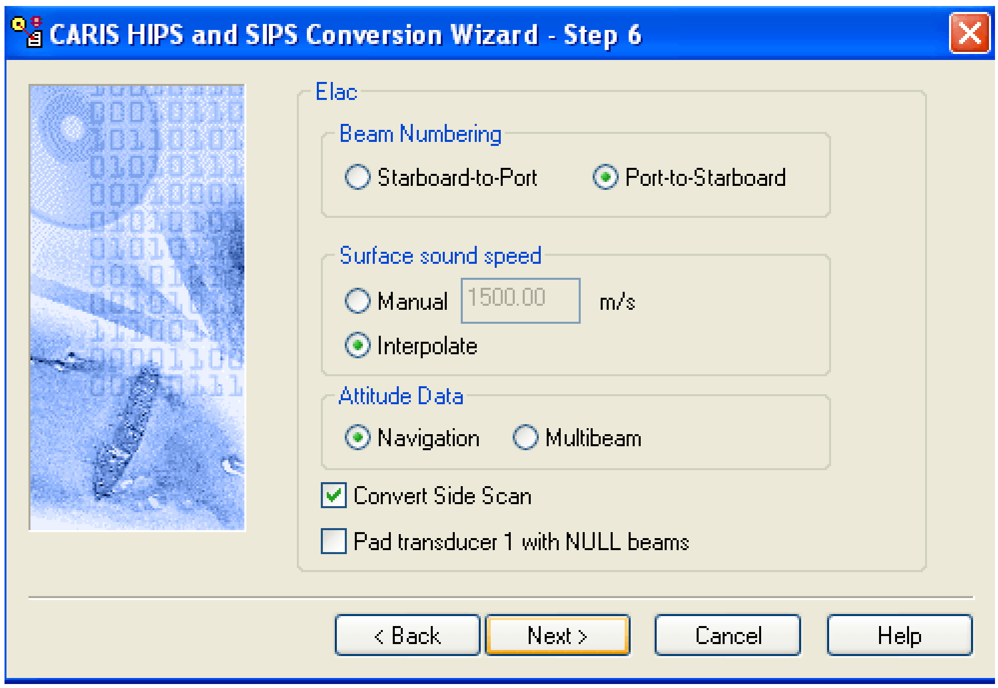
Since EIVA allows data logging from multiple devices, the *Data Source* option enables you to select the source for heading, attitude, and navigation data. The data for each device is tagged with a number (from 0 to 4), that represents one of up to five devices collecting each type of data. (If you select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

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

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

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

Elac



Beam numbering

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

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

Surface sound speed

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

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

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

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

Attitude data

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

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

Convert Side Scan

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

Survey mode

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

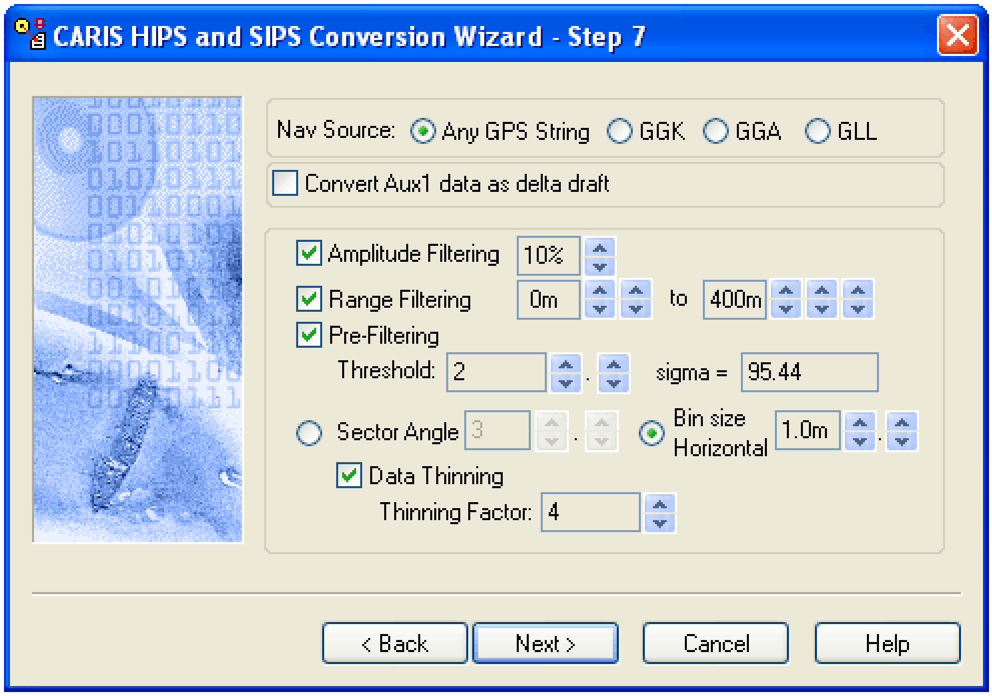
7. Click the *Pad Transducer 1 with NULL beams* check box so that the soundings associated with transducer 1 are padded to have 63 beams.
8. Click **Next**.

Furuno

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

GeoAcoustics

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



2. If you want to convert data from Aux1 as delta draft, select the *Convert Aux1 data as delta draft option.*

Amplitude Filtering

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

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

Range Filtering

The *Range Filtering* option converts soundings within a specified slant range, as determined by travel time times sound speed, regardless of the direction of travel. Soundings outside of this slant range are not converted.

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

Pre-Filtering

The *Pre-Filtering* option controls which soundings in each swath are considered for conversion, by calculating the mean depth and standard deviation within a swath sector or horizontal bin. Any soundings that fall outside the set multiple of the standard deviation are not converted.

This pre-filtering is done in two passes. The first removes the most obvious outliers. Then a second pass computes a new mean using a “better” set of soundings.

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

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

The equivalent confidence value is displayed as a percentage.

Sector Angle

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

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

3. Select the *Sector Angle Interval* option and choose a degree level (to a maximum of 10º) by clicking the up or down arrow buttons.
4. Alternatively, select the *Horizontal Bin Size* option choose an across track distance by clicking the up or down arrow buttons.

Data Thinning

Soundings that survive *Pre-Filtering* are sorted according to depth, and those that are closest to the mean depth are selected, as specified by the *Thinning Factor* you set.

Data Thinning is done on a per swath basis, based on the set of soundings that are closest to the mean value computed.

The *Thinning Factor* reduces the number of soundings on a per swath basis using a powers of two reduction control. Setting a factor of, for example, 4, will mean that the one-fourth of the soundings that are closest to the calculated mean will be converted per swath sector.

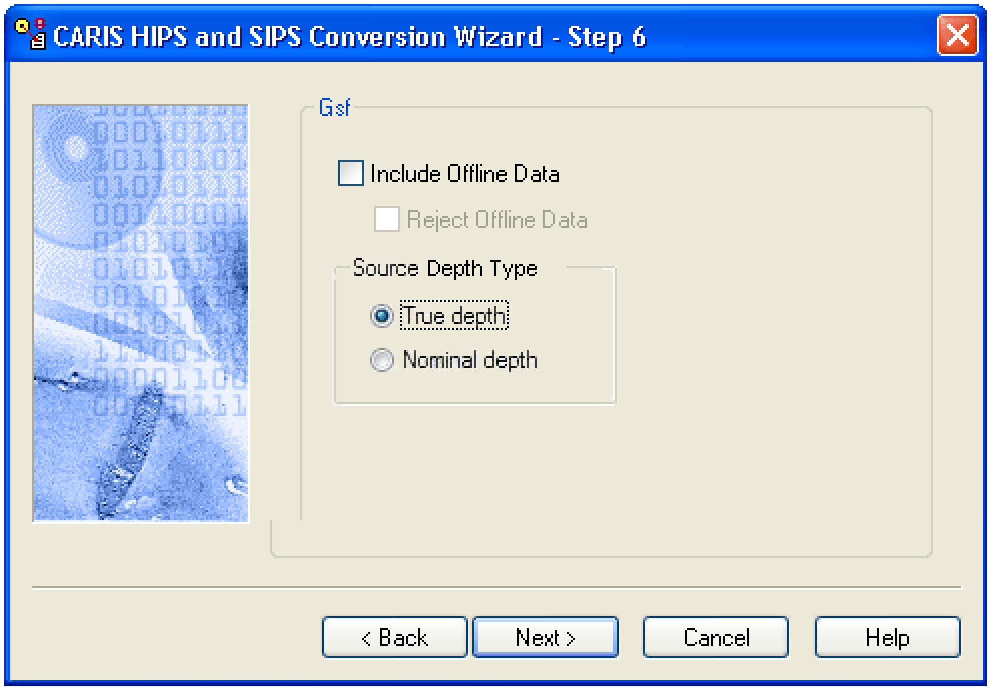
1. Select the *Data Thinning* check box to implement the thinning option.
2. Select a *Thinning Factor* value by clicking the up or down arrow buttons. Values between 2 and 256 may be set (values between one-half and $^{1}/256^{th}$ can be selected.

3. Click **Next**.

This converter reduces data to fewer than 2000 soundings in a profile, as follows:

- If data contains more than 2000, and you select the *Pre-filtering* and *Data Thinning* options, these will be applied to the data.
- If data contains more than 2000, and you do not select *Pre-filtering* and *Data Thinning* options, data is automatically reduced by the value in *Thinning Factor* field (using the last factor entered).

GSF

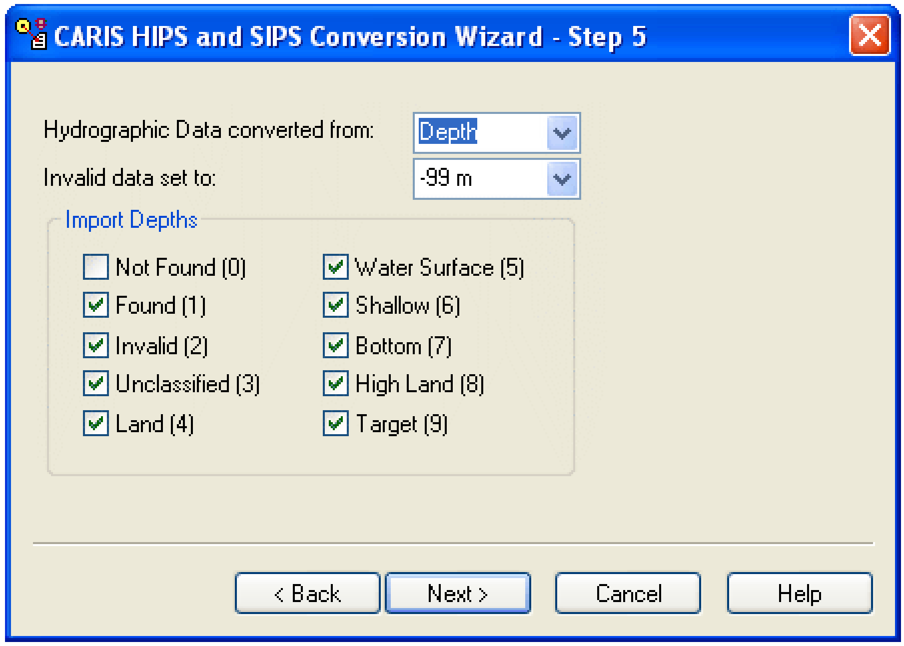


Data recorded during line turns is marked as Offline data in GSF. This data can be included or excluded during conversion. When included it can also be automatically rejected. One purpose for including the Offline data would be to maintain the same swath profile numbers in HIPS format.

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

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

Hawkeye



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

There are two kinds of Hawkeye files, with _HD.bin and _TD.bin extensions. The two kinds of files have identical data structures, but differ in contents:

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

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

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

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

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

Use the *Invalid data set to option* to apply a substitute depth of either +99m (below chart datum) or -99m (above chart datum) for data rejected by disabled beam. This data will be marked as “Rejected” and display this depth value.

2. Select -99m or +99 m as substitute depth.
3. Select the depth classes to import to HIPS. (Default sets all classes to be imported.)

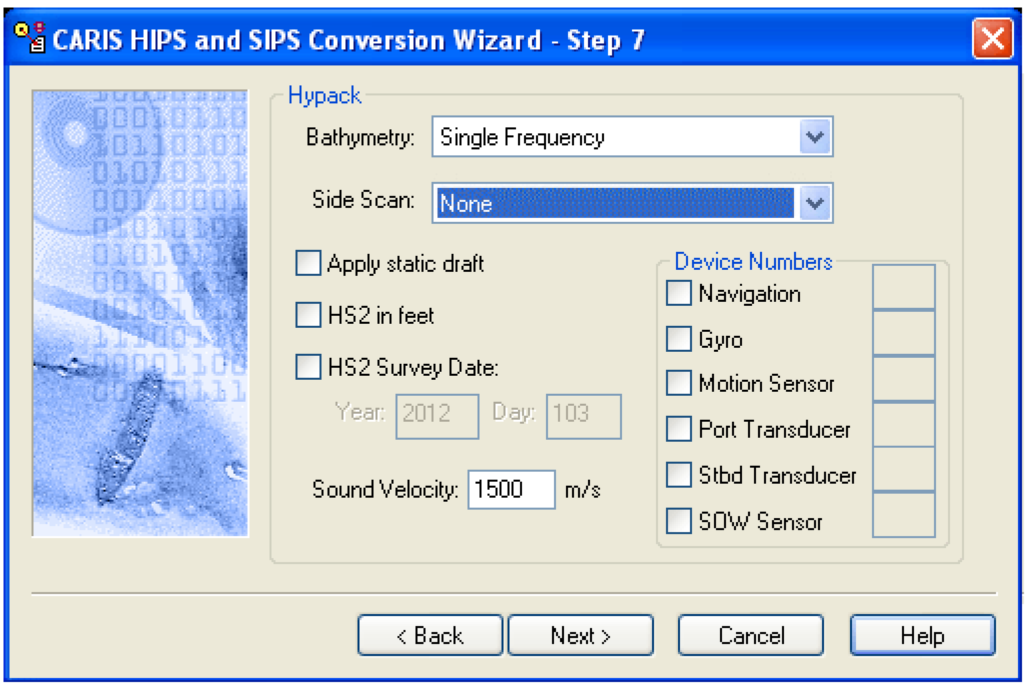
Hypack

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

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

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

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



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

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

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

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

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

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

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

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

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

4. Select *HS2 in feet* check box.

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

If you want to include another date value:

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

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

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

Device Numbers

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

The following sensors are affected by the device number setting:

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

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

Imagenex

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

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

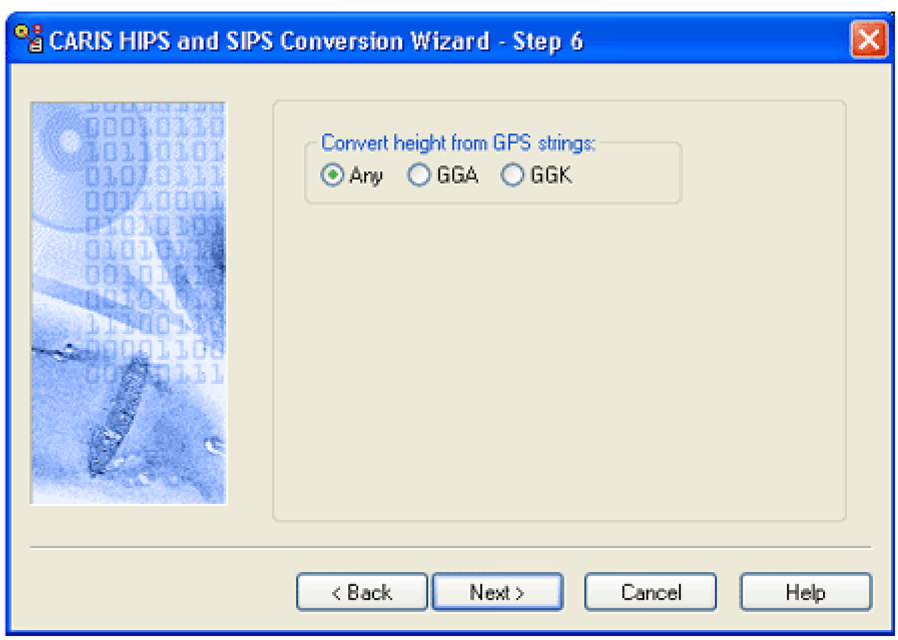
If you expect head #1 to generate no more than 200 beams, set the beam number in the HVF to 200.

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

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

To read GPS height data from Imagenex files:

1. Select the height string to be read: *GGA, GGK* or *Any*.



LADS

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

There are four sounding classifications in the LADS format:

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

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

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

There are no dialog box options for the LADS format.

LAS

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

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

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

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

MarineSonics

There are no specific options for the MarineSonics format.

Navitronics

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

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

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

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

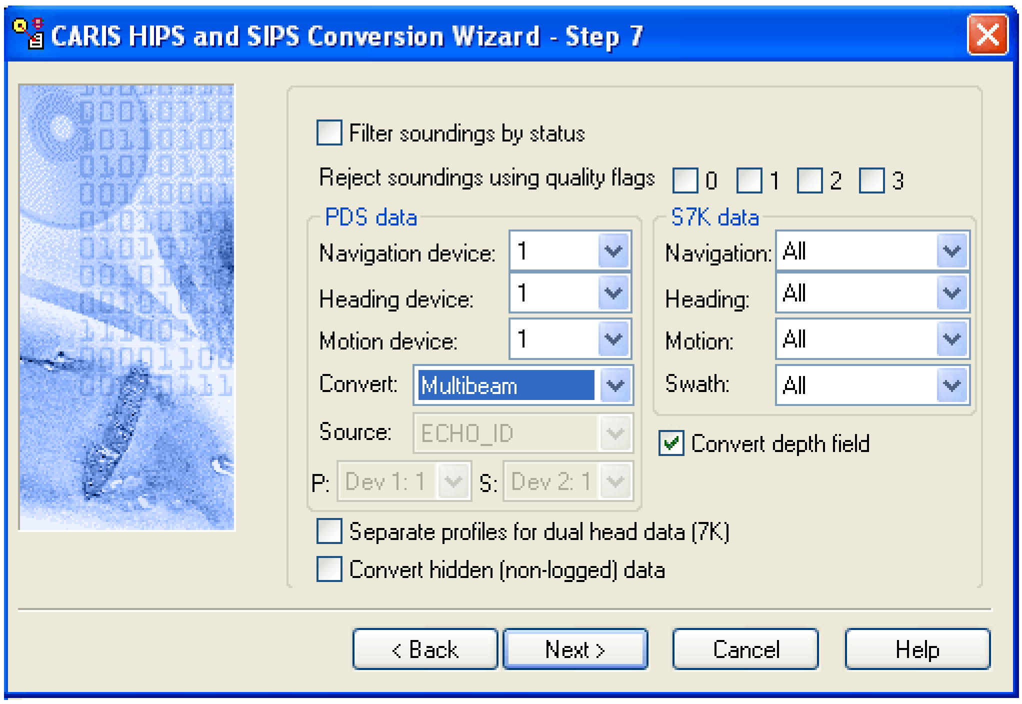
4. Select the *Scale* option if you want to average the data into 8-bit format, or select *Shift* and type the value for the first bit.
5. Click **Next**.

Qmips

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

Teledyne Reson PDS

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



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

PDS data

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

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

If converting singlebeam or dual frequency data you will need to specify which echo sounding device was used. If you care converting singlebeam data, you can choose from the ECHO_ID (raw data) or CF_DEPTH (processed data) attributes from the

Source drop-down list. If converting dual frequency data, the only source available is ECHO_ID.

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

Singlebeam

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

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

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

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

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

Dual Frequency

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

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

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

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

S7K data

It is possible to have 7004/7006 and 7027 records stored in the same file. In this case, you may want to set the options on the Swath drop-down to either “Bathymetry 7004/7006” or “Raw Detection” 7027. Leaving the setting as “All” may cause a mismatch between observed depths and SonarError records.

9. Indicate the source of *S7K data* by selecting an option from the drop-down list. The default value is “All”.

   For example, the options for *Heading* are Altitude (1016), Heading (1013) or Navigation (1015).)

In the case of dual head data stored in the Reson 7K format, the data from transducer #1 is combined with that from #2 to form a single HIPS profile.

To convert the two sets of data separately,

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

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

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

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

12. Select the Convert depth field option

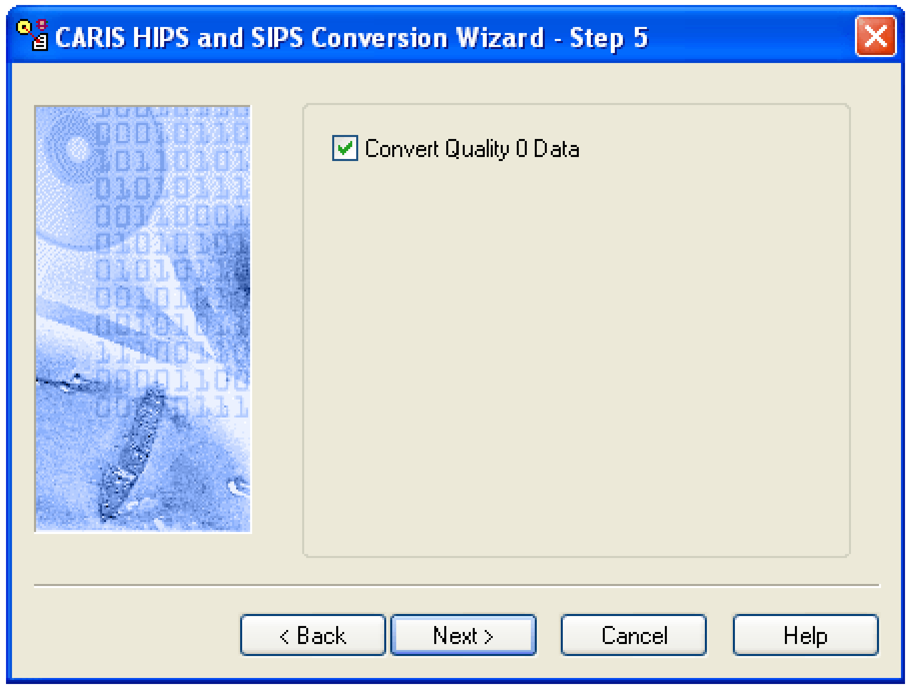
This will populate Delta Draft file in the line directory.

13. For information applying Delta Draft, see “Delta Draft” on page 172.
14. Click **Next**.

SAS

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

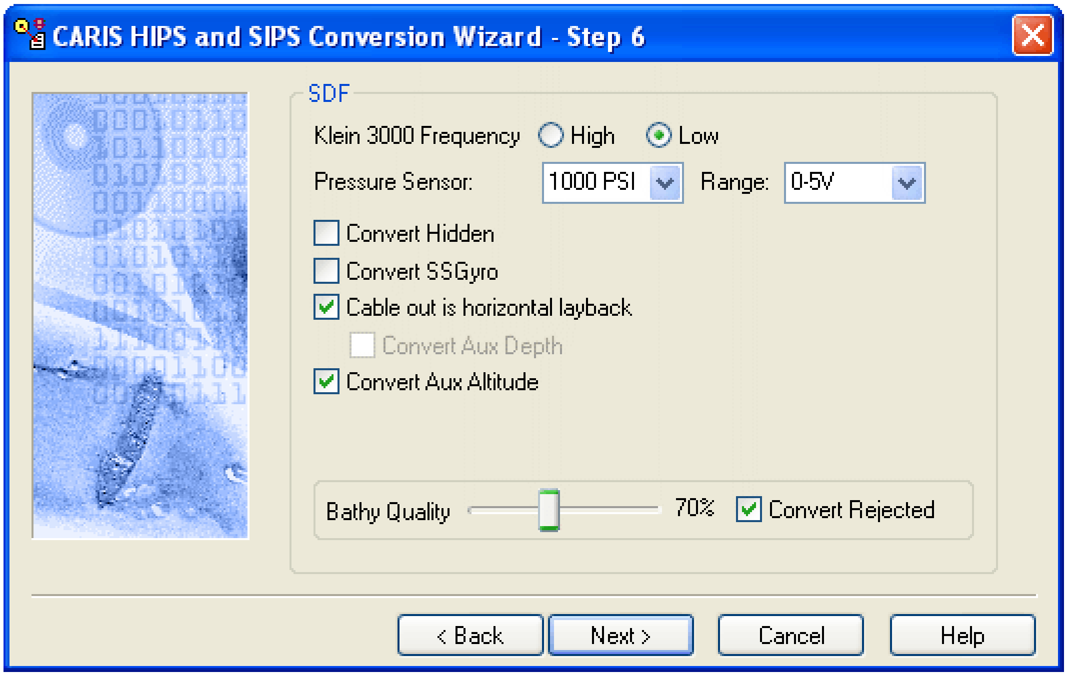
1. Select the check box to convert Quality 0 data .



Scripps

There are no options specifically for the Scripps format.

SDF



1. Select a Klein 3000 *High* or *Low* frequency.
2. Select a *Pressure Sensor* reading from the drop-down list to set the depth of the towfish.
3. Select the *Range* of voltage for the pressure sensor, either 0-5V or 1-5V.
4. Hidden data consists of repeated pings over the same area. Select the *Convert Hidden* check box to include hidden data in conversion.
5. Select the *Convert SSGyro* check box to include side scan sonar gyro data.
6. Select the *Cable Out is Horizontal Layback* option to omit sensor depth during conversion. When cable out is converted as horizontal layback, sonar depth is set to 0.0 metres.
7. Select *Convert Aux Depth* to use the data in the auxiliary depth field. This option will override the sonar depth converted into HIPS and SIPS from the depth field. (This option is disabled if the *Cable Out is Horizontal Layback* option is selected.)
8. Select the *Convert Aux Altitude* check box, to convert sonar altitude from the auxiliary altitude field in the sonar record.

Bathymetry data associated with Klein 5000 V2 data contains quality values. You can filter for these values using the *Bathy Quality* slider control.

9. Set the slider to a value between 50 and 100%.

You can retain the rejected data in the conversion to HIPS. The data will be marked as Rejected.

10. To include the rejected data in the converted dataset, select the *Convert Rejected* check box.

All sonar data is stored using 16 bits.

Seabeam

1. Make sure the *Convert Side Scan* option is checked to include side scan data with bathymetric data.
2. Select the *Shorten Line Names* check box to modify the file name to a 12-character CARIS name (YYDDD_HHMMDD).

Seafalcon

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

1. Select the thinning option, if needed.

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

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

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

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

4. Select the swath number(s) for import.

SEGY

There are no options for the SEGY format.

SHOALS

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

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

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

1. Use the mouse to select a cutoff value.

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

2. Select either the *First Pulse* check box to convert the return from the top of the vegetation canopy or the *Second Pulse* check box to convert the return from the ground.

*(Scanning Hydrographic Operational Airborne LIDAR Survey)

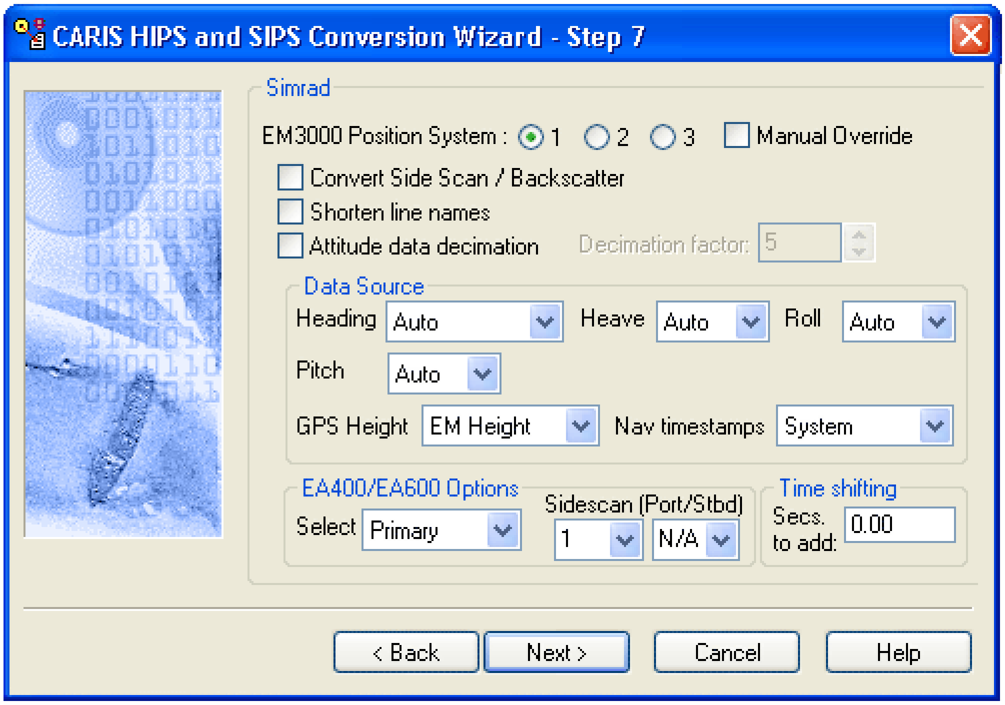
Simrad

1. If you are using the EM3000 sonar, select the channel used for the primary position.
2. Select *Manual Override* to read from the navigation system data that is marked inactive by Simrad.
3. Click the *Convert Side Scan/Backscatter* box if you want to convert this data.

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

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

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



example, if you select 10 as the *Decimation Factor*, then one out every tenth attitude record is converted.

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

Data Source

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

7. For *Heading*, select the *Auto* option to read gyro data from the active motion reference unit, or select an inactive unit by selecting *MRU1/ MRU2* options. Select the *H. Diagrams* option to read gyro from the headings datagram.
8. For *Heave, Roll,* and *Pitch*, select the *Auto* option to read data from the active motion reference unit, or select an inactive unit by selecting *AutoMRU1/MRU2* options.
9. For *GPS Height*, select a source from either the *EM Height* datagrams or the *GPS String* in the navigation datagrams.

Navigation time stamps can be selected from either the logging system or from a source specified by Simrad Installation datagram (the GPS string in the navigation datagram or the logging system).

10. Choose *System* to select the logging system or choose *Automatic* to select the timestamp specified by the Installation datagram.

EA400/EA600 Options

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

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

Time shifting

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

14. Enter the value to be added as a constant.

Spawar

There are no options specifically for the Spawar format.

SWATHPlus

1. Type a value in the *Speed of sound* field to apply sound velocity to the imported data.

Teledyne

There are no other options specifically for the Teledyne format.

UNB

1. Select one or two transducers.
2. Select the *Convert Sonar* option, if you want to import side scan data.

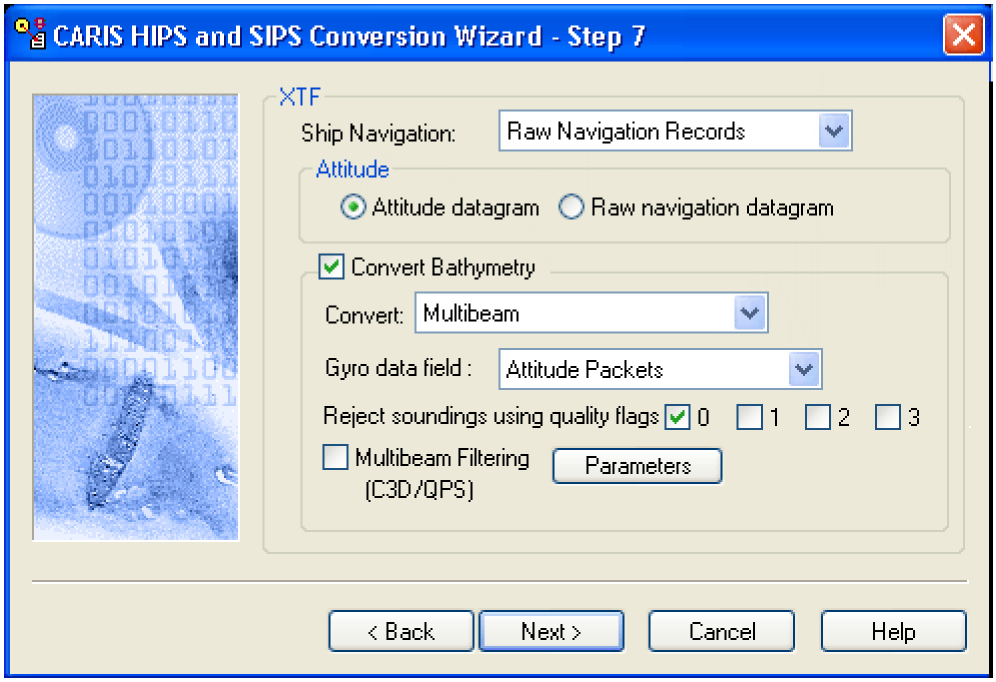
Winfrog

1. Select *Single* or *Dual Frequency*.
2. Select attitude records to be converted.

3. Specify the speed of sound, so that slant range data can be generated from the measured sounding depths.
4. Choose the navigation records to be converted.

XTF

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



Ship Navigation and Attitude

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

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

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

2. Select either Attitude diagram or Raw Navigation datagram.

Convert Bathymetry

To convert bathymetry data:

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

     If there are two channels of data, you can identify which channel contains the primary data

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

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

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

Select gyro data field

The *Gyro data field* options in the pull-down list are determined by the selection made in the Ship Navigation field. (See "Ship Navigation and Attitude" on page 126.)

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

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

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

Set reject soundings flags

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

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

If you are converting C3D or QPS data, you can apply filtering before moving to the next step. See "Multibeam Filtering" on page 128.

If you do not want to apply filtering

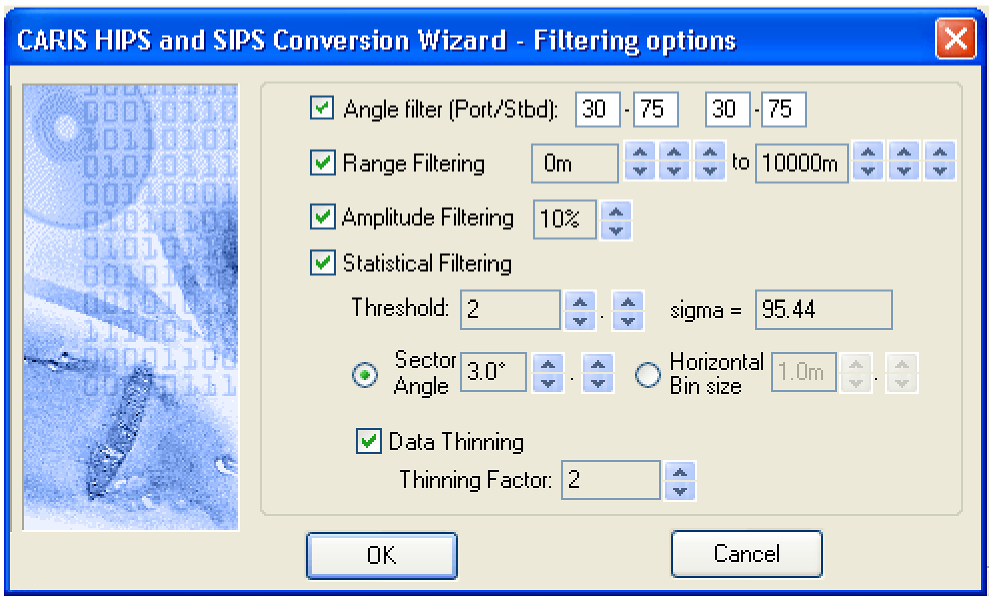
6. Click **Next** to continue to the next step.

Multibeam Filtering

To apply filters for Benthos C3D or QINSy QPS data

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

The Filtering options dialog box is displayed.



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

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

All other options are applied to QPS data only.

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

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

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

4. Select the Amplitude Filtering check box to implement this option.

5. Click the up or down arrow buttons to select a percentage value for rejecting soundings.

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

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

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

The equivalent confidence value is displayed as a percentage.

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

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

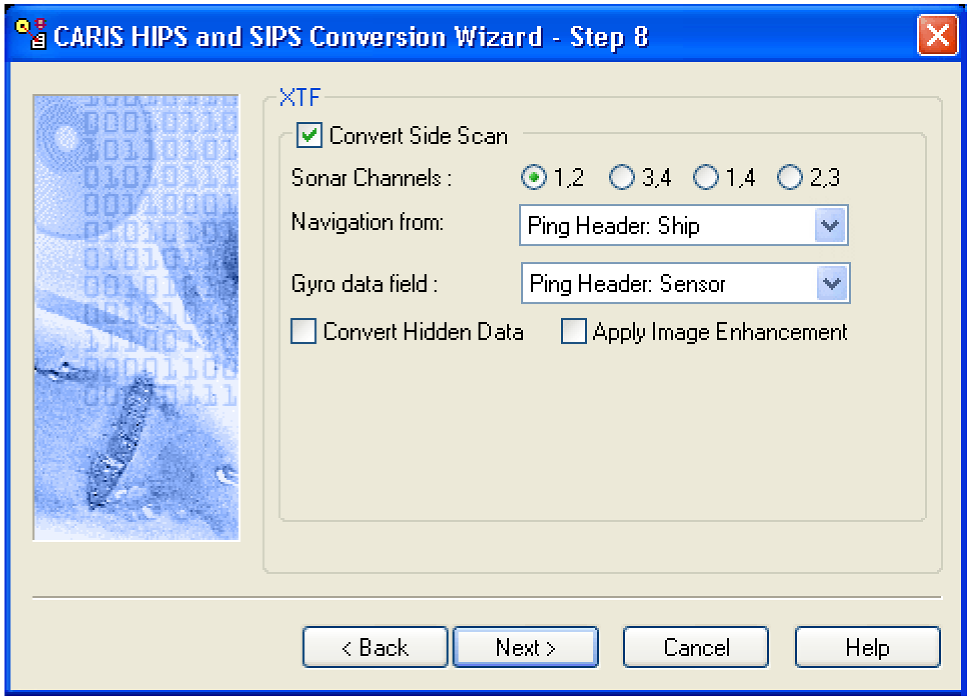
1. Select the *Sector Angle Interval* option and choose a degree level (to a maximum of 10º) by clicking the up or down arrow buttons.
2. As an alternate method, Select the *Horizontal Bin Size* option choose an across track distance by clicking the up or down arrow buttons.

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

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

Step 8: Convert Side Scan data

If you do not want to convert side scan data, click **Next** and go to "CONVERT DATA" ON PAGE 94.



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

To convert side scan data:

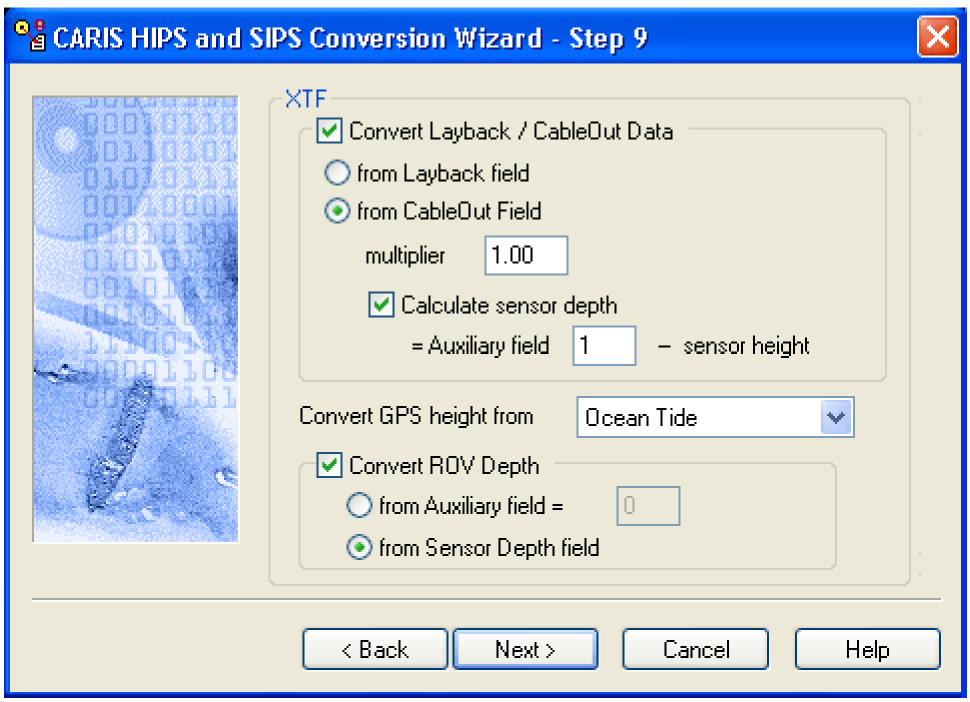
6. Click the *Convert Side Scan* check box.
7. Select one of four *Sonar Channels* pairs by clicking the radio button.
8. Select one of the following as the source for reading the *Navigation* data for the towfish:
   - Raw Navigation Records
   - Position Data Records
   - Ping Header: Ship
   - Ping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

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

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

Step 9: Convert Layback

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



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

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

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

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

GPS Height

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

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

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

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

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

The auxiliary and sensor depth fields are enabled.

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

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

Click **Next** to go to the step in the Conversion Wizard, to “CONVERT DATA” ON PAGE 94.

5

Sound Velocity Correction

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

In this chapter...

Sound Velocity Processing 134
Sound Velocity Correction 136

Sound Velocity Processing

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

Sound velocity processing is not available for all types of multibeam and single beam sonar data. Only those sonar formats that provide raw travel time and angle from the transducer are currently supported in HIPS. This includes multibeam bathymetry from XTF, Hypack, XSE and Simrad EM formats.

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

HVF

For effective sound velocity correction, the Sound Velocity section of the HIPS Vessel file must be set up to define the transducer offsets. These offsets are used in Sound Velocity Correction, together with the Sound Velocity profile data. (See "SVP" ON PAGE 47 for these sensor configurations.)

SVP file

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

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

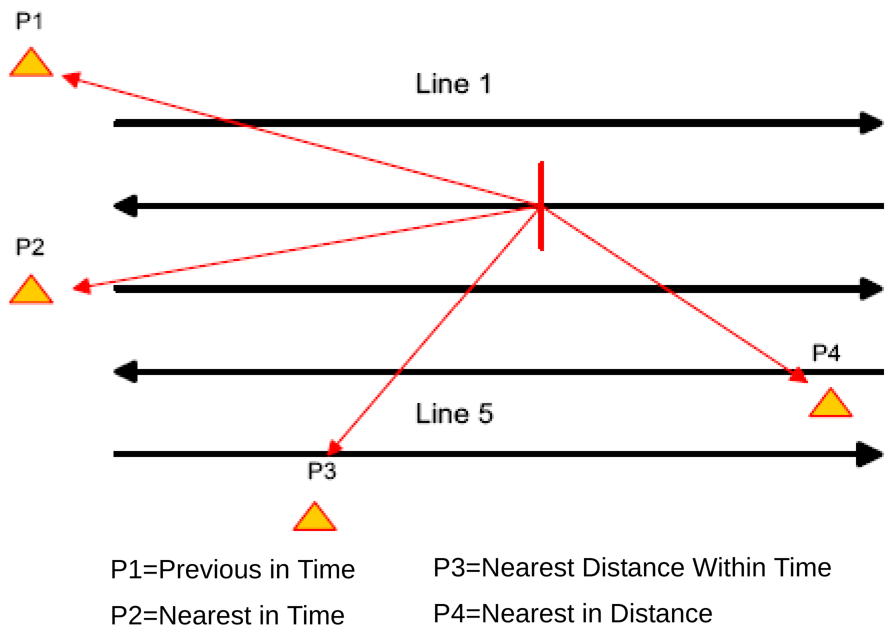
Profile selection method

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

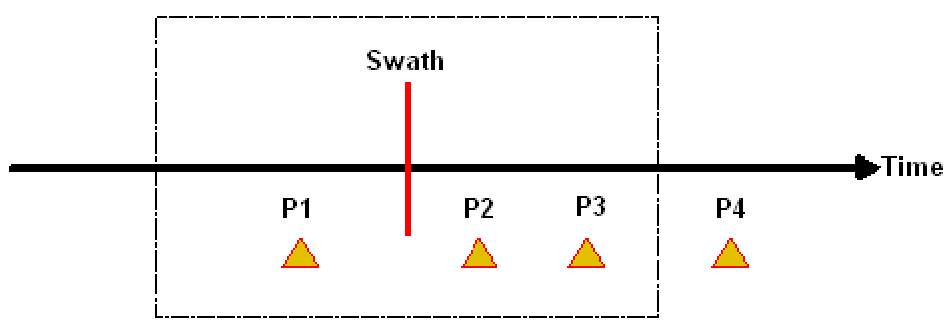
- *Previous in Time*: Select the profile with a time stamp prior and nearest to the time of the swath.
- *Nearest in Time*: Select the profile with a time stamp nearest to the time of the swath. This can either be prior to or after the time.
- *Nearest in Distance*: Select the profile with a position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

- *Nearest Distance Within Time*: Select the profile with a position nearest to the swath and has a timestamp nearest the time of the swath within a selected range.

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



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



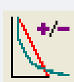
Sound Velocity Correction

The sound velocity correction process uses a ray tracing algorithm to apply the sound velocity profiles. You can use either regular attitude data or smoothed data for applying vessel motion data to the ray-tracing algorithm (see “Attitude Filter” on page 286).

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

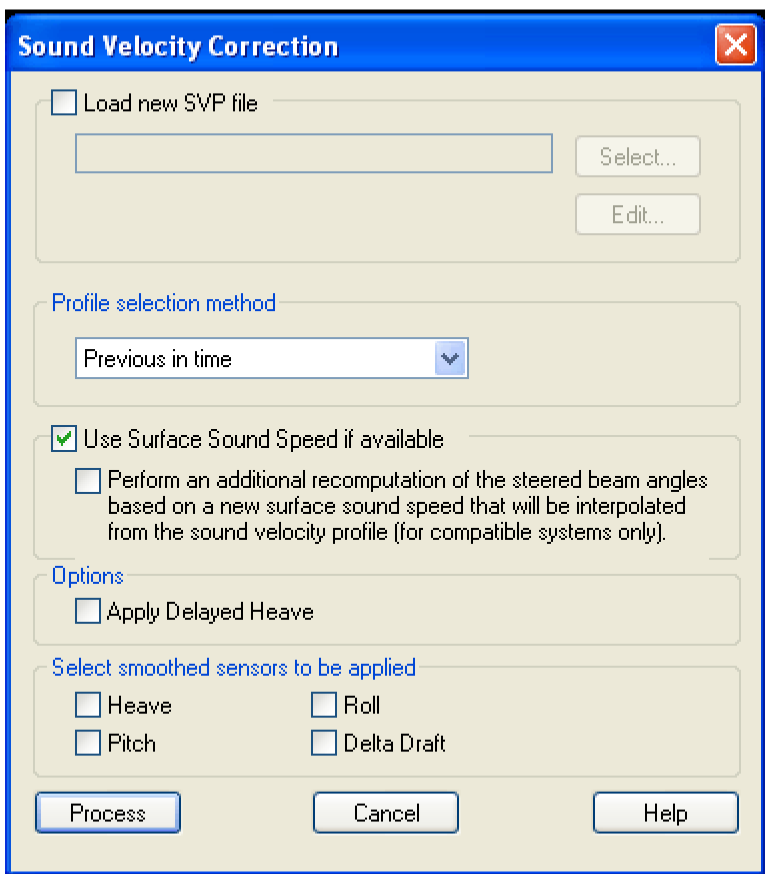
> It is recommended that sound velocity correction be applied to bathymetry before cleaning the data. Whenever SVC is applied, the Merge command should also be applied to the data

To apply Sound Velocity Correction:

Menu	Process > Sound Velocity Correction
Tool	

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

The Sound Velocity Correction dialog box is displayed.



SVC will use either the sound velocity profile from your converted data, or the last SVP applied to the data. You can choose to load another profile, and you can edit a profile to use.

To use a new profile:

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

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

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

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

If you want to edit the data, click **Edit** to open the SVP Editor. (See “Edit SVP Data” on page 140.)

Profile selection method

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

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

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

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

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

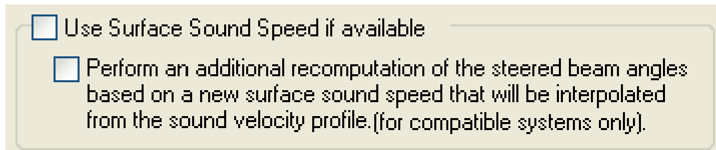
Use Surface Sound Speed

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

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

To use available SSP data in sound velocity correction:

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



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

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

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

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

Apply Delayed Heave

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

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

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

Smoothed Sensors

11. Select which smoothed sensor data is to be applied during sound velocity correction: Heave, Pitch, Roll, Delta Draft. (Sensor data can be smoothed in the Attitude Editor. See "Filter and Smooth" on page 16).

Note: Waterline values are not interpolated during SVC.

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

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

As well, the SVP Corrected field in the Selection tab will show “Yes” for any selected line to which SVC has been applied.

Query Applied SVC Profiles

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

- querying the SVC status of lines in the Selection tab
- examining the logfile.xml.
- display the SVP files as background data
- colour coding line display by SV profile

Query lines

To query lines which have been sound velocity corrected:

1. Select lines in the Display.

Selected lines will be listed automatically in the Selection tab.

2. Scroll through the Selection tab to the *SVP Corrected* column.

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

Process Log

Select SoundVelocityCorrect in the leftmost column of the Process Log. This will display profile information in the right column of the window. See also Process Log Window in the Reference Guide.

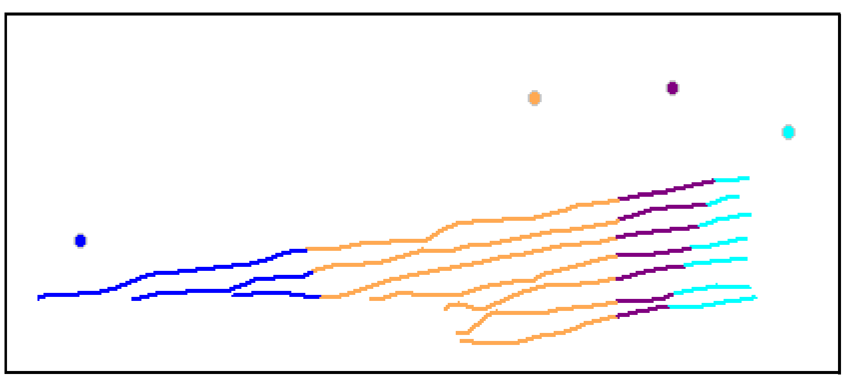
Colour by SV Profile

You can also determine which profiles have been applied to lines by turning on the Colour by SV Profile property.

To view the profiles that have been applied in SVC:

1. Select the Ship Track LInes layer in the Layers tab.
2. Open the Properties window.
3. Set the *Colour by SV Profile* value to true by selecting the check box.
4. Refresh the Display.

All the lines will display a colour for each cast, as in the following example.



Display profiles

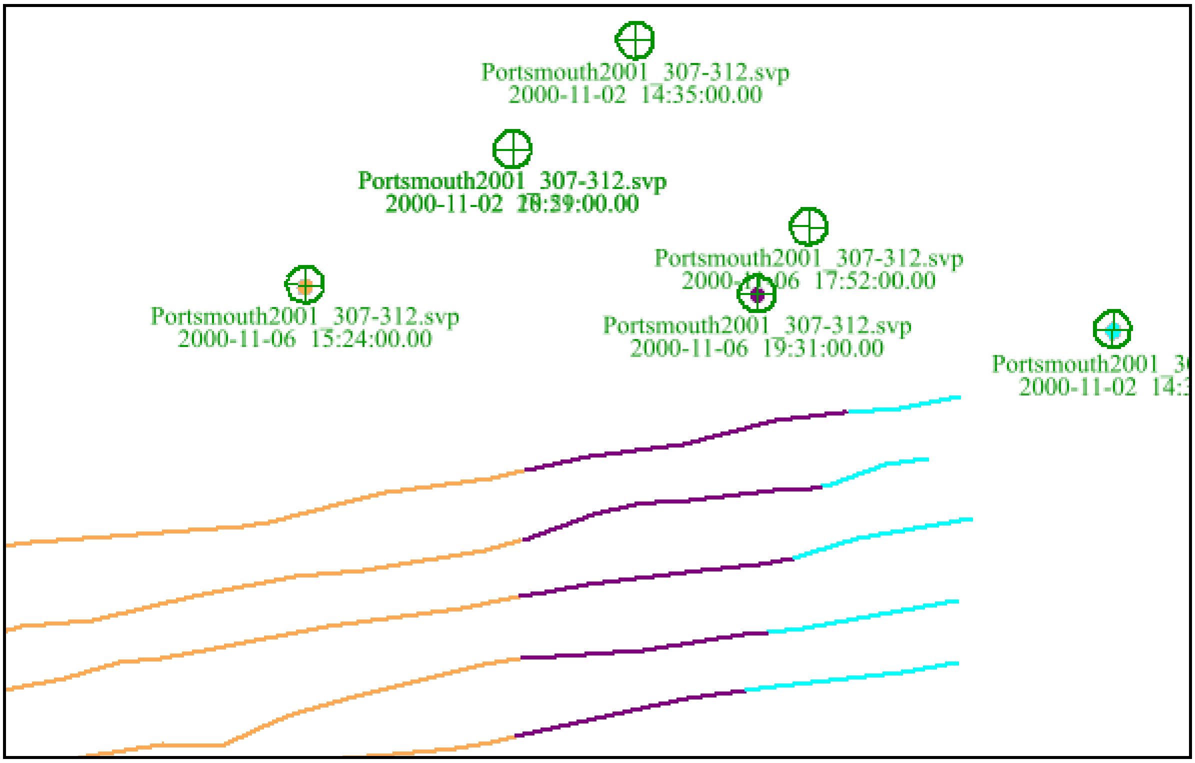
SVP files can be opened as background to a project, in order to see which profiles have been applied. To view the SVP files:

1. Select the Open Background Data command.

Menu	File > Open Background Data

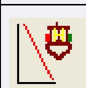
2. In the Open dialog box, browse to the location of the SVP file that was applied to the lines.
3. Click **Open**.

The open file displays the various profile locations, as in the example below.



If the SVP values are not displayed, make sure that the colour set for Tide / SVP in Tools > Options > Display tab is not set to the same colour as your Display window background.

1.

Menu	Edit > Sound Velocity Profiles
Tool	

6

Correct for Tide

Tidal data must be loaded for every track line to generate final depths. Tide files can be edited or new files created using Tide Editor.

In this chapter...

Load Tide 144
Tide Files 146
Tide Zone Files 146
GPS Tide 149

Load Tide

Tidal observation data must be loaded for every track line before the Merge process can be executed. Tide data is used to generate final depths relative to the tide datum by subtracting the tide from the sounding depth.

There are two options for tide loading:

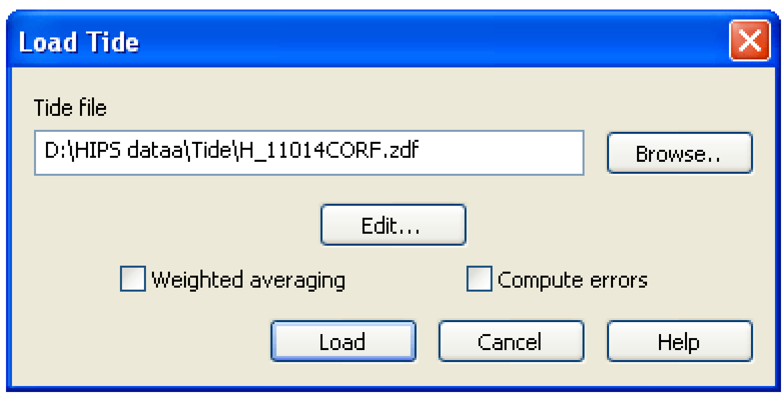
- load a *.tid file to apply tide observations from a *single* station. (For information on these files, see “Tide Files” on page 146.)
- load a *.zdf file to apply tide observations from *multiple* tide stations to a tide zone. (For information on these files, see “Tide Zone Files” on page 146.)

To apply tide observation data to selected HIPS track lines,

1. Select the track line(s).
2. Select the Load Tide command.

Menu	Process > Load Tide
Tool	

The Load Tide dialog box is displayed.



3. Click **Browse…** to choose a tide (.tid) file (single station data) or a tide zone (.zdf) file.
4. Click **Open**.

The tide file name is displayed in the field.

5. [Optional] To revise the contents either type of tide file, click **Edit** to open the Tide Editor.

If you are loading a tide zone (.zdf) file, you have the option of applying weighted averaging from multiple tide stations. Weighted Averaging gives priority to data from the closest tide station.

6. Select the *Weighted Averaging* check box to apply this option. (see “Tide Stations and Weighted Average” on page 148). If the selected tide file doesn’t contain any averaging options, this option is not available

7. Select the *Compute Errors* check box to have the Load process save computed errors to a tide error file for each line. These tide error files will then be used in computing TPU. If the selected tide file does not contain any error parameters, this option is unavailable.
8. Click **Load**.

The tide data is loaded into the selected track lines from the tide zone definition file.

Load Tide status

To determine if tide has been loaded for a line or lines:

1. Select the trackline(s).
2. Select the Selection tab.

The Selection tab will display line data for the selected line(s) in columns.

The *Tide Loaded* column shows “Yes” when tide has been loaded for the selected line(s).

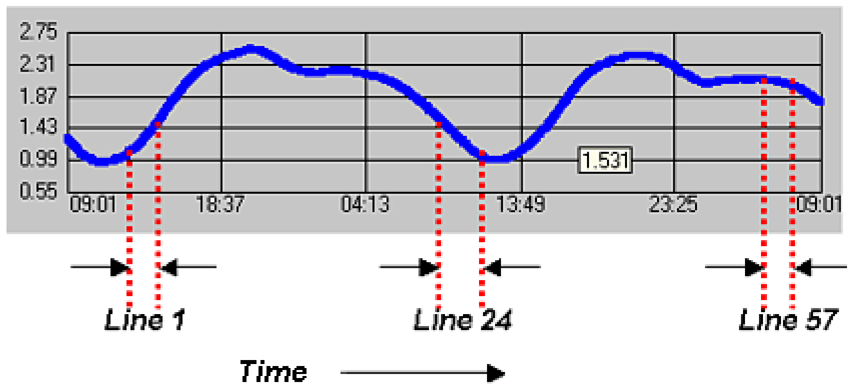
The *GPS Tide* column shows “Yes” when tide has been loaded for the selected line(s).

The *Tide Applied* column can display one of four values:

- **None**: No tide has been applied (Merge has never been run, and no ProcessedDepths exist.)
- **Observed**: The line has been Merged, and choose to apply Observed tides from loaded TID file(s).
- **GPS**: The line has been Merged, and a GPS tide file has been applied.
- **Zoned**: The line has been Merged, and a previously loaded ZDF file has been applied.

Tide Files

When tide observations from a *single* station are applied to a track line (or lines), the time extent of the line(s) is used. The illustration below shows tide values from a single station applied to three selected track lines, based on the time each line was recorded.



Tide data from a single station is loaded from files with the extension *.tid. These files can be created or edited in a text editor (such as Notepad), or with the Tide Editor.

HIPS supports a standard Canadian Hydrographic Service tide format called COWLIS, as well as two NOS/NOAA formats and the NHS file format. HIPS also supports a basic format consisting of just the required date, time, and tide values.

Track lines in areas without water level changes, such as rivers or lakes, must also have a tide file. In this case, or if tidal data cannot be obtained, a “zero tide” file—where all tide/time values are zero—is used so merging can proceed.

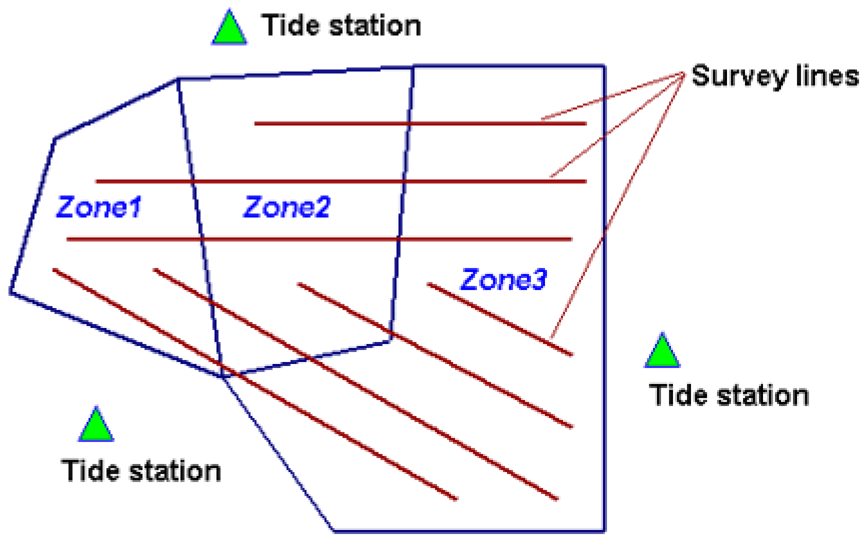
All tide files, including zerotide.tid, are located by default in...\Hips*ver*\Tide.

For more detail, see "“Tide File Formats” on page 147 in the Reference Guide.

Tide Zone Files

When tide observations from *multiple* tide stations are applied to a line or lines, tide zones are used.

Where a survey line falls within two or more tide zones, the tide data to be loaded for each part of a line is determined by the zone in which that part of the line falls (as illustrated below). Note that the tide zones do not overlap each other.



The tide data file must span the entire time frame of the track line. The time zone of the tidal observation data must match the time zone of the survey data to which it will be applied.

The output from the Load Tide function (see "Load Tide" on page 144) attaches a binary HIPS tide-time series to each track line.

The tide zone option assigns tide observations to track lines from multiple tide stations based on the time period in defined geographic zones.

Each tide zone is defined by a closed polygon with tide, time and range corrections for a primary station, plus up to three backup secondary tide stations. Tide zone should not overlap each other.

HIPS uses a Zone Definition File (with the extension *.zdf) for defining tide zone data. This is an ASCII file and can be prepared with any text editor.

For complete description of the ZDF format, see ""Tide Zone Definition Files" on page 153 in the Reference Guide .

Display tide zones

Tide zones and station locations can be displayed as background data.

Menu	File > Open Background Data

1. Select the Open Background Data command.

A standard Windows Open dialog box is displayed.

2. Navigate to the desired tide zone file (*.zdf).
3. Click **Open**.

The outlines of the tide zones, and their labels, are drawn in the Display window. The tide zones file is listed in the Layers tab of the Control window.

Tide stations and weighted average

When an average tide is calculated from multiple tide stations, the weight given to a tide station's data is inversely proportional to its distance from the vessel at that time. In other words, the further a station is from the survey line, the less weight is accorded to the data from that station.

Tide Editor

Tide Editor presents tide file information in both graphical and tabular formats. You can use the editor to edit an existing file or to create a new tide file to apply to survey lines before the Merge process.

Tide Editor is a separate application that launched from the HIPS and SIPS interface.

GPS Tide

The Compute GPS (Global Positioning System) Tide function provides an alternative to normal tidal observation for reducing soundings to the sounding datum. A single sounding datum height, or a datum model file with a grid of datum heights, can be applied with the GPS height during the computation.

As of HIPS and SIPS 8.0.3, any open surface layer or TIN layer can be selected as the input “model” for computing GPS tide.

For details on sounding datum models and a sample *.info file, see “GPS Tide format” on page 156 and “INFO file” on page 160 of the Reference Guide.

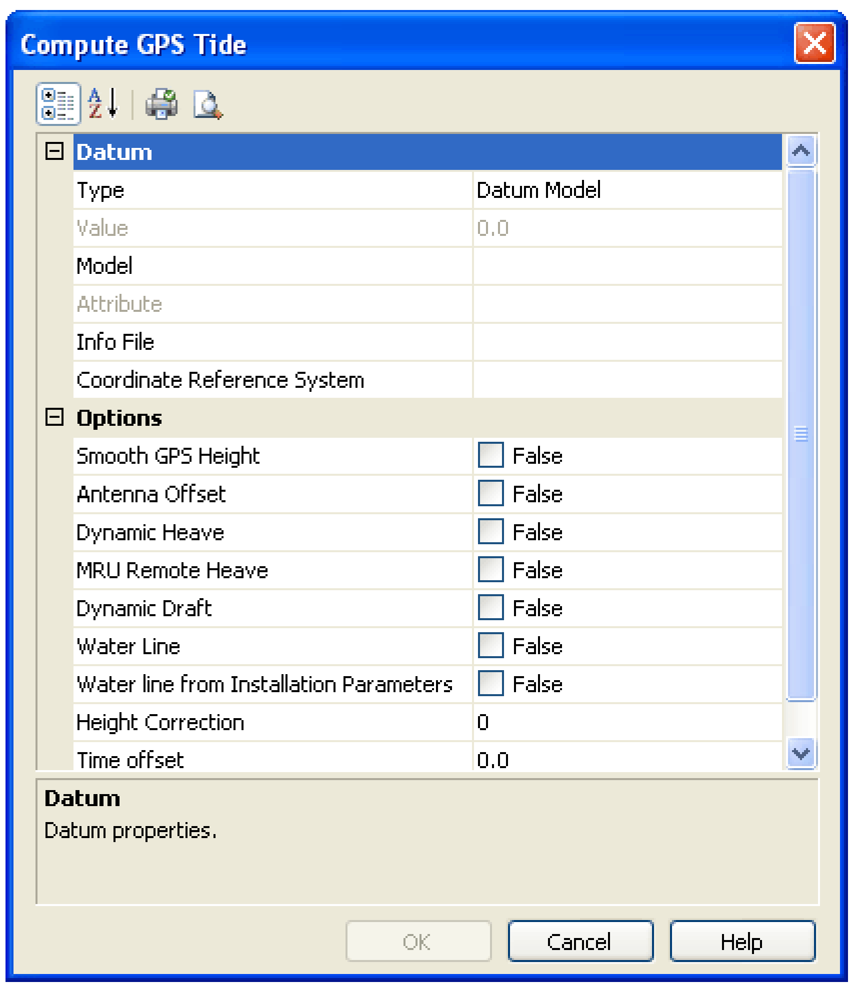
To compute GPS tide:

Menu	Process > Compute GPS Tide

1. Select a track line or group of track lines.
2. Select the Compute GPS Tide command.

The Compute GPS Tide dialog box is displayed.

.



When a surface is used as the datum model, an elevation attribute must be set. This can be selected from a drop-down list in the *Attribute* field.

When parsable ASCII file is used as the datum model, but is not recognized as a known format, you must also open a *.info file that will control the parsing of the ASCII data. As well, a coordinate system must be selected. The *Info File* and *Coordinate Reference System* fields are enabled so that you can enter this information.

3. In the *Type* field, select either Single Value or Datum Model as the type of datum to be applied.
4. If Single Value is selected as *Type*, enter a single datum height value in the *Value* field to apply a single distance from the ellipsoid to the antenna.
5. If Datum Model is selected as *Type*, use the drop-down list in the *Model* field to select a currently open surface, or to select <Browse> and open a model file. The field will display the surface name, or the filename of the model file.

   If you have selected a surface as the datum, select an appropriate attribute from the list in the *Attribute* field.

   If you have selected a datum model file of an unknown format, click Browse button in the *Info File* field to select the appropriate .info file. This field is disabled if the selected file is a known format with an available info file (which will be applied by default).

   In the *Coordinate Reference System* field, click the Browse button to open the Select Projection dialog box and set the relevant projection.
6. Select the check boxes or type in values to apply the following options during the process:

Smooth GPS Height	Set to “True” to apply smoothing to GPS height.
Dynamic Heave	Set to “True” to apply dynamic heave (either regular vessel heave, or delayed heave, if it exists).
MRU Remote Heave	Set to “True” to apply remote heave resulting from vessel roll/pitch on an offset-mounted MRU.
Dynamic Draft	Set to “True” to apply dynamic draft (as a result of interpolating the draft table in the HIPS Vessel File (HVF), or as stored in HIPS as time series data).
Water Line	Set to “True” to apply the waterline offset in the HVF.
Water Line from Installation Parameters	Set to “True” to apply the waterline value from the InstallationParameters.xml file generated by the converter.
Height Correction	Type an amount as a static offset to the GPS antenna.
Time Offset	Type a value in seconds to apply as a time offset.

7. Click **OK** to apply.

The GPS Tide is calculated for the selected line(s). The settings in the dialog box are saved in a GPS Tide file in the line directory.

You can open and view the GPS tide data in the Attitude Editor ("Attitude Editor" on page 9).

Sounding datum model files in gridded binary (.bin) or ASCII (.xyz) format can be opened in HIPS and SIPS as background data. See "Open Background Data" on page 46 in the Reference Guide.

7

Compute TPU

Total Propagated Uncertainty (TPU) is used to assign a horizontal error estimate (HzTPU) and a depth error estimate (DpTPU) to each sounding.

In this chapter...

Total Propagated Uncertainty 154
Compute TPU 156
TPU Filtering 159
Apply Filter 161
Load Error Data 163

Total Propagated Uncertainty

Total Propagated Uncertainty (TPU) is derived from a combination of all individual error sources. The following errors (among others) contribute to TPU:

- nav/gyro/heave/pitch/roll/tide errors
- latency error estimate
- sensor offset error estimates
- individual sonar model characteristics

TPU is essential for the following functions:

- creating S-44 or S-57 compliant datasets (see "TPU Filtering" on page 159)
- calculating Bathymetry Associated with Statistical Error (BASE) Surfaces weighted by uncertainty (see "Uncertainty Weight" on page 189)

Error values are located in the vessel file for the survey and the DeviceModels.xml file in …\HIPS\System. Other errors are directly entered in the Compute TPU dialog box.

The following sonar error-models in the DeviceModels.xml have been tested by the University of New Hampshire (UNH).

- Atlas Hydrosweep DS
- Elac Nautik 1180
- Seabeam 2112
- Teledyne Reson SeaBat 8101
- Teledyne Reson SeaBat 9001
- Teledyne Reson SeaBat 9003
- Teledyne Reson SeaBat 8125
- Simrad EM300
- Simrad EM1000
- Simrad EM1002
- Simrad EM3000
- Simrad EM3000D

Other sonars listed in the file may make assumptions that could produce less rigorous results than these error models supplied by UNH.

The Compute TPU function must be performed after Sound Velocity correction. Whether it is applied before or after Merge will depend on the workflow you are following. When you compute TPU it will always be done on the Observed depths, therefore tide and draft offsets will not affect the values.

Compute TPU

The Compute TPU (Total Propagated Uncertainty) command applies Horizontal TPU (HzTPU) and Depth TPU (DpTPU) values to each sounding in the observed depths.

DpTPU is the uncertainty associated with the depth value of a sounding. It is a random error and is scaled to 95% (1.96 sigma).

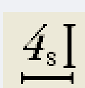
HzTPU - The uncertainty associated with the position of a sounding. It is a random error and is scaled to 95% (1.96 sigma).

The HzTPU and DpTPU values are stored line-by-line with indexing to profiles and beams. You can view the specific error values for individual soundings by querying the soundings in Swath, Single Beam, or Subset Editor. The HzTPU and DpTPU values are displayed in the Selection tab.

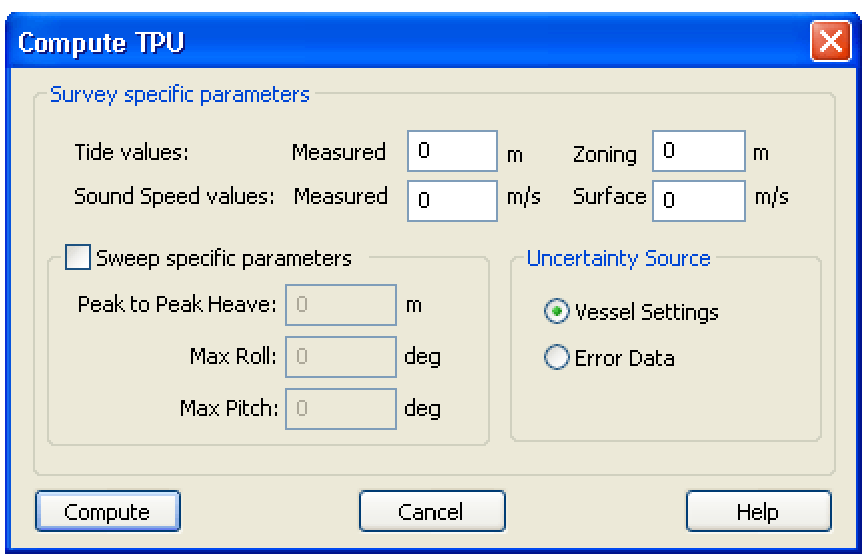
Static errors (such as those related to sensor position) that are used to compute the TPU are read directly from the project's vessel file and from the DeviceModels.xml file. These errors are directly entered in the TPU computation.

Dynamic errors due to tide and sound velocity values, which can change from time to time, are entered via the Compute TPU dialog box.

1. Select a track line in the Display window so it is highlighted.
2. Select the Compute TPU command.

Menu	Process > Compute TPU
Tool	

The Compute TPU dialog box is displayed.



3. In the *Tide Values* fields, enter the vertical uncertainty values due to tide. These will be applied to all lines being processed.

- The *Measured* offset is the error value for the tide station. It is equivalent to the standard deviation of the TideGauge measurements.
- The *Zoning* offset is the vertical uncertainty value in range calculation for a tide zone file. (HIPS does not model the TPU for the tide zone. Instead a single user-defined single value may be used here.)

4. Enter error offsets for *Sound Speed Values*.
    - The *Measured* offset value is used in computing range error to compensate for inaccuracies in SVP measurements.
    - The *Surface* offset value is used to account for errors in surface sound speed measurements that alter the beam angle. (This parameter is only applicable to systems that require accurate surface sound speed measurements.)

Transducer motion offsets for sweep surveys are entered in the *Sweep specific parameters* fields.

5. Type the estimated observed heave offset in the *Peak-to-Peak Heave* field.
6. Type the roll values for the transducers in the *Max Roll* field.
7. Type the pitch values for the transducers in the *Max Pitch* field.

You can use previously loaded RMS data in the TPU computation instead of data from the vessel file.

8. Select *Vessel Settings* to use vessel data. (If you select Vessel Settings and RMS data has been loaded it will not be used.)
9. Select *Error Data* to use RMS data.

    If you select *Error Data* and no RMS data is available, vessel settings will be used.

    If you select *Error Data* and one or more of the RMS data attributes are not available, then a zero value will be used for the missing attribute.

Note: If you choose to use RMS data in computing TPU, and true heave RMS has been loaded (via the Load True Heave process), Compute TPU will override the Down/Heave RMS values loaded from the Load Error Data process, and use the true heave RMS data loaded with Load True Heave.

10. Click **Compute**.

The results are displayed in the Output tab of the Worksheet window. A log file is also created showing the results.

TPU Filtering

TPU filtering can be applied with one of the HIPS and SIPS automatic filters. Flittering parameters are set in the Set Filters dialog box and applied using the Apply Filters command.

The filter compares the HzTPU and DpTPU values for each sounding against the depth and horizontal error limits for a specific S-44 survey order, or S-57 zone of confidence (CATZOC) attribute. All soundings with HzTPU and DpTPU values outside those limits (set by International Hydrographic Organization (IHO) standards) are rejected (or accepted).

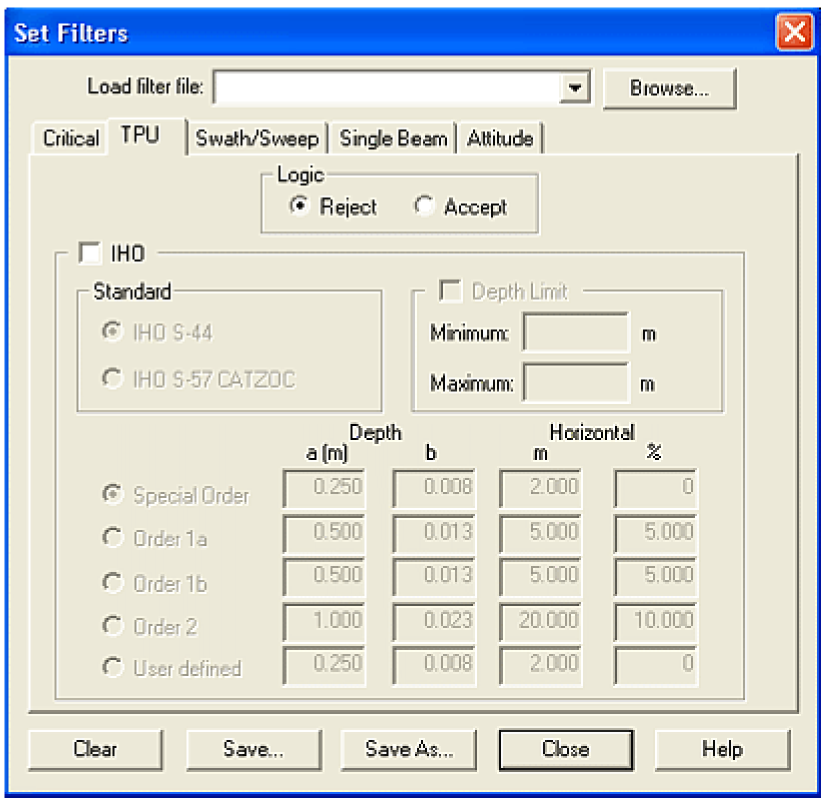
To set TPU filters:

1. Open a project.

Menu	Tools > Set Filters
Tool	

2. Select the Set Filters command.

The Set Filters dialog box is displayed.



3. Select the TPU tab.

The TPU filter fields will display any values previously entered. Parameters set in these fields are retained until you change them or load a saved filter file.

4. [Optional] Select an existing HIPS Filter File from the list or click **Browse** to select a file.

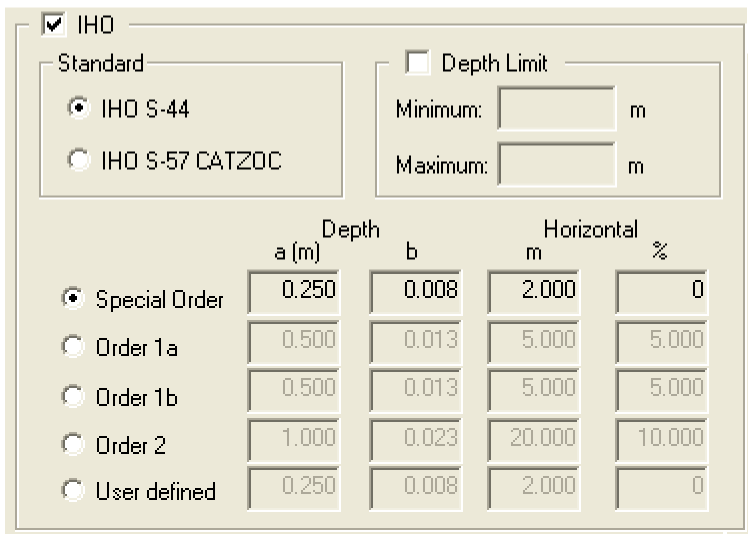
If you load a filter file, all the values used in that file are shown in the fields. You can change any of the values.

5. Select either a *Reject* or *Accept* logic command to apply to the filtered data.
6. Select the IHO check box.

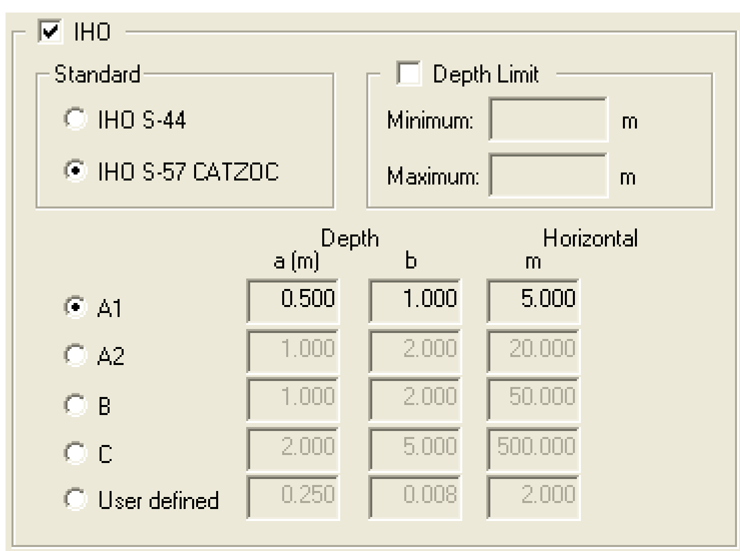
This will enable the TPU filter so the values will be applied to data when the Apply Filters command is used.

7. Select the standard to apply, either *IHO S-44* or I*HO S-57 CATZOC,* and complete data in the appropriate fields.

Depending on which you select, the dialog box is refreshed to display the options associated with that standard.



S-44



S-57

Each survey order or zone of confidence contains the following fields:

- *a*: constant depth error (i.e. the sum of all constant errors)

- *b*: factor of the depth dependent error
- *m*: position limit for horizontal errors
- *%*: percentage of depth used to calculate horizontal error (S-44 only).

Depth error limit

To calculate the error limits for depth accuracy, the constant depth error (a) and the factor of the depth dependent error (b) are combined with depth (d) and the depth dependent error (b * d) in the following formula:

Error limit for depth accuracy = $\pm\sqrt{a^2 + (b \times d)^2}$

When the filter is run, soundings with a vertical uncertainty outside the error limit for the survey order or zone of confidence are rejected (or accepted).

Horizontal error limit

To calculate horizontal accuracy, the limit for horizontal error (m) and (if applicable) the depth percentage (%) for the selected survey order or zone of confidence are compared against a sounding's HzTPU. Soundings with a horizontal uncertainty outside the limits are rejected (or accepted).

Values entered in the TPU filter will be retained when the dialog box is closed. However, you can also save the settings in a HIPS Filter File (.hff) for use on other data.

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

Your filter can now be applied to your data using the Apply Filters command.

Apply Filter

To apply the filtering values set in the Set Filters dialog box:

Enable the TPU filter by selecting the IHO check box in the

1. Select either the *ISO 2-44* or *S-57 CATZOC* options.

The IHO frame of the TPU tab is refreshed to display the S-44 survey order options (Special Order, Orders 1a, 1b, 2) or the S-57 zones of confidence (see above).

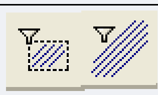
You can apply filtering to a selected depth range by entering the maximum and minimum soundings for that range. Leave these fields empty to apply filtering to every sounding along the track line.

2. Select the *Depth Limit* check box to apply the filters to a depth range.

3. Type depth levels in the *Minimum* and *Maximum* fields to set a range for the filters.
4. Select a survey order or zone of confidence option.

All fields in the *User defined* option are activated when this option is selected.

5. Type *Depth* and *Horizontal* values.
6. Run one of the Filtering commands.

Menu	Tools > Apply Filters > Selected Lines/All Lines
Tool	

For other automated filters, see "ATTITUDE FILTER" ON PAGE 286, "AUTOMATIC FILTERING" ON PAGE 325, "SINGLE BEAM FILTERING" ON PAGE 105, and "PROTECT CRITICAL SOUNDINGS" ON PAGE 304.)

Load Error Data

Use this process to override error values in the HIPS Vessel File with real-time error values. The Load Error Data process generates a HIPS RMS (root-mean-square) file which can be used in TPU computations.

Applanix POS MV files and Applanix POSProc files contain real-time attitude, navigation and RMS error values recorded during survey. This means that different error values are recorded for different conditions, resulting in more precise error estimates than the static error values recorded in the HIPS Vessel File.

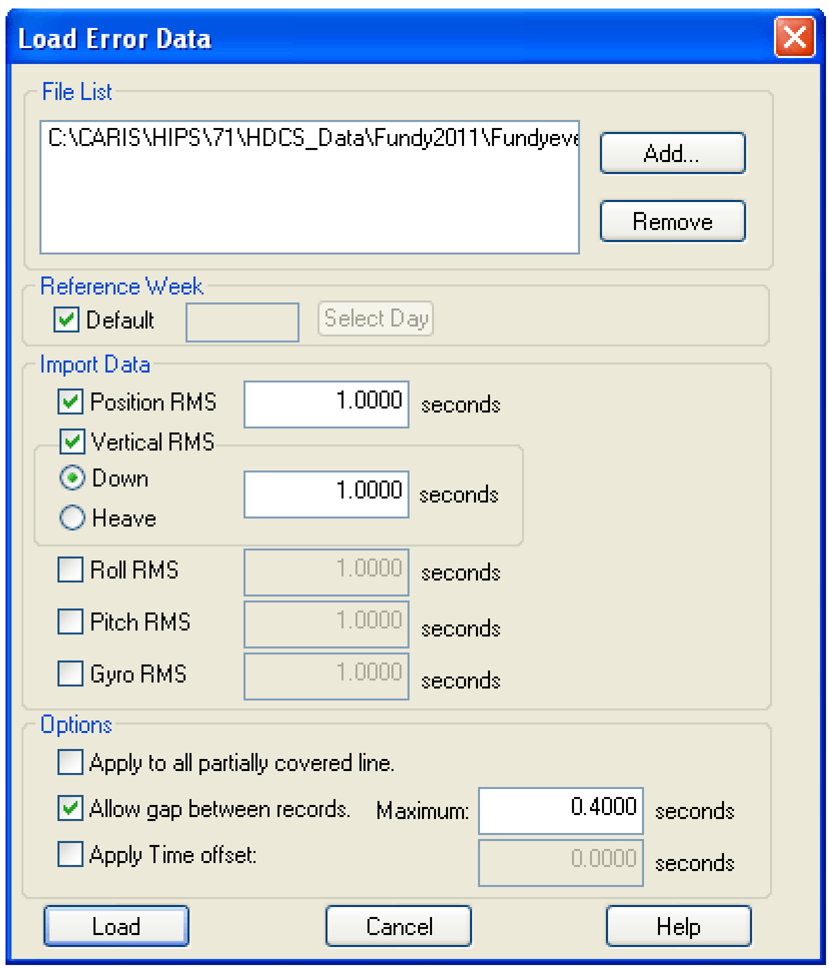
The files must be timestamped with valid GPS entries to fix the time of the data, and must contain the Julian date of the day the data was acquired in the form: YYYY-JJJ-filename or filename YYYY-JJJ

1. Select a track line in the Display window.

Menu	Process > Load Error Data

2. Select the Load Error Data command.

The Load Error Data dialog box is displayed.



3. Click **Add** to select files. You can load either POS MV files OR load POSProc files, but not both at the same time.
4. To remove a file from the *File List* field, select the file and click **Remove**.

Reference Week

Reference weeks start on Sunday at midnight and are time-stamped in GPS seconds, starting at 0. If the survey week runs from, for example, Wednesday to Tuesday, the start day for the survey week can be set, so Sunday is “rolled over” and data for the full survey week is applied.

5. [Optional] To accommodate a GPS week rollover event, click **Select Day** and select the start date from the calendar. This will enter the Julian date.

Import Data

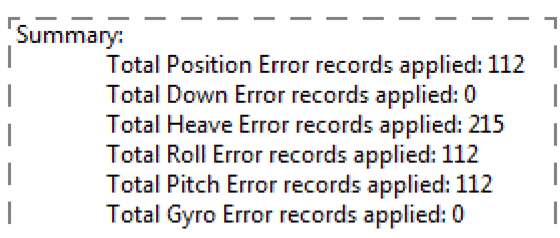
Choose which error records in the Applanix files you want to import into the line by clicking on the appropriate check boxes. Then enter an output interval (in seconds). This value determines how frequently the data is written to the HIPS RMS file.

6. Select Vertical RMS to load vertical RMS data, then select one of the following alternatives:
   - *Down* to load Down RMS data stored with Group 2 records, or
   - *Heave* to load Heave RMS data stored with Group 111 records.
7. Select *Roll RMS*, *Pitch RMS*, *Gyro RMS* to load roll error, pitch error and gyro error values from the Applanix files to the track line.
8. Type an output interval.

Options

9. The *Apply to all partially covered line* option will load the error data even if the data does not cover the entire line.
10. Select the *Allow gap between records* check box to limit the interval allowed between consecutive data records in the file. If an interval exceeds the amount specified in the *Maximum* field, the data will not be loaded and the process will stop.
11. If the time stamp does not match the time stamps in the project, you can select the *Apply Time offset* option and enter a time value (in seconds).
12. Click **Load**.

The Output tab displays a summary of the records loaded.



SBET files loaded

Menu	Process > Detailed Line Query

Use the Detailed Line Query function to confirm which files have been loaded:

1. Select a line or lines in the Display window.
2. Select the Detailed Line Query command.

This opens the Detailed Line Query window, displaying the records for the selected line(s) in columns.

3. Scroll through the table to the *SBET RMS File* column.

The Load Error Data process generates a HIPS RMS file. If this RMS file exists, the name of the SBET file used to generate it, will be displayed in this column. If no RMS file exists, the text "*** Not Loaded ***" is displayed.

> Only newly loaded SBET RMS data stores the file name. Data loaded previously will only show a blank field.

See also "Detailed Line Query" on page 274.

8

Merge

The Merge process converts along track/across track depths into latitude, longitude, and depth by combining the ship navigation with the horizontal and vertical offsets from the HIPS vessel file. This geographically references the sounding position and depth.

In this chapter...

Merge Process 168
Apply Merge... 170
Delta Draft... 172

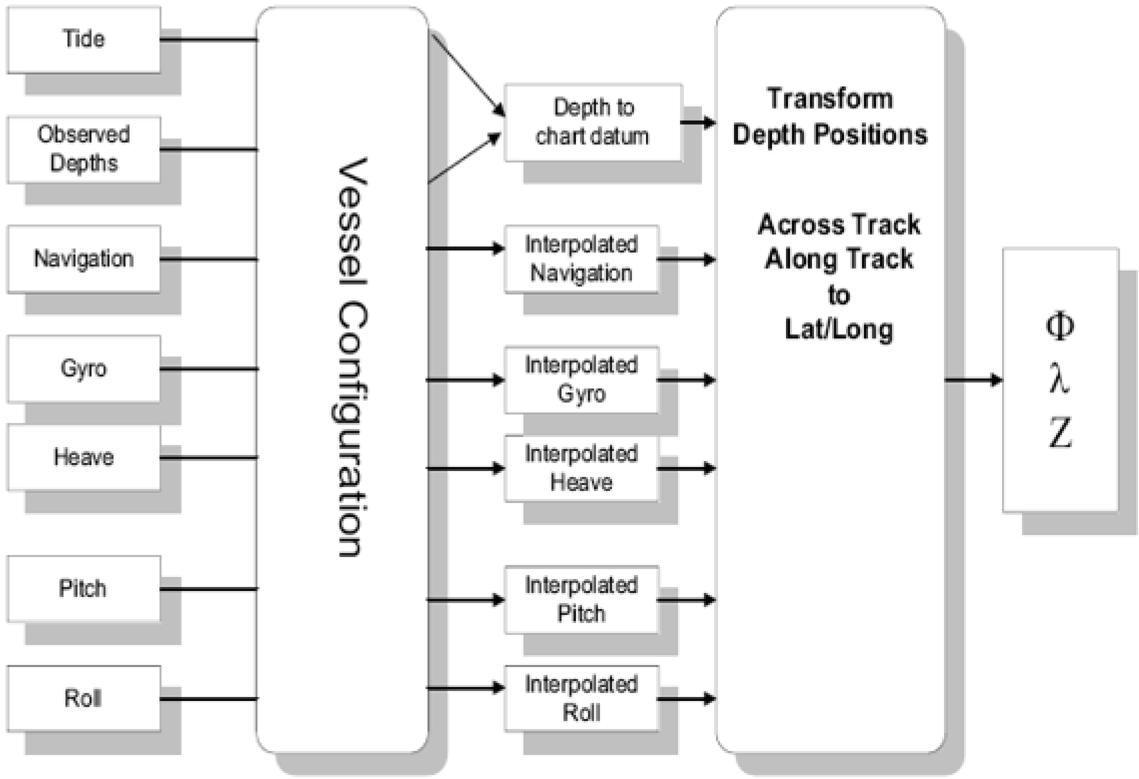
Merge Process

Once Sound Velocity Correction has been applied, (see "Sound Velocity Correction" on page 136) the final sounding depth and geographical position needs to be calculated. This is done in the Merge process.

Trackline data must be Merged before BASE surfaces can be created and before surface cleaning can be performed.

Merge takes into consideration pertinent vessel configuration offsets and the following values, many of which are set in the HIPS Vessel file:

- Navigation
- Gyro - based on the settings in the HIPS Vessel file.
- Dynamic draft - based on the settings in the HIPS Vessel file.
- Smoothed sensor data
- Waterline (if not applied in the SVC process)
- Motion data: Heave, Pitch, and Roll (if not applied in the SVC process)
- Tide or GPS tide
- Delta draft
- Refraction coefficients
- Observed depths



The following operations take place when merging:

- Recorded sensor information is compensated for constant time errors as noted in the vessel file.
- For each time-tagged depth record, the position for the centre of the swath profile is calculated based on the interpolated position fix at that time.
- For each depth record, the position of the sounding is calculated, based on the profile centre position and any gyro/heave/pitch/roll corrections that are appropriate (this is echo sounder system dependent).
- Tide is interpolated and applied to each depth record.
- Draft is applied.

A processed depths file is created for each line. This file contains the final computed geographic position for each depth record.

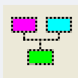
If any of these offsets or parameters is changed the data must be Merged again.

Apply Merge

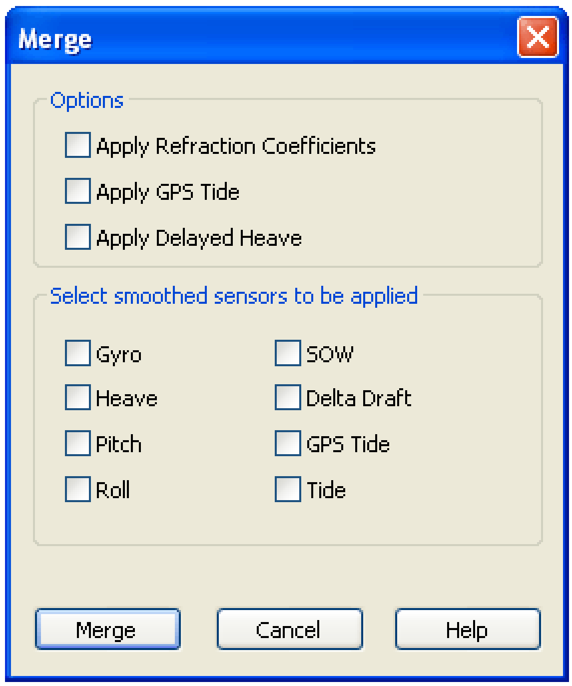
Lines must have tide loaded before they can be merged.

To apply Merge:

1. Select a line or group of lines.
2. Select the Merge command from the Process menu or toolbar.

Menu	Process > Merge
Tool	

The Merge dialog box is displayed.



3. Select the *Apply refraction coefficients* check box to apply any data created in Refraction Editor. (See "Correction for Refraction Artifacts" on page 334)
4. Select the *Apply GPS tide* check box to use GPS tide data instead of tidal observation data.
5. Select the Apply Delayed Heave check box to apply delayed heave values instead of regular heave.

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

If you have a SmoothedCoefficients file (see "Filter and Smooth" on page 16) for a sensor, you can apply this file during merge to smooth selected sensor data.

6. Click the check box beside a sensor to select the appropriate data that is to be smoothed during Merge.
7. Click **Merge**.

You can view the progress of the Merge process in the Output tab of the Worksheet window.

The Merge is now complete. Other functions requiring geo-referenced soundings can now be used such as the Subset Editor, Surface Cleaning, BASE Surface and others. The settings in the dialog box are remembered by the application.

> If a line is merged and then changes are made to the sensor data used by merge, the track line is marked in the Project tab with the Outdated icon. As well, the Outdated field for that line is set to “Yes” in the Selection tab. An Outdated track line must have Merge applied again.

Delta Draft

The Load Delta Draft function lets you import your model for dynamic changes in draft and have these changes applied to the soundings during the Merge process. Delta Draft represents a change in draft that is positive as draft increases and negative as draft decreases. The data, which is loaded from a ASCII text file, overrides the Dynamic Draft information in the vessel file. The Delta Draft value is interpolated between timestamps in Merge and added to the Observed Depth to obtain the Final Depth.

The input format for the Delta Draft text file is similar to the COWLIS tide format. Each record is in the following format:

```
YYYY/MM/DD HH:MM:SS.SSS 1.234
```

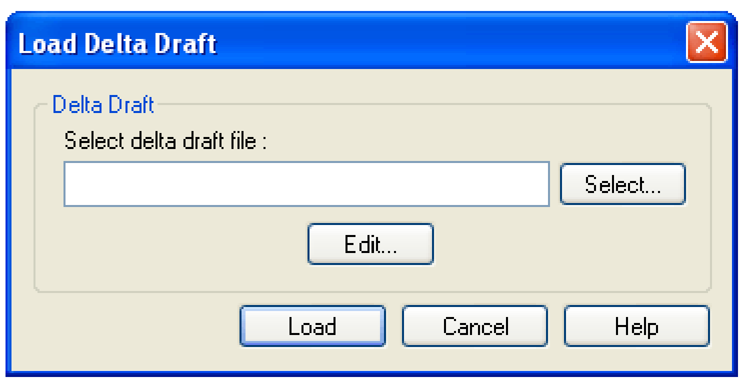
The Delta Draft function can also be used to load recorded depth versus time data for an underwater platform on which the multibeam sonar is mounted, such as a ROV (Remote Operated Vehicle) or AUV (Autonomous Underwater Vehicle), or a towed vehicle.

1. Select the track line(s).

Menu	Process > Load Delta Draft

2. Select the Load Delta Draft command.

The Load Delta Draft dialog box is displayed.



3. Type the path of the file or click **Select** to locate the file.
4. To make changes to the file, click **Edit**.

The file is opened in the default text editor (such as Notepad). Make any changes and save the file.

5. Click **Load**.

The delta draft is loaded into the line folder.

9

Create Field Sheets

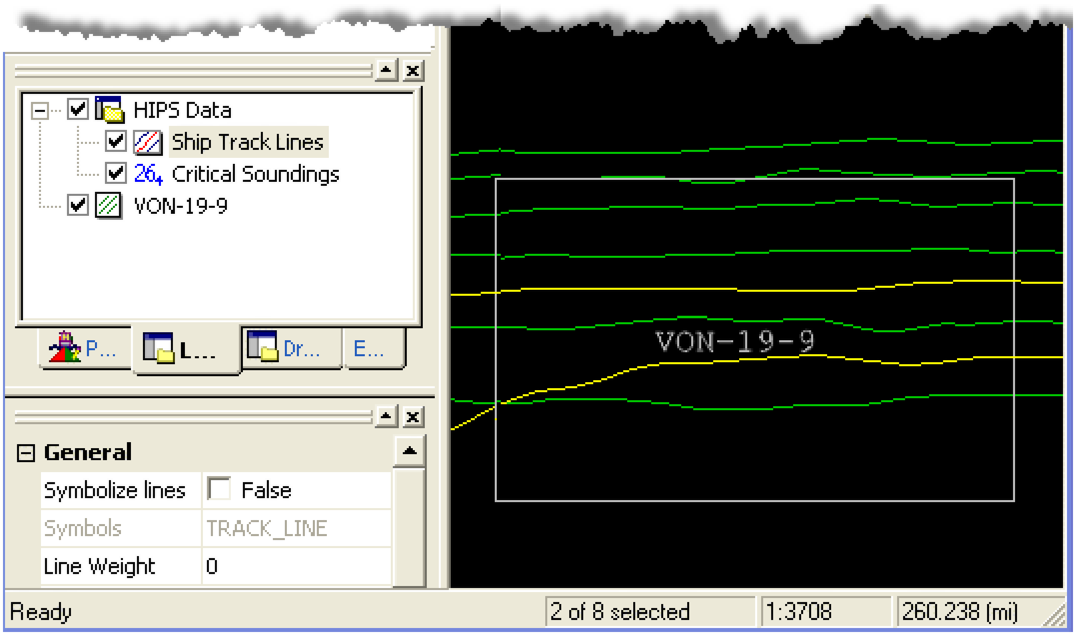
Field sheets in HIPS and SIPS are used to organize data products created from cleaned and processed bathymetry and side scan data.

In this chapter...

Create a Field Sheet 174
Open and Close Field Sheets 179
Set Field Sheet Properties 181
Field Sheet Projection Grid 182

Create a Field Sheet

Field sheets have a role in almost every process in HIPS and SIPS. They are essential for the creation of BASE surfaces. The image below illustrates a newly created field sheet over an area of track lines.



A field sheet consists of a data directory with a field sheet definition file and a CARIS file for storing vector products. The definition file contains information about the geographic boundary of the field sheet and the coordinate system to be used for the data products.

Field sheets are used to:

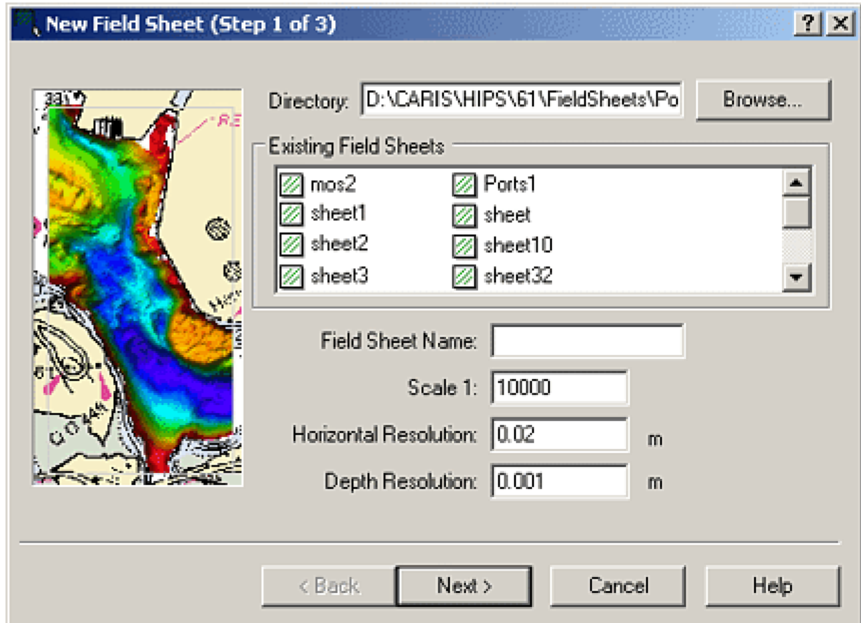
- create BASE surfaces. ("Create a BASE Surface" on page 193)
- create a layer of contours from a BASE surface or a layer of tiles. ("Contours" on page 361)
- create a layer of tiled soundings.
- generate a profile from a BASE surface.
- create a mosaic from side scan data.
- select soundings from a layer of tiles. ("Sounding Selection" on page 383)

To define an area for a new field sheet:

Menu	Process > New Field Sheet

6. Select the new New Field Sheet command.

The New Field Sheet Step 1 dialog box is displayed.



7. Type a name for the field sheet in the *Name* text box. Names should be alphanumeric characters only and cannot contain any spaces.
8. Define the scale of the field sheet.

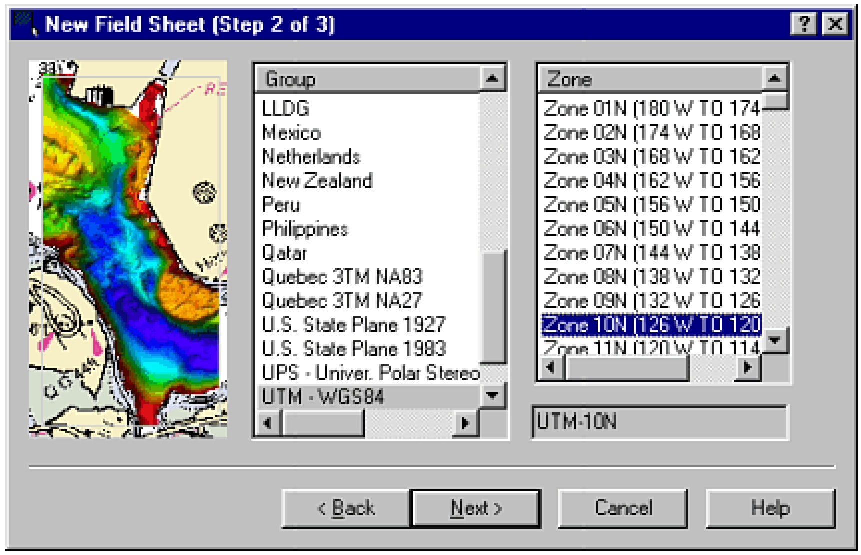
The *Horizontal Resolution* field sets the minimum distance between two adjacent coordinate values in the field sheet. The first time you create a field sheet this field will contain the default value of 0.020 (the minimum allowable resolution for S-57). You can reset this field to default to a different value.

9. Type a new value in the *Horizontal Resolution* field, if needed.

The *Depth Resolution* sets the minimum distance between two adjacent depth values. The *Depth Resolution* field contains a default value of 0.0010 (default resolution used in CARIS maps for heights/depths).Type a new value in the *Depth Resolution* field, if needed.

10. Click **Next**.

The New Field Sheet Step 2 dialog box is displayed.



This dialog box lists projections available to display your field sheet data. These are based on the coordinate systems and ellipsoid parameters as defined in the editable text file `..\Hips\System\mapdef.dat`.

11. Select a projection from the *Group* column.

The pre-defined projections specific to that projection system are listed in the *Zone* column.

12. Select a projection from the Zone column.

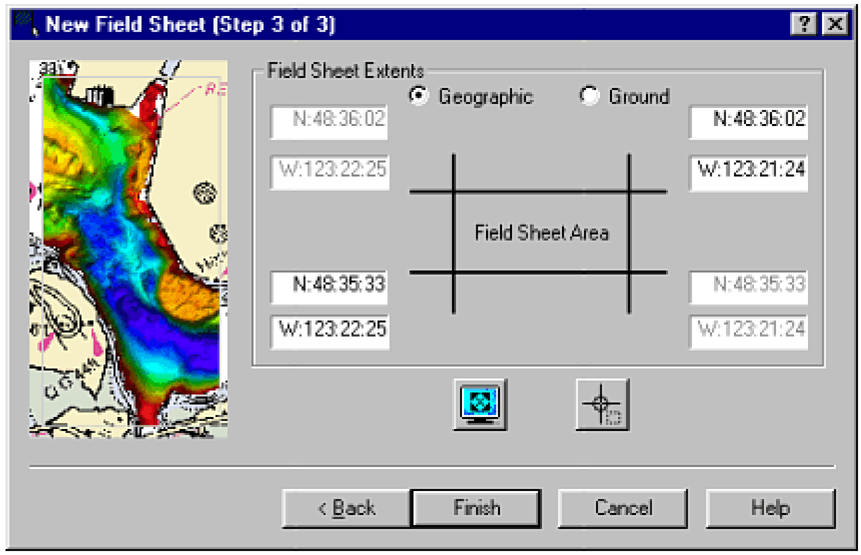
13. Click **Next**.

The New Field Sheet Step 3 dialog box is displayed.

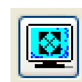
14. Select either the *Geographic* or *Ground* coordinates option.

15. Define the extent of the field sheet by one of three methods:

- use the current extent of the Display window, or
- use the mouse to create a box defining the extent, or
- enter the geographic or ground coordinates.

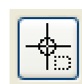


Use current display

1. Zoom in on the area you want to include field sheet so that it fills the Display window.
2. Click the Current Display Extent button. 

OR

Draw a bounding box

1. Click the Bounding Box button. 

The cursor in the Display window is shown as a cross-hair.

Press and hold the mouse button, and drag the cursor across the area where you want to create the field sheet.

A rectangular box is drawn across the area where the cursor was dragged.

- If you want to move the bounding box, position the cursor inside the box so it becomes a four-headed arrow. Press and hold the mouse button, and then drag the box to a new location.
- If you want to resize the bounding box, position the cursor on any of the box handles so the cursor becomes a two-headed arrow. Press and hold the mouse button and drag the handle to resize the box.

OR

Enter coordinates

1. Enter the northeast and southwest extents by clicking in one of the appropriate fields and typing the information, or by using the arrow keys to toggle the values into the entry fields.

After the area for the field sheet has been defined,

16. Click **Finish** to create the field sheet.

The new field sheet is outlined in the Display window.

The new field sheet data and files are located, by default, in ..\Hips\Fieldsheets\ProjectName\FieldSheetName.

Open and Close Field Sheets

Open a field sheet in a project or session.

NOTE: Opening a session file automatically opens all field sheets that were open when the session file was last saved.

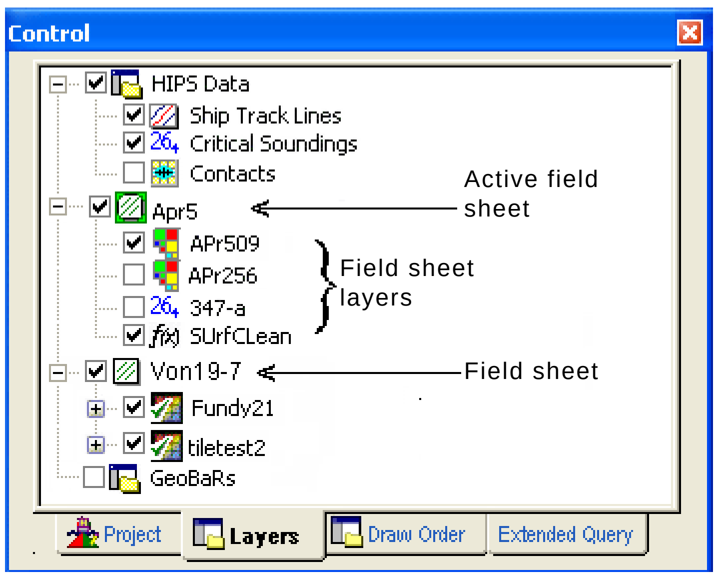
Menu	File > Open Field Sheets

1. Select the Open Field Sheets command.

The Open Field Sheets dialog box is displayed.

2. Select a field sheet from the list of available field sheets.
3. Click **Open**.

The field sheet outline is displayed in the Display window. Any layers that have been created within the field sheet are listed in the file tree in the Layers tab of the Control window, and are visible in the Display window.



You can toggle the display of the layers on and off using the check box next to a layer. (You may have to Refresh the display to view the data).

Set as active field sheet

You can have field sheets with different coordinate systems open in the same session. But only one field sheet can control the coordinate system used in the Display window.

The Set as Active Field Sheet command identifies the field sheet that controls the coordinate system in the Display window. By default, the first field sheet you open in a session is the active field sheet.

To change the active field sheet:

Menu	Process > Set as Active Field Sheet
Pop-up	Set Active

1. Select a field sheet in the Layers tab of the Control window.
2. Select the Set as Active Field Sheet command.

The icon for the active field sheet is highlighted green in the Layers tab. As well, the Active Field Sheet value in the Properties window will be set to “True” for the active field sheet.

Turn off field sheet

Individual layers in a field sheet can be activated or turned off by clicking the check box next to the layer. To make it easier to view data when there are many layers, or when more than one field sheet is open, you can turn off an entire field sheet with one command.

To remove a field sheet from the display without closing it:

1. Highlight the field sheet in the Layers tab.
2. Select Turn Off Field Sheet from the right-click menu.

All the previously selected layers of the field sheet are un-selected. To display the field sheet or any of its layers again, select the check box beside the layer.

Hide a surface

To hide a surface (e.g., a BASE surface) and its attribute layers, while continuing to display the field sheet:

Pop-up	Turn Off Layers

1. Highlight the surface in the Layers tab.
2. Select Turn Off Layers from the right-click menu.

The selected surface and all its attribute layers are no longer visible in the Display window.

Delete field sheet

Pop-up	Delete

1. Select the field sheet by name in the Layers tab of the Control window.
2. Select Delete from the right-click menu.

The field sheet is permanently removed.

Close all field sheets

Menu	File > Close Field Sheets

1. Select the field sheet by name in the Layers tab.
2. Select the Close Field Sheets command.

The field sheet outline is removed from the Display window and the file name is no longer displayed in the Layers tab.

Close single field sheet

Pop-up	Close

1. Select the field sheet by name in the Layers tab.
2. Right-click on the field sheet layer and select the Close command from the pop-up menu.

The field sheet is no longer open in the interface.

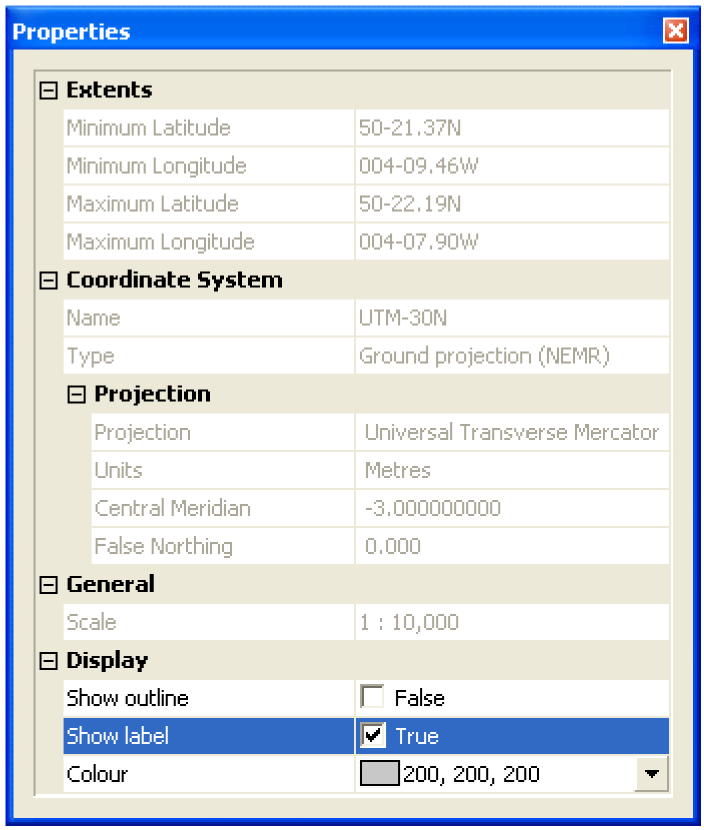
Set Field Sheet Properties

Menu	Window > Properties
Tools	
Pop-up	Properties

To set the display properties of a field sheet:

1. Select a field sheet layer in the Layers tab of the Control window.
2. If the Properties window is not open, select a Properties window command.

The Properties window for field sheets is displayed.



The *Extents, Coordinate System, Projection* and *Scale* fields are read-only and cannot be modified. You can modify the following properties:

3. Set the *Show outline* check box to “True” to display a border around the field sheet area.
4. Set the *Show label* check box to “True” to display the name of the field sheet in the Display window.
5. Set *Active Field Sheet* to “True” to make one field sheet the active sheet, when more than one field sheet is open.
6. Select a border *Colour* from the colour picker, or, create a custom colour from the standard Windows Colour Palette.

Field Sheet Projection Grid

The Projection Grid feature lets you display a grid of ground projection coordinates within a field sheet.

(The grid will cover the area of the field sheet only. To display a grid over the full extent of the display, use the Projected Grid option in the Tools > Options > Display dialog box. See "Projected Grid" on page 108 in the Reference Guide.)

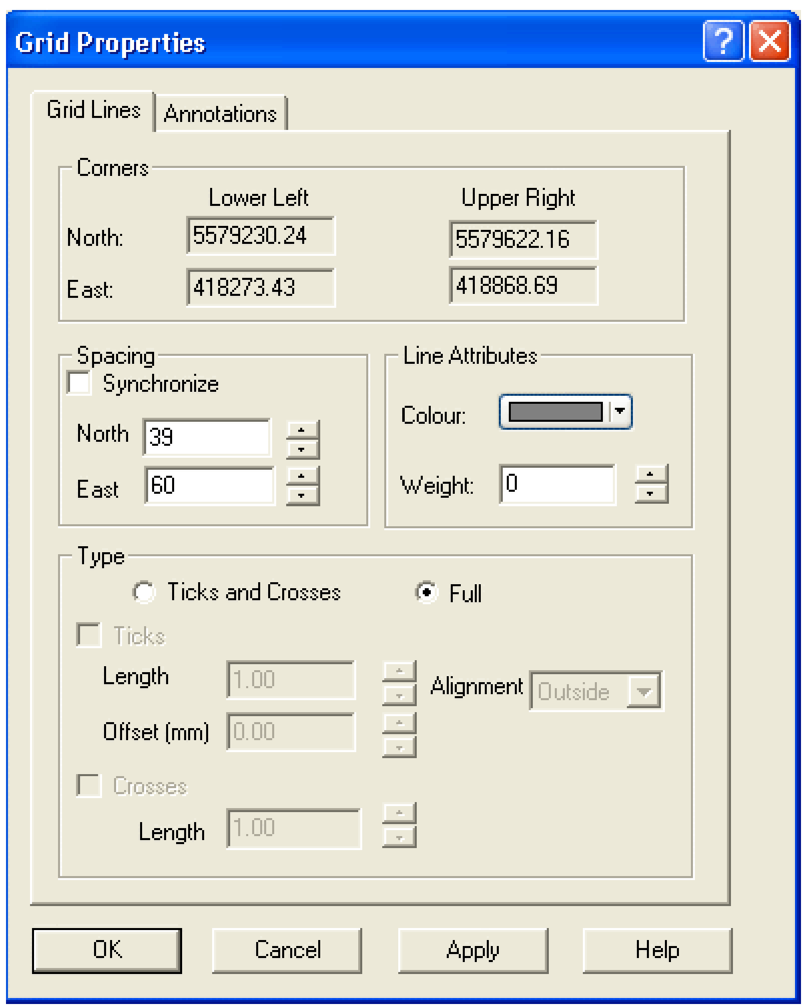
The projection grid has grid line properties such as thickness, shape and colour. You can also annotate the grid with the coordinates of your field sheet data, and set the look and positioning of these annotations. These settings for the grid are saved with the field sheet.

1. Select a field sheet in the Layers tab.

Menu	Process > Products > Projection Grid

2. Select the Projection Grid command.

The Grid Properties dialog box is displayed.



3. Set the options for positioning and style of grid lines as described in the table below.

Grid Lines Prop

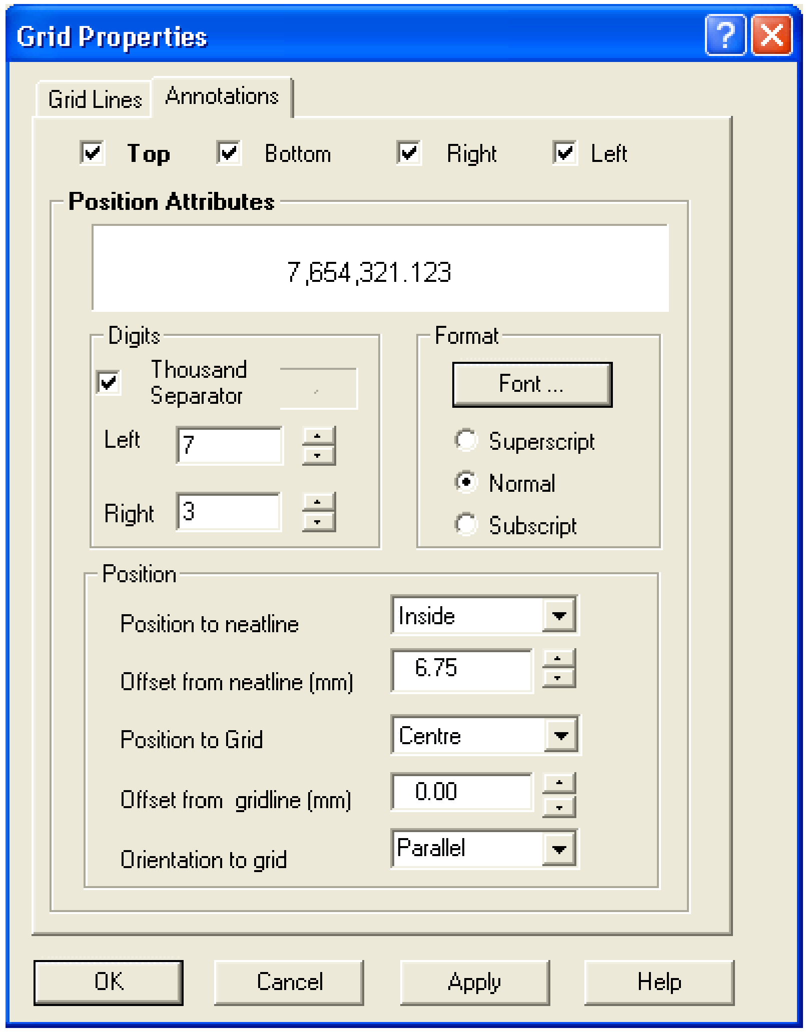
GRID LINES		
Property		**Description**
Corners		The lower left and upper right corners of the rectangular field sheet determine the corners of the grid.
	Lower Left	The latitude (North) and longitude (East) coordinates of the lower left corner of the field sheet.
	Upper Right	The latitude (North) and longitude (East) coordinates of the upper right corner of the field sheet.
Spacing		The spacing of the grid lines is calculated by adding the spacing values to the 0,0 position until the grid fills the area defined by the corners.
	Synchronize	To space the grid lines the same interval apart, enable Synchronize.
	North	Set the latitude interval between grid lines (North)
	East	Set a longitude interval (only available if Synchronize is not enabled.)
Line Attributes		Set a colour and line weight for the grid lines.
	Colour	Click on the **Colour** button and select a colour for the grid lines from the drop-down list.
	Weight	Set the thickness of the lines . Units are millimetres at display scale.
Type		Select the type of grid: solid lines or ticksand.or crosses.
	Full	The grid will be formed of solid lines.
	Ticks and Crosses	If you enable Ticks, you can set the following options: • Length (length of the markings in mm at display scale) • Offset • Alignment If you enable Crosses, you can set the following option: • Length (the dimensions of the crosses) You can use both ticks and crosses in the same grid.

4. Set Annotations options. See "Annotations" on page 184.

Annotations

This dialog box provides controls for annotating the projection grid with the coordinates of your field sheet data.

5. Click the Annotations tab to set position and format of annotations for the grid.

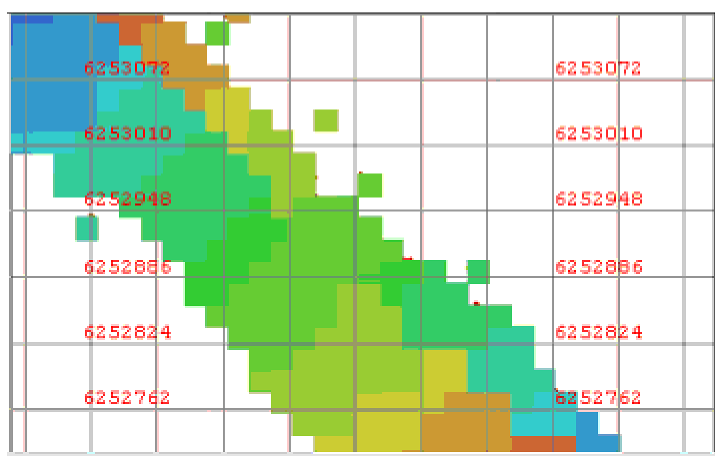


ANNOTATIONS	
Property	**Description**
Position Attributes	Select any combination of **Top, Bottom, Right, Left** positions for the annotations relative to the grid lines.

ANNOTATIONS	
Property	**Description**
Digits	Set the format of the digits used in the annotations. The field at the top of the dialog box shows how digits are displayed. By default, the annotation is displayed in full. • Thousand Separator: uses a comma to separate the digits to the right of the decimal point into groups of three, (e.g., 1,004,532). • Left: The number of digits to the left of the decimal point (e.g., 4 digits displays 4,532). • Right: The number of digits to the right of the decimal point (e.g., 3 digits displays 4532.123)
Format	Select a font for the digits, and choose to have it positioned above or below the normal line of digits.
Position	• Position to neatline: placement of annotation relative to neatline, either Inside or Outside the field sheet • Offset from neatline: distance of the annotation from the neatline in millimetres • Position to Grid: Choose Above /Left, Centre, or Below/ Right • Offset from grid line: the distance between the annotation and the grid line in millimetres • Orientation to grid: Parallel to the grid or at right angles to it.

6. Click **OK** to create the grid.

A grid of coordinates is generated and displayed in the Display window. A Projection Grid layer is added to the Layers tab.



To change the settings of the projection grid:

1. Select the Projection Grid layer in the Layers tab.
2. Open the Properties window and select the Settings field.

This will open the grid Properties dialog box where attributes other than the extents of the grid can be changed.

10

Create BASE Surfaces

The Bathymetry Associated with Statistical Error (BASE) Surface can be generated to view data for safety of navigation or for a detailed examination of the seafloor.

Three kinds of BASE surfaces can be created:

- Uncertainty
- Swath angle
- Combined Uncertainty and Bathymetry Estimator (CUBE)

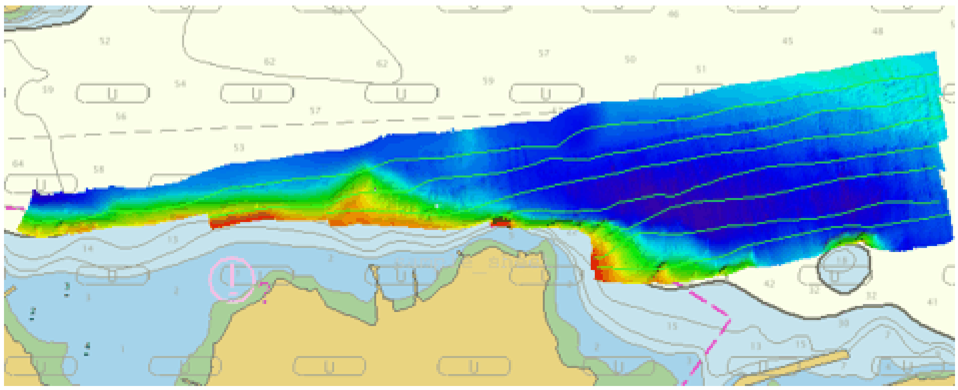
In this chapter...

Overview 188
Four types of BASE surfaces 189
Create a BASE Surface 193
Properties of BASE Surfaces 204
Vertical Reference System 215
BASE Surface Commands 218
BASE Surface QC Report 238
Surface Filtering 244

Overview

A Bathymetry Associated with Statistical Error (BASE) Surface is a georeferenced image of a multi-attributed, weighted-mean surface. A BASE Surface can also contain a visual representation of horizontal and vertical uncertainty.

An example of a BASE Surface image is shown below.



The BASE Surface is used for the following purposes:

- as a background visual aid when cleaning data interactively or automatically.
- to create a georeferenced TIFF that can be exported to other software such as real-time navigation systems.
- to export data as an XYZ text file so it can be imported as soundings in a CARIS map or used to build digital terrain models in other software.
- as a Product Surface that can be used to build contours, sounding selections, profiles, and other data layers. This data can later be used to create an ENC or raster chart.
- as a quality control feature for data. By viewing the uncertainty values for nodes, you can identify bad data and determine if an area needs to be re-surveyed.

A BASE Surface file is saved within the CARIS Spatial ARchive (CSAR) framework. This file contains the metadata for the surface.

Four types of BASE surfaces

Four different surface types can be created:

- Uncertainty uses a weighting scheme based on depth uncertainty as specified by a selected S-44 survey order.
- Swath angle uses a weighting scheme based on a beam's intersection angle with the seafloor.
- Shoalest Depth True Position stores the shoalest depth within a given node in the depth layer. The true position of this depth is also stored on the BASE Surface.
- Combined Uncertainty and Bathymetry Estimator (CUBE) surface uses multiple hypotheses to represent potential depth variances along the seafloor (see ““CUBE Processing” on page 247).

Uncertainty Weight

The weight that a sounding contributes to a node is inversely proportional to the predicted depth uncertainty of the sounding.

The sounding’s depth uncertainty is scaled as a function of the sounding’s distance from the node. The propagation of the depth uncertainty to the node takes into account the sounding’s distance from the node AND the sounding’s horizontal uncertainty. Thus, both depth uncertainty and horizontal uncertainty play a role in determining the weight a sounding contributes to a node.

The sounding’s area of influence is a radius that determines the number of nodes to which the sounding can be applied. This radius is determined by the depth uncertainty of the selected IHO S-44 survey order (see formula below).

$$\text{Depth Uncertainty} = \sqrt{a^2 + (b \times d)^2}$$

Once the depth uncertainty is propagated beyond the S-44 survey order requirement that has been selected for depth accuracy, then it has reached its area of influence limit.

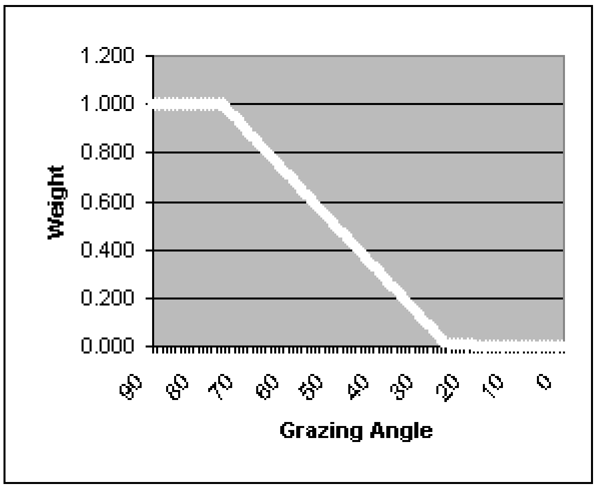
All distance computations are based on the ground coordinate system defined for the BASE Surface—they are not simplified to cell distances. All of the nodes within a sounding’s area of influence are located rigorously. No simplified weight matrix scheme is used.

Swath Angle Weight

The weight a sounding contributes to the Surface also varies with the sounding's grazing angle with the seabed.

This weighting value is important in areas with adjacent or overlapping track lines. The swath angle weight ensures that higher weight is given to beams from the inner part of a swath than to outer beams from adjacent track lines.

In the following graph, beams with a grazing angle between 90 and 75 degrees are given a weight of 1.0. The weight decreases linearly to 0.01 as the angle with the sea floor decreases to 15 degrees.



This is the default swath angle weight scheme as defined by the file...\Hips\System\GrazingAngleWeights.txt. The default file can be customized to another weighting scheme.

Range weighting

All BASE Surfaces use range weighting to determine how a sounding is applied to a node. Range weighting is based on distance—soundings close to a node are given greater weight than soundings further away. The calculated node positions are determined by the corner coordinates of the field sheet.

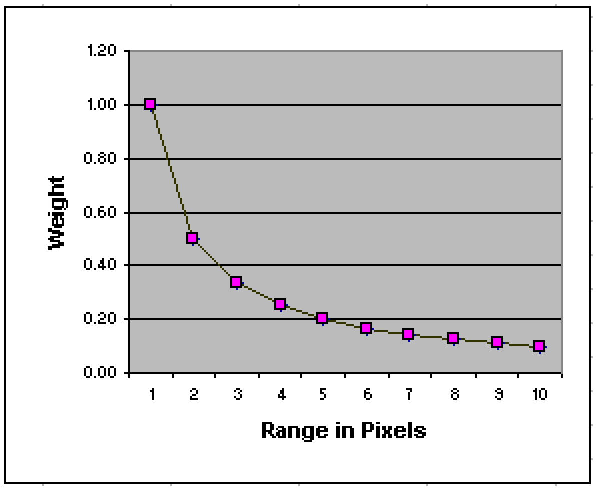
The range weight is inversely proportional to distance from the node: soundings closer to a node are given a greater weight than soundings further away.

The number of nodes each sounding is applied to is determined by the size of the beam footprint. The beam footprint is calculated using depth, sonar beam width, and the grazing angle.

The sonar that is used, with the appropriate beam width, is defined in the BASE Surface wizard.

A list of multibeam systems and corresponding beam widths is listed in ..\Hips\System\DeviceModels.xml.

The following graph demonstrates range weighting using distance from a node in units of pixels (multiples of the BASE Surface resolution).



Shoalest Depth True Position

This BASE surface stores the shoalest depth within a given node in the depth layer. It can also create a true position surface layer for which Horizontal and Vertical Uncertainty can be specified as optional output layers.

If true position data does not exist for the lines selected, the surface creation will continue without these attributes. If, at a later time, uncertainty data becomes available, a surface recomputation can be invoked and the surface will be updated accordingly.

CUBE

The Combined Uncertainty and Bathymetry Estimator (CUBE) generates a surface that contains multiple hypotheses representing potential depth variances on the seafloor. Examining these hypotheses in Subset Editor, you can determine if they are valid or not. The hypotheses can be replaced by alternative hypotheses (if necessary).

Using a CUBE filtering tool soundings that fall outside specified parameters can be rejected and another CUBE Surface generated. Thus, CUBE is used in an iterative process to filter data to produce the best-possible representation of the surveyed area.

For generation of CUBE surface and hypothesis editing see "Generating CUBE Surfaces" on page 251.

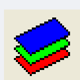
Create a BASE Surface

BASE Surfaces are created using a wizard. You can select specific track lines to include in the BASE Surface or let the wizard select them for you automatically. You can generate as many BASE Surfaces as necessary within a field sheet.

> All track lines must be merged before a BASE Surface can be created.

To create a BASE Surface:

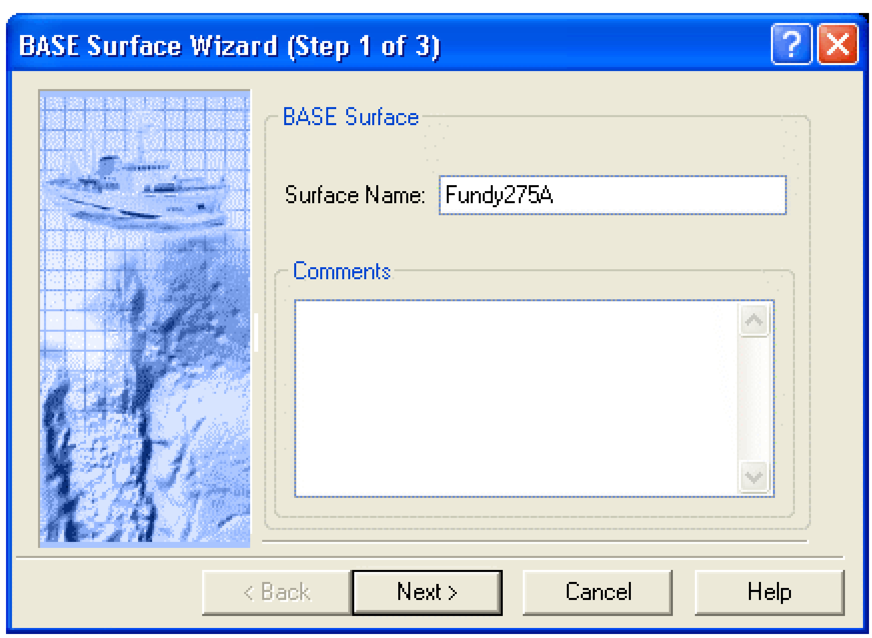
1. Open a field sheet of the area for the BASE Surface:
   - To create a BASE Surface for the entire field sheet area, select the field sheet in the Layers tab in the Control window.
   - To create a BASE Surface for a single track line, select a track line within the field sheet and then select the field sheet layer in the data tree.
2. Select the New Base Surface command.

Menu	Process > BASE Surface > New
Tool	

The number of steps in the wizard is determined by the type of surface you are creating.

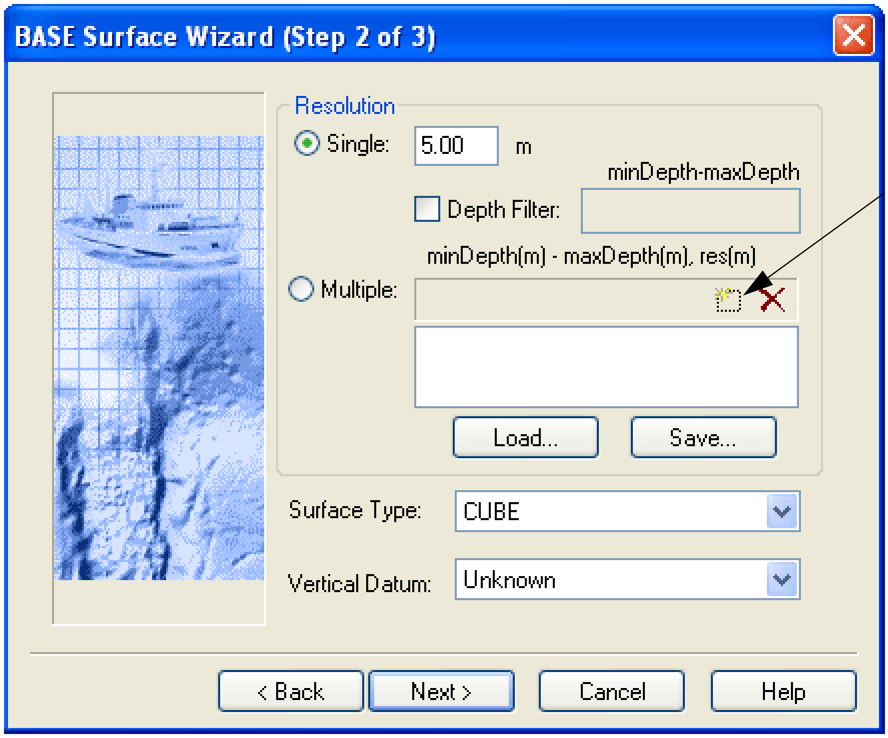
BASE Surface (Step 1)

The BASE Surface Wizard (Step 1) dialog box opens.



1. Type a *Name* for the BASE Surface.
2. Type any necessary *Comments* in the text box, and click **Next** to display the BASE Surface Wizard (Step 2) dialog box.

BASE Surface (Step 2)



Click to activate depth range and resolution fields.

The *Resolution* value(s) sets the distance(s) between BASE Surface nodes. You can use the same single resolution value for the entire surface, or use different resolutions for each range of depths.

1. Select *Single* and enter a resolution value, or select the *Multiple* option and enter a range of values.

If you select the *Single* option, you can determine the range of depths to include in the BASE surface by using the *Depth Filter* option.

2. Select *Depth Filter* and enter a range of depths.

If you select the *Multiple* option, the depth ranges and related resolution values can be saved as a template to apply when creating other new BASE surfaces.

3. Click Load to use a saved file.

   OR

4. Click  to activate the fields for depth ranges and the desired resolution in metres, and type the depth and resolution values.
   - Click  to delete contents of the field.
5. Click **Save** to save the settings as a template.

Values are entered in the following format:
`minimum depth - maximum depth, resolution` (all in

metres). For example, `0.0-10.0,5.0` which associates depths between zero and 10 metres with a resolution of 5 metres.

6. Select a Surface Type: *Swath angle, Shoalest Depth True Position, Uncertainty* or *CUBE* from the drop-down list.
7. Select the *Vertical Datum*, if known, from the list.

The Vertical Datum set for the BASE surface will be maintained in products created from it, such as finalized or interpolated surfaces. The Vertical Datum setting for a selected BASE surface can be seen and changed in the Properties window.

8. Click **Next**.

The content of the next dialog box is determined by the choice of surface type made in step 5:

"BASE Surface (Step 3) - CUBE" on page 253

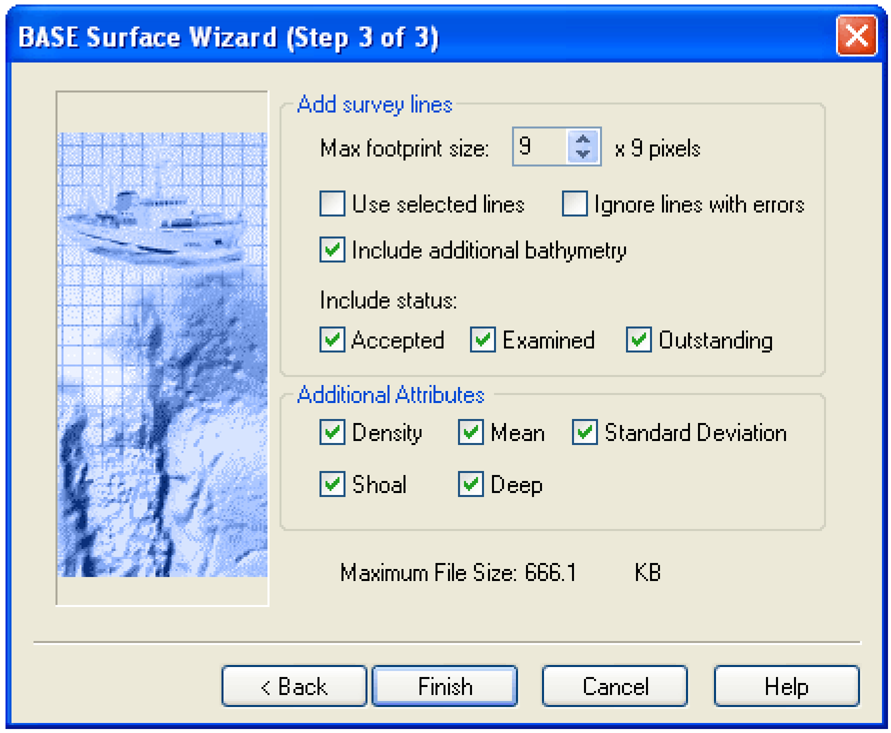
"BASE Surface (Step 3) - Swath Angle Weighting" on page 197

"BASE surface (Step 3) - Uncertainty Weighting" on page 199

"BASE surface (Step 3) - Shoalest Depth True Position" on page 201

BASE Surface (Step 3) - Swath Angle Weighting

If you selected the *Swath Angle Weighting* option at Step 2, the (Step 3) dialog box is displayed.



Add Survey Lines:

The *Max Footprint Size* defines the maximum area in the Surface to which a sounding is applied. Use this value to prevent over-expansion of the footprint due to large depth spikes or very shallow grazing angles.

1. Click the up and down arrow buttons in the *Max Footprint Size*.

If one or more track lines were selected before the BASE Surface process was started, the *Use Selected Lines* check box is enabled.

2. Check *Use Selected Lines* if you want to create the BASE Surface only on the selected line or lines.

Use the *Ignore Lines with errors* option so that the surface creation is not interrupted if it encounters bad data. The process will skip the part of the line containing bad data and continue creating the surface with the next line. Once the surface is complete you can decide whether to remove the partial line.

3. Check *Ignore Lines with errors* to have bad line data omitted from the surface.
4. Check *Include Additional Bathymetry* to include water column bathymetery added to project.
5. Select either *Accepted, Examined* or *Outstanding* check boxes to include soundings with these status flags in the BASE Surface process.

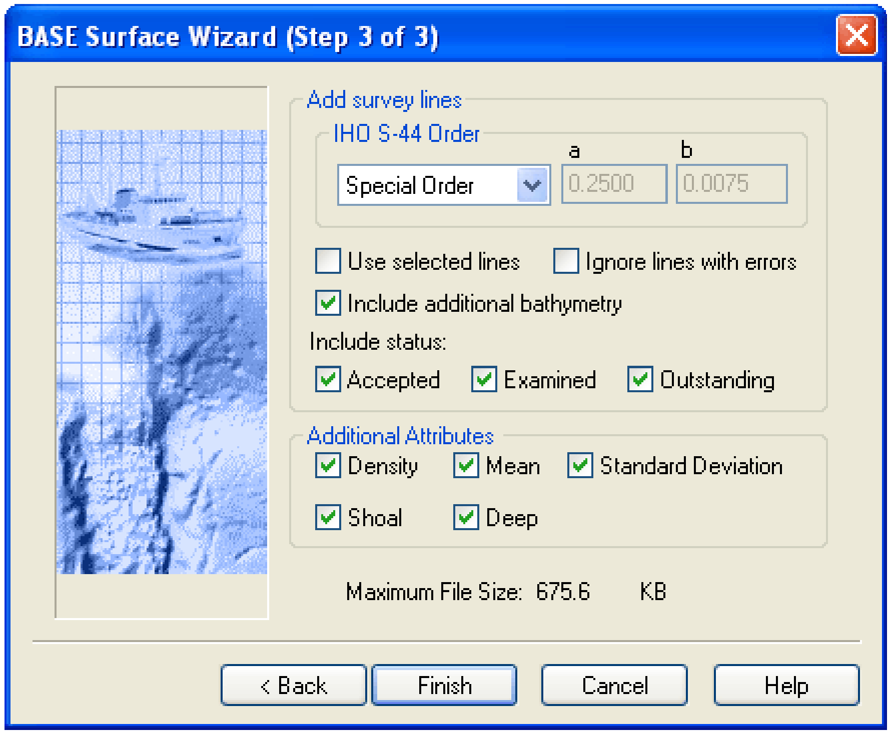
Additional Attributes

6. Select one or all *Additional Attributes* to create an attribute layer of the BASE Surface that displays that attribute.
   - *Density*: create an attribute layer that displays the density of soundings contributing to a node.
   - *Mean*: create an attribute layer that displays the mean of all soundings contributing to a node.
   - *Standard Deviation*: create an attribute layer that displays the standard deviation from the mean.
   - *Shoal*: create an attribute layer that displays the shoalest soundings contributing to a node.
   - *Deep*: create an attribute layer that displays the deepest soundings.
7. Click **Finish**.

The Swath Angle BASE Surface is created.

BASE surface (Step 3) - Uncertainty Weighting

If you selected the *Uncertainty* option, the (Step 3) dialog box is displayed.



Add Survey Lines

1. Select an *IHO S-44 Order* from the drop-down list.

The *a* (constant depth error) and *b* (factor of depth dependent errors) fields are automatically filled when a survey order is selected. The values are read from ..\HIPS\System\IHO_Standards.xml.

2. If one or more track lines were selected before the BASE Surface process was started, the *Use Selected Lines* check box is enabled. Select this check box if you want to create the BASE Surface for the selected line or lines.

Use the *Ignore Lines with errors* option so that the surface creation is not interrupted if it encounters bad data. The process will skip the part of the line containing bad data and continue creating the surface with the next line. Once the surface is complete you can decide whether to remove the partial line.

3. Check *Ignore Lines with errors* to have bad line data omitted from the surface.
4. Check *Include Additional Bathymetry* to include water column bathymetery added to project.

Additional Attributes

5. Select either the *Examined* or *Outstanding* check boxes to include data with these status flags in the BASE Surface process. The *Accepted* check box is selected by default.

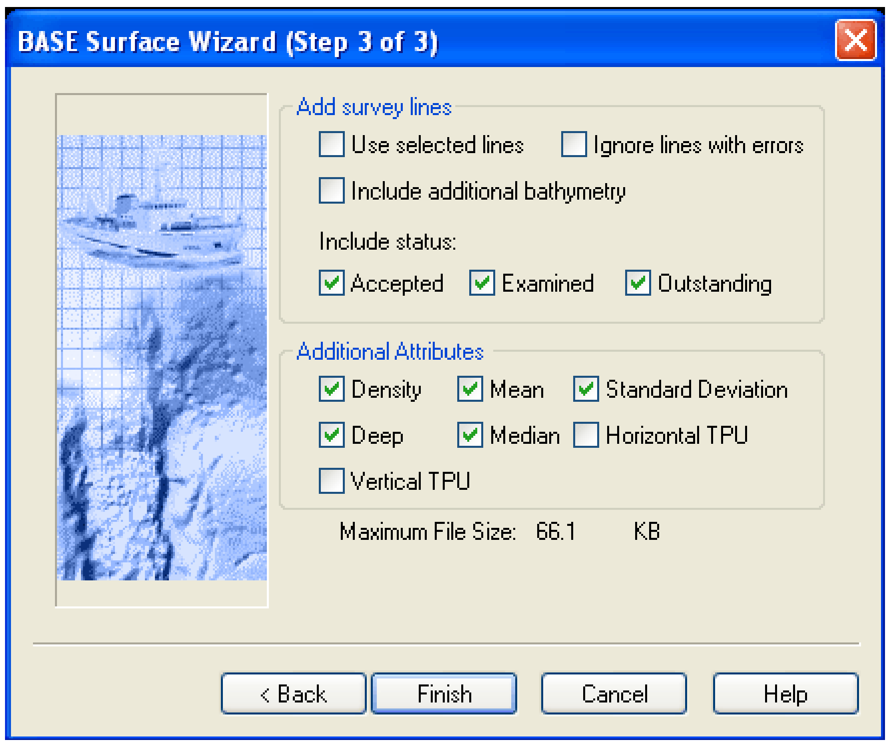
6. Select one or all *Additional Attributes* to create an attribute layer of the BASE Surface that displays that attribute.
   - *Density*: create an attribute layer that displays the density of soundings contributing to a node.
   - *Mean*: create an attribute layer that displays the mean of all soundings contributing to a node.
   - *Standard Deviation*: create an attribute layer that displays the standard deviation from the mean.
   - *Shoal*: create an attribute layer that displays the shoalest soundings contributing to a node.
   - *Deep*: create an attribute layer that displays the deepest soundings.
7. Click **Finish**.

The Uncertainty BASE Surface is created.

BASE surface (Step 3) - Shoalest Depth True Position

If you selected Shoalest Depth True Position in Step 2, the Step 3 dialog box displays the following options. The options specific to this surface type are Median, Horizontal Uncertainty and Vertical Uncertainty, which enable you to create surface attribute layers from those kinds of data .

If such data does not exist for the lines selected, the surface creation will continue without these attributes. If, at a later time, uncertainty data becomes available, a surface recomputation can be invoked and the surface will be updated accordingly.



Add Survey Lines

1. Check *Use Selected Lines* if you want to create the BASE Surface only on the selected line or lines.

Use the *Ignore Lines with errors* option so that the surface creation is not interrupted if it encounters bad data. The process will skip the part of the line containing bad data and continue creating the surface with the next line. Once the surface is complete you can decide whether to remove the partial line.

2. Check *Ignore Lines with errors* to have bad line data omitted from the surface.
3. Check *Include Additional Bathymetry* to include water column bathymetery added to project.
4. Select either *Accepted, Examined* or *Outstanding* check boxes to include soundings with these status flags in the BASE Surface process.

Additional Attributes

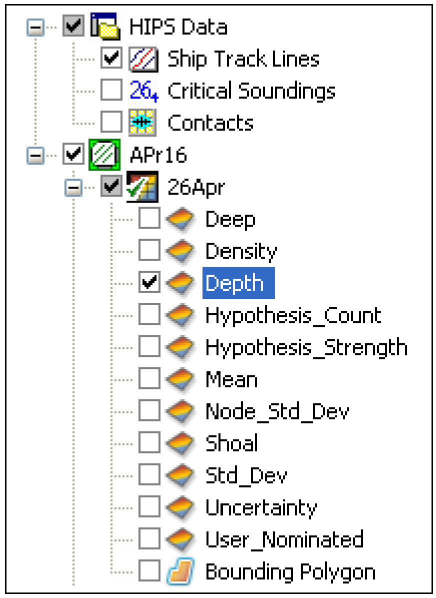
5. Select one or many of the *Additional Attributes* to create a layer of the BASE Surface that displays that attribute.

Density	creates an attribute layer that displays the density of soundings contributing to a node.
Mean	creates an attribute layer that displays the mean of all soundings contributing to a node.
Standard Deviation:	creates an attribute layer that displays the standard deviation from the mean.
Deep	creates an attribute layer that displays the deepest soundings.
Median	creates an attribute layer that displays the median values of soundings contributing to the node
Horizontal TPU	creates an attribute layer that displays the Horizontal TPU value for the shoalest sounding used for the Depth Attribute layer.
Vertical TPU	creates an attribute layer that displays Depth TPU value for the Shoalest sounding used for the Depth attribute layer

6. Click **Finish**.

View BASE Surfaces

In the HIPS interface, when a BASE Surface layer is created it is displayed in the Layers tab of the Control window. Attributes such as Depth are displayed as child layers of the BASE Surface.



1. Click Expand (+) icon beside the Field Sheet file name to expand the directory tree in the Layers tab.
2. Display the BASE Surface by selecting the check box beside the layer.

The following attributes can be displayed in the BASE Surface (depending on the weighting method you selected in Step 6 and the selected attributes in Step 7):

- **Depth**: Depth at the node.
- **Density**: The number of soundings contributing to a node.
- **Standard Deviation**: The number of standard deviations that the node is from the mean.
- **Uncertainty**: The depth uncertainty assigned to each node.
- **Mean**: The mean depth calculated from all soundings contributing to a node.
- **Shoal**: The shoalest sounding contributing to a node.
- **Deep**: The deepest sounding contributing to a node.

3. To view or hide layers, select or clear the check boxes beside the layer.

Properties of BASE Surfaces

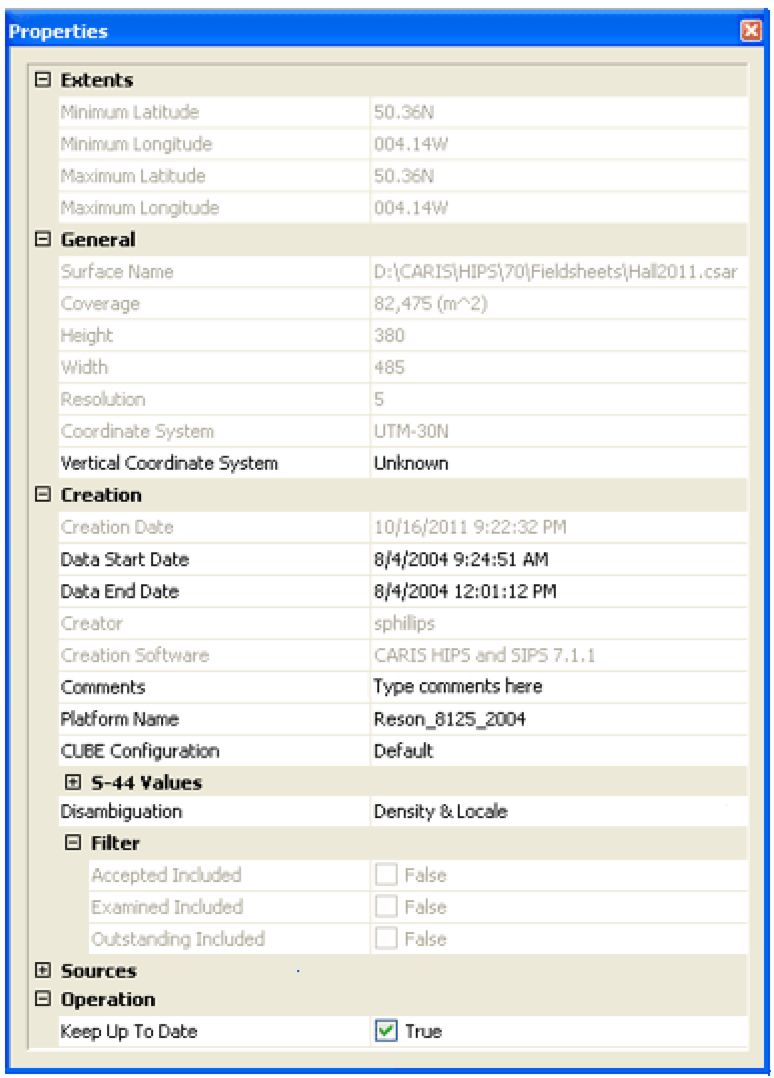
Use the Properties window to view information about a BASE Surface or to set the display options for its attribute layers.

View Surface Properties

1. Select a BASE Surface layer in the Layers tab of the Control window.
2. Select the Properties window command, or right-click the layer and select Properties from the pop-up window.

Menu	Window > Properties
Tools	
Pop-up	Properties

The Properties window displays the BASE Surface information. The image below shows the properties of a CUBE surface parent layer.



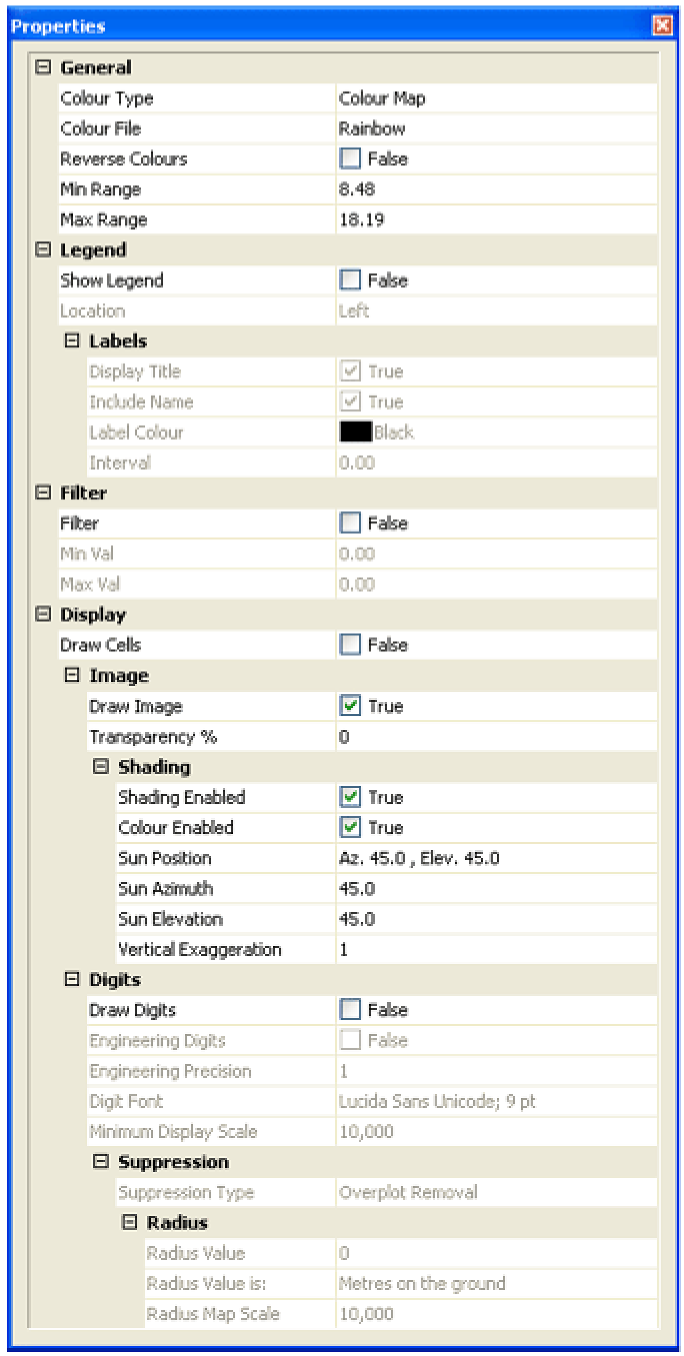
Most of this information is read-only, however, the following fields can be edited:

Editable fields:

Properties	Function
General	
Vertical Coordinate System	Select the appropriate vertical datum for the surface data from a drop-down list.
(Coverage)	(This field cannot be edited, but the coverage value can be updated using the Update Coverage command on the pop-up menu.)
Creation:	
Data Start Date	Date can be modified using the drop-down calendar
Data End Date	Date can be modified using the drop-down calendar
Comments	This field is populated with the contents of the Comments field from Step 1 of the BASE Surface creation. This may be the mandatory <abstract> element for 19115 metadata, or other information about the surface entered during creation. This content can be edited.
Platform Name	Displays the vessel name.
CUBE Configuration	Displays the configuration selected in the last step of creating a CUBE surface. If a configuration other than Default was selected, or created in the Advanced Options dialog box, click on the Browse button [...] to see the values applied in that configuration. (See also “Advanced Options for CUBE surface” on page 255.)
S-44 Values:	(Present only if selected surface is a CUBE surface.)
Disambiguation	Select the process used the create the CUBE surface, form a drop-down list.
Operation:	
Keep Up To Date	Set to True to automatically update the currently selected surface. (*Automatic BASE Surface update* must also be set in Tools > Options > General.)

Attribute Layer Properties

All of the properties for an attribute layer can be edited, however some fields are only activated when others are set. For example, *Sun Position* and *Vertical Exaggeration* fields are greyed out unless *Shading Enabled* is set to "True".



> The properties of the Bounding Polygon layer are different from those of the other attribute layers. See "Bounding Polygon Properties" on page 213.

BASE Surface Properties are organized into four groups:

"General options" on page 207

"Legend options" on page 208

"Filter options" on page 209

"Display options" on page 210

General options

General properties control the colour display for the attribute layer. Use either a colour map file or a colour range file to display data values as colours in the display. Colours can be reversed when using a colour map file.

See "Colour Map Editor" on page 39 for more information.

:

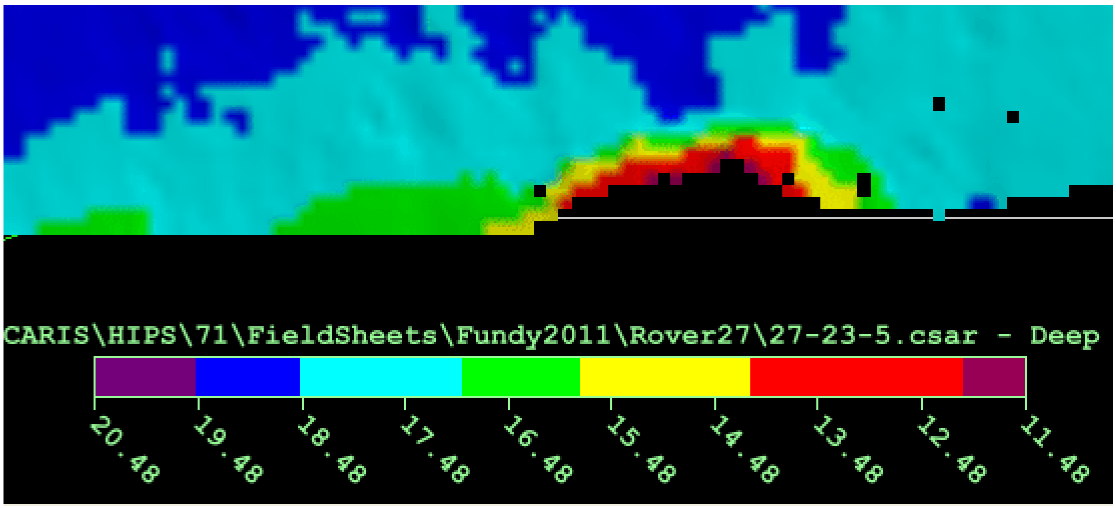
Property	Function
General	
Colour Type	Select Colour Map or Colour Range (Colour Map activates other fields).
Colour File	Displays the active colour file based on the *Colour Type* selected above. The default colour map is Rainbow. If you chose *Colour Range*, but no colour range files exist in the directory, the surface will automatically be coloured grey. By default, the first colour range file in the drop-down list will be selected.
Reverse colours	Select to apply colour mapping from maximum to minimum values instead of from minimum to maximum.
Min range	The minimum value for the attribute layer to which the colour file is applied, e.g., the shoalest data on the Depth layer or the minimum number on Density layer.
Max range	The maximum value for the attribute layer to which a colour is applied e.g., the deepest sounding on the Depth layer or highest number on the Density layer.

The *Min Range* and *Max range* values determine the upper and lower extents to which the colour file is applied. The Minimum or the range is mapped to the first value in the colour file, and the maximum to the last value.

The Min and Max range is taken from the data for the selected layer, and can be adjusted. For example, if the minimum of the range is increased, all data falling below that value will be mapped to the first colour defined in the colour file. Similarly, if the maximum in lowered, all data falling above that maximum will be coloured with the last colour in the colour file.

Legend options

You can add a raster legend to the display, to identify which attribute values (e.g., depths) are represented by which colours.

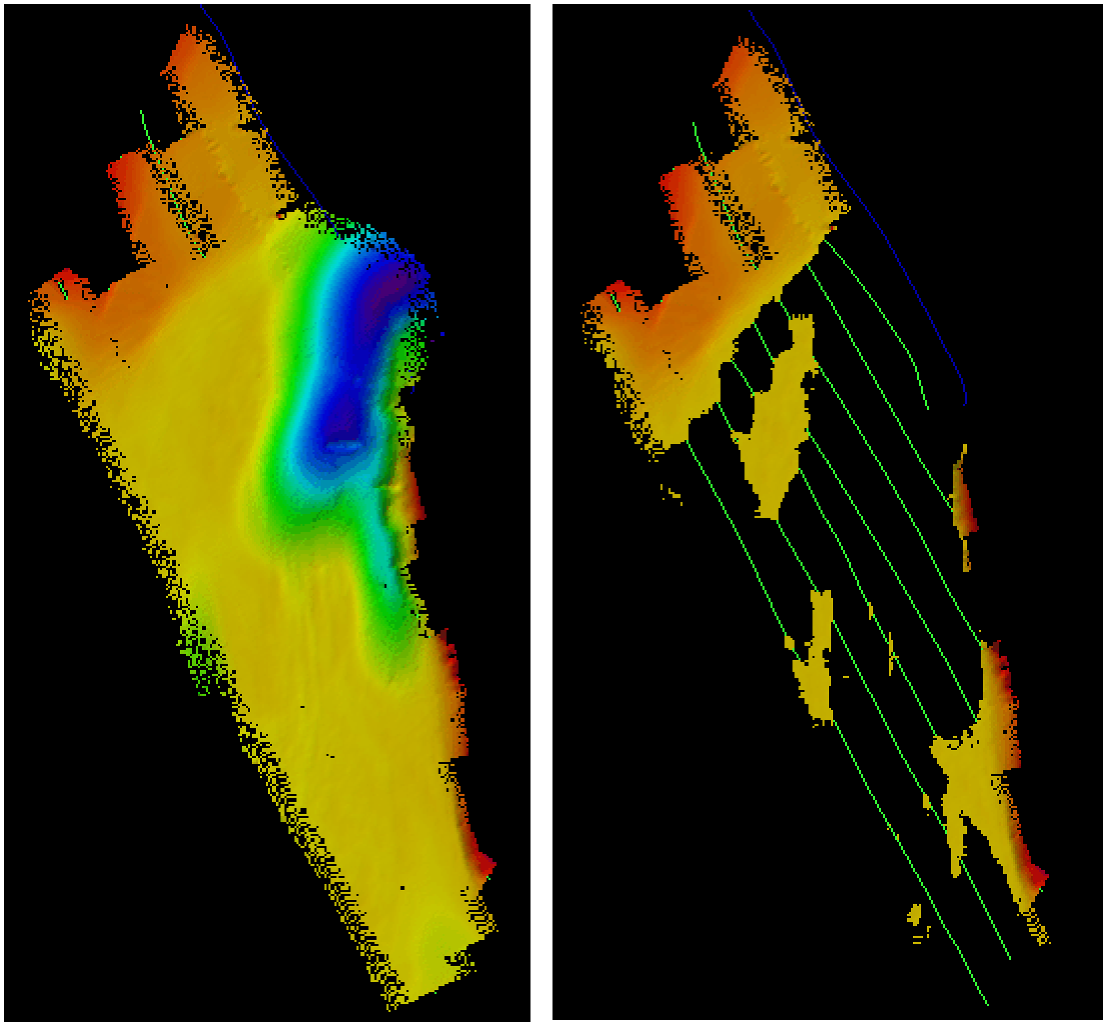


See also "Raster Legend" on page 38.

Property	Function
Legend:	
Show Legend	Set to True to display a legend to identify the range of values to which each colour is mapped.
Location	Select the position of the legend, e.g., Left of the display, bottom of the display etc.
Labels:	These are options for descriptive text on the legend.
Display Title	Set to True to display the name of the attribute layer and units, e.g., Depth (m).
Include Name	Set to True to display the full path and name of the attribute layer e.g., C:\CARIS\HIPS\71\Fieldsheets\Project\Fieldsheet name\BASEsurfacename.csar. - Depth (m).
Label Colour	Set the colour of the labels on the legend. The default colour is black.
Interval	Set an interval between the values displayed on the legend scale

Filter options

Use filter controls to limit the display of data to that which falls between the minimum and maximum values. The image below illustrates filtering to display a range of shoalest depths.



Property	Function
Filter:	
Filter	Set to True to display only the surface data with values between the Filter minimum and maximum values (as set in the fields below).
Min val	Set a minimum value below which data will not be displayed.
Max Val	Set a maximum value above which data will not be displayed.

Display options

Display properties include controls for display of images and digits. The Draw Cells property will display a grid representing the nodes of the surface layer. With large data sets, you may need to zoom in to see this grid, as in the illustration below.

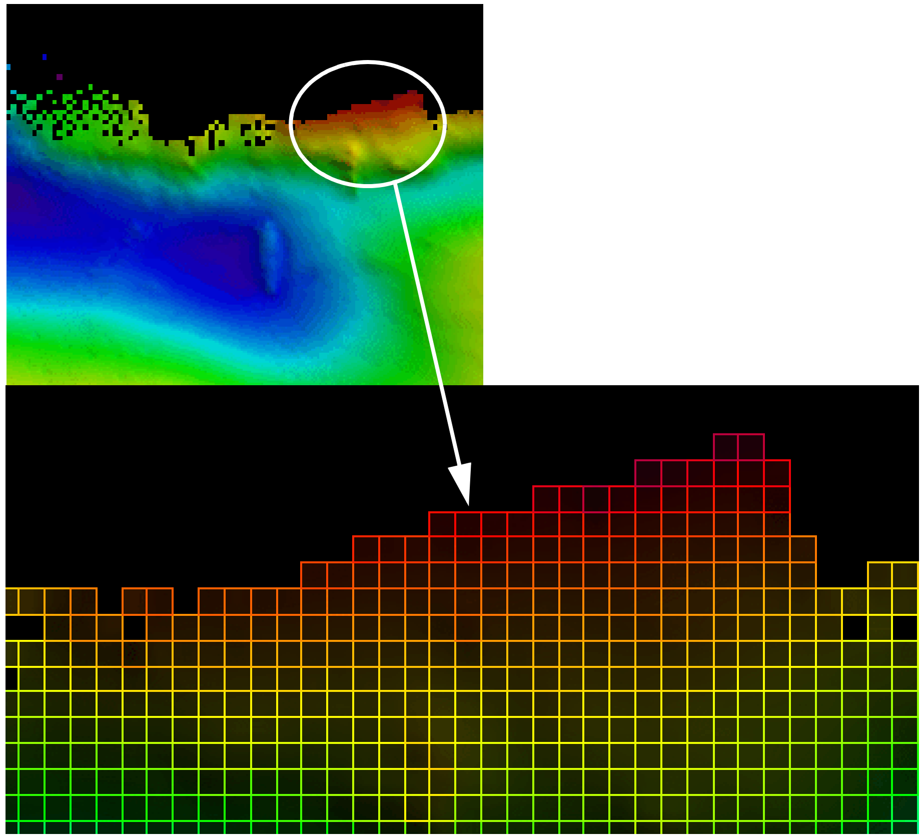


Image properties control whether or not the selected attribute layer is displayed, and options such as transparency and shading. Shading effects reflect the sun elevation and azimuth values.

Properties for display of digits include the option to display data in engineering format instead of hydrographic format. For example, hydrographic format would show a depth as *12*^{7}. Engineering format would display the same value as 12.7 (to as many decimal places as set).

When using engineering digits for depth values, sounding rounding rules are applied. However, only the last digit in the sounding will be rounded to ensure that the desired precision is not lost.

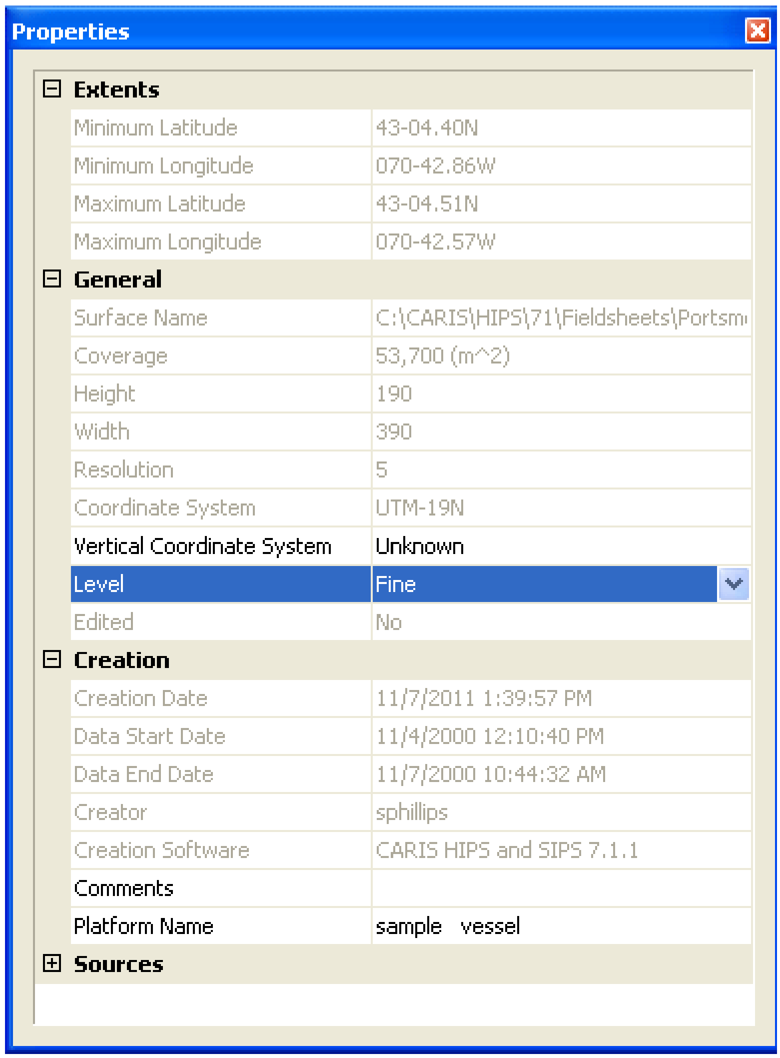
Property	**Function**
Display:	
Draw Cells	Set to True to display a grid representing the nodes of the surface.
Image:	
Draw Image	Set to True to display the selected layer or grid. Default setting is True.
Transparency	Type a *Transparency* percentage for the selected layer: the higher the percentage value, the more transparent the layer will appear.
Shading:	
Shading Enabled	Set to True to turn on shading to heighten the ridge detail for fine-scaled features.
Colour Enabled	Set to display the attribute layer using the full colour map or file. If set to False, the layer will be displayed as grey only.
Sun Position	5. Click in the field to display the sun compass. 6. Move the symbol to shift the light direction. 7. Click in the display to see the change in shad-ing. When the sun symbol is in the centre of the compass, the light is represented as directly above the image. The height and direction from which the light is displayed is shown by the *Sun Elevation* and *Sun Azimuth* values.
Sun Azimuth	An adjustment here is reflected in the *Sun Position* field.
Sun Elevation	An adjustment here is reflected in the *Sun Position* field.
Vertical Exaggeration	Increase the exaggeration value to heighten the vertical detail.
Digits:	
Draw Digits	Set to True to display data values.
Engineering Digits	Set to True to display the digits of the data nodes with decimal places instead of superscripts, and in regular (un-slanted) text.
Engineering Precision	Set the number of decimal places to include when values are displayed in engineering digits. Default is 1 decimal place.
Digit Font	Click the Browse button to select the font from the Fonts dialog box.
Minimum Display Scale	Set the value of the smallest scale at which digits will be displayed.

Property	Function
Suppression:	Filter out specific data based on the number of soundings in a selected area. Fields are activated when Draw Digits is set to “True”.
Suppression Type	Apply Overplot Removal (to remove overlapping soundings), Radius (filter out soundings within a set radius), or None.
Radius:	Size of the area to which the suppression is applied.
Radius Value	The radius defined as metres on the ground, or as the number of millimetres between soundings at map scale.
Radius Map Scale	Value to apply to scale.

Bounding Polygon Properties

A bounding polygon displays properties similar to those of a parent BASE surface (Extents, General, Creation, and Sources). (See "Properties of BASE Surfaces" on page 204.)

One property specific to bounding polygons is the ability to adjust the boundary so that it more closely delineates the extent of the surface.



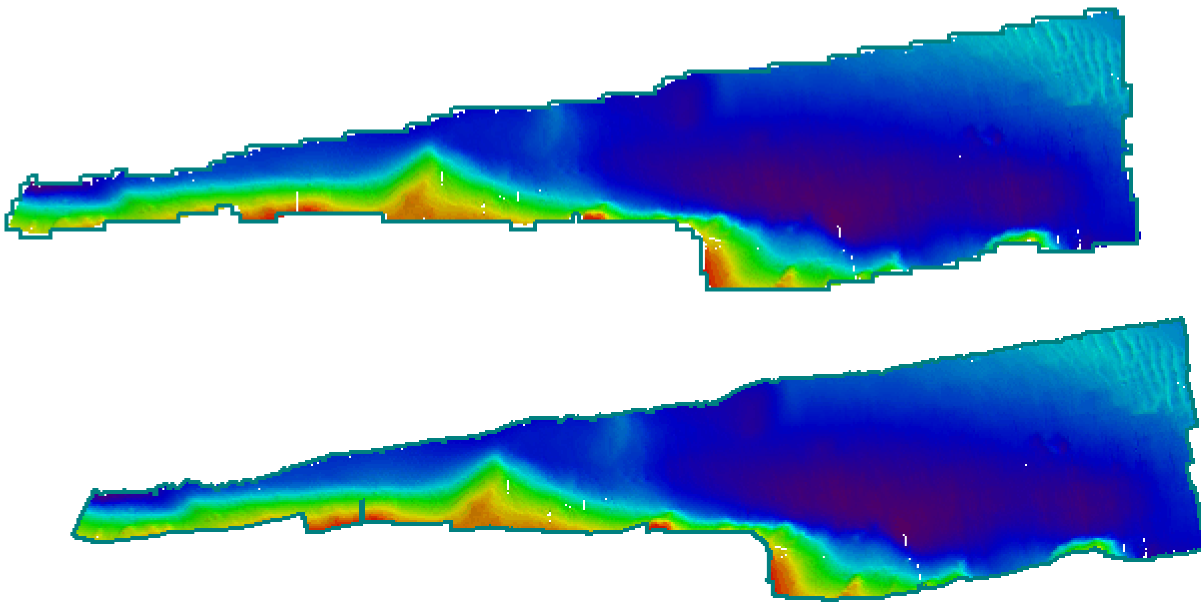
How closely the bounding polygon fits the extents is determined by the level of detail used to create the polygon. Settings in the *Level* field in the Properties window can be used to rebuild the polygon with a different resolution. (By default, polygons are created at a Medium level.) The polygon can be recreated using a Coarse level (less detail), or a Fine level (more detail).

To change the Level property:

1. Select the Bounding Property layer in the Layers tab.

2. Select the *Level* field in the Properties window.
3. Choose a different setting from the drop-down list.
4. Right-click on the Bounding Polygon layer, and select Rebuild from the pop-up menu.

The bounding polygon will be rebuilt. The image below shows the difference between the polygon drawn at the default level and rebuilt at a Fine level.



Vertical Reference System

A vertical reference system is a 3D coordinate reference system in which position is defined by latitude, longitude, height or depth and a linear unit of measure, for example, feet or metres.

When the Z values in a vertical reference system represent depths, the Z axis direction is “positive-down”. (This is consistent with the HIPS and SIPS depth convention.)

In HIPS and SIPS the identification of the vertical reference system information used for a project can be added to a BASE surface during its creation. Alternatively, it can be added to the Vertical Coordinate System field in the Properties of an existing BASE surface.

A database of defined vertical reference systems in available in HIPS and SIPS through the Vertical System Editor. This information is drawn from the European Petroleum Survey Group (EPSG) database of reference systems, which can be found at: http://www.epsg-registry.org.[1]

Vertical Reference System Editor

The existing datum, ellipsoid and reference system entries in the database are read-only. Each entry has a unique name and numerical identifier.

Used-defined entries can also be added to the database using the Editor.

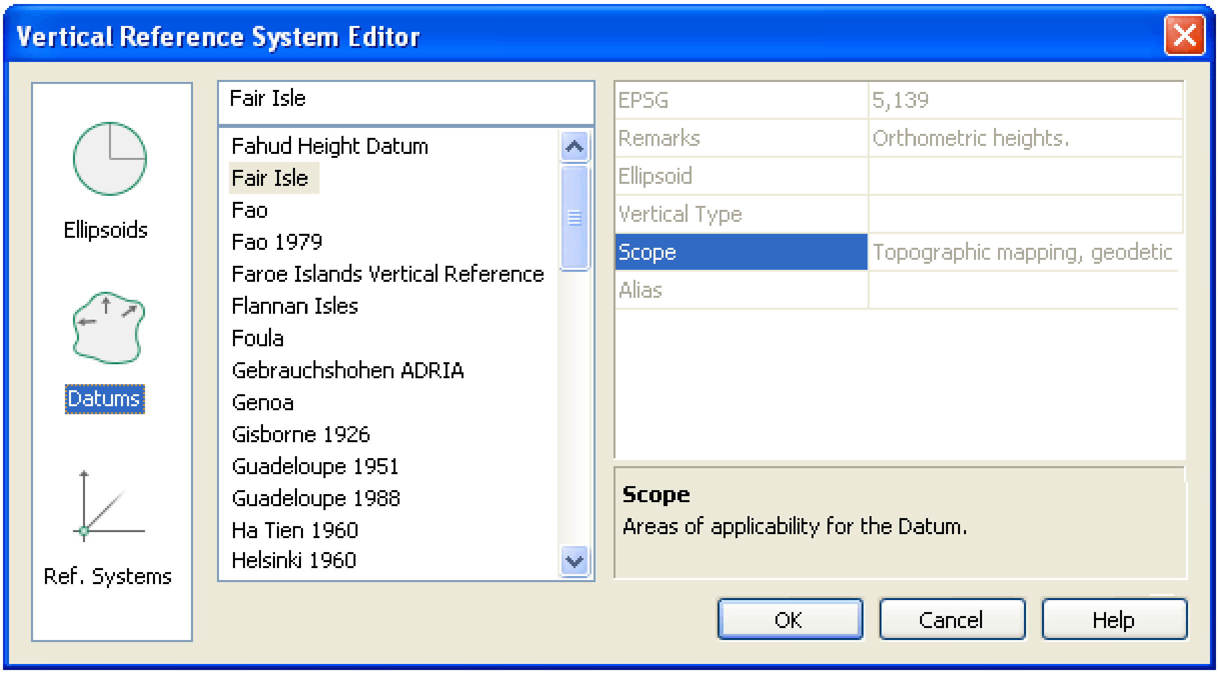
To open the Vertical Reference System Editor:

Menu	File > Vertical Reference System Editor

1. Select the **Reference System Editor** command.

The Vertical Reference System Editor is displayed.

1. Use of material from OGP Surveying and Positioning Committee’s EPSG Geodetic Parameter Registry does not imply endorsement by OGP Surveying and Positioning Committee of CARIS products or services.



The leftmost column lists the types of coordinate system elements for which you can view properties.

2. Select the category of data you want to view, for example, Datums.

This will activate the listing of all the entries for that category. to view the properties of a specific entry,

3. Select an item from the list. (The example above shows the Fair Isle datum selected.)

The properties for the selected datum (or ellipsoid or vertical reference system) are displayed in the pane on the right. As you select each property field, a brief description of the field is displayed below the properties. In the example illustrated above, the content of the *Scope* field show “Topographic mapping and geodetic surveying” as the areas of applicability for the selected datum.

> The first property field contains the EPSG code number that identifies the selected datum, ellipsoid or reference system.

Editing the Vertical Reference database

You can add your own entries to the database, provided the names of the entries do not duplicate existing names. User-defined entries can be deleted, but original entries cannot.

To add an item:

1. Select the icon for the category of data that you want to add. For example, to add an ellipsoid, click the Ellipsoids icon.
2. Right-click anywhere in the ellipsoids listing, and select New from the pop-up menu.

A New entry is added to the list.

3. Click on the new entry and type the name for your entry, over-writing “New”.

In order for your new entry to be saved to the database, it must have values defined for the properties. If you do not define the properties, the new item will be removed from the list when the editor is closed.

4. Type appropriate information in the properties fields in the list on the right.
5. Click **OK** to save your changes to the database and close the dialog box.

Deleting entries

While the original items in the Vertical Reference System database are read-only, any user-created entries can be deleted.

To delete a user-defined entry:

1. Select the entry.
2. Right-click in the pane and select “Delete” from the pop-up menu.
3. Click OK in the dialog box to confirm the deletion.

BASE Surface Commands

Open

To open a BASE surface you first need to open the field sheet in which it was created.

Menu	File > Open Field Sheets

1. Select the Open Field Sheets command.

The Open Field Sheets dialog box is displayed.

2. Select one (or more) field sheet files from the list.
3. Click **Open**.

The field sheet filename is displayed in the Layers tab of the Control Window and the field sheet outline(s) is shown in the Display window.

4. Click the (+) icon to expand the field sheet file tree.
5. Select the BASE Surface check box.

The BASE Surface is now visible in the Display window.

Delete

Permanently delete a BASE surface from a field sheet.

Pop-up	Delete

1. Right-click on a BASE surface layer in the Control Window, and select the Delete command from the pop-up menu.

The BASE Surface layer is permanently deleted from the field sheet.

Add to BASE Surface

The Add to BASE Surface command applies new track lines to an existing surface without the need to regenerate the surface. You can add specific lines to the Surface or you can let the program select the lines to be added automatically.

1. Select the track line layer in the Layers tab of the Control window.
2. Select the track line(s) you want to add to the BASE Surface so it is highlighted.

Menu	Process > BASE Surface > Add to
Tool	

3. Select the Add to BASE Surface command.

The Select Surface dialog box is displayed.

4. Select the surface to which the line is to be added.
5. Click **OK**.

The BASE Surface is updated to include new track lines.

Remove From BASE Surface

Remove a survey line from a Swath Angle or Uncertainty BASE Surface.

1. Select the survey line in the Project tab of the Control window.
2. Select a BASE Surface layer in the Layers tab of the Control window.
3. Select the Remove from BASE Surface command.

Menu	Process > BASE Surface >Remove From
Tool	

The BASE Surface is regenerated without the data from the selected survey line.

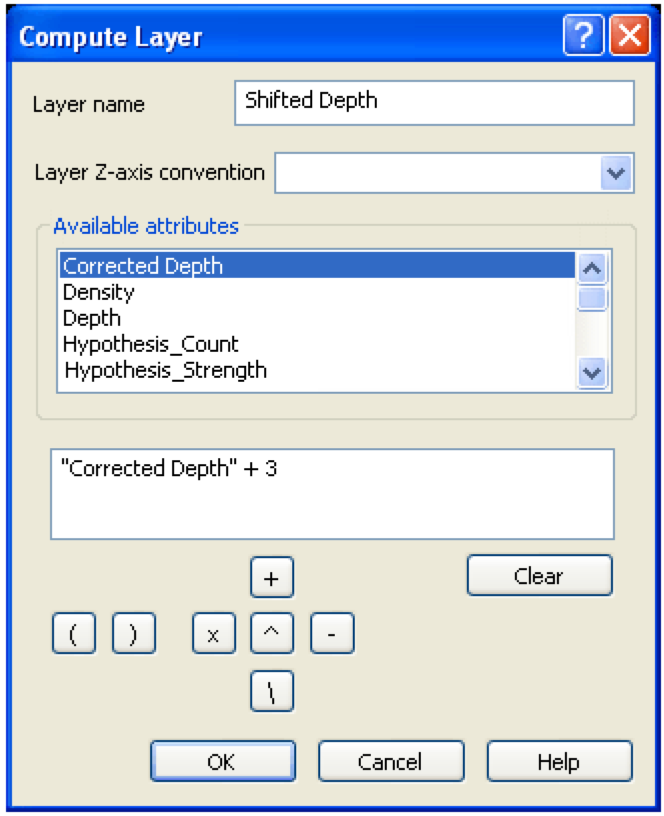
Add Layer

Use the Add Layer command to create a new sub-layer for an existing BASE Surface. The new attribute layer can be created from one or more existing attribute layers, numerical operators and numeric values in an equation. You cannot create a layer for an attribute that is not present in the surface.

1. Select a BASE Surface (parent) layer in the Layers tab of the Control window.
2. Select the Add Layer command from the right-click menu.

Pop-up	Add Layer (Control window only)

The Compute Layer dialog box is displayed.



3. Type a name for the new surface layer in the *Layer Name* field.

The *Layer Z-axis convention* field defines the Z-axis convention for the data in the new layer. The options are:

- Depth with Z-axis positive down: to be used if the data in the new layer will contain depth values that are positive down.
- Height with Z-axis positive up: to be used if the data in the new layer will contain elevation values that are positive up.
- Not applicable: to be used if the data in the new layer will represent generic, non-Z values (e.g., Std_Dev).

4. Select a Layer Z-axis convention from the drop-down list.

The Available Attributes list displays the attribute layers currently present in the selected surface.

5. Double-click an attribute in the *Available Attributes* list to insert it into the conditions field below.

> If there are spaces in the name of the selected attribute, place quotation marks around the attribute in the conditions field, (e.g., "attribute name").

Use your keyboard and the condition buttons to create the equation for the new layer. The example above shows the equation to create a new depth layer with values increased by 3 metres.

6. Click any of the operators to add it to the field.

Operator	Function	Displays
( )	parentheses	(Depth)
+	addition	Depth + 4
X	multiplication	Depth * 4
–	subtraction	Depth - 4
\	division:	Depth / 4
^	exponentiation (Depth^{X})	Depth ^ 4

To add values:

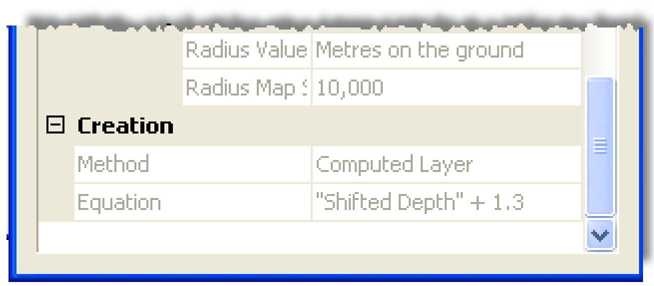
7. Click at the point in the equation where you want to enter a numerical value, and type the number.

Use **Backspace** to remove parts of an equation, or click **Clear** to remove everything from the conditions field.

8. Click **OK** to compute the new layer.

The new layer is drawn in the Display window and is listed as a BASE Surface child layer in the Layers tab.

The properties of the new layer can be viewed in the *Creation* fields of the Properties window: the *Method* field shows it is a computed layer, and the *Equation* field shows the variables in the equation.



Recompute surface

The Recompute function rebuilds a BASE Surface and regenerates the surface image.

If you have updated the BASE Surface since it was interpolated, you can apply the Recompute command to rebuild the interpolated surface.

Menu	Process > BASE Surface > Recompute
Tool	

1. Select a Surface name in the field sheet file tree so it is highlighted.
2. Select the Recompute BASE Surface command.

Only the track lines used previously in the Surface are processed during rebuilding.

The Recompute operation checks to see if the surface makes use of HIPS data that is no longer present.

If data is missing, a dialog box lists what is missing. You then have the option of terminating the Recompute process, or continuing without the missing data. If you choose to continue, the replacement surface will have the missing data lines removed from its list of contributing lines.

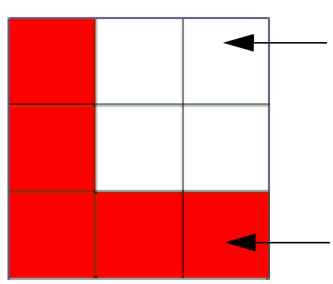
> When recomputing a CUBE BASE Surface all CUBE Surface editing, including nominations, hypothesis and node rejections are considered no longer valid and will thus not be retained after recomputation of the BASE Surface.

Interpolate BASE Surface

The Interpolate command eliminates small holes that appear in areas of sparse data. This sometimes happens in the outermost beams along the outside edges of surveyed areas where there is little or no overlapping coverage.

This command examines each Surface node to determine if it contains a pixel value. If the node does not contain a value, the neighbouring pixels are examined to determine if enough of them contain pixel values to justify interpolating a value for the hole. This limits the interpolation to holes in the BASE Surface and prevents it from expanding the surface outward from the survey area. A copy of the BASE Surface, with these gaps filled, is then created.

In the following example of a 3 x 3 Surface node area, five nodes have pixel values (shown in red).



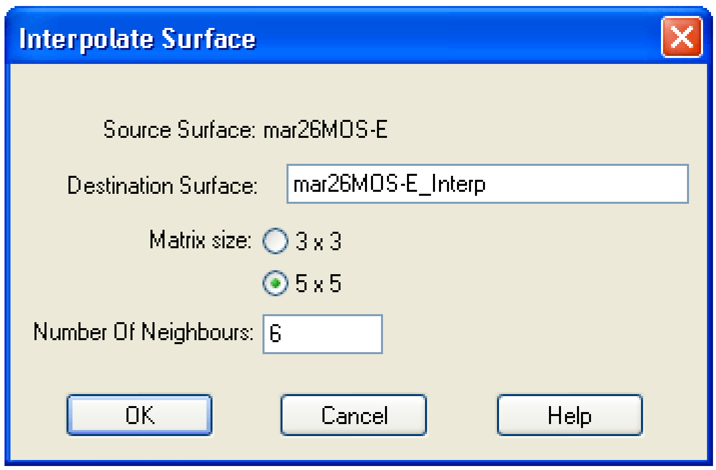
This means if you select five or less as the value for the *Number of Neighbours*, a pixel value is created for the node without a value from the neighbouring nodes.

To interpolate a surface:

1. Select a BASE Surface layer in the Control window so it is highlighted.
2. Select the Interpolate command.

Menu	Process >BASE Surface> Interpolate
Popup	Interpolate

The Interpolate Surface dialog box is displayed.



The *Destination Surface* field displays the file name of the copy that is created from the original BASE Surface. The default name is the original Surface name with _Interp appended to it. You can change the name, if necessary.

3. [Optional] Type a new name in the *Destination Surface* field.

The *Matrix Size* determines the number of nodes closest to the node that can be used to interpolate a pixel value.

4. Select either the *3 x 3* or the *5 x 5* option.

The *Number of Neighbours* field is a threshold level used to determine the minimum pixels in the matrix area that must be present to interpolate a new pixel value. The matrix size determines the maximum and minimum values for the neighbours field. For example, if you select 3 x 3 for the matrix size, then the possible maximum number of neighbouring nodes with pixels is 9 and the minimum number is 3.

5. Type the number of neighbouring nodes used to interpolate pixels.
6. Click **OK**.

The interpolated Surface is created and displayed with an _Interp extension, in the field sheet data tree in the Layers tab.

As you continue to re-interpolate the Surface, the effects accumulate to remove any remaining gaps. To interpolate again:

1. Select the interpolated Surface in the Layers tab.
2. Select the Interpolate Surface command.

The second interpolation is applied to the selected Surface layer. It does not create a new surface layer.

Every time an interpolated Surface is re-interpolated, the parameters are stored. Thus, when the Recompute command (see "Recompute surface" on page 221) is used on an interpolated BASE Surface, all interpolations are performed on the Surface layer.

An interpolated BASE Surface cannot be updated with the Automatic BASE Surface Update setting in the Options dialog box.

Finalize BASE Surface

A finalized BASE Surface is a finished version of the surface that is ready for export or for further processing (for example, to create a Product Surface). Most importantly, finalizing a surface ensures that designated soundings are included in the final surface, to be carried through to bathymetric products.

These three operations can be applied when finalizing a BASE Surface:

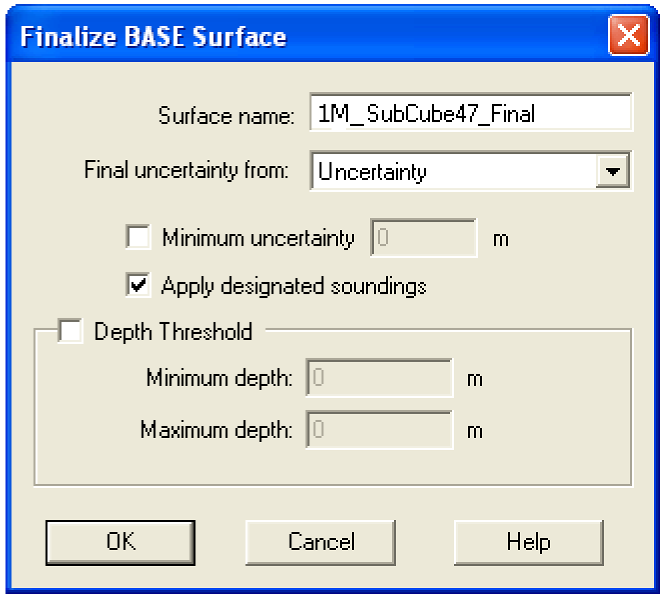
- **Minimum uncertainty**: Vertical uncertainty can fall to almost zero if a node contains too many soundings. This function applies a user-defined vertical uncertainty to each node to compensate for this situation.
- **Designated soundings**: If the data set contains designated soundings, the Surface is regenerated and the designated soundings' depth values are applied to the nearest nodes. (See "Critical Soundings" on page 297).
- **Depth thresholds**: A Finalized BASE Surface can be modified to represent a range of depths levels in the field sheet. When the Finalized Surface is generated, only these depths are displayed.

To create a finalized BASE Surface:

1. Select a BASE Surface layer in the Layers tab of the Control window.
2. Select the Finalize BASE Surface command.

Menu	Process > BASE Surface > Finalize

The Finalize BASE Surface dialog box is displayed.



The *Surface name* field shows the name that will be applied to the finalized surface. This defaults to the name of the source BASE Surface with _Final appended to the end.

To change the name:

1. Type a different name for the finalized surface.

The *Final uncertainty from* drop-down list gives three options as the source of values for calculating the uncertainty of the finalized surface:

- Uncertainty: The vertical uncertainty values from the Uncertainty layer of the source surface.
- Std Dev.(scaled to 95% CI): The standard deviation values from the source surface (scaled to 95% confidence interval).
- Greater of the two: The larger of the two above values.

2. Select a final uncertainty option from the drop-down list. (This option is greyed out if the source surface does not contain uncertainty values, for example, if it is a swath angle surface.)
3. Select the *Minimum Uncertainty* check box and type a depth value to represent the minimum vertical uncertainty value.
4. Select the *Apply Designated Soundings* check box to use this option.

   When this option is selected, designated soundings take precedence over other soundings during the soundings selection process.

   If this option remains un-selected, designated soundings are processed no differently from other soundings.

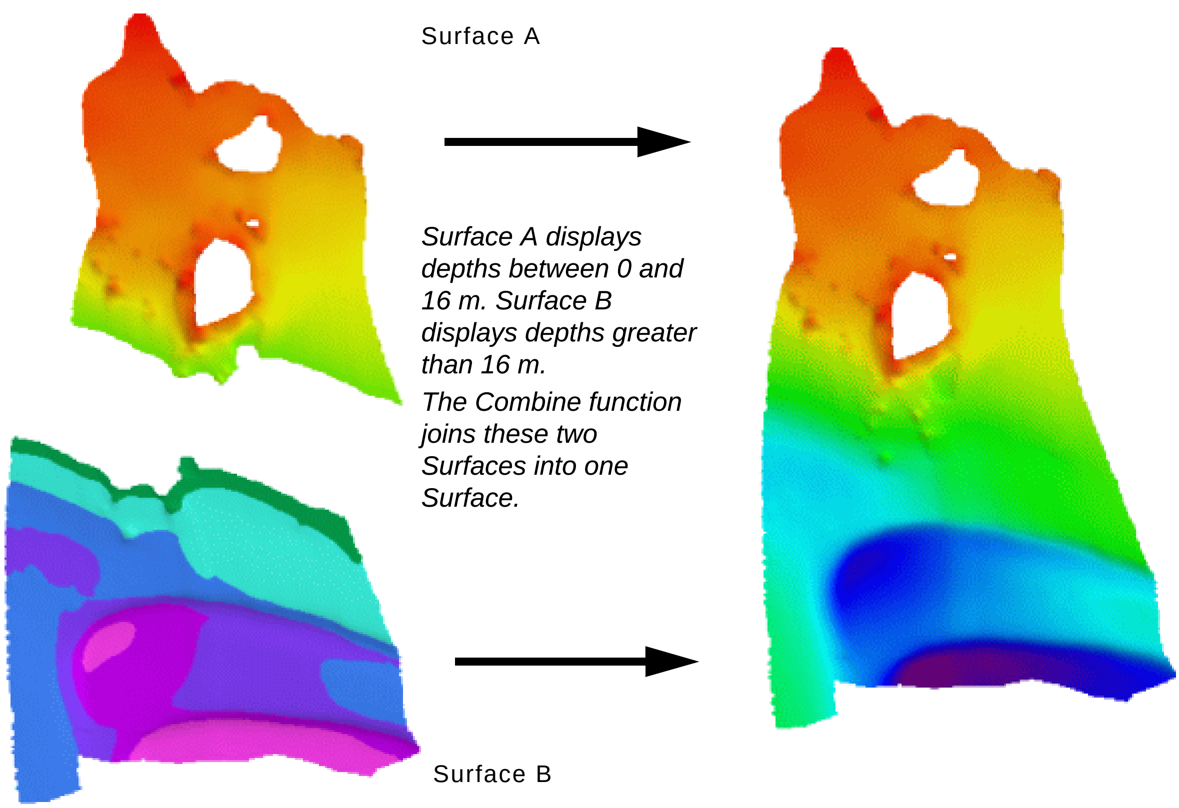
Using Depth Threshold values allows the finalized surface to contain only a specified range of depth nodes. The finalized surface will be “clipped” so that the surface will only contain depths where the surface resolution is optimal.

5. Select the *Depth Threshold* check box to create a finalized Surface that displays only node depths between the minimum and maximum depth values.
6. Type the minimum and maximum depth values in their respective fields.
7. Click **OK**.

A finalized BASE Surface is generated and visible in the Display window. The finalized BASE Surface is listed in the Layers tab of the Control window.

Combine

The Combine options stitches two or more BASE Surfaces together to form one surface. This option is used when you have created finalized BASE Surfaces with different depth thresholds (see “Finalize BASE Surface” on page 224), or when you have two adjacent Surfaces with different resolutions. The geographical extent of the combined BASE Surface is determined by the geographical extent of the field sheet.

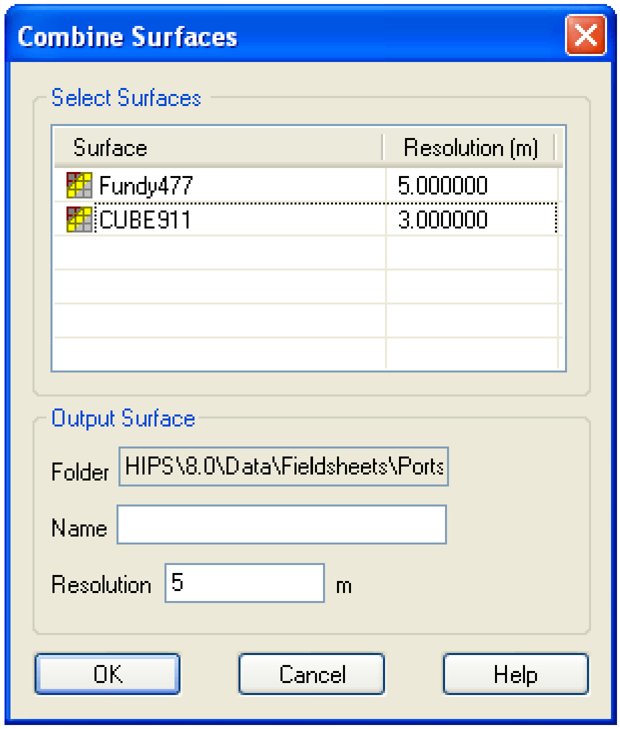


Surface A displays depths between 0 and 16 m. Surface B displays depths greater than 16 m.

The Combine function joins these two Surfaces into one Surface.

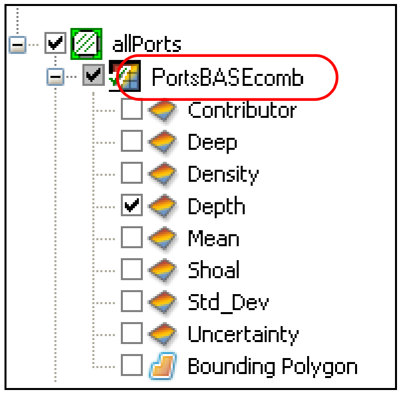
Menu	Process > BASE Surface > Combine
Pop-up	New > Combine Surfaces (Control window only)

1. Select the field sheet that contains the BASE Surfaces you want to combine.
2. Select the Combine command to display the Combine BASE Surfaces dialog box.



3. Select the BASE Surfaces from the file tree by two methods (you may have to click the expand icon to view all the available Surfaces):
   - Press <Ctrl> and select individual surfaces with the mouse.
   - Press <Shift> and select a range of surfaces by clicking the first and last surface in the range. All surfaces between the first and last surface are selected.
4. The Output Folder field is read-only and displays the directory path of the selected field sheet. The combined BASE Surface is saved to this folder.
5. Type a name for the combined Surface.
6. Type a resolution for the combined Surface
7. Click **OK**.

A new BASE Surface that combines the selected Surfaces is generated. The new combined surface is listed in the Layers tab.



The combined BASE Surface reproduces all the attribute layers contained in the source Surfaces.

Difference Surfaces

Create a surface showing the differences in attribute values between two BASE Surfaces.

The Difference Surface function is useful for comparing changes to an area. A surveyed area can change considerably over time due to the redistribution of sediment with the currents. This redistribution can cause changes in depths. A difference surface can be used to find these changes by comparing two surfaces for the same area that were created at different times.

Note: the difference surface is not part of a field sheet and is not saved with the project. However, it can be opened using the Open Background Data command.

To use this function, the surfaces to be compared must cover part or all of the same area, and the data in the attribute layers being compared must be of the same type.

The data types include:

Type	Size/Format	Range
Short	16-bit (2-byte), integer	-32,768 to 32,767 (no decimal points)
Integer	32-bit (4-byte), integer	-2,147,483,648 to 2,147,483,647 (no decimal point)
Float	single-precision, IEEE 754 32-bit (4-byte), floating-point	1E-44 to 3.4E+38 (positive or negative)
Double	double-precision, IEEE 64-bit (8-byte), floating-point	negative values: -1.79769313486231570E+308 to -4.94065645841246544E-324 positive values: 4.94065645841246544E-324 to 1.79769313486231570E+308

For example, Depth layers can be compared, but you can't compare a Density layer to a Depth layer, since the former contains Integer data and the latter is Float.

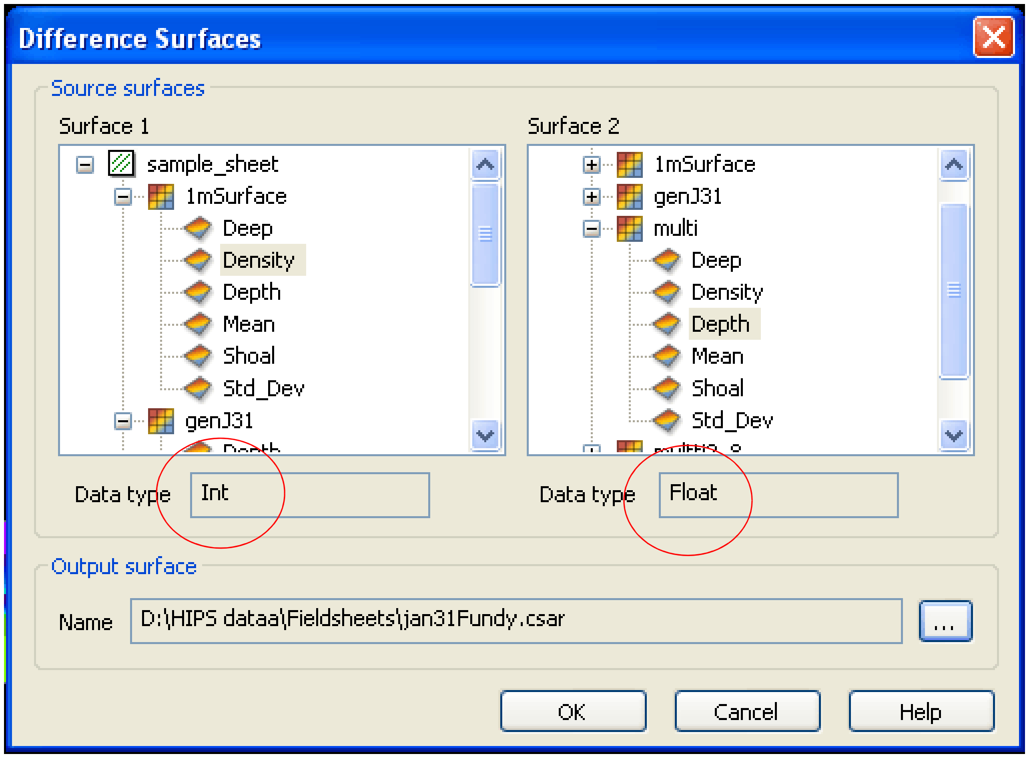
When you select a layer in the Difference Surface dialog box, its data type is displayed in the *Data Type* field.

To compare two open surfaces:

1. Select a surface or attribute layer in the Layers tab.
2. Select the Difference Surfaces command.

Menu	Process > BASE Surface > Difference Surfaces

The Difference Surface dialog box is displayed.



Any open surfaces are listed in the Surface fields.

3. Click the expand (+) icon beside a layer in the *Surface 1* list to view the available attribute layers.
4. Select an attribute layer to use as the data source for the difference calculation.

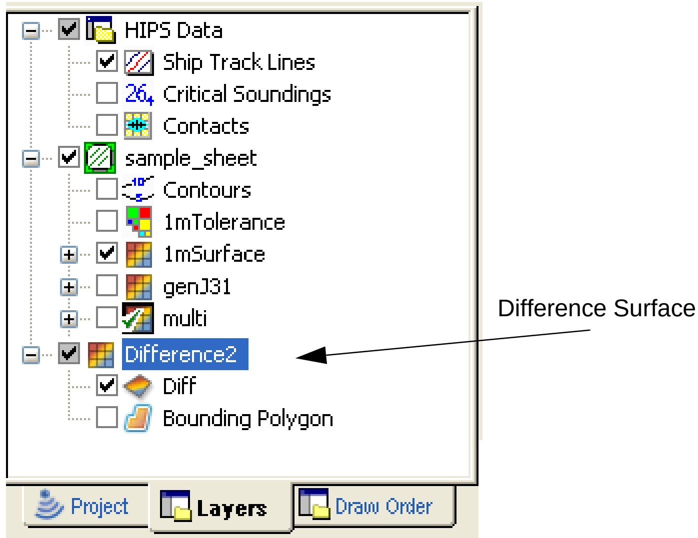
The *Data Type* field displays the data type of the selected layer.

5. Expand the *Surface 2* list and select an attribute layer with the same data type as the layer selected for *Surface 1*.

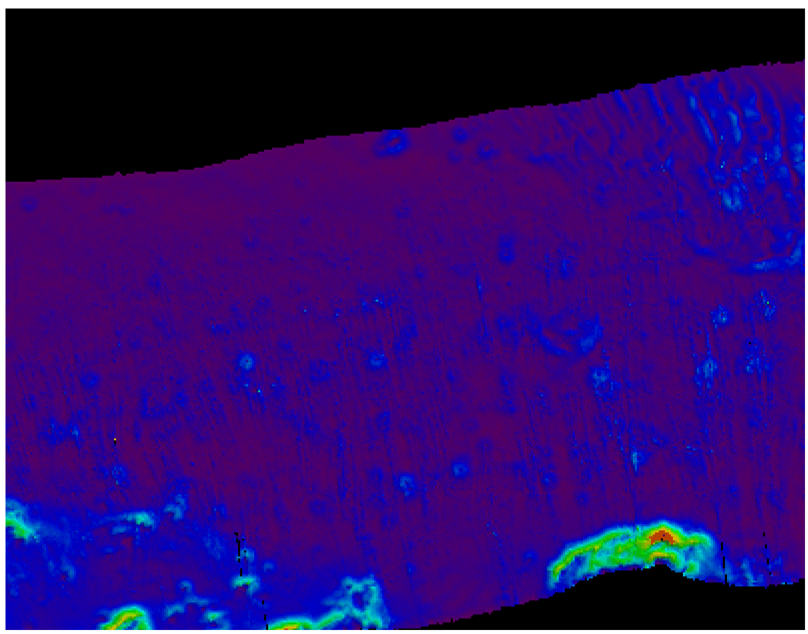
The *Data Type* field displays the data type of the second selected surface attribute.

6. Click the Browse button (**...**) to define the name and location for the new surface file, which is saved in CSAR format.
7. Click **OK**.

A new surface is generated, with an attribute layer named Diff and a Bounding Polygon layer. The resolution of the new surface will be the same as the resolution of the surface selected as Surface 1.



An example of a Difference Surface is shown below.



The colours represent the range of discrepancies between the two compared surfaces. The dark areas of this surface represent greater depth differences than the light areas. If the contrast is not clear, you can reverse the colours, or change the colour map settings or set a colour range, in the Properties window. (See "Colour Map Editor" on page 39 and "Colour Range Editor" on page 43 of the Tools guide.

Compute Surface Statistics

The Compute Statistics command generates statistics on the nodes in an attribute layer of a surface, such as a BASE surface, mosaic or GeoBaR.

The statistics calculated are:

- Minimum value in the attribute layer of the surface
- Maximum value in the attribute layer
- Mean of the values in the layer
- Area
- Standard Deviation (Std_dev)
- Total count

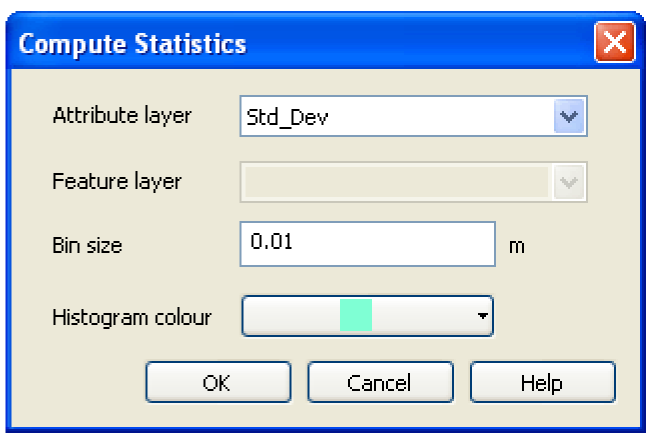
These statistics are reported together with a histogram depicting the distribution of the points/nodes within the min/max range of values.

To compute the statistics of an attribute layer:

1. Select the parent layer of the attribute layer.
2. Select the Compute Statistics command.

Menu	Process > BASE Surface > Compute Statistics
Pop-up	Compute Statistics (Layers window only)

The Compute Statistics dialog box is displayed.



3. Select the *Attribute layer* for which you want to compute statistics, for example, the Depth or Std_Dev layer.

Bin Size

The Bin size field defines the range of each grouping of data that will be displayed as a column in the histogram. Each bin or column will contain all the data within the range set by the bin size.

The unit of measure for the Bin size field is controlled by the type of data in the selected attribute layer. For example, Depth, Standard Deviation and Mean layers are reported in the units set for the data in Tools > Options > Display > Units. Density and Hypothesis Count are unitless.

4. Enter a *Bin size* for the histogram.

> The Min and Max Range values in the Properties for the selected attribute layer can help you set a useful bin size for the histogram.

For example, if the range of values for the Std_Dev layer is 0 to 16, a bin size of .02 m will yield a more useful histogram than would 2m bin size.

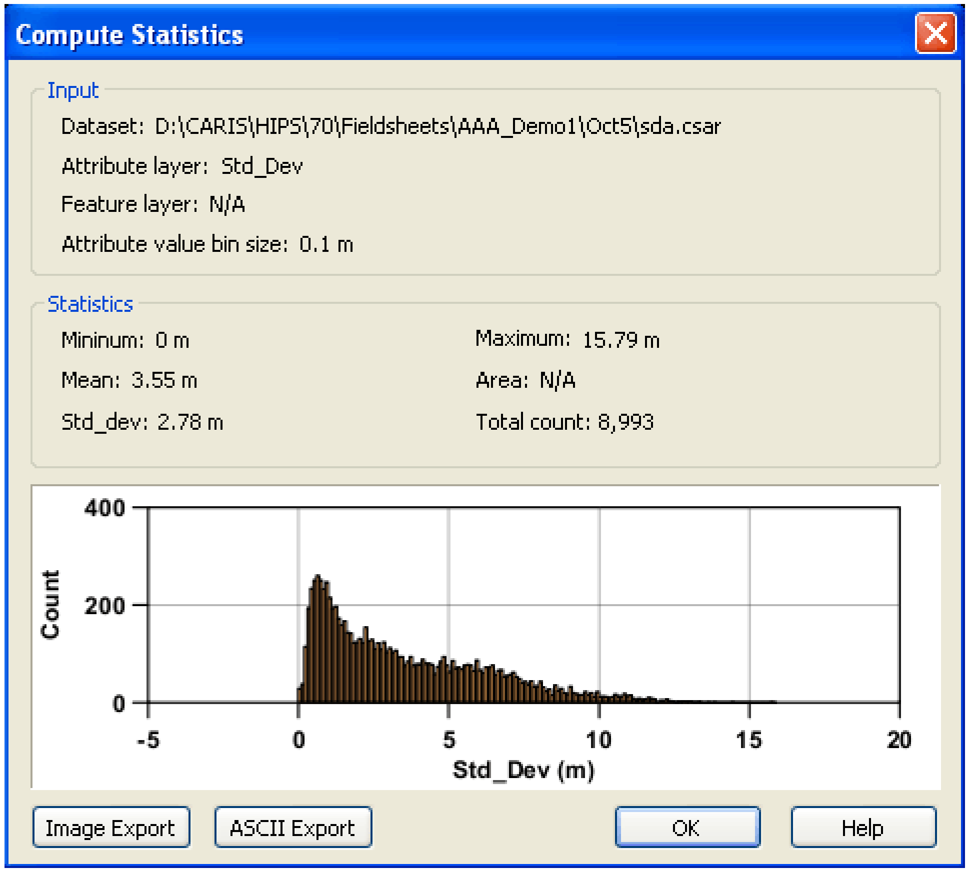
5. Select a colour for the content of the columns from the *Histogram colour* drop-down list. The default colour is white (outlined in black).
6. Click **OK**.

Statistics output

The Compute Statistics dialog box displays the input data and the computed statistics as values and as a histogram.

- The *Input* section displays the settings that were used to calculate the statistics: data set, attribute layer and bin size.
- The *Statistics* section provides the results of the analysis of the data in the selected attribute layer.
- The histogram displays the distribution of the data. The X axis is labelled with the source layer name and unit of measure. The Y axis contains the count of the data points in each bin or column.

The histogram below displays the statistics generated for the standard deviation layer of a BASE surface.



To save these statistics, export them to image or ASCII formats. When exporting to an image format, only the histogram is exported.When exporting to ASCII, the *Input*, *Statistics* and histogram data will all be exported.

See "Export Histogram to Image" on page 234 and "Export Statistics to ASCII" on page 237 for more information.

7. Click **OK** to close the dialog box when finished.

Export Histogram to Image

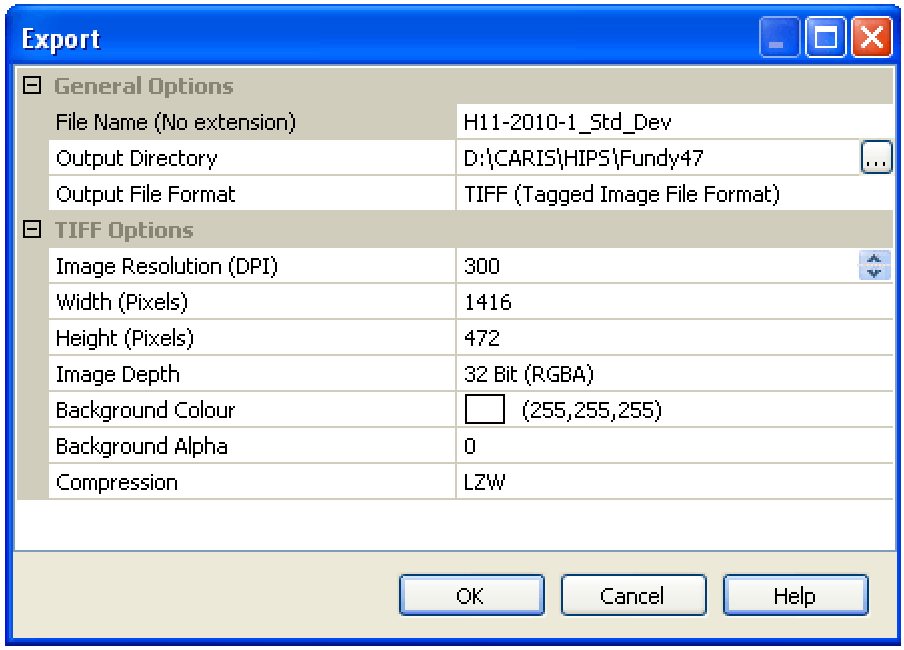
Use the Image Export option in the statistics results dialog box to export an image of the current histogram as:

- PDF (Portable Document Format)
- PS (PostScript)
- SVG (Scalable Vector Graphics)
- TIFF (Tagged Image File Format)

To export an image of the histogram:

1. Click **Image Export**.

The Export dialog box is displayed.



Use the Export dialog box to define the settings for exporting the image. Click on each row to activate the options for that field. The example above displays the options for export to TIFF format.

By default, the *File Name* field is populated with a name combining the names of the surface and the attribute layer that was computed (`<surfacename>_<layername>`). You can change this file name.

2. [Optional] Type a *File Name* for the image file.
3. Click within the *Output Directory* field to enable the **Browse** (...) button.

4. Click **Browse** and select a location for the exported file.
5. Select the *Output File Format* from the drop-down list.

Once you select a format, specific options for that format are displayed in the lower section of the dialog box.

6. [Optional] Enter or select a value for the options as needed.

Export options:

PDF and PostScript options	
Paper size	Set the page size of the exported file.
Width/Height	Set the width and height of the histogram in the resulting file.
Data Orientation	Set the orientation of the histogram in the resulting file. Portrait is a vertical position; Landscape is a horizontal position. Landscape is selected by default.
Margins	Set the width of the area between the histogram and the edge of the file. The default is 5mm.

Image Export DPI (Quality)	Set the resolution used to export the histogram. The higher the resolution, the better the quality of the image when zoomed in. If the file will be used in the electronic form with the image being zoomed in tightly, a higher resolution is recommended. If the file will simply be printed, a smaller resolution is adequate.
SVG format options	
Width/Height	The width and height of the histogram in the resulting file.
Image Export DPI (Quality)	Set the resolution used to export the histogram. The higher the resolution, the better the quality of the image when zoomed in. If the file will be used in the electronic form with the image being zoomed in tightly, a higher resolution is recommended. If the file will simply be printed, a smaller resolution is adequate
TIFF format options	
Image Resolution	The resolution (quality) at which to export the image. The higher the resolution, the closer the image can be zoomed, but the larger the file.
Width/Height (Pixels)	Set the width and height of the resulting image in pixels. These fields are controlled by the resolution of the image. As the DPI is increased or decreased, so is the number of pixels.
Image Depth	Set the number of values applied to each pixel in the image (one value for each colour, plus one for transparency if using 32 Bit). If you would like a transparency setting applied to the background colour, the 32 Bit (RGBA) option must be selected.
Background Colour	The colour displayed in the background of the histogram. The default is black.
Background Alpha	(This option is only available when the Image Depth is set to 32 Bit RGBA). Set the degree of transparency applied to the background colour. The default setting is zero transparency.
Compression	Compress the resulting image file during export to decrease the file size. This option is only available when using the TIFF format. There are various compression methods available, each of which performs differently. By default, LZW is used.

7. Click **OK**.

The image file is saved to the specified location.

Export Statistics to ASCII

Use the ASCII Export option in the Compute Statistics results dialog box to export all of the statistical information to an ASCII text file. This file will include:

- Information about the computed surface
- The resulting statistics
- The centre value of each bin and the count of data points within the bin.

To export to an ASCII file:

1. Click **ASCII Export**.

A standard Save As dialog box is displayed. By default, the *File Name* field is populated with a name combining the names of the surface and the attribute layer that was computed (`<surfacename>_<layername>`).

2. [Optional] Enter a new *File name.*
3. Select a location for the file and click **Save**.

An ASCII text file is created in the specified location.

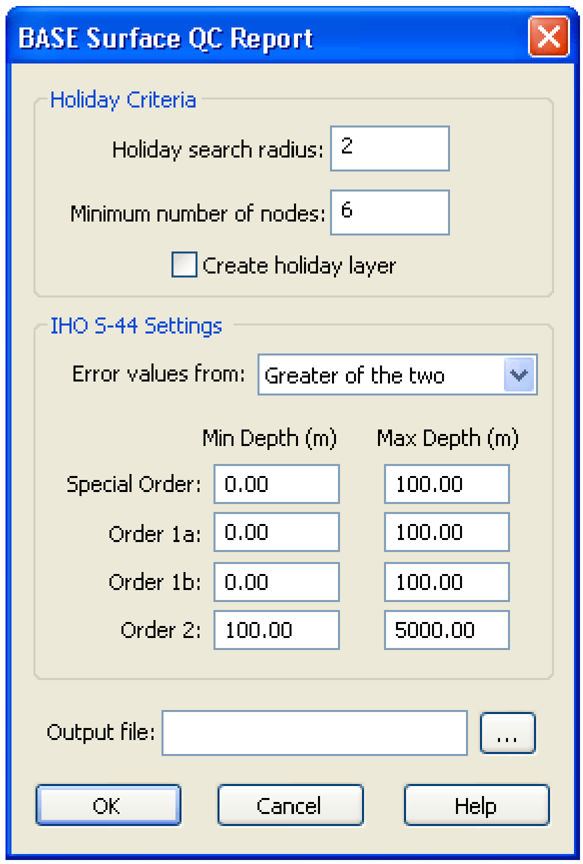
BASE Surface QC Report

The BASE Surface Quality Control report lists significant data gaps within the surface. It also analyzes how much of your data conforms to the confidence levels set by the S-44 standard for the four orders of survey.

Menu	Process > BASE Surface > Surface QC Report
Pop-up	QC Report (Control window only)

1. Select the BASE Surface parent layer in the Control window.
2. Select the Surface QC Report command.

The BASE Surface QC Report dialog box is displayed.



Holiday Criteria

The top section of the dialog box sets the criteria for finding significant data gaps (known as “holidays”). A holiday is formed when enough gaps are clustered together within a specified radius. You can create a layer that displays these holidays. This layer does not display every gap in the surface, but only those gaps that meet the criteria.

1. Type the number of nodes to use as a radius for searching for gaps in the *Holiday Search Radius* field.
2. Type the *Minimum number of nodes* that must contain data so that a data gap holiday is not formed.

For example, if the value is 6, then a node without data must be surrounded by five or less nodes with data to be included (with other empty nodes) in a holiday.

3. Select the *Create holiday layer* check box to create a layer in the Layers tab displaying the data gaps selected by the search criteria.

IHO S-44 Settings

The lower section compares uncertainty values contained in the surface to the S-44 standard. These values are drawn from the Standard Deviation and Uncertainty layers. This comparison will show how many nodes in the surface fall within the S-44 categories.

4. Select the source of error values from the drop-down list:
   - Uncertainty layer (which contains the uncertainty “model” values entered when the surface was created),
   - Standard Deviation (the values showing how the data varies from the “model”),
   - Greater of the two (uncertainty or standard deviation, whichever is larger value)
   - Lesser of the two.

> For a swath angle surface, the error values come from the Standard Deviation layer only. (The option will be greyed out, but it is applied.)

The depth levels shown in the dialog box are the minimum and maximum depth levels for each survey order. You can change these to conform with the minimum and maximum depths in your surface.

5. [Optional] Type new depth levels for any of the survey orders. (You can have overlapping depth levels.)
6. Click **Browse** to select a directory path and name for the QC report. The path and name is shown in the *Output File* field.
7. Click **OK**.

The report is shown in the Output window, and in the Output file you designated. The Holiday layer is displayed in the Layers tab under the other layers for the surface.

QC Report File example

```
BASE Surface QC Report
Date and Time: 2/4/2010 3:39:49 PM
Surface:
d:\CARIS\HIPS\70\Fieldsheets\FundyData\PASG1\pagstest1.csar
Holiday Search Radius: 2
Holiday Minimum Number of Nodes: 12
Holiday layer created: No
Error values from: Greater of the two

Number of nodes processed: 8859
Number of nodes populated: 8858 (99.99%)
Number of holidays detected: 0
IHO S-44 Special Order:
    Range:  0.000 to 100.000
    Number of nodes considered: 8858
    Number of nodes within: 2523 (28.48%)
    Residual mean: 0.743
S-44 Order 1a:
    Range:  0.000 to 100.000
    Number of nodes considered: 8858
    Number of nodes within: 3995 (45.10%)
    Residual mean: 0.473
S-44 Order 1b:
    Range:  0.000 to 100.000
    Number of nodes considered: 8858
    Number of nodes within: 3995 (45.10%)
    Residual mean: 0.473
S-44 Order 2:
    Range: 100.000 to 5000.000
    No depths within the specified range
```

Sections of a QC report file

The first section contains information about the report file and the data holiday options set in the dialog box:

- The data and time that the QC report file was created.
- The BASE Surface on which the QC report is based
- The holiday search radius.
- The minimum number of nodes required to omit a node (without data) from being included in a holiday.
- If a holiday layer was created or not.
- The source of error values (the layer).

The second section contains information on the number of nodes containing data:

- The total number of nodes processed in the surface.
- The total number of nodes that contains data.
- Percentage of the surface that contains data.
- Number of data holidays located.

The third section contains information about each S-44 survey order:

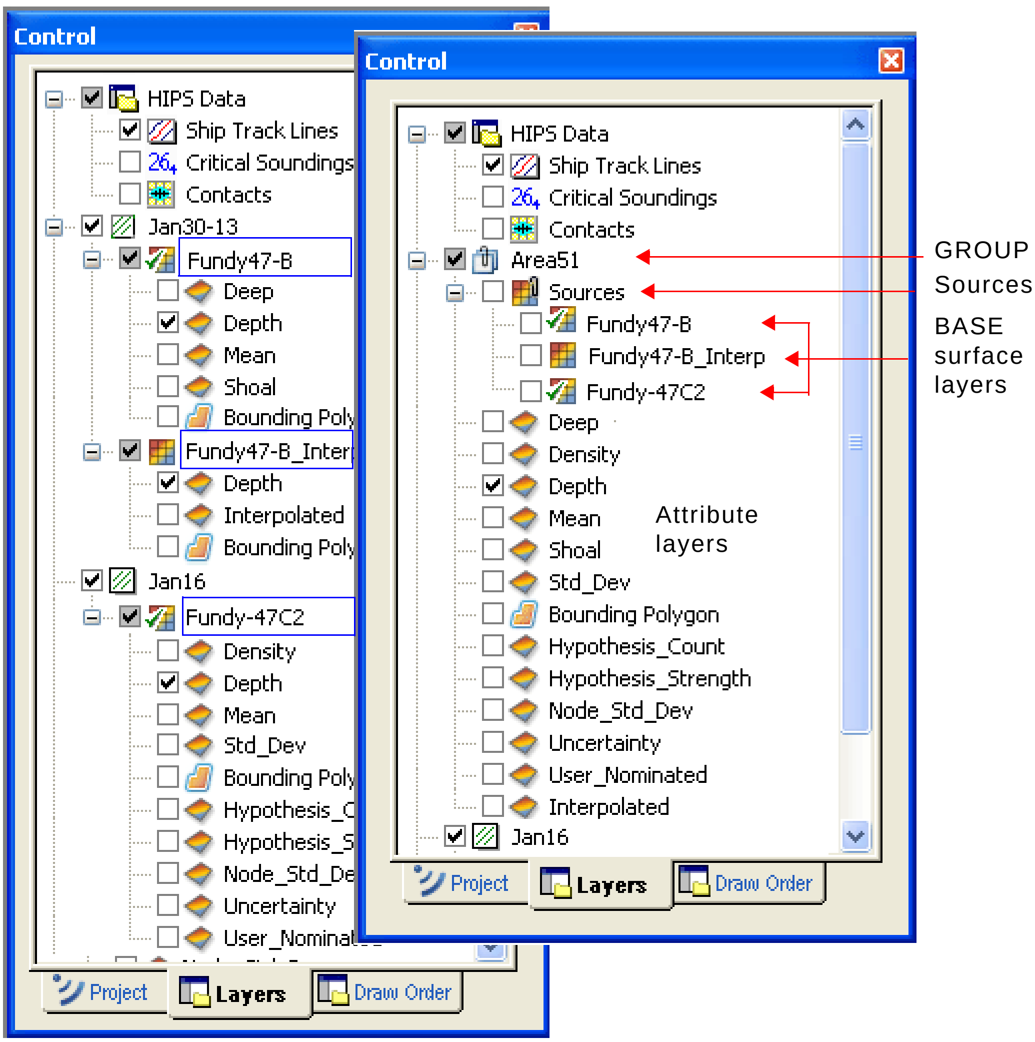
- The depth range for the survey order (set in the dialog box).
- The number of nodes within that depth range with uncertainty values that were considered for the survey order.
- The number of nodes that fall within the depth error limits for that survey order.
- The residual mean is derived from the computed and reported uncertainty.

  If the value is negative, the uncertainty values meet the requirements for the selected survey order. If the value is positive, the uncertainty values do not meet the requirements for the selected survey order. Therefore, if no nodes were considered for the survey order, the residual mean would have to be above zero.

Group multiple surfaces

Multiple BASE surfaces created in different field sheets can be grouped together, so that the same display properties can be applied consistently to all the grouped surfaces. (See "PROPERTIES OF BASE SURFACES" ON PAGE 204.

Grouped surfaces are listed together under a new Sources layer, followed by their combined attribute layers, as in the illustration below.



When a surface is added to an existing group, the properties of the first of each type of attribute layer included in the group are applied to all other attribute layers of the same type.

For example, if the Depth layer for the grouped surfaces has Filter values set, the Depth layer for a surface added to the group will display data within the filter range, regardless of its settings before it was added to the group.

Create a group

To group multiple surfaces:

1. Right-click on a surface parent layer in the Layers tab.
2. Select the Create New command from the Group item on the pop-up menu.

Pop-up	Group > Create New

The New Group dialog box is displayed.

3. Type a name for the group layer in the Group Name field.
4. Click **OK**.

The group is created in the Layers tree, and the selected surface is added to the group.

Add surface to group

To add another surface to the group:

1. Right-click on the surface to be added.
2. Select Group from the pop-up menu.

The pop-up sub-menu will list any created and available groups.

(If a selected surface already belongs to a group, that group is greyed out in the pop-up menu, to prevent a surface from being added to a group twice.)

Pop-up	Group > *group name*

3. Select the group you want to add the surface to.

The surface is added to the group, under the Sources layer.

Remove a surface from a group

You can remove a surface from a group. It will be returned to its place in the field sheet (if the field sheet is open)—otherwise the surface is listed with other layers in the Layers tab.

To remove a surface from a group:

1. Right -click on the surface you want to remove from the group.
2. Select Group from the pop-up menu.

Pop-up	Group > None

3. Select None from the sub-menu.

The surface is removed from the group and restored to a position in the Layers tab.

Ungroup

To disband the group and restore the component surfaces, use the Ungroup command:

1. Right-click on the group layer.

Pop-up	Group > Create New

2. Select Ungroup from the pop-up menu.

The group is removed from the layers tree.

Surface Filtering

Surface filtering uses a scaled standard deviation or uncertainty threshold to reject soundings that fall outside that threshold. You can also filter sounding depth by a direct threshold amount, for example, 2 metres from the surface.

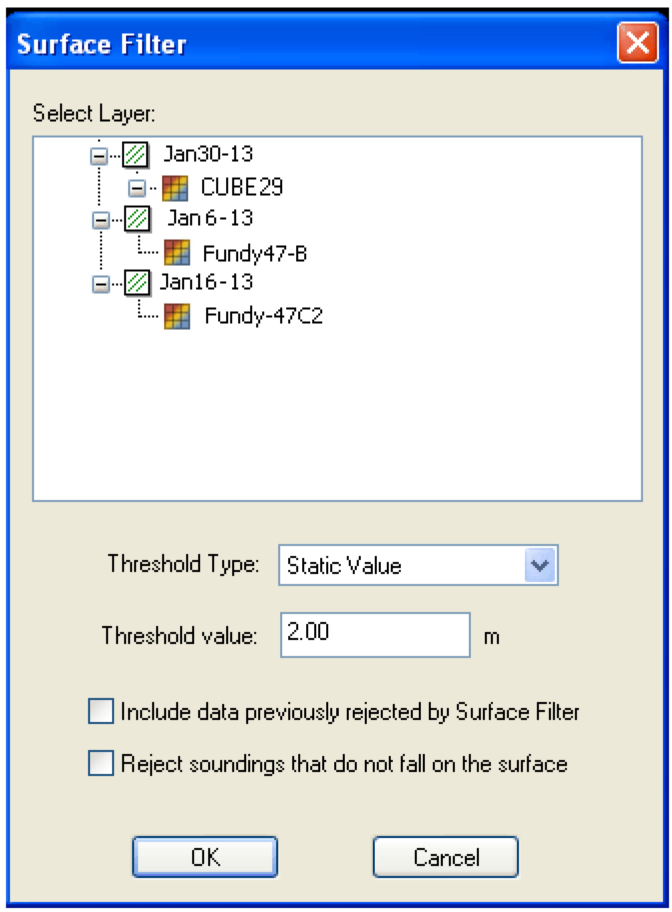
Soundings that fall outside the threshold are assigned a Rejected by Surface Filter flag and are no longer available for processing (unless they are given an Accepted flag again).

This can greatly reduce the number of manual edits required to produce a clean sounding set.

Menu	Tools > Apply Filters > Surface Filter

1. Select the Surface Filter command.

The Surface Filter dialog box is displayed.



2. Select the BASE surface to be filtered from the *Select Layer* tree.
3. Select the *Threshold Type* from the drop-down list:
   - Standard Deviation
   - Uncertainty
   - Greater of the two
   - Lesser of the two.

- Static Value

4. Set the desired *Threshold value.*
   - For standard deviation or uncertainty thresholds, the value entered will show the confidence interval next to the field.
   - Static values are entered in units set in Tools > Options > Display > Units > Vertical units, e.g., metres.
5. Select the *Include data previously rejected...* check box to include rejected data when running the filter.
6. Select the *Reject soundings* check box to reject soundings that are offset from the surface.
7. Click **OK**.

Surface filtering can also be used in Subset Editor, where you can apply filters to a *subset* of a BASE surface. See "Subsets and BASE Surfaces" on page 350.

11

CUBE Processing

A Combined Uncertainty and Bathymetry Estimator (CUBE) surface uses multiple hypotheses to represent potential depth variances along the sea floor.

In this chapter...

OVERVIEW 248
GENERATING CUBE SURFACES.................................. 251
HYPOTHESIS EDITING.. 259

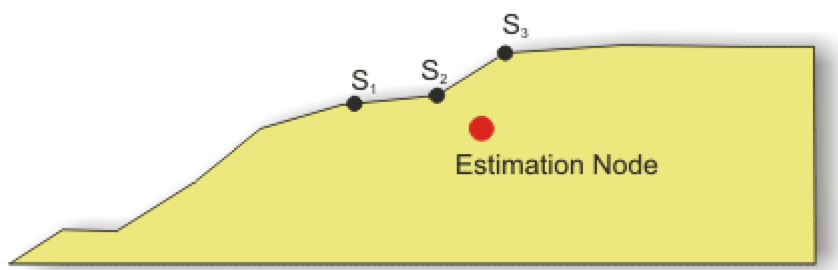
Overview

The CUBE process uses sounding propagation along with disambiguation to create and select hypotheses.

When soundings are propagated to a grid of estimation nodes:

- Soundings with a low vertical uncertainty are given more influence than soundings with high vertical uncertainty
- Soundings with a low horizontal uncertainty are given more influence than soundings with a high horizontal uncertainty.
- Soundings close to the node are given a greater weight than soundings further away from the node.

Generally, as soundings are propagated to a node, a hypothesis (depth value) is developed at that node. If a sounding's value is not significantly different from the previous sounding then the same or modified hypothesis is used. If the value does change significantly, a new hypothesis is created. A node can contain more than one hypothesis.



In the above graphic, two soundings—S_1 and S_2—have similar values and therefore are part of the same hypothesis at the estimation node. However, sounding three (S_3) has a significantly different value so it forms a new hypothesis.

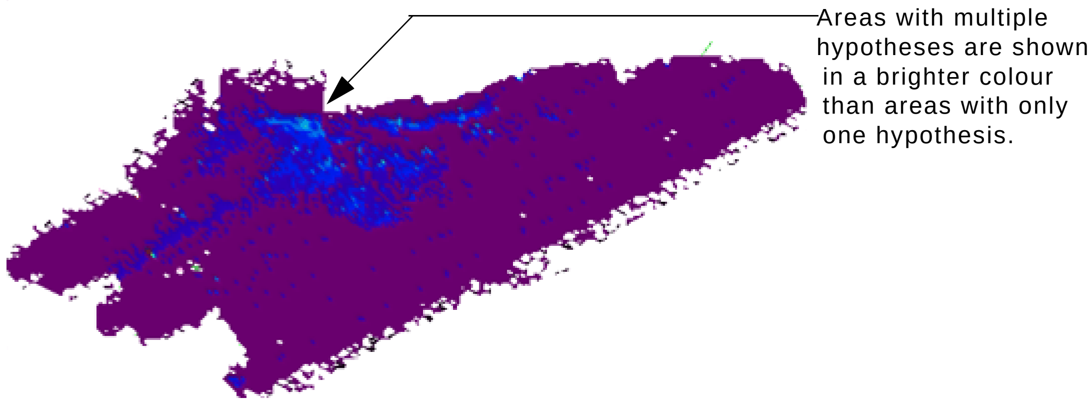
Disambiguation

The final process in CUBE is disambiguation. Disambiguation selects one hypothesis over others. There are four disambiguation options:

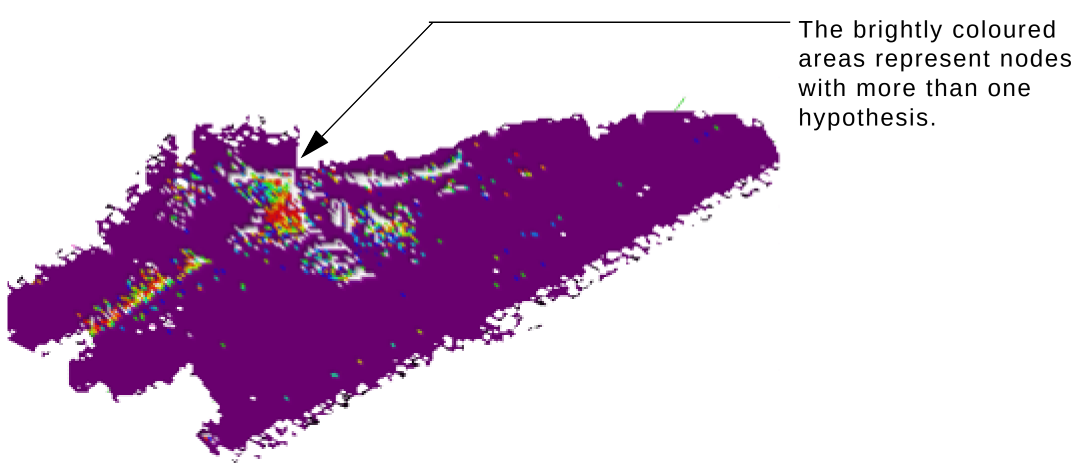
- Density: Select the hypothesis with the greatest number of sounding samples.
- Locale: Select the hypothesis that is most consistent with the surrounding nodes that have only one hypothesis.
- Locale and Density: Select the hypothesis the contains the greatest number of soundings and is also consistent with neighbouring nodes.
- Initialization: Select the hypothesis that is nearest to a node value of a previously created BASE Surface. Initialization differs from the other methods because it filters potential outlier soundings just prior to disambiguation.

When a surface using the density and locale options is generated, two layers specific to the CUBE Surface are displayed.

- The Hypothesis Count layer is a visual representation of hypothesis density at a node. A Surface with a Hypotheses Count layer is displayed below.



- The Hypothesis Strength layer is a visual representation of the mathematical confidence of a chosen hypothesis. Each node is given a value ranging from 0.0 (high confidence) to 5.0 (low confidence). Nodes with one hypothesis have a confidence value of 0.0 while nodes with multiple hypotheses will have confidence values greater than 0.0. A Hypothesis Strength layer is displayed below.



More than one hypothesis or a low confidence value does not necessarily mean an error. Uneven areas (slopes, for example) show more than one hypothesis because of the changing terrain. Nodes with multiple hypothesis should be examined in the Subset Editor. See “Hypotheses Cleaning” on page 266.

When you select the Initialization method of disambiguation, two additional layers are created:

- Guide_Depth: Depths from the initialization surface.
- Guide_Uncertainty: Vertical uncertainty values from the initialization surface.

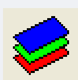
The User Nominated layer displays the nominated hypotheses that were chosen over the hypotheses selected by CUBE disambiguation (see “Nominate Alternative Hypotheses” on page 263).

Generating CUBE Surfaces

CUBE Surfaces are created using the BASE Surface wizard. All track lines must be merged and TPU must be computed before a CUBE Surface can be created.

To create a CUBE Surface:

1. Open a field sheet of the area for the Surface:
    - To create a Surface for the entire field sheet area, select the field sheet in the Layers tab in the Control window.
    - To create a Surface for a single track line, select a track line within the field sheet and then select the field sheet layer in the data tree.

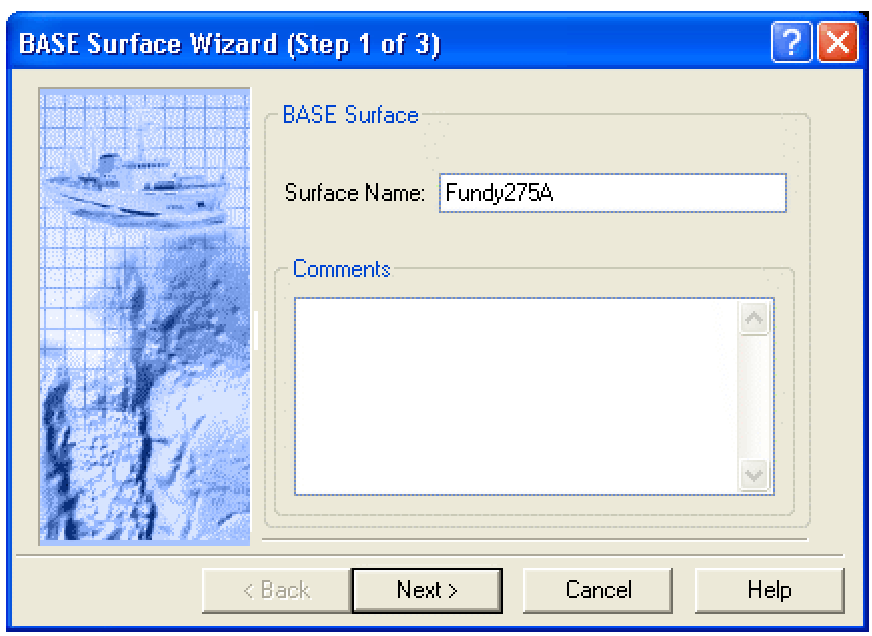
Menu	Process > BASE Surface > New
Tool	

2. Select the New Base Surface command.

The number of steps in the wizard is determined by the type of surface you are creating.

BASE Surface (Step 1)

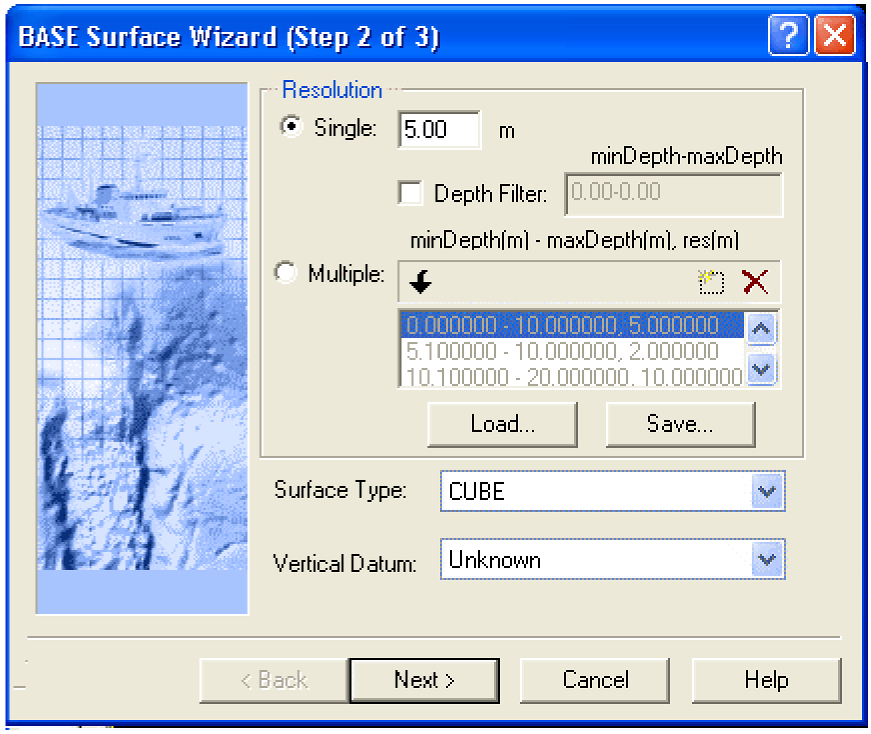
The BASE Surface Wizard (Step 1) dialog box opens.



1. Type a *Name* for the BASE Surface.
2. Type any necessary *Comments* in the text box, and click **Next**.

BASE Surface (Step 2)

The BASE Surface Wizard (Step 2) dialog box is displayed.



The *Resolution* value(s) sets the distance(s) between BASE Surface nodes. You can use the same single resolution value for the entire surface, or use different resolutions for each range of depths.

1. Select *Single* and enter a resolution value, or select the *Multiple* option and enter a range of values.

If you select the *Single* option, you can determine the range of depths to include in the BASE surface by using the *Depth Filter* option.

2. Select *Depth Filter* and enter a range of depths.

If you select the *Multiple* option, the depth ranges and related resolution values can be saved as a template to apply when creating other new BASE surfaces.

3. Enter the series of depth ranges and the desired resolution in metres, and click **Save** to save the settings as a template.

   OR

4. Click Load to use a saved file.

Values are entered in the following format:
minimum depth - maximum depth, resolution (all in metres. For example, 0.0-10.0,5.0 which associates depths between zero and 10 metres with a resolution of 5 metres

5. Select CUBE from the *Surface Type* list.
6. Select the *Vertical Datum*, if known, from the list.

7. Click **Next**.

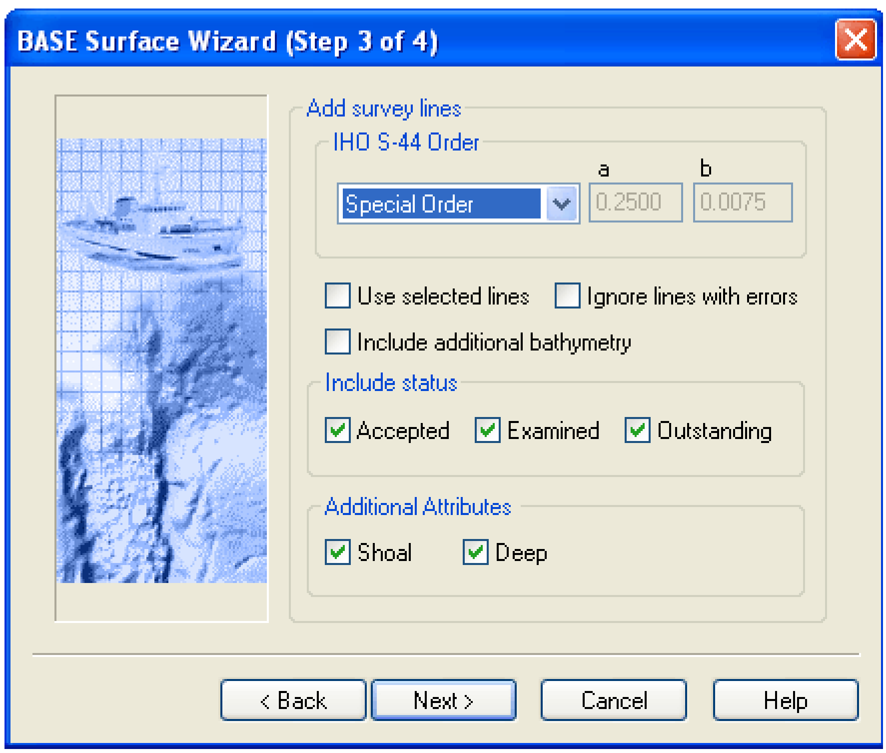
The content of the next dialog box is determined by the choice of surface type made in step 5:

"BASE Surface (Step 3) - CUBE" on page 253

BASE Surface (Step 3) - CUBE

The S-44 survey order is used to determine the area of influence that a sounding can apply to the nodes.

A sounding's area of influence (the number of nodes to which a sounding is applied) is determined by the vertical uncertainty limit in the selected S-44 survey order. When a sounding's vertical uncertainty increases beyond the requirement for the survey order then it cannot contribute to the node.



1. Select an *IHO S-44 Order* from the drop-down list.

The *a* (constant depth error) and *b* (factor of depth dependent errors) fields are automatically filled when a survey order is selected. These values are read from ..\HIPS\System\IHO_Standards.xml.

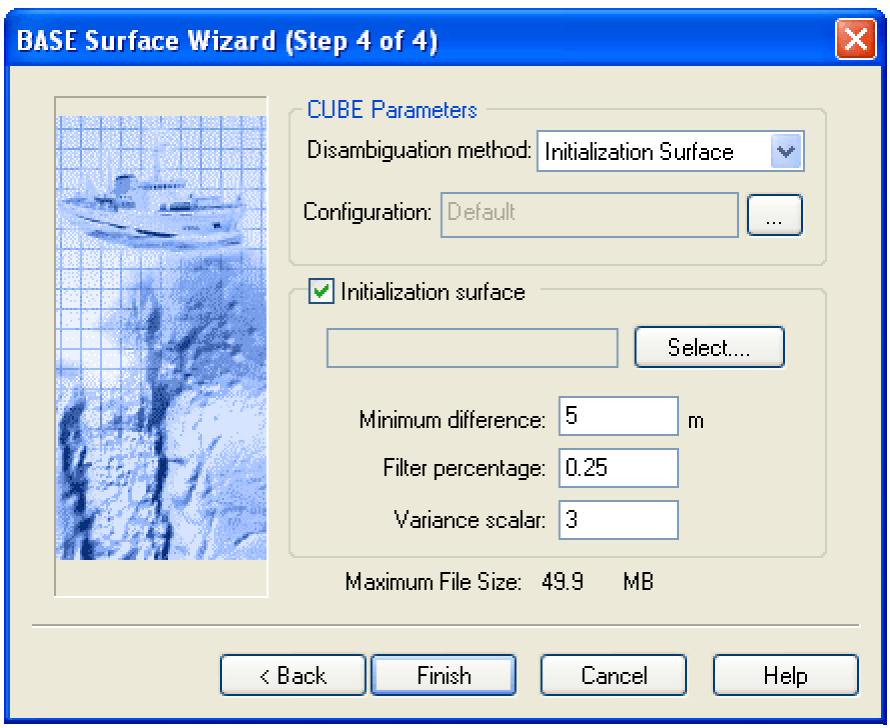
If one or more track lines were selected before the BASE Surface process was started, the *Use selected lines* check box is enabled.

2. Clear this check box if you want to apply the BASE Surface to the entire field sheet area.

Use the *Ignore Lines with errors* option so that the surface creation is not interrupted if it encounters bad data. The process will skip the part of the line containing bad data and continue creating the surface with the next line. Once the surface is complete you can decide whether to remove the partial line.

3. Check *Ignore Lines with errors* to have bad line data omitted from the surface.
4. Check *Include Additional Bathymetry* to include water column bathymetery added to project.
5. Check either *Accepted*, *Examined* or *Outstanding* to include data with these status flags in the CUBE surface.
6. Select *Shoal* or *Deep* (or both check boxes) to include shallowest or deepest soundings data in the CUBE Surface
7. Click **Next**.

BASE Surface (Step 4) - CUBE



1. Select a disambiguation method from the drop-down list:
   - Density
   - Locale
   - Density and Locale
   - Initialization Surface

   (These methods are described in "Disambiguation" on page 248.)

Default Configuration values will be applied unless you select otherwise. To apply other configuration options, click the [...] **Browse** button to open the Advanced configuration settings. (See "Advanced Options for CUBE surface" on page 255.)

If you selected *Initialization* as the method for disambiguation, the *Initialization Surface* check box is automatically selected, so you can set values for the filtering tests.

2. Type the path and name of an existing BASE Surface, or click **Select** and browse to the BASE Surface.

If you have selected a disambiguation method based on density or locale, the *Initialization Surface* tests can be used as an optional filtering method.

Initialization Surface filtering uses the depth and vertical uncertainty values in an existing BASE Surface (the Initialization Surface) to exclude potential outliers from the new surface. There are three filtering tests:

- *Minimum Difference* is a set distance from an existing BASE Surface node. For example, if the *Minimum Distance* is set at 10 metres, all soundings that are not within 10 metres of the node are excluded when a new Surface is generated.
- *Filter Percentage* is a percentage of depth at a node. For example, if the depth at a node is 20 metres and the percentage value is 0.25, all soundings that are more than 5 metres from the node (20x0.25) are excluded from the new Surface.
- *Variance Scaler* is a multiplier of vertical uncertainty at a node. For example, if the scaler is set to three and depth uncertainty for a node is one metre then soundings more than three metres (1x3) from the node are excluded from the processing of the new BASE Surface.

The shoalest values from the three tests are then used as the threshold to filter outlier soundings.

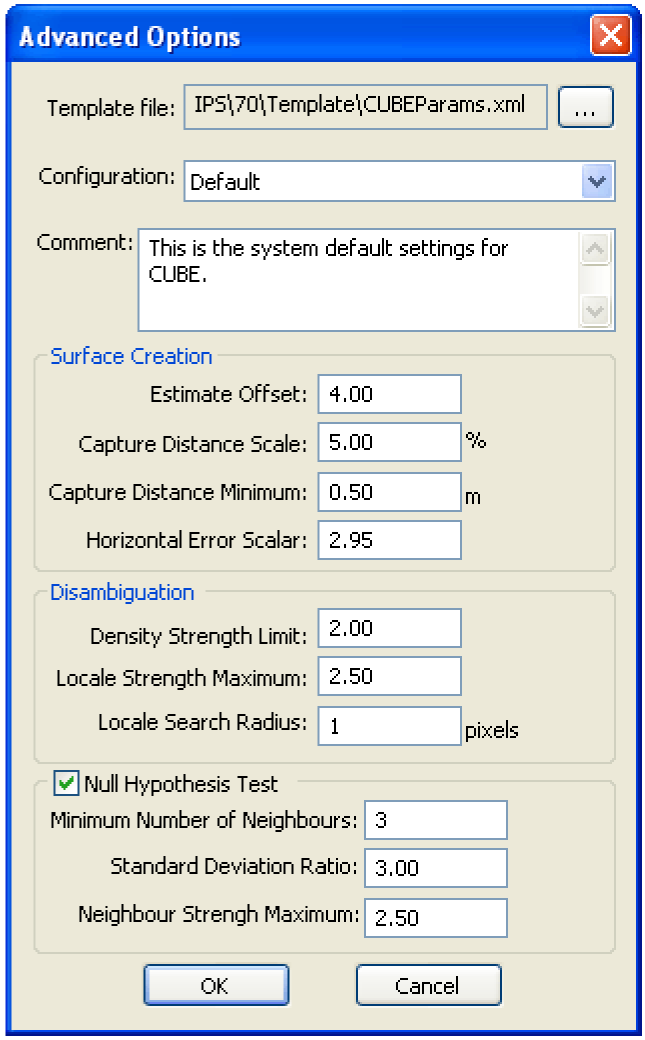
3. [Optional] Changes the default values set for the filtering tests.
4. Click **Finish**.

A CUBE Surface is shown in the Display window and the layers associated with the Surface are listed in the Control window. See "View BASE Surfaces" on page 203 for more information on display options.

Advanced Options for CUBE surface

The Advanced Options dialog box displays the detailed configuration settings applied to the CUBE surface. These settings are contained in the CUBEparams.xml file, found in the installed Template folder.

You can use the settings in this file or select a custom configuration file. If you use a custom file, make sure that you use the same syntax and structure as the CUBEparams.xml file.



1. [Optional] Click **Browse** to select another configuration file. The file path is displayed in the *Template File* field.
2. Select a configuration file from the *Configuration* drop-down list: Default, Deep or Shallow.

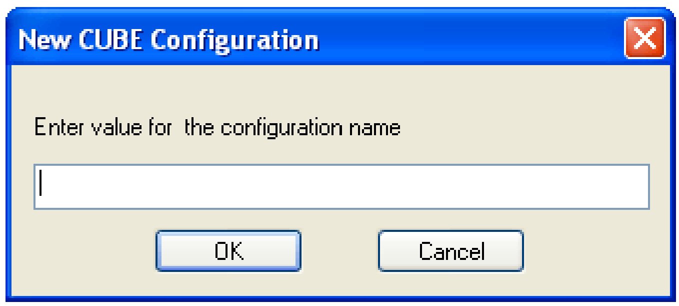
You can adjust the configuration settings can in these files, but you must save changes before they can be applied. If you do not want to overwrite the CUBEparams.xml file, select New and create a new template file.

- Deep: This configuration is intended to be used in areas where small features are not likely (shifting sand shoals), not important (steep grades and deep water), or separately located with sidescan.

This parameter set corresponds with NOAA's 2006 Complete gridding standard.

- Shallow: This file defines the parameters for the CUBE algorithm and are intended to be used in areas of critical underkeel clearance, in areas with numerous small features, and multibeam that can stand on its own without feature-by-feature sidescan correlation. This parameter set corresponds to NOAA's 2006 Object Detection gridding requirements.
- New: select “New” to save any changes to the default settings, or to create a new configuration. Otherwise the changes will be saved to the CUBEparams.xml file.

3. If you select “New”, the New CUBE Configuration dialog box is displayed.



4. [Optional] Type the name for the new configuration setting and click **OK**.
5. [Optional] Type any additional information in the *Comments* field.
6. The default values in the following fields can be adjusted, as needed. You will be prompted to save your changes before the configuration can be applied. *Estimate Offset*: The threshold for significant offset from current estimate to warrant an the creation of a new hypothesis. The value must be between 0.1 and 10.0.

- *Capture Distance Scale*: Scale on predicted or estimated depth for how far out to accept data. Value is a percentage of depth used to limit the radius of influence a sounding may have on the grid. Value must be between 1.00 and 100.00.
- *Capture Distance Minimum*: The minimum value (in metres) on predicted or estimated depth for how far out to accept data. This value is used in conjunction with Capture Distance Scale to limit the radius or influence of a sounding. Value must be between 0.0 and 100.00.
- *Horizontal Error Scalar*: The value used to scale the horizontal error of each sounding when used in the radius of influence computation. Value must be between 0.0 and 10.00.
- *Density Strength Limit*: The strength value used to switch from the 'density' disambiguation method to the 'locale' version when using the density & locale algorithm. Value must be between 0.00 and 5.00.
- *Locale Strength Maximum*: The maximum strength value allowed as part of the mean in the locale algorithm. Value must be between 0.00 and 5.00.

- *Locale Search Radius*: The radius of the search when computing the trimmed mean. The values are in pixels and must be greater than zero. The original system default value is 1.
- *Null Hypothesis Test*: Flag used to control the application of the NULL hypothesis test. Value must be either True or False. Default value is False.
- *Minimum Number of Neighbours*: During the Null Hypothesis test, this controls the minimum number of neighbours that a node must have in order to be considered for the Standard Deviating Ratio test. If the node has less the specified number, it is automatically marked as 'Null'. Values must be between 2 and 8.
- *Standard Deviation Ratio*: During the Null Hypothesis test, this represents the cut-off limit for the standard devotion ratio including the node in question to the standard deviation of the qualified neighbouring nodes. If the computed ratio exceeds the specified value, the node is marked as 'Null'. Values must be between 0.0 and 10.0.
- *Neighbour Strength Maximum*: The maximum strength value that is allowed to be considered as part of the standard deviation computations in the NULL hypothesis test. Value must be between 0.00 and 5.00.

7. Click **OK** to save and apply settings.

Hypothesis Editing

CUBE surfaces can be displayed in the Subset Editor for examination and editing, using an iterative process to choose which of multiple hypotheses of depth values are the best ones to represent the sea floor.

When a CUBE surface is created, soundings are weighted and contribute to surface grid nodes based on TPU values and distance from the nodes. The CUBE surface allows for multiple depth estimates or hypotheses to exist at a single grid node, depending on the variation of the sounding data. CUBE then uses “Disambiguation” to determine which hypothesis at each node is the most “correct”.

You can verify and, if necessary, override, a CUBE decision in Subset Editor, by nominating an alternative hypothesis as the depth. Once these kinds of changes have been made to the CUBE surface, the surface is updated.

A surface filter can then be applied to the data. Any sounding data that is not in agreement with the selected hypotheses will be flagged as rejected. By applying this surface filtering, the number of manual edits required by the hydrographer to produce a clean sounding data set is greatly reduced.

> When following a CUBE workflow, this hypothesis editing process is performed instead of other filtering such as Swath or Subset Editor filters.

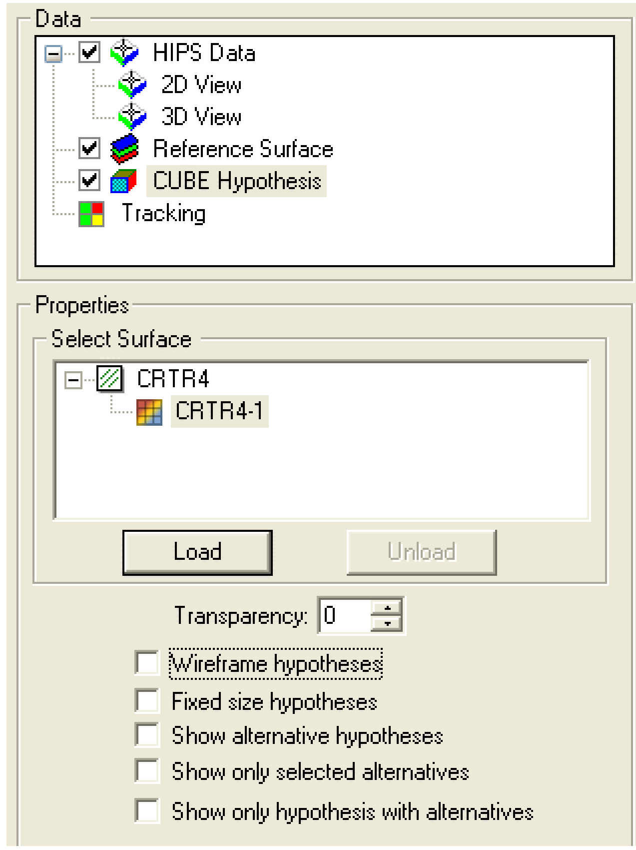
To examine an open CUBE Surface in Subset Editor:

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

1. Select the Subset Editor command.
2. Define the subset with a bounding box.
3. Load the subset.

Data contained within the bounding box is loaded into the 3D Subset view.

4. In the Subset Editor tab, select the CUBE Hypothesis layer.



5. Expand the Select Surface data tree and select a BASE Surface containing CUBE data.
6. Click **Load** to view the CUBE surface.
7. To return to the normal display in the 3D and 2D Views, click **Unload**.

The display options for the hypotheses are listed at the bottom of the Subset Editor tab.

"Wire frame Hypotheses" on page 261

"Fixed Size Hypotheses" on page 262

"Nominate Alternative Hypotheses" on page 263

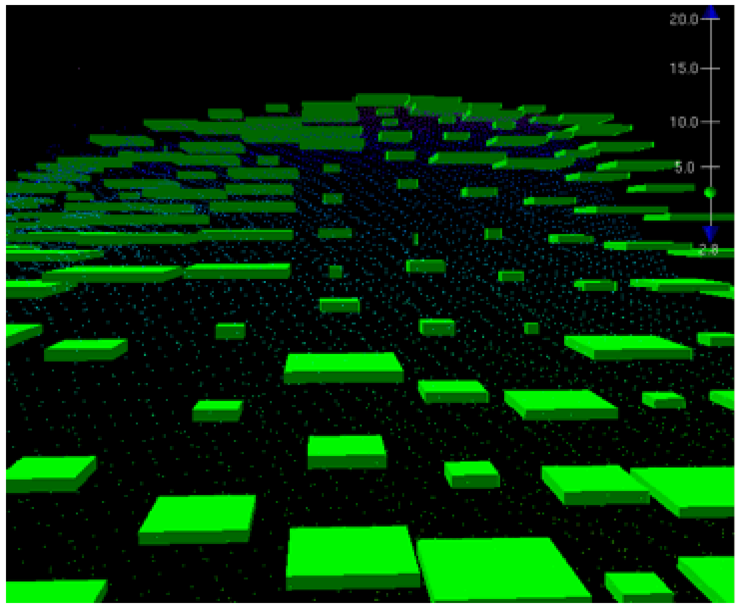
"Show only Selected Alternatives" on page 264

"Show Only Hypotheses with Alternatives" on page 264

8. View and edit the hypotheses. See "Hypotheses Cleaning" on page 266
9. Update the surface. See "Updating a CUBE Surface" on page 267

Hypotheses views

Below is an example of an enlarged section of a CUBE surface loaded in the 3D View.



Hypotheses are represented as squares. The squares have different dimensions to reflect the confidence level given to each hypothesis. These confidence levels range from 0.0 (the highest value) to 5.0 (the lowest value). The size of a square is related to the numerical confidence level: large squares represent nodes with the high confidence values while smaller squares represent nodes with low confidence values.

The thickness of the square represents the vertical uncertainty assigned a node (to a 95% confidence interval). The thicker the square, the greater the vertical uncertainty.

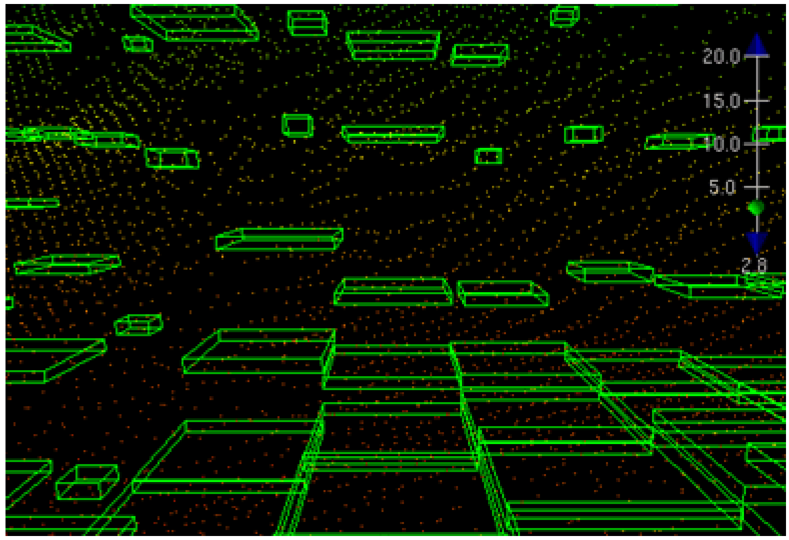
More than one hypothesis or a low confidence value does not necessarily mean an error. Uneven areas (slopes, for example) show more than one hypothesis because of the changing terrain.

Wire frame Hypotheses

Hypothesis squares can also be displayed without a fill so that they are transparent except for the outlines of the squares.

1. Select the *Wireframe Hypotheses* check box.

The 3-D View is refreshed to display the squares without fill, as illustrated below.

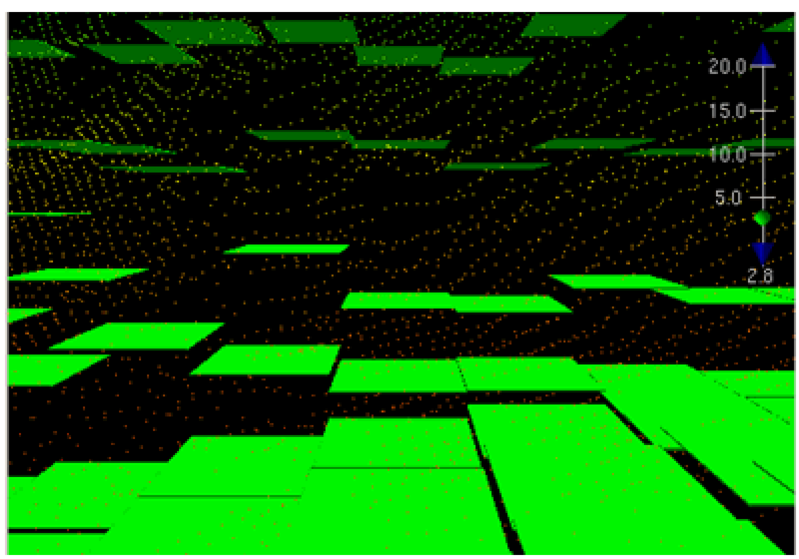


Fixed Size Hypotheses

The *Fixed Size Hypotheses* option flattens the hypotheses squares and redraws them to a standard size. This option is useful if you want to examine the dataset for holes or to see the exact position of a square in the subset.

1. Select the *Fixed Size Hypotheses* check box.

The 3-D View is refreshed to display the squares as flattened and in a standard size.



Nominate Alternative Hypotheses

Some nodes have alternative hypotheses that were not selected during disambiguation You can view alternative hypotheses and compare them against the selected ones. If needed, the alternative hypotheses can be nominated to replace the established hypotheses.

"Show alternative hypotheses" on page 263

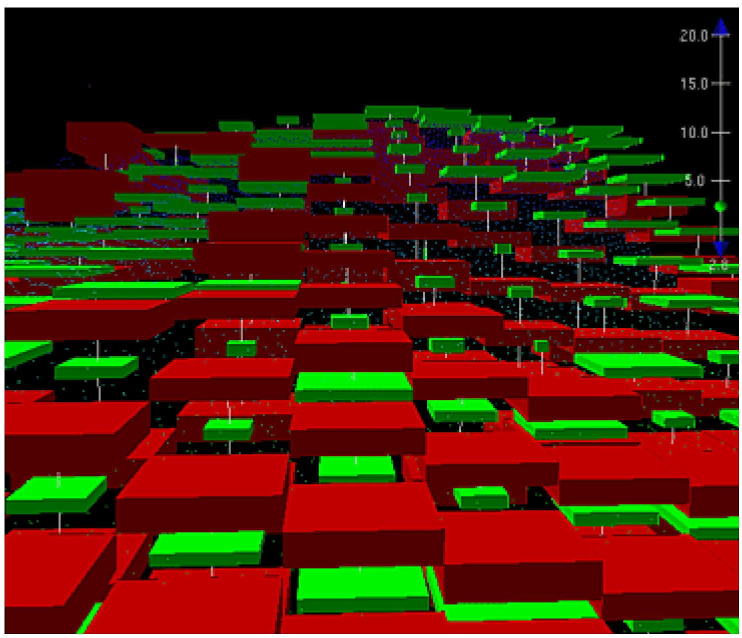
"Show only Selected Alternatives" on page 264

"Show Only Hypotheses with Alternatives" on page 264

Show alternative hypotheses

1. Select the *Show Alternative Hypotheses* check box and click **Apply**.

The 3-D and 2-D Views are refreshed to highlight the alternative hypotheses. The following image shows a 3-D View with the alternative hypotheses displayed in red.



The red squares represent hypotheses that were not selected during disambiguation. The vertical lines that run between the nodes are viewing aids to match these alternative hypotheses to the ones selected by CUBE.

1. To replace an hypothesis, select an alternative hypothesis in the 2D or 3D window.

The Nominate and Clear commands are now active.

Menu	Tools > Subset Editor > Nominate
Tool	

2. Select the Nominate command.

The alternative hypothesis is now highlighted in blue. This means that the nominated hypothesis has replaced the established one.

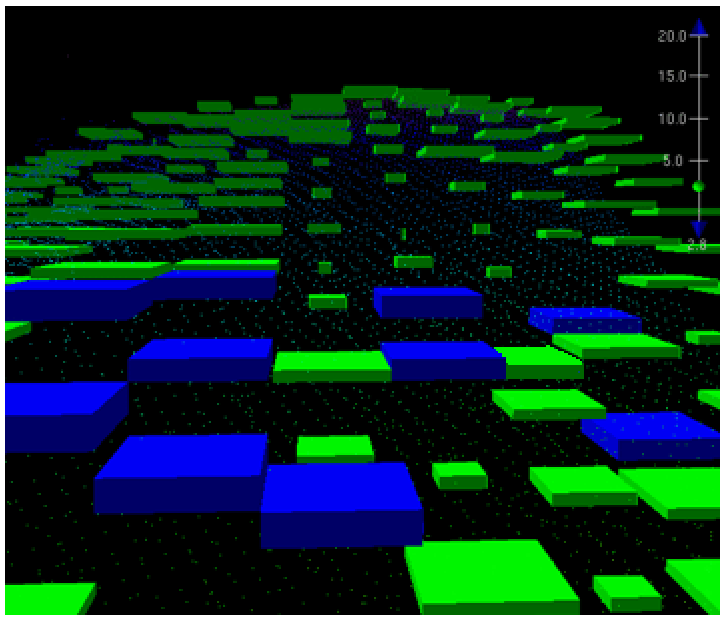
All nominated hypotheses are given the highest confidence value of 0.0.)

Show only Selected Alternatives

To compare nominated nodes against the existing nodes,

3. Clear the *Show Alternative Hypotheses* check box
4. Select the *Show Only Selected Alternatives* check box.

The 2D and 3D Views are refreshed to show only the nominated and established hypotheses. The following image shows the 3-D View with only nominated and established hypotheses displayed.

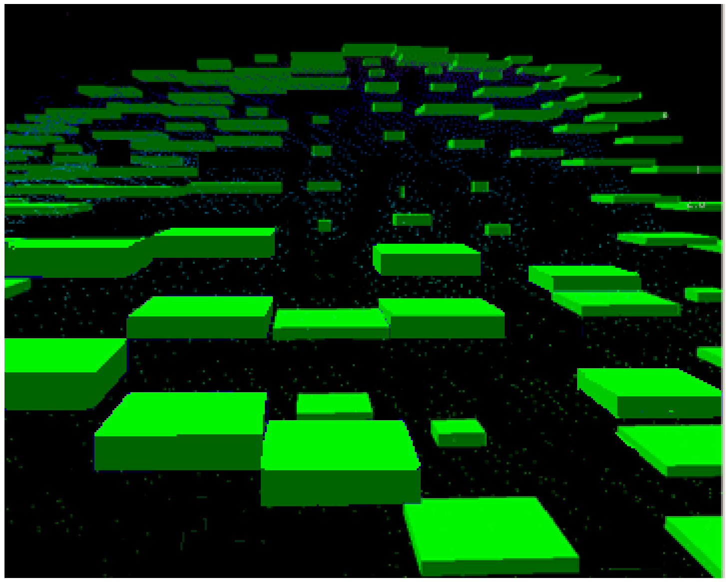


Show Only Hypotheses with Alternatives

Some hypotheses will not have alternatives. To view only those hypotheses selected by CUBE that *do* have alternative hypotheses:

5. Select the *Show Only Hypotheses with Alternatives* check box.

The Views are refreshed to show only the hypotheses for which there are alternative hypotheses, as illustrated below.



When you have finished nominating alternative hypotheses to replace the hypotheses selected by CUBE disambiguation, the User Nominated layer will display these nominated hypotheses.

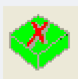
Menu	Tools > Subset Editor > Clear
Tool	

6. Select the Save command to save your data.
7. To remove the nomination flag, select the Clear command.

Hypotheses Cleaning

Like soundings in Swath and Subset Editor, hypotheses can be given a rejected flag. When a hypothesis is rejected, it is retained in the Surface but is excluded from processing when disambiguation is run again or when filtering is applied.

Reject

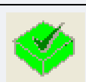
Menu	Tools > Subset Editor > Reject
Tool	

1. Select a hypothesis.
2. Select the Reject command.

The hypothesis is now flagged as rejected.

To change rejected back to the accepted status (and therefore available for further processing).

Accept

Menu	Tools > Subset Editor > Accept
Tool	

1. Select a hypothesis.
2. Select the Accept command.

The hypothesis is now flagged as accepted and is available for further processing.

Reject Node

To reject all hypotheses at a node:

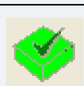
Menu	Tools > Subset Editor > Reject Node
Tool	

1. Select a hypothesis.
2. Select the Reject Node command.

All hypotheses associated with the node are rejected.

Accept Node

To return hypotheses associated with a node to their normal status.

Menu	Tools > Subset Editor > Accept
Tool	

1. Select a hypothesis.
2. Select the Accept command.

All hypotheses associated with the node are flagged as accepted.

Updating a CUBE Surface

If changes are made to the data contributing to the surface, the CUBE Surface can be rebuilt to show these changes using the following methods:

Menu	Process > BASE Surface > Recompute
Tool	

- Select the Recompute BASE Surface command . (See "RECOMPUTE SURFACE" ON PAGE 221).

> When recomputing a CUBE BASE Surface, all CUBE BASE Surface editing, including nominations, hypothesis and node rejections are considered no longer valid and *will not be retained* after recomputation of the BASE Surface.

- Select the *Automatic BASE Surface Update* check box in Tools> Options > General. After edits are made, this will automatically update the area of the CUBE surface contained within the subset bounding box. (See GENERAL for more information).

> When using the Automatic BASE Surface Update option, all CUBE BASE Surface editing, including nominations, hypothesis and node rejections *could be retained* if no soundings were rejected in the proximity of the nodes.
>
> If soundings have been rejected after CUBE BASE Surface editing, these editing adjustments will be lost.

12

Data QC

Use the BASE surface to highlight problems with sensor data that may exist, and if necessary, edit problem areas.

In this chapter...

Workflow 270
Tools for examining and editing data 272
Query Data 273
Rejecting and Accepting Data 277
Examine Navigation Data.. 278
Shift Navigation Data... 284
Attitude data... 285
Delayed Heave... 289
Load Attitude/Navigation Data 291
Classify Lines.. 293
Reject Line.. 294
View Rejected Soundings 295
Critical Soundings 297
Designate Critical Soundings from a Surface 298
Restart Cleaning 305
Quality Control Reports...................................... 307

Workflow

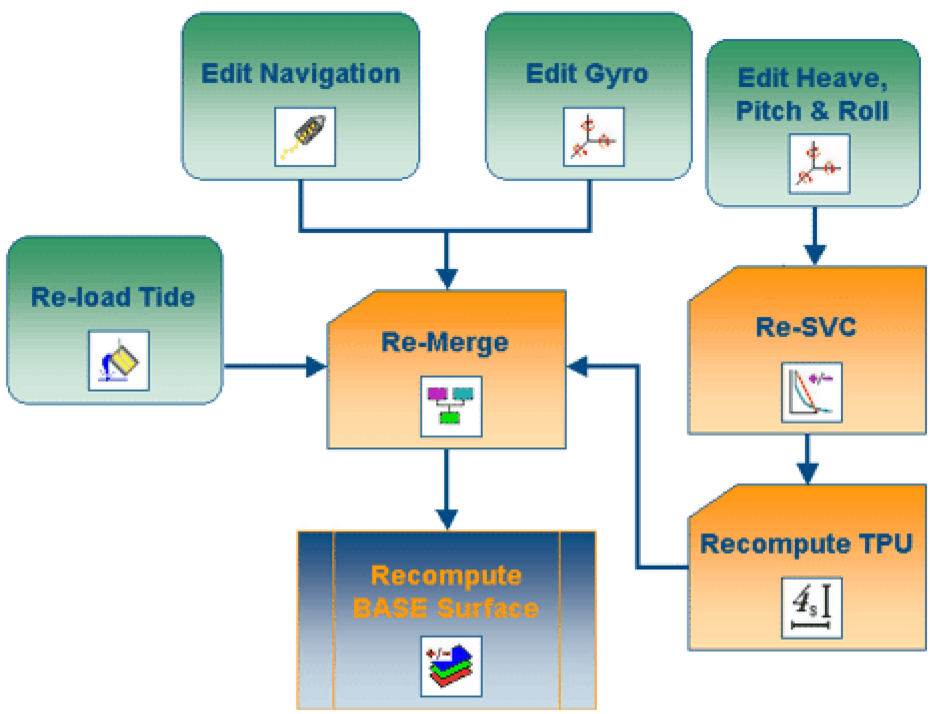
Problems with sensor offsets, auxiliary sensor data, and so forth, become readily apparent in the BASE Surface. Relying on a surface to highlight problems in the auxiliary sensors can improve processing efficiency, because you are no longer required to investigate the motion and navigation for each survey line.

In the HIPS multibeam workflow described here, the auxiliary sensor data was not investigated prior to creating the surface. So at this point, you can use the BASE or CUBE surface to highlight problem areas that may exist.

Certain edits will require that data be Merged again. For example, if you reload tide, edit navigation, or edit gyro, you will need to Merge the modified lines again.

If you edit heave, pitch or roll, you will need to re-apply Sound Velocity Correction. If you re-apply SVC, recompute TPU before Merging again.

If any project lines are re-Merged, the BASE surface will need to be re-generated. See "Recompute surface" on page 221.



After data QC, the following processes can be applied to further clean data:

- statistical surface cleaning (see "Creating Tiles and Cleaning Data" on page 314)
- "Subset Cleaning" on page 339

- "Swath Cleaning" on page 324
- filters
  - "Protect Critical Soundings" on page 304,
  - "Attitude Filter" on page 286,
  - "TPU Filtering" on page 159,
  - "Single Beam Filtering" on page 105.

Tools for examining and editing data

There are a number of tools available to assist examining soundings, and in tracking the progress of data cleaning. For example, you can:

- examine information about one or more selected track lines (“Query Data” on page 273).
- set a flag to indicate the type of survey line (“Classify Lines” on page 293).
- reject an entire track line to remove it from processing (“Reject Line” on page 294)
- show selected types of rejected soundings (“View Rejected Soundings” on page 295)
- preserve and highlight shallowest soundings (“Critical Soundings” on page 297)
- easily view Designated, Examined and Outstanding soundings when working with a large data set (“Critical Soundings” on page 297)
- reverse previous depth cleaning by re-setting status flags from “Rejected” to “Accepted” (“Restart Cleaning” on page 305)

Query Data

Line information, such as project and date of survey, and status of corrections such as SVC and Merge, can be viewed in the Selection tab and the Detailed Line Query window.

This line profile information is displayed in columns. The columns which are displayed, and the order in which they are displayed, is controlled by a pop-up menu. (See "Column Settings" on page 20 of the Reference guide for details on adjusting the columns displayed.)

Similar information for data in HIPS editors can be viewed when selected and queried using the Query or Query Line command.

Queried data can be saved to a text file from the display, or a line report can be generated using the Process > Line Report command.

"View Line Data" on page 273

"Query Line Data" on page 275

"Query Data in Editors" on page 274

"Line Report" on page 275

View Line Data

To view data profiles for one or more track lines selected in the Display window:

1. Select a track line in the Display window or in the Project tab of the Control window.
2. Open the Selection tab in the Worksheet window.

The Selection tab automatically displays the data profile for each selected line. As well as Project, Vessel, Day and Line identification, the profile data displayed can include this information:

- Min Time
- Max Time
- Total Time
- Merged
- Outdated
- Speed
- Line Reject
- Line Class
- Heading
- Length
- SR Corrected
- GPS Tide
- Tide Loaded
- Raw Range
- SVP Corrected
- TPU Computed
- Nav Examined
- Del Dft Loaded
- Tide Applied
- Locked

The information is read-only and cannot be modified.

3. [Optional] Select Save As from the right-click menu to save the line data to a text file.

(See "Column Settings" on page 20 of the Reference guide for details on selecting and adjusting the columns displayed.)

Detailed Line Query

In addition to the line data displayed in the Selection tab, a Detailed Line Query can display an additional 30 columns of line data in the Detailed Line Query window.

To view this detailed line information:

1. Select a line or lines in the Display window.

Menu	Process > Detailed Line Query

2. Select the Detailed Line Query command.

The Detailed Line Query window opens. If you are processing a large number of lines, a progress bar will display in the left side of the Status bar.

You can cancel loading of the selected lines into the Line Query window:

3. Select **Cancel** on the Processing dialog box.

To query another line or lines:

4. Select the lines.
5. Select the Detailed Line Query command again.

The display in the Query window will be refreshed to show your new selection.

As with the Selection tab, the Detailed Line Query window will display the line information in a table. These read-only records can be sorted by column by clicking on a column header.

Which data is visible and in what order it is displayed is controlled using the Column Settings dialog box, opened from the right-click menu. (See "Column Settings".) Changes made to the column selection and order are retained until you change them.

This detailed information can be saved to a text file.

6. [Optional] Select Save As from the right-click menu to save the line data to a text file.

You can also save a customized selection of the data using the "Line Report" on page 275.

Query Data in Editors

Use the Query function to examine data selected in any HIPS and SIPS interactive editor. For example, position data selected in Navigation Editor, or pings selected in Side Scan Editor.

1. Select the data so it is highlighted in one of the editor windows.

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

2. Select a Query command.

The selected records are displayed in the Selection tab. For example, in Navigation Editor, this data is displayed for each numbered record:

- Time
- d-Time
- Lat (DMS)
- Lon (DMS)
- Distance (m)
- d-Distance (m)
- Speed (m/s)
- d-Speed (m/s)
- CMG
- d-CMG
- Status

This data is read-only. However, individual records can be rejected or accepted using the pop-up menu in the Selection tab.

(See “Column Settings” on page 20 of the Reference guide for details on selecting and adjusting the columns displayed.

Query Line Data

To view records for entire lines in a HIPS and SIPS Editor, use the Query Line command.

1. Select data in any editor window.

Menu	Edit > Query Line
Pop-up	Query Line

2. Select a Query Line Command.

The records for the entire line are displayed in the Selection tab, in the same tabular format as the Query function provides. This data can be rejected or accepted from the pop-up menu in the Selection tab.

Line Report

Use the Line Report command to save line profiles directly to a text file.

The saved report will contain a table of values under the same column headings that can be viewed in the Detailed Line Query window.

As well as the detailed line data, the report also shows the totals of the values for these specific columns:

- Total Time
- Length
- Total Nav
- Accept Nav
- Reject Nav
- Total Depth
- Accept Depth
- Reject Depth

The text file can be opened in a text editor, but column data can be best viewed in a spreadsheet.

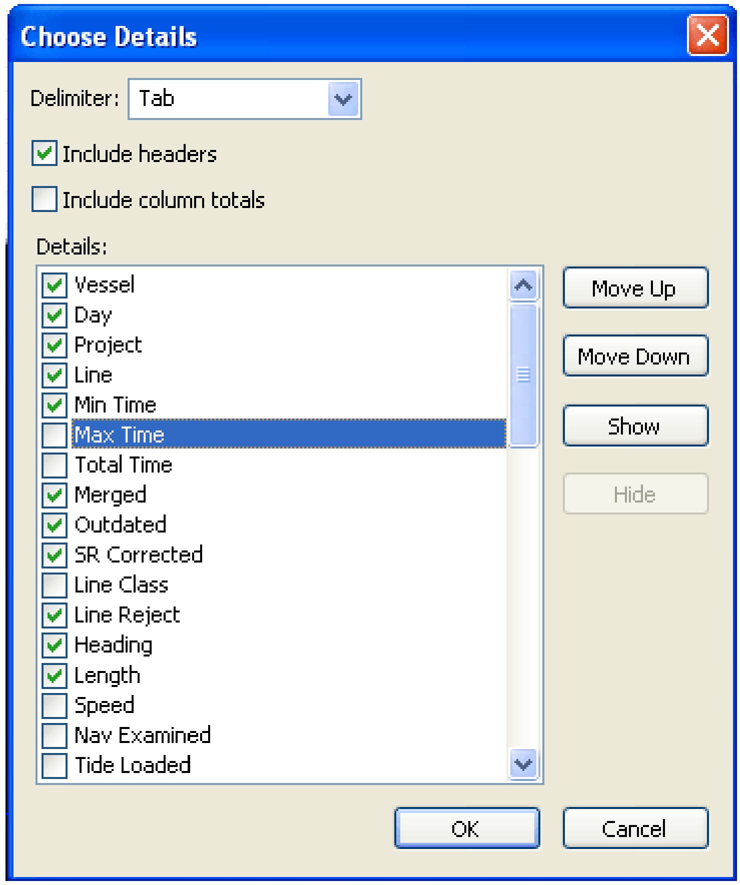
To create a line report:

1. Select a line in the Display window.

Menu	Process > Line Report

2. Select the Line Report command.

The Choose Details dialog box is displayed.



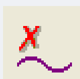
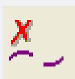
3. Select a *Delimiter* to use to indicate the separate columns. The default is Tab.
4. Select *Include column totals* to have this information included in the text file.
5. From the *Details* list select the column headings to include in the text file.
6. Click **Move Up** and **Move Down** to change the order of the columns.
7. Click **OK**.
8. In the Save As dialog box, set a name and destination folder for the text file.
9. Click **Save**.

Your selection of Details and the order of the columns, are retained for the next time you use Line Report.

Rejecting and Accepting Data

Data can be rejected or accepted in HIPS and SIPS Editors. All data is marked as “Accepted” until flagged otherwise.

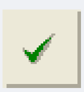
Reject Data

Menu	Edit > Status Flag > Reject-With Interpolation/Reject-Break Interpolation
Tool	 
Pop-up	Reject > Reject-With Interpolation/Reject-Break Interpolation

1. Select and highlight the data in the View window of the editor.
2. Select a Reject command (Reject with Interpolation or Reject_ Break Interpolation).

The selected data is now flagged as rejected.

Accept Data

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

Use this command to revert the status of rejected data back to “Accepted”.

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

The previously rejected data is now flagged as Accepted.

Examine Navigation Data

Navigation outliers can affect the final positions of soundings during the Merge process. Therefore, the navigation data should (but is not required to be) be examined and cleaned for outliers.

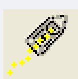
In Navigation Editor you can examine track lines to accept or reject, and query data, as necessary.

There are two methods for locating navigation outliers:

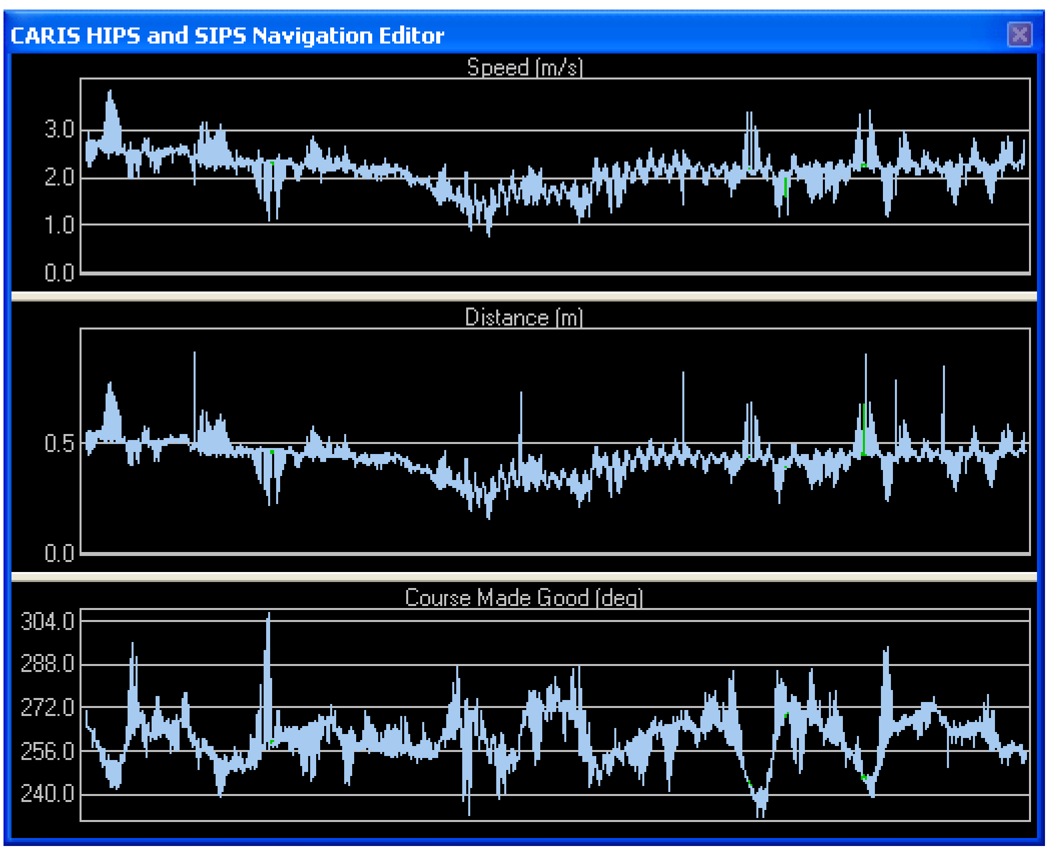
- Examine the track lines and time graphs that show the computed speed, distance, and course made good between navigation points.
- Find speed and time jumps using the search functions in the Navigation Editor tab in the Control window. (See "Navigation Editor tab" on page 46.)

To edit lines in Navigation Editor:

1. Select a track line in the Display window.
2. Select the Navigation Editor command.

Menu	Tools > Navigation Editor
Tool	

Navigation Editor time graphs open in the HIPS and SIPS interface and display relevant speed, distance, and course-made-good data for the selected line.



3. Select data in either the Display window or the time graphs by pressing and holding the mouse button while dragging the cursor across the data.

The selected data is highlighted.

4. Release the mouse button when finished.
5. You can now select a Reject option (see "Reject Data" on page 277) or Accept (see "Accept Data" on page 277) or Query the selected data (see"Query Data in Editors" on page 274).

You can also use the Spike Detection options in the Navigation editor tab:

"Speed Jump" on page 279

"Time Jump" on page 279

Speed Jump

The Speed Jump option searches the track line for changes in vessel speed that are same as, or greater than, the Speed Jump value.

1. Select the *Automatic Query* check box if you want to display the selected position fixes in the Selection tab of the Worksheet window.
2. Enter a value in the *Speed Jump* check box.
3. Click the *List Buffer* up or down arrow keys to select the number of adjacent position fixes that are highlighted when a jump is found. For example if you choose 5, the two position fixes to the right and left of the selected fix are also selected.
4. Click **Find**.
5. Select Accept ("Accept Data" on page 277), one of the Reject options "Reject Data" on page 277) or Query (see"Query Data in Editors" on page 274).
6. Click **Find** again to move along the track line.

Time Jump

The Time Jump option searches the track line for changes in time (in seconds) that are the same as, or greater than, the Time Jump value.

1. Ensure that the Control window is open and the Navigation Editor tab is displayed.
2. Select the *Automatic Query* check box if you want to display selected position fixes in the Selection tab of the Worksheet window.
3. Enter a value in the *Time Jump* box.

4. Click the *List Buffer* up or down arrow keys to select the number of adjacent position fixes that are highlighted when a jump is found. For example if you choose 5, the two position fixes to the right and left of the selected fix are also selected.
5. Click **Find**.
6. Select one of the Reject options ("Reject Data" on page 277) or Accept ("Accept Data" on page 277) or Query (see"Query Data in Editors" on page 274).
7. Click **Find** again to move along the track line.

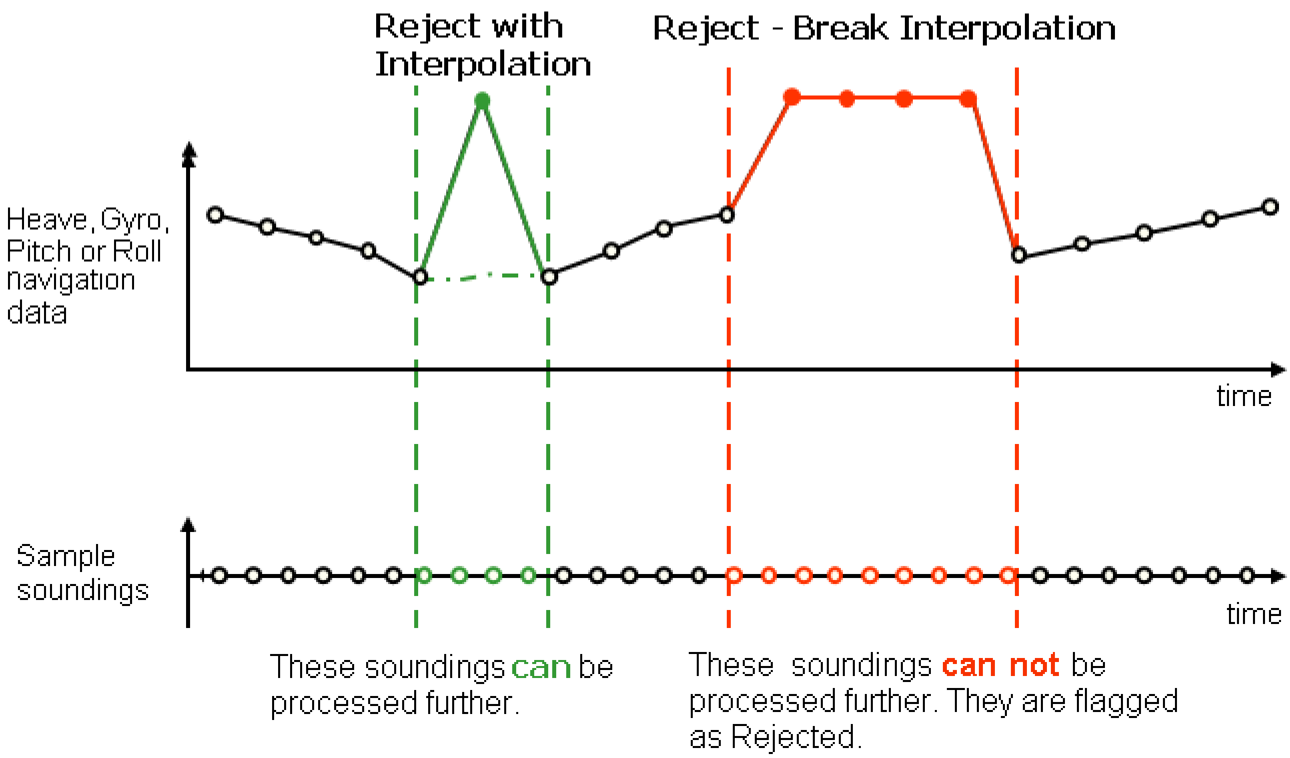
Rejecting Navigation Data

All data is marked as Accepted until flagged otherwise. Data can be queried for position fixes, and rejected.

Data can be rejected using either

- **Reject - With Interpolation:** Soundings associated with this navigation data are processed further since they have positions interpolated for them.
- **Reject - Break Interpolation:** Soundings associated with this navigation data are flagged as rejected and cannot be processed.

These alternatives are illustrated below.



Line Interpolation

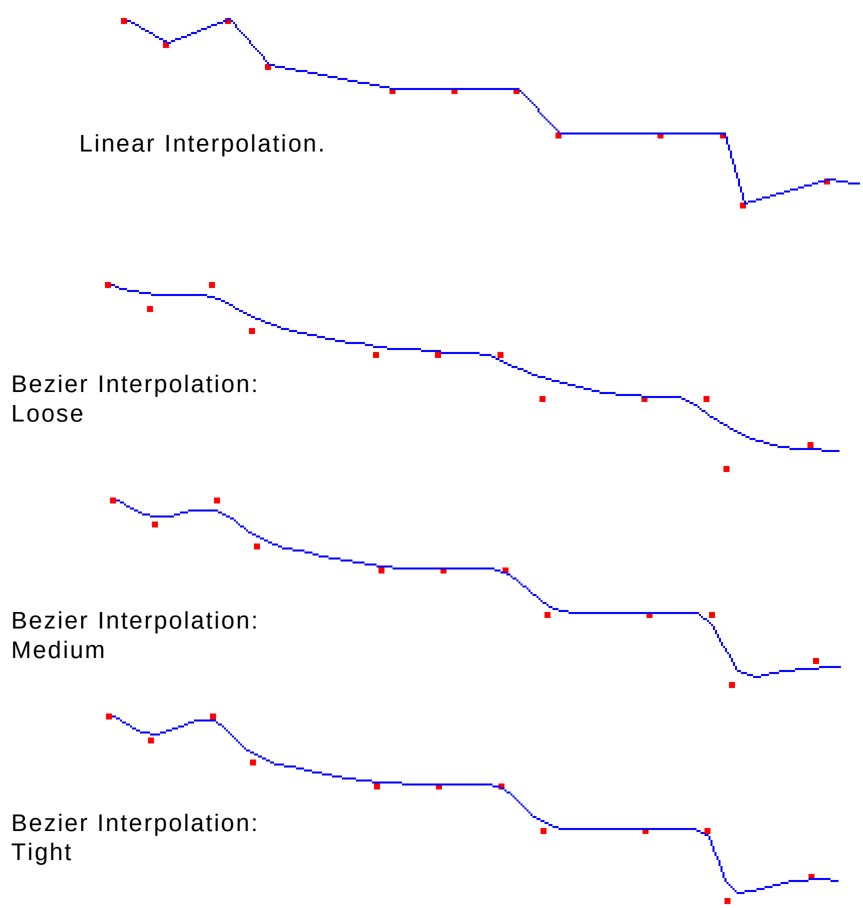
Position observations do not usually occur at exactly the same instant as a depth observation (ping). Thus it is unlikely that there is position data for every ping.

For example, positions observations may be taken every second, (1 Hz frequency), while pings may be observed 10 times a second (10 Hz frequency). So, in most cases it will be necessary to interpolate positions to match the time for each ping.

You use either linear or Bezier method of interpolation.

- **Linear**: Calculate new positions by connecting consecutive positions using straight lines. This is the default method.
- **Bezier**: Calculate new positions by connecting points using a Bezier curve that does not necessarily connect all navigation positions. Bezier curves are available in three types: tight, medium and loose.

Linear interpolation is suitable if the original navigation positions are clean and do not significantly deviate from neighbouring positions. Bezier interpolation is suitable if the original data is noisy.



In the Navigation Editor the interpolation method is set in the Interpolation method section of the Navigation Editor tab.

1. Select the *Navigation Editor* tab in the Control window.
2. Select either *Linear* or *Bezier Curve* by selecting the appropriate check box.
3. If the *Bezier Curve* option is enabled, select either a *Loose*, *Medium* or *Tight* format from the drop-down menu.

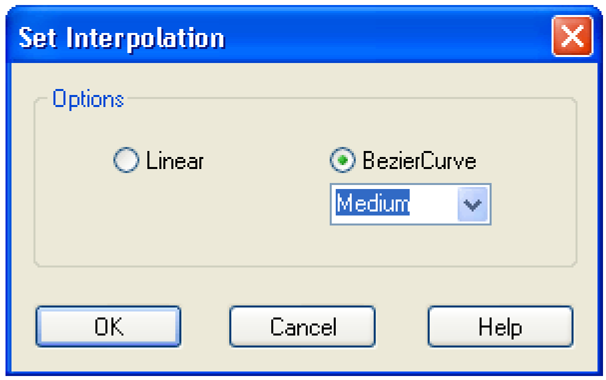
To set an interpolation option for one or more survey lines without opening the Navigation Editor:

1. Select a line.

Menu	Tools > Set Interpolation

2. Select the Set Interpolation command from the Tools menu.

The Set Interpolation dialog box is displayed.



3. Select an interpolation option: *Linear* or *Bezier Curve*
4. Click **OK**.

Shift Navigation Data

Menu	Tools > Shift Navigation

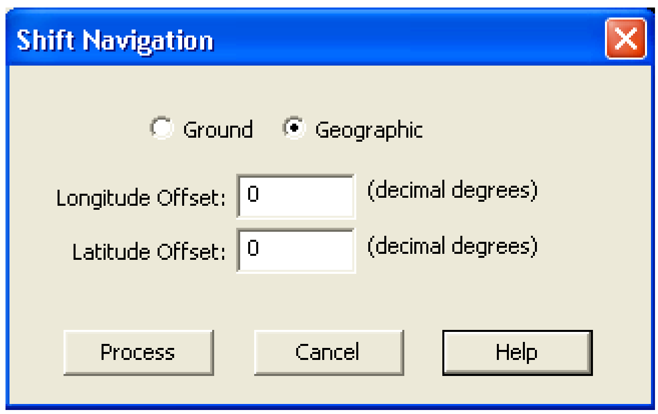
Use this to apply a shift in latitude and longitude to an entire survey. Latitude and longitude can be shifted independently to a maximum of 1 minute (0.016667 in decimal degrees and 1800 metres).

After data is shifted this way, Merge must be applied.

To shift survey data to a new position:

1. Select one or more lines.
2. Select the Shift Navigation command.

The Shift Navigation dialog box is displayed.



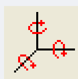
3. Select either Ground or Geographic units.
4. Type the amount of shift in one or both of the *Offset* fields. (Offset value can be positive or negative.)
5. Click **Process**.
6. [Optional] Click **Cancel** to cancel the process, and click **Yes** to confirm the cancel.

Attitude data

You can examine and clean the motion data of the vessel or towed transducer using the Attitude Editor, within the HIPS and SIPS interface. (See also "Attitude Editor Interface" on page 10.)

To open the Attitude Editor:

1. Select a track line so it is highlighted.
2. Select the Attitude Editor command.

Menu	Tools > Attitude Editor
Tool	

The Attitude Editor is displayed, showing graphs for Gyro, Pitch and Roll (in degrees) and Heave (in metres) for the selected line.

- Gyro data is displayed as a positive value when a clockwise rotation is experienced.
- Pitch data is displayed as positive when the vessels bow is down.
- Roll data in is displayed as positive when the vessels starboard side is up.
- Heave data is displayed as a positive value when the vessel is heaved upwards.

Other sensors can be added by selecting them from the Available Sensors list on the Sensor Layout dialog box.

1. Select the Sensor Layout command.

Menu	Tools > Sensor Layout

> Use the space bar to move forward along a track line and <Ctrl> + <spacebar> to move back.

Query data

Use the Query command to display information about selected data.

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

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

The following data is displayed in the Selection tab:

- time stamp
- d-time difference in seconds from last time stamp
- sensor value in either degrees or metres/feet
- d-value difference from last value
- status

You can change the data status flag to rejected or accepted.

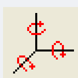
1. Select the data in the Selection tab so it is highlighted.
2. Select a Reject or Accept command.

The data is marked as accepted or rejected, depending to the option you selected.

Filter Attitude Data

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

1. Select a track line so it is highlighted.
2. Select the Attitude Editor command.

Menu	Tools > Attitude Editor
Tool	

The Attitude Editor is displayed. (See "Attitude Editor Interface" on page 10.)

Attitude Filter

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

The Attitude filter can be used to:

- Smooth data: The Smooth function is used to even out localized variability. The parameters for creating the smoothed data are saved to the SmoothedCoefficients file in the HCDS_Data\Project\Vessel\Day\Line folder.

  This file can be applied to the track line during any process that supports smoothing.
- Filter data: The Filter function is used to reject attitude that falls outside of defined boundaries. Soundings with the same time stamp as the rejected data are also flagged as rejected. You can choose to use interpolation when rejecting data.

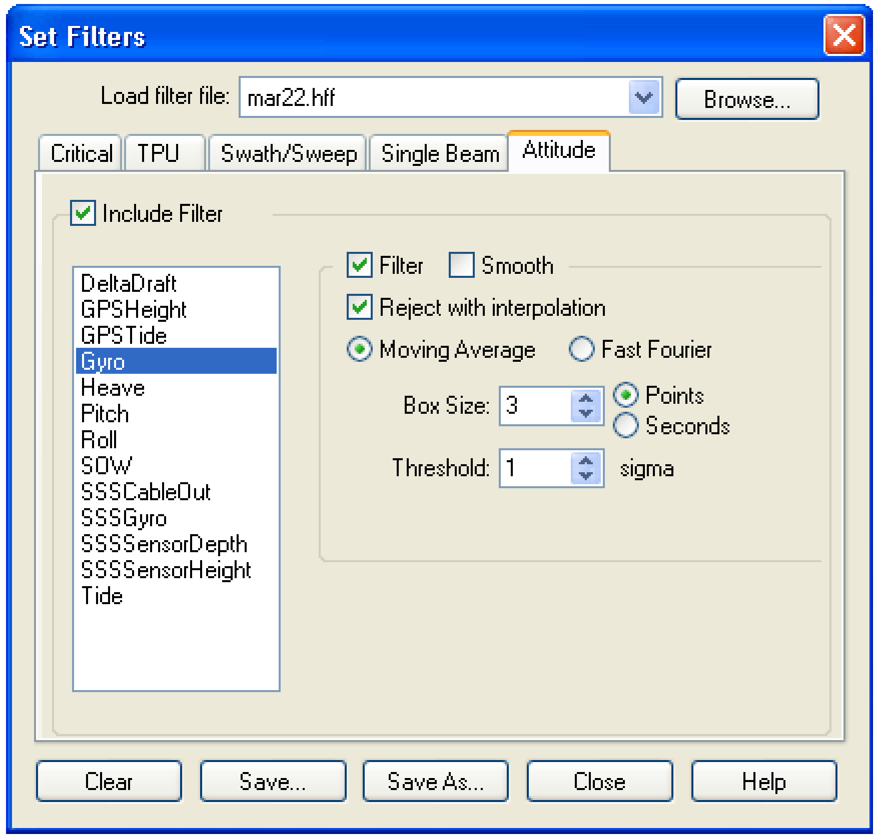
(These filter and smooth functions are also available in the Attitude Editor.)

To set filtering options for attitude:

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

Menu	Tools > Set Filters
Tool	

The Set Filters dialog box is displayed.



3. Select a sensor from the list.
4. Select the *Filter* check box.

By default, the filter is set to “Reject with interpolation” so that soundings with the same time stamp as the rejected data are not also flagged as rejected.

5. Clear the Reject with Interpolation check box if you want soundings associated with the rejected sensor data to also be rejected.

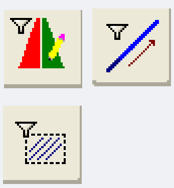
There are two filtering options: Moving Average and Fast Fourier.

- The *Moving Average* calculates a mean for each data point, by calculating an average sensor value over a window of data using the Box Size parameter. The window is defined as a number of data points or seconds centred on a point. The average value for all of the values in the window is calculated and will be given to the central point.
- *Fast Fourier* applies a common wave-smoothing algorithm to a data point. The Fast Fourier transformation performs a low-pass filter on the selected sensor. You select a Box Size in either points or seconds.This value is converted into seconds and then inverted to become the cut-off frequency.

6. Select either the *Moving Average* or *Fast Fourier* filtering option.

7. Select the Box Size parameter by clicking the up or down arrow buttons. When using the Moving Average, the Box Size determines the size of the window for averaging the data point. When using the Fast Fourier, the Box Size determines the cut-off frequency that is applied to all data.
8. Determine if the Box Size is in (data) points or time, by selecting the Points or Seconds option.
9. Click the Box Size up or down arrow buttons to select the number of adjacent data points.
10. Click the *Threshold* up or down arrow keys until you reach a desired value. Threshold is a multiple of the standard deviation (sigma).
11. Click Save As to save the filter settings to a HIPS filter file.

To use the filter:

Menu	Tools > Apply Filters > 1 Screen/To End of Line/Selected Lines
Tool	

12. Select an Apply Filter command from the Tools menu.

The corresponding attitude data is rejected.

For other automated filters, see "Automatic Filtering" on page 325, "TPU Filtering" on page 159, "Single Beam Filtering" on page 105, and "Protect Critical Soundings" on page 304.)

See also "Filter and Smooth" on page 16.

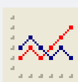
Delayed Heave

Real-time heave values are stored in HIPS during the conversion process. Some systems also provide post-processed heave data, which must be separately loaded to the track line using the Load Delayed Heave function. If the data is not stored in a supported format, the Generic Data Parser may be used to import post-processed heave data stored in an ASCII format.

HIPS currently supports these delayed heave formats: TrueHeave® and PFreeHeave®.

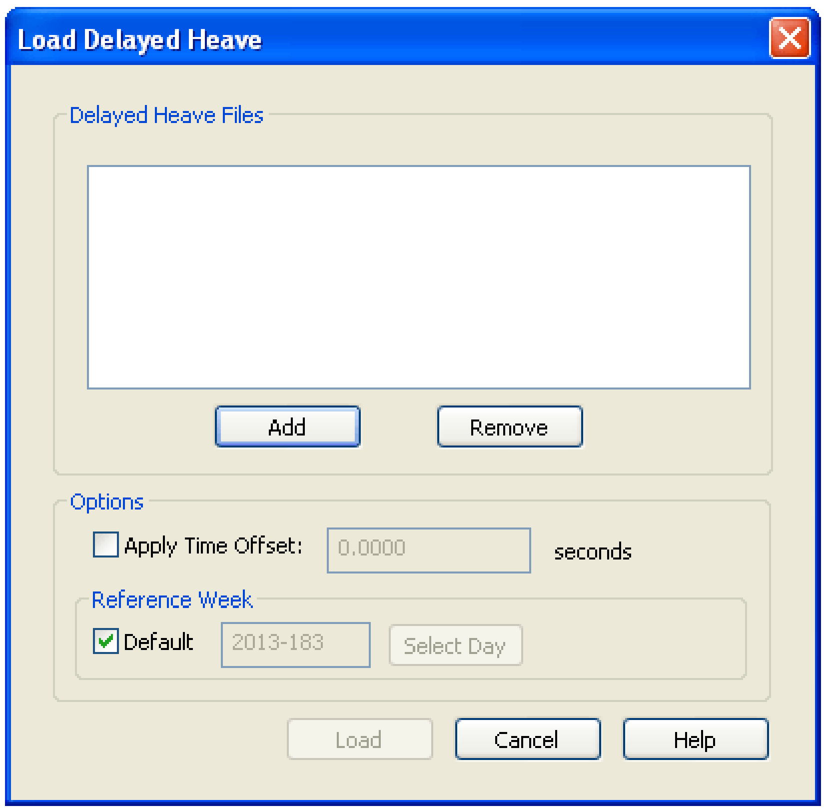
To load delayed heave data:

1. Select a track line.

Menu	Process > Load Delayed Heave
Tool	

2. Select the Load Delayed Heave command.

The Load Delayed Heave dialog box is displayed.



3. Click **Add** and select the Delayed Heave file. (The location of delayed Heave files can be set from the Directories tab of the Tools > Options dialog box.)

The file name is displayed in the dialog box.

4. To remove a file, select it and click **Remove**.

Options

5. Select the *Apply Time Offset* if the time stamps in the delayed heave files do not match the time stamps for the rest of the project.

6. Type the offset value (in seconds) that will synchronize the time stamps in the delayed heave files with the project time stamps.
7. Clear the *Default* check box and click **Select Day** to override the year/ Julian date with a calendar date. If *Default* is checked, the function will proceed as if the delayed heave file originates in the same GPS reference week as each of the HIPS data lines.
8. To apply the delayed heave data to the track line, click **Load**.

NOTE: if True Heave RMS data has been loaded for a line, it will override Down/Heave RMS data loaded by the Load Error Data process when Compute TPU is run.

See also "Load Error Data" on page 163.

See also "Compute TPU" on page 156.

Load Attitude/Navigation Data

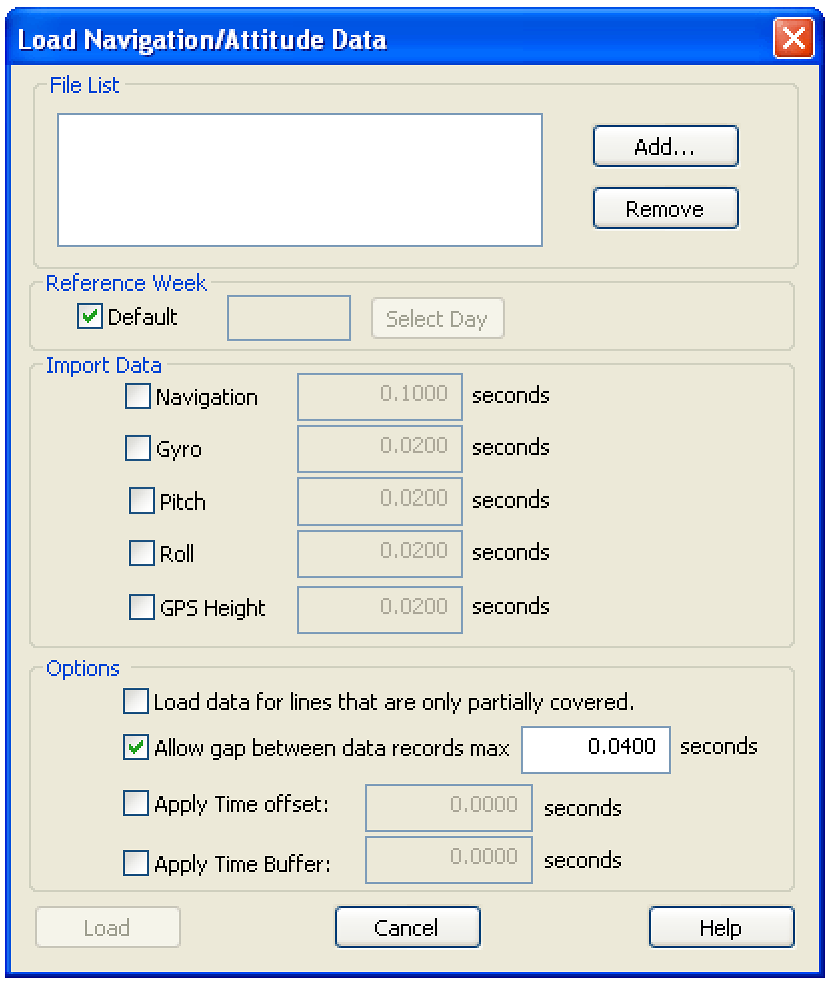
Load attitude and navigation data from Applanix POS MV or POSProc files to selected track lines. (Applanix POS MV files and Applanix POSProc files contain real-time attitude, navigation and RMS error values recorded during survey.)

1. Select one or more track lines.

Menu: Process > Load Attitude/Navigation Data

2. Select the Load Attitude/Navigation command.

The Load Navigation/Attitude Data dialog box is displayed.



3. Click **Add** to select the files by browsing to their location.
   - You can load either POS MV files OR load POSProc files, but not both at the same time.
4. To remove a file from the *File List*, select the file and click **Remove.**

Reference Week

You can specify a specific calendar date to override the year and Julian date that forms part of the SBET file name. This can also be used to accommodate a GPS week roll-over event. (The rollover cannot span more than 7 days.)

5. Clear the *Default* check box and click **Select Day** to override the year/ Julian date with a calendar date.

Import Data

6. Click the appropriate check boxes to choose which records you want to import into the track line(s) from the attitude/navigation files.
7. To reduce the volume of large attitude records to a manageable level, enter an output interval (in seconds).

Options

8. The *Load data for lines that are partially covered* option will load the navigation data even if the data does not cover the entire line.
9. Select the *Allow gap between data records* check box to limit the interval allowed between consecutive data records in the file. If an interval exceeds the amount specified in the *Maximum* field, the data will not be loaded and the process will stop.
10. If the time stamp in the file does not match the time stamps in the project, you can select the *Apply Time Offset* option and type a time value (in seconds).
11. To set a time buffer to extend the loading of navigation /attitude beyond the time extent of the line's observed depths data, select the *Apply Time Buffer* check box and type the value of the buffer in seconds.
12. Click **Load**.

Use the Detailed Line Query function to confirm that the files have been loaded:

1. Open the Detailed Line Query window.

Menu	Window > Detailed Line Query

2. Select a line or lines in the Display window.

The records for the selected line(s) are displayed in a table in the Detailed Line Query window.

3. Scroll through the table to the *SBET Nav/Att File* column.

If the navigation data has been modified by the Load Nav/ Attitude command, the name of the loaded SBET is listed in the *SBET Nav/Att File* column. If navigation data has not been altered, the field will display *** Not Loaded ***.

(The SBET RMS File column will list the SBET file used when user performs the Process->Load Error Data function. See "SBET FILES LOADED" ON PAGE 165.)

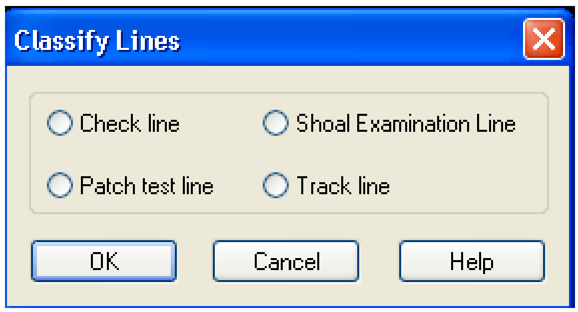
Classify Lines

Use this command to set or change a classification flag for a track line, to help distinguish between line types.

1. Make sure the Ship Track Lines layer is selected in the Layers tab of the Control window.
2. Select a track line in the Display window or Control window.
3. Select the Classify Lines command.

Menu	Edit > Classify Lines

The Classify Lines dialog box is displayed.



4. Select from four options for setting the line classification flag:
   - *Check Line*: A line that is run perpendicular to the survey and is used to check the validity of survey data.
   - *Patch Test Line*: A line that is run over a surveyed area to calibrate sensor and sonar offsets.
   - *Shoal Examination Line*: A line that is run over a surveyed area to re-check possible shoals on the sea floor.
   - *Track Line*: A line that is part of the main survey grid.
5. Click **OK**.

The line is now flagged according to one of four categories.

8.

Reject Line

Use the Reject Line command to remove an entire track line from data processing.

When a rejected line is merged, the profiles are marked as rejected, which prevents the soundings from being used in BASE surfacing, surface cleaning, and mosaicking. However, status flags on the individual soundings are maintained. A rejected line is not processed in the tiling operations or exported to a CARIS map or ASCII file.

1. Select a track line.

Menu	Edit > Reject Line

2. Select the Reject Line command.

The rejected line is displayed in the Project tab in the Control window with a red **X** beside the file name.

To change status of a rejected line back to accepted:

Menu	Edit > Accept Line

3. Select the Accept Line command.

The line is displayed in the Project tab of the Control window with a red exclamation point (!) beside the file name. This icon indicates the line has been flagged as outdated. You must merge the soundings again before they are available for further processing (see "Merge" on page 167).

View Rejected Soundings

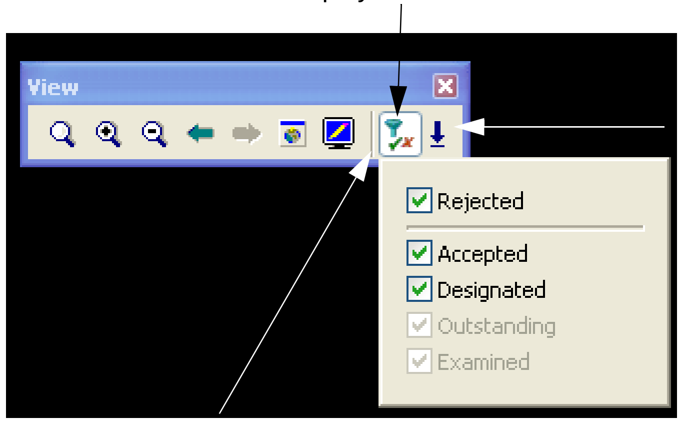
You can show or hide rejected data in Attitude, Navigation, Single Beam, Swath, and Subset Editor. Soundings that were rejected in the Subset Editor can also be viewed in Swath Editor.

Use the two-part Display Filter button on the View tool bar to display rejected data in an Editor. (This control will also display data in Accepted, Designated and other critical sounding states.)

Use the right half of the button to set which data to display. When the left half of the button is depressed, the selected data type is displayed. When the button is not active, all data is displayed.

.

Display Filter button



Click to choose a filter

Click to toggle visibility

For example, to toggle between displaying all the data open in a HIPS editor, and displaying only rejected data:

1. Open data in an Editor.
2. Click the arrow part of the Display Filter button and select the *Rejected* check box from the drop-down list.
3. Depress the Display Filter button to make the Rejected data visible.
4. Click the button again to see all data.

Reason for Rejection

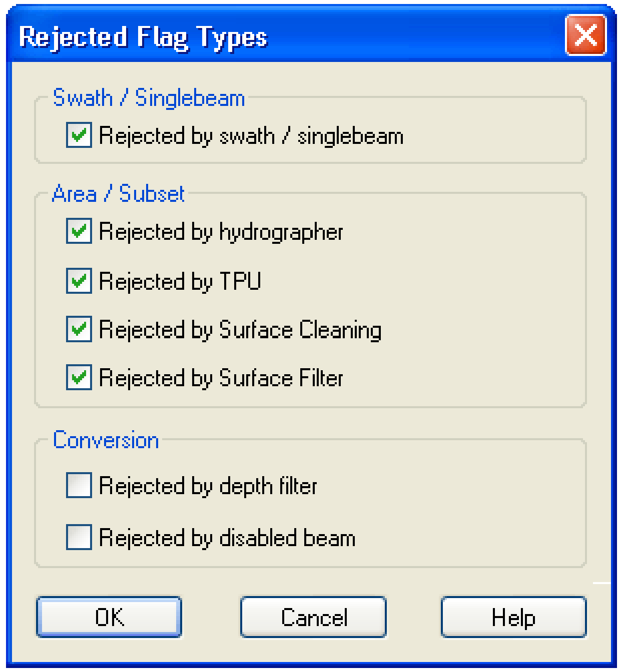
You can also control which type of rejected soundings are displayed in the Single Beam, Swath, and Subset Editors.

To display specific types of rejected data:

Menu	View > Rejected Flag Types

1. Select the View Rejected Flag Types command.

The Rejected Flag Types dialog box opens.



2. Place a check mark beside the box with the type of rejected data you want to view.
   - *Rejected by Swath/Single Beam Editor*: View soundings rejected in Swath or Single Beam Editors or by the Swath or Single Beam filters.
   - *Rejected by Hydrographer*: View data rejected interactively in the Subset Editor by the hydrographer.
   - *Rejected by TPU*: View soundings rejected through the Total Propagated Uncertainty filtering process.
   - *Rejected by Surface Cleaning*: View soundings rejected through the Surface Cleaning process.
   - *Rejected by Surface Filter*: View soundings rejected through surface filtering.
   - *Rejected by Depth Filter*: View soundings rejected by the depth filter in the Conversion Wizard or the Generic Data Parser when it was converted to HIPS format.
   - *Rejected by Disabled Beam*: View soundings rejected due to a disabled beam status during conversion. This occurs when the beam is disabled in the (legacy) vessel configuration or if the sonar system flagged the beam as a bad detection in the raw data file.
3. Click **OK**.

The rejected data types you selected are displayed in the editors.

Critical Soundings

Three types of critical soundings can be flagged in HIPS:

- *Designated*: soundings that have been flagged in the Swath or Subset Editors as being the shoalest sounding in an area.
- *Outstanding*: soundings that have been flagged in the Subset Editor as needing further examination.
- *Examined*: soundings that have been flagged in the Subset Editor as having been examined and verified

The Designated flag identifies the shallowest (also called "shoalest") sounding on important features. For example, upright masts on a significant wreck would be flagged as Designated.

The purpose of the Designated flag is to ensure that the shallowest depths over significant seabed features are maintained for standard hydrographic products.

When these soundings are applied to a BASE Surface, the Surface is regenerated so that the Designated sounding's depth value is applied to the nearest node (see "Finalize BASE Surface" on page 224).

Identify Designated Soundings

Designated Soundings can be identified in two ways:

- automatically with the Critical Sounding Detection command (see "Designate Critical Soundings from a Surface" on page 298)
- manually as part of processing in Single Beam, Swath and Subset Editors (see "Find and Designate Soundings in HIPS Editors" on page 300)

Designated soundings can exported to ASCII, CARIS, GSF, and HOB formats.

The location of designated soundings can be viewed in the Display window. They can also be viewed in the HIPS editor in which they are designated. (See "View Critical Soundings in HIPS Editors" on page 301 and "View Critical Soundings in the Display window" on page 302.)

Designate Critical Soundings from a Surface

The Critical Soundings Detection function automatically finds and designates shoal or deep soundings in a surface or surface attribute layer.

The process contours the selected surface, between minimum and maximum depth at a step interval equal to half the resolution of the surface. For instance, if a surface has a min/max depth of 5/40 metres, with a resolution of 1 metre, the process will contour from 5-40 meters at every 0.5m, resulting in contours at 5.0, 5.5, 6.0, ... , 39.5, 40.0.

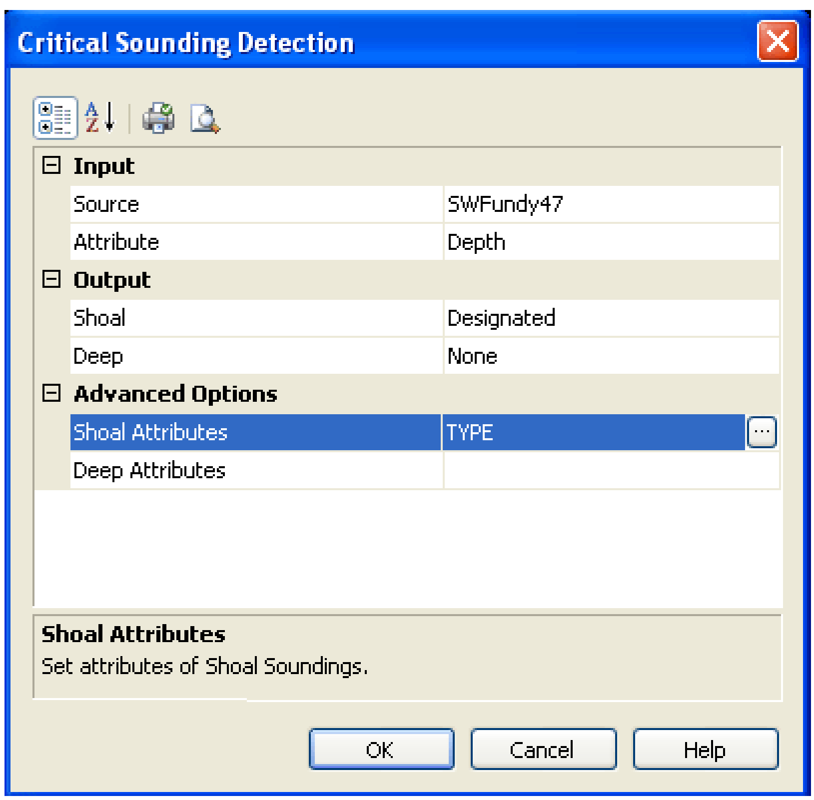
Any isolations (closed contours which have no contours inside them) are identified. The direction within these isolations is known (i.e. either shoaling or deepening). Based on the direction within the isolation, a shoal or deep is flagged on the HIPS data. You can define these flags, and possible attribute values as well, in the Critical Soundings Detection dialog box.

To automatically designate shoal or deep soundings:

1. Open a BASE Surface.
2. Select the Critical Sounding Detection command.

Menu	Process > BASE Surface > Critical Sounding Detection
Pop-up	Right-click on layer and select Critical Sounding Detection

The Critical Sounding dialog box is displayed.



First, define the *Input* to the process:

3. Click in the *Source* field to select a surface by name from the list of surfaces currently open in HIPS. (If you select a surface in the Layers tab before opening the Critical Soundings Detection dialog box, that surface will be displayed by default.)
4. Click in the *Attributes* field to select a surface attribute layer, such as Depth, from the drop-down list.

Next, define the *Output*:

5. Click in the *Shoal* field to select Designated, Outstanding or Examined from the drop-down list. This will determine which soundings are flagged.
6. [Optional] Click in the Deep field and do the same.

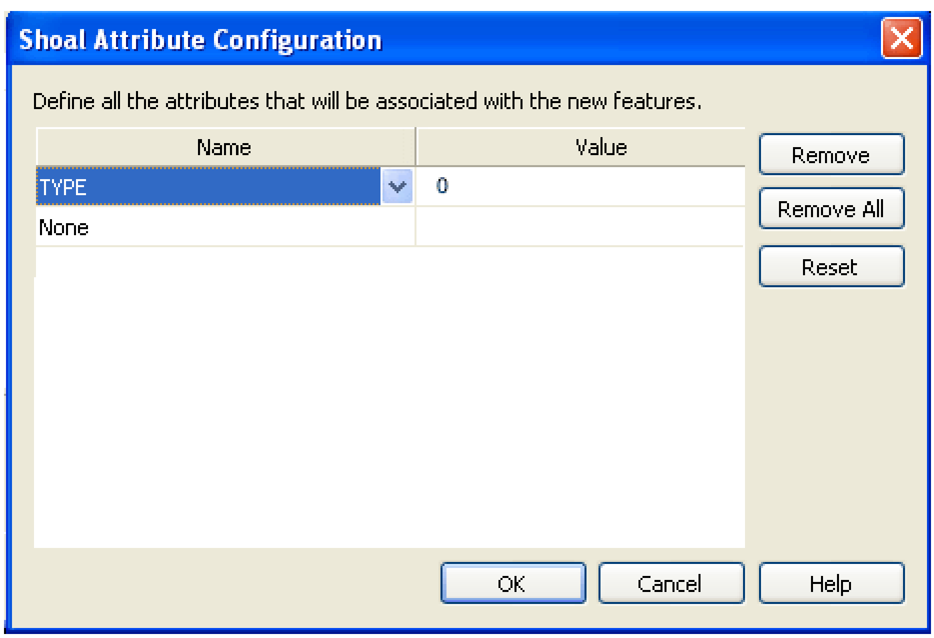
At least one Output Type needs to be set to run the process.

Advanced Options are available to set additional user-defined attributes for the detected critical soundings. Any attributes set here are applied to the detected soundings when the process is run.

7. Click in the *Shoal Attributes* field to open the Shoal Attribute Configuration dialog box.

Attributes in this list and their values are defined in the Catalogue Editor and contained in the catalogue for the current project.

In the image below, an attribute called “Type” has been selected. If this “Type” were to represent, as an example, the type of process which designated the sounding, then the Value that would be set would be either, say, 0 if detected automatically, or 1 from manual processing.



The attributes defined will be listed in the Shoal Attributes field in the previous dialog box, when you click **OK**. When the Detection process is run, the attributes and their set values will be saved with the detected soundings.

8. Click in the Name field and select an attribute from the drop-down list.
9. Click in the Value field to set a value.

To clear an added attribute row, click **Remove.** Remove All will clear all the entries from this dialog box and from the Advanced Options fields.

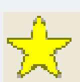
Reset will reset this list to the entries listed when the Shoal Attributes Configuration dialog box was opened.

10. Click **OK** to return to the previous dialog box.
11. [Optional] Repeat the process to set additional attributes for Deep.
12. Click **OK** to run the detection process.

Find and Designate Soundings in HIPS Editors

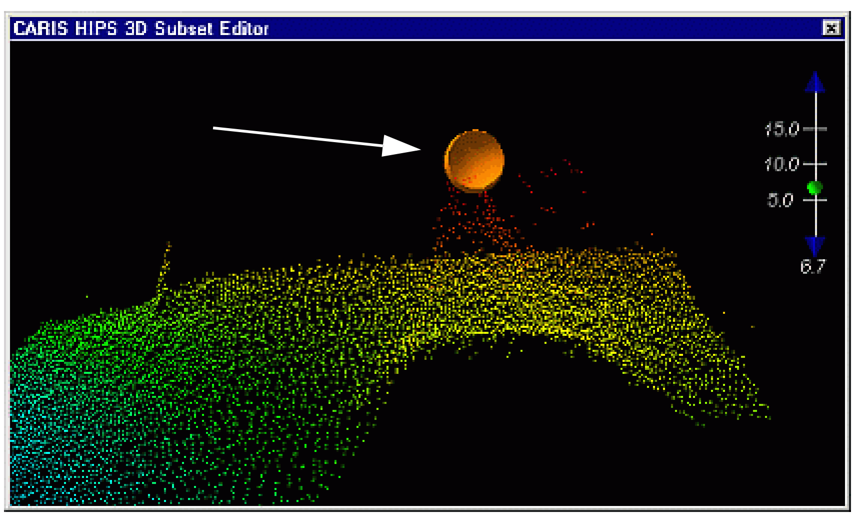
To flag a shallow sounding as the shoalest:

1. Open either Single Beam, Swath or Subset Editor, and load data.
2. Select the shallowest sounding from a cluster of soundings around a shoal feature.

Menu	Edit > Status Flag > Designate
Tool	
Key	<D>

3. Select the Designate command.

The sounding will be flagged as Designated in the Selection tab, and highlighted in the editors by a special symbol (as shown in the zoomed-in view below).



Find Designated Soundings

The Find Designated command automatically selects the shallowest sounding in cluster of highlighted soundings. This feature reduces the time needed to designate shallowest soundings.

To automatically find and designate the shallowest sounding:

1. Use the cursor to highlight soundings in Swath, Single Beam or Subset Editor.
2. [Optional] Use the Query command to view the soundings in the Selection tab.
3. Select the Find Designated command.

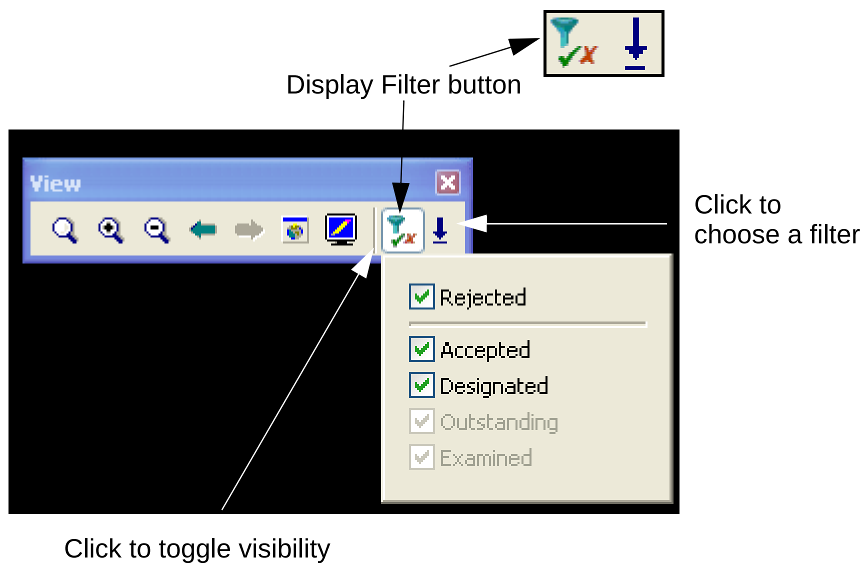
Menu	Edit > Status Flag > Find Designated
Tool	

View Critical Soundings in HIPS Editors

To view critical soundings in an editor such as Swath Editor, use the two-part Display Filter button on the View tool bar. This control will display data in Accepted, Designated, and other critical sounding states.)

Use the right half of the button to set which data to display. When the left half of the button is depressed, the selected data type is displayed. When the button is not active, all data is displayed.

.



For example, to toggle between displaying all the data open in a HIPS editor, and displaying only Designated soundings:

1. Open data in an Editor.
2. Click the arrow part of the Display Filter button and select the *Designated* check box from the drop-down list.
3. Depress the Display Filter button to make the Designated data visible.
4. Click the button again to see all data.

View Critical Soundings in the Display window

Critical soundings that have status flags set during subset and swath cleaning are contained in a HIPS Data layer in the Layers tab.

To view these critical soundings in the Display window:

1. Select the Critical Soundings layer in the Layers tab of the Control window.
2. Refresh the display.

The Designated, Outstanding and Examined soundings will be displayed.

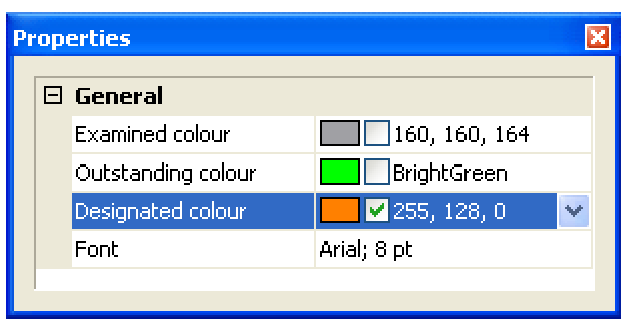
Each type has a default display colour, based on status. These colours by status are set in the Display tab of the Tools > Options dialog box. See "Status" on page 115 in the Reference guide.

Properties

You can over-ride these default colours using the Properties for the Critical Soundings layer. Settings changed in the Properties window are retained in the session file.

1. Highlight the Critical Soundings layer in the Layers tab.
2. Open the Properties window.
3. Highlight a status field (in the example below, Designated is selected).
4. Set a new colour from the drop-down list.
5. Refresh the Display.

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



To limit the type of critical soundings visible in the Display window, for example, to display only the Designated soundings:

1. Highlight the Critical Soundings layer in the Layers tab.
2. Open the Properties window.
3. Remove the check mark from the check box for the status types you don't want to view, leaving the desired status type active (as in the illustration above).

To change the type face and font size of the soundings:

4. Highlight the Font field in the Properties window, and click the browse button to select from the standard Font dialog box.

These settings are saved when you save your session.

Outdated layer

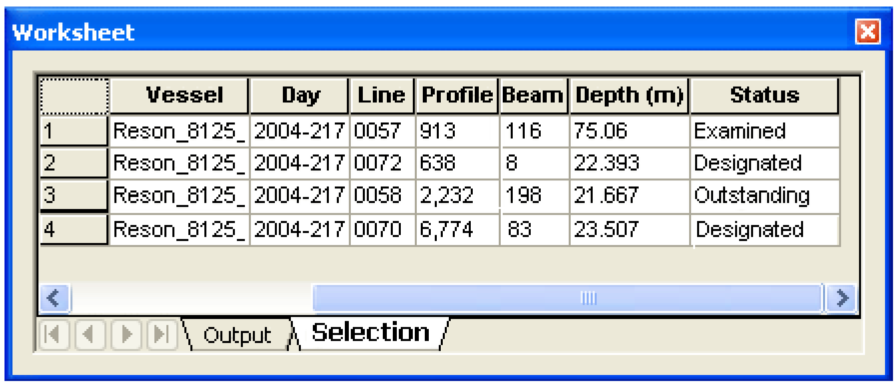
If changes are made to the Critical Soundings layer, the icon for that layer will indicate that the layer is outdated.

To update the Critical Soundings layer:

1. Right-click on the layer, and select Regenerate from the pop-up menu.

Query Soundings

To view information on specific soundings, select the soundings in the Display window and use the Query command. (See "Query Data" on page 273.) Information about the soundings will be displayed in the Selection tab of the Worksheet window.



	Vessel	Day	Line	Profile	Beam	Depth (m)	Status
1	Reson_8125_	2004-217	0057	913	116	75.06	Examined
2	Reson_8125_	2004-217	0072	638	8	22.393	Designated
3	Reson_8125_	2004-217	0058	2,232	198	21.667	Outstanding
4	Reson_8125_	2004-217	0070	6,774	83	23.507	Designated

(See also "Subset Cleaning" on page 339 and "Swath Cleaning" on page 324.) For more information on sounding status flags, see "Status Flags and Values" on page 138 in the Reference Guide.

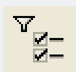
Protect Critical Soundings

You can set a protective sphere around critical soundings to protect any other soundings within the sphere from being rejected when you apply automated filters.

The size of this protective area is determined by setting a radius. You can also select which type of critical soundings the protection is applied to.

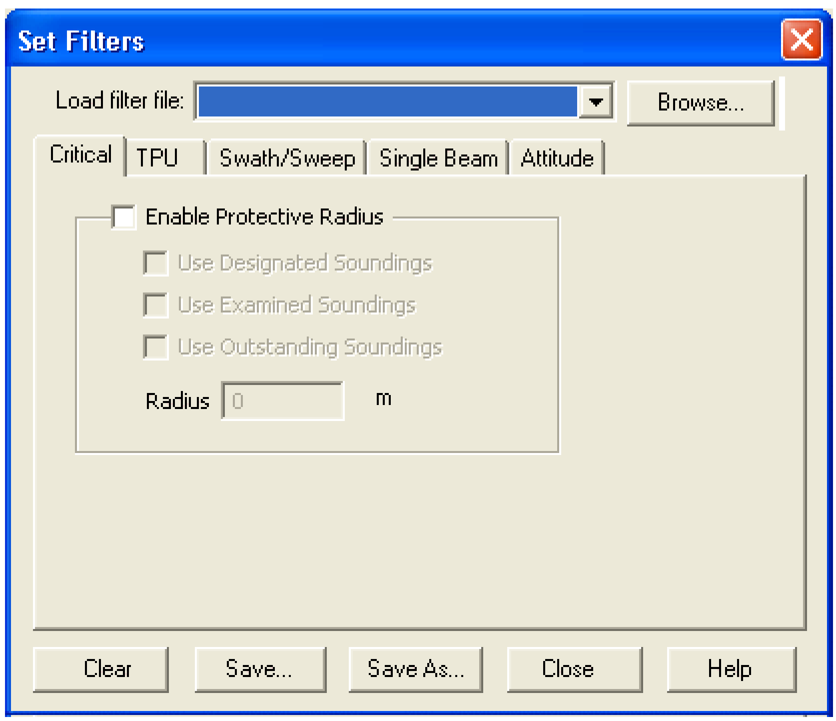
> Data must be merged before critical sounding filtering is applied.

To prevent the neighbours of critical soundings from being rejected during automated filtering:

Menu	Tools > Set Filters
Tool	

1. Select the Set Filters command from the Tools menu.

The Set Filters dialog box is displayed.



2. Select the Critical tab.
3. Select the *Enable Protective Radius* check box.
4. Select which type(s) of soundings to apply the radius to.
5. Type the dimensions of the protective area in the *Radius* field. This value is in the default units set in Tools > Options > Display > Units.
6. Click **Save** or **Save As** to save your filter file so it can be applied with the Apply Filters command.

Restart Cleaning

Edits made to sounding, attitude, and navigation data using the various data cleaning tools can be undone by using the Restart Cleaning function. The Restart Cleaning function resets status flags for each selected track line from “Rejected” to “Accepted”.

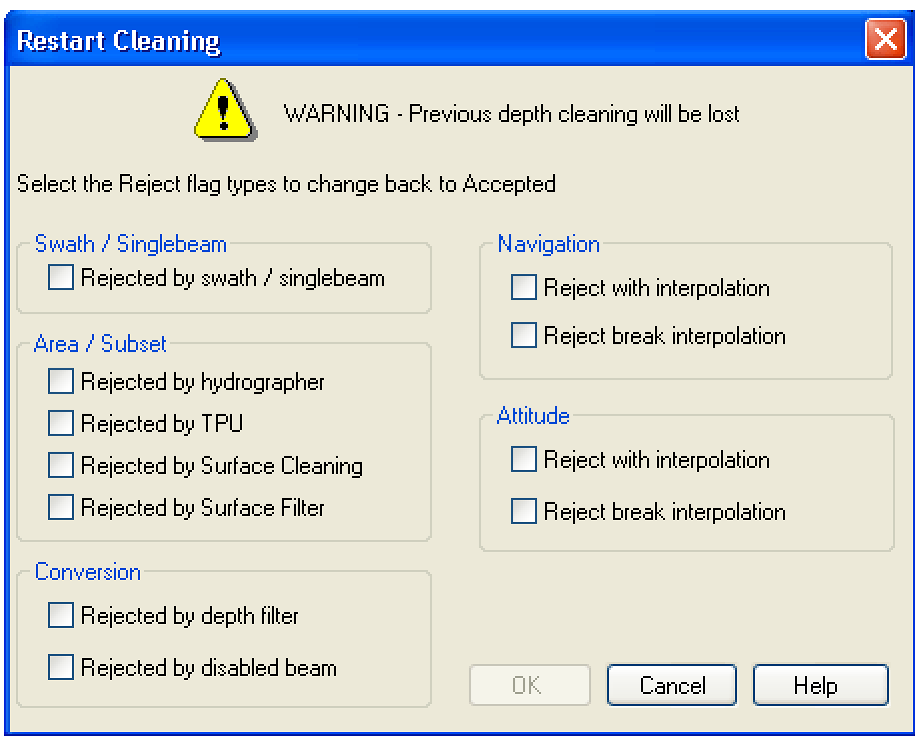
You can set which editing process will be reversed by selecting the appropriate check boxes in the dialog box.

> **Warning**: All previous depth cleaning is lost once this command is implemented.

1. Select the track line(s).
2. Select the Restart Cleaning command.

Menu	Edit > Restart Cleaning
Tool	

The Restart Cleaning dialog box is displayed.



3. Select any of the following check boxes to select a rejected data criteria.
   - *Rejected by Swath Editor/Single Beam Editor*: Reset to “Accepted”, any sounding data that was rejected in Swath or Single Beam Editor, or by the Swath or Single Beam filters.

- *Rejected by Hydrographer*: Reset to “Accepted”, any sounding data that was interactively rejected in Subset Editor.
- *Rejected by TPU*: Reset to “Accepted”, the data that was rejected during Total Propagation Uncertainty (TPU) filtering.
- *Rejected by Surface Cleaning*: Reset to “Accepted”, the data that was rejected during Surface Cleaning.
- *Rejected by Surface Filter*: Reset to “Accepted”, data that was rejected during CUBE filtering.
- *Rejected by depth filter*: Reset to “Accepted”, data that was rejected by the depth filter during conversion.
- *Rejected by disabled beam*: Reset to “Accepted”, data that was rejected due to the disabled beam flag during conversion.
- *Rejected with interpolation (Navigation)*: Reset to “Accepted”, any navigation data rejected with interpolation in Navigation Editor.
- *Rejected break Interpolation (Navigation)*: Reset to “Accepted”, any navigation data rejected without interpolation in Navigation Editor.
- *Rejected with Interpolation (Attitude)*: Reset to “Accepted”, any attitude data that was rejected with interpolation in Attitude Editor.
- *Rejected break Interpolation (Attitude)*: Reset to “Accepted”, any attitude data that was rejected without interpolation in Attitude Editor.

4. Click **OK**.

The selected data is now flagged as accepted.

Quality Control Reports

The Quality Control (QC) process in HIPS is typically used to statistically compare soundings recorded from check lines against selected attribute values from a BASE Surface in the same survey area.

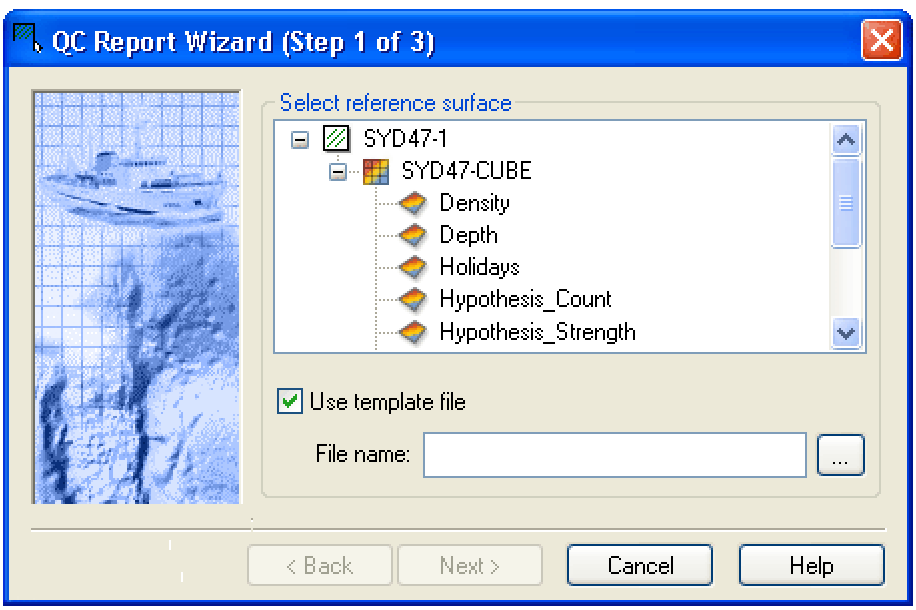
Statistical variance between the check-line soundings and the attribute values are shown in a table in QC Reports tab of the Worksheet window once the QC tool is run.

1. Open data.
2. Open a field sheet.
3. Select a survey line in the Display (or you can select a Project, Vessel, or Day folder in the Project tab).
4. Select the Quality Control command.

Menu	Process > QC Report

QC Report - Step 1

The QC Report - Wizard Step 1 dialog box is displayed.



1. Select a BASE Surface attribute from the *Select Reference Surface* list.

A template file is an XML file that contains output from the Quality Report. You can save the output as a template file or open an existing template file. If you open an existing template file, the settings saved in that file are applied to the options in the other dialog boxes.

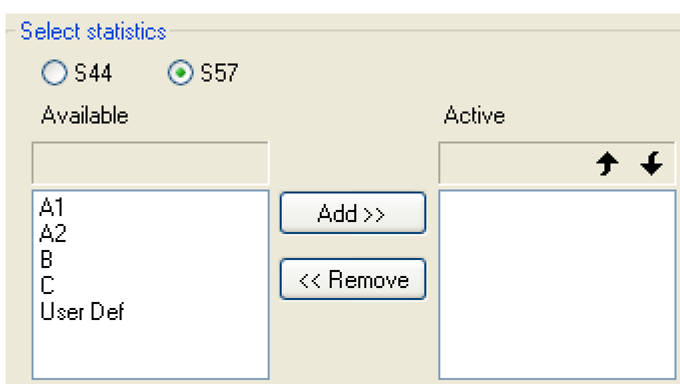
2. Select the *Use template file* check box to create or open a template file.
3. Click **Browse** to select a location and name for the file or select an existing file.
4. Click **Next**.

QC Report - Step 2

The QC Report - Wizard Step 2 dialog box is displayed.

This dialog box selects the IHO standard for determining what percentage of soundings fall within a selected error limit for depth accuracy. You can use the S-44 Ed. 4 survey orders or S-57 CATZOC zones of confidence. (For more information on how error limit for depth accuracy is calculated, see “TPU Filtering” on page 159).

S44 is selected by default. If you select *S57*, the dialog box is refreshed to show the s-57categories relevant to S-57CATZOC zones of confidence.



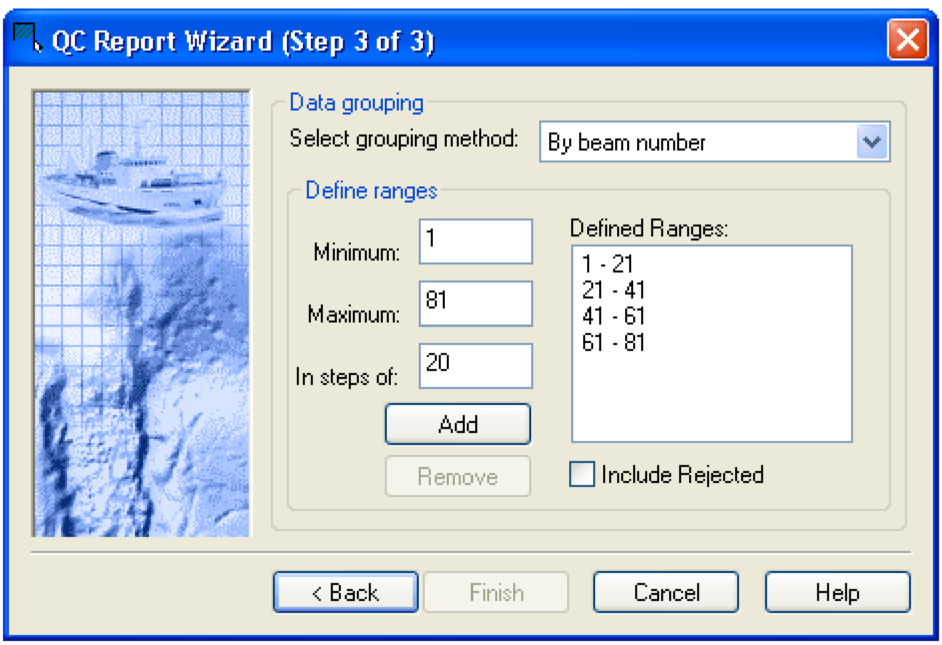
1. Select *S44* or *S57*.
2. To display a category in the QC report, select it in the *Available* list and click **Add** to move to the *Active* list.

If you are using the user defined (*User Def*) category, the *a* (constant depth error) and *b* (factor of the depth dependent error) fields become active once it is moved to the *Active* list.

3. [Optional] Type the values for the user defined category in the *a* and *b* fields.
4. To remove a category, select it in the *Active* list and click **Remove** to move it back to the *Available* list.
5. Select a category and click the up or down arrow buttons at the top of the *Active* list to determine its order in the *Active* list. This will determine the order that data is displayed in the Worksheet window when the QC Report is generated.
6. Click **Next**.

QC Report - Step 3

The QC Report - Wizard Step 3 dialog box is displayed.



This dialog box defines how soundings are grouped in the QC Report. There are three options available:

- *Beam number*: Soundings are grouped according to across track beam number.
- *Angle*: Soundings are grouped according to beam angle from nadir.
- *Distance*: Soundings are grouped according to across track distance from nadir.

1. Select a method for grouping soundings from the drop-down list.

Soundings can be further organized by range. The range values must correspond to grouping method selected above.

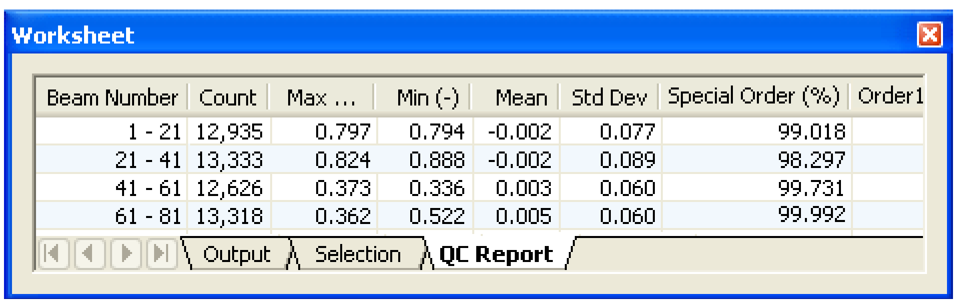
2. Type the range in the *Minimum* and *Maximum* fields.

Ranges are incremented according to the value entered in the *In steps of:* field. For example, for beam numbers 1 to 100 with an increment of 10, the beams are then displayed in the QC Report tab in 10 rows.

3. Type a value to increment the ranges.
4. Click **Add** to display the range in the *Defined Ranges* list.
5. To remove a range, select it in the *Defined Ranges* list and click **Remove**.
6. To include rejected soundings, select the *Include Rejected* check box.
7. Click **Finish** to complete the wizard and close the dialog box, or click **Cancel** to close the wizard without implementing changes.

Quality Control Statistics

The quality control report is displayed in the QC Reports tab of the Worksheet window.



Beam Number	Count	Max ...	Min (-)	Mean	Std Dev	Special Order (%)	Order1
1 - 21	12,935	0.797	0.794	-0.002	0.077	99.018	
21 - 41	13,333	0.824	0.888	-0.002	0.089	98.297	
41 - 61	12,626	0.373	0.336	0.003	0.060	99.731	
61 - 81	13,318	0.362	0.522	0.005	0.060	99.992	

These fields are:

Field	Definition
Beam Number	Range grouping as defined in the third dialog box in the wizard.
Count	Total soundings in range.
Max	Maximum distance of soundings above surface.
Min	Maximum distance of soundings below surface.
Mean	Mean difference of soundings to surface.
Std Dev	Standard deviation of mean differences is presented at one sigma or 68% Confidence Interval.
(Selected S-44 or S-57 categories)	Percentage of soundings that fall within the selected S-44 survey orders or S-57 CATZOC zones of confidence categories.

13

Statistical Surface Cleaning

The surface cleaning function is used to perform area-based statistical data cleaning on multibeam data in HIPS.

The process uses advanced tiling technology for subdividing the entire area into variable size and density cells. It then applies polynomial regression statistics to compute the surface.

In this chapter...

Process 312
Polynomial Surface Types 313
Creating Tiles and Cleaning Data 314
Query data in tiles 320
Tile Histogram 319

Process

In surface cleaning, tiling is used to subdivide the area into manageable cells where each tile is considered separately during the polynomial regression and statistical threshold testing. The tile split criteria is based either on the density of soundings within the tile, or the tile size, or both.

The Surface Cleaning function automatically creates a Starting Level Tile set (even if you do not select the tiling option). The Starting Level Tile Set is created by taking the largest dimension of the field sheet and dividing the area by 3.

For each tile, a polynomial solution is computed using iterative least-squares re-weighted regression. The objective of the regression is to generate a surface defined by a set of polynomial terms, which closely matches most of the real depths. However, since the number of terms defining the polynomial surface are less than the number of points considered, then the points do not fit the surface exactly. Using the residual values of each point as a goodness measure, outliers can be detected by testing for high residual values.

The process is iterative within each tile, and for each new iteration the soundings are re-weighted inversely proportional to their previous residual. You can fix the re-weighting scheme to apply full weight to all soundings within a defined limit from the surface. This re-weighting limit can be based on a fixed distance or a multiple of the standard deviation of the residuals within the tile.

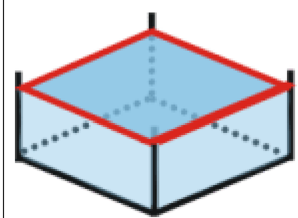
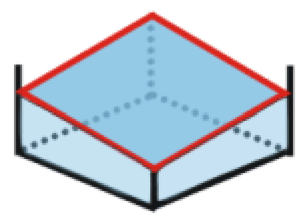
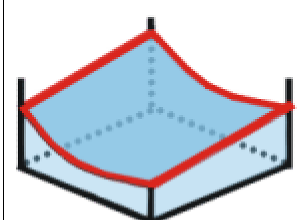
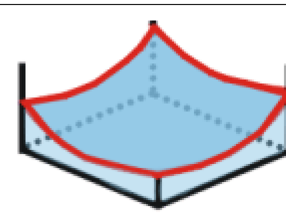
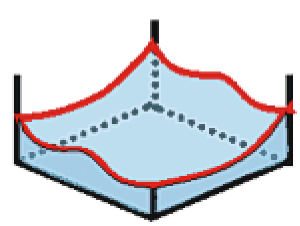
You can also define the maximum number of iterations. The buffer around each tile determines how much extra data is used to compute the tile polynomial surface. This helps eliminate edge effects during statistical testing.

The second stage of surface cleaning is the testing of the residuals once the surface has been computed. The relevant parameters are the rejection threshold scaling values in both the positive and negative directions (above and below the surface), and the minimum absolute residual of an outlier. The latter parameter specifies that no point within this distance (either above the surface or below) are rejected. This is an important parameter if the local standard deviation of a tile is small, because the scaling values are multiplied by a small standard deviation and may reject points which are only slightly removed from the surface. This parameter is location dependent and maybe used, for example, to prevent the rejection of rocks of a specific height. The rejection threshold values are also location dependent and surface dependent.

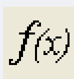
Polynomial Surface Types

The following table lists each of the polynomial surface types with the corresponding equation and a graphical example of each surface type.

Polynomial Surface Types

Surface	**Equation**	**Example**
Mean	$-f(xy) = b0$	
Tilted Plane	$-f(xy) = b0 + b1x + b2y$	
Curved Tilted Plane	$-f(xy) = b0 + b1x + b2x + b3xy$	
Quadratic	$f(xy) = b0 + b1x + b2x + b3xy + b4x2 + b5y2$	
Cubic	$f(xy) = b0 + b1x + b2x + b3xy + b4x2 + b5y2 + b6x2y + b7xy2 + b8x3 + b9y3$	

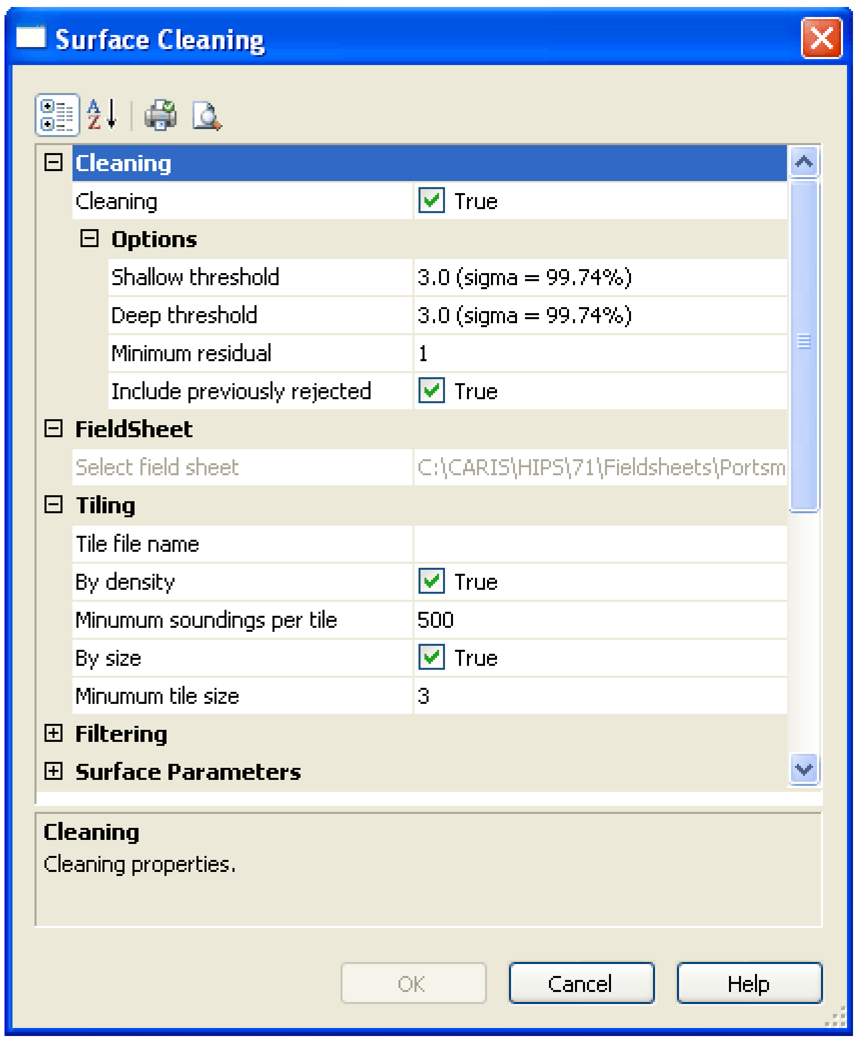
Creating Tiles and Cleaning Data

Menu	Tools > Surface Cleaning
Tool	
Popup	New > Surface Cleaning

Data must be merged before surface cleaning can be performed.

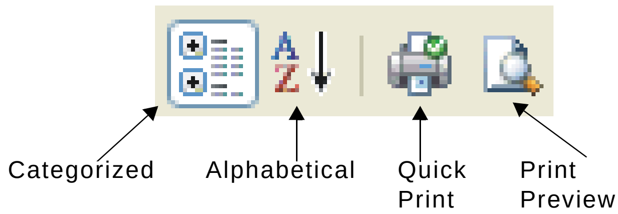
1. Select a field sheet.
2. Select a Surface Cleaning command.

The Surface Cleaning dialog box is displayed.



By default, the dialog box fields are listed by category. To view these fields in alphabetical order:

3. Click on the Alphabetical button in the toolbar.



Cleaning

4. Set the *Cleaning* check box to *True* to enable the surface cleaning functions.
5. Enter cleaning Options, as needed, in the following fields. (The sigma value is a multiple of the standard deviations of residuals in the tile. The equivalent confidence values are displayed as a percentage.)
   - *Shallow threshold*: The threshold above the polynomial surface. The value is selected by clicking the up and down arrow buttons beside shallow threshold field.
   - *Deep threshold*: The threshold below the polynomial surface. The value is selected by clicking the up and down arrow buttons beside deep threshold field.
   - *Minimum residual*: A sounding that fails the threshold test will not be rejected if the absolute value of the residual is less than the minimum.
   - *Include preciously rejected*: Include any data previously rejected by Surface Cleaning in this computation.

Field sheet

The Field sheet field displays the name of the selected field sheet.

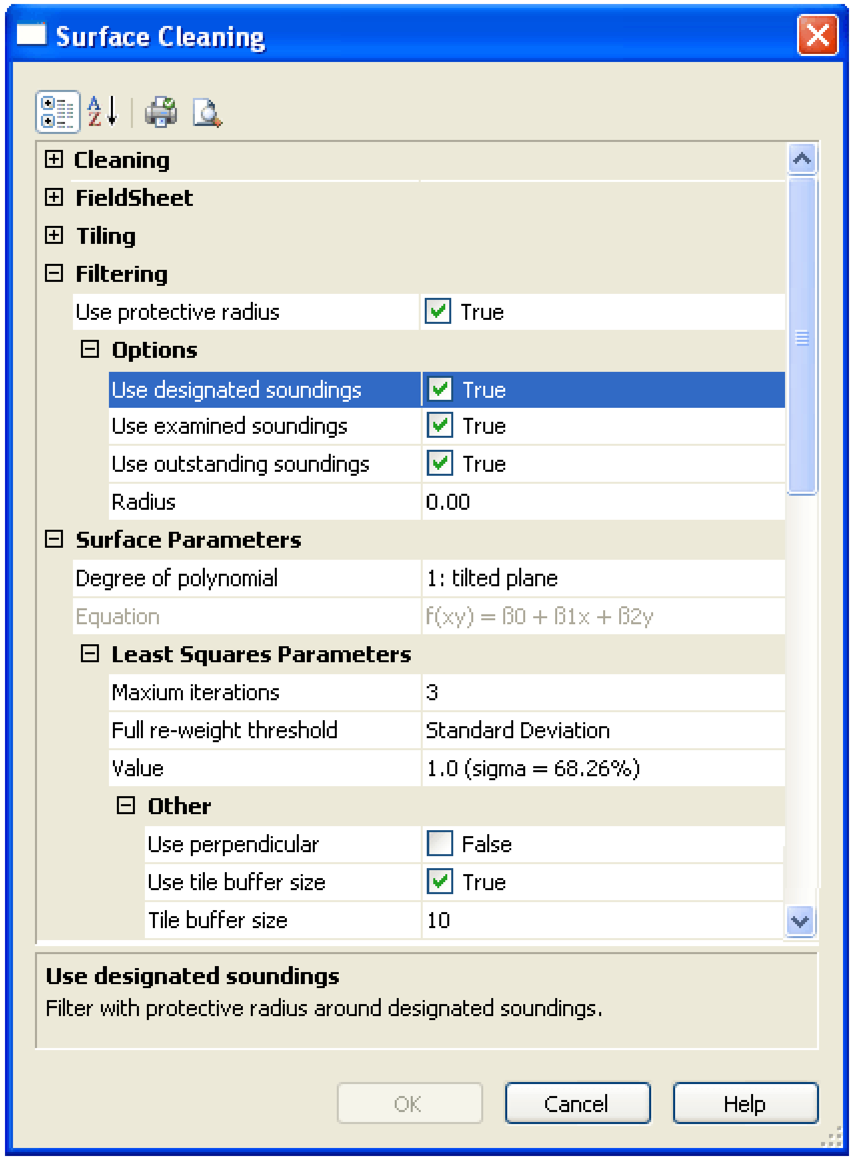
Tiling

Choose the method to create tiles: either *By Density* of tiles or *By Size* of tiles.

6. Set *By Density to* True. Each tile is continuously sub-divided until one of the sub-tiles has less than the minimum number of soundings.

   Type a *Minimum soundings per tile* value for the minimum number of soundings that will be covered by a tile.
7. Set *By Size* True. This will continuously sub-divide each tile until the sub-tiles reach the minimum size. (This size cannot be larger than the Starting Level Tile Set. The Starting Level Tile Set is created by taking the largest dimension of the field sheet and dividing the area by 3.)

   Set the *Minimum size of tile*, in default measurement units.
8. Type a name for the tile file. Surface Cleaning cannot be applied unless a tile file is set. (All tile files are assigned an .mts extension. Also created is a .mmi file that records the list of track lines used to create the tiles.)



Filtering

You can set a protective sphere around critical soundings to protect any other soundings within the sphere from being rejected when you apply automated filters. (See "Critical Soundings" on page 297.)

The size of this protective area is determined by setting a radius. You can also select which type of critical soundings the protection is applied to.

9. Set *Use Protective Radius* to True.
10. Set to True the type(s) of soundings to apply the radius to.
11. Type the dimensions of the protective area in the *Radius* field. This value is in units set as default Horizontal Length in Tools > Options > Display > Units.

Surface Parameters

Set advanced parameters, as needed:

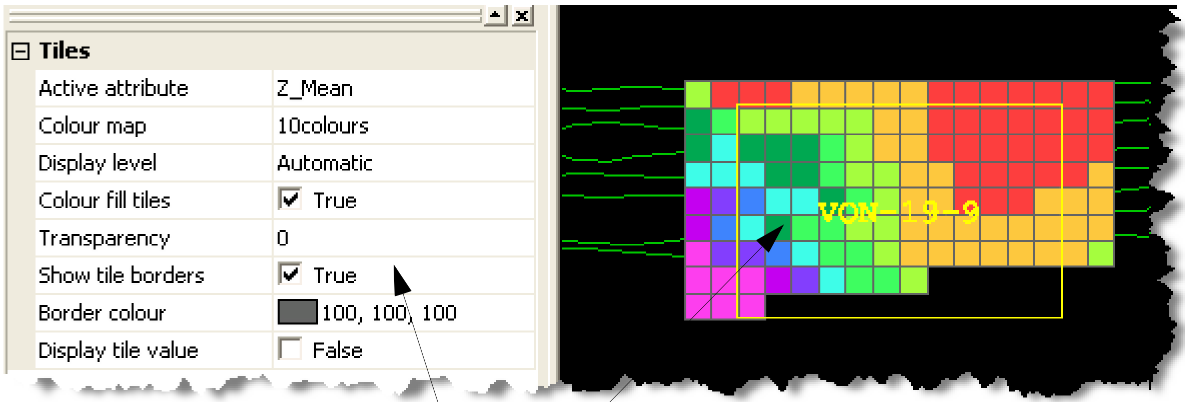
12. *Degree of polynomial*: From the drop-down list, select one of the available polynomials to match the expected complexity of the sea floor.
    - The equation for the selected degree will be displayed in the field below. (See "Polynomial Surface Types" on page 313 for polynomial descriptions.)
13. Set the *Least Squares Parameters*:
    - *Maximum iterations*: Set the number of times the least squares computation of the polynomial will process the soundings. This re-weighting continues until there are no additional rejected soundings or the maximum number of iterations is reached.
    - *Full re-weighted threshold*. Select either by:

      Standard deviation: The full re-weighting of soundings for each iteration can be a multiple of the standard deviation of the residuals. The *Value* field will show a multiple of the standard deviations of residuals in the tile. The equivalent confidence values are displayed as a percentage.)

      Residual surface value: The full re-weighting of soundings for each iteration can be fixed as a distance from the surface. The *Value* field will show in units as set by the default Horizontal Length units in Tools > Options > Display > Units.
14. Set other options
    - *Use perpendicular*: When this option is set to True, the residual is calculated as the perpendicular distance between the sounding and the nearest point on the polynomial surface. The default calculation for the residual uses the depth difference between the sounding and the polynomial surface at the location of the sounding.
    - *Use tile buffer*: When this is set to True, an extra area around the tile is included in the surface computation.
    - *Tile buffer size*: a percentage of the tile size that describes the width of the buffer around the tile.
15. Click **OK**.

The tiles are displayed in the Display window and the tile surface is listed (by the file name that was assigned in Step 4) in the Layers tab under its parent field sheet layer.

The properties of the tiles can be viewed in the Properties window.



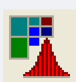
Properties of the displayed tiles

Tile Histogram

The Tile Histogram dialog box displays the distribution of residuals within a tile.

If the histogram resembles a bell curve then the selected polynomial is well suited for the area. However, if the histogram has many peaks or a wide centre, then the polynomial surface is not suited to the area. If there are severe shifts in data distribution, then the area may have to re-tiled using a higher degree of polynomial.

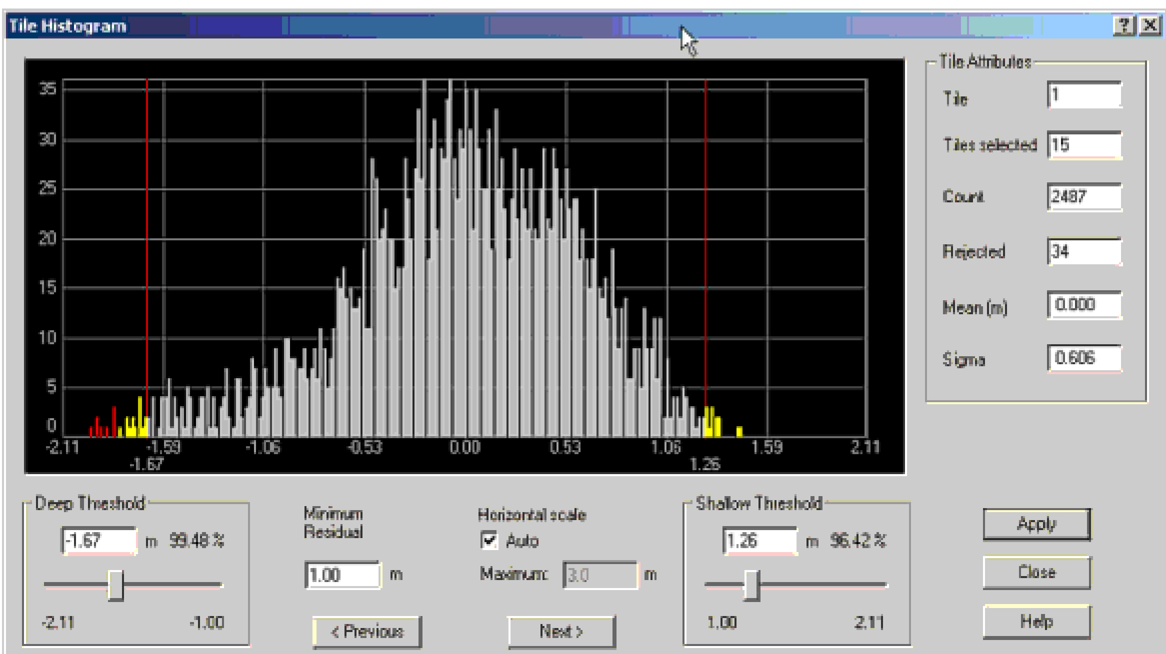
You can also use the histogram to change the threshold values and reject soundings.

Menu	Tools > Surface Cleaning > Tile Histogram
Tool	

1. Select the tiles you want to examine using the tile selection command.
2. Select the Tile Histogram command.

The Histogram dialog box is displayed.

Information about the tile is displayed on the right-hand side of the dialog box.



3. To view data for another tile, click **Next Tile** or **Previous Tile**.
4. Use the *Shallow Threshold* or *Positive Threshold* sliders to adjust these values.

The corresponding confidence intervals and Maximum Residual Distance values change as the bars are moved.

5. Click **Apply**.

The rejected residuals are displayed in red.

Query data in tiles

There are three ways to examine data in tiles:

- "Select and View Tile Data" on page 320
- "Query Soundings in Tiles" on page 320
- "Query Tile Display" on page 321

Select and View Tile Data

To see data about the tiles created for surface cleaning:

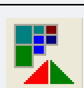
1. Open surface cleaning tiles in the Display window.
2. Select the tile layer in the Layers tab of the Control window.
3. In the Display window, select tiles to be queried.

The data for the selected tiles is automatically displayed in the Selection tab, including among other attributes:

- X and Y coordinates for the centre of the tile
- Z mean, Z minimum and maximum
- Res mean and Res Sigma
- Count of the number of soundings within the tile area, and the number of Rejected soundings within the tile area
- Tile level and tile size

Query Soundings in Tiles

To view data on the soundings in selected tiles:

Menu	Tools > Surface Cleaning > Query Tile Soundings
Tool	

1. Open surface cleaning tiles in the Display window.
2. Select the tile layer in the Layers tab of the Control window.
3. In the Display window, select tiles over the soundings to be queried.
4. Select the Query Tile Soundings command.

The data for each sounding in the selected tiles is displayed in the Selection tab. As well as the Project/Vessel/Day/Line data for each sounding, other attributes are also displayed, such as:

- *X*: and *Y* coordinates for the soundings
- Profile, Beam and Status data
- *Depth*: depth of sounding.
- *Surface Depth*: depth of polynomial surface at the location of the sounding.

- *Residual*: percentage difference between the sounding and the polynomial surface.
- *Shallow Threshold*: depth of rejection threshold above the polynomial surface.
- *Deep Threshold*: depth of rejection threshold below the surface.

Query Tile Display

You can also use the Tile Display Query function to create an SQL-type query that will display only those tiles with selected attributes. For more information, see "Tile Display Query" on page 407.

14

Cleaning Swath Data

Data cleaning in Swath Editor consists of interactively selecting and rejecting soundings and using proven and efficient filtering functions to automatically detect and reject outliers.

You can also flag Designated Soundings.

(For descriptions of the Swath Editor interface and controls, see "SWATH EDITOR" ON PAGE 143 in the Editors guide.)

In this chapter...

SWATH CLEANING 324
AUTOMATIC FILTERING 325
MANUAL CLEANING 328
UPDATE BACKSCATTER 332
FLAG DESIGNATED SOUNDINGS 333
CORRECTION FOR REFRACTION ARTIFACTS 334

Swath Cleaning

During data conversion, the original data file is separated into a number of distinct files, each storing a different type of information. All are related by time. One of these files is the Observed Depths file. This file contains measured single beam, sweep, or swath soundings as reported by the data acquisition system.

The Swath Editor displays the contents of the file for sweep and swath systems in the Display window in graphic form so you can perform cleaning and filtering operations on the data.

Each soundings begins with Accepted status. If rejected (for any reason) during processing it can be restored to Accepted status without loss of data.

Sounding outliers can be rejected by applying automatic swath filters (see "Automatic Filtering" on page 325) or interactively using manual cleaning tools (see "Manual Cleaning" on page 328).

For information on the Swath Editor interface, see Swath Editor in the HIPS and SIPS Editors Guide.

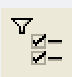
Automatic Filtering

When dealing with high-volume bathymetric datasets, it is advantageous to be able to reject outliers automatically. Using Swath filters can be an effective way to reduce the time it takes to clean large volumes of data.

There are three basic types of swath filters:

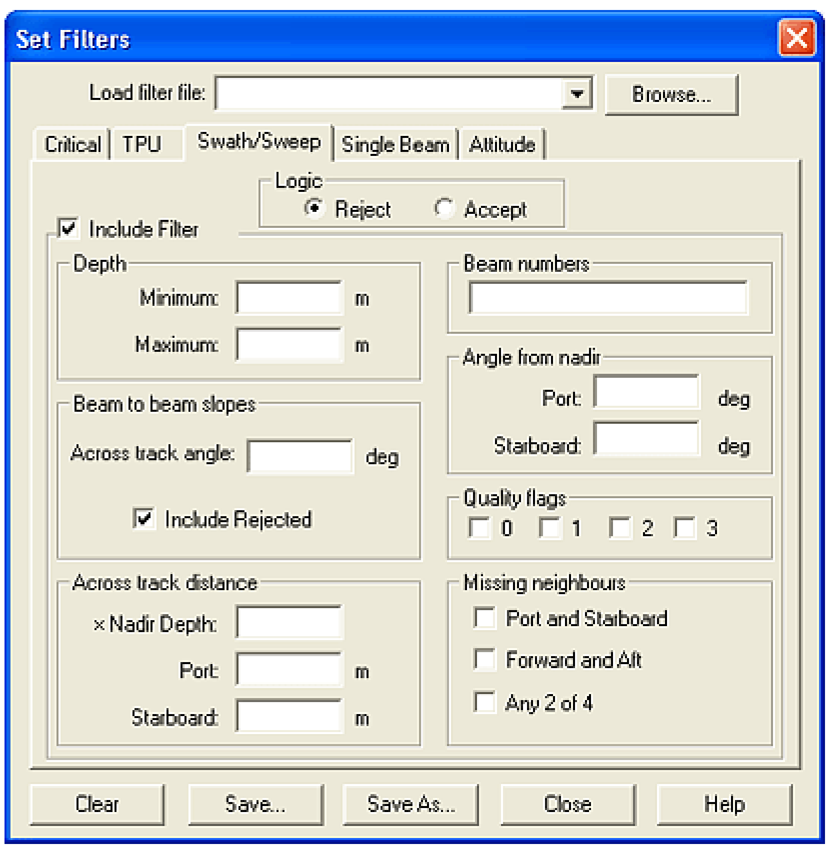
- **Spike detection**: The minimum and maximum depth filters and the beam-to-beam slope filters.
- **Swath reduction**: across track distance filters and nadir angle filters.
- **Sonar quality**: Quality values set by the sonar and missing neighbours

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

Menu	Tools > Set Filters
Tool	

1. Select the Set Filters command.

The Swath/Sweep tab of the Set Filters dialog box is displayed.



1. [Optional] Select an existing HIPS Filter File from the list or click **Browse** to select a file.

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

2. Set the logic to *Accept* or *Reject*.
3. Select the *Include Filter* check box to activate the filtering options.
4. Select the filter parameters used to flag data (see below).

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

5. Click **Close**, or click **Clear** to remove data from the fields.
6. Click **Save As** to save the settings to a filter file.

The following filters are available.

- *Minimum Depth*. Reject any soundings that are shallower than the depth specified (e.g., 10 m).
- *Maximum Depth*. Reject any soundings that are deeper than the depth specified (e.g., 50 m).
- *Across track angle* Beam-to-beam slopes, across track. For each beam on the same swath, calculate the slopes in degrees to the prior and post beams, and if both slopes exceed the defined value and are of opposite sign then reject the beam.

  The *Include Rejected* option includes previously rejected soundings when recalculating the slopes during multiple runs of the filter.
- *Across track distance: X Nadir Depth*. Any soundings with an across track distance, from the centre beam, greater than the nadir depth times this number (e.g,. 3.0) are rejected. For example, in 30 metres of water, with a setting of 3.0, all soundings with an across track distance greater than 90.0 metres are rejected.
- *Across track distance: Port*. Filter a sounding if the across track distance on the port side is greater than the value entered in this field.
- *Across track distance: Starboard*. Filter a sounding if the across track distance on the starboard side is greater than the value entered in this field.
- *Beam Numbers*. Specify beams to be rejected. Separate individual beams with a space and specify a range of beams with a hyphen. For example, entering 12 32 56-60 72 rejects beams 12, 32, 56, 57, 58, 59, 60, and 72.
- *Angles from Nadir: Port and Starboard*. Filter beams that are outside of the designated angles (e.g., 60 degrees). Angles are computed from the nadir using depths and across track distances (roll corrected).

Quality Flags. If your system attaches quality codes to soundings, and those codes are transferred to HIPS during conversion, a sounding can be rejected according to its value.

> Quality flags are currently supplied by Teledyne Reson, Elac, Hypack, and GSF multibeam systems/formats.

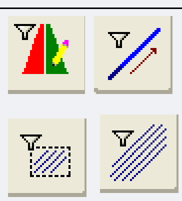
Missing neighbours are beams rejected during the conversion process into HIPS. This occurs if the beam is disabled in the vessel configuration or if the sonar system flags the beam as a bad detect in the raw data file.

- *Port-Starboard*. Beams are rejected if beams on each side of the swath are rejected.
- *Forward-Aft*. Beams are rejected if the adjacent beams in the previous and next swath are rejected.
- *Any 2 of 4*. Beams are rejected if two of the four neighbouring beams are rejected.

During filtering, only accepted soundings are examined. Soundings rejected after one filter pass are not considered in the next.

Apply Filters

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

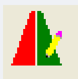
Menu	Tool > Apply Filters > 1 Screen/To End of Line/Selected Lines/ All Lines
Tool	

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

You have four options for applying the filters:

- **Filter 1 Screen**: Apply the filters only to the swaths that are currently visible in the Plan View of Swath Editor.
- **To End of Line**. Apply the filters to the track line currently open in Swath Editor, from the first swath currently visible in the Plan view to the end of the line.
- **Selected Lines**. Apply the filters to all of the currently selected track lines. This function does not require that Swath Editor be open.
- **All Lines**. Apply the filters to all track lines currently open in the Display window. This function does not require that Swath Editor be open.

Manual Cleaning

Menu	Tools > Swath Editor > Open
Tool	

1. Select the Ship Track Lines layer in the Layers tab.
2. Select a track line.
3. Select an Open Swath Editor command.

The Swath Editor opens with all views displayed.

(For a complete description of the Swath Editor interface, and ways of viewing data display options and controls, see SWATH EDITOR in the HIPS and SIPS Editors Guide.)

4. Select soundings to be cleaned. (When soundings are selected they are highlighted in yellow.)
5. Apply the Reject or Reject Swaths command o the sounding as appropriate.

Tools

To assist with manual cleaning, use:

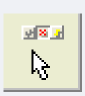
"AUTO CURSOR MODE" ON PAGE 328

"FIND SOUNDING" ON PAGE 328

"QUERY DATA" ON PAGE 331

Auto Cursor Mode

Auto Cursor Mode can be used to speed up the interactive process. This mode combines the selection and Accept/Reject/Query functions into a single procedure.

Menu	Edit > Status Flag > Auto Cursor
Tool	

1. Select the Auto Cursor command.
2. Select a Reject/Accept/Query command.
3. Press the mouse button and draw a bounding box over the data.
4. Release the mouse button.

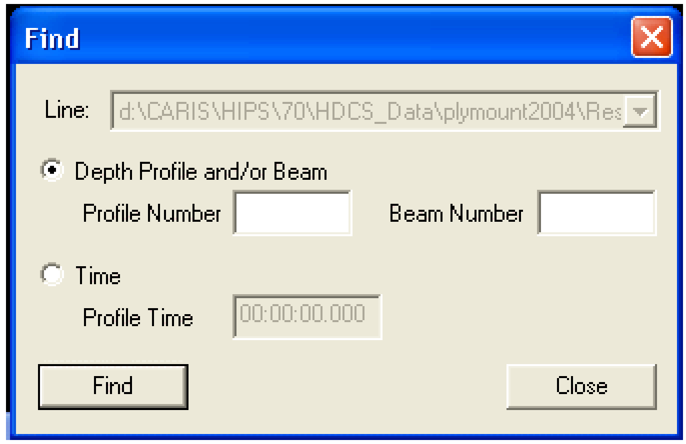
Find Sounding

Use the Find function to search a survey line for a specific sounding(s) by beam and/or profile number.

Tool	
Pop-up	Find

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

The Find dialog box is displayed.



3. To search by the swath or beam number, select the *Profile and/or Beam* option.
4. Type the swath or beam number in the *Profile Number* or *Beam Number* field.
5. To search for a swath that was recorded at a specific time, select the *Time* option.
6. Click inside the *Profile Time* box and type the time in the relevant fields (hour:minutes:seconds:decimal fractions of a second), using the 24-hour clock.
7. Click **Find**.

The selected beam or swath is highlighted in the Plan, Rear and Side Views.

8. Click **Query** to display the data in the Selection tab.

Rejecting Data

A sounding rejected within Swath Editor has the Reject by Swath Editor flag turned on. This flag is displayed in the Status field of the Selection tab when the sounding is queried.

To reject a sounding:

1. Select the data you want to reject.

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

2. Select a Reject command.

The Reject by Swath Editor status flag is now on and the sounding may or may not be displayed depending whether the Show Rejected function is turned on. (See “View Rejected Soundings” on page 295 for description.)

A rejected sounding can always be returned to the “Accept” status without loss of data. To revert rejected soundings back to “Accept”:

1. Select the rejected data so it is highlighted.

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

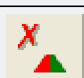
2. Select the Accept command.

The soundings are now flagged as accepted.

Reject Swaths

In some instances it may be necessary to reject all soundings in one or more swaths. This can be done with the Reject Swaths command without having to select all of the data in the swaths.

1. Select some sounding in the swath you want to reject.

Menu	Edit > Status Flag > Reject Swaths
Tool	
Key	<S>

2. Select the Reject Swaths command.

All soundings in the swath (whether individually selected or not) now have the Rejected by Swath Editor status flag turned on. The sounding will not be displayed unless the Show Rejected function is turned on (see “View Rejected Soundings” on page 295).

Query Data

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

Select soundings to view status and other information.

1. Select the data in one of the Swath Editor windows.
2. Select a Query command.

The following data fields for the selected soundings are displayed in the Selection tab:

- time of the swath profile
- d-time from the previous profile
- profile number
- beam number
- across track distance
- along-track distance
- depth
- d-depth from the previous profile
- d-depth from the previous beam
- amplitude / phase detect
- quality value
- status flag
- horizontal error value
- vertical error value

You can reject queried soundings selected in the Selection tab.

1. Query a selection of soundings.
2. Highlight certain soundings in the Selection tab.
3. Select an Accept or Reject command.

You can also use the keyboard to set the flag: “A” key for Accept, “Q” for Query, “R” for Reject, and “S” for Reject Swath.

Update Backscatter

This command updates the backscatter imagery for a line after swath cleaning is complete.

Backscatter data is referenced directly to the individual beams of bathymetry data. After soundings have been rejected in Swath Editor, Update Backscatter is applied to remove the intensity values for the rejected soundings from the backscatter image.

1. Select a track line.

Menu	Tools > Update Backscatter

2. Select the Update Backscatter command.

Flag Designated Soundings

The Designated Sounding flag identifies the shallowest, or "shoalest" sounding in a feature. For example, in a cluster of soundings surrounding an outcrop of rocks, the shoalest sounding among that cluster is identified as Designated.

The purpose of the Designated Sounding flag is to ensure that the shoalest depths over significant seabed features are maintained in charts and other standard hydrographic products.

Soundings can be designated in both Swath Editor and Subset Editor.

For more information, see "Critical Soundings" on page 297.

1. Select the shoalest sounding from a cluster of soundings around a feature in Swath Editor.

Menu	Edit > Status Flag > Designate
Tool	
Key	<D>

2. Select the Designate Soundings command.

The sounding is flagged as Designated.

Correction for Refraction Artifacts

Since it is not possible to completely control the sound speed variations in water we apply sound velocity profiles to correct for this. However, refraction artifacts may still remain, especially if incorrect or insufficient sound velocity profiles are applied either during acquisition or during post processing.

To correct for this, you could choose to limit the usable swath angle (and therefore survey more lines), or you can use the Refraction Editor, which simulates the effects of altering the SVP to derive a better refraction solution.

Since refraction errors are most pronounced in the outer parts of the survey line coverage, they tend to create characteristically curved (upward or downward) refraction artifacts.

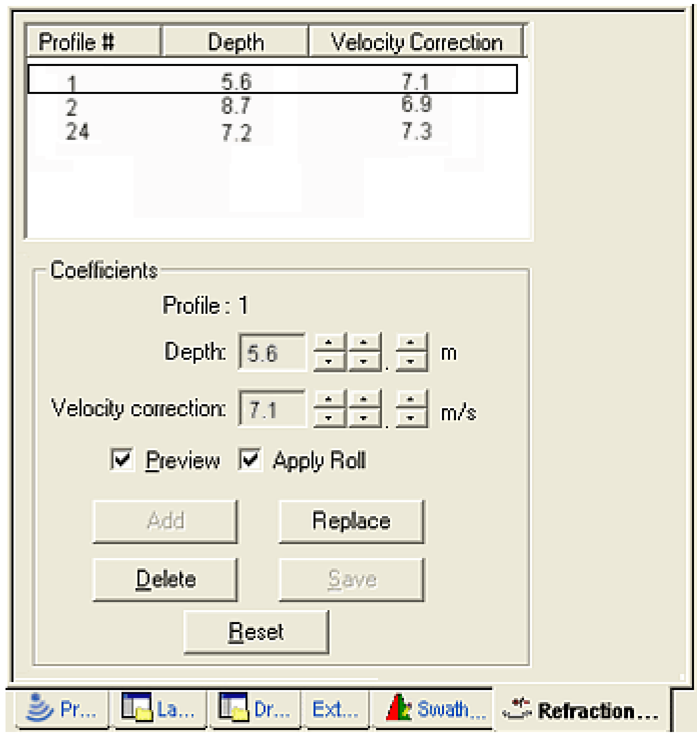
Apply the velocity correction at a user-defined depth. As you adjust the values, you can observe the changes in the shape of the swath profiles. When you arrive at the values which give you the desired correction, you can save these coefficients. Later, during the Merge process, you can apply these Refraction Editor coefficients.

To open the Refraction Editor:

1. Select a track line.
2. Open Swath Editor, then select the Refraction Editor command.

Menu	Tools > Swath Editor > Refraction Editor

The Refraction Editor tab is displayed next to the Swath Editor tab in the Control window.



3. Set the values for depth and velocity correction until the desired effect is observed.
4. Click **Add** to add the new entry to the list of coefficients.
5. Click **Reset** before clicking **Add** to set the values to their original setting.
6. To change an existing coefficient, select it, adjust the settings and click **Replace** to update it.
7. To remove an entry from the list select it and click **Delete**.
8. Click **Save** to save your entries.

As you change the values in the Refraction Editor tab you will be able to see the effects of these changes in the Swath Editor display.

9. Select the *Preview* check box to view the effects of your changes.

If the *Preview* option is selected, you can also enable the *Apply Roll* option. When you first open the Refraction Editor, this control is set to be the same as the setting in the Roll section of the Vessel file (HVF). This control allows you to quickly observe the effects of applying (or not applying) dynamic roll to the sounding data.

> If there is no roll data, or if the appropriate Roll section cannot be found in the HVF, or if the survey line has already been sound velocity corrected, toggling the *Apply Roll* switch will have no effect.

10. Select the *Apply Roll* check box to see the effects of roll on the data

The velocity correction and depth are recorded with the current swath profile number. While paging through the track line during cleaning, new swath/correction/depth values can be recorded.

Interpolation

When you add the first Refraction Coefficient, the correction starts at that profile and goes to the end of the line with that same correction. If you add a second Refraction Coefficient after the first, the correction applied between the two profiles is interpolated. This is done so that there is no "jump" in the profile when the correction at the second Refraction Coefficient is applied to the data.

This interpolation is linear. That is, if the selected profile is between two refraction entries, the depth and delta velocity are both interpolated from these entries.

Querying

Querying the data in Swath Editor will always give the results from the display, so if you change the display with the *Preview* option of the Refraction Editor turned on, the query will return the results of the change. The Observed Depth data (across track, along track and depth) does not change.

The effects of the Refraction Editor can be turned on or off during Calibration. As well, the effects on the final merged depths can be modified any time by editing the coefficients and re-merging, or the effects can be removed by re-merging and choosing not to apply the coefficients.

15

Process Data in Subsets

Create subsets to efficiently visualize and clean data from multiple track lines at the same time, and use subset tiles to track the progress of processing.

In this chapter...

Data Cleaning Workflow 338
Subset Cleaning 339
Designate Soundings 342
Track Progress with Subset Tiles 343
Subsets and BASE Surfaces 350

Data Cleaning Workflow

A subset is a user-defined rectangular area that contains some of the soundings of a survey project. Dividing the area of the survey into subsets makes the cleaning of a large data set more manageable.

The following workflow is a suggested method for cleaning data with Subset Editor.

1. Open a field sheet with a BASE surface.

   (See "Create a Field Sheet" on page 174)
2. Create Subset Tiles in the field sheet area.

   (See "Track Progress with Subset Tiles" on page 343)
3. Open Subset Editor and create a subset area within the tiled area. (See Subset Editor Interface in the HIPS and SIPS Editors Guide.)
4. Use the 3D View Controls to adjust the view of the data. (See Subset 3D View in the HIPS and SIPS Editors Guide.)
5. Clean subset data.

   (See "Subset Cleaning" on page 339)
6. Classify data in subset area.

   (See "Tracking Cleaning Status" on page 349)
7. Define another subset and repeat the process.

If you are processing a small amount of data, using subset tiles to track progress can be omitted (Steps 1 and 2), and Subset Editor can be directly opened into the track lines.

However, if you are processing large areas of data, the subset tiling method is a useful way to track changes.

(For hypothesis editing a CUBE surface in Subset Editor, see "Hypothesis Editing" on page 259.)

Subset Cleaning

Soundings in the 2D and 3D Views can be interactively examined and their status flags can be modified.

To open Subset Editor:

Menu	Tools > Subset Editor
Tool	

1. Select the Subset Editor command.
2. Define the area in the Display window to be included in the subset.

   See “Open Subset Editor and Load Data” on page 111 in the Editors Guide.

Menu	Tools > Subset Editor > Load
Tool	

3. Select the Load Subset command to the load the data into the Subset Editor windows.

The subset of your data is generated and displayed in the 2D and 3D windows in Subset Editor

If data for any line has not been merged, the depth information may not be accurate. In this case, the subset will be loaded, but the Output window will warn you that the processed depths for a specified line are outdated and require Merge. (See “Merge” on page 167.

Reject

Each sounding in the HIPS format carries several status flags that are turned on and off as needed. Each sounding begins as an Accepted status. If rejected during processing, it can always be returned to the accepted status without loss of data.

Note: When you select data in the 3D View, all data in the highlighted area is selected, not just the visible data.

1. In either the 2D or 3D View, select the data you want to reject so it is highlighted.

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

2. Select the Reject command.

If you query the rejected soundings, you will see that they now display the status flag “Rejected (Subset-Hydrographer)”. As well, the soundings may or may not be displayed depending on whether the Show Rejected button is toggled to visible.

Note: When you select data in the 3D View, all data in the highlighted area is selected, not just the visible data.

Accept

Change rejected soundings back to the accepted status.

1. Select the rejected data that you want to return to accepted status.
2. Select the Accept command.

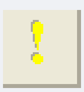
Menu	Edit > Status Flag > Accept
Tool	
Pop-up	Accept
Key	<A>

The soundings are now marked as accepted.

Outstanding

Flag a sounding as requiring further examination.

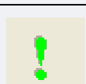
1. Select the data that you want flag as Outstanding.
2. Select the Outstanding command.

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

Examined

Flag questionable soundings as having been examined and verified.

1. Select the data that you want to flag as Examined.
2. Select the Examined command.

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

Query

View information on selected data.

1. Select the data that you want to query.
2. Select the Query command.

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

The data is displayed in the Selection tab.

Note: When you select data in the 3D View, all data in the highlighted area is selected, not just the visible data.

Designate Soundings

Shoal soundings over significant seabed features can be flagged as “Designated” to ensure that these depths are maintained in charts and other standard hydrographic products.

Use the Designate commands to flag, for example, the top of a mast in a wreck, or the shoalest sounding in a cluster of soundings identifying an outcrop of rocks.

For more information, see “Critical Soundings” on page 297.

1. Select the shoalest sounding from a cluster of soundings around a feature in Subset Editor.

Menu	Edit > Status Flag > Designate
Tool	
Key	<D>

2. Select the Designated Soundings command.

The sounding is flagged as Designated.

Shoalest soundings can also be designated in Swath Editor.

Find and Designate

The Find and Designate command automatically selects the shallowest sounding in cluster of highlighted soundings. This feature reduces the time needed to designate shallowest soundings.

1. Use the cursor to highlight soundings in Swath or Subset Editor.
2. [Optional] Use the Query command to view the soundings in the Selection tab.

Menu	Edit > Status Flag > Find and Designate
Tool	

3. Select the Find and Designate command.

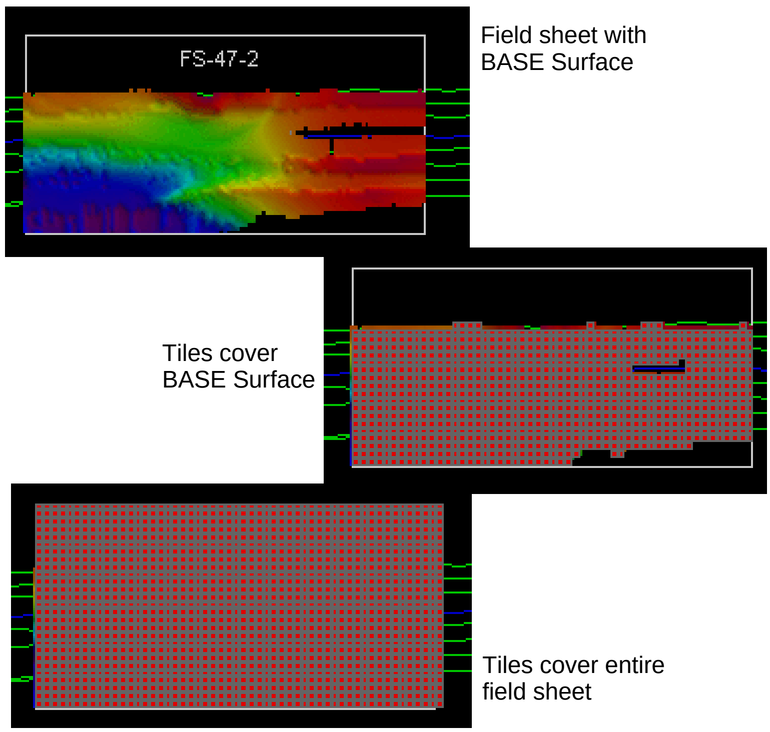
The shoalest sounding among the cluster of highlighted soundings is now flagged as Designated in the Selection tab, and displays the Designated symbol when viewed in Swath or Subset Editor.

Track Progress with Subset Tiles

Subset Tiles are used to track the process of examining and cleaning areas within the survey project. Each tile within a Subset Tile layer has one of three tracking settings: Incomplete, Partially Complete, or Completed (see “Tracking Cleaning Status” on page 349).

The extent of the subset tile area can be determined in two ways:

- the tiles cover the entire field sheet area, or
- the tiles cover the area of the field sheet that contains a height source, such as a BASE Surface.



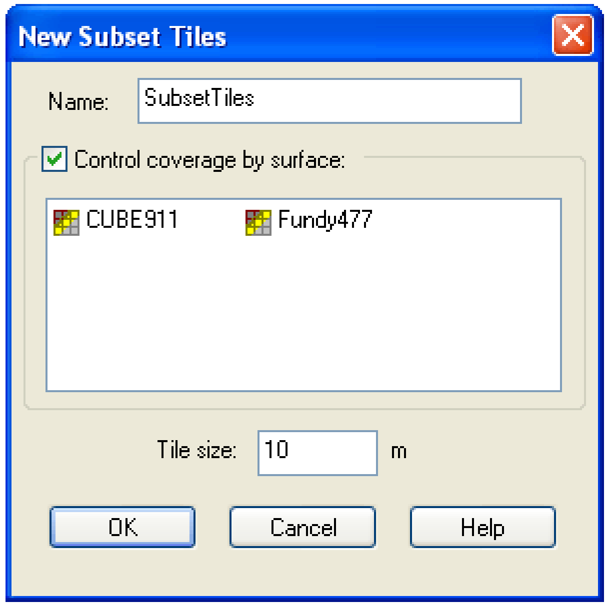
The option to cover the entire field sheet area is useful if you are cleaning newly acquired data with Subset Editor while the area is being surveyed. You would not have to create a new subset tile layer because of additional data in your project.

Subset tiles are produced in field sheets. To create subset tiles for tracking:

1. Select a field sheet in the Layers tab of the Control window.
2. Select the New Subset Tiles command.

Menu	Tools > New Subset Tiles
Pop-up	New Subset Tiles

The New Subset Tiles dialog box is displayed.



The *Control Coverage by Surface* option constrains the new subset tile area to the selected BASE Surface. When this check box is cleared, the subset tiles will cover the entire field sheet area.

To tile the entire field sheet,

3. Clear the *Control Coverage by Surface* check box.
4. Type the tile size.
5. Click **OK**.

A Subset Tile layer is created and displayed within the extent of the field sheet area.

To tile the area of a specific BASE surface,

1. Select the *Control Coverage by Surface* check box to apply the subset tiles to a BASE Surface.
2. From the list, select the BASE surface to be used as the template for subset tiles.
3. Type the tile size.
4. Click **OK**.

The Subset Tile layer name is added to the Layers tab. The names are automatically generated beginning with SubsetTiles1. You can show or hide the layer with the check box beside the Subset Tile file name in the Layers tab.

Subset tile properties

You can set display properties for tiles in the Properties window. For example, you can set colours to display the progress of subset cleaning, or make the tile layer transparent so the features below it are more visible.

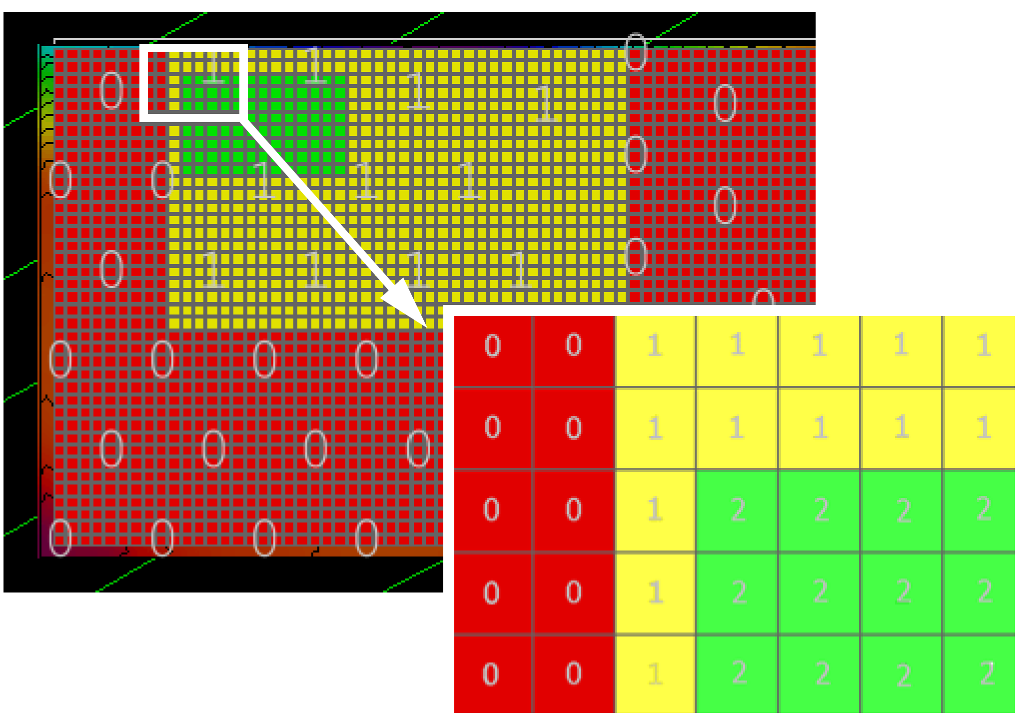
The *Active Attribute* field lists all attributes included in the tiles. When you select an attribute, such as cleaning status, tiles are displayed according to the values of those attributes.

For example, if Cleaning_Status is selected as the active attribute, and the Colour fill tiles is set to "True", the tiles will be displayed in

- red, if the tiles have not been examined. (status = Incomplete)
- yellow, if the tiles are partially complete. (The tiles were only partially covered by a subset that has been marked as complete.)
- green, if the cleaning of the tiles is complete.

As well, the numeric value for the attribute status can be displayed, using the *Display tile value* field.

When this field is set to "True", the tiles will display the value 0 for Incomplete, 1 for Partially complete or 2 for Completed. The example below shows tiles with "Cleaning status" set as the active attribute, and both *Colour fill tiles* and *Display tile value* set to "True".



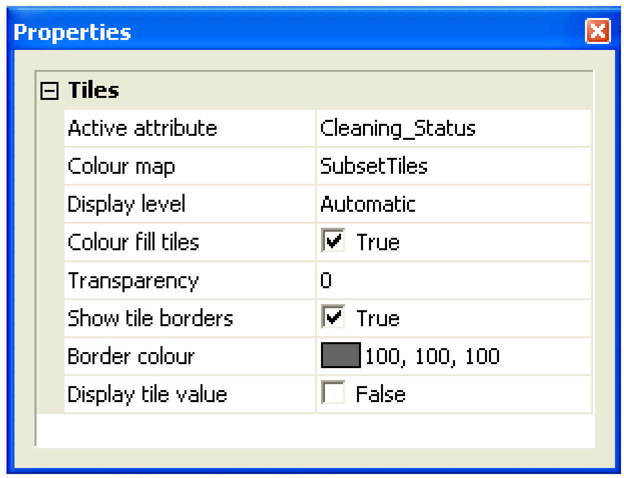
Menu	Window > Properties
Pop-up	Properties

These cleaning status values can also be viewed in the Selection tab of the Worksheet window when tiles are selected.

To set properties for the tiles layer,

1. Select a tile layer in the Layers tab of the Control window.
2. Select the Properties window command.

The Properties window shows the display options for tiles.



3. Select an attribute from the drop-down *Active Attribute* list.
4. Select a colour map from the *Colour Map* drop-down list.
5. Change how tiles are rendered to the Display Window by selecting the *Display Level*, either:
   - *Maximum*: Draw all tiles
   - *Automatic*: Draw tiles to fit the current extent of the Display window.

The *Colour Fill Tiles* check box determine how attribute values are drawn in the Display window.

- If the *Colour Fill* check box is set to “True”, the tiles are drawn according to the selected colour map.
- If the check box is unchecked and set to “False”, the tiles are drawn in outline only.

6. Select a *Transparency* percentage to make features visible through the tiles.

The *Show Tile Borders* check box determines how the border lines dividing the tile areas are displayed.

- If the *Show Tile Borders* check box is checked, the value is set to “True” and the lines are colour coded according to the colour selected in the *Border colour* field.
- If the *Show Tile Borders* check box is clear, the value is set to “False”, and the borders of the tiles are not displayed.

7. [Optional] Select an alternative colour for the border lines from the colour picker in the *Border colour* field.

8. Select the Display tile value check box to set the value to “True” so the appropriate cleaning status value (0=Incomplete, 1=Partially Complete and 2=Complete) is displayed in each tile.

Edit tiled subset data

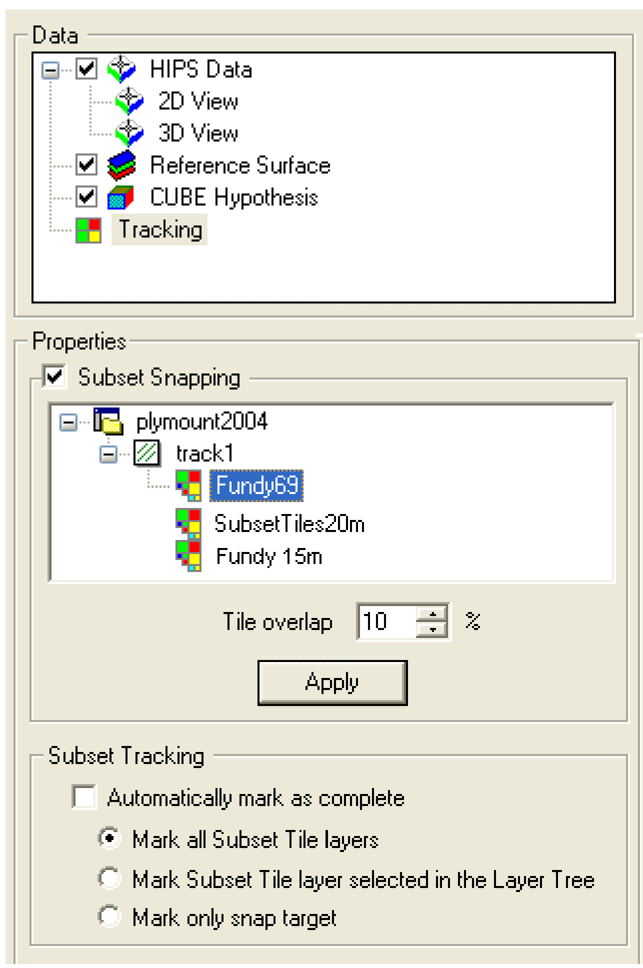
Once the tracking tiles are created, you can examine and edit the data by opening Subset Editor and defining a subset. (See “Open Subset Editor and Load Data” on page 111). You can clean data from the open subset (see “Subset Cleaning” on page 339). You can track the progress of cleaning by the colours of the tiles, or by selecting the tiles and viewing status in the Selection tab of the Worksheet window.

Tracking Layer Options

The layer containing the tracking tiles has options which can be set in the Subset Editor tab of the Control window (see “Subset Editor Tab” on page 115).

1. Select the Tracking layer in the Subset Editor tab.

The tab is refreshed to display the tracking options.



Use the Subset Snapping check box so that the subset bounding box encloses entire tiles.

2. Select the Subset Snapping check box.
3. Select the tile tracking layer.
4. Click **Apply**.

You can also set an amount by which the subset snapping will overlap tiles as you move from completed to unprocessed tiles.

This helps ensure coverage of all of the data that lies within tiles along the edges of the subset.

5. Select a percentage in the *Tile Overlap* field.
6. Click **Apply**.
7. In the Subset Tracking options, select the *Automatically mark as complete* check box to flag the data as examined and cleaned. This can be changed later (see “Tracking Cleaning Status” on page 349).

When the *Automatically mark as complete* check box is selected, two other options are made active.

8. Select the *Mark all Subset Tile layers* option to flag the tiles in ALL tile layers as completed, or select *Mark only snap target* to select tiles only in the selected tile layer as completed.

Tracking Cleaning Status

Once you have examined and cleaned a subset, the subset area can be assigned one of three classifications:

- **Complete**: All data in the selected area is clean and ready for further processing or export.
- **Partially Complete**: Not all the data within the area has been viewed and verified as clean in Subset Editor.
- **Reset**: Data has not been verified or cleaned.

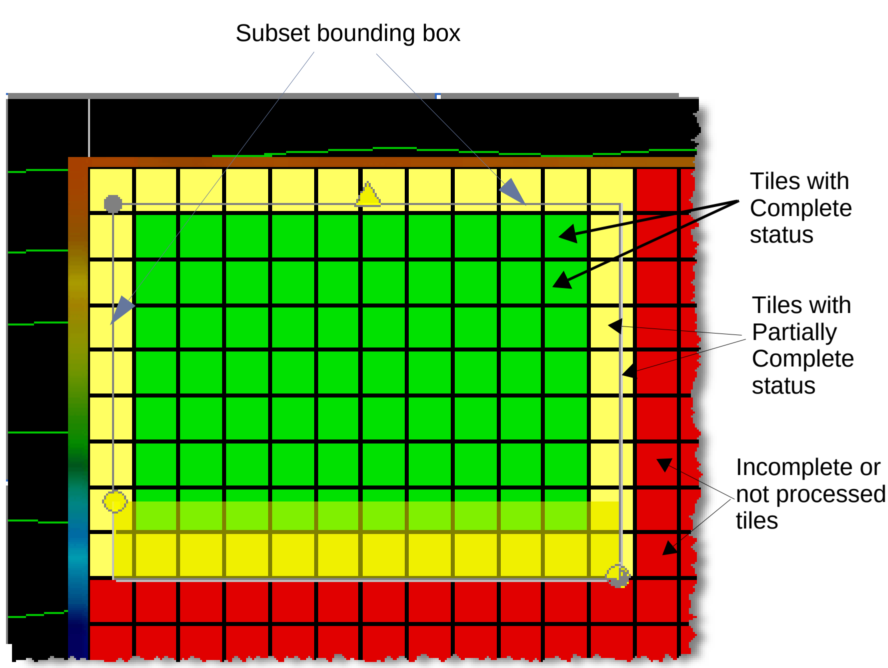
The area is colour-coded in the Subset Tile according to the selected cleaning status classification (and the settings in the Properties tab):

- Complete = green (2)
- Partially Complete = yellow (1)
- Incomplete or not processed = red (0).

To assign a cleaning status to a subset area,

1. Select the appropriate status from the Subset Editor toolbar or from the Tools >Subset Editor submenu.

The flagged tiles will be displayed with the colour appropriate to their status. Only tiles totally encompassed by the subset bounding box are flagged with the selected status. Tiles only partially covered by the bounding box are flagged as Partially Complete, as in the example below.



Subsets and BASE Surfaces

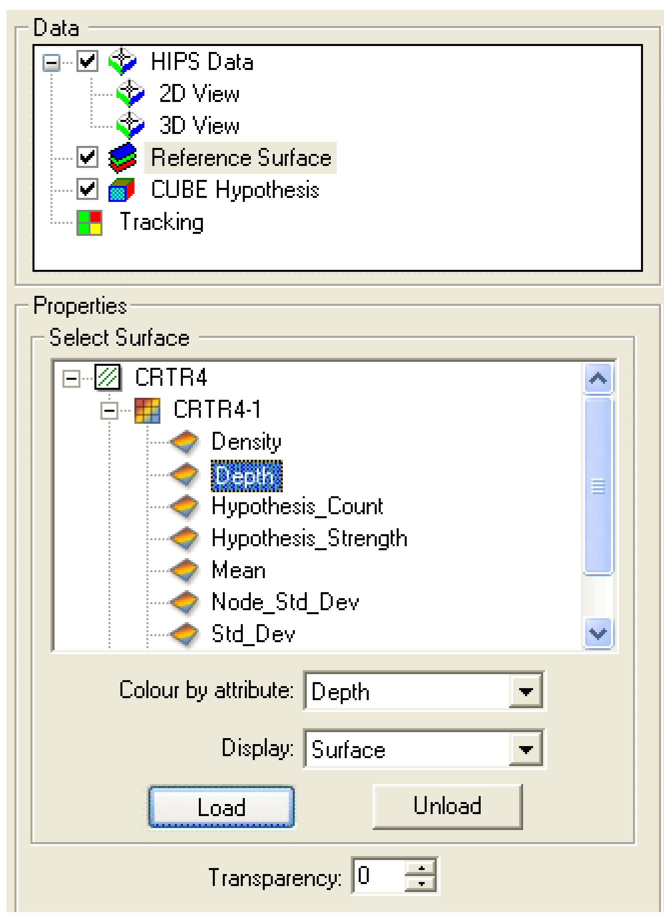
BASE surfaces can be opened in Subset Editor and displayed in the 2D and 3D Views. This enables you to determine the position of Designated soundings against an existing BASE Surface, and whether Designated soundings have been correctly applied to a Finalized BASE Surface. (See “Critical Soundings” on page 297.)

As well you can apply surface filtering to a subset of any BASE surface in Subset Editor.

.Open BASE surface in Subset Editor

1. Open the BASE surface for your project.
2. Open Subset Editor and define a subset over the BASE surface.
3. Load the subset.
4. Select the *Reference Surface* layer in the Subset Editor tab.

The Subset Editor tab is refreshed to show the property options for the Reference Surface layer.



5. Select a BASE Surface layer from the *Select Surface* file tree.

Although the Depth attribute is used to display the surface, you can colour the surface by another attribute by selecting that attribute from the *Colour by Attribute* drop-down list.

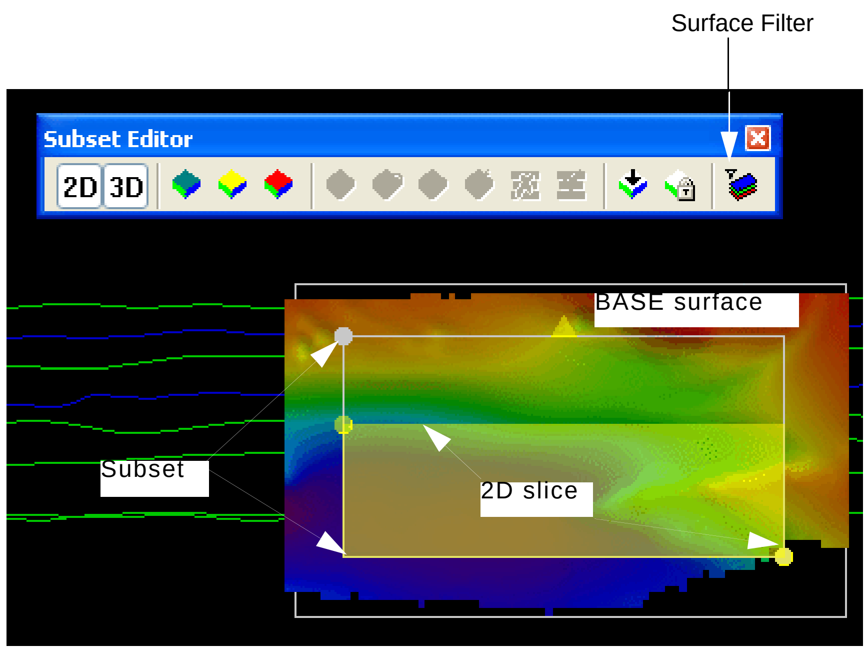
6. Display the BASE surface as points, wire frame or a complete surface by selecting an option from the *Display* drop-down list.
7. Select a transparency level for the Surface, using the up and down arrow buttons.
8. Click **Load**.

The BASE Surface is displayed in the Views according to the above options.

9. [Optional] Click **Unload** to remove the BASE Surface from the Views.

Surface filter

You can load any type of BASE surface to Subset Editor and apply filtering to the part of the surface within the subset slice, or to the part of the surface defined within the full subset.



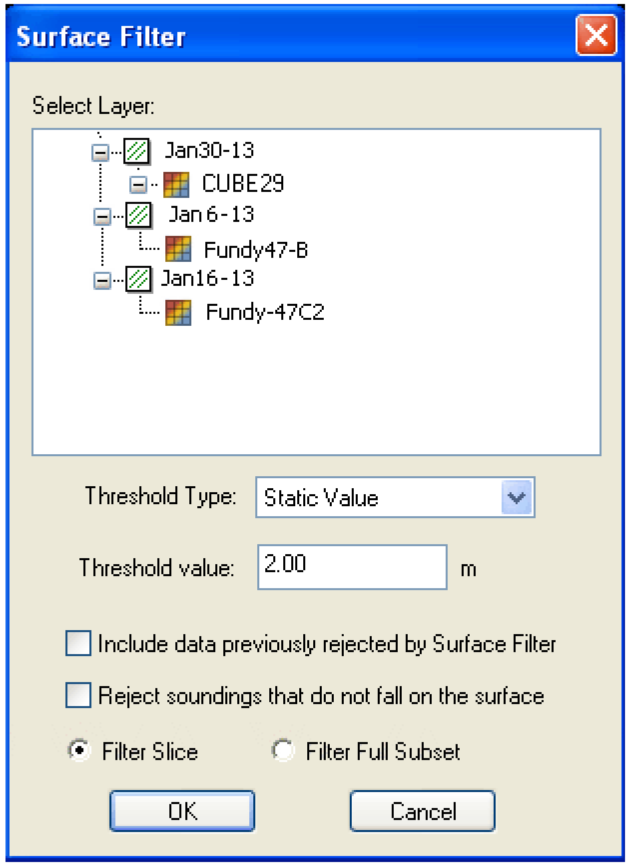
1. Open a BASE surface.
2. Open Subset Editor and define a subset over the BASE surface. (See “Open Subset Editor and Load Data” on page 111.)
3. Load the subset.
4. Select the *Reference Surface* layer in the Subset Editor tab.
5. Select the BASE Surface Depth layer from the *Select Surface* file tree in the Properties.
6. Click **Load**.

When the surface is loaded, the Surface Filter button on the Subset Editor toolbar is activated.

Menu	Tools > Subset Editor > Surface Filter
Tool	 Subset Editor toolbar

7. Select the Surface Filter command.

The Surface Filter dialog box is displayed.



Once you select the surface in Reference Surface, it is selected in the Filter dialog box and cannot be changed.

8. Select the *Threshold Type* from the drop-down list:
   - Standard Deviation
   - Uncertainty
   - Greater of the two
   - Lesser of the two.
   - Static Value
9. Set the desired *Threshold value.*
   - For standard deviation or uncertainty thresholds, the value entered will show the confidence interval next to the field.
   - Static values are entered in units set in Tools > Options > Display > Units > Vertical units, e.g., metres.
10. Select the *Include data previously rejected* ... check box to include rejected data when running the filter.
11. Select the *Reject soundings that do not fall on the surface* check box to reject soundings that are offset from the surface.
12. Select Filter Slice to filter only the soundings within the subset slice, or
13. Select Filter Full Subset to filter all the soundings within the subset bounding box.

14. Click **OK** to apply the filter.

For description of applying surface filtering to an entire BASE surface, see "SURFACE FILTERING" ON PAGE 244.

16

Create Product Surfaces

Create product surfaces from finalized BASE surfaces. Product surfaces maintain the designated soundings from the finalized BASE Surface and can be used to create more cartographically correct contours. Product surfaces can also be created from tile layers.

Use product surfaces for building contours, sounding selections, profiles, and other data layers.

In this chapter...

Product Surfaces 356
Contours 361
Profiles 367
Sounding Selection 383
Colour Options for Soundings and Contours 394
Tiling a field sheet 396

Product Surfaces

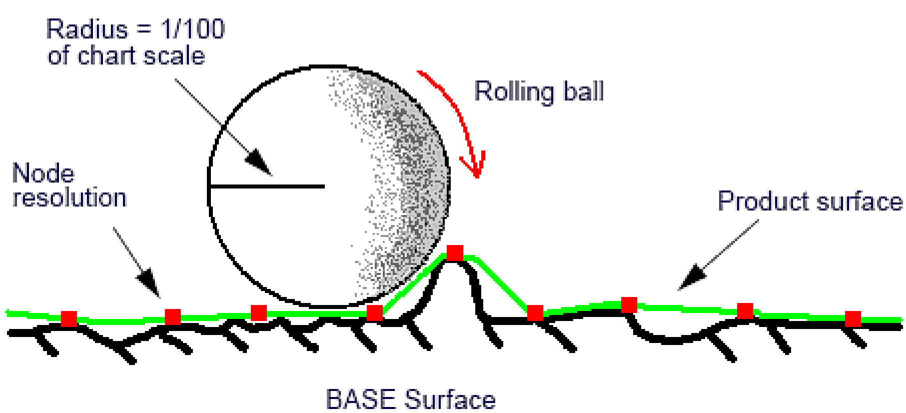
A Product Surface is a shoal-biased surface that can be used for creating more cartographically correct contours for navigation aids (for example, Electronic Navigational Charts).

The Product Surface is created from a finalized BASE surface, t is down-sampled so that some of the finer surface details are no longer visible, while the shoals are preserved. This surface can be used to directly generate contours and selected soundings (instead of generating contours or selected soundings from tiles).

Process

A Product Surface is created through the process of 3D Double Buffering. This process smooths the surface to reduce clutter and highlight significant shoals.

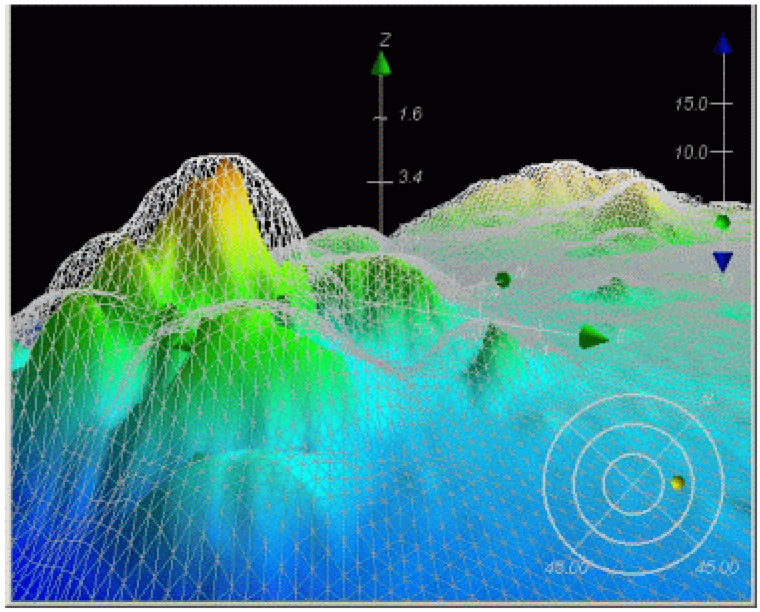
Double buffering is like rolling a ball over the surface at an interval determined by the surface's node resolution. The radius of the ball is determined from the chart scale (radius = 1/100 of chart scale). As the ball is rolled over the surface, the surface is smoothed, but the shoals are retained.



Defocusing

This optional process can also be applied to the generalization process. Defocusing spreads each shoal depth over an area defined by the horizontal error. The process uses an ellipsoid of rotation around each node with the horizontal error as the semi-major axis. Nodes within the ellipse are adjusted up to the surface of the ellipse.

In the following image, the webbed surface overlaying the BASE Surface represents the Product Surface area after defocusing has taken place. The shoals are preserved, but without the sharply defined detail of the original BASE surface.

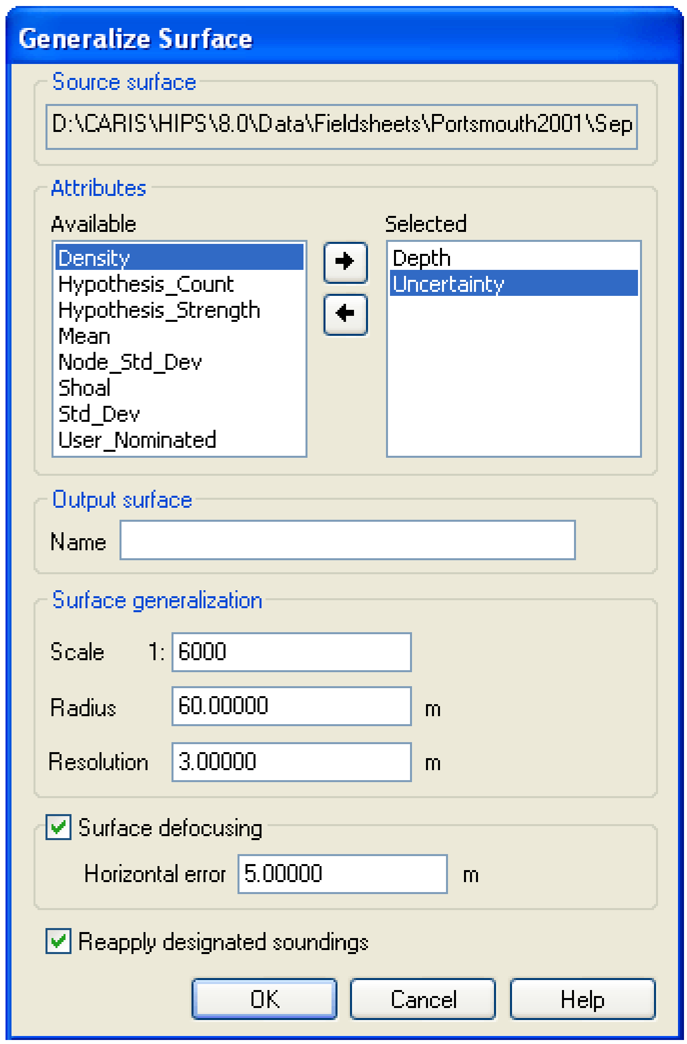


Create a Product Surface

Menu	Process > BASE Surface > Product Surface
Pop-up	Product Surface

1. To create a Product Surface from a finalized BASE surface: Open the BASE surface.
2. Select the BASE Surface layer in the Layers tab.
3. Select the New Product Surface command.

The Generalize Surface dialog box is displayed.



The path and filename of the selected BASE surface is displayed at the top of the dialog box.

4. [Optional] Select one or more attribute layers to add to the Depth layer in your product surface.
5. Type a name for the new product surface in the *Output* field. (The file is saved to the same field sheet directory as the BASE surface.)
6. Type the scale ratio that best suits the type of chart.

The scale of the Surface should match the scale of the product being created. For example, if you are creating a Product Surface that is going to be used for an ENC approach to a harbour, then use the appropriate scale for an approach.

The scale determines the radius used for generalizing the contours of the Surface. The *Radius* value changes as the *Scale* value is changed.

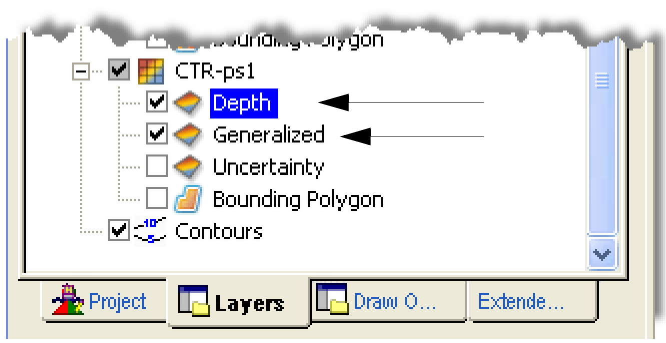
The *Resolution* value sets the node spacing of the generated Product Surface.

7. Type a *Resolution* value, if needed.
8. [Optional] Select *Surface Defocusing* to implement this option.

The defocusing operation requires you to apply a horizontal error value. This value must be derived from the errors values associated with the data.

9. Type a *Horizontal Error* value.
10. Select the *Reapply Designated Soundings* check box to make sure that designated soundings are applied to the surface.
11. Click **OK**.

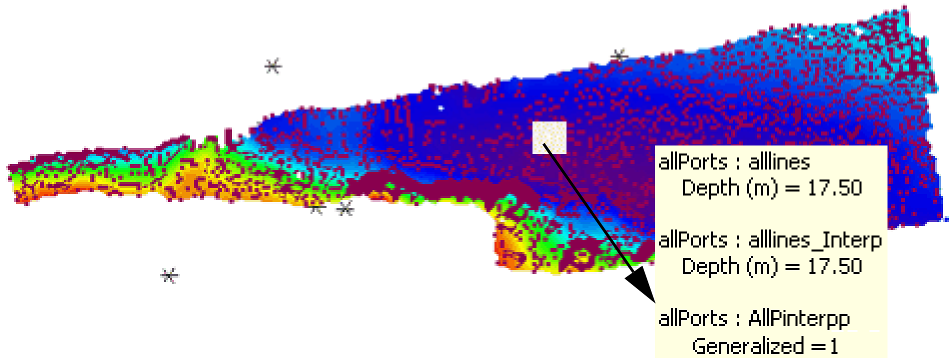
When the Product Surface is created, it is listed in the Layers tab and shown in the Display window.



Also created is a Generalized layer which displays the nodes that were affected during the product surface-creation process.

- Nodes that have been modified are given a status flag of 1.
- Nodes that retain their original values are given a status flag of 0.

The status of the nodes is displayed in the tool tip, as illustrated below, which shows Generalized = 1.



In this example, the filter in the Properties for the Generalization layer has been set to show only the nodes which have been modified by the Product Surface. These are displayed as red dots in the example above.

When you create selected soundings from the Generalization layer, you can use SQL-type filters to omit nodes that have been modified. See "Enable Filter" on page 390.

Contours

Contours can be generated from a Product Surface, a finalized BASE surface or a tile layer.

Contour smoothness is related to the amount of detail in your depth source. A finely detailed depth source will result in rougher contour lines. For smoother contour lines, use a smoothed Product Surface as a source for contouring.

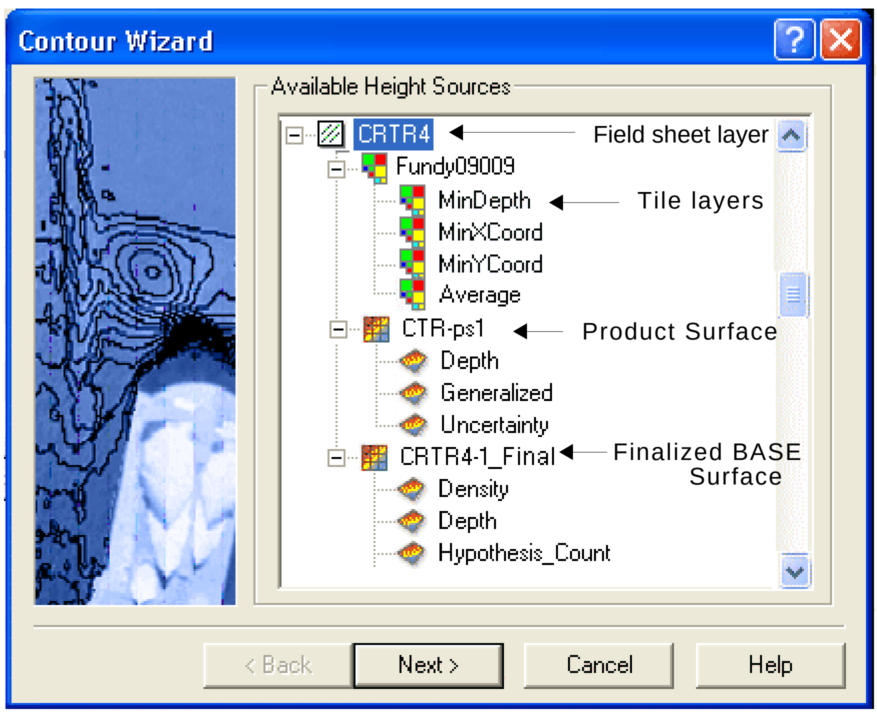
You can label contours with depth values, adjust the size and colour of contours, and mask the lines behind the contour label.

Create Contours

1. Open a project and a field sheet containing a BASE surface.
2. Select the field sheet layer in the Layers tab.
3. Select the New Contour Layer command.

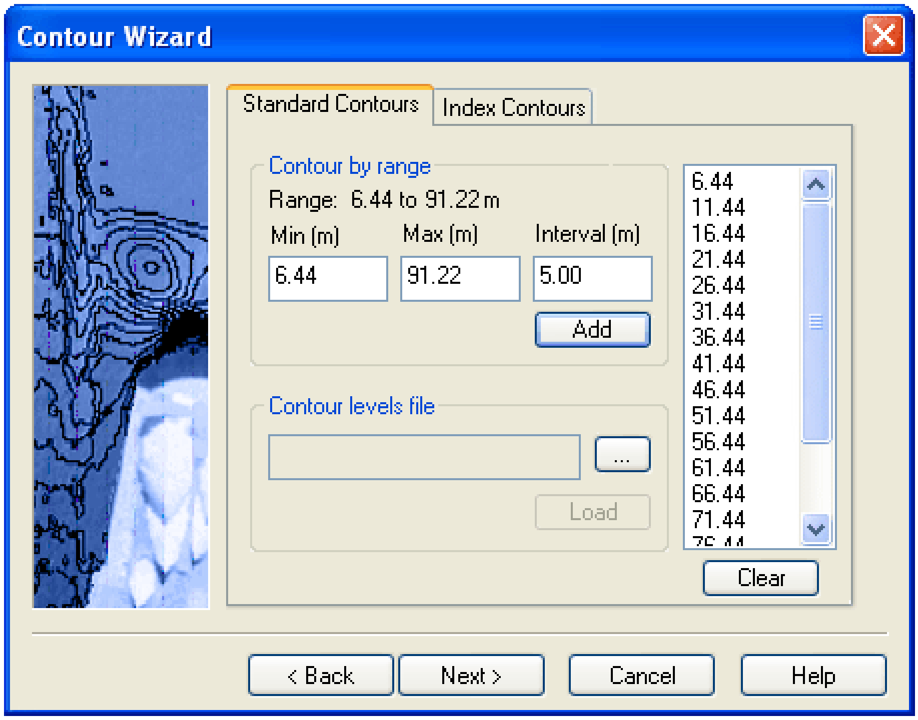
Menu	Process> Products > Contours > New Contour Layer
Pop-up	(Layers tab) New > Contours…

The Contour Wizard - Step 1 dialog box is displayed.



4. Expand the layer tree and select a tile, Product Surface or BASE Surface attribute layer to be the height source for the contours.
5. Click **Next**.

The Contour Wizard - Step 2 dialog box is displayed.



At this step you enter the intervals for standard and index contours. You can create contours at regular intervals, or you can load a file listing specific depths at which to create the contours.

The dialog box displays the *Range* of depth values derived from the contour source (BASE Surface, bin or tile). You can contour over this entire range, or limit contouring to a specific range of values.

The depth intervals you set are saved in the project field sheet directory as ContourLayerName_DepthList.txt.

1. [Optional] Type the *Minimum* depth level for the contour interval.
2. [Optional] Type the *Maximum* depth level for the contour interval.
3. Type an *Interval* value to set the distance between contour lines.
4. Click **Add**.

The contour depths are listed. As you change the *Interval* value, the number of contour depths will vary accordingly.

5. If you are satisfied with the values, click **Next**.

Contour levels file

Contour levels files, or depth list files, are text files that set a range of depths for tiling and contouring data, where values are:

- negative when representing heights above the datum, and
- positive when representing depths below the datum.

These files are automatically generated during the contouring process. You can use a contour levels file to set the values for contours.

6. Click the browse button and select a contour levels file.
7. Click **Load**.

The contour depths are listed. You can edit these values using the commands on the pop-up menu.

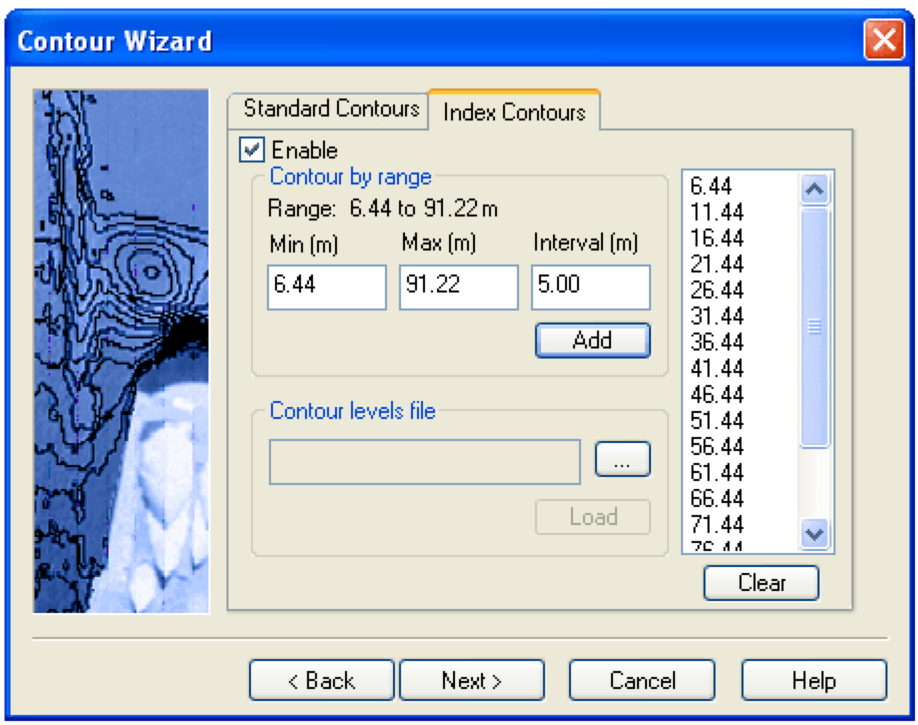
8. Right-click in the depth values list, and select a command.
   - Insert: manually add a specific depth value. It will be added to the list in order.
   - Delete: deletes the selected value. You can select multiple values.
   - Save: saves the values as a contour levels text file.
   - Clear: removes all the values from the list. (Clicking **Clear** will do this also.)

Index Contours

Index contours visually emphasize certain contour levels, for example, every 5th contour. These index contours are optional.

To create index contours:

1. Select the Index Contours tab.



2. Select the *Enable* check box.

This activates depth list fields identical to those in the Standard Contours tab. Follow the same process as for standard contours, however, enter a larger interval value.

- For example, to index every 5th contour where contours are every 5m, set the interval for indexing to 25m.

3. Click **Next**.

The Contour Wizard - Step 3 dialog box is displayed.

You can save the contours as a field sheet layer or save the contours to a new or existing HOB file.

1. Select either the *Contour to Field Sheet* option or the *Contour to HOB File* option. (See "Contour to HOB file" on page 364.)

Contour to field sheet

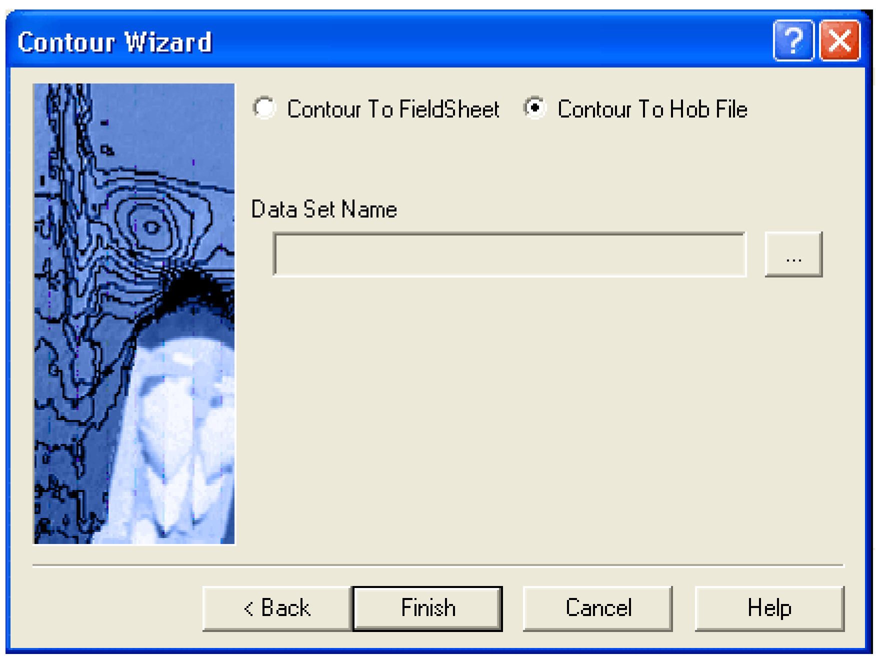
The next step is to describe the various CARIS attributes for the new contours.

2. Enter the following information for standard and index contours:
   - Name
   - Theme Number
   - Feature Code
3. Click **Finish**.

Contours are created for the selected field sheet and shown in the Display window. The standard and index contours are listed in the Control window.

Contour to HOB file

If you selected the *Contour to HOB* option, the dialog box is refreshed to display the Contour Wizard - Step 3 (HOB) dialog box.



1. Click **Browse** [...] and select an existing HOB file, or type the path and name for the file. You can also enter the name of a new file.
2. Click **Finish**.

The contours layer has now been added to the existing HOB file, or a new file with contours has been created. The contours can be viewed in the Display window by choosing the File Open command and selecting the HOB file.

Contour Labelling

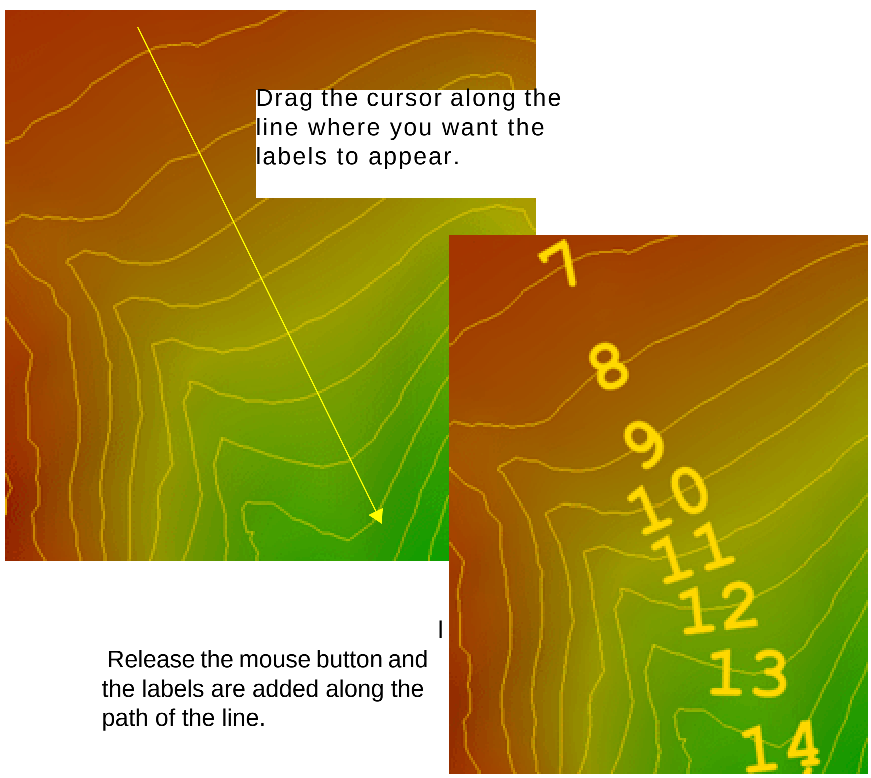
You can label contour lines with their associated depth values. You set the path for the labels by drawing a line across the contours where you want the depth values to appear. Labels are added at the point where the line intersects the contours.

Menu	Process > Products Contours > Add Labels
Pop-up	(Layers tab) Add Labels

1. Select the Contours layer in the Layers tab of the Control window.
2. Select the Add Labels command.

Your cursor changes to a digitizing tool so that you can set the path for the labels.

3. In the Display window, click on the part of the field sheet where you want to start the line and drag the cursor to the part of field sheet where you want the line to end.



Labels are now added to the contours.

You can change the size and colour of contour labels. To set the size of the labels in mm,

1. Select the contour layer.
2. Select Set Label Size from the pop-up menu.

3. Type the new value in mm.
4. Click **OK**.

The colour of the labels can be changed from the default (black) by checking the *Override Colour* option in the Properties window and selecting a colour from the drop-down menu.

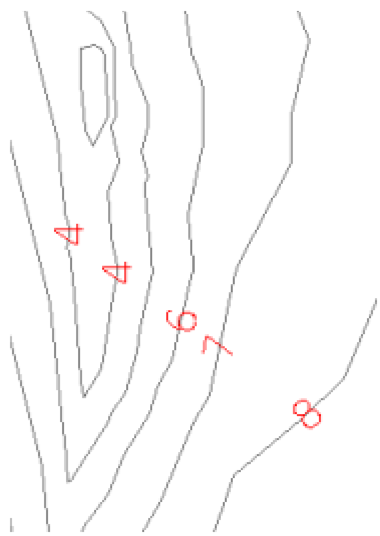
Labels can also be removed from the display.

Menu	Process > Product Contours > Remove Labels
Pop-up	(Control window) Remove Labels

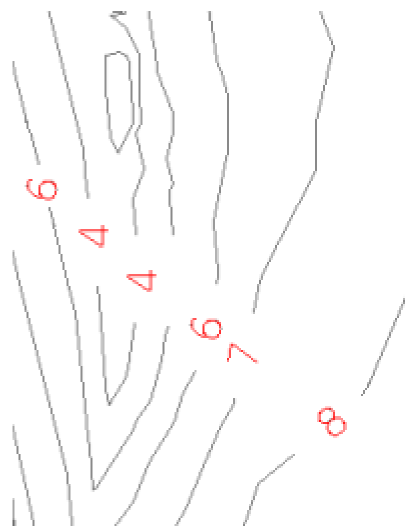
1. Select the Remove Labels command.

Masking Lines Behind Contour Labels

Concealing the part of the contour line behind a label can make a map area easier to read.



Contours with labels



Contours lines masked behind labels

To mask the area behind contour labels:

Menu	Process > Product Contours > Mask Contours
Pop-up	(Control window) Mask Contours

1. Make sure all previous work is saved.
2. Select the Contour Masking command.

The parts of the contour lines that intersect the labels are now hidden.

This masking process can be reversed with the Unmask contours command.

Profiles

Use the Profile command to draw a cross-sectional view of a height source, such as a BASE Surface attribute layer.

Profile graphs are created from a line digitized on a surface, either manually drawn or along a superselected trackline.

A profile is graphed in a dockable window, and can be viewed in real time as the profile is digitized or edited. Profile lines can be edited, and exported as an image or to an ASCII file.

Profiles are saved as field sheet layers.

Digitize a Profile

Creating a profile by digitizing a line involves these basic steps:

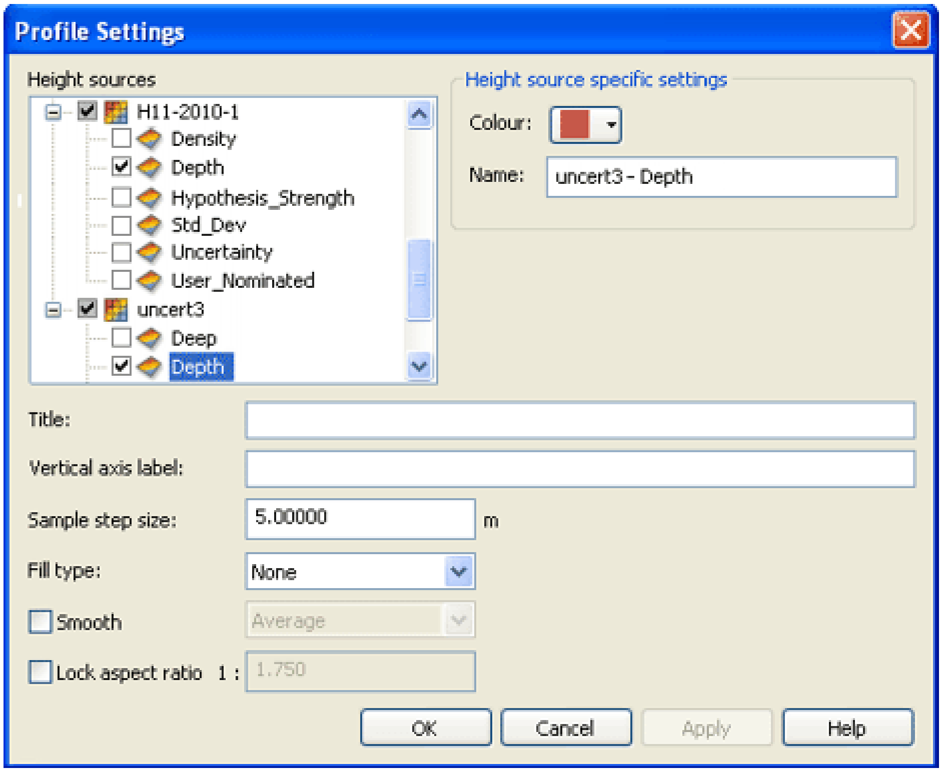
- select the field sheet containing the surface
- select layers to profile
- [Optional] set options for the graph display
- digitize the line across the surface

To create a profile:

1. Select the field sheet layer containing the BASE surface to be profiled.
2. Select the Profile by Digitizing command.

Menu	Process > Products >Profile > By Digitizing
Pop-up	New >Profile

The Profile Settings dialog box is displayed.



In this dialog box you can set all the options for the profile, or, you can select one or more height sources, generate the profile, then re-open this dialog box to adjust settings while viewing the effect in the Profile window.

3. Select the height source check box for the layer(s) you want to profile.

You can select more than one layer as height sources for the profile. Each source that you select will generate its own profile line in the graph. Lines with the same values will overlap.

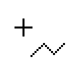
4. [Optional] Set a *Sample step size*. Default value is 5m.

5. Set other options as desired. (See "SETTING PROFILE OPTIONS" ON PAGE 370.)

6. Click **OK**.

The Profile window opens and displays the message "Empty profile line".

Digitize the profile line:

The cursor changes to digitizing mode. 

7. Click once on the surface to fix the anchor point for your profile line.

8. Click to add a series of points to define the line.

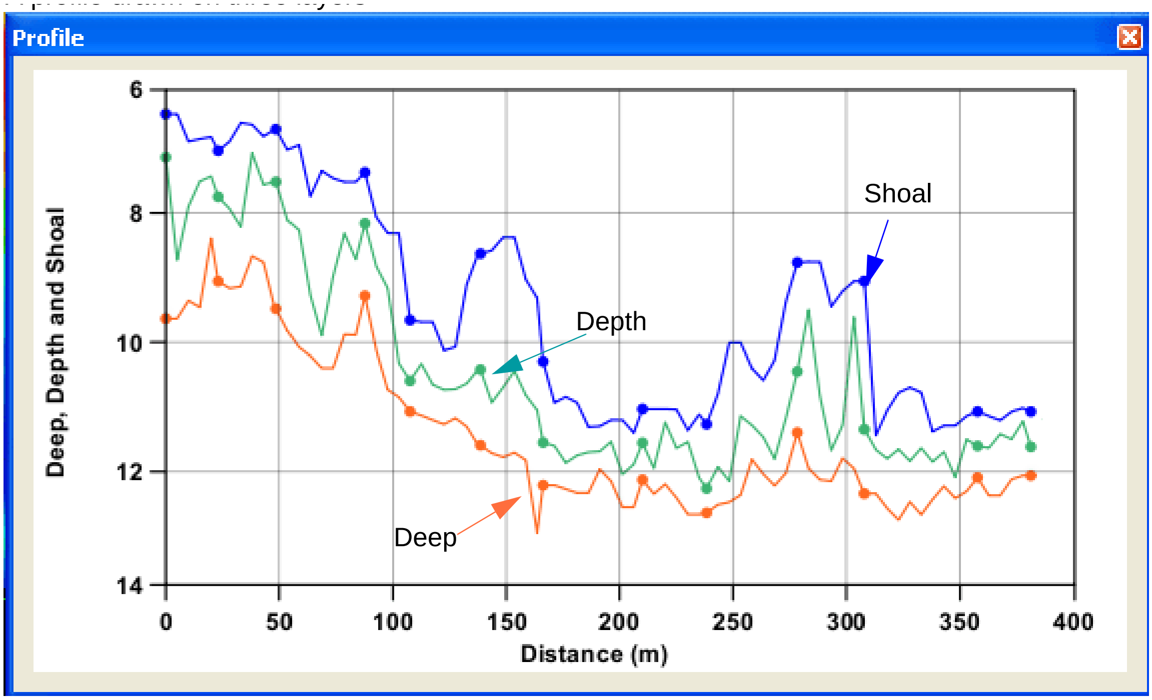
The profile line is automatically drawn between the points. The Profile window dynamically displays the profile as you add more points to the digitized line.

9. To remove points before the line is completed, right-click and select Remove Last from the pop-up menu. Repeat as needed. (Points can

also be removed from a completed line in Edit mode. (See "LINE EDITING OPTIONS" ON PAGE 376.)

10. Click the **Enter** key to complete the line, or right-click on the line and select End line from the pop-up menu.

The Profile is displayed in the Profile window. The example below shows the profiles on different height sources, generated by the same digitized line. The dots in the graph represent the digitized points of the line.



Create Profile by Superselection

You can create a profile that follows a superselected line, such as a track line. The line must fall within the bounds of a field sheet.

To create a profile for superselected track line:

1. Open the field sheet layer containing the BASE surface to be profiled.
2. Select the ShipTrack Lines layer in the Layers tab.
3. Select a track line.
4. Select the Profile by Superselection command.

Menu	Process > Products >Profile > Superselection

5. In the Profile Settings dialog box, select the height source check box for the layer(s) you want to profile.

You can select more than one height source for the profile. Each layer that you select will have its own profile line in the graph. Lines with the same values will overlap.

6. [Optional] Set a *Sample step size*. Default value is 5m.

7. Set other options as desired. (You can also generate the profile and then re-open this dialog box to adjust settings.)
8. Click **OK**.

The Profile is displayed in the Profile window.

Setting Profile Options

Profile options are set in the Profile Settings dialog box. You can set options here before you create your line, or use this dialog box to adjust or edit the settings for an existing profile.

To open the Profile Settings dialog box:

1. Select the Profile layer in the Layers tab.
2. Select the profile line in the Display window.
3. Right-click in the Profile window, and select Settings from the pop-up menu.

The Profile settings box is displayed.

- If you are setting options for a new profile, set the options and click **OK**, and draw the line.
- If you are changing or setting options to an already digitized profile, click **Apply** to dynamically see the effect of the option in the Profile window.

Profile colour

Use the *Colour* field to assign a different colour to each line graphed in the Profile window.

1. Highlight a layer in the *Height sources* list.

The *Name* field is automatically populated with the name of the surface and the layer you selected (e.g., SurfaceName - LayerName). You can edit this name.

2. [Optional] Replace the automatically generated name.
3. Select a colour from the colour picker. (Default colour is black.)
4. Repeat to assign a different colour for each profile.

Graph labels

5. Enter a *Title* for the profile graph.

The vertical axis in the graph shows the attribute values from the surfaces. The horizontal axis is always distance along the profile.

6. Enter a *Vertical axis label* for the profile graph.

Sample step size

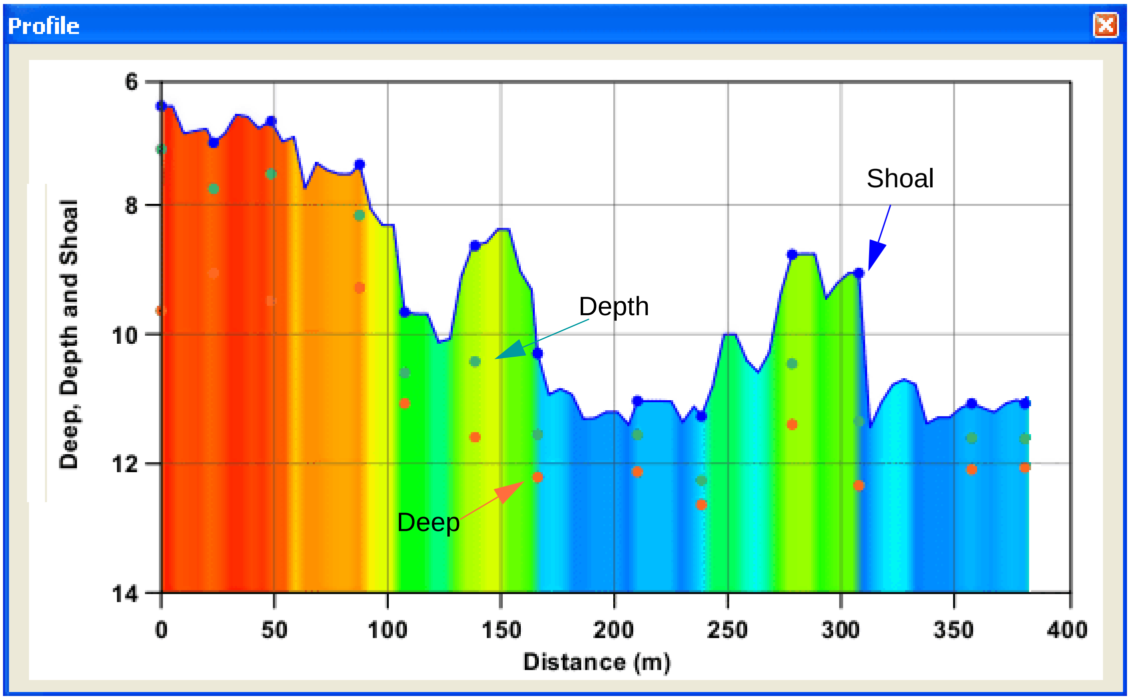
The *Sample step size* field controls the frequency with which the surface is sampled. HIPS samples the data at the specified interval and uses it to draw the profile line between the digitized points. The smaller the step size, the more detail in the profile.

7. Enter a value for *Sample step size*.

Fill

The *Fill type* option allows you to colour the profile according to the elevations in the data. You have the option of applying the colours horizontally or vertically.

This is an example of a profile with vertical fill.



8. Select an option from the **Fill type** drop-down list.

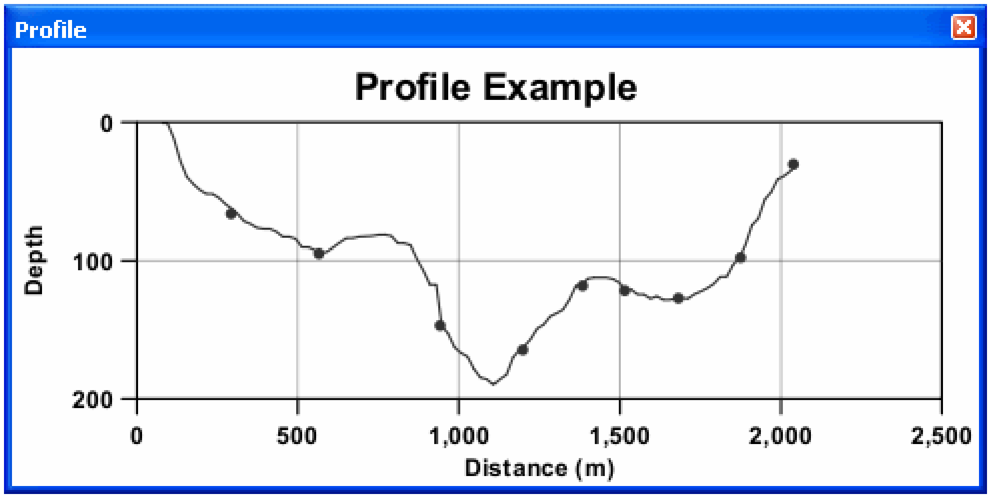
Smooth

The *Smooth* option allows you to smooth the profile line in the profile graph. Smoothing is applied reduce the number of peaks in the graph if it has a high number of sample points. The number of sample points is based on the *Sample step size* setting. There are three types of smoothing to choose from:

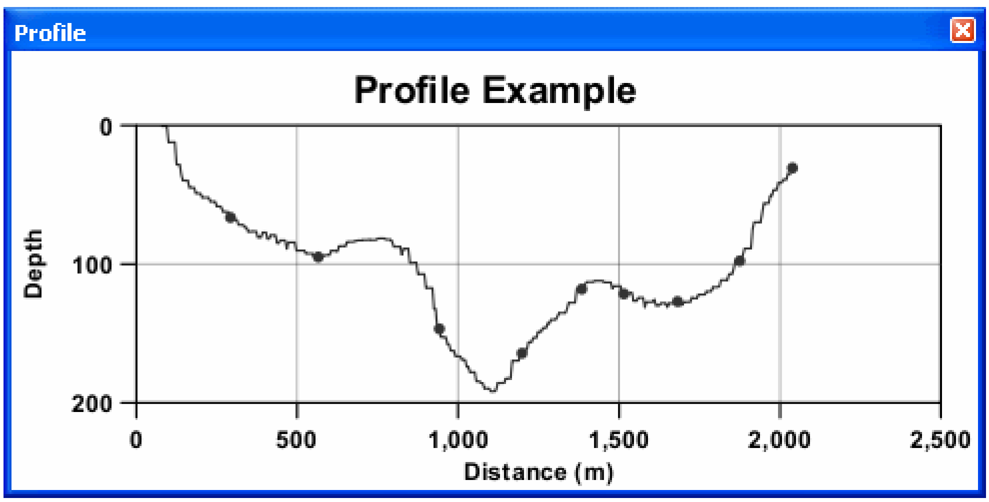
- *Average*: This method will create the graph using the average elevation values within each sample distance.
- *Shoal*: This method will create the graph using the shoalest values within each sample distance.
- *High/Low*: This method will create two lines in the graph; one for the minimum values in each sample distance and one for the maximum values in each sample distance.

Below is an example of a profile with and without shoal smoothing applied.

With Shoal Smoothing



Without Smoothing



9. Click the check box to enable the **Smooth** option.
10. Select a smoothing type from the drop-down list.

Optionally, you could apply smoothing to the digitized line. See "Edit the Profile line" on page 375.

Lock aspect ratio

Use *Lock aspect ratio* option to maintain the ratio of the width of the profile graph to its height, if the Profile window is re-sized.

11. Click the check box to enable the **Lock aspect ratio** option.

The current ratio of the graph will be displayed in the aspect ratio field.

12. [Optional] Enter a new ratio value for the graph.

> The settings for Vertical axis label, Fill type, Smooth and Lock aspect ratio will be remembered the next time the dialog box is opened.

Resize Profile window

The size of the Profile window can be changed to provide a better view of the profile. As the window is re-sized, the view is automatically scaled and the axes values updated dynamically.

- The vertical axis represents the height source of the profile and can be assigned a name in the Profile Settings dialog box.
- The horizontal axis represents the length of the line feature being used to generate the profile. The unit of measure for the values on this axis are controlled by the *Horizontal Length* setting in Tools > Options > Display > Units.

For information on re-sizing, see "Display, Reposition and Resize Windows" on page 25 of the HIPS and SIPS Reference.

Import Profile

Profiles saved in a field sheet in earlier versions of HIPS and SIPS may be listed in the Layers tree when the field sheet containing the surface is opened.

These profiles will be listed by name and can be viewed in the Profile window.

1. Right-click on the named layer, and select Display from the pop-up menu.
2. In the Profile Settings dialog box, select a height source (and other options as desired).
3. Click **OK**.

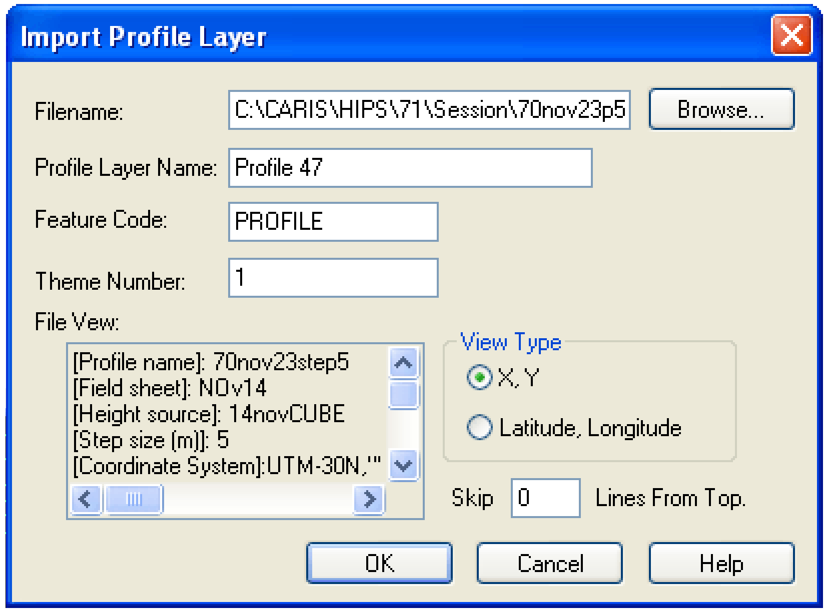
The profile will be seen in the Profile window.

To import a profile in ASCII format:

1. Select the field sheet layer containing the surface to which you want to import the profile.
2. Select the Import Profile command.

Menu	Process > Products >Profile > Import

The Import Profile Layer dialog box is displayed.



3. Click **Browse** to select the ASCII file containing the profile to import.
4. Type a *Profile Layer Name*. The default name is Profile1.
5. Type a *Theme Number*. The default is 1.
6. Select the View Type: X,Y or Latitude, Longitude. This value should be the same as the type set when the profile was exported to the ASCII file.

The *File View* field shows the contents of the ASCII file, which you can scroll through. To skip extraneous content in the file, you can set where the data begins.

7. Set a value for the number of lines from the top that should be disregarded when the file is imported.
8. Click **OK**.

The imported profile will be listed as named under the field sheet in the Layers tab. It can now be viewed, edited and exported as any other profile.

Edit the Profile line

Profile lines are saved with the field sheet in the Profile layers. Edits which can be done on a profile line include:

- add, remove or move points to change the path of the line
- move the entire line to see the profile on a different part of the surface
- rotate the entire line around a selected point

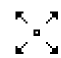
To edit a profile:

1. Select the child Profiles layer in the Layers tab.
2. Select the profile line you want to edit in the Display window.
3. Select the Edit profile command.

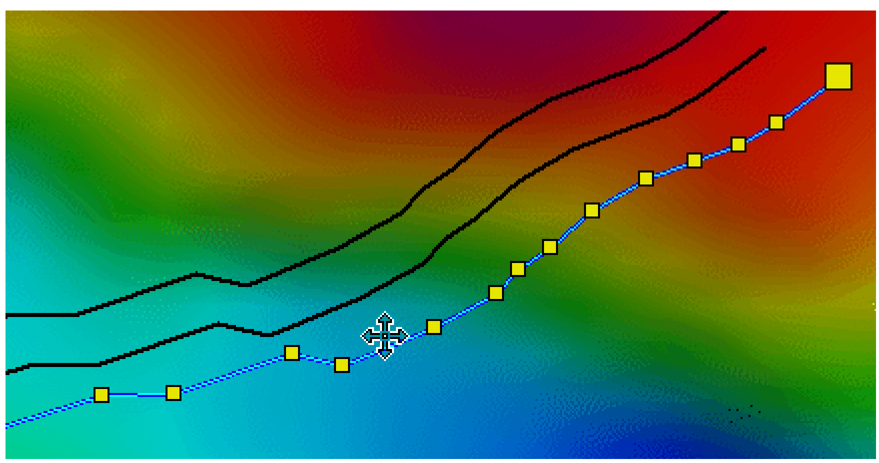
Menu	Process > Products >Profile > Edit

The profile line is placed in edit mode. Square vertex markers show the points which digitized the line.

If you hover your cursor over the line, the cursor will change shape to indicate the edit function that you can perform.

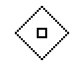
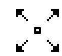
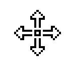
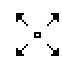
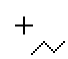
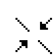
When the cursor is over a point it looks like this: 

In the image below the cursor shape indicates it is ready to edit the line, for example, to drag the entire profile line to another position in the Display.



(The black lines are the previous positions of the profile line. When the Display is refreshed, these lines will disappear.)

Line Editing Options

Edit Function	Procedure	Cursor changes to:
Add a point to the profile line	1. Select the profile line. 2. Press and hold the <Ctrl> key. 3. Move the cursor to the place on the where you want to add a point. 4. Click once.	
Reposition a a point on the line	1. Select the profile line. 2. Click on the point you want to move. (The cursor changes shape and the selected point square turns red.) 3. Drag the point to its new location.	
Move an entire profile line	1. Select the profile line. 2. Move the cursor over the line until the cursor changes shape. 3. Drag the line to a new location.	
Rotate a profile line around a selected point	1. Select the line 2. Select a point on one of the lines. (The point turns red.) 3. Hold down the <Shift> key. 4. Move the cursor over the line until the cursor changes shape. 5. Click then drag to rotate the line around the selected point.	
Delete a point	1. Select the profile line. 2. Select a point. (Selected point square will turn red.) 3. [Optional] Press <Ctrl> and click on other points to select them. 4. Press <Delete>, or right-click and select Delete from the pop-up menu.	
Redigitize parts of the profile line	1. Select the line. 2. Select a point at either end of the line or on a specific point within the line. 3. Right-click on the line and select Redigitize from the Edit Line pop-up menu. (The cursor changes to the line digitizing shape.) 4. Click to a point, then continue to digitize.	
Shorten or extend a line	1. Select the line. 2. Right-click on the line and select Trim/Extend from the pop-up menu. To trim a line, do one of the following: •Click on the line at the point you want to shorten it to, OR • Click on the end point and drag it back along the line to the desired length. To extend a line, drag an end point out to a new location.	

Remove Profiles

Profile lines in the Profiles layer can be removed from the layer if necessary.

To remove a profile line:

1. Select the **Profiles** layer in the Layers window.
2. Select the profile line you want to remove in the Display window.
3. Select the **Remove Profile** command.

Menu	Process > Products > Profile > Remove

The profile line is removed from the display and the profile graph is removed from the Profile window.

Export Profiles

To save profile information to use in another project, you can export the profile to an ASCII file or as an image to various formats.

See "Export Profile to ASCII" on page 377 and "Export Profile to Image" on page 380 for more information.

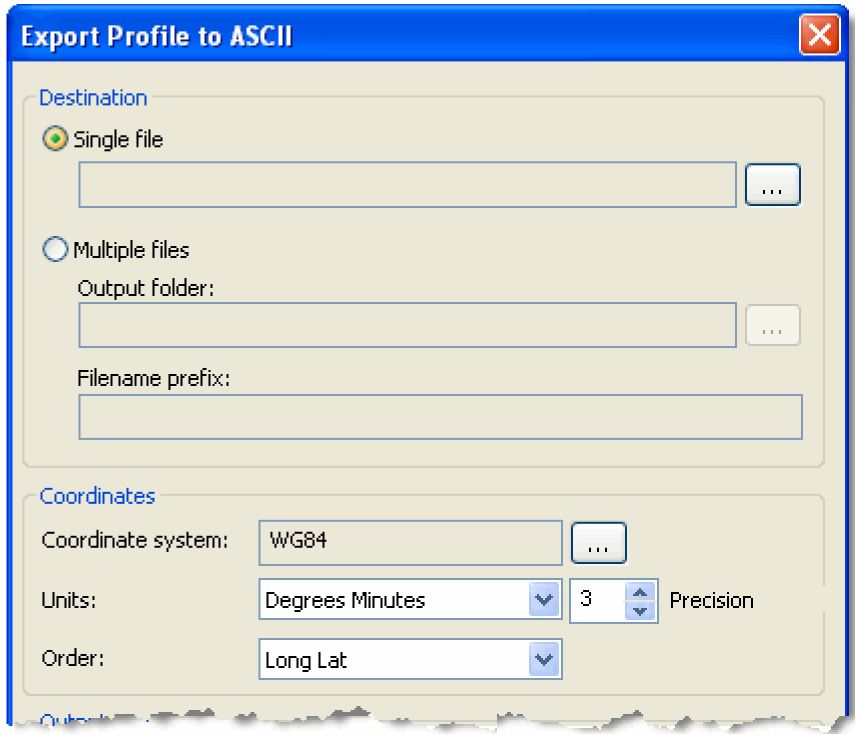
Export Profile to ASCII

Use the ASCII Export option in the Profile window pop-up menu to export profile lines to ASCII files.

To export to ASCII:

1. Right-click the Profile graph in the Profile window.
2. Select the **ASCII Export** command.

The Export Profile to ASCII dialog box is displayed.



Destination

If you have created a profile line on multiple layers, you have the option of exporting the profile as one file or to one file for each layer.

To export to a single file:

3. Select *Single File* and click **Browse** to select the destination folder.
4. Type a name for the file.

When you export to multiple files, the ASCII files will be named with a combination of the surface name and layer name. You can add a prefix to the file name. The format of the filename with this option enabled is:

Prefix-SurfaceName - LayerName.txt

5. Select the *Multiple Files* output option.
6. Click **Browse** (...) to select the output folder.
7. [Optional] Enter a *Filename prefix* to add to each file name.

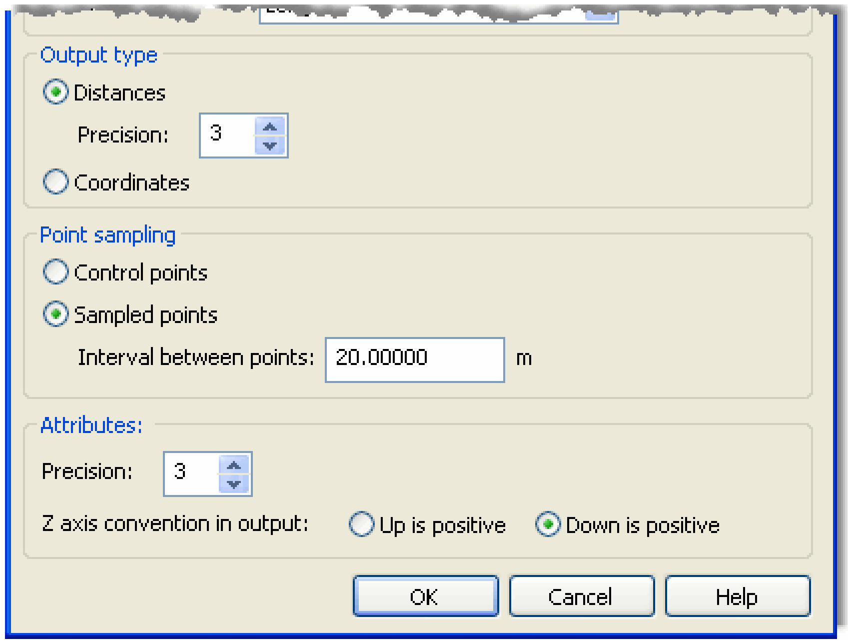
Coordinates

By default, the Geographic (Lat/Lon) - WGS84 coordinate system is set as the coordinate system for the output.

> If you change the coordinate system here, the new system will be set the next time the Export to ASCII command is used.

8. [Optional] Click **Browse (...)** to select another output coordinate system.
9. Select an option from the *Units* drop-down list.

10. Select a *Precision* value to define the number of decimal places to apply to the exported coordinate values.
11. Use the *Order* drop-down list to specify the order of the coordinate columns in the output file, e.g., Long/Lat or Lat/Long.



Output type

The ASCII file will contain the depth values at specific points along the profile and the locations of those points. Depending on the Output type selected, the locations may be reported as distances from the start of the profile (in metres, feet, etc.), or as geographic coordinates.

12. Select an *Output type* option.

If exporting locations as *Distances*, you can also define the number of decimal places to apply to the distance values.

13. Select a Precision value for the *Distances* field.

You have the option of exporting depths at all control points in the profile, or at a sampling of points.

- The *Control points* option exports depth values at the points that were digitized to create the profile line.
- The *Sampled points* option selects points at a specified interval and exports the values at those locations.

14. Select a *Point sampling* option.
15. [Optional] If exporting sampled points, enter a value for the *Interval* between points.

The Interval setting defaults to the Sample step size setting of the profile graph, but you can export sampled points at a different interval if desired.

The profile was created using attribute values from the selected source layer, e.g., Depth. You can define the precision for these values as well as the Z-axis convention to use in the output.

16. Select a Precision value to define the number of decimal places to include in attribute values.
17. Select the Z-axis convention to use in the exported values.
18. Click **OK** to perform the export.

The data in the Profile window is exported to an ASCII file. Header information includes:

- The name of the profile line.
- The coordinate projection of the data.
- The unit of measure for the attribute values.
- The Z-axis convention of the data.
- The unit of measure for distance values.
- The start and end coordinates of the profile line.
- The headings for the column order.

Export Profile to Image

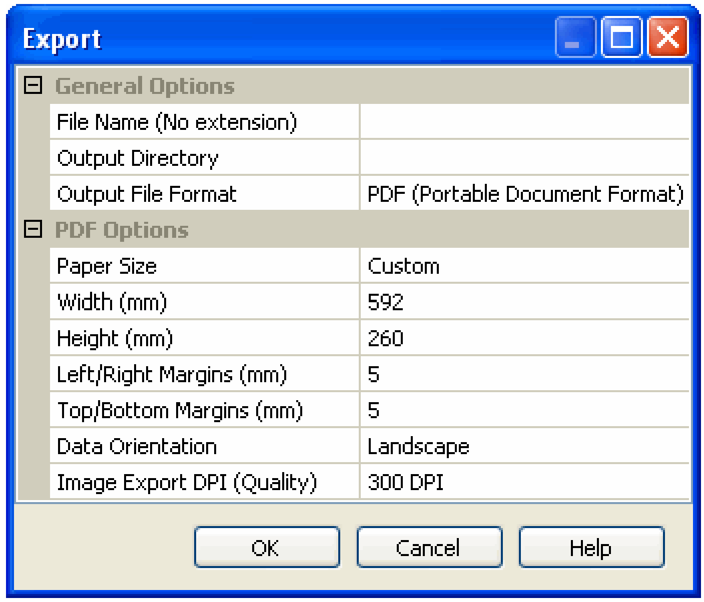
Use the Image Export command in the pop-up menu of the Profile window to export an image of the current profile graph. The profile image can be exported to:

- PDF (Portable Document Format)
- PS (PostScript)
- SVG (Scalable Vector Graphics)
- TIFF (Tagged Image File Format)

To export an image of the profile graph:

1. With the profile displayed, right-click the Profile window and select choose **Image Export** from the pop-up menu.

The Export dialog box is displayed.



Use the Export dialog box to define the settings for exporting the image. The example above displays the options for export to TIFF format.

2. Type a *File Name* for the image file.
3. Click within the *Output Directory* field to enable the **Browse** (...) button.

4. Click **Browse** and select a location for the exported file.
5. Select the *Output File Format* from the drop-down list.

Once you select a format, specific options for that format are displayed in the lower section of the dialog box.

6. [Optional] Enter or select a value for the options as needed.

Export options

PDF and PostScript options	
Paper size	Set the page size of the exported file.
Width/Height	Set the width and height of the histogram in the resulting file.
Data Orientation	Set the orientation of the histogram in the resulting file. Portrait is a vertical position; Landscape is a horizontal position. Landscape is selected by default.
Margins	Set the width of the area between the histogram and the edge of the file. The default is 5mm.
Image Export DPI (Quality)	Set the resolution used to export the histogram. The higher the resolution, the better the quality of the image when zoomed in. If the file will be used in the electronic form with the image being zoomed in tightly, a higher resolution is recommended. If the file will simply be printed, a smaller resolution is adequate.

SVG format options	
Width/Height	The width and height of the histogram in the resulting file.
Image Export DPI (Quality)	Set the resolution used to export the histogram. The higher the resolution, the better the quality of the image when zoomed in. If the file will be used in the electronic form with the image being zoomed in tightly, a higher resolution is recommended. If the file will simply be printed, a smaller resolution is adequate
TIFF format options	
Image Resolution	The resolution (quality) at which to export the image. The higher the resolution, the closer the image can be zoomed, but the larger the file.
Width/Height (Pixels)	Set the width and height of the resulting image in pixels. These fields are controlled by the resolution of the image. As the DPI is increased or decreased, so is the number of pixels.
Image Depth	Set the number of values applied to each pixel in the image (one value for each colour, plus one for transparency if using 32 Bit). If you would like a transparency setting applied to the background colour, the 32 Bit (RGBA) option must be selected.
Background Colour	The colour displayed in the background of the histogram. The default is black.
Background Alpha	(This option is only available when the Image Depth is set to 32 Bit RGBA). Set the degree of transparency applied to the background colour. The default setting is zero transparency.
Compression	Compress the resulting image file during export to decrease the file size. This option is only available when using the TIFF format. There are various compression methods available, each of which performs differently. By default, LZW is used.

7. Click **OK**.

The image file is saved to the specified location.

Sounding Selection

Survey data contains so many soundings that they cannot all be displayed legibly on a chart or map. HIPS enables users to make a meaningful selection of soundings to be displayed.

A selection of soundings is created as a sub-layer of a field sheet. The sounding selection is generated from a tile or BASE Surface attribute layer. The selection mechanism is shoal- or deep-biased and allows control over the final density of the selected sounding layer. The sounding layer can be written to a CARIS file or to a Hydrographic Object Binary (HOB) file.

This process does not physically remove the soundings from the HIPS file. It merely determines which soundings are displayed.

You can colour code the selected soundings according to depth, if necessary. See “Colour Options for Soundings and Contours” on page 394.

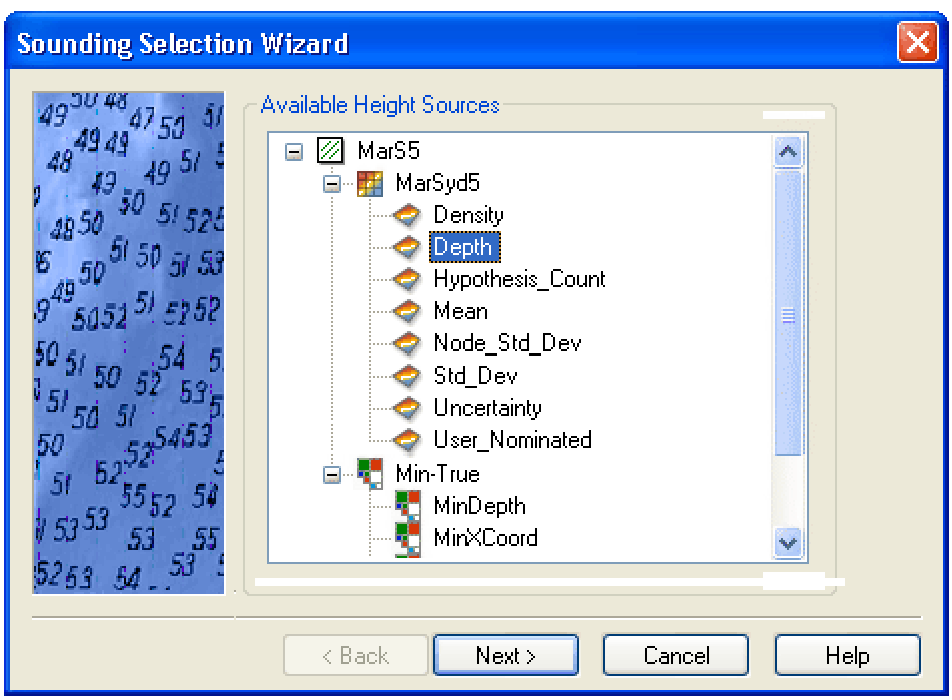
To create a layer of selected soundings:

1. Open the field sheet containing the tile layer or BASE surface.
2. Select the Selected Soundings command.

Menu	Process > Products >Selected Soundings
Pop-up	New > Selected Soundings…

The Sounding Selection Wizard is displayed.

Select Height Source



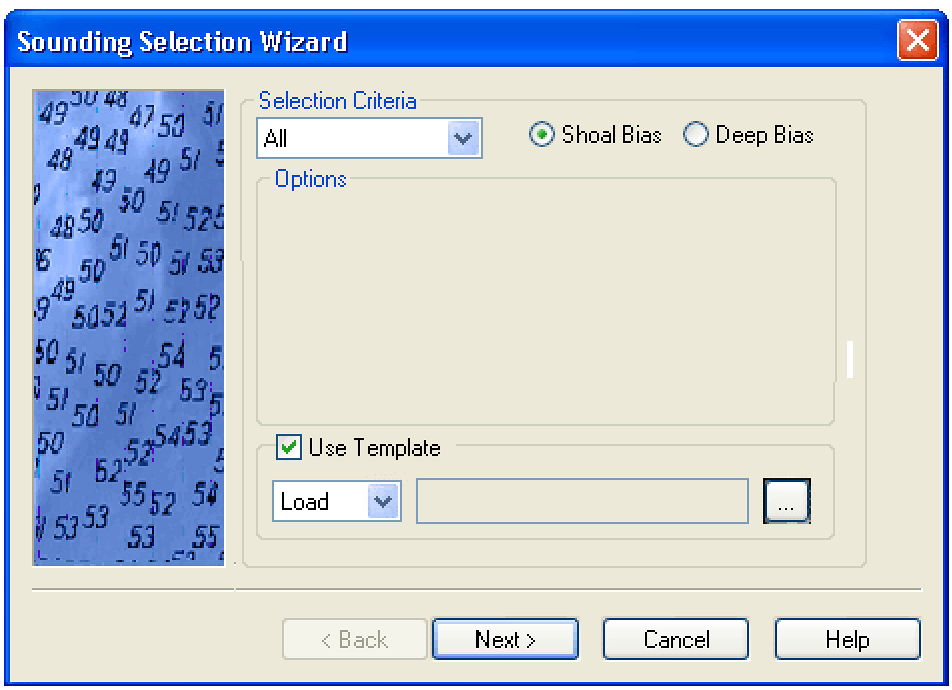
The Selected Soundings function uses tile layers (including bin layers) and BASE Surface attribute layers as data sources.

From the chosen layer, one value from each tile/bin or attribute layer node is used to represent all of the soundings from that cell. In the case of a bin layer, there is only one value present, so it is used. In the case of a tile layer, each tile contains a number of attributes and a primary sounding. The primary sounding is the value that is used in the Selected Soundings operation.

3. Select a BASE Surface attribute layer or tile layer. You can view all layers by clicking the + icon to expand the file tree.
4. Click **Next**.

Selection Criteria

The settings in this dialog box determine the density of the selected soundings in the layer.



Selection is based on two main options:

- a sounding criteria method (e.g., radius or overplot removal)
- a conflict resolution method (shoal bias or deep bias)

Selection criteria

You can apply one of three methods for sounding selection:

- *All:* selects soundings from:
  - node values from the BASE Surface attribute layer
  - the primary sounding from every tile

- the sounding stored in every bin

- *Radius:* selects only the (shoal or deep) soundings within a set distance
- *Overplot Removal:* selects only certain soundings in a dense data set so that they can be plotted within the field sheet without overlapping.

Conflict resolution

You can set whether the shoalest soundings or the deepest soundings will be included when conflicts are encountered during selection. For example, if you select *Shoal bias,* and two soundings overlap, the shoalest of the two will be selected.

1. Set either *Shoal Bias* or *Deep Bias* as the method of resolving any conflicts during the selection process.
2. Select one of the three criteria from the *Selection Criteria* drop-down list.

If you select *Radius* or *Overplot Removal*, the dialog box will display the options associated with those criteria.

Template

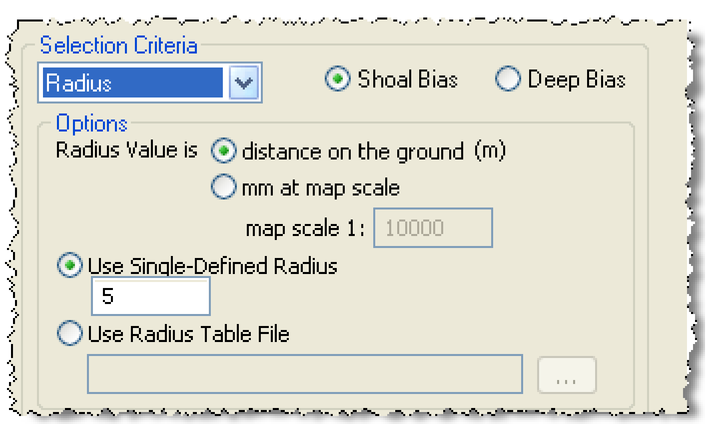
Settings in the wizard can be saved to a Sounding Selection Template (CSST) file. This file can be opened and reused.

3. Select the *Use template* option to open or create a template file that contains sounding selection options.
4. Select *Load* to open an existing template file, or

   select *Save* to create a new template file from the options you are setting in the wizard.
5. Click the Browse button to open a file, or create a new file.
6. Click **Next**.

Radius

If you choose *Radius* as your selection criteria, the following option fields are displayed:



The radius value you set (either as a specific value or from a radius table file) will either be applied as metres on the ground, or as the number of millimetres between soundings at map scale.

Once you have set how the radius value will be applied, you can either set the same interval value to be applied to each radius, or have varying values applied from a radius table file.

For example, if you selected Shoal bias, and set a single-defined radius of 5, then only the shoalest sounding within each 5m radius on the ground, or within 5mm at map scale, will be displayed.

Sounding Radius Table

A radius table sets the minimum radius between soundings for various depth ranges. Each depth range will have its own radius value.

The sounding radius table is a text file containing three columns:

- minimum depth of the range
- maximum depth of the range
- radius value for the depth range.

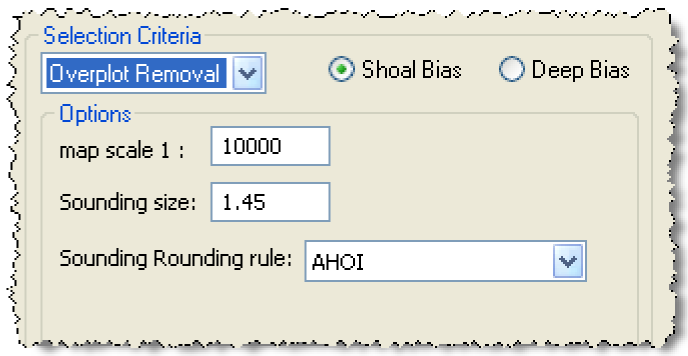
For example, using the example radius table below, 8m soundings would be a minimum of 1mm apart at map scale, while 80m soundings be 10mm apart..

Min depth	Max depth	Radius value for range
`0.0`	`10.0`	`1.0`
`10.0`	`20.0`	`2.0`
`20.0`	`50.0`	`5.0`
`50.0`	`500.0`	`10.0`

1. Set the *Radius Value*.
   - Select distance on the ground (m), or
   - Select *mm at map scale*, and type a scale value in the *Map Scale 1:* field (default is 1:10000).
2. To set a standard minimum distance between soundings, select the *Use single defined radius* and enter the interval.
3. To use a radius table to set the minimum distance between soundings, select the *Use radius table file* option and click **Browse** to select the file.
4. Click **Next**.

Overplot Removal

If you choose the *Overplot Removal* option from the *Selection Criteria* list, the dialog box is refreshed to show the following options.



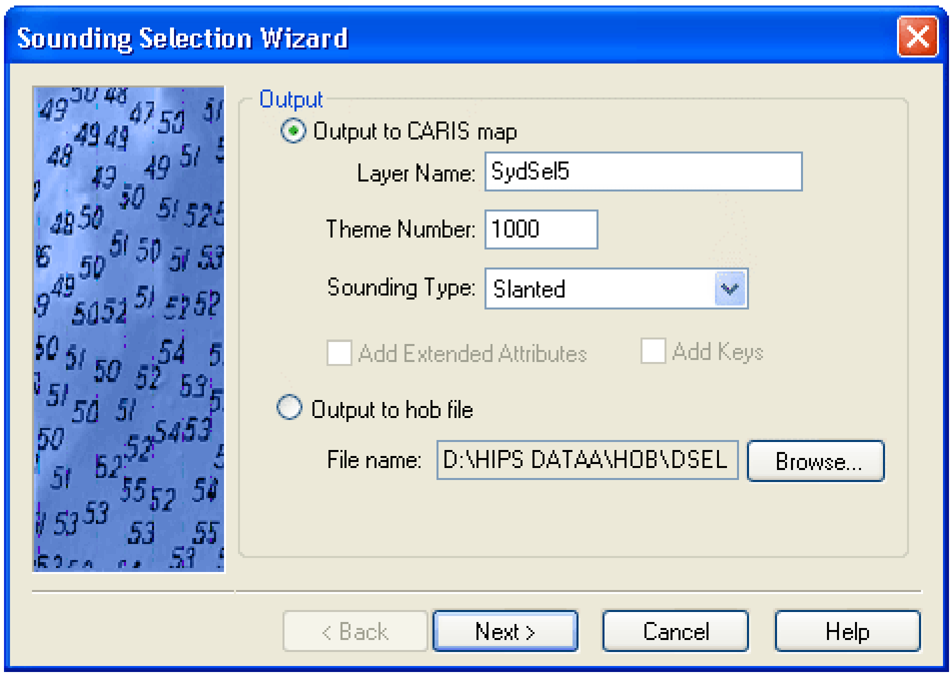
Overplotting occurs when soundings overlap or are so close together that they are plotted on top of each other. Two soundings with depths of 11 and 22 might look like 1212 if they were overplotted.

Overplot Removal places a buffer zone around each sounding, then suppresses either the deepest or shoalest soundings which overlap these buffer zones.

5. Type the map scale for the selected soundings (default is 1:10000).
6. Type the size of the soundings (that you want them to appear in the Display window) in the *Sounding Size* field.
7. Select a *Sounding Rounding Rule* from the drop-down list. Click **Next**.

Output Selection

Use this dialog box to set the output format options for the selected soundings.



The soundings can be exported to two formats: CARIS map or HOB file. The CARIS map file is opened as a layer in your currently open project. The HOB file is stored in the same directory as the field sheet. The default directory is ..\Hips\Fieldsheets\ProjectName.

"Output to CARIS map" on page 388

"Output to HOB file" on page 389

Output to CARIS map

To display selected soundings in HIPS:

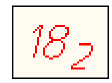
1. Select the *Output to CARIS Map* command.
2. Type a name for the layer in the *Layer Name* field.

The *Theme Number* is used to organize information in the CARIS file. It is important to maintain each layer on a separate theme, therefore be sure to change it from the default number each time you create a new sounding selection.

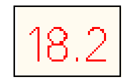
3. Type a number in the *Theme Number* field.

Soundings can be displayed in two formats.

- *Slanted*: This format displays the digits of the sounding in a slanted font with any value less than 1 in subscript.
- *Engineer Soundings*: This format displays the digits of the sounding in regular text with values to the right of a decimal point where appropriate.



Slanted



Engineer

4. Select either *Engineer* or *Slanted* sounding styles from the drop-down menu.

The *Add Extended Attributes* option adds useful additional information to each sounding. The *Add Keys* option gives each sounding a unique identifier or key. This key provides a link back to the original sounding in the HIPS file and is required to open a specific sounding in the HIPS Swath/Single Beam or Subset Editors.

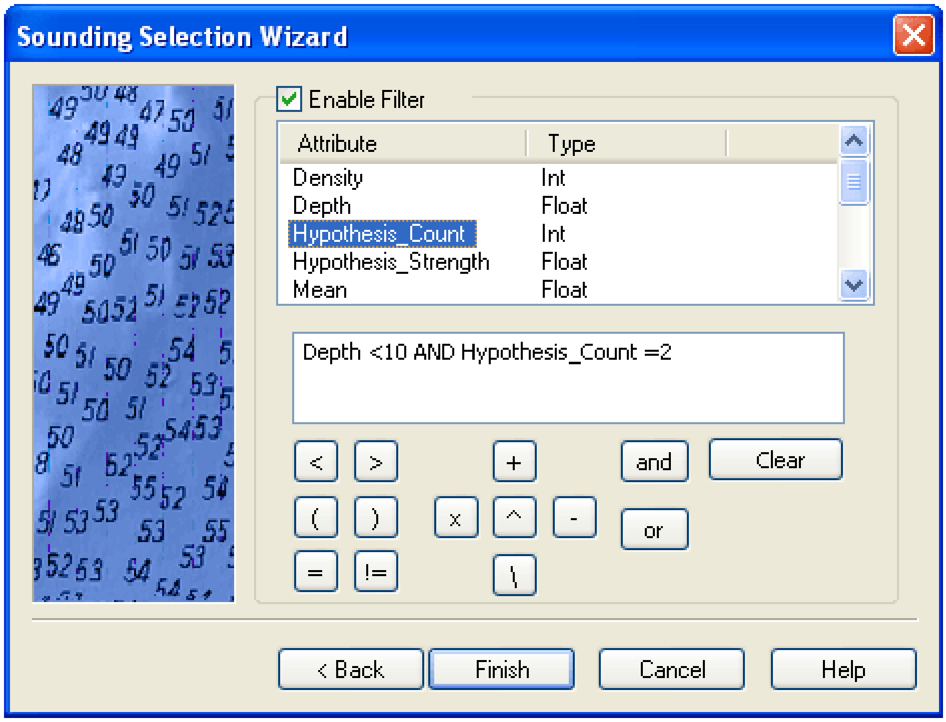
5. Select the *Add Extended Attributes* option if you want to include time stamp, launch ID, accuracy value, and tidal reduction value with the soundings.
6. Select the *Add Keys* option to give each sounding a unique identifier. The key format is XXXPPPPPBBB where XXX is a random value, PPPPP is the profile number and BBB is the beam number.
7. Click **Next.**

Output to HOB file

A HOB file contains S-57 feature objects. Attributes to help identify the soundings are also included in the HOB file. The wizard enables you map these attributes to corresponding S-57 attributes.

1. Select the *Output to Hob File* option.
2. Click **Browse** to select a directory and a name for the HOB file.
3. Click **Next.**

Enable Filter



The *Enable Filter* option allows you to extract and display only the soundings that meet the conditions you set for the filter.

1. Select the *Enable Filter* check box.
2. Use a combination of attributes, operators and values to construct your filter.
3. Use the Backspace key to delete a character or **Clear** to remove the entire filter string

For example, to display only the soundings with a Depth less than 10 metres, and a hypothesis count of 2:

- Double-click on Depth in the list of available *Attributes* to place it in the filter field.
- Click the “**<**” button, and type 10.
- Click the **and** button and double-click on the Hypothesis_Count attribute.
- Click the “**=**“ button, and type 2.

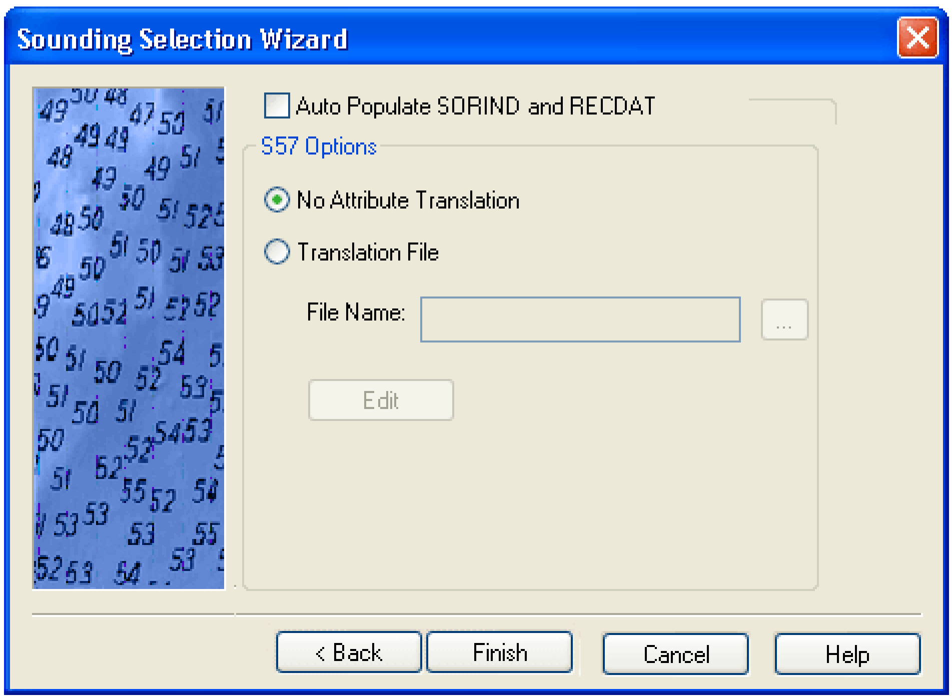
HIPS and SIPS will automatically adds the necessary spaces between the parameters in the filter string. However, if you choose to type the filter parameters into the filter field, you must manually add those spaces.

4. Click **Next** if a HOB file is your output, or click **Finish** if you selected CARIS Map in Step 3.

HOB S-57 Options

When sounding selection output goes to a HOB file, the process generates S-57 SOUNDG objects. These can have certain attributes populated.

Use the options on this dialog box to automatically enter information into *SORIND* and *RECDAT* attribute fields, and to map BASE surface attributes to specific S-57 attributes.



Auto Populate SORIND and RECDAT

These attributes can be automatically populated:

- SORIND (information about the source of the sounding) and
- RECDAT (date the object was captured, edited or deleted).

1. [Optional] Select the *Auto Populate SORIND and RECDAT* check box to have information on these attributes automatically added to the soundings in the HOB file.

> Note: auto populating the SORIND and RECDAT attributes can increase save and export times as well as file size, particularly with large, high-density data sets.

S-57 options

The other S-57 option is to map the BASE surface attributes to S-57 attributes, via a translation file. The translation file has a BSST file extension and is in XML format. The *No Attribute Translation* option is set by default.

To continue without mapping any attributes:

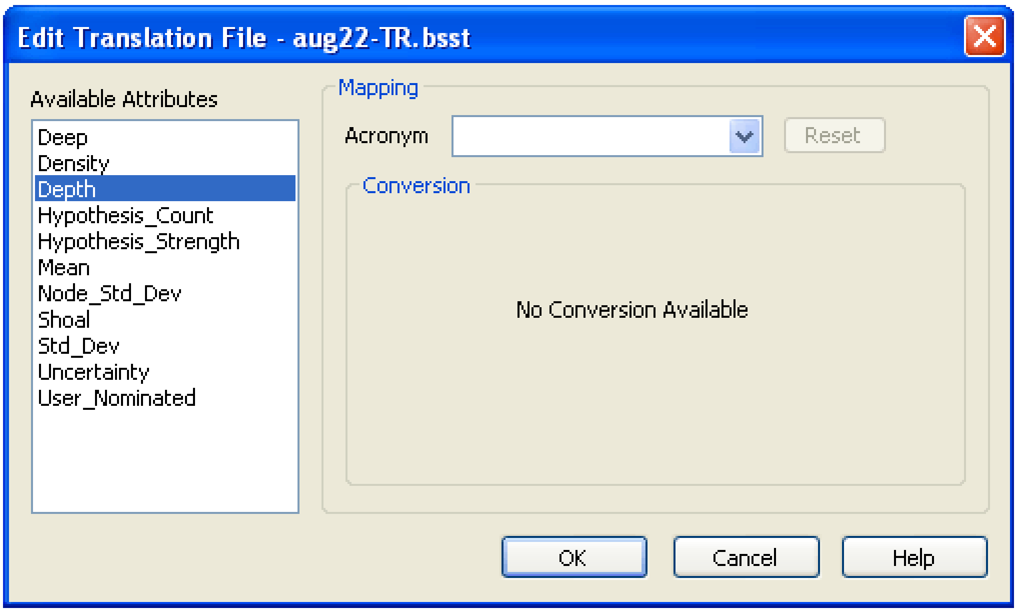
1. Click **Finish**.

Translation file

To use a translation file to map attributes:

1. Select *Translation File*.
2. Click **Browse** to select an existing BSST translation file (or to specify a name for a new translation file).
3. Click **Edit** to set the mapping for BASE surface attributes to S-57 attributes.

The Edit Translation File dialog box is displayed.

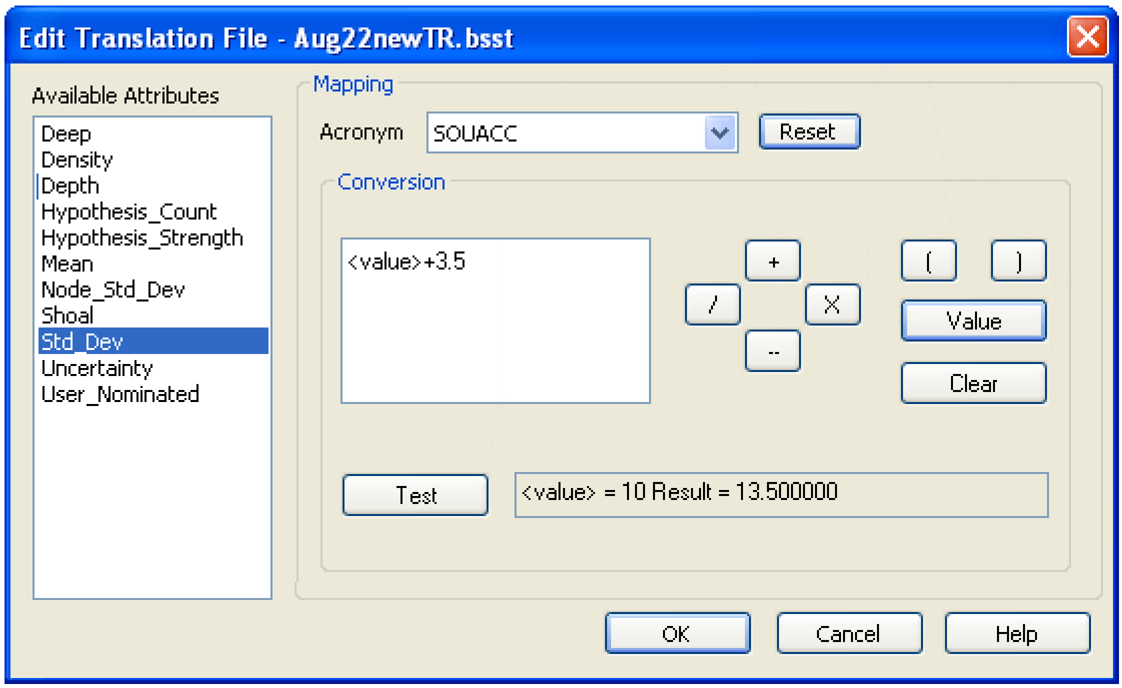


1. Select a BASE surface attribute from the *Available Attributes* list.
2. Select an appropriate S-57 attribute from the drop-down list.

If no mapping is possible between the selected attribute and the chosen S-57 attribute acronym, “No conversion available” is displayed.

If conversion can be applied, the dialog box will display the available values that can be mapped.

HIPS attributes of Integer data type, may be appropriate to map to S-57 attributes, using a conversion formula. A conversion formula is created by inserting values and appropriate mathematical operators into the *Conversion* area, as in the example below.



To create a conversion formula, for example, to map Std_Dev to SOUACC, (sounding accuracy attribute):

1. Click **Value** to use the value of the selected BASE surface attribute in the conversion formula.
   - This places the <value> variable in the formula window.
2. Insert mathematical operators using the **+**, **-**, **X**, **/** buttons.
3. Type the numeric value by which to adjust the original value, if needed.
   - In this example, adjust the original value by adding 3.5.
4. Click **Test** to see the formula.
5. If you want to erase the formula, click **Clear**.
6. Click **Reset** to remove the mapping of BASE surface attribute to S-57 attribute.
7. Click **OK** to save the mapping settings and return to the HOB S-57 Options dialog box.
8. Click **Finish.**

Selected soundings are saved to the HOB file, as SOUNDG features. The original BASE surface attributes, e.g., Std_Dev, are converted to the S-57 attributes to which they were mapped. In the example above, the Std_Dev value was 10. In the HOB file, this is translated to a Sounding accuracy of 13.5.

View the HOB file using the Open Background data command. The values mapped to the SOUACC attribute are displayed in the Sounding accuracy column in the Selection tab, when the soundings are selected and queried. See also "View soundings in HOB file" on page 51 in the Reference Guide.

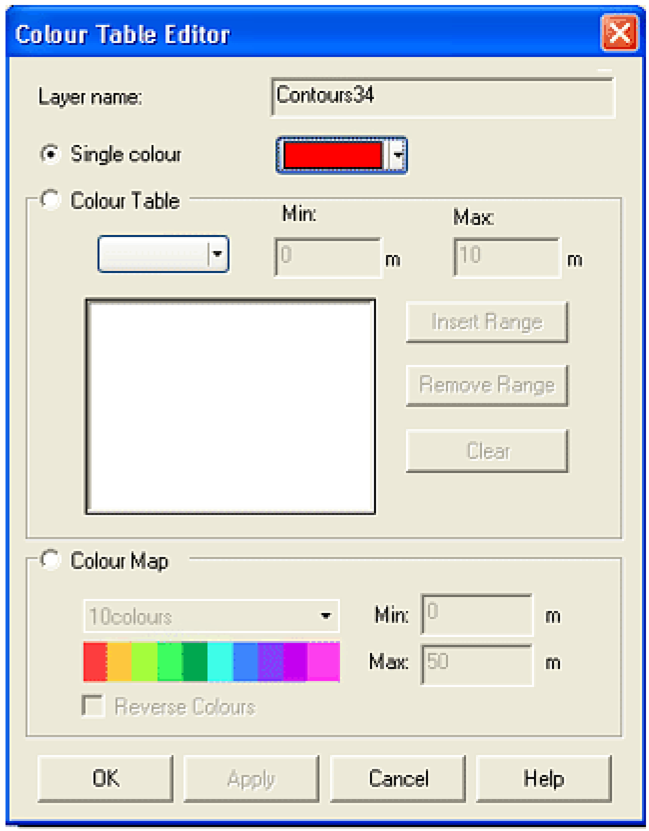
Colour Options for Soundings and Contours

The Colour Table Editor lets you colour code soundings and contours for easier display and interpretation.

1. Select a sounding or contour layer in the Layers tab.
2. Select the Edit Colours command from the right-click menu.

The Colour Table Editor dialog box is displayed.

.



Single colour

Use this option to display a soundings or contours in a single colour.

3. Select the *Single Colour* option.
4. Select a colour from the colour picker, or create a custom colour from the standard Windows colour palette.
5. Click **OK** to save and apply settings.

Colour table

To apply different colours to different depth ranges:

1. Select the *Colour Table* option.
2. Select a colour from the colour palette, or create a custom colour from the standard Windows colour picker.
3. Type a minimum range depth in the field beside the palette.
4. Type a maximum range depth in the next field
5. Click **Insert Range**.

The colour and range values are displayed in the Colour Table dialog box.

6. Repeat Steps 2 to 5 to continue adding a colours for each range.
7. To remove a single colour range, select a range so it is highlighted and click **Remove Range**.
8. To remove all selected ranges, click **Clear**.
9. Click **OK** to save and apply settings.

The soundings or contours are colour-coded according to the range values you selected earlier.

Standard colour map

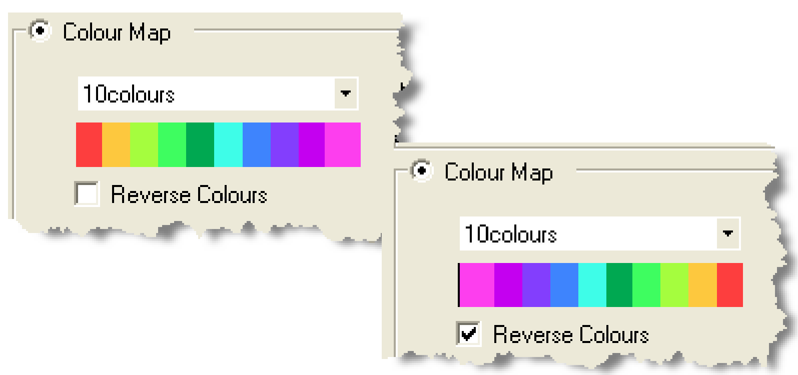
Use this option to select a standard colour map.

1. Select the *Colour Map* option.
2. Select a colour map from the drop-down menu.
3. Type the minimum and maximum range values for the colour map.
4. Click **OK** to save and apply settings.

The layer objects are displayed according to the colour map settings.

Reverse colours

Select this option to reverse the order in which the colour map is applied (see illustration below).

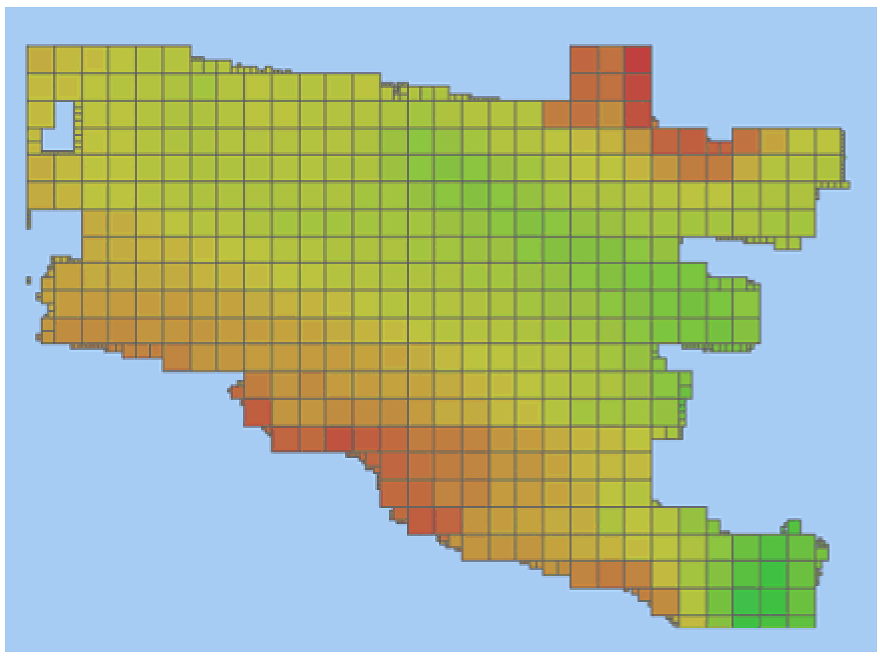


Tiling a field sheet

The tiling of a dataset allows for data points, in this case geographically referenced soundings, to be gathered into tiles depending on user-defined statistical criteria. Once the tiles have been created, products such as contours and selected soundings can be created using the tiles as a height source.

Generally, the tiling process splits a selected area into sections (tiles) and continues sub-dividing until a network of squares cover the data.

Below is an example of a tiled field sheet in the Display window. These tiles are all the same size.



A special type of tiling is used to track the progress of subset data cleaning, (see "Track Progress with Subset Tiles" on page 343) and for performing statistical surface cleaning on sounding data (see "Statistical Surface Cleaning" on page 311).

Tiling methods

There are three methods of tiling a field sheet: tiling, binning and QTC tiles. If you need to examine the data for more detailed statistical information then the tiling option should be selected.

Tiling

The process divides the field sheet into four sections (tiles) and continues sub-dividing each section by four until the criteria for creating the tiles is reached. The result is a network of tiles covering the data. Tiles can be the same size or vary in size.

Binning

Binning splits the surface of a field sheet into squares or “bins” based on a user-defined size. Within each bin, the shallowest or deepest sounding can be selected. This option is less computationally intensive than tiling as there are no additional statistical criteria applied.

QTC Tiles

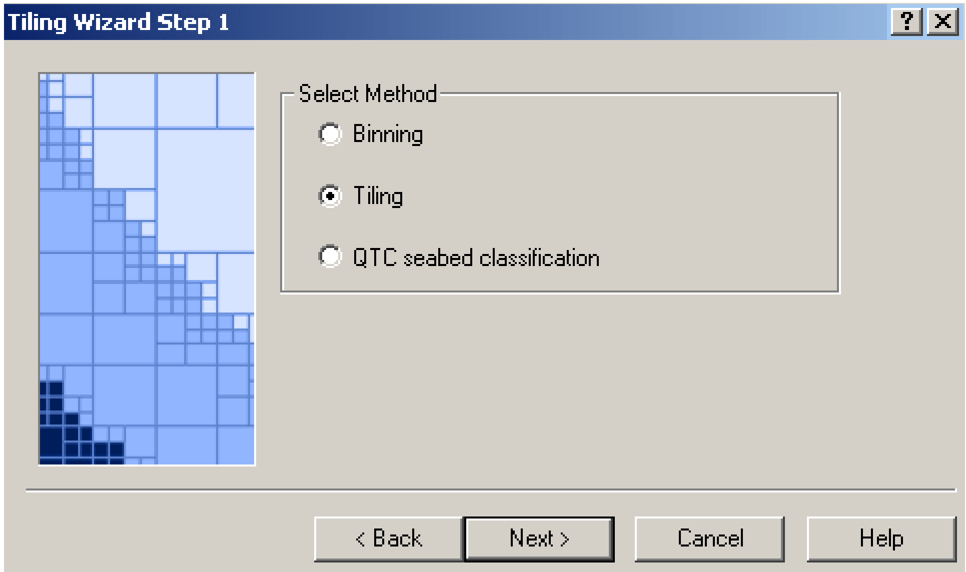
Sea floor classification is a procedure in which the amplitude and shape of the reflected acoustic signal is used to interpret the physical nature of the bottom. Quester Tangent Corporation has developed a program called QTC View to perform this analysis. The analysis in QTC View groups the return signals into classes which correspond broadly with physical properties of the sea floor. The results are summarized in a text file (.seabed). HIPS and SIPS generalizes these results by creating tiles based on the class values in the *.seabed files.

Create Tiles

1. Open a field sheet.
2. Select the field sheet layer in the Layers tab.
3. Select the Tiling command.

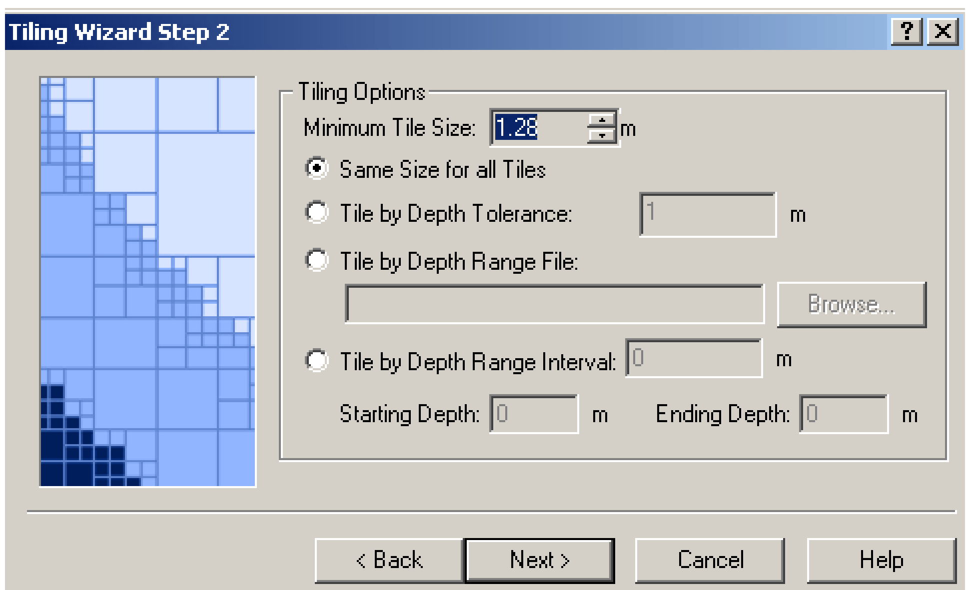
Menu	Process > Products > Tiles
Pop-up	New > Tiles

The Tiling Wizard - Step 1 dialog box is displayed.



4. Select the method of tiling. (In this example, Tiling is selected.)
5. Click **Next**.

The Tiling Wizard - Step 2 dialog box is displayed.



Tiling options

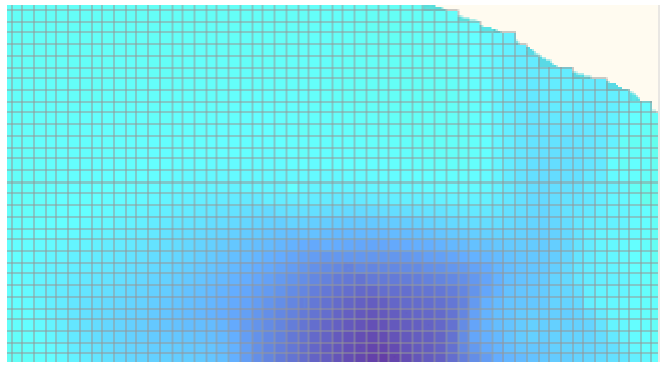
This step is to define which of the tiling methods will be used to create tiles: tiling by size, by depth tolerance or by depth range. First, a minimum tile size must be set, since it is an important criteria in each of these methods.

6. Set the minimum tile size by clicking the arrow buttons.

Next, select a tiling option from the following:

- *Same Size For All Tiles*: Set all tiles to the minimum tile size.

This method will subdivide the data until all tiles are equal to the selected size. No other statistical criteria will be applied. The result will be equal sized tiles similar to the result from Binning, as illustrated below.

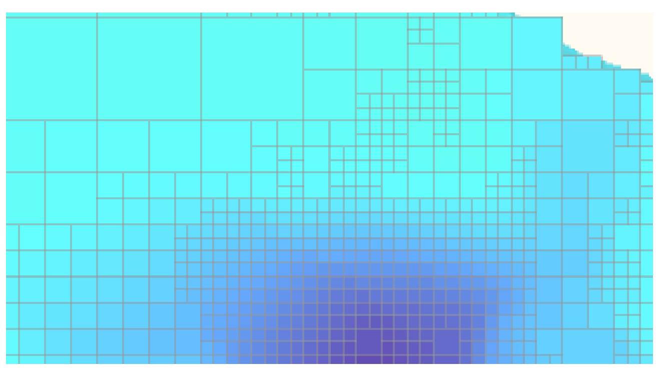


- *Tile by Depth Tolerance*: This method makes use of two criteria: a tolerance value as well as the minimum tile size.

  The tolerance value entered here will determine the variation of depths allowed in a single tile. For example, if a value of 2 is entered as the tolerance value, the tiles will be subdivided until all the depths contained in a single tile are within 2m of each other.

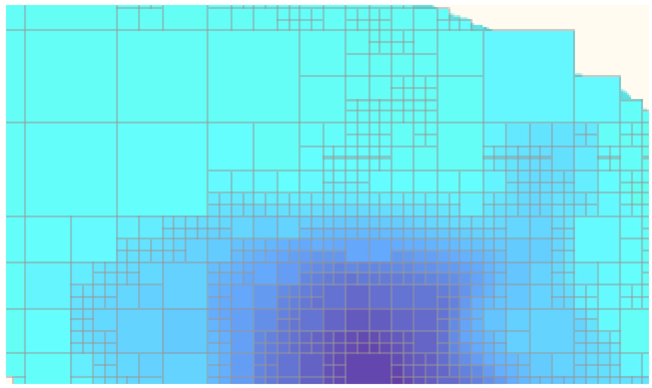
  The second criterion to be met is the Minimum Tile Size. If the first criterion is not met for certain tiles, the tiles will only be subdivided until they meet the Minimum Tile Size.

  Illustrated below is a tile set created with a depth tolerance of 2m and a minimum tile size of 1.28m.



- Tile By Depth Range: Uses range criteria as well as the minimum tile size. Based on the user-defined interval, the tiles will be subdivided until no tile crosses the specified depth interval. The subdivision will only take place between the specified starting and ending depths. However, if the depth range criterion cannot be met before the tiles met the minimum tile size, the subdivision of the tiles will stop.

This method allows for greater sampling at an interval change. Tiling by Depth Range can be applied by selecting a Depth Range file, or by entering a Range Interval, Starting Depth and Ending Depth. In the example below, a depth interval is set to 5m.



7. Select a tiling option.

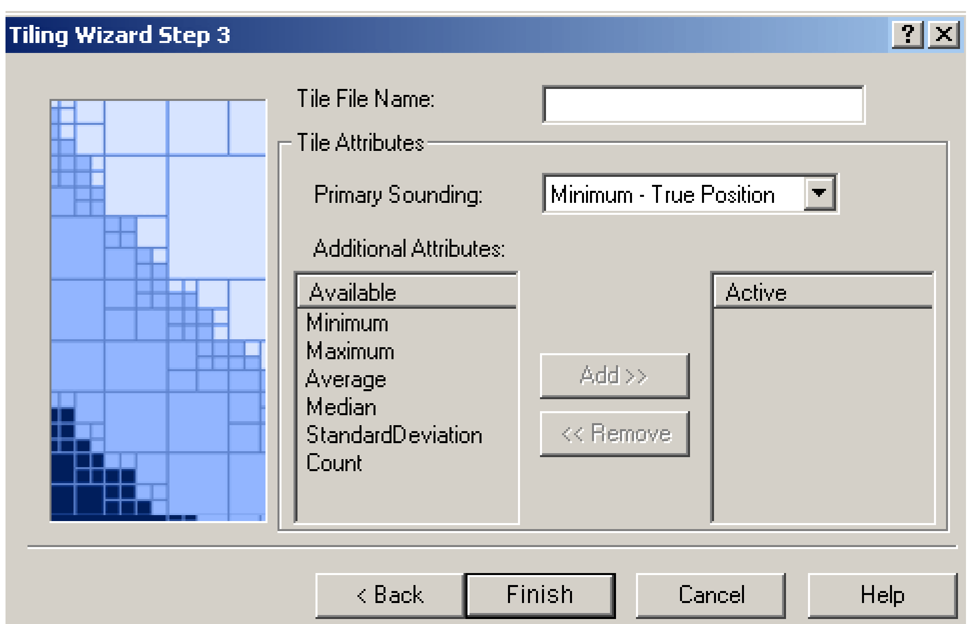
   If you select Tile by Depth Tolerance, type the value to be applied.

   :*If you choose Depth Range File,* click **Browse** to select the file. The tiles are created using the depths listed in the file. (See "Depth Range Files" on page 409.)

   If you select *Tile By Depth Range Interval,* type a value for the depth interval and for the starting and ending depths.

8. Click **Next**.

The Tiling Wizard - Step 3 dialog box is displayed.



9. Type a name for the tile file.

Primary sounding

10. Select a primary sounding as the starting point for the tiling process. Choose from five options:
    - *Minimum - True Position*: Use the shoalest sounding at its actual position.
    - *Maximum - True Position*: Use the deepest sounding at its actual position.
    - *Average - Tile Centre*: Use the mean value of the soundings as the centre of the tile.
    - *Minimum - Tile Centre*: Use the shoalest sounding as the centre of the tile.
    - *Maximum - Tile Centre*: Use the deepest sounding as the centre of the tile.

You can also add other attributes to the tile set. These attributes (such as maximum depth, average depth, etc.) add to the value of the tile set as they can later be selected as the height source for the tile display or used to produce contours and selected soundings.

11. Choose other tile attributes by selecting an attribute in the Available list and clicking **Add** to move it to the Active list.You can move the attribute from Active list by clicking **Remove**.
12. Click **Finish** to complete the tiling process.

A progress indicator on the status bar shows how much of the tiling process has been completed. Once the tiling is completed, it is shown in the Display window and the tile layer is listed in the Control window.

Create Bins

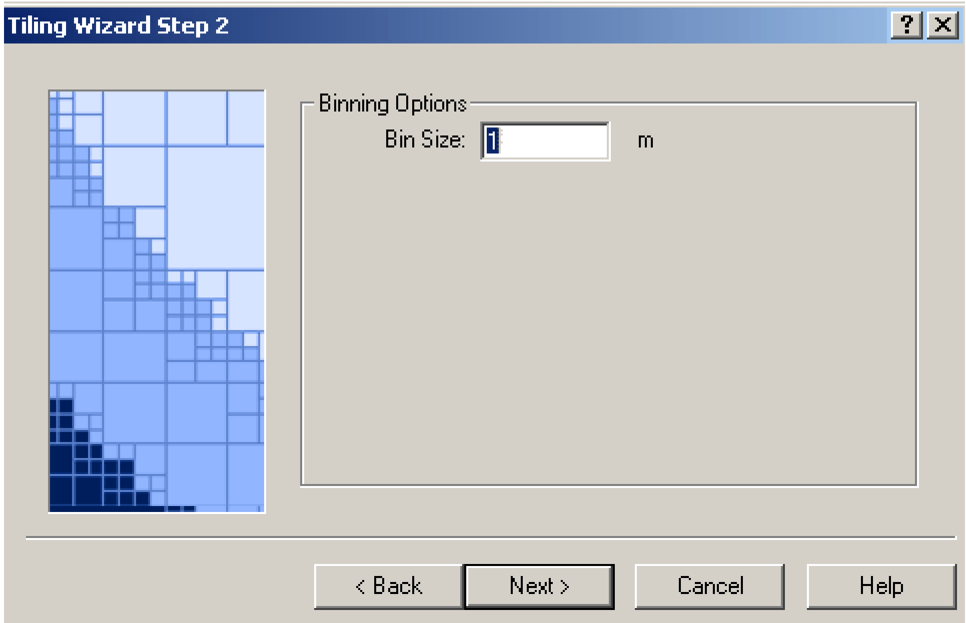
1. Select the field sheet.
2. Select the field sheet layer in the Layers tab.
3. Select the Tiles command.

Menu	Process > Products > Tiles
Pop-up	New > Tiles

The Tiling Wizard - Step 1 dialog box is displayed.

4. Select the Binning option.
5. Click **Next**.

The Tiling Wizard - Step 2 dialog box is displayed.



6. Type a size for the bin tiles.
7. Click **Next**.

The Tiling Wizard - Step 3 dialog box is displayed. The *Available Attributes* section is dimmed.

8. Type a name for the file
9. Select a minimum or maximum sounding position as the starting point for the binning process.
10. Click **Finish**.

A progress indicator on the status bar shows how much of the process has been completed. Once the binning is completed, a bin file is created and displayed in the Display window. A bin layer is also added to the Layers tab.

Create QTC Tiles

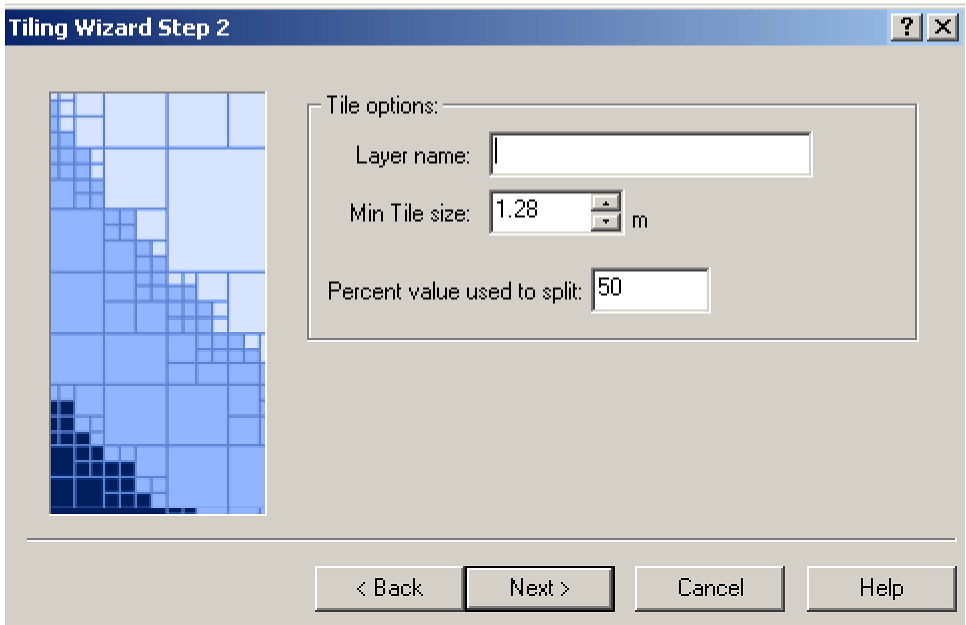
1. Select the field sheet.
2. Select the field sheet layer in the Layers tab.
3. Select the Tiles command.

Menu	Process > Products > Tiles
Pop-up	New > Tiles

The Tiling Wizard - Step 1 dialog box is displayed.

4. Select the QTC Seabed Classification option.
5. Click **Next**.

The Tiling Wizard - Step 2 (QTC) dialog box is displayed.

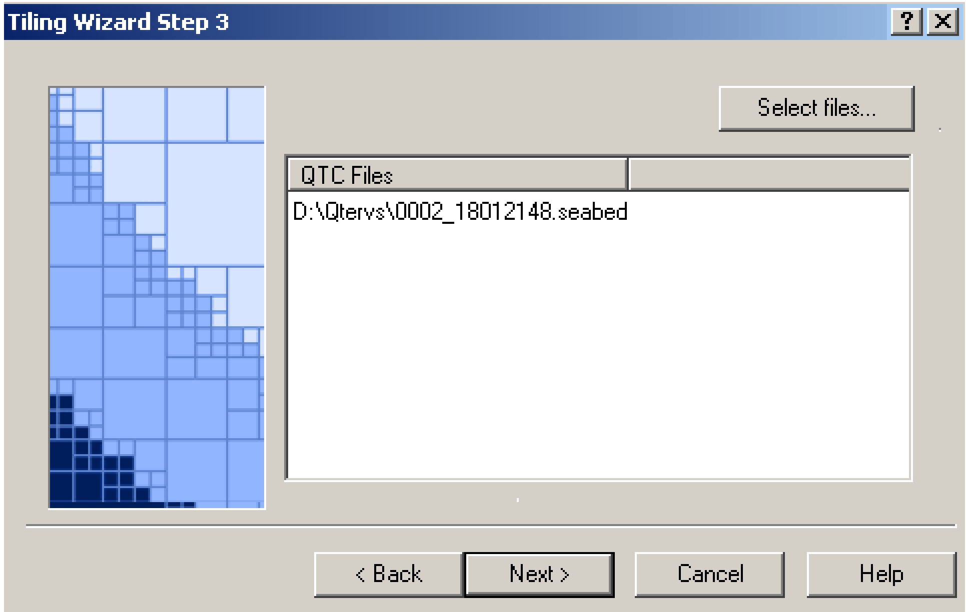


6. Type a name for the QTC tile layer.
7. Set the minimum tile size by clicking the up or down arrow buttons. As you increment upwards, the tile size is doubled.

The *Percent Value Used to Split* option continuously splits the tiles until the total amount of a specific class value in a tile meets or exceeds the entered percentage. For example, if the percentage is set at 50, then a tile is continuously sub-divided into smaller tiles until one of the tiles contains at least 50 per cent of a class value.

8. Type the percent value.
9. Click **Next**.

The Tiling Wizard - Step 3 (QTC) dialog box is displayed.



10. Click **Select files** to browse to the *.seabed file used to create the tiles.

The selected file is displayed in the *QTC Files* text area.

11. Click **Finish**.

A progress bar displays the percentage of tiles created. When the process is completed, the tiles are drawn to the Display window, and listed in the Layers tab.

Open and Close Tiles

Tiles are opened when the field sheet with which they are associated is opened.

Menu	File > Open Field Sheets

1. Open a field sheet.

The field sheet file name is displayed in the Layers tab of the Control window.

2. Expand the field sheet file tree by clicking the Expand (+) icon beside the field sheet so the tile file layer is visible.
3. Select the check box beside the tile layer name, to make the tile visible in the Display window.

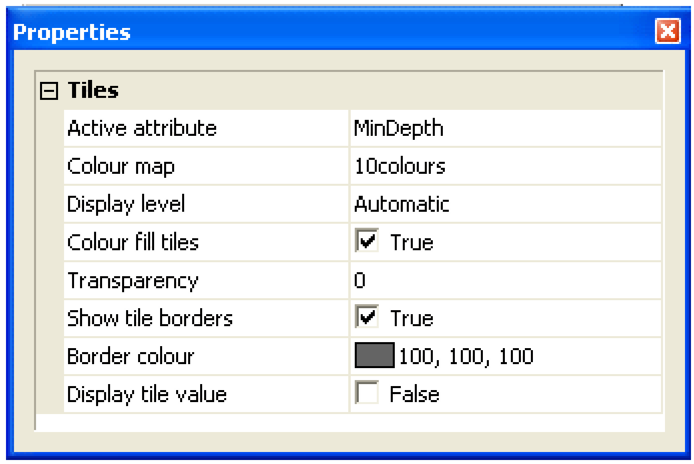
Tile layers are closed when the field sheet they are associated with is closed.

Set Tile Display Properties

Set the display properties for tiles.

1. Select a tile layer in the Layers tab of the Control window.
2. Select the Properties window command, or right-click the layer and select Properties from the pop-up menu.

The Properties window displays the tile layer properties.



The *Active Attribute* list contains all attributes included in the tile (such as the Profile number). When you select an attribute, the tile is displayed according to the attribute values

1. Select an attribute from the drop-down *Active Attribute* list: MinDepth, MinXCoord or MinYCoord. By default MinDepth is selected.
2. Select a colour map from the drop-down list. A preview of the colour map is displayed below the list.
3. Change how tiles are rendered to the Display Window by selecting a *Display Level* option:
   - *Maximum*: Draw all tiles
   - *Automatic*: Draw tiles to fit the current extent of the Display window.

The *Colour fill tiles* check box determines how attribute values are drawn in the Display window.

- If the *Colour fill tiles* check box is cleared, these values are drawn according to the selected colour map, but the tiles are left blank.
- If the *Colour fill tiles* check box is selected and set to True, the individual tiles are filled according the colour map, but the attribute values are shown in black.

4. Select a *Transparency* percentage to make features visible through the tiles: the greater the percentage, the more transparent the tiles appear.

The *Show tile borders* check box determines how the border lines dividing the tile areas are displayed.

- If the *Show tile borders* check box is cleared, the lines are colour coded according to the colour map and attribute values.

- If the *Show tile borders* check box is set to True, you can choose a single colour (using the Border colour map) for the tile border lines.

5. Set the *Show tile borders* check box to True to display a border line colour.
6. Select a colour for the tiles borders from the *Border colour* map.
7. Set the *Display Tile Value* check box to True to display the values for the active attribute, for example, MinDepth will display sounding.

View Tile Attributes

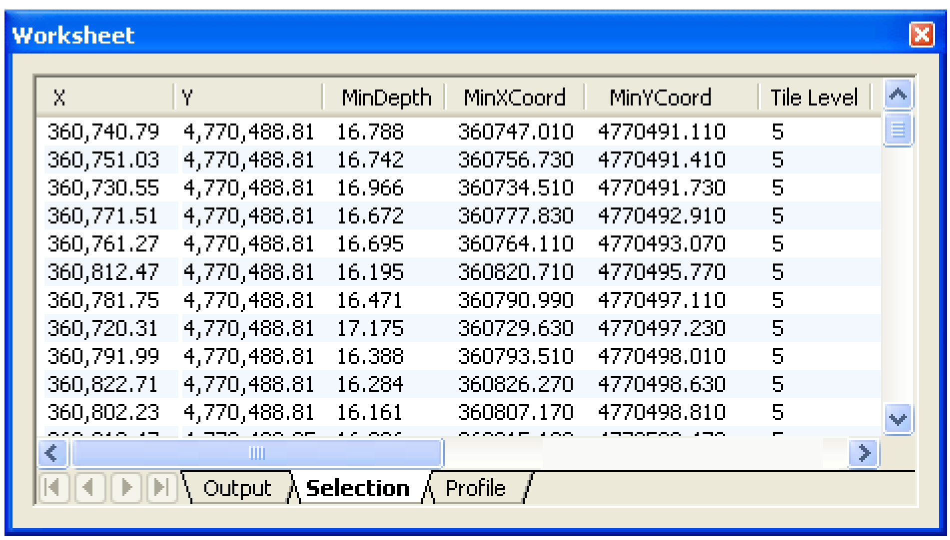
You can view information on currently selected tiles in the Selection tab of the Worksheet window.

1. Open tiles are open in the Display window.
2. Select the tile layer in the Layers tab of the Control window.
3. Select the tiles to be viewed.

The selected tiles are highlighted in the Display.

4. Select the Selection tab.

The tile data for the selected tiles is displayed in the Selection tab.



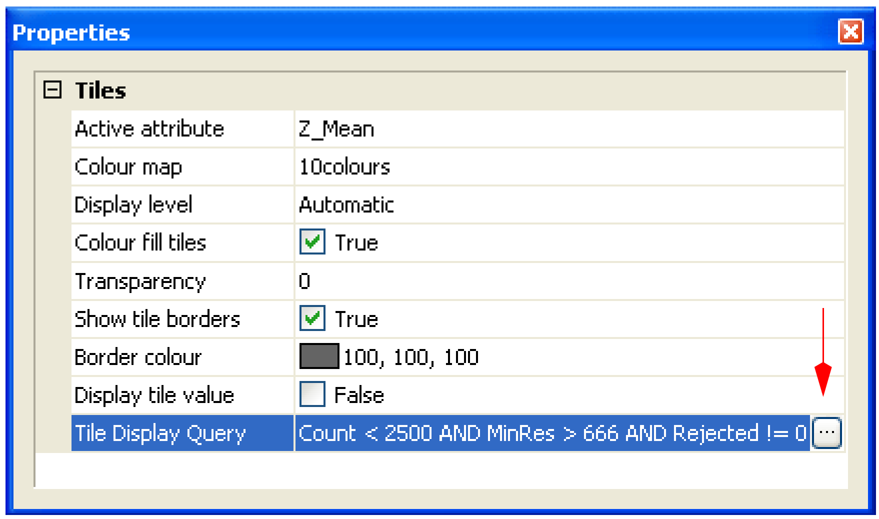
X	Y	MinDepth	MinXCoord	MinYCoord	Tile Level
360,740.79	4,770,488.81	16.788	360747.010	4770491.110	5
360,751.03	4,770,488.81	16.742	360756.730	4770491.410	5
360,730.55	4,770,488.81	16.966	360734.510	4770491.730	5
360,771.51	4,770,488.81	16.672	360777.830	4770492.910	5
360,761.27	4,770,488.81	16.695	360764.110	4770493.070	5
360,812.47	4,770,488.81	16.195	360820.710	4770495.770	5
360,781.75	4,770,488.81	16.471	360790.990	4770497.110	5
360,720.31	4,770,488.81	17.175	360729.630	4770497.230	5
360,791.99	4,770,488.81	16.388	360793.510	4770498.010	5
360,822.71	4,770,488.81	16.284	360826.270	4770498.630	5
360,802.23	4,770,488.81	16.161	360807.170	4770498.810	5

Tile Display Query

Using the Tile Display Query dialog box you can create an SQL-type query based on selected tile attributes and defined values. Only those tiles that meet the search criteria are displayed in the Display window. The query results can be saved in the Tiles Properties when the session is saved.

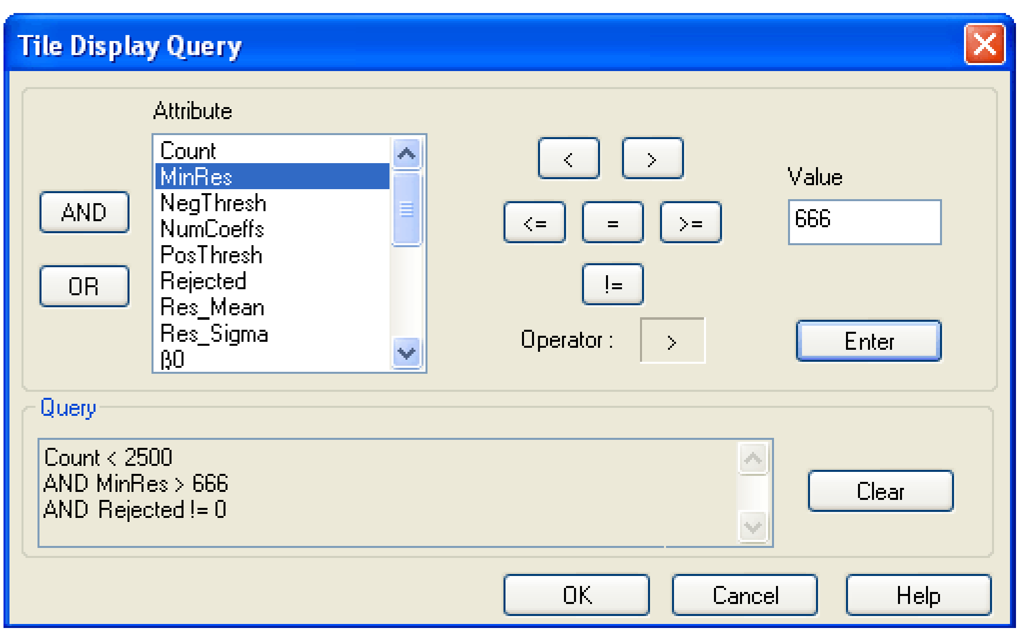
The Tile Query dialog box is opened from the Properties window, when the tile layers are selected.

1. Select the tile layer in the Layers tab.
2. Open the Properties window for the Tiles layer, as seen below.



3. Click the **Browse (...)** button in the Tile Display Query field.

The Tile Display Query dialog box is opened



4. Select an attribute from the list.
5. Click one of the following operator buttons:
   - (<) less than (e.g., to select attributes that have a value less than the entered amount)
   - (>) greater than
   - (<=) less than or equal to
   - (=) equals
   - (>=) greater than or equal to
   - (!=) not equal to
6. Type a value for the attribute in the *Value* field, and click **Enter**.

The search criteria you selected in the dialog box are displayed in the *Query* section.

7. To add other search criteria, click either of these logic operators:
   - **AND**: This operator combines two queries together so the results of both are shown in the Display window.
   - **OR**: This operator searches and displays either one of two search criteria in the Display window.

For example, to select only the tiles with a count more than 2500 soundings that also have a resolution of 500, the query string would be:

```
Count < 2500 AND MinRes = 500
```

8. Repeat Steps 3 through 6 to add as many search criteria as needed.
9. Click **OK**.

The Display window is refreshed to show only the tiles that meet the criteria you entered. The Query string is displayed in the Tile Display Query field in the Properties window.

If you save your session, the query will be saved and restored when that session is loaded again.

To clear the Tile Display Query field:

10. Select the tile layer.
11. Click the **Browse** button in the Tile Display Query field.
12. In the Tile Display Query dialog box, click **Clear**.

This will remove the criteria and display all the tiles for the selected tile layer.

Depth Range Files

Depth range files are text files that set a range of depths for tiling and contouring data. These files are automatically generated during the contouring process (see "Create Contours" on page 361).

The convention for setting depth values is similar to that of other CARIS files:

- Drying heights above the datum are represented as negative values
- Depths below the datum are represented with positive values.

Below is an example of a range depth file..

```
-50
-30
-20
-10
-5
0
5
10
15
20
30
50
```

In the above file the negative values (-30 to -5) are above the datum, and the positive values (5 to 50) are below the datum.

17

Process Imagery Data

Side scan and multibeam backscatter imagery data can be processed directly into mosaics, which can then be examined for anomalies and corrected.

In this chapter...

Overview 412
Workflow for Processing Imagery Data 413
Processing Engines 414
Geocoder workflow for multibeam data 415
Open Mosaic Editor 416
Create GeoBars 417
Beam Pattern Correction 426
Edit GeoBaRs 429
Create Mosaic from GeoBaRs 434
Edit Mosaics... 438
Interpolate Mosaic 442
Sediment Analysis.. 446
Contacts 457

Overview

Multibeam backscatter and side scan sonar data can be processed for the purpose of creating mosaics. But as well as being end products, mosaics can be used as tools to more efficiently process imagery data.

Imagery data processed in Mosaic Editor is stored as **Geo**referenced, **Ba**ckscatter **R**asters, or GeoBaRs. GeoBaRs are the basis for all mosaics created in HIPS and SIPS

A GeoBaR is essentially a mosaic of imagery data for a survey line, which can be edited in georeferenced space. Once imagery has been corrected a full mosaic can be compiled from the data. Sediment analysis can be performed on data processed with the Geocoder engine.

Most users will be able to perform all their imagery processing directly in Mosaic Editor.

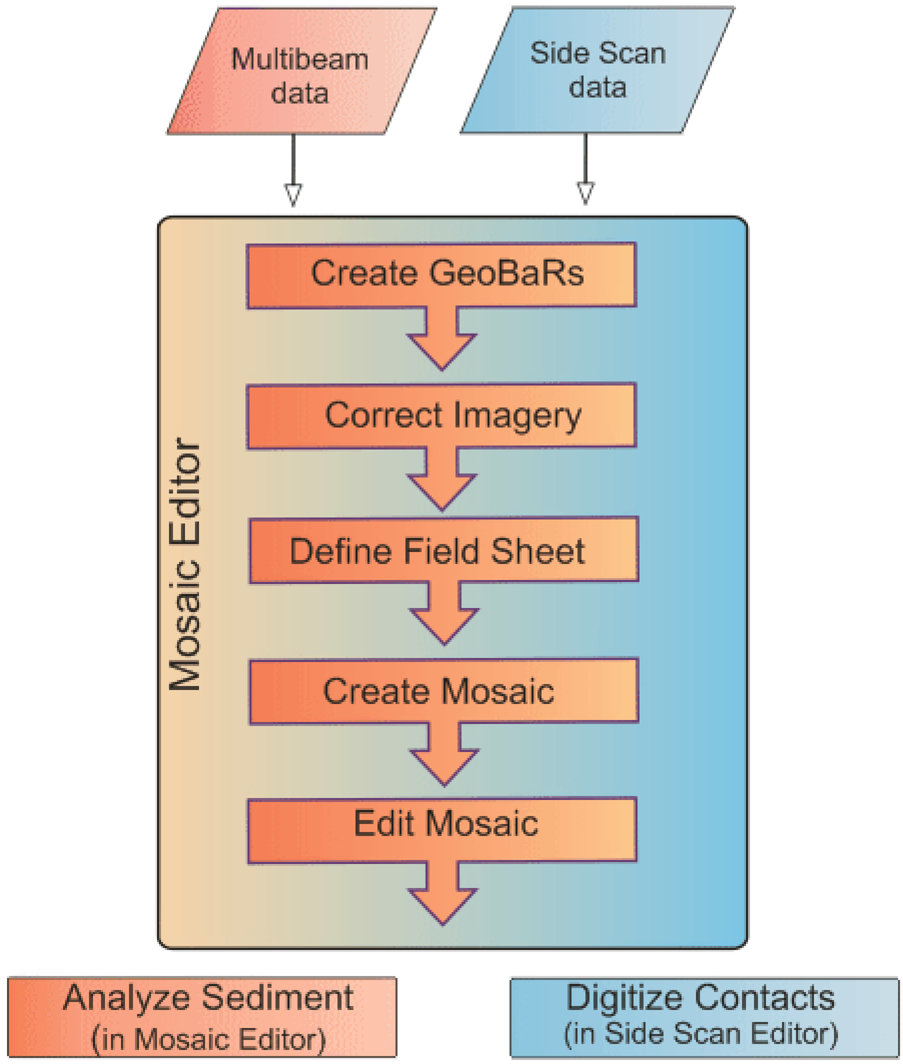
However, if you:

- need to edit altitude data for side scan data, or
- will be digitizing and editing contacts, or
- prefer to use a waterfall display for editing, or
- intend to apply advanced TVG controls

then these functions must be performed in Side Scan Editor. (This is described in the SIDE SCAN EDITOR section of the Editors Guide.)

Workflow for Processing Imagery Data

There are differences in the workflow depending on the kind of data being processed in Mosaic Editor. However, the basic workflow is the same:



For multibeam and backscatter processing, see "GEOCODER WORKFLOW FOR MULTIBEAM DATA" ON PAGE 415.

> Multibeam data must be fully processed and merged before creating mosaics. Side scan data must have digitized altitude before creating mosaics.

For side scan processing in Side Scan Editor, see "SIDE SCAN EDITOR WORKFLOW" ON PAGE 50 of the Editors Guide.

Processing Engines

Mosaic Editor has two processing engines: Geocoder Engine and SIPS Engine.

Geocoder Engine

Geocoder processes multibeam backscatter, both beam-averaged and the higher resolution time-series returns, as well as side scan sonar data.

The Geocoder engine has multibeam-optimized routines and data handling, such as automated gain corrections, suitable for normalizing data from specific sonar types with varying acquisitions settings, as well as automatic slope correction using inherent multibeam bathymetry.

> Multibeam data must be converted to HIPS format, processed with the bathymetric editors (Sound Velocity Correction, Apply Tide etc.— see Multibeam "Workflow diagram" on page 16), and Merged (see "Merge" on page 167) before the imagery can be corrected in Mosaic Editor.

SIPS Engine

The new Side Scan Editor workflow configures correction values that are retained and can be used to create mosaics from lines selected in the Display window, without the need to create GeoBaRs. However the option to create GeoBaRs and mosaics from them is still avaiable.

Corrections which originally were options when mosaics were created with the SIPS engine, are now automatically applied, based on settings in the Side Scan Editor Properties window.

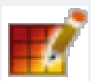
Geocoder workflow for multibeam data

Main steps to process backscatter imagery

- convert data to HIPS
- edit navigation, gyro, heave, load tide, compute TPU etc.
- Merge
- open Mosaic Editor
- create at least one GeoBaR using only default corrections
- create a Beam Pattern correction file (using patch test data if available)
- set options for processing:
    - source data type
    - auto corrections
    - beam pattern corrections
    - despeckle strength
- create GeoBaRs applying desired corrections
- edit GeoBaRs to adjust the imagery “on-the-fly”
- generate Mosaics from selected edited GeoBaRs
- edit Mosaics - adjust the display of the imagery and the contributing GeoBaRs
- Sediment Analysis - analyze sea bottom sediment from files generated by the Geocoder
- digitize contacts in Side Scan Editor

Open Mosaic Editor

To open Mosaic Editor:

Menu	Tools > Mosaic Editor > Open
Tool	

1. Select the Open Mosaic Editor command.

The Mosaic Editor and Beam Pattern toolbars are displayed. The Control Window now displays the three control tabs of Mosaic Editor.

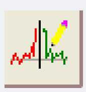
The Mosaic Editor and Beam Pattern toolbars are also displayed.

For a description of the Mosaic Editor interface, see MOSAIC EDITOR in the Editors guide.

Side Scan Editor

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

1. Open Mosaic Editor.

Menu	Tools > Side Scan Editor > Open
Tool	

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

The Side Scan Editor window will open, displaying the waterfall view.

Create GeoBars

GeoBaRs are created using the options selected in the Process tab of the Mosaic Editor. Different options are available, depending on the processing engine selected.

You can create multiple GeoBaRs on a survey line to:

- Correct multiple sources of data (i.e., time series, beam averaged, and/or side scan) from the same file, if applicable.
- Have different resolutions per GeoBaR.
- Visually compare data.

The simplest way to create a GeoBaR is to use the default values displayed on the Process tab, and then view and correct the GeoBaR imagery by adjusting these values. Then you can apply these settings from the created GeoBaR to other track lines.

GeoBaRs are displayed in the Display window and are listed by name in the Layers tab. A GeoBaR has three child layers:

- **Intensity**: displays the current intensity (including edits).
- **Original Intensity**: displays the intensity when the GeoBaR was created (before edits to GeoBaR).
- **Weights**: displays the computed weights based on sonar geometry. These values are used when compiling a mosaic based on the Auto-Seam or Full Blend methods.

GeoBaRs remain open when Mosaic Editor is closed. Mosaic Editor will recognize GeoBaRs that are already open and will not try to load an open GeoBaR.

Naming GeoBaRs

Each GeoBaR should be given a unique filename, particularly when there will be many created in a project. A convenient way to standardize naming, and at the same time ensure unique file names for all GeoBaRs, is to use formatting tags combined with text.

> If you try to create a GeoBaR with the same name as an existing one, you will be warned that the new GeoBaR will overwrite the older one, and be given the option to proceed.

Formatting tags available for GeoBaR naming are:

Formatting Tag	Adds to GeoBaR name:
%Y	Current year (YYYY)
%M	Current month (MM)
%D	Current date (DD)
%h	Current hour (hh) based on 24-hour clock
%m	Current minute (mm)
%s	Current second (ss)
%j	Current Julian date (jjj)
%p	Project name
%v	Vessel name
%d	Data date, e.g., 2007-244
%l	Line identifier, e.g., 244-2327
%i	Imagery type selected in the *Source Data Type* field (SS =side scan, TS= time series, BA=beam average)
%e	Processing engine selected (SIPS/Geocoder)
%r	Value and units from the *Resolution* field (Note: a decimal point is represented by underscore, for example, 4_5m is 4.5.

Tags will automatically add strings of identifying information such as the date and time the GeoBaR is created.

For example, this string:
`%Y-%M-%D(%h’%m”%s)_%e(%i@%r)`

will name the GeoBar as:
`2009-10-21(11’13”29)_Geocoder(TS@1.0m).`

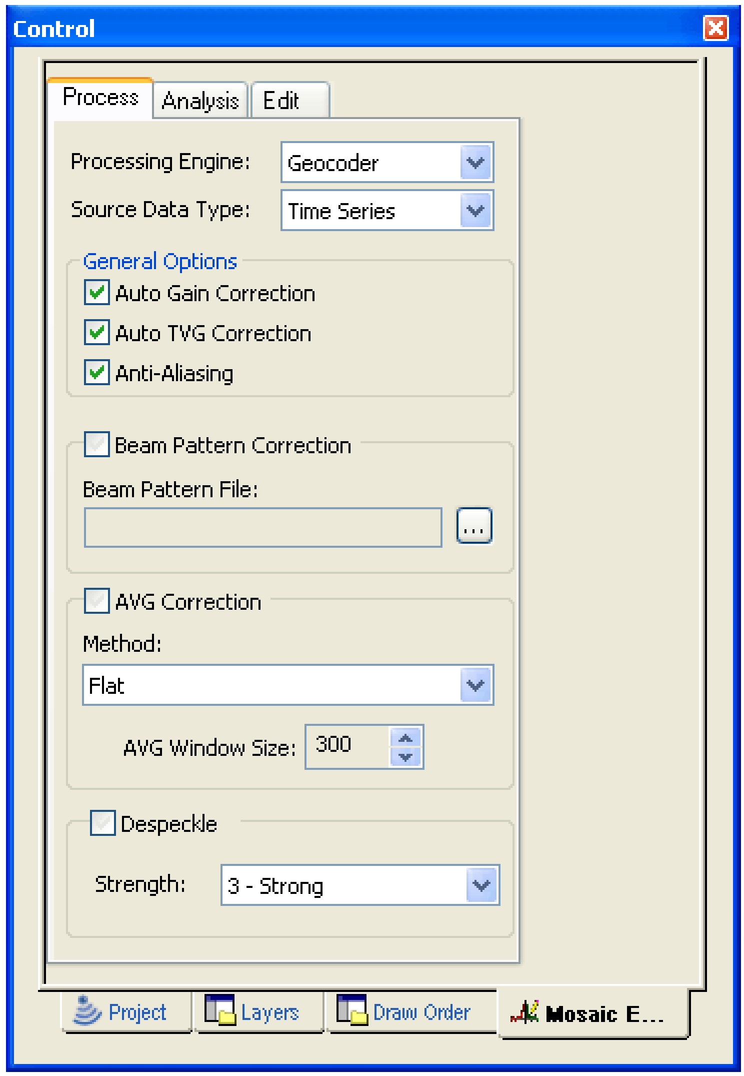
This example can be used repeatedly to provide a standardized series of names differentiated by timestamp.

> GeoBaR names must not contain any spaces.

Create GeoBaR with the Geocoder Engine

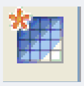
To create a GeoBaR:

1. Open Mosaic Editor.
2. Select a survey line (or lines) in the Display window.



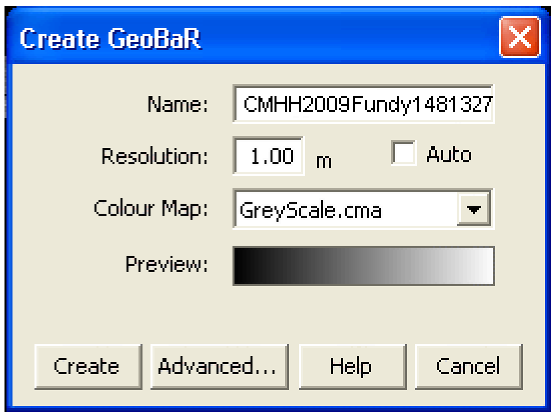
3. In the Process tab, select Geocoder from the *Processing Engine* drop-down list.
4. Select the type of data being processed: Side Scan, Beam Average or Time Series.
5. By default, the *General Options* of *Auto Gain, Auto TVG* and *Anti-Aliasing* are selected. It is recommended that these automatic correction options be left on.
6. [Optional] Select the Beam Pattern Correction check box, and select a Geocoder Beam Pattern Correction file (*.bpt). (If no file is available it will need to be created.) See “Beam Pattern Correction” on page 426.

7. [Optional] Set the AVG Correction check box and select a method from the drop-down list.
8. Set the AVG window size using the arrow buttons.)
9. [Optional] Select the Despeckle check box and select a strength of adjustment to filter out pixels with intensity values inconsistent with their neighbours.

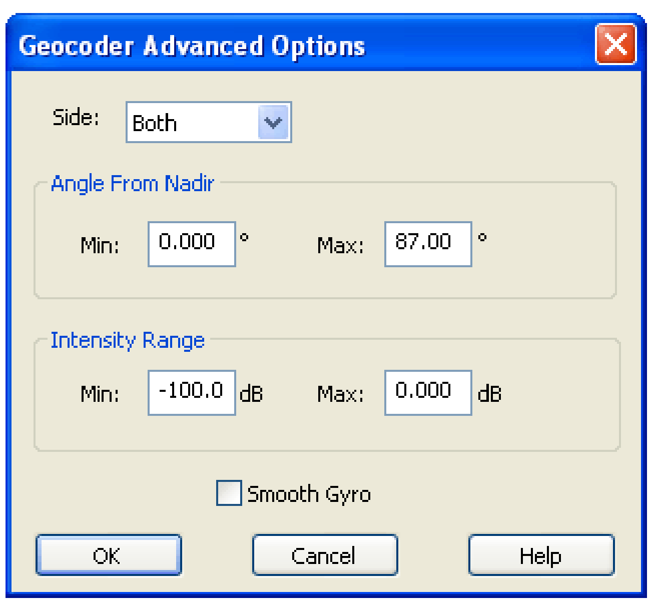
Menu	Tools > Mosaic Editor > Create GeoBar
Tool	

10. Select the Create GeoBar command.

The Create GeoBaR dialog box is displayed.



11. Type a name for the GeoBar. The name must not include any spaces. (See “Naming GeoBaRs” on page 418.)
12. Type a resolution value. With the Geocoder engine you have the option of selecting *Auto* to have an optimal resolution set automatically based on the beam footprint.
13. [Optional] Select a colour map for the new GeoBaR. The colour map will be displayed in the *Preview* field. The default map is GreyScale.
14. [Optional] Click **Advanced** for other options.



In the Geocoder Advanced Options dialog box:

15. Select Port, Starboard or Both sides on which to create a GeoBaR.
16. Set minimum and maximum values for the range of the *Angle from Nadir*. Default range is from minimum of 0 to maximum of 87 degrees.
17. Set minimum and maximum values in dB for the *Intensity Range*, for example, a minimum of -100 and maximum of 0. (Intensity values are expressed as negative numbers.)
18. Select *Smooth Gyro* to apply gyro that has been smoothed in Attitude Editor.

In some cases it may be more useful to maintain the full intensity range when creating the GeoBaR, and limiting this range only in the final Mosaic.

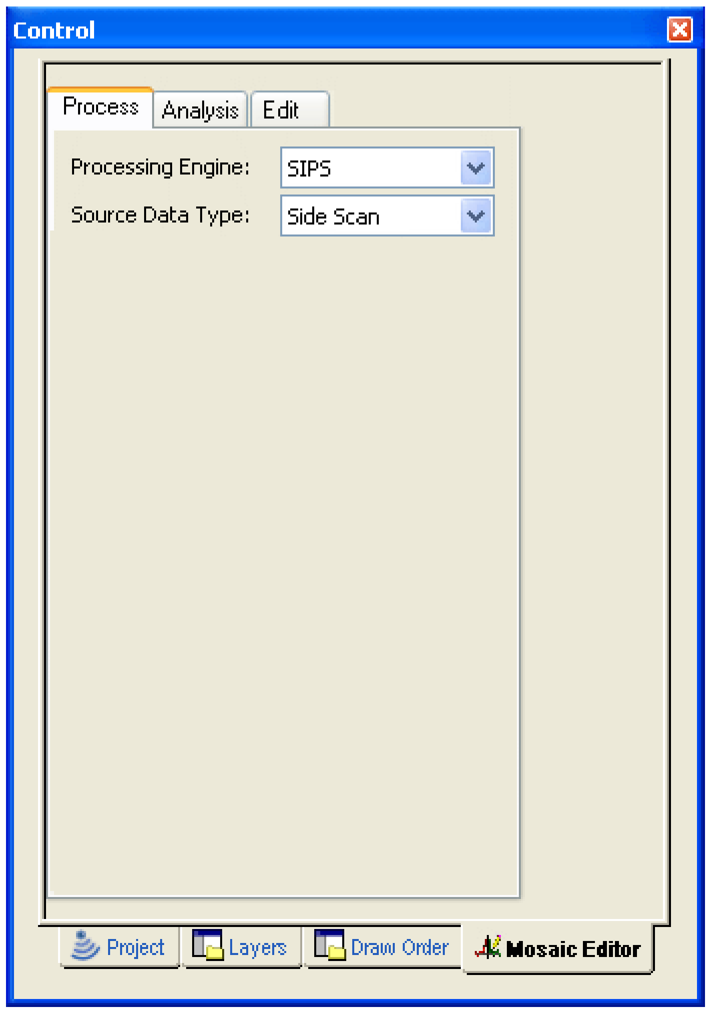
19. Click **OK** to return to the Create GeoBaR dialog box.
20. Click **Cancel** to return to the Create GeoBaR dialog box, discarding any changes that were made.
21. Click **Create**. The new GeoBar is displayed and listed in the Layers tab.

For description of GeoBaR layer properties, see “Properties of Mosaics and GeoBaRs” on page 23 of the Editors reference.

Create GeoBaRs with the SIPS Engine

Corrections which originally were options when mosaics were created with the SIPS engine, are now automatically applied, based on settings in the Side Scan Editor Properties window.

1. Select a line or lines in the Display window.
2. On the Mosaic Editor Process tab, select SIPS from the *Processing Engine* drop-down list.



3. Select the type of data being processed: Side Scan, or Beam Average.
4. Select the Create GeoBar command.
5. .In the Create GeoBar dialog box, type a name for the GeoBar.

Menu	Tools > Mosaic Editor > Create GeoBar
Tool	

GeoBaRs must have unique names. If you try to create a GeoBaR with the same name as an existing GeoBaR you will be warned that the new GeoBaR will overwrite the older one. See “Naming GeoBaRs” on page 418 for more information.

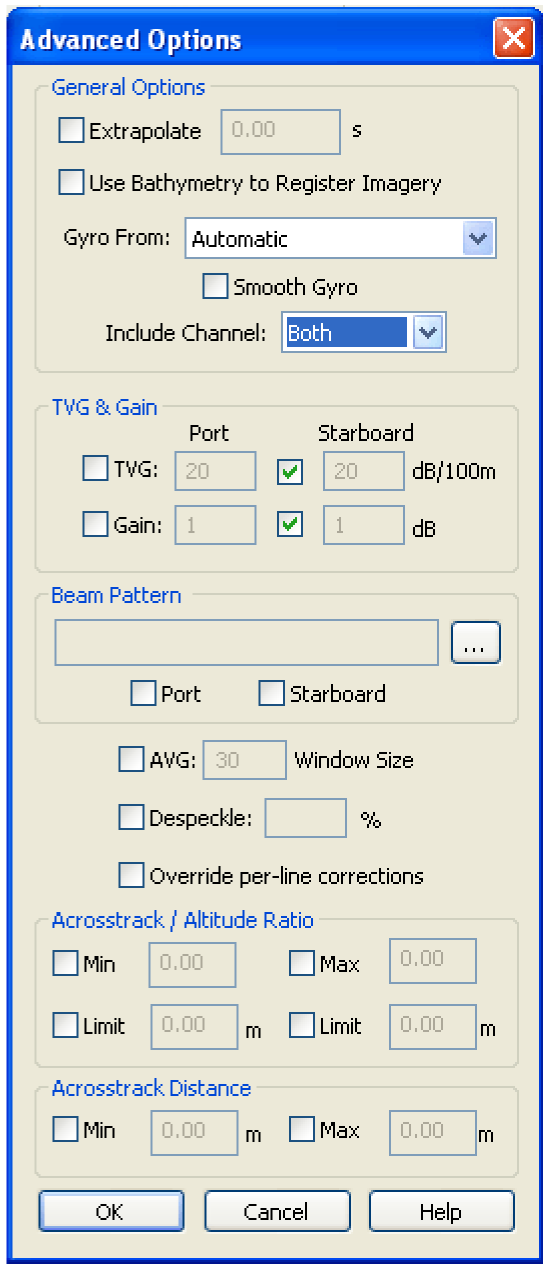
6. Type a resolution value, or accept the default value. (The *Auto* function is only available when using the Geocoder engine.)

7. [Optional] Select a colour map for the new GeoBaR. The colour map will be displayed in the *Preview* field. The default map is GreyScale.

The imagery corrections that are applied in the creation of GeoBaRs with the SIPS engine, are those which are set in the Properties tab in Side Scan Editor.

To modify these settings:

8. Click **Advanced** to set or change correction options.



SIPS Engine Advanced Options settings:

General Options	
Extrapolate	Select Extrapolate to extend the heading and/ or navigation data at each end of the line to a maximum specified by the selected value.
Use Bathymetry to Register Imagery	Select TOuse the bathymetry location data from the same swath to register the side scan imagery. If not selected, a flat seafloor assumption will be used. (This function is only applicable to systems which provide bathymetry and imagery information at the same time in the same swath.)
Gyro From:	Select source from the drop-down list Select the orientation source from the *Gyro* drop-down list: Automatic, Course Made Good, Ship Gyro or Towfish Gyro.
Smooth Gyro	Select *Smooth Gyro* to apply gyro that has been smoothed in Attitude Editor.
Include Channel	Select Port or Starboard, or both.
TVG & Gain	
TVG	Select the boxes to apply to Port or Starboard or both. • Type values in dB/100 m in the fields
Gain	Select the boxes to apply to Port or Starboard or both . • Type values in dB/100 m in the fields
Beam Pattern	Click **Browse** to select the Beam Pattern file to apply. • Check the boxes to apply to Port or Starboard or both.
Gain Normalization	Enable and set the window size for the moving average filter.
Despeckle	Set the Despeckle threshold value.
Override per-line corrections	Select this option to apply these correction settings to all lines in the mosaic.
Acrosstrack/Altitude Ratio	The Acrosstrack/Altitude Ratio excludes data starting from nadir extending across track to a distance determined by the towfish altitude and the given ratio.
Min/ Max	Select the check boxes to activate the value fields so you can type in desired values
Limit	Select the check boxes to activate the value fields so you can type in desired values

Acrosstrack Distance	The *Acrosstrack Distance* option can also be set to exclude a fixed distance across track, regardless of altitude.
Min Max	Select the check boxes to activate the value fields so you can type in desired values

9. Click **OK** to return to the Create GeoBaR dialog box
10. Click **Cancel** to return to the Create GeoBaR dialog box, discarding any changes that were made.
11. Click **Create**.

The new GeoBaR will be displayed in the Display window, and listed by name in the Layers tab.

For description of GeoBaR layer properties, see "PROPERTIES OF MOSAICS AND GEOBARS" ON PAGE 23 of the Editors reference.

Beam Pattern Correction

Beam Pattern Correction is applied to remove acoustic artifacts from the imagery caused by imperfections in the sonar. Beam pattern correction relies on a user-generated beam pattern file to identify and remove this effect. Once the beam pattern file is created it can be applied to the survey data to uniformly correct the selected data. This also results in a more consistent appearance in imagery.

Applying Beam Pattern Correction with the Geocoder engine selected removes sediment artefacts and multibeam sonar system artefacts from data. Angular artefacts are removed from side scan data if the SIPS engine is selected.

For consistent results, you should create your beam pattern file over as flat, homogenous, and featureless an area as possible, preferably sand. For best results, it is recommended that a “patch test” be done during a survey over an area known to have such properties.

The process to apply Beam Pattern Correction in Mosaic Editor consists of:

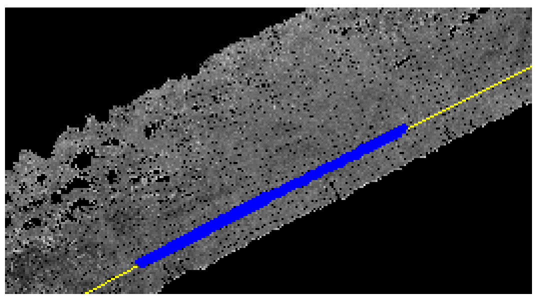
- selecting the processing engine: Geocoder or SIPS
- creating a GeoBaR from the patch line, preferably with only the default corrections applied.
- creating the Beam Pattern file
- setting the Beam Pattern options for the type of data you are correcting
- creating GeoBaRs and applying Beam Pattern Correction

Create Beam Pattern file

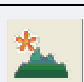
1. Make sure the Ship Track Lines layer is active in the Layers tab.
2. Load one or more GeoBaRs.
3. In the Display window, select a trackline and then *superselect* an area of the line by pressing the down arrow key on the keyboard while range-selecting over a suitable area of the GeoBaR.

> In order to create a robust beam pattern file, a minimum number of pings should be selected (usually 250), so that sufficient data is included for statistical analysis.

The superselected area will be highlighted in blue at nadir of the GeoBaR, as in the example below.



4. [Optional] Query the superselected area to determine the number of pings in the superselection.
5. [Optional] From the data in the Selection tab, subtract the value of the End ping from the Start ping.
6. Select the Geocoder or SIPS processing engine in the Process tab of Mosaic Editor.
7. Select the Create Beam Pattern command.

Menu	Process > Beam Pattern Correction > Create
Tool	

   (The Create Beam Pattern command is only active in Mosaic Editor while part of a line is superselected.)

The Save Beam Pattern Correction file dialog box is displayed.

8. Name the file and click **Save** to save and load the file.

The created file will have the extension *.bp.

Despeckle

Imagery may have isolated pixels with intensity levels that are inconsistent with neighbouring pixels. It is possible to filter the isolated light and/or dark pixels through the Despeckle process. This will produce smoother, more consistent imagery.

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

Despeckling uses a calculated value based on neighbouring intensity levels to replace the current pixel's intensity if it is outside the specified range.

To apply the Despeckle tool:

1. Select the Despeckle check box.

When using the Geocoder engine, the value calculated from the strength of neighbouring intensities.

2. Select a value from Weak to Very Strong from the Strength drop-down list.

When using the SIPS engine, the despeckle value is derived from applying a mean filter or a median filter with a specific threshold, calculated from a matrix of neighbouring pixels. The default values are to apply a Mean Filter with a 3x3 matrix cross and a threshold of 100%.

To change the default options:

1. Click **Change**.
2. Select the Median Filter from the *Method* drop-down list.
3. Use the up and down arrows to select a *Threshold* for defining an acceptable range in which the pixel value will not be replaced.

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

4. Select a pattern of neighbouring pixels from the *Matrix Size* drop down list be used in calculating the mean/median.
5. Click **Set** to return to the Process tab.

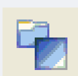
Edit GeoBaRs

Editing GeoBaRs includes adjusting brightness, contrast, visibility options and the draw order of GeoBaRs. These corrections are done with the tools on the Edit tab.

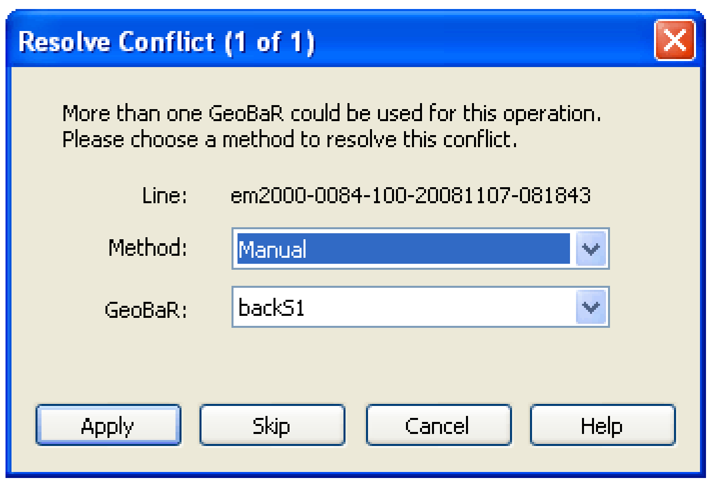
> The Edit tab is activated when you load a GeoBaR for a selected line. If there is more than one GeoBaR on a line, the Edit tab is activated when you select a specific GeoBaR on that line, using the Resolve Conflict dialog box.

Editing is done one GeoBaR at a time.

To open a GeoBaR for editing:

Menu	Tools > Mosaic Editor > Load Single GeoBaR
Tool	
Menu	Tools > Mosaic Editor > Load all GeoBaRs
Tool	

1. Select the survey line for the GeoBaR you want to edit.
2. Use a Load GeoBaR command to load a GeoBaR for the line.
3. If there is more than one GeoBaR for the selected line, use the Resolve Conflict dialog box to chose the GeoBaR to be edited. See “Conflict Resolution” on page 444.



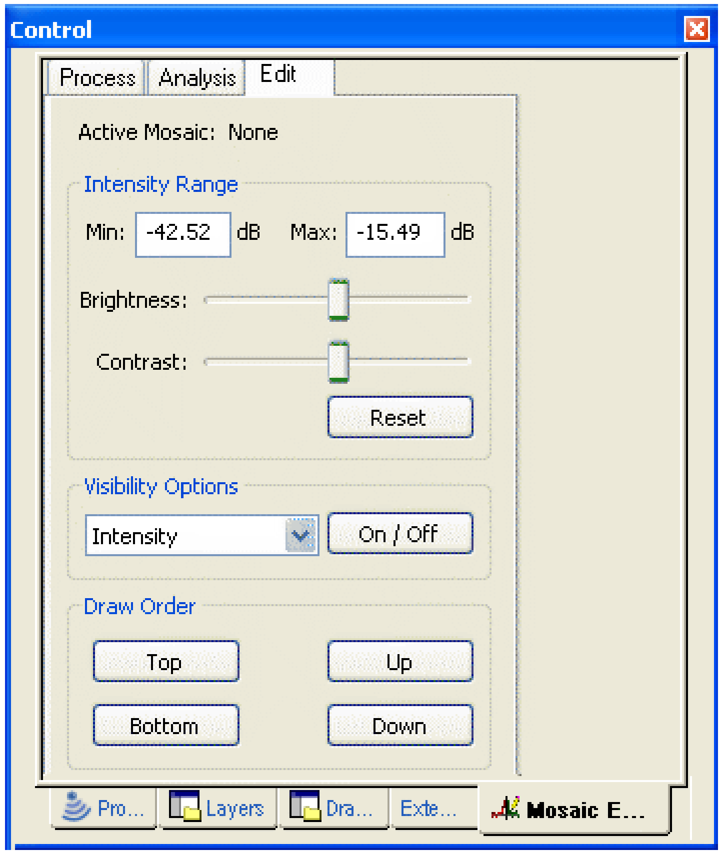
This will activate the functions on the Edit tab.

You can also edit GeoBaRs that are already loaded and visible in the Display window.

1. Select the survey line for the GeoBaR you want to edit. (View the Properties for the GeoBaR to determine the source line.)

This will activate the functions on the Edit tab.

If there is more than one GeoBaR loaded for the selected line, the Resolve Conflict dialog box will open so you can select the one to edit.



Editing

The Intensity Range fields show the minimum and maximum intensity values for the selected GeoBaR.

1. To adjust brightness, move the sliders to the right or left to change the intensity of the image.
2. To adjust contrast, slide the Contrast slider to the right for more contrast and to the left for less contrast.

As you move the sliders, new values are displayed in the Intensity Range fields.

Use **Reset** to restore the GeoBaR to the original intensities it had when it was created.

3. Use the Visibility Options to turn on or off the various child layers of the GeoBaR.
4. Manipulate the *Draw Order* of the GeoBaR using the **Top**, **Bottom**, **Up** and **Down** buttons. This will change the layer at which the GeoBaR is drawn in the Display window. (This function can be important if you intend to make your final mosaic using the *Overwrite* method.)

Image Classification

Image classification groups return-signal intensities (represented in the mosaic as pixel values) into classes. These classes are displayed in the mosaic using specific colours.

To determine the range of intensities:

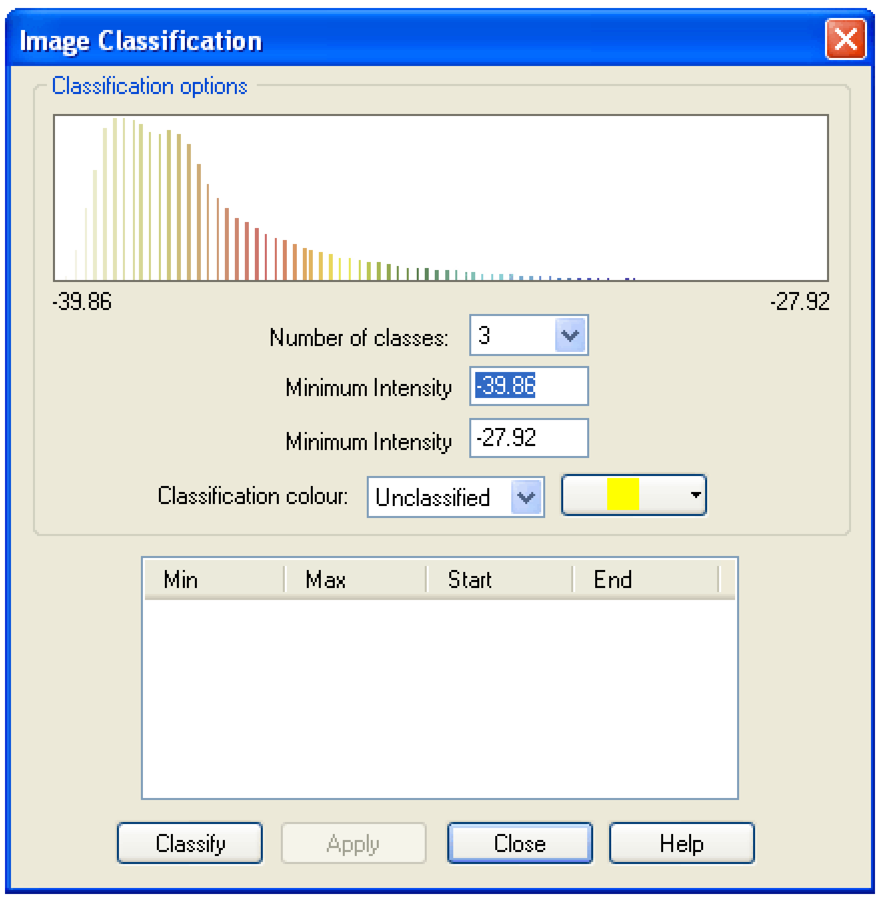
1. Select the Intensity layer of the mosaic in the Layers tab.
2. In General field of the Properties window, view the Min/Max Ranges for the layer.
   - You can also move the cursor across the mosaic in the Display window. As you move the cursor, screen tips show the inensity values in dB.

To apply Image Classification:

3. Select the Mosaic layer.
4. Select the Classify command to open the Image Classification dialog box.

Popup	Classify

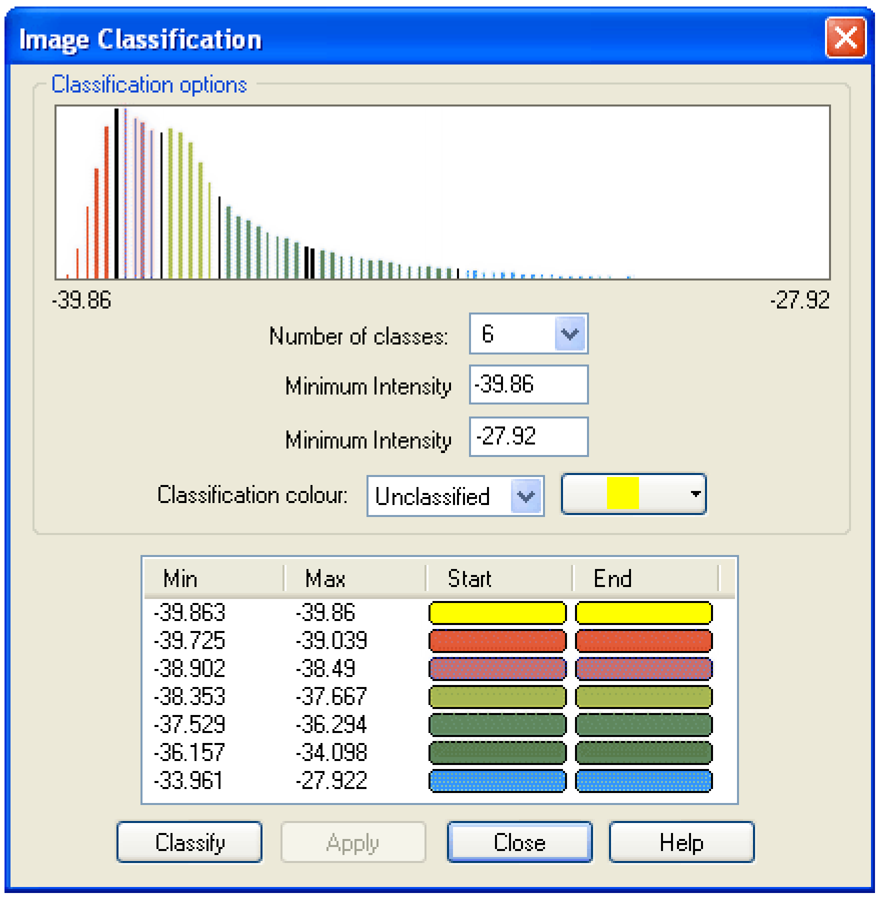
The Image Classification dialog box is displayed.



5. Use the *Number of classes* drop-down list to set the total classes in the mosaic.

6. Set a range of values for a class by typing values for *Minimum intensity* and *Maximum Intensity* in their respective fields.
7. [Optional] Select a colour for a class by selecting a class from the *Classification* colour drop-down list, and the colour picker.
8. This colour will be displayed beside the class name.
9. [Optional] Select another colour from the colour picker, or create a custom colour from the standard Windows colour palette.
10. Repeat Steps 4 to 7, as needed.
11. Click **Classify**.

A colour table is generated according to the number of classes you selected in Step 3. Pixels are grouped according to class starting from the minimum to the maximum pixel value.



The histogram is refreshed to show the pixels coloured according to the class designation. When the dialog box is closed, the intensity values in the mosaic will be displayed with these colour classifications.

To save the colours and their classes to a .CRF (colour range file) for future use:

12. Click **Apply**.
13. In the Save As dialog box, type a name for the file and click **Save**.

Delete GeoBaR

To delete a GeoBaR:

1. Right-click on the GeoBaR layer in the Layers tab.
2. Select Delete GeoBaR from the pop-up menu.

This deletes the GeoBaR permanently from your disk.

Create Mosaic from GeoBaRs

A mosaic can be created from two or more GeoBaRs. As with other HIPS and SIPS surfaces, a mosaic is created in a field sheet.

(Mosaics of side scan data can also be created directly from a selection of track lines. See "Create Mosaics" on page 78.)

To create a mosaic from GeoBars in Mosaic Editor:

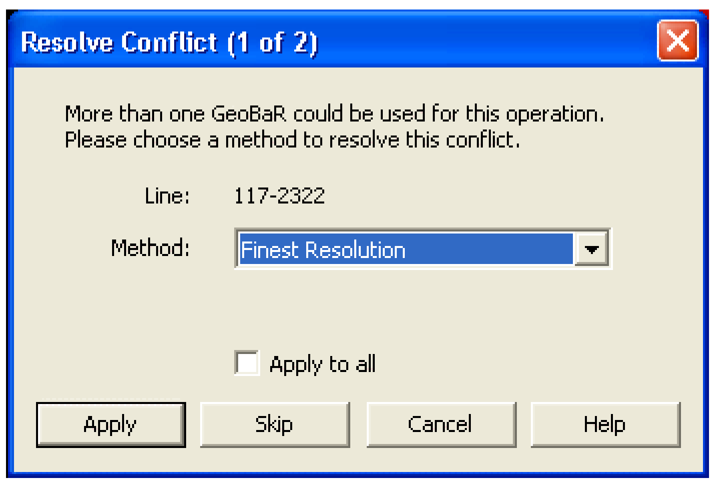
1. Open or create a field sheet that encompasses the area you want to mosaic.
2. Select the Ship Track Lines layer in the Layers tab.
3. Select two or more track lines for which GeoBaRs have been created.

The Create Mosaic command and toolbar button are activated.

Menu	Tools > Mosaic Editor > Create Mosaic
Tool	

4. Select a Create Mosaic command.

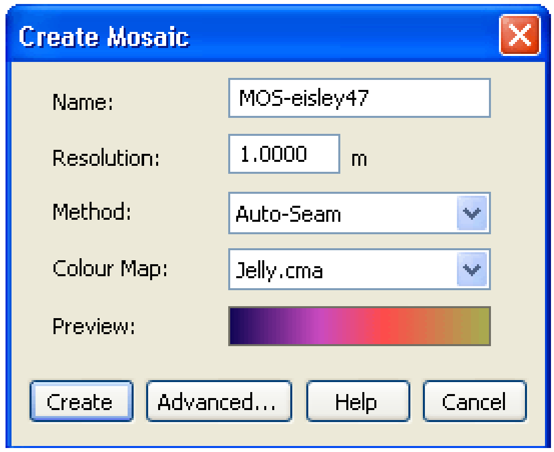
Mosaics are created using one GeoBaR per line. When there is more than one GeoBaR loaded on a selected line, the Resolve Conflict dialog box will appear so you can choose which GeoBaR you want to include in the mosaic.



There are three methods to resolve a conflict: Newest GeoBaR, Finest Resolution, and Manual. (For more information on conflict resolution, see "Conflict Resolution" on page 444.)

5. Select a method to resolve the conflict.

When any conflicts in selecting GeoBaRs are resolved, the Create Mosaic dialog box is displayed.



Name the mosaic.

Every mosaic should be given a unique filename. A new mosaic created with the same name as an existing one will overwrite the older mosaic.

Names must not contain any spaces.

A convenient way to standardize naming, at the same time as ensuring unique file names, is to use formatting tags combined with text.

Formatting Tag	**Adds this to the mosaic name**
%Y	Current year (YYYY)
%M	Current month (MM)
%D	Current date (DD)
%h	Current hour (hh) based on 24-hour clock
%m	Current minute (mm)
%s	Current second (ss)
%j	Current Julian date (jjj)
%r	Value and units from the *Resolution* field (Note: a decimal point is represented by underscore, for example, 4_5m is 4.5 metres.

Using tags will automatically add strings of identifying information such as the date and time the mosaic was created.

For example, this string:
`Fundy_%Y-%M-%D(%h’%m”)_%r`

will name the mosaic:

`Fundy_2010-04-24(14’13”)_@1.0m.`

The above example tag string could be used repeatedly to provide a standardized series of names that would be differentiated by timestamp.

6. Type a name for the mosaic

Image Resolution

If one or more of the GeoBaRs to be included in the mosaic have a lower resolution than what you set for the mosaic, the mosaic you create may have empty cells. To avoid this, choose one of these options from the Mosaic Creation warning dialog box:

- keep the resolution you have set for the mosaic, and re-sample the GeoBaRs with the lower resolution, or
- keep the resolution you have set for the mosaic, and use all GeoBaRs regardless of resolution, or
- keep the resolution you have set for the mosaic, but don’t include the GeoBaRs with lower resolution, or
- set the resolution for the mosaic to that of the lowest resolution GeoBaR.

7. Enter a resolution value in metres.

Smaller values will result in finer resolution than larger values, but involve a longer creation time and larger file size.

Overlapping data

GeoBaRs often overlap. You can set how you want any overlapping data to be handled in the mosaic. There are five methods to choose from:

- Auto-Seam: This method determines which pixels have higher significance when there is overlap. A weighting function assigns lower significance to pixels at the edge of the swath than to pixels near the centre. The first few pixels near the nadir are assigned low significance due to the low resolution in that region.
- Full Blend: This method uses Geocoder’s blend operation to smooth pixel values. It also relies on a weighting scheme.

> Auto-Seam and Full Blend use the Weights layer generated for each GeoBaR. Auto-Seam picks one pixel or the other when there is overlap; Full Blend takes a weighted average of the two.

- Overwrite: Uses the draw order to decide which pixel will be used in an overlapped area.
- Shine Through: The highest intensity pixel will be used.
- Underlay: The opposite of Overwrite. The pixel is added only if no other point exists.

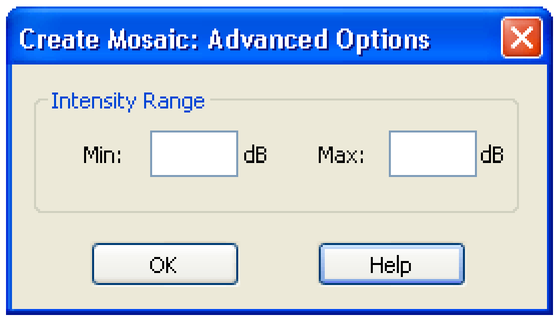
8. Select a method to process overlapping data from the drop-down list.

You can set the display colours for the mosaic.

9. Select a colour map from the drop down list. The range of colours will be displayed in the Preview field below.

When a GeoBaR is created, limits can be set on the intensity range. In some cases it may be more useful to maintain the full intensity range in creating the GeoBaR, then limiting this range in the final Mosaic.

10. [Optional] Click **Advanced** to set intensity options.



11. Set minimum or maximum values in dB for the *Intensity Range*. This will exclude nodes that fall outside of the range defined by the min/max values.
12. Click **OK** to return to the Create Mosaic dialog box.
13. Click **Create**.

The mosaic that is created will appear in the Layers tab, with its colour-coded child layers: Intensity, Original Intensity and Weights (as for GeoBaRs) plus a layer called Contributor.

Delete Mosaic

To delete a Mosaic:

1. Right-click on the Mosaic layer in the Layers tab.
2. Select Delete Mosaic from the pop-up menu.

This will permanently delete the mosaic from your disk. (It does not delete the component GeoBaRs.

Edit Mosaics

Mosaics can be edited for brightness, contrast, and visibility. As well, component GeoBaRs can be added, modified or removed.

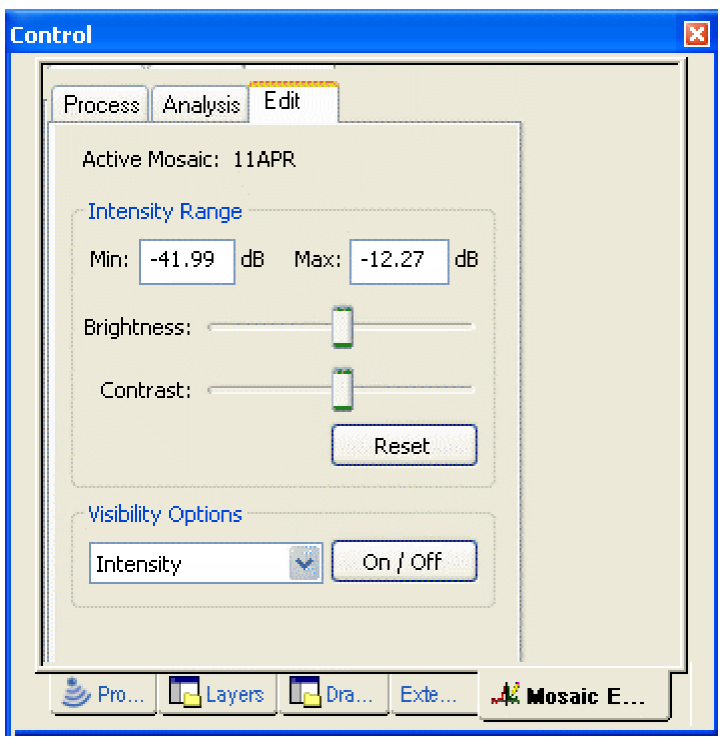
While you may have more than one mosaic open in the display, only one mosaic can be edited at a time.

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

To edit a mosaic:

1. Open Mosaic Editor.
2. Right-click on a mosaic layer in the Layers tab of the Control window.
3. Select Edit Mosaic from the pop-up menu.

The functions on the Edit tab are displayed. The top line of the Edit tab displays the name of the mosaic being edited



1. To adjust brightness, move the sliders to the right or left to change the intensity of the image.
2. To adjust contrast, move the Contrast slider to the right for more contrast and to the left for less contrast.

As you move the sliders, the new values are displayed in the Intensity Range fields.

3. Use **Reset** to restore the values of the Original Intensity layer. (Removes the effects of the adjustments made with the sliders.)
4. Use the Visibility Options to turn on or off the various child layers of the mosaic.
   - Select the layer (Intensity, Original Intensity or Weights) from the drop-down list and click **On/Off**.

Edit Mosaic Components

GeoBaRs can be added to and removed from an existing mosaic, and the way in which the component GeoBaRs contribute to a mosaic can modified.

To edit a GeoBaR that is a component of a mosaic:

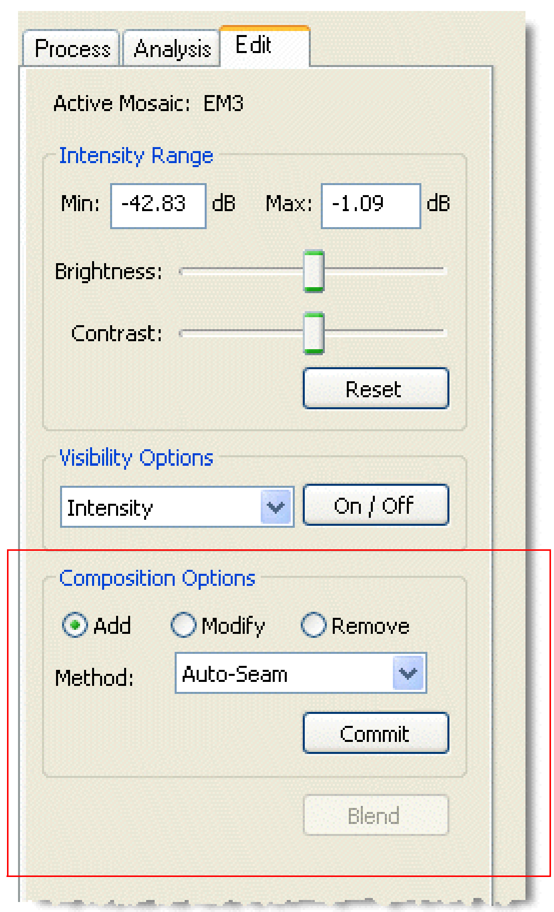
1. Open the field sheet containing the mosaic.
2. Right-click on the mosaic layer in the Layers tab, and select Edit Mosaic from the pop-up menu.

This will activate the Edit tab of Mosaic Editor. The top line of the tab will display the name of the active (selected) mosaic.

3. With the Edit tab activated, select the Ship Track Lines layer in the Layers tab.
4. In the Display window, select a line that is part of the mosaic.

Since mosaics are created from one GeoBaR per line, selecting a line automatically selects the component GeoBar, whether or not that GeoBaR has been opened.

The Edit tab is updated to display *Composition* options for the selected component.



The methods available to edit the components of a mosaic are the same as used to create the mosaic: Auto-Seam, Full Blend, Overwrite, Shine Through, and Underlay.

Also available is the blending method used to create the original mosaic. This is listed as Default.

Add a new component

To add another GeoBaR to the mosaic:

1. Select an open GeoBaR that is not a ready part of the mosaic.
2. In the Edit tab, select the *Add* option.
3. Select a method for processing overlap from the drop-down list.
4. Click **Commit**.

Modify Components

To modify the method by which a component GeoBaR contributes to the mosaic:

1. Select a line that is part of the mosaic.
1. Select *Modify*.
2. Select the preferred method from the drop-down list.
3. Click **Commit**.

Remove GeoBaR from mosaic

To remove a component from a mosaic,

1. Select *Remove*.
2. Select an option from the drop-down list:

- *Remove Complete* will re-mosaic the portion of the mosaic covered by selected GeoBaR, without any contributions from the selected GeoBaR.
- *Remove Explicit* will keep any blended pixels, but remove any pixels that are explicitly from the selected line.

3. Click **Commit** to save the changes.

Manual Blend

You can select a GeoBaR and perform a manual blending operation with it.

1. Click **Blend** to open a slider dialog that you can use to blend a GeoBar into the existing mosaic.
2. Move the slider to see the effects of blending. When you have a desired blend, click **Commit**.
3. To back out of the Blend function without saving, click **Cancel**.

Other functions

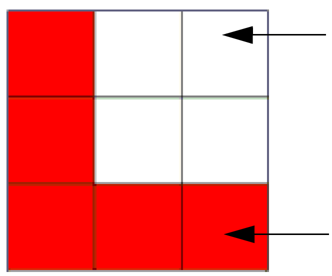
- When a component GeoBaR is opened in Edit tab, the Intensity Range fields display the minimum and maxiumum intensity values for the GeoBaR. You can adjust these values for the selected GeoBaR as you would for a mosaic. See "Edit Mosaics" on page 438
- You can use Compute Statistics command to generate the statistics for a mosaic attribute layer, and output them to an ASCII file or to image. See "Compute Surface Statistics" on page 232.

Interpolate Mosaic

The Interpolate Surface command eliminates small holes that appear in areas of sparse data, for example, in the outside edges of surveyed areas where there is little or no overlapping coverage.

This command examines each node in the mosaic surface to determine if it contains a pixel value. If the node does not contain a value, the neighbouring pixels are examined to determine if enough of them contain pixel values to justify interpolating a value for the hole. This limits the interpolation to holes in the mosaic surface and prevents it from expanding the mosaic outward from the survey area. A copy of the mosaic surface, with these gaps filled, is then created.

In the following example, in a 3 x 3 surface node area, five nodes have pixel values (shown in red).

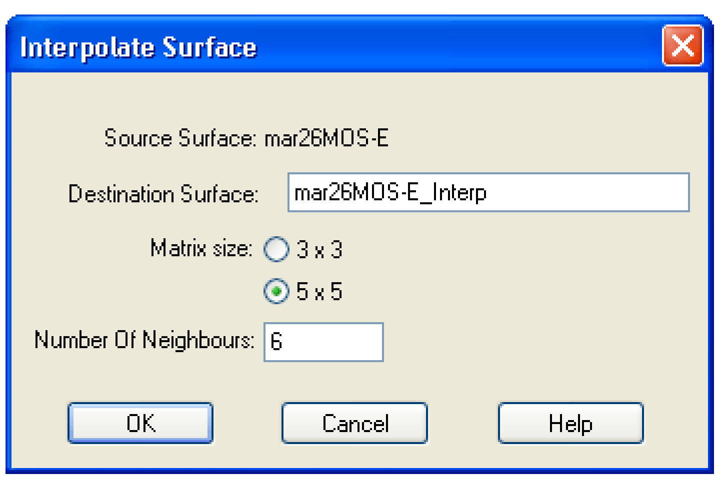


This means if you select five or less as the value for the *Number of Neighbours*, a pixel value is created for the node without a value from the neighbouring nodes.

1. Right-click on the mosaic in the Layers tab.
2. Select the Interpolate command.

Pop-up	Interpolate

The Interpolate Surface dialog box is displayed.



The *Destination Surface* field displays the file name of the copy that is created from the original Surface. The default name is the original Surface name with _Interp appended to it. You can change the name, if necessary.

3. [Optional] Type a new name in the *Destination Surface* field.

The *Matrix Size* determines the number of nodes closest to the node that can be used to interpolate a pixel value.

4. Select either the *3 x 3* or the *5 x 5* option.

The *Number of Neighbours* field is a threshold level used to determine the minimum pixels in the matrix area that must be present to interpolate a new pixel value. The matrix size determines the maximum and minimum values for the neighbours field. For example, if you select 3 x 3 for the matrix size, then the possible maximum number of neighbouring nodes with pixels is 9 and the minimum number is 3.

5. Type the number of neighbouring nodes used to interpolate pixels.
6. Click **OK**.

The interpolated mosaic is created and displayed with an _Interp extension, in the data tree in the Layers tab.

As you continue to re-interpolate the mosaic surface, the effects accumulate to remove any remaining gaps. To interpolate again:

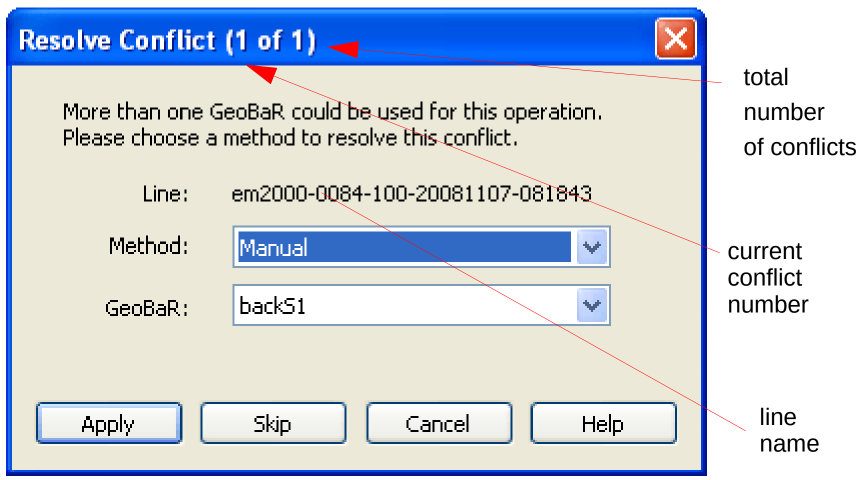
1. Select the interpolated mosaic in the Layers tab.
2. Select the Interpolate command.

The second interpolation is applied to the selected mosaic layer. It does not create a new layer.

Conflict Resolution

The Resolve Conflict dialog box can appear in any operation where there are multiple GeoBaRs for a line, but only one GeoBaR can be used for the operation. (For example, when loading a GeoBar on a line that has more than one GeoBaR created, or when creating a mosaic where multiple GeoBaRs have been created for the lines.)

Use the Resolve Conflict dialog box to choose a GeoBaR, based on attributes such as resolution or source type, as well as the option to select a GeoBaR manually from a list.



The title bar of the Resolve Conflict dialog box shows you the number of the current conflict and the number of conflicts there are for the line.

The line is identified, and options to resolve the conflict are listed in the *Method* field. These options vary with the task you are performing. If there is more than one conflict on a line, you can use the *Apply to all* option to apply the same method of resolution to all conflicts.

The dialog box also has three buttons: **Apply**, **Skip**, and **Cancel**. All these buttons close the dialog box.

- **Apply** uses the current settings to resolve this conflict.
- **Skip** will not resolve this conflict but will allow you to continue with any other conflicts.
- **Cancel** will stop the process regardless of how many conflicts are remaining.

Ways to Resolve Conflict

To resolve a conflict when loading a single GeoBaR:

1. Select either
   - *Newest GeoBaR (*the most recently created) or
   - *Finest Resolution* (e.g. in a group of GeoBaRs with resolution ranging from 1m to 10 metres, the GeoBaR with resolution of 1m will be selected), or
   - *Manual* (select the GeoBaR by name from a drop-down list of the conflicted GeoBaRs).

Selecting a GeoBaR

To resolve a conflict when you want to select a loaded GeoBaR and there is more than one GeoBaR loaded for the line, you can select by the specific source type of the GeoBaR. If more than one available GeoBaR is of the specified type, then a new Resolve Conflict dialog box is displayed containing only the GeoBaRs that share the source type in conflict.

1. Select either:
   - *Newest GeoBaR*
   - *Finest Resolution*, or
   - *Manual*, or
   - *Source Type*.

If you select *Source Type* as the method then

2. Select either Side Scan, Beam Average, or Time Series from drop down list.

Selecting Component GeoBaRs

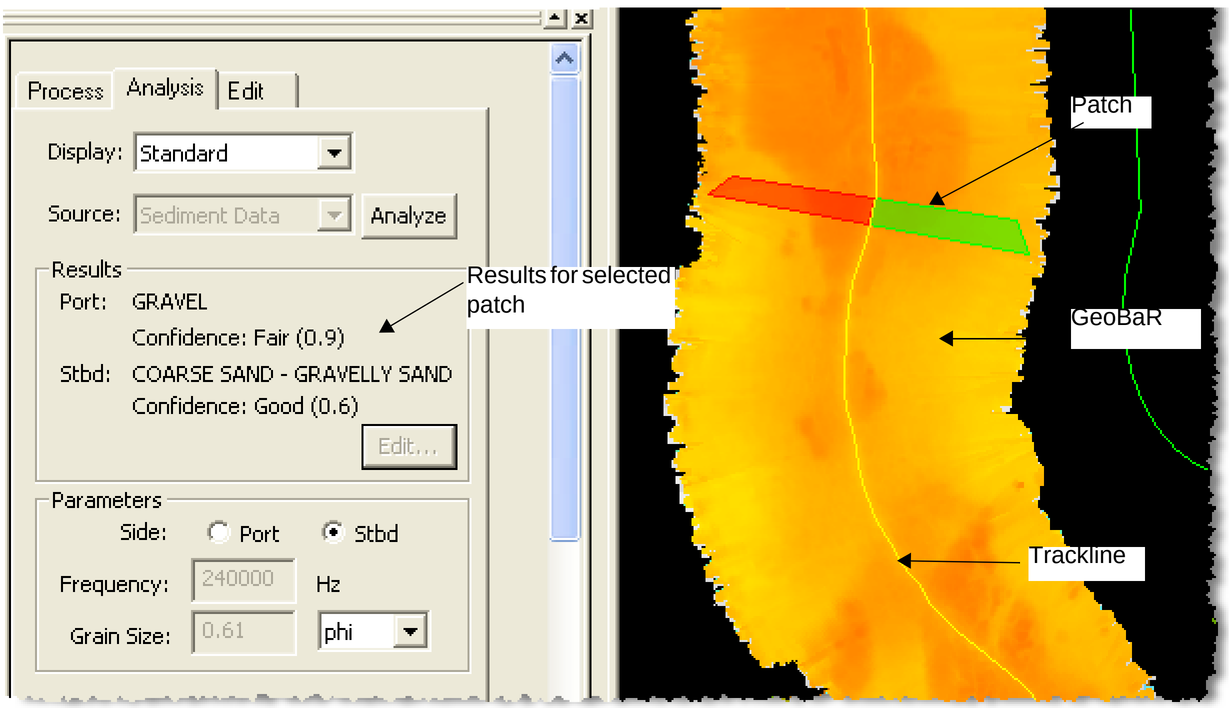
If you are editing a mosaic, and there are a number of GeoBaRs loaded, (any of which could have contributed to the mosaic), the Resolve Conflict dialog will show the option to select the specific GeoBaR used to create the mosaic.

1. Select Component from the *Method* list to select the GeoBaR which contributed to the mosaic.

Sediment Analysis

Use the Sediment Analysis tool in Mosaic Editor to determine sediment type, by analyzing the angular responses of data processed in the Geocoder engine.

When a GeoBaR is created using the Geocoder engine, a binary sediment analysis file is also created for the line. Sediment analysis is performed over a certain number of pings, controlled by the Patch Size in the Sediment Analysis Options dialog box. (See "Set Sediment Analysis options" on page 446.)



Data is analysed with respect to the sediment angular response models in order to determine an average grain size. This average grain size is then cross-referenced to the customizable look-up table in order to provide a textual response. (See "Grain Size table" on page 451.)

The sediment analysis results are displayed on the Analysis tab. They can also be seen in the Sediment Analysis Graph window that displays the average angular response for a given range of data. (See "Sediment Analysis Graph" on page 453.) Sediment analysis results can also be exported to ASCII. (See "Sediment Analysis to ASCII" on page 539.)

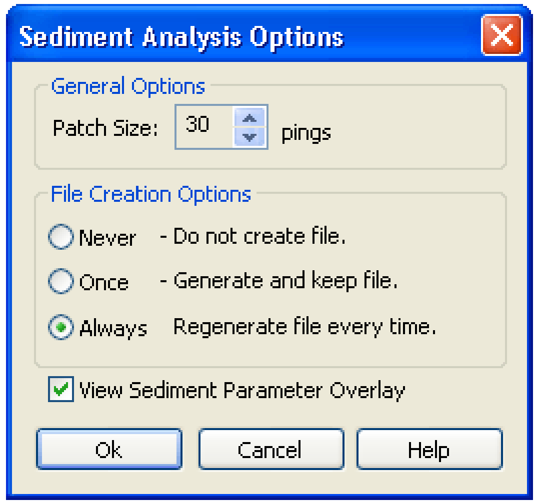
Set Sediment Analysis options

1. Select the Analysis tab in Mosaic Editor.

Menu	Tools > Mosaic Editor > Sediment Analysis Options
Tool	

2. Select the Sediment Analysis Options command.

The Sediment Analysis Options dialog box is displayed.



3. Set the number of pings in the *Patch Size* over which the sediment analysis will be performed.

By default, a sediment analysis file is created every time a new GeoBaR is created for that line. This new sediment analysis file will overwrite any previous file for the line. However, you can change this option so that a sediment analysis file is never created, or so that the file is only generated once per line

4. Set the File Creation Option by selecting *Never*, *Once* or *Always*.

Use the View Sediment Parameter Overlay option to overlay certain sediment analysis parameter data on a selected GeoBaR.

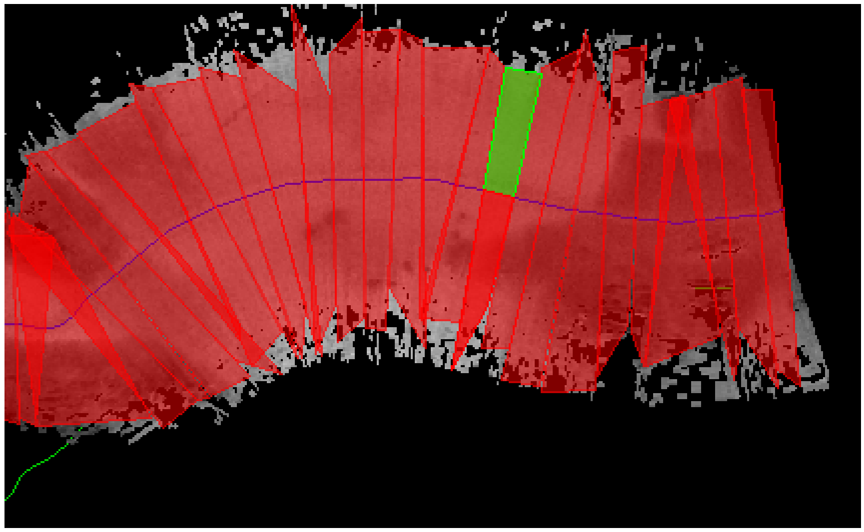
5. Set View Sediment Parameter Overlay to display representations of the sediment analysis data. (See “View Sediment Analysis Overlay” on page 455 for more information.

Auto Analyze

The Auto Analyze function will analyze sediment data for entire lines for which GeoBaRs have been created. This automated process can replace the labour-intensive process of analyzing each patch individually.

1. Select the layer named “GeoBaRs”.
2. In the Properties window, set the Show All Sediment Analysis Patches to “True”, and set Auto Adjust Histogram to “False”. See also “GeoBaR layers” on page 24
3. Activate the Analysis tab of the Mosaic Editor tab in the Control window.
4. Select one or more lines displaying open GeoBaRs.

All sediment analysis patches available on the lines will be highlighted. In the image below the sediment patches are in red; one selected patch shows in port and starboard colours.



Menu	Tools > Mosaic Editor > Auto Analyze Sediment Data
Tool	

5. Select the Auto Analyze Sediment Data command.

A dialog box will appear confirming that the analysis is taking place. The Status bar will show the progress of the analysis. When the process is complete the outcome is reported in the Output window. Results displayed on the Analysis tab.

Analyze selected patch

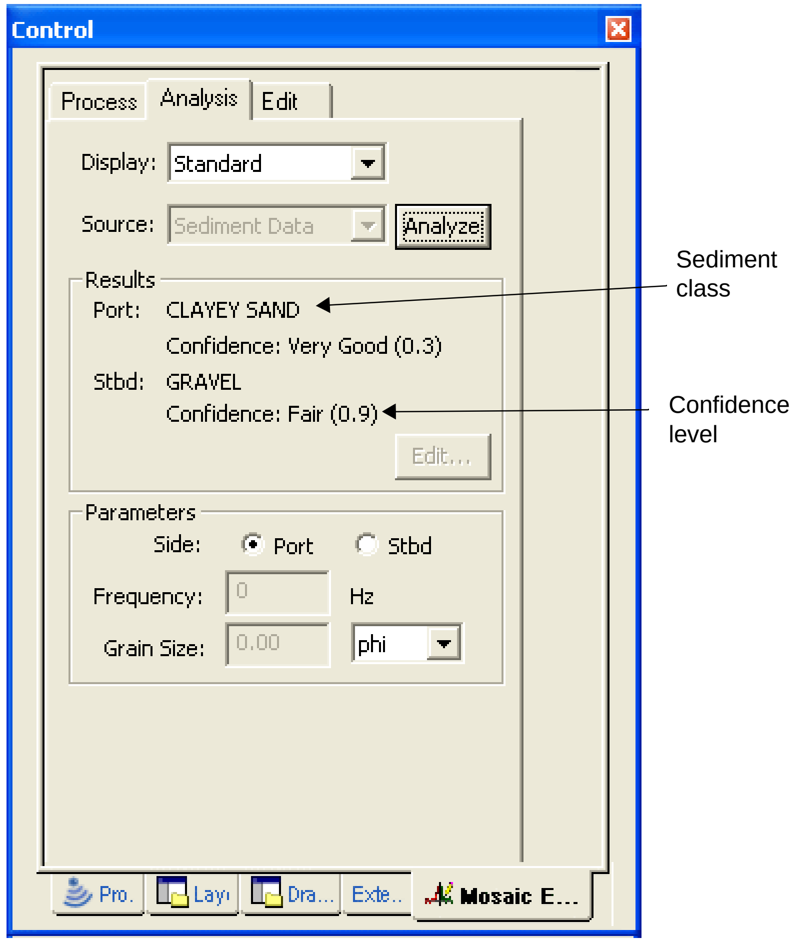
1. Select the layer named “GeoBaRs”.
2. Set the Show All Sediment Analysis Patches option in the Properties window.
3. Select the Analysis tab of the Mosaic Editor tab in the Control window.
4. Select one or more lines displaying open GeoBaRs.
5. Select a patch from the patches highlighted.

The port and starboard sides of the selected patch will be highlighted.

Menu	Tools > Mosaic Editor > Sediment Analysis Graph
Tool	

6. Open the Sediment Analysis graph.
7. Select Standard from the *Display* field drop-down list.
8. Click **Analyze**.

The graph window will draw the lines resulting from the analysis. The Results section of the Analysis tab will display the sediment class and confidence level of the analysis.



The results are displayed as the type of sediment, based on the values set in the Grain Size table. (See "Grain Size table" on page 451.) Also shown is the confidence level in the result (the smaller the value, the higher the confidence level).

Auto Analysis of sediment data can be done in either Standard or Advanced mode.

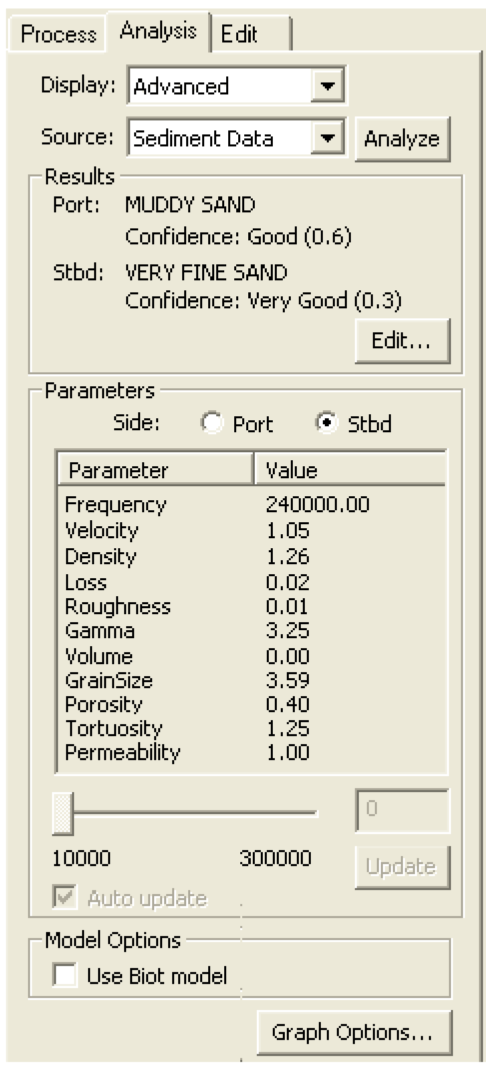
Advanced Mode

The Advanced mode on the Analysis tab is an extension of the Standard mode and displays more parameters. This analysis compares the grain size from the data to the values on a customizeable lookup table. (See "Grain Size table" on page 451.)

(The Advanced Mode is likely to be of more interest to users doing scientific research rather than a processing workflow.)

1. Highlight a sediment analysis patch.
2. Select Advanced from the *Display* field.
3. Select Sediment Data from the *Source* field on the Analysis tab.
4. Click **Analyze**.

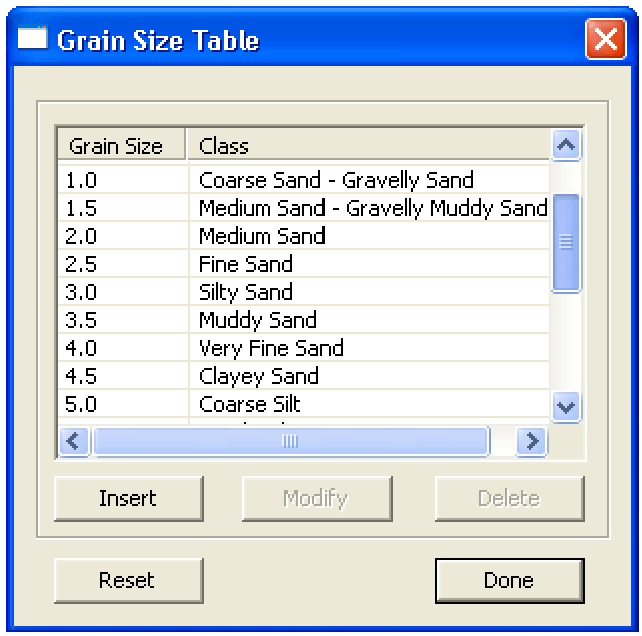
The sediment analysis results for that patch are displayed in the *Results* field. The input parameters for the results are shown in the *Parameters* fields. These values are read-only.



Grain Size table

The classes of sediment which is displayed by the sediment analysis are based on values in the Grain Size table. This table can be customized while in Advanced mode only.

1. Select the Analysis tab.
2. Select Advanced.
3. Click **Edit** to open the Grain Size table.



Grain size values and their descriptions can be edited using the **Modify** button.

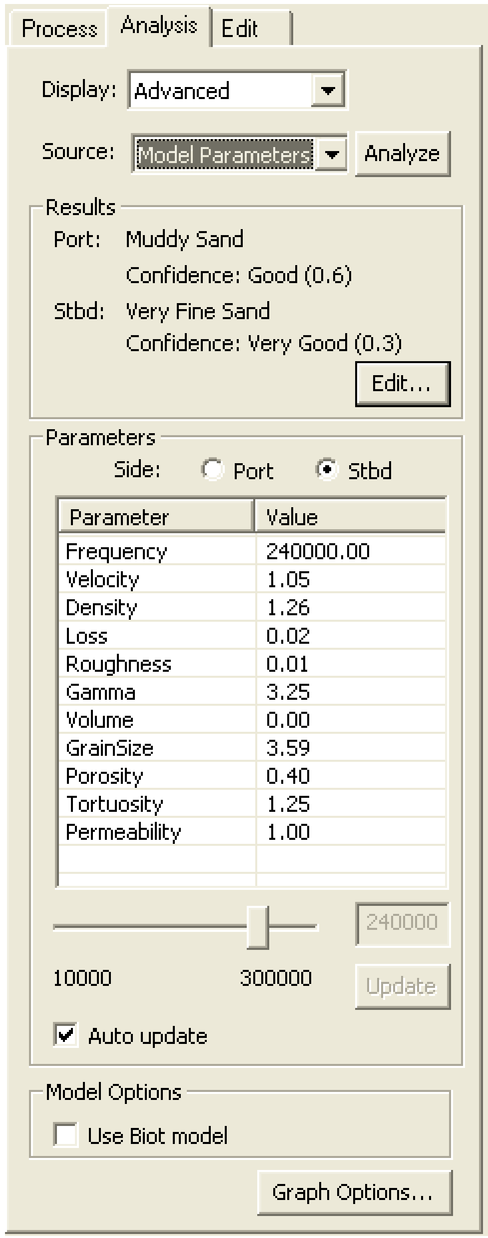
New classes can be added using the **Insert** button, and removed from the table using **Delete**. Use **Reset** to restore the default values.

Model Parameters

The Model Parameters mode enables the specialized user to define the backscatter curve for their own purposes by manipulating parameter values for port and starboard data or applying a Biot model.

1. Highlight a sediment analysis patch.
2. Select Model Parameters from the *Source* field on the Analysis tab.

The Parameters list box is activated.



You can modify values in the list using the slider below it.

3. Select a parameter, e.g., porosity, from the list.
4. Use the slider below the list to adjust the value of that parameter.

The values for the parameter automatically update as you move the slider. The range of possible values varies with the parameter selected.

Alternatively, you can manually update each parameter as you adjust the value.

5. Click **Update** to set the new value in the list.

Sediment Analysis Graph

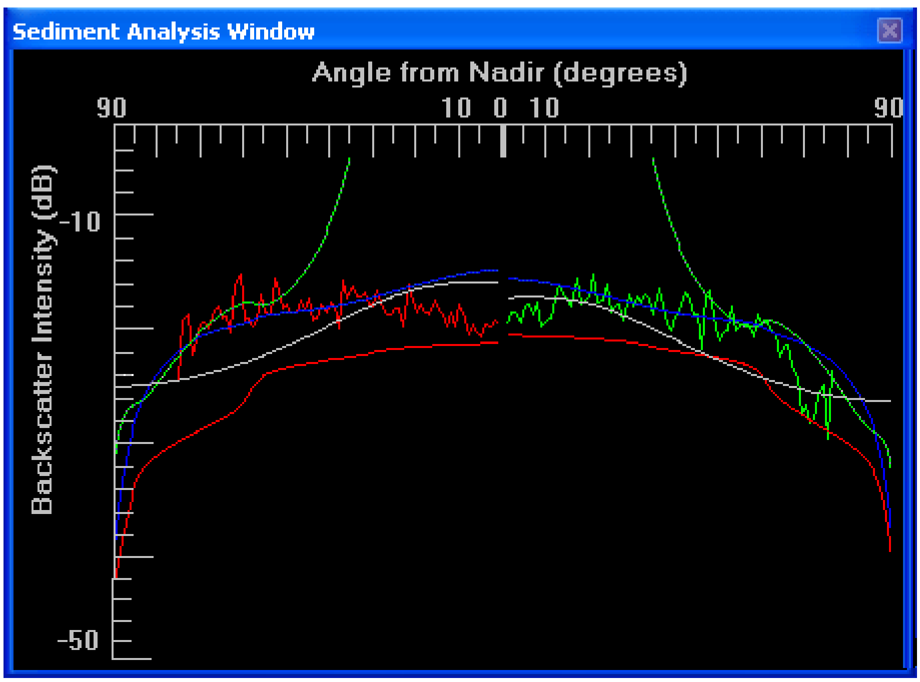
You can view the results of sediment analysis graphically in the Sediment Analysis Graph window. This graphs the Interface, Volume and Kirchhoff backscatter intensity values to angle of nadir in degrees.

To open this window:

1. Click open the Analysis tab, and select Advanced mode from the *Display* drop-down list.
2. Select Sediment Data from the *Source* drop-down list.
3. Select a survey line.
4. Select the Open Sediment Analysis graph command.

Menu	Tools > Mosaic Editor > Sediment Analysis Graph
Tool	

The graph is displayed.



The graph can display three kinds of backscatter, or the total backscatter or all four, colour coded as follows:

- Light green line is Interface Backscatter: occurring at the sea floor/water interface, this is the main component of the initially returned acoustic energy.
- Light red line is Volume Backscatter: secondary in time, this is sound energy returned or scattered from within the sediment. The less homogeneous the sediment, the more the sound wave is disturbed.

- Yellow line is Kirchhoff Backscatter: This model accounts for the roughness of the sea floor particularly for grazing angles close to 90 degrees.
- Blue line is Total Backscatter: the combined backscatter from the three backscatter sources. This is analyzed to determine approximate grain size.

The line colours can be changed from these defaults in Tools > Options > Display > Mosaic Editor dialog box.

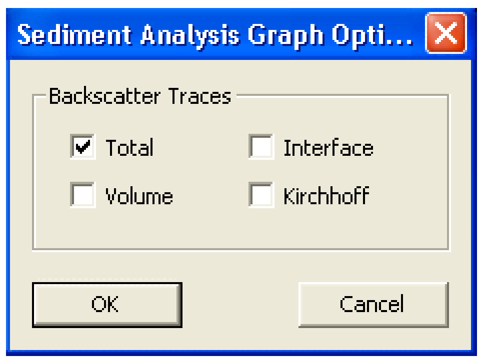
Also shown are port and starboard lines in their traditional colours.

Graph options

To set which backscatter forms will be displayed in the graph window:

5. Select Advanced from the *Display* field on the Analysis tab.
6. Click the **Graph Options** button at the bottom of the Analysis tab.

The Sediment Analysis Graph Options dialog box is displayed.



7. Select the check boxes for the types you wish to display. Click **OK**.

Close graph

To close the graph window:

8. Toggle the Open Sediment Analysis graph command off, or click the toolbar button again.

View Sediment Patches

This option displays the patches as a vector overlay on the main display.

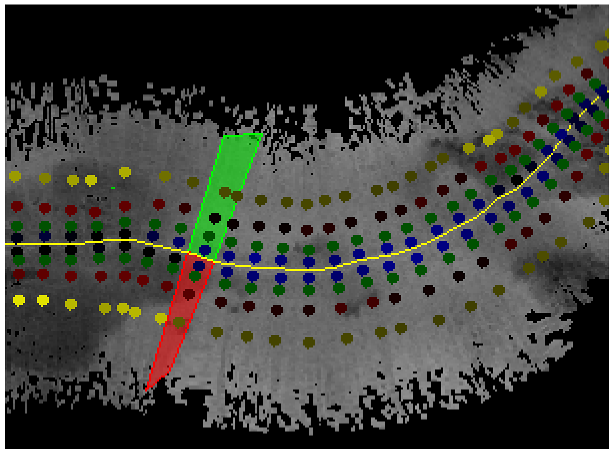
To locate a specific patch along a line:

1. Select the GeoBaR layer.
2. In the Properties window, set Show All Sediment Analysis Patches to True.
3. Click open the Sediment Analysis tab.
4. Click on the line in the Display window at the point at which you want to see the sediment values.

The patch for that part of the line will be highlighted.

View Sediment Analysis Overlay

Use this feature to overlay certain sediment analysis parameter data on a selected GeoBaR.



Four data points are shown per patch, per side. Starting from nadir and going outward they are: Grain size (or phi), roughness, volume, and impedance. The data points are shown in 256 shades of blue, green, red and yellow respectively.

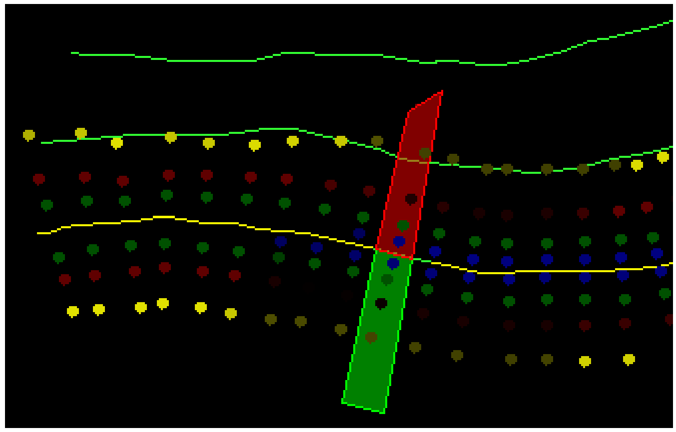
The data is scaled against absolute minimums and maximums for each parameter type (rather than relative to the line's max/min for each). This makes comparisons between different lines' data meaningful.

The data will be displayed for any patch that was analyzed prior to selecting the line, meaning that if you change the data through further analysis while the line is selected, the overlay will not be refreshed.

At this time, this overlay feature is only available for one selected line/GeoBaR at a time, and does not work with multiple lines or Mosaics.

1. Open Mosaic Editor.
2. Set the View Sediment Analysis Overlay option in the Sediment Analysis options dialog box.
3. Click the Analysis tab in the Control window.
4. Select a line.

The overlay for the line is displayed.



Since the overlay is created from the sedimentAnalysis.xml file it can be displayed even when GeoBaRs are not loaded.

Contacts

Contacts are representations of object and feature on the sea floor, such as wrecks or other shoal features, that are visible in the sonar data.

In Side Scan Editor you can georeference contacts and attach attributes attached to them using the Add Point or Add Line Contacts tools. Contacts are placed directly in the waterfall view. A contact record is created for each contact, stored in the contact file for the survey line. A contact record includes sufficient information to be subsequently exported to a CARIS map or to a database application.

Contact are described in "Contacts" on page 81 in the Side Scan Editor section of the Editors Guide.

18

Process Water Column Data

Water column data can be viewed, selected and queried in Subset and Swath Editor. the data can also be imported into Additional Bathymetry.

The display of water column data can be manipulated in the editors, and bathymetry derived from water column imagery can be added to an existing project.

In this chapter...

Water Column Data in HIPS 460
Water Column Data in Swath Editor...................... 462
WCI Controls in Swath Editor 466
Add WC Bathymetry to Project............................. 468
WCI Data in Subset Editor.................................... 469
Additional Bathymetry.. 473

Water Column Data in HIPS

Water column data can be integrated into existing bathymetry workflows. Water column image (WCI) data can be displayed in both Swath Editor and Subset Editor, and its bathymetry data added to a project. In Swath Editor water column data is displayed as a curtain image. In Subset Editor it is displayed in 3D.

Currently, water column imagery data from Teledyne Reson 7K (.s7k) and Simrad (.all, .wcd) sonars is supported in HIPS and SIPS.

"Water Column Data in Swath Editor" on page 462

"WCI Data in Subset Editor" on page 469

Projects

Projects containing water column data are organized in the same Project-Vessel-Day-Line hierarchy as other sonar projects.

When data is converted to HIPS there is the option of copying the original data files to the Processed folders. By default, raw data files are not carried over.

When opening water column data, HIPS will look for WCI data in the line directory. If the data is not found there, HIPS looks in the location of the raw data files as recorded during conversion. If the raw data no longer exists in its original location, you will be prompted to have HIPS search for it, or to search for it yourself.

Processing Workflow for Water Column Data

Open a project to which the water column data will be added.

Open WCI data in Swath Editor (Across and Along track and Stacked views)

- Filter based on intensity
- Overlay Bottom Detection
- Assign Colour Map

- Line editing - Query /Reject/Accept bathymetry
- Select and data to add to project bathymetry
- Re-evaluate intensity filter range

Open data in Subset Editor (2D and 3D views)

- Filter by intensity ranges determined in Swath Editor
- Query/ Reject /Accept bathymetry data
- Designate critical soundings
- Select and add points to additional bathymetry layers.

Working with Additional Bathymetry Layers

Toggle views between WCI, project bathymetry and additional bathymetry layers.

Compute TPU for additional bathymetry.

If required, undo all processing of Additional Bathymetry, with Restart Cleaning command on pop-up menu.

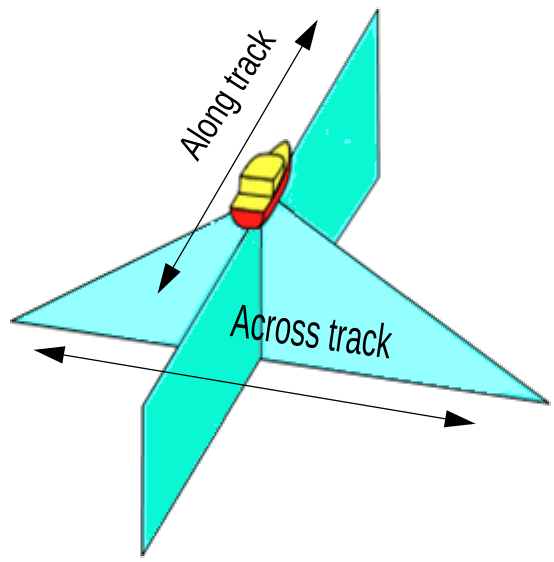
Regenerate additional bathymetry layers if project data has new corrections and new merge applied.

Add additional bathymetry to BASE surfaces.

Water Column Data in Swath Editor

In Swath Editor. water column imagery is displayed in two windows:

- Across track: the water column data as viewed from the stern of the survey vessel. There is an across track image for each selected profile (ping). (Also referred to as a profile view, polar intensity plot or swath profile.)
- Along track: displays the water column data viewed from the starboard side of the vessel. This could also be referred to as a side view or vertical profile. The along track image is made up of the beams which point to nadir for the selected profiles.



Stacked view

WCI can also be displayed in a “stacked” view, to give an idea of what may be in the water column based on multiple profiles and beams. In the across track view all the profiles represented in the Plan view are stacked one on top of another.

In the along track view the beams within a profile are stacked with the highest intensity beams showing through. See “Stacked view” on page 466.

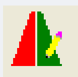
Bottom detections

Bottom detections can be plotted on top of the water column image in the across track view. This can be used to view how the existing bathymetry compares to that which is present in the water column data. This display can be toggled on and off from the toolbar. See “Water Column Toolbar” on page 465.

Basic filtering is also present so the WCI data can be filtered based on intensity values.

All existing Swath Editor options and functionality, where relevant, have been connected within the WCI views, to keep work flows consistent and familiar.

Display water column data

Menu	Tools > Swath Editor > Open
Tool	
Pop-up	Tools > Swath Editor

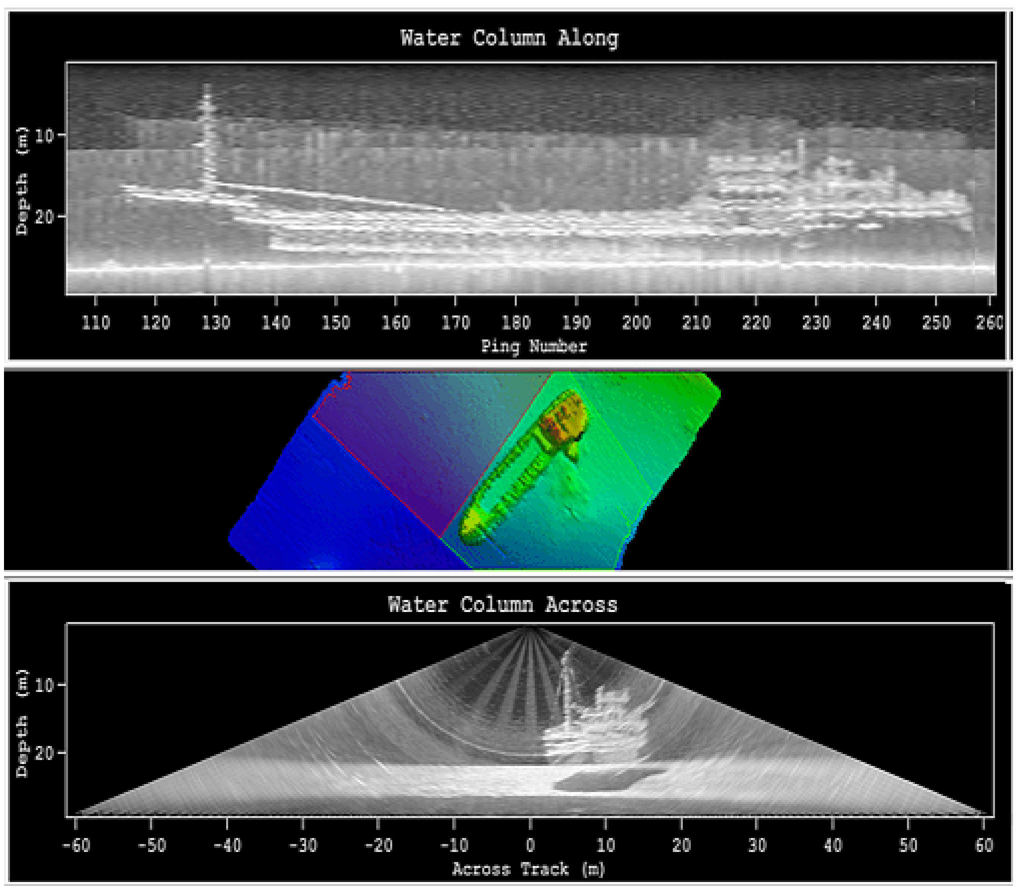
To view WCI data in Swath Editor:

1. Select a survey line in the Display Window.
2. Select the Open Swath Editor command.

If the selected line contains WCI data:

- the data will be loaded into the Along and Across track windows,
- the Water Column Editor toolbar will be displayed,
- and the WCI tab will be displayed in the Swath Editor Control page.

The example below shows WCI data displayed in the across and along track windows, and bathymetry in the Display window (shown as a BASE surface). [1]



3. Select a profile, for example, in the Plan View.

The related WCI data is displayed in the water column windows

1. Image courtesy of John Hughes Clarke, Ocean Mapping Group, University of New Brunswick.

Scroll through the Plan view using the arrow keys, or scroll buttons, to see the water column image in the across and along track views.

To highlight a WCI profile in the Swath Editor views:

4. Click the middle mouse button in the Along track window.

As each profile is highlighted, the display in the Swath Editor windows is automatically refreshed.

As well, there is a slider bar in the Across track window, similar to that of the Profile window, which you can use to scroll through the data.

Automatic scrolling

As well as viewing WCI data profile by profile, you can view the data in sequence using the playback controls on the Water Column toolbar. (These work in a similar fashion to the playback controls in the Side Scan Editor.)

To scroll automatically through the data:

1. Click on the Plan View window.
2. Click **First Profile** on the Water Column Data toolbar to select the first profile visible in the Plan View.
3. Click **Start playback** to start scrolling the data.

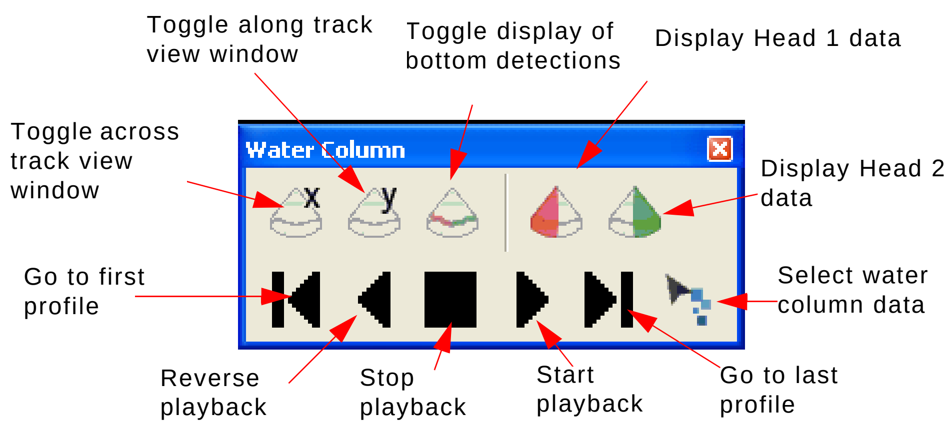
HIPS will move up the track line, profile by profile, changing the view of the water column data, until it reaches the last profile loaded in the Plan View.

4. Click **Reverse playback** to scroll back down the track line.
5. Click **Stop playback** to pause the scrolling.
6. Click **Last Profile** to reset the view to the last profile loaded in the [plan View.

Playback is not available when data is stacked.

Water Column Toolbar

The Water Column toolbar contains the controls to open and close the Across track and Along track windows, to display the bottom detection and to control the play back of the imagery.



The **Display across track** and **Display along track** tool buttons open and close their respective windows.

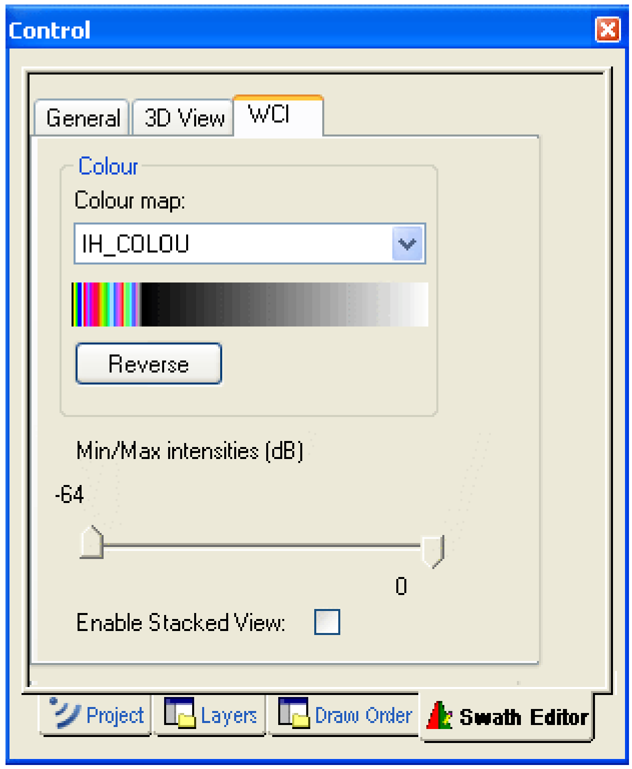
The **Bottom Detection** button toggles the bottom detection on and off. In the across track it will indicates the current swath bathymetry and in the along track, the centre beam bathymetry If activated when Stacked View is enabled, the bottom detections are also stacked.

Head 1 data and **Head 2 data** are used when there is data from both heads of a dual head multibeam configuration. You can toggle between the heads to determine which data should be plotted on top: Head1 (port data) or Head 2 (starboard data).

The **Select** button enables you to select water column data in the across track or along track windows, for example, to add data to the Additional Bathymetry layer.

WCI Controls in Swath Editor

When there is water column data open in Swath Editor, the Control window tab will include a WCI tab containing options for displaying water column data.



Use the *Colour* controls set the colour of the data displayed in the Across track and Along track windows.

7. Select a colour from the *Colour map* field.
8. Click **Reverse** to change the current colours to their inverse.

Water column data can be filtered by intensity values, so that only data within the set range is displayed. To filter the data:

9. Use the sliders to set a filter range between -64 dB and zero.

These intensity settings are retained so that the same filter range can be applied when the data is loaded into Subset Editor.

Stacked view

Use the *Enable Stacked View* option to see all the across track views layered on top of each other in the Across track window.

1. Make a selection of profiles in the Plan view window.
2. Select the *Enable Stacked View* check box.

The Across track view will display the stacked images.

3. Use the scroll bar on the Plan View, or your middle mouse button, to move the selection to other profiles.

The stacked view will be refreshed with the new data.

> You cannot use playback when the data is in Stacked mode.

Add WC Bathymetry to Project

Viewing WCI data in Swath Editor enables you to determine critical data that can then be selected and added as bathymetry in the project. As well, you can then use the editing tools to Reject, Accept and Designate the new additional bathymetry data

To select data in the Across or Along track windows:

Tool 

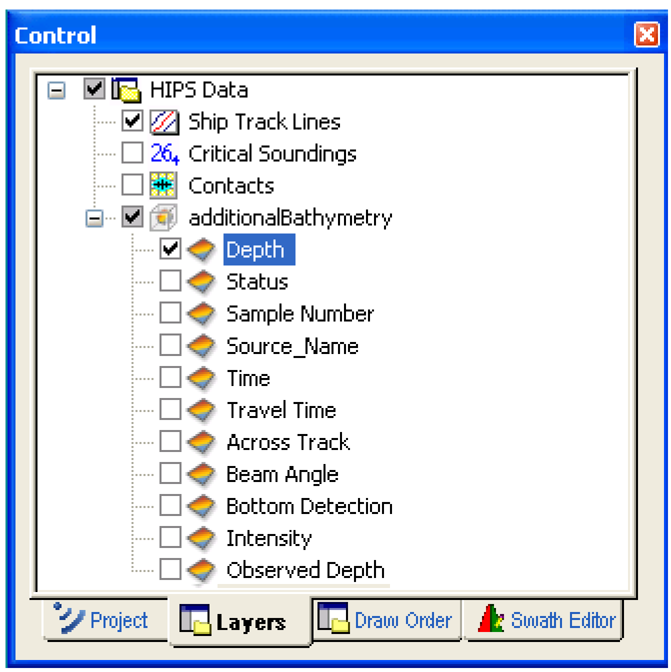
1. Click the Select button on the Water Column toolbar.
2. Use Select by Range or Select By Lasso to select data.
3. Right-click on the selected data and select Query from the pop-up menu to view in the Selection tab.

To add selected data to the project as additional bathymetry:

Pop-up: Add selection to Additional Bathymetry

4. Right-click on the selected data and select Add to Additional Bathymetry.

The data will now appear as “additionalBathymetry” layers in the Layers tab.



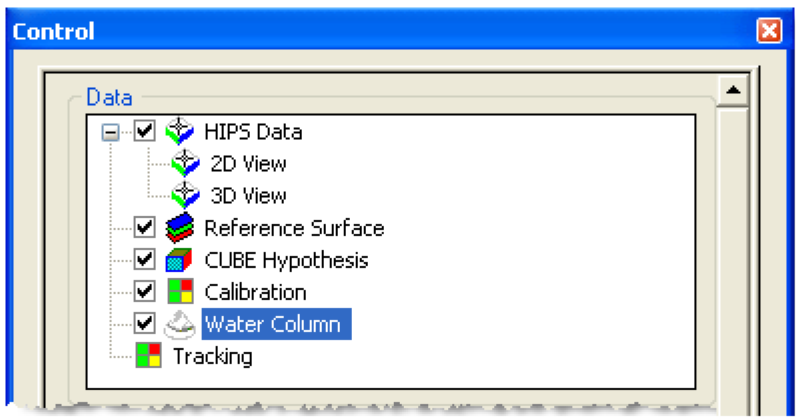
As well, the Output tab will report the number of WCI samples exported to the bathymetry layer.

WCI Data in Subset Editor

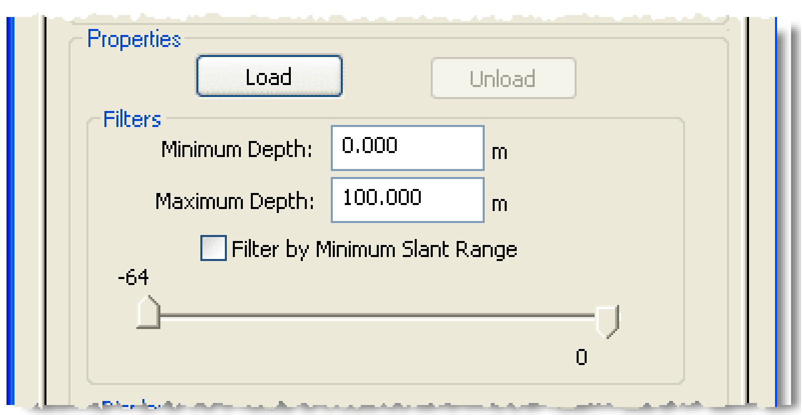
In Subset Editor, WCI data is loaded as a temporary 3D point cloud. Data can be filtered by intensity as well as by a minimum slant range at nadir. WCI data in Subset Editor can also be added the the Additional Bathymetry layer and processed in the standard bathymetry workflow.

To open water column data in Subset Editor:

1. Define and load a subset of the data. (For information on loading a subset see, "Open Subset Editor and Load Data" on page 111.)
2. In the Control window, select the Subset Editor tab.
3. In the Data tree, select the Water Column layer.



The tab is refreshed to display the controls for loading and unloading WCI data, filtering by depth, slant range, and intensity.



4. [Optional] Set a range of depths to load using the Minimum and Maximum Depth fields.

Applying a minimum slant range filter will load only the strong echoes and minimize background noise. Background noise within the minimum slant range has a backscatter strength of lower than -50dB. Therefore it is logical to load only those data points which have a stronger echo than - 50 dB.

This option is turned off by default. To set slant range filtering:

5. Select the *Filter by Minimum Slant Range* check box.

Water column data can be filtered by intensity values, so that only data within the set range is displayed. This filter is the same as in Swath Editor. Settings are retained so that the same filter range can be applied in both editors.

To filter the data:

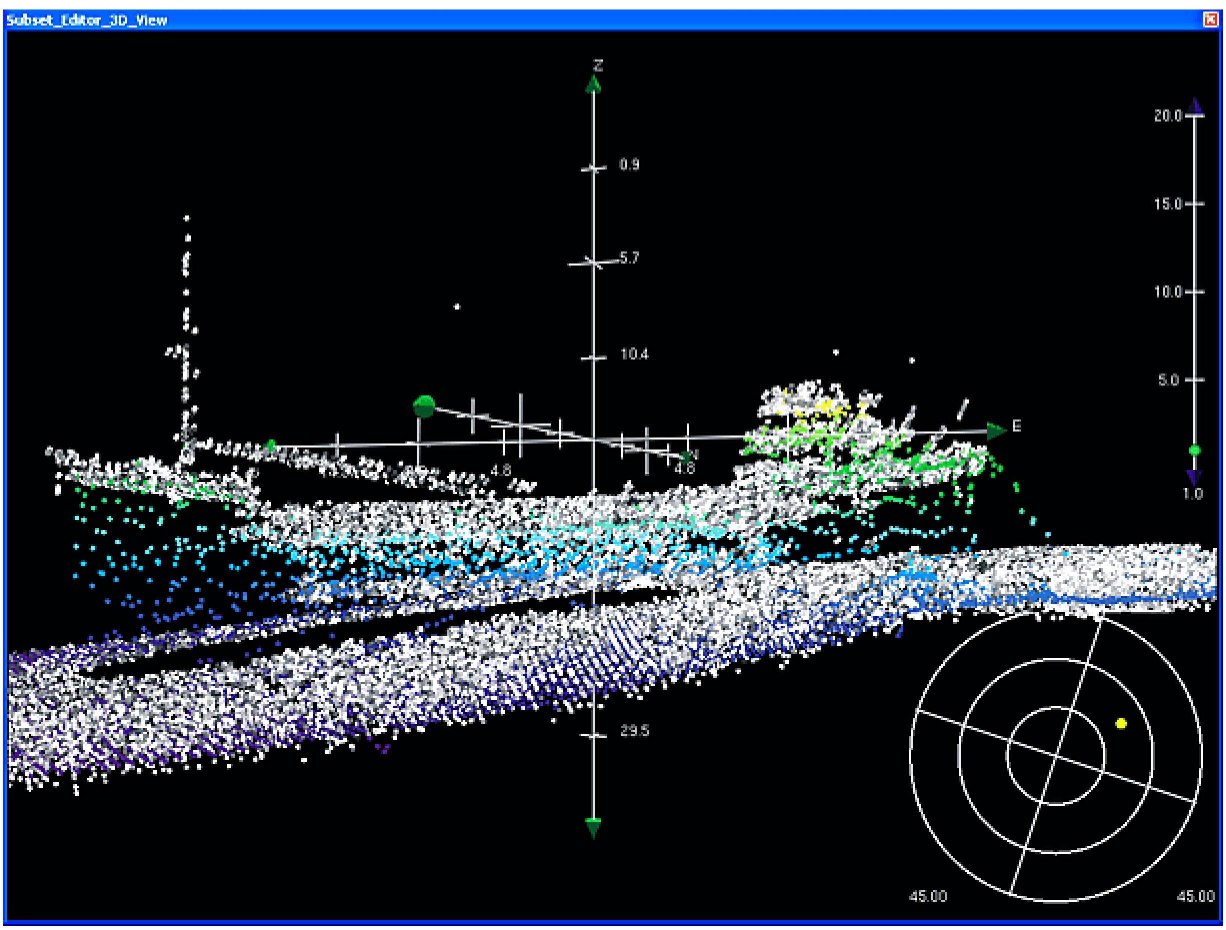
6. Use the sliders to set a filter range between -64 dB and zero.
7. Click **Load**.

The face of the **Load** button shows the loading progress. Loading can be cancelled by clicking the **Cancel** button.

When the data is loaded, the button label is changed to **Open**.

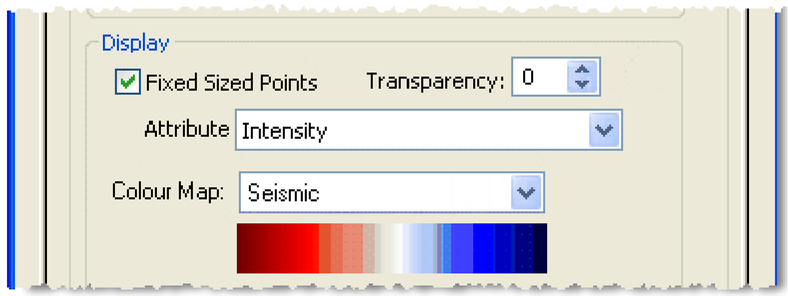
8. Click **Open** to display the data in the Subset Editor windows.

The image below shows WCI data displayed in the Subset Editor 3D view:



Display options

You can set the size of the displayed data points, their transparency and colour them by attribute.



The default setting for the size of points is set in the HIPS Data layer, where the size can be set from 1(smallest) to 10 (largest).

You can override this default setting for the WCI data. To have stronger returns display as larger points, and weaker returns as smaller points, (while other points in the subset are not resized):

1. Select the Water Column layer in the Data tree.
2. Clear the *Fixed Sized Points* check box.

If you want the WCI points to be displayed at a constant size:

3. Select the *Fixed Sized Points* check box.

You can toggle this display option on and off while data is loaded.

Attributes of the WCI data can be displayed one layer at a time. Each attribute (except Bottom Detection) has its own colour map. You can change the assigned colour mapping.

The attributes include:

- Intensity
- Depth
- Line
- Day
- Vessel
- Project

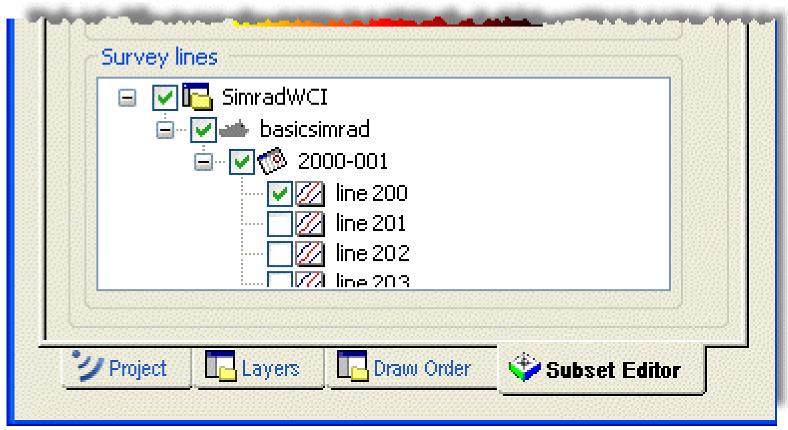
To view the data of a particular attribute:

4. Select the attribute from the drop-down list.
5. Select a colour map from Colour Map drop-down list.

Two colour settings for “Bottom Detection” indicate whether a point is detected bottom or not. You can change the colours of these flag settings also using the colour picker.

Survey lines

The Survey Lines field in the lower part of the Control window shows a data tree in the familiar P/V/D/L layers.



Use the check boxes beside each layer to turn on and off the WCI data associated with individual lines.

Additional Bathymetry

In both Swath Editor and Subset Editor additional bathymetry data can be queried and edited. As well critical soundings can be Designated, and others flagged as Outstanding or Examined.

Restart Cleaning

If you want to reverse the editing of additional bathymetry:

1. Right-click on the additional bathymetry layer, and
2. Select Restart Cleaning from the pop-up menu.

This will restore the status of the data to Accepted.

Show Rejected

The additionalBathymetry (parent) layer has similar properties to those of a BASE surface: extents, identification, creation date, etc. See “View Surface Properties” on page 204.

As with BASE surface parent layers, there are few editable fields, however, for the additionalBathymetry layer there is the option to display any additional bathymetry data that has been rejected in an editor.

1. Set *Show Rejected Points* to True to show rejected data.
2. Set the option to False to hide rejected points.

The attribute (child) layers have the same properties options as do BASE surface attribute layers. See “Attribute Layer Properties” on page 206.

Compute TPU

TPU can be applied to the additionalBathymetry layer. This will add two child layers to the additionalBathymetry attributes: DepthTPU and PositionTPU. See “Compute TPU” on page 156.

Regenerate

If corrections such as tide, SVP etc., have been applied to project data during processing, requiring that Merge be re-applied, the additional bathymetry is *not* updated.

To update the additional bathymetry:

1. Right-click on the additionalBathymetry layer.
2. Select Regenerate from the pop-up menu.

19

Create Publications

Use Plot Composer to publish HIPS and SIPS products to PDF or paper.

In this chapter...

PLOT COMPOSER 476

Plot Composer

HIPS and SIPS data can be formatted for printing to paper or to Portable Document Format (PDF) using CARIS Plot Composer.

Plot Composer is a plotting utility provided with various CARIS applications to produce paper products containing raster and vector geo-spatial information.

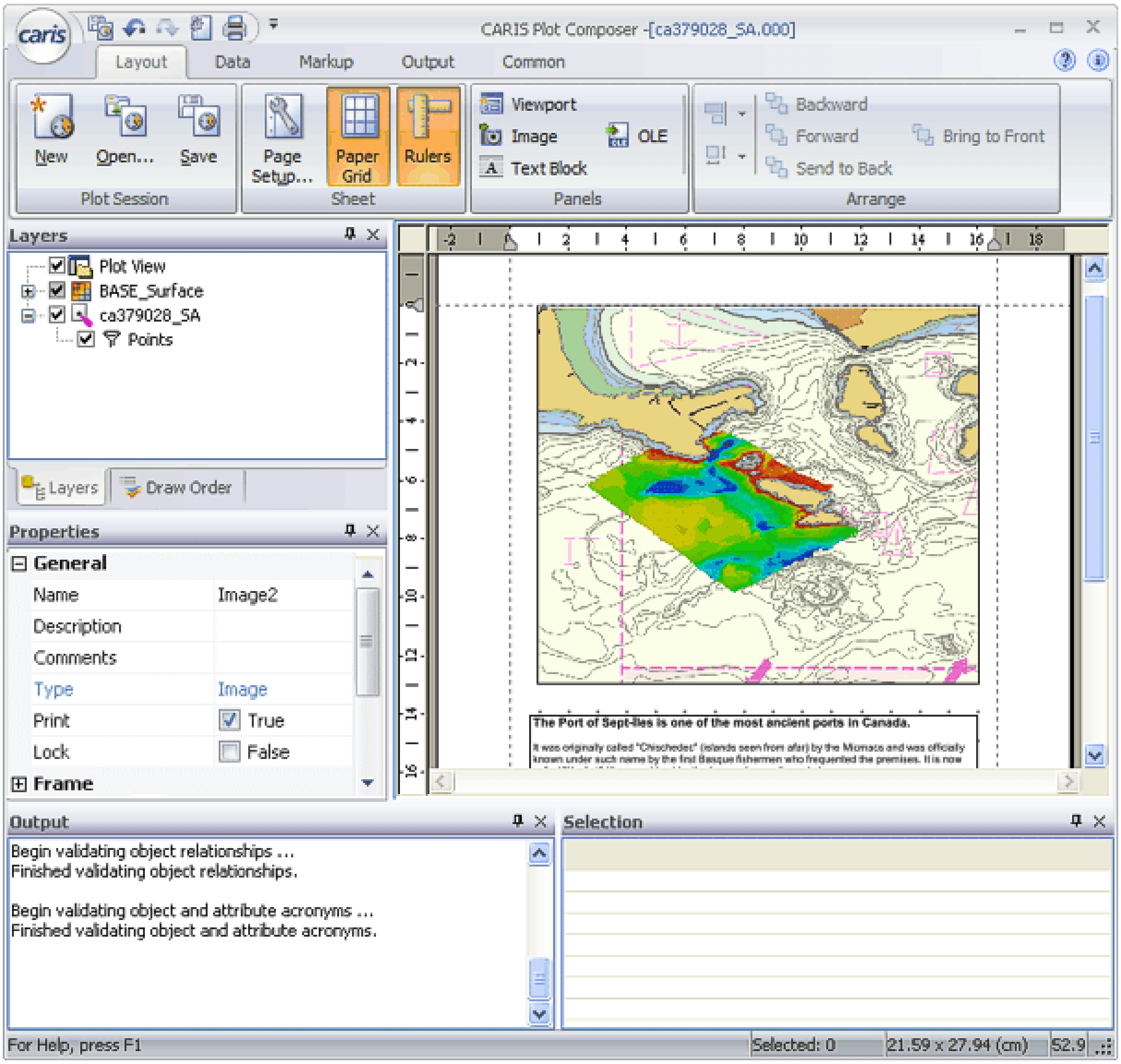
Plot Composer can use data from other CARIS products such as Bathy DataBASE, in formats such as GeoTIFF, BSB, JPEG 2000, S-57, Shapefiles, AutoCAD, as well as HIPS and SIPS data, to produce paper products.

Some of Plot Composer's key functions are to:

- Capture data views from HIPS and SIPS in the scale displayed
- Configure data and add raster and vector data formats supported by CARIS applications
- Display geo-referenced in the CARIS supported formats
- Add geographic or projection grids, borders and scale bars
- Configure data views to be *north-up* or rotated
- Set coordinate system
- Add OLE objects, images and text blocks
- Output to PostScript or TIFF file.

Open Plot Composer from the Start > Programs > CARIS > Utilities folder, (which also contains the Plot Composer Users Guide).

The following is an example of a BASE surface and background data being formatted in Plot Composer.


CARIS Plot Composer -[ca379028_SA.000]
Layout
Data
Markup
Output
Common
New
Open...
Save
Plot Session
Page Setup...
Paper Grid
Rulers
Sheet
Viewport
Image
OLE
Text Block
Panels
Backward
Forward
Bring to Front
Send to Back
Arrange
Layers
Plot View
BASE_Surface
ca379028_SA
Points
Layers
Draw Order
Properties
General
Name
Image2
Description
Comments
Type
Image
Print
True
Lock
False
Frame
Output
Begin validating object relationships ...
Finished validating object relationships.
Begin validating object and attribute acronyms ...
Finished validating object and attribute acronyms.
Selection
The Port of Sept-Iles is one of the most ancient ports in Canada.
For Help, press F1
Selected: 0
21.59 x 27.94 (cm)
52.9

20

Export Data

Sounding and contour layers can be exported to S-57 format, or to DXF. Surface metatdata can be exported to an XML file. Surface data can also be exported to BAG.

As well, data products can be exported in various formats with the Export Wizard.

In this chapter...

Export to S-57 480
Export Surface Metadata 481
Export Surface to BAG... 482
Export Surface to ESRI ASCII Grid 486
Export Selection to DXF.. 487
Export Selection to Formatted ASCII 489
Export Selection to GML 490
List of Formats Exported 492
Export Wizard 494
Coordinate System 541

Export to S-57

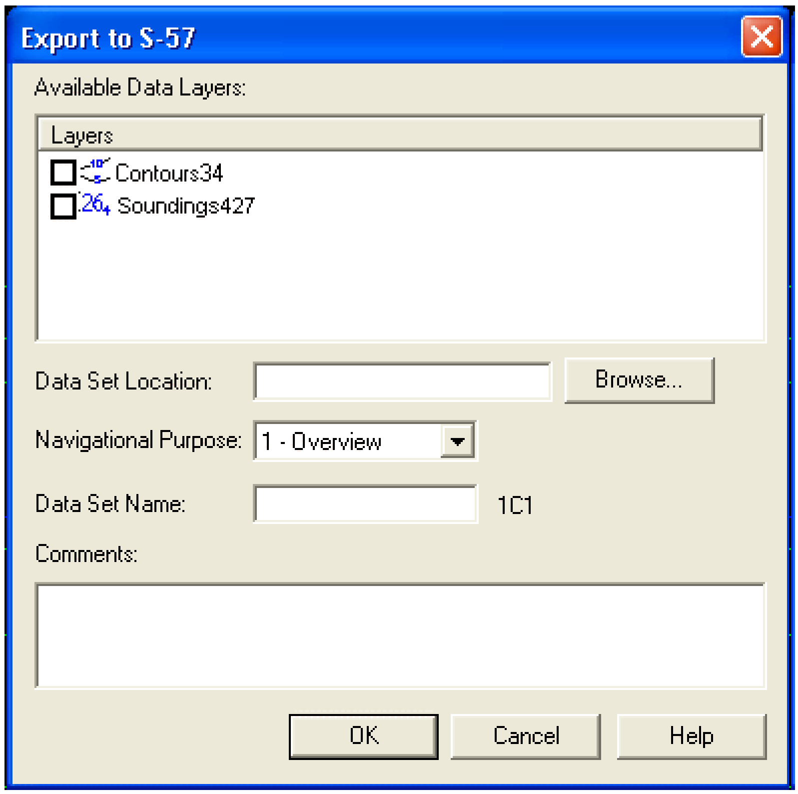
You can export contour and sounding features to an S-57 file.

To export to S-57:

1. Open a project and field sheet containing a contour layer and/or a sounding layer.
2. Select the field sheet layer in the Layers tab.
3. Select Export to S-57 from the File menu.

Menu: File > Export

The Export to S-57 dialog box opens.



4. Select a data layer to export from the list of available layers.
5. Type the full path or click **Browse…** to enter the full path to the data.
6. Choose a Navigational Purpose layer from the drop-down list of chart types.
7. Type a 5-character data set name for the file. This name will be appended to the characters displayed to the right of the field (1C1 in this example).
8. Click **OK**.

Export Surface Metadata

Save the metadata from a CSAR surface to a 19115 / 19139 compliant metadata file. Metadata can be viewed in the Properties tab for the selected surface.

See "View Surface Properties" on page 204.

The metadata file is saved in XML format.

1. Select the surface in the Layers tab.

Menu	File > Export > Surface Metadata

2. Select File> Export > Surface Metadata command.

The Save As dialog box is displayed.

A default file name is displayed, created from the name of the selected surface with the word "metadata" appended to it, as in: `BASE421_metadata.xml`.

3. [Optional] Type a name for the metadata file.

The default location for the exported file is CARIS > HIPS > *version* > Session.

Export Surface to BAG

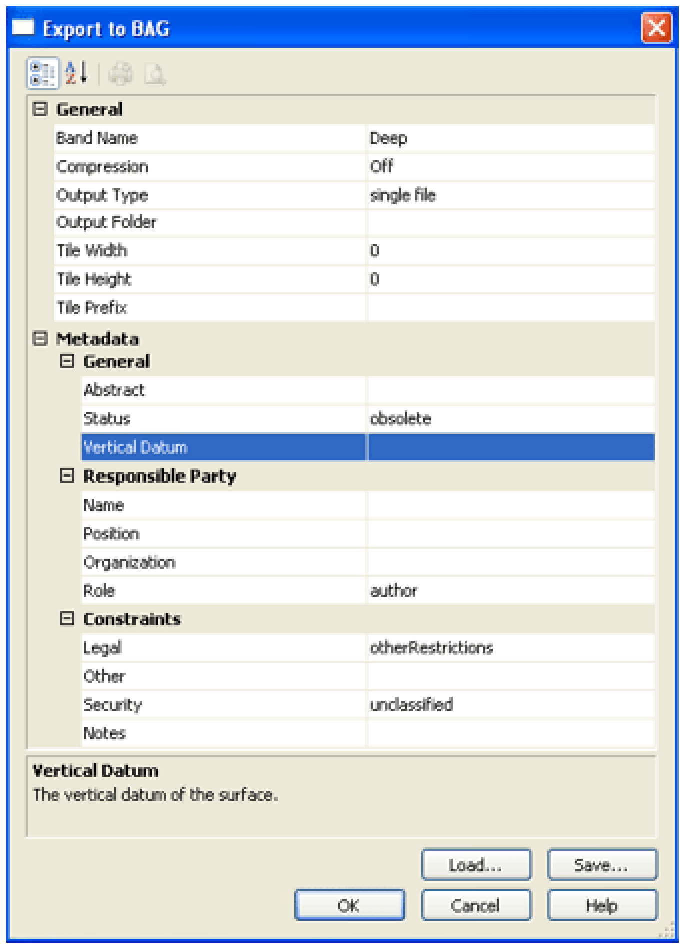
Export a BASE Surface to a BAG (Bathymetric Attributed Grid) file. You can save your export settings to a template file that can be loaded and applied when exporting to BAG again.

1. Select the surface in the Layers tab.

Menu	File > Export > Surface to BAG

2. Select the Export Surface to BAG command.

The Export to BAG dialog box displays the fields for General and Metadata settings for the export.



3. Enter the export settings, or load a previously saved template file.

See "Export Settings" on page 483.

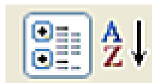
See "Save and Load Templates" on page 484.

4. Click **OK** to perform the export.

The surface is exported using the selected settings. The Output tab shows the status of the export.

Export Settings

You can view the export settings fields in two ways. In Category view the fields are listed under headings. In Alphabetical view the fields are listed in A to Z order. Toggle between these views using the buttons at the top right of the of dialog box.



Category Alphabetical

Field	Function
General (define the input and output settings fro the export)	
Band Name	Select the surface layer to be exported.
Compression	
Output Type	Export the selected BASE surface to a single *Output File*, or as a tile set to multiple files in an *Output Folder*
Output File (single file)	Click **Browse** to save the name and location for the exported file. Name appears in the field.
Output Folder (multiple files)	Click **Browse** to locate the destination folder for the exported tile set
Tile Width	Define width of the multiple tiled files. The units are those set in Tools > Options > Display > Units.
Tile Height	Define height of the multiple tiled files. The units are those set in Tools > Options > Display > Units.
Tile Prefix	Type a prefix for exported tile files. The files will have a tile number appended to the file name, based on the location of the tile in the surface. The tile number is in the format RxC where R and C are the row and column index for the current tile. With a prefix such as RTFM, tiles would be numbered RTFM_1x1, RTFM_2x2 etc.
NOTE: Attempting to export a large surface to a single file may cause an out-of-memory error if there is limited memory allotted to the application. For larger surfaces, it's recommend that you export a tile set.	
Metadata (provide metadata for the exported files)	
General	
Abstract	Type a brief description of the data being exported.

Field	Function
Status	[Optional] Identify the status of the data at the time of export. Select from the drop-down menu: • completed • historicalArchive • obsolete • onGoing • planned • required • underDevelopment
Vertical Datum	Values above the vertical datum are positive.
Responsible Party	identifying information
Name	Identify person responsible for the data
Position	Position title for the responsible person
Organization	Organization of the responsible person.
Role	User role(s) assigned to the responsible party with regard to this data.
Constraints	
Legal	Specify whether the file is legally restricted from being copied, altered, or referenced without permission from the responsible party. Set the constraint from the drop-down list. If you select otherRestrictions, enter details in the Other field.
Other	Type details of legal constraints on the data.
Security	Identify restrictions on access to the exported data by setting its security classification level from the drop-down list.
Notes	Additional information about security constraints.

Save and Load Templates

Some field values are entered automatically when the dialog box is displayed. To automatically populate *all* fields automatically, you can create a template, by setting options for all the fields manually and then saving the settings to an XML file.

This XML template file can then be loaded, so that the fields populated from the settings in the template are applied to another export.

To save export settings as a template:

1. Enter fields settings.
2. Click **Save** to create a template of the export settings.

3. In the Save dialog box, define a name and location for the template file and click **Save**.

To load a saved template:

1. Select the surface in the Layers tab.
2. Select the Export Surface to BAG command.
3. Click **Load**, select the desired template and click **Open**.

The fields are populated with the saved settings.

Export Surface to ESRI ASCII Grid

Save a BASE surface layer to an ESRI ASCII grid. This export creates three output files:

- an *.asc file containing the grid data (this can be opened as a background file in HIPS),
- a *.prj file, which contains coordinate system information,
- and an *.asc.aux.xml file containing metadata.

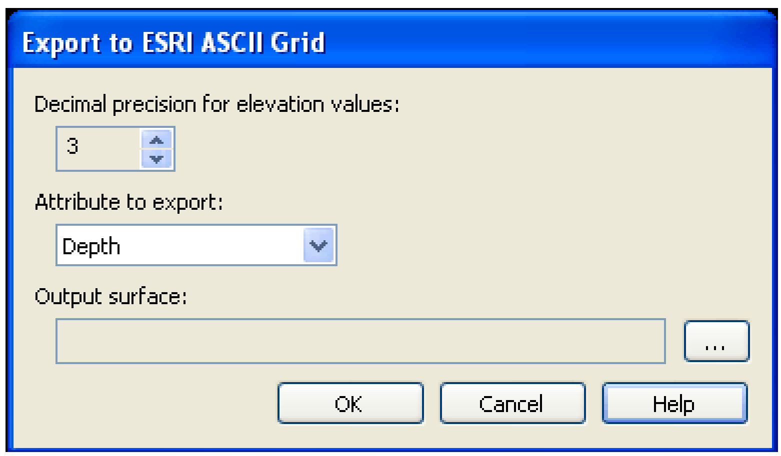
These files can be viewed in a text editor.

1. Select the BASE surface parent layer in the Layers tab.

Menu	File > Export > Surface to ESRI ASCII Grid

2. Select the Export to ESRI grid command.

The Export to ESRI ASCII Grid dialog box is displayed.



Use the *Decimal Precision for elevation values* field to define the number of decimal places to apply to the elevation values in the output files. The maximum number of decimal places is12.

3. Select the value for *Decimal Precision for elevation values* that will be applied to the output data.
4. Select a surface Attribute to export, e.g., Depth or Density, from the drop-down list.
5. Click **Browse** (**...**) in the *Output surface* field and define a name and destination for the exported surface.
6. Click **OK.**

An ESRI ASCII Grid file is created.

> ESRI ASCII and Binary grid formats can be opened in HIPS as “Surfaces” with the Open Background Data cmmand from the File menu.

Export Selection to DXF

Export a selection of data to a DXF (AutoCAD) file. Contours, soundings, lines and bounding polygons can be exported to DXF format.

Data is saved in two files:

- The DXF file contains the exported data.
- The DXF_RXL file contains the coordinate system, unit type and unit scale of the data.

These files can then be opened in CAD applications or viewed in HIPS and SIPS as background data.

Export contours, soundings or bounding polygon

To export from HIPS to DXF:

1. Open the field sheet containing the contours, soundings or bounding polygon to be exported.
2. In the Layers tab, highlight the specific layer, for example, Contours or Depth.
3. In the Display window, select the the data to be exported.

Menu	File > Export > Selection to DXF

4. Select the Export to DSF command.

A Save As dialog box is displayed.

5. Enter a file name and browse to the folder where you will save the exported file.
6. Click **Save**.

The Output window will report the progress of the export and the number of objects exported. The data is saved to the specified folder.

Export Lines or Critical Soundings

To export selected lines or selected critical soundings to DXF:

1. In the Layers tab, highlight the Ship Track Lines or Critical Soundings layer.
2. In the Display window, select the the lines or soundings to be exported.

Menu	File > Export > Selection to DXF

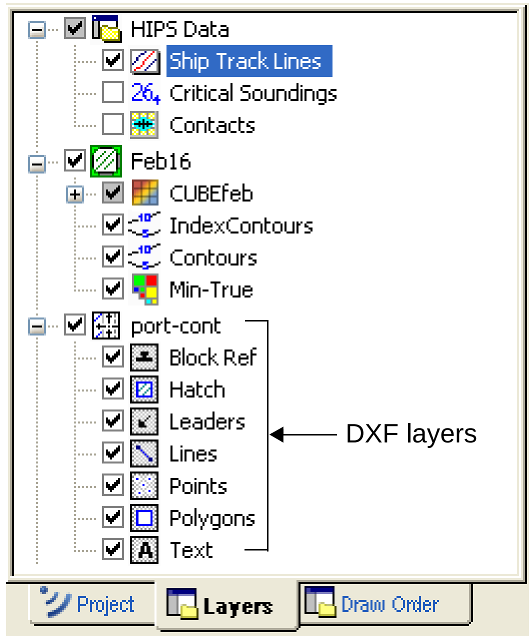
3. Select the Export to DXF command.

A Save As dialog box is displayed.

4. Enter a file name and browse to the folder where you will save the exported file.
5. Click **Save**.

The data is saved to the specified folder.

DXF files can be opened in HIPS and SIPS as background data. The opened file is listed in the Layers tab, as illustrated below.



Export Selection to Formatted ASCII

Export selected soundings and lines to formatted ASCII. The exported files will have the extension *.WKT (well-known text).

To export a selection, for example, of lines to a formatted ASCII file:

Menu	File > Export > Selection to Formatted ASCII

1. Select the lines.
2. Select Export > Selection to Formatted ASCII command.
3. In the *Save As* dialog box, select the folder to export to, and type a file name.
4. Click **Save**.

The selected data is exported to the named WKT file.

Export Selection to GML

Export information about selected features (lines soundings, contacts) to a GML (Geography Markup Language) file.

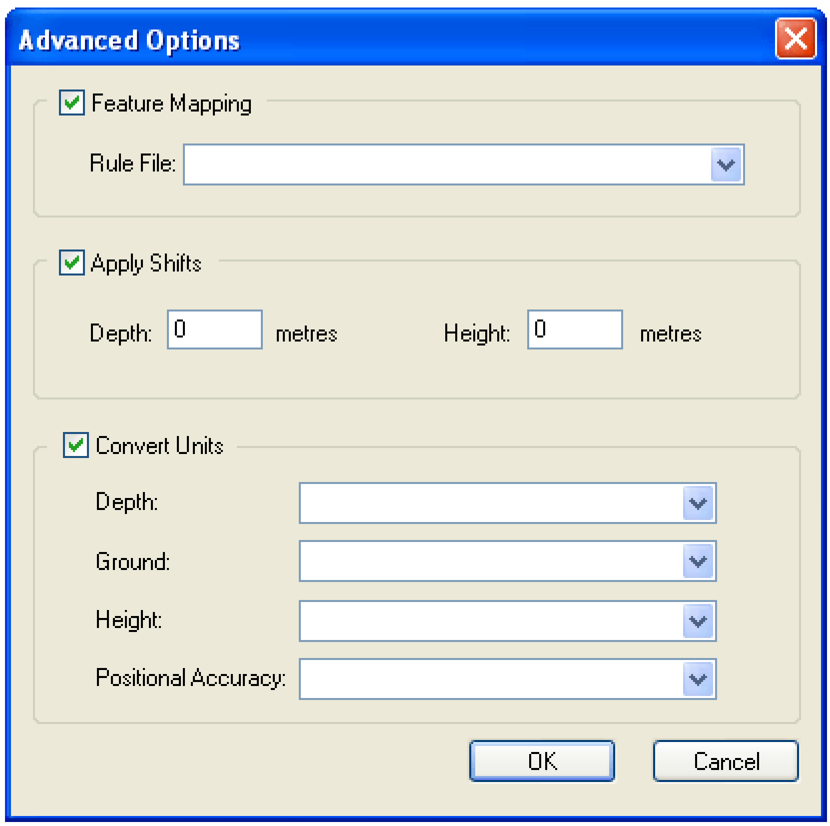
To export to a GML:

1. Select the features to be converted.

Menu	File > Export > Selection to GML

2. Select the Export Selection to GML command.
3. In the Save As dialog box, type a name and select a location for the file.
4. [Optional] Select the *Run System command after export* check box and enter command parameters in the *Command* field. This system command will then be applied to the new GML file when it is created.
5. Click **Save**.

The Advanced Options dialog box is displayed



Feature mapping

There are no default GML mapping files in HIPS. If you want to use a mapping file you must create it before using the Feature Mapping option.

To map the selected HIPS features to GML:

6. Click the check box to enable the Feature Mapping option.
7. From the Rule File drop-down list, select a file to use for mapping features .

Apply features

To apply a custom shift to the depths or height:

8. Click the check box to enable the Apply Shifts option.
9. Enter values in Depth and Height fields.

Convert Units

To convert Depth, Ground, Height or Positional Accuracy units to another unit of measure:

10. Click the Convert Units check box to enable the option.
11. Select a new unit of measure from the drop-down list for each type you want to change.
12. Click **OK**.

Formats Exported

The HIPS Export Wizard is a separate application launched from the HIPS and SIPS interface. TheExport Wizard converts HIPS data products (soundings, BASE Surfaces, mosaics, etc.) to the following formats:

- ASCII text file
- CARIS map
- Bathymetric Attributed Grid (BAG)
- (FAU)
- Generic Sensor Format (GSF) files
- Hydrographic Object Binary (HOB) file
- Hydrographic Transfer Format (HTF) files
- TIFF image
- Unified Sonar Imaging Processing System (UNISIPS) format

See the table below to view export options.

Export Formats

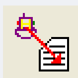
Format	**Description**	**Refer to...**
BAG to ASCII	Export data in Bathymetric Attributed Grid format to ASCII format	"BAG to ASCII" on page 497
BASE Surface to ASCII	Export BASE Surface position and attribute data to a text file.	"BASE Surface To ASCII" on page 498
BASE Surface to Image	Export BASE Surface attribute data as a georeferenced image for further processing in a CARIS product or other application.	"Surface To Image" on page 501
Contacts	Export contacts to a CARIS map, an ASCII file, and/or a TIFF image.	"Contacts" on page 504
GeoBaR to ASCII	Export one or more open GeoBaRs to ASCII format files.	"GeoBaR to ASCII" on page 507
GeoBaR to Image	Export one or more open GeoBaRs to Image files.	"Surface To Image" on page 501
GSF to CARIS map	Export soundings and track lines from one or more Generic Sensor Format (GSF) files to a CARIS map. The GSF data does not have to be converted to HIPS format prior to using this utility.	"GSF to CARIS Map" on page 511
GSF to HOB	Export GSF data to a Hydrographic Object Binary (HOB) file that stores feature objects and associated attribute data. The HOB file can contain links to CARIS spatial objects.	"GSF to HOB" on page 514
HIPS Tide to ASCII	Export tide data from track lines to text format.	"HIPS Tide to ASCII" on page 516

Format	Description	Refer to...
HIPS to ASCII	Export HIPS data to a customized text listing of soundings that you can load into other software systems.	"HIPS To ASCII" on page 517
HIPS to CARIS map	Export HIPS data to a CARIS map. This option contains three modes of exporting data: soundings, track lines and swaths.	"HIPS To CARIS Map" on page 523
HIPS to FAU	Export HIPS data to FAU files.	"HIPS to FAU" on page 527
HIPS to GSF	Export processed HIPS data to GSF format. A new GSF file is created for each trackline that is exported from HIPS. Conversely, when HIPS data is created from GSF data, a copy of the original GSF file is maintained inside the HIPS directory structure. The Export to GSF function updates these GSF files.	"HIPS To GSF" on page 528
HIPS to HOB	Export HIPS data to HOB files that store feature objects and associated attribute data. The HOB file can contain links to CARIS spatial objects. This option contains three modes of exporting data: soundings, track lines and swaths.	"HIPS To HOB" on page 529
HIPS to HTF	Export data to a Hydrographic Transfer Format (HTF) file. The HTF is a text file that contains a header section and a sounding record. The Export wizard can save a header section so it can be loaded into multiple HTF files.	"HIPS To HTF" on page 531
Mosaic to ASCII	Export mosaic layers to an ASCII file.	"Mosaic to ASCII" on page 535
Mosaic to Image	Export a mosaic as a TIFF image for further processing in a CARIS product or other application.	"Surface To Image" on page 501
Sediment Analysis to ASCII	Export the results of Sediment Analysis done in Mosaic Editor.	"Sediment Analysis to ASCII" on page 539
SIPS to UNISIPS	Export SIPS data to a Unified Sonar Imaging Processing System (UNISIPS) format.	"SIPS To UNISIPS" on page 540

Export Wizard

The first step to exporting data with the Export Wizard is to select the file format to be exported. The number of dialog boxes displayed in the subsequent steps depends on the export format selected in the Step 1 dialog box.

To use the HIPS Export Wizard:

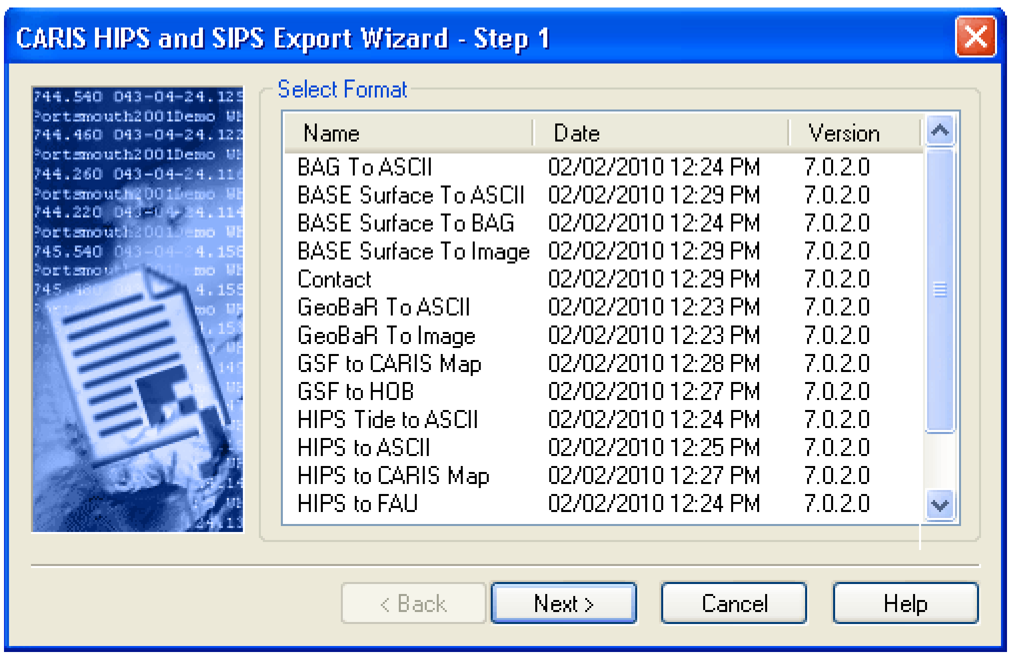
Menu	File > Export
Tool	

1. Select the Export command.

The Export Wizard is displayed.

Step 1: Select Format

The Select Format dialog box lists all the formats available for exporting data.



1. Select an export format.
2. Click **Next** to select data to export.

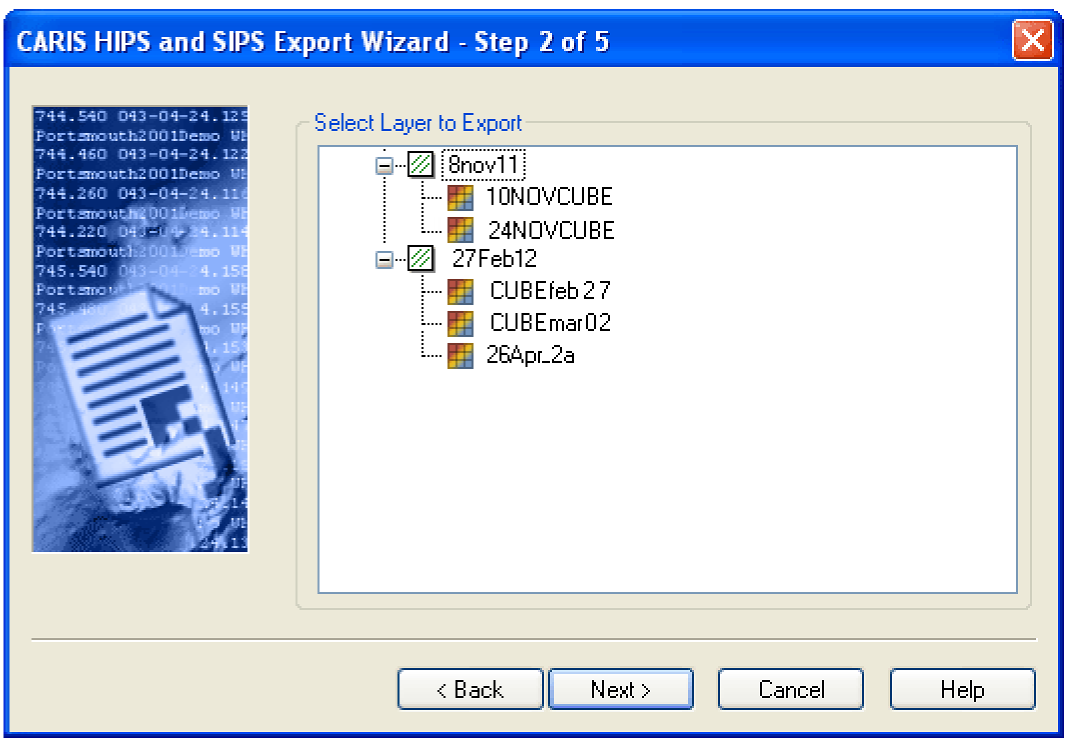
Step 2: Select Files

Depending on the format being exported, this dialog box will display either available field sheet layers or a project tree. For example, when exporting a BASE surface, available layers are displayed; when exporting HIPS data, the Project/Vessel/Day/Line tree for all available project files is displayed.

> If you are exporting line data, for example, HIPS to ASCII or HIPS to HOB, you can select specific track lines before you activate the Export Wizard. You can also adjust this selection in the Step 2 dialog box.

Export a surface

If you select a BASE Surface, a mosaic, a GeoBaR (or multiple GeoBaRs) to export, the Select Layer to Export dialog box will display the open files of that type below the associated field sheet.



1. Expand the file tree and select the layer you want to export.

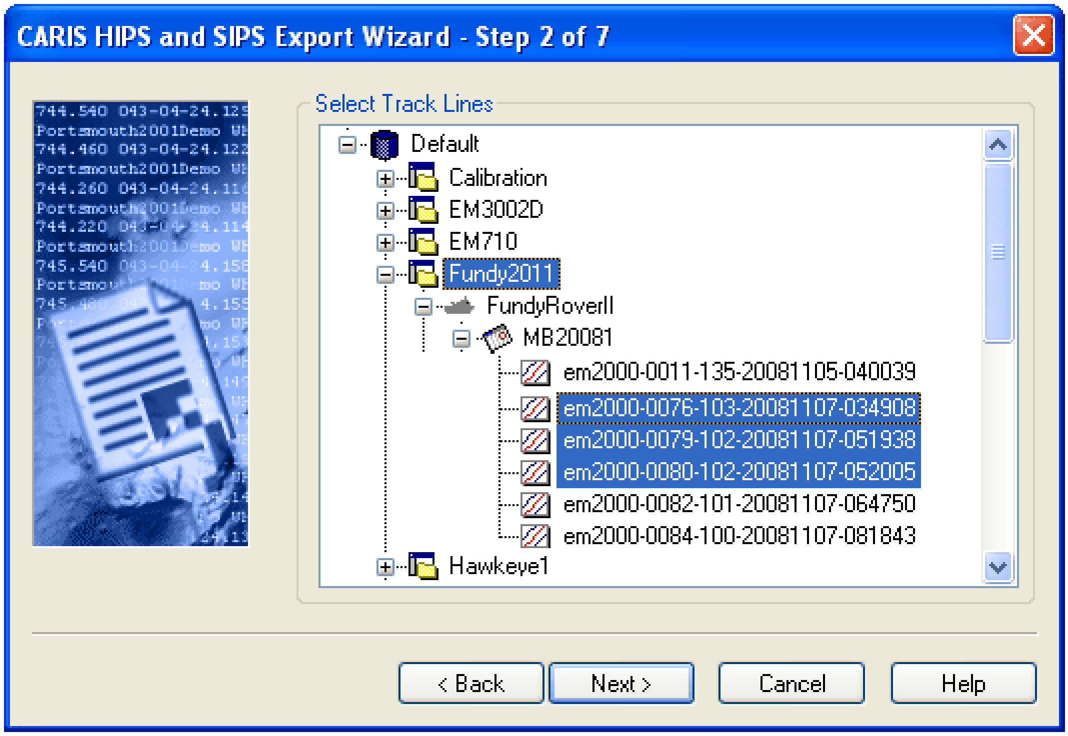
You can export multiple GeoBaRs to ASCII or to Image.

2. Hold down the <Ctrl> key while selecting the GeoBaRs to export specific GeoBaRs.
3. To export all the listed GeoBaRs, hold down the <Shift> key and select the first and last in the list.

Export contacts or line data

For these file formats the dialog box displays the existing projects, organized in a Project/Vessel/Day/Line structure.

.



1. Expand the file tree to select the trackl ines to export.

If you have selected lines to export before opening the Export Wizard, they will be selected here.

To add lines to this selection:

2. Hold down the <Ctrl> key and select additional lines.

BAG to ASCII

Data from a Bathymetric Attributed Grid (BAG) file can be exported using the BASE Surface to ASCII options.

1. Open the Export Wizard.
2. Select BASE Surface to ASCII from the list in the Step 1 dialog box.
3. Click **Next**.
4. Select the BAG surface from the layer tree at Step 2.
5. Click **Next** .

See "BASE Surface To ASCII" on page 498.

To export metadata for the BAG surface:

Menu	File > Export > Surface Metadata

1. Open the BAG file as background data.
2. Select the BAG layer.
3. Select the Export Surface Metadata command.

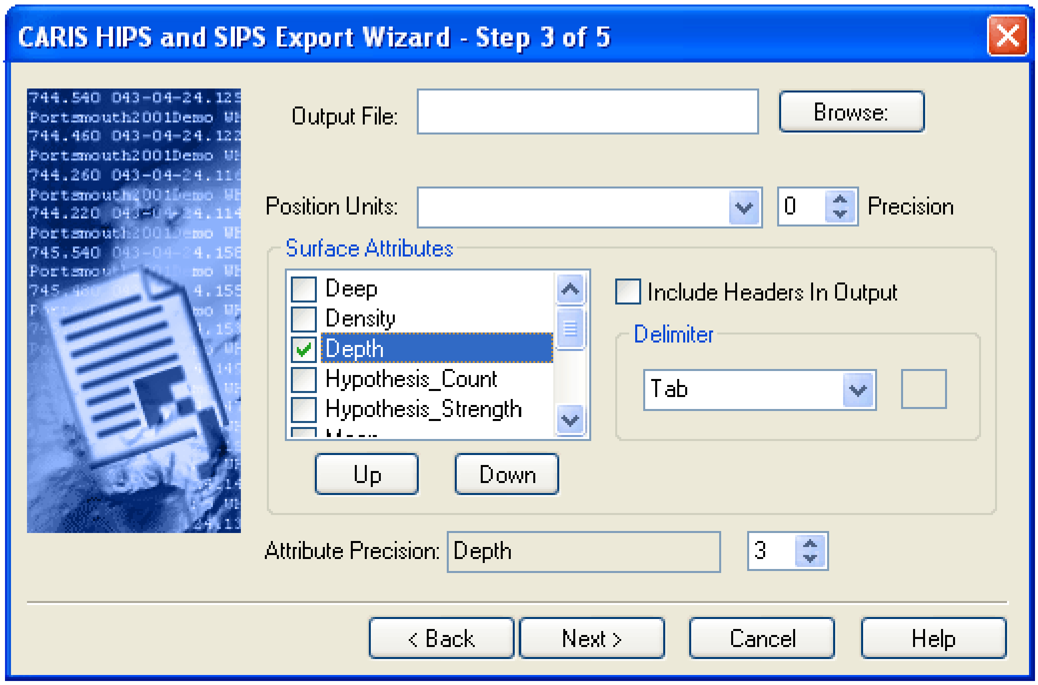
The Save As dialog box will auto-fill the *File name* field with the name of the BAG file with `_metadata` added to the file name. You can type an alternate name.

4. Click **Save**.

BASE Surface To ASCII

Export BASE Surface attributes to an ASCII text file.

BASE Surface to ASCII (Step 3)



1. Type the path and file name for the output file , or **Browse** to the destination folder.
2. Type the file name and click **Save**.
3. Select the *Position Units* for data from the drop-down list:
   - Ground (East, North): position displayed as Easting, Northing to the precision defined (for example:`360425.000, 4770355.000`)
   - Geographic (DMS): position displayed as Latitude, Longitude in degrees, minutes, seconds to the precision defined (for example: `43-04-23.03N, 070-42-51.89W`
   - Geographic (Unformatted DMS): displays the position without using hyphens or direction indicators, for example, `43 04 23.03, -070 42 51.89.`
   - Geographic (Decimal Degrees): position displayed as Latitude, Longitude, in decimal degrees to the precision set, for example, 43.073 -070.714.
   - Geographic (Degree Minutes): position displayed as Latitude, Longitude, in degrees and minutes, to the precision set, for example, 34-23.610N,119-52.585W.

Note that for positions in the northern hemisphere, longitude West values are negative; in the southern hemisphere, the South latitude values are negative. For example, a position near Portsmouth NH, USA is `43 04 23.03, -070 42 51.89`. A position near Sydney Australia is `-33 51 00.84 151 11 43.7`

4. Specify the precision for the display of position units, by using the *Precision* up or down arrow button to set the number of digits to be displayed to the right of the decimal point.
5. Select the check boxes for the *Surface Attributes* values to be recorded in the ASCII file.
6. Click **Up** and **Down** to move the attributes into the order you wish them to appear in the file.
   - All attributes checked in the list will be included in the ASCII output.
7. [Optional] Select *Include Headers in Output* to have the appropriate column headers included in the ASCII file.
8. Select either Space, Comma, or Tab from the list to use these characters to separate fields in the output file. If you select Other, type a character to use a delimiter.
9. Set the *Attribute Precision* value (number of digits to the right of decimal point) for the selected attributes.

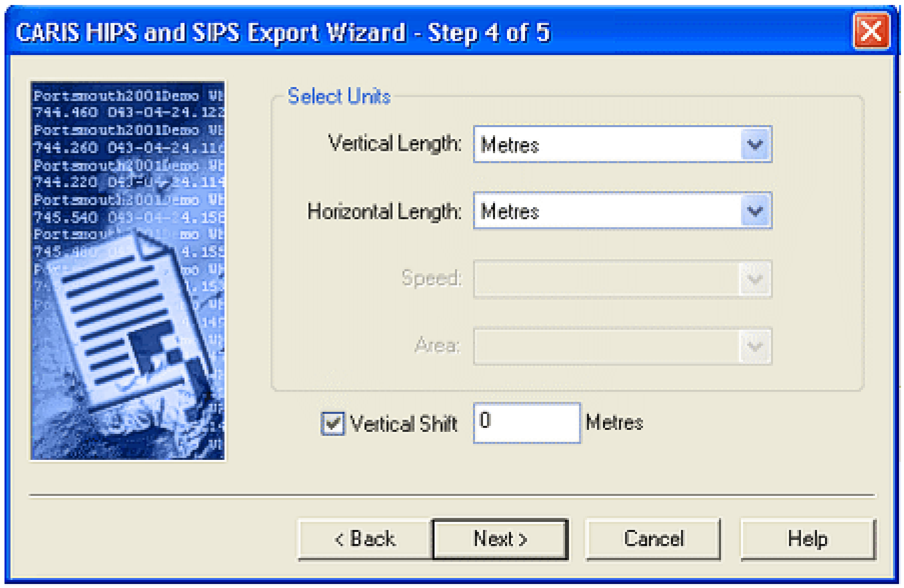
The example below shows a comma-delimited ASCII file for the Depth attribute of a BASE surface with the positions in decimal degrees, and column headers turned on.

```
//Lat (DD) Long (DD)Depth  Mean Shoal
43.073, -070.714,  5.989, 5.99, 5.895
43.073, -070.714,  6.459, 6.46, 6.045
43.073, -070.714,  6.906, 6.93, 6.490
43.073, -070.714,  7.443, 7.44, 6.936
43.073, -070.714,  8.002, 8.00, 7.491
43.073, -070.714,  6.628, 6.63, 6.130
```

10. Click **Next**.

Select Export Units

This dialog box sets the unit type for recording data. The fields that are active in this dialog are determined by the attributes selected in the previous dialog box.



Available units are:

- metres and kilometres
- fathoms (US and international)
- feet (US and international)
- yards (US and international)
- miles (US, international, and nautical)

1. In the *Vertical Length* field, select units for depth and uncertainty values.

The *Horizontal Length* field is only active if you selected Ground in the *Position Units* field in the previous dialog box.

2. Select the units for position data.

To apply a static vertical shift to data during export:

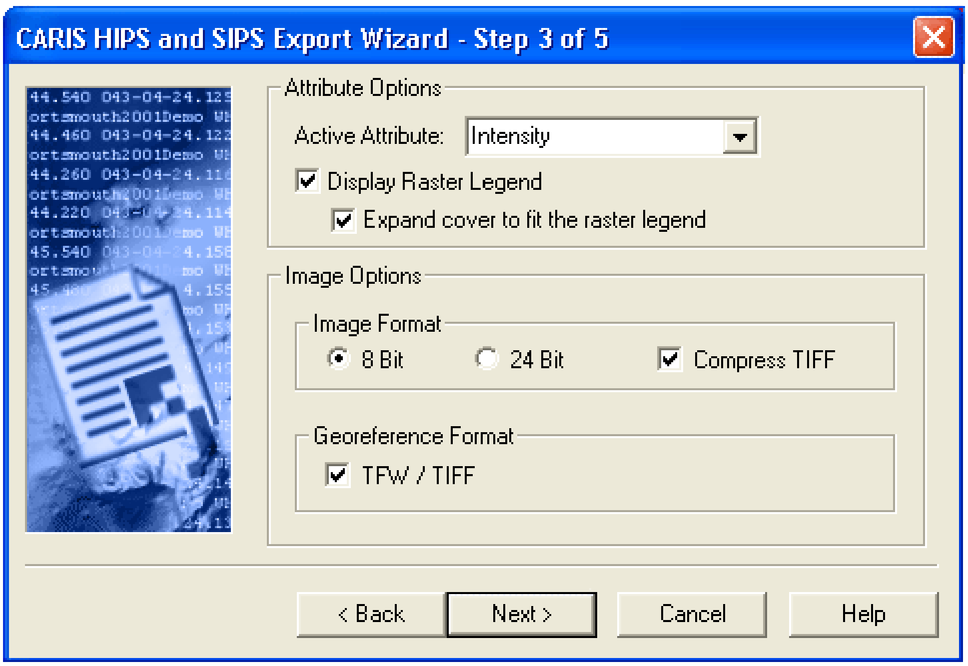
3. Select the *Vertical Shift* check box and type the value for the shift (in metres).
4. The shift will be applied to any layer of the surface containing vertical data.
5. Click **Next** to continue setting export parameters. "Export" on page 542

If you are exporting attribute data from a BASE Surface created prior to HIPS and SIPS 6.1, this dialog box is not displayed because the default measurement unit is metres.

Surface To Image

Export a BASE Surface, a mosaic, or one or more GeoBaRs to an image file.

Attribute and Image Options



Attribute Options

If the selected attribute layer has a legend associated with it, you can include this legend in the exported image. The legend will be overlaid on the surface, in the position and with labels as set in the Properties for the selected attribute layer.

1. From the *Active Attribute* field, select the layer to display as a geo-referenced image.
2. Select *Display Raster Legend* to include the raster legend for the selected attribute layer in the exported image.

   Select *Expand cover to fit the raster legend* to expand the area of the image so that the legend does not overlay the surface.

Image Options

Image format options can help reduce file sizes:

3. Select a colour format for the image.
   - *24 Bit (*16 million colours)
   - *8 Bit* (256 colours)
4. Select *Compress TIFF* to decrease the file size of the TIFF during export.

Georeference Format

Selecting the TFW / TIFF check box will generate two ASCII files in addition to the TIFF image. One, with file extension CSPRPOJ, contains Coordinate system information for the image. The other, with extension TFW, contains the ground coordinates. These can be read using a text file reader.

5. Select *TFW/TIFF* as to produce additional files.
6. Click **Next**.

Image Output Options

You can export a BASE surface layer, a single GeoBaR layer, or a Mosaic layer to a TIFF image file, or to a tiled set of images, or to both formats at the same time.

More than one GeoBaR can be exported at a time, in which case the TIFF files are saved either to a single folder, or to folders which follow the PVDL (Project/Vessel/Day/Line) structure.

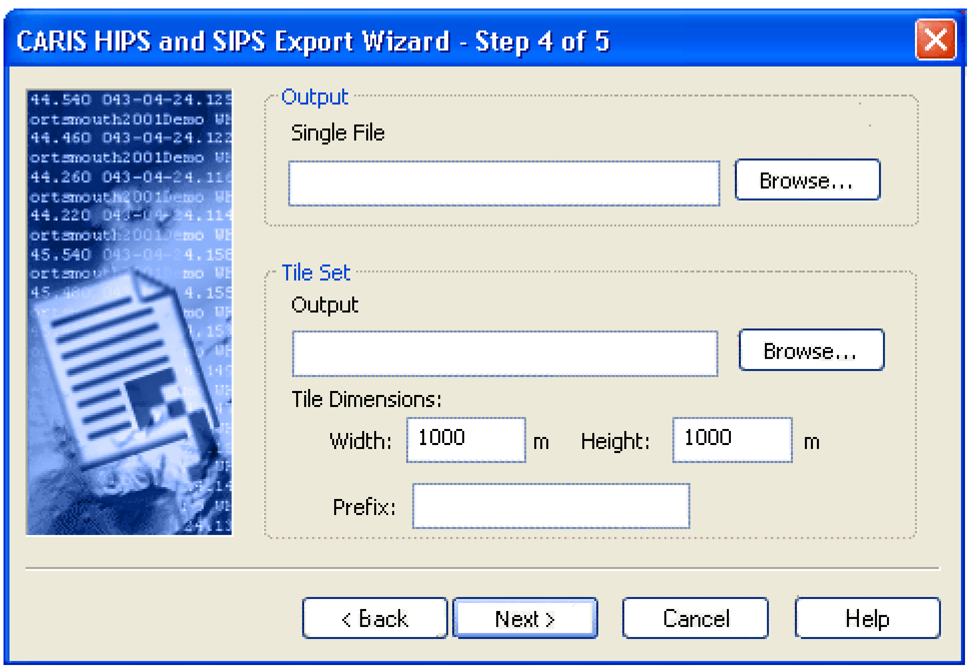


Image Output

1. Click **Browse**, and set the *Output* file path.
2. Type a name for the file name and click **Save**.

If you are exporting multiple GeoBaRs:

3. Click **Browse** to select the folder where the exported files will be saved. (The TIFF files in the output folder will have the same names as the GeoBaRs they were created from.)

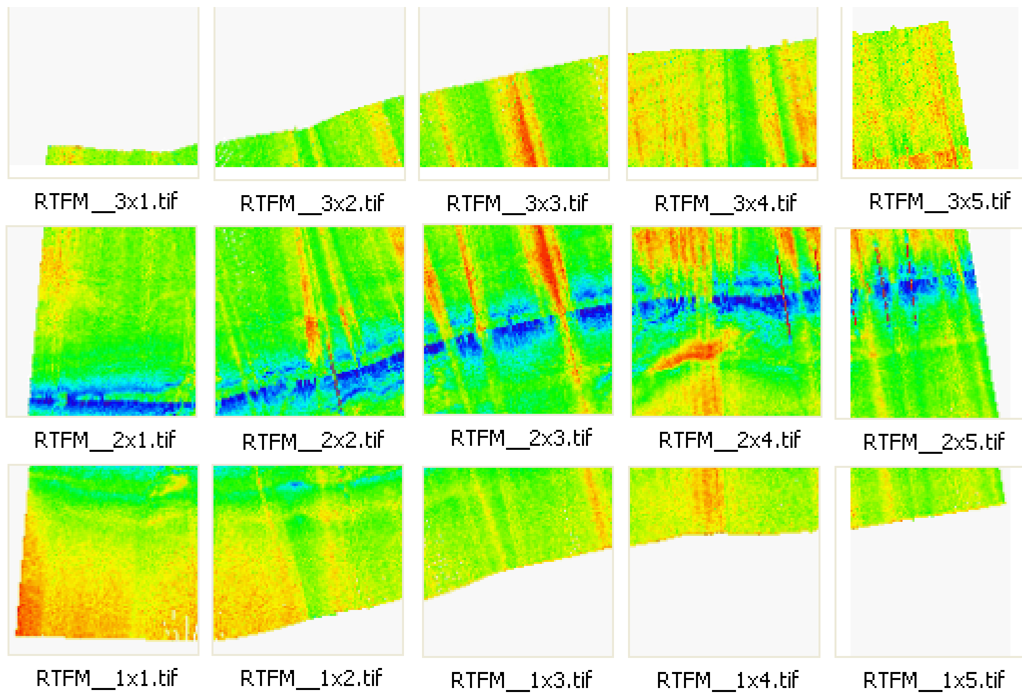
   Select *Maintain PVDL Hierarchy* to save each TIFF file within its Project/Vessel/Day/Line structure in the folder you designate.

Tiled image output

Some attribute layers may be too large to view effectively as one image. In this case, you can export the layer divided into a set of tiled images, with set dimensions.

These files are given a prefix, to which a tile number is appended. The tile number is in the format “RxC” where “R” and “C” are the row and column index for the current tile.

For example, with prefix RTFM, the first row of tiles would be numbered RTFM_1x1, the second RTFM_1x2 etc., as illustrated below.



The tiled sets of TIFF images are all saved to the folder selected in the Tile Set *Output* field.

4. Click **Browse**, and select the path to the *Output Folder* for the tile images.
5. Set the Tile Dimensions to specify the size of each tile. The units used here are those set for *Horizontal Length* in Tools > Options > Display > Units.
6. Type a prefix for the exported tile files.
7. Click **Next**.

“Export” on page 542

Contacts

Export contact data for your entire project to a CARIS map and/or an ASCII text file. The ASCII files are structured to be easily imported into a relational database. You can also export contact snapshot images to TIFF files.

If you select ASCII, you will have to browse to a folder location and type a file name. This name is used as the first component name of four files that will be created to store the contact information:

- **filename_ContactLine.txt** contains information identifying the survey line and indexes to the other files.
- **filename_ContactSingle.txt** contains all the information for single point contacts.
- **filename_ContactMulti.txt** contains all information for line contacts.
- **filename_ContactMultiPoints.txt** contains all location information for multipoint contacts.

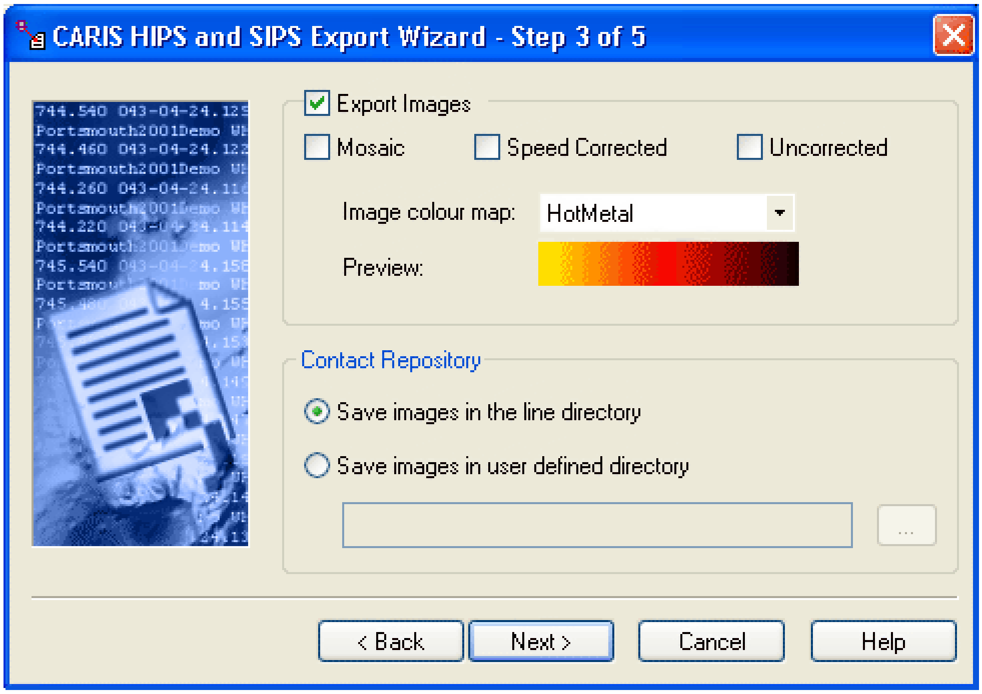
For information on the structure of exported contact files, see CONTACT FILE FORMATS ON PAGE 139 OF THE HIPS AND SIPS REFERENCE GUIDE

Contacts (Step 3)

1. Select the *Export Image* check box to export the contact(s) in TIFF image format.
2. If you select *Export Image*, select from the following image formats:
   - *Mosaic*: This option creates a georeferenced TIFF image from contact data.
   - *Speed Corrected*: This option calculates the distance between profile lines in the contact and corrects for vessel speed.
   - *Uncorrected*: This option does not correct for vessel speed and renders a one-to-one correspondence between the side scan image and the contact.
3. To export the colour map that was selected in the Side Scan Editor, select the *Use Colour Map Specified in Contact File*.
4. To export the contact with a new colour map, select the *Use Other Colour Map* option and select the map from the drop-down list.

The selected colour map is displayed in the *Preview* field.

5. Select the directory location for the contact. You can select from two options:



- *Save Images in the Line Directory*: This option saves the contact data to the line file in the HIPS/SIPS Project/Vessel/Day directory structure that you selected in the second dialog box.
- *Save Images in User Defined Directory*: This option saves the contact data in another directory. Click **Browse** to select the folder where you want to save the data.

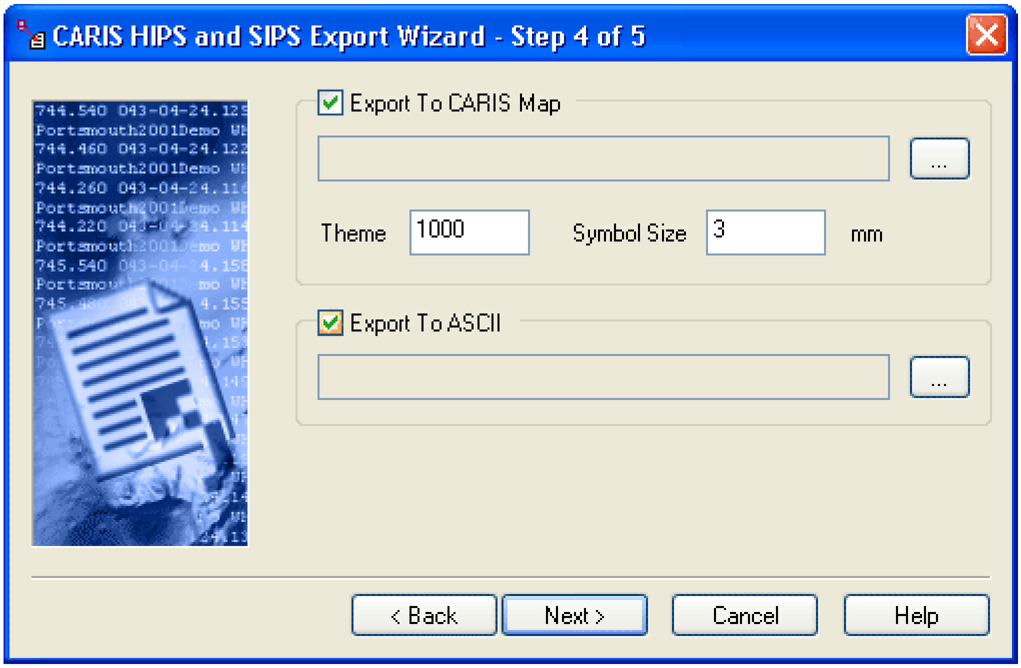
6. Click **Next**.

Contacts (Step 4)

This dialog box contains options for exporting contact data to an ASCII or CARIS file.

1. Select either the *Export to CARIS Map* or *Export to ASCII* check boxes. Both can be selected at the same time.
2. Type a file name and path to store the contact data after export, or click **Browse** to select the file path (both CARIS data and ASCII text files are typically stored in ..\Hips\CARIS_Files).
3. Type a layer number for the contact data in the *Theme Number* field.
4. Type a symbol size for the contact data in the *Symbol Size* field.
5. Click **Next**.

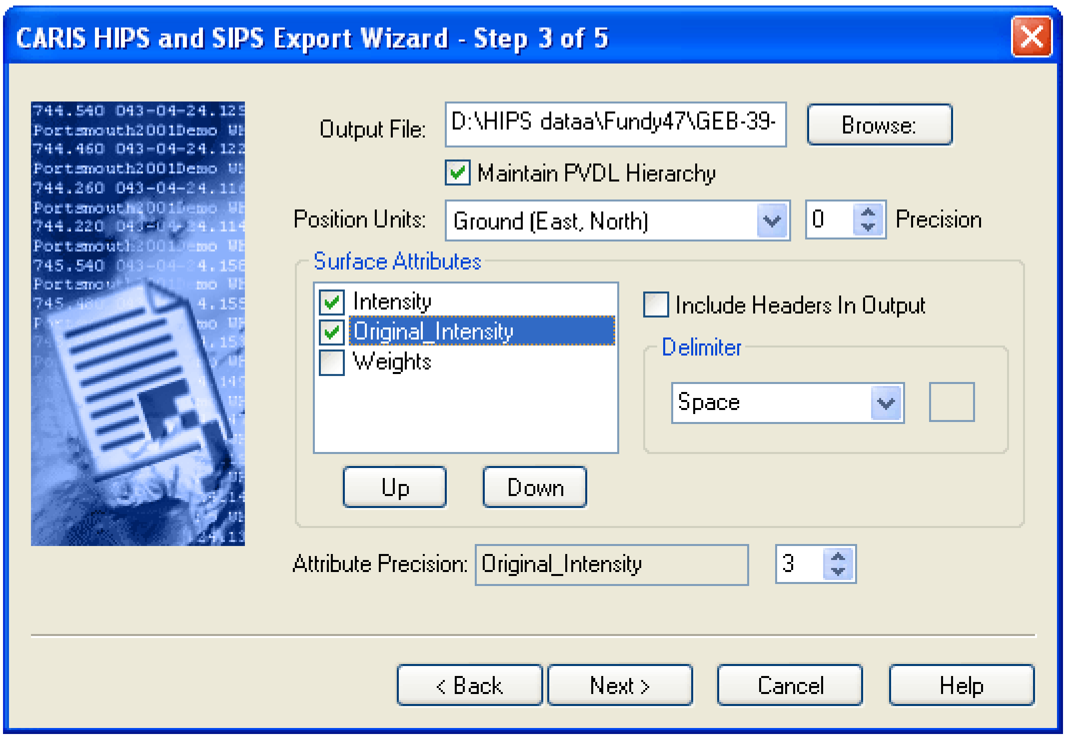
"Export" on page 542


CARIS HIPS and SIPS Export Wizard - Step 4 of 5
Export To CARIS Map
...
Theme
1000
Symbol Size
3
mm
Export To ASCII
...
< Back
Next >
Cancel
Help

GeoBaR to ASCII

Export GeoBaR attributes to an ASCII text file.

GeoBaR to ASCII Output Options



Output file location

If you are exporting one GeoBaR:

1. Type the path and file name for the output, or click **Browse** to go to the destination folder.
2. Type the file name and click **Save**.

If you are exporting multiple GeoBaRs:

3. Click **Browse** to select the folder where the exported files will be saved. (The ASCII files in the output folder will have the same names as the GeoBaR they were created from.)
4. Select *Maintain PVDL Hierarchy* to save each ASCII file within its Project/Vessel/Day/Line structure in the folder you designate. (The files will have the same names as the GeoBaR they were created from.)

Position units

5. Select the *Position Units* for data from the drop-down list:
   - Ground (East, North): position displayed as Easting, Northing to the precision defined (for example:`360425.000, 4770355.000`)
   - Geographic (DMS): position displayed as Latitude, Longitude in degrees, minutes, seconds to the precision defined (for example: `43-04-23.03N, 070-42-51.89W`

- Geographic (Unformatted DMS): displays the position without using hyphens or direction indicators, for example, `43 04 23.03, -070 42 51.89.`
- Geographic (Decimal Degrees): position displayed as Latitude, Longitude, in decimal degrees to the precision set, for example, 43.073 -070.714.
- Geographic (Degree Minutes): position displayed as Latitude, Longitude, in degrees and minutes, to the precision set, for example, 34-23.610N,119-52.585W.

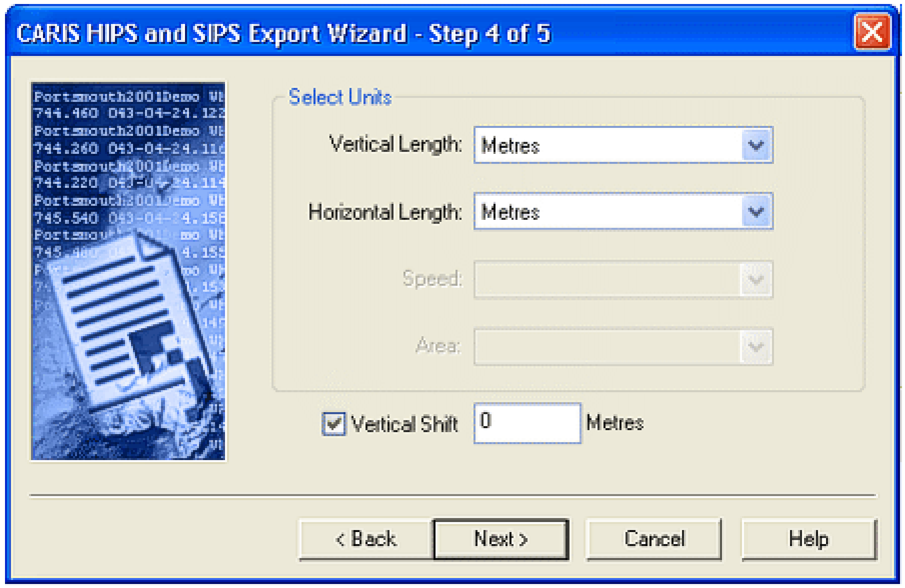
Note that for positions in the northern hemisphere, longitude West values are negative; in the southern hemisphere, the South latitude values are negative. For example, a position near Portsmouth NH, USA is `43 04 23.03, -070 42 51.89.` A position near Sydney Australia is `-33 51 00.84 151 11 43.7`

6. Specify the precision for the display of position values, by using the *Precision* up or down arrow buttons to set the number of digits to be displayed to the right of the decimal point.
7. Select the BASE Surface attribute values to be recorded in the ASCII file by selecting the attribute in the *Available* column and clicking **Add** to move it to the *Active* column.
8. To remove an attribute, select the attribute from the *Active* column and click **Remove** to move it back to the *Available* column.
9. Choose a *Attribute Precision* value (number of digits to the right of decimal point) for an attribute in the *Active* column by selecting an attribute and clicking the up or down arrow buttons.
10. Select the *Headers* check box to print attribute names at the top of the attribute rows in the output file.
11. Select either Space, Comma, or Tab from the list to use these characters to separate fields in the output file. If you select Other, type a character to use a delimiter.

Click **Next**.

Select Export Units

This dialog box sets the unit type for recording data. The fields that are active in this dialog are determined by the attributes selected in the previous dialog box.



Available units are:

- metres and kilometres
- fathoms (US and international)
- feet (US and international)
- yards (US and international)
- miles (US, international, and nautical)

1. In the *Vertical Length* field, select the units for depth and uncertainty values.

The *Horizontal Length* field is only active if you selected Ground in the *Position Units* field in the previous dialog box.

2. Select the units for position data.
3. Click **Next**.

"EXPORT" ON PAGE 542

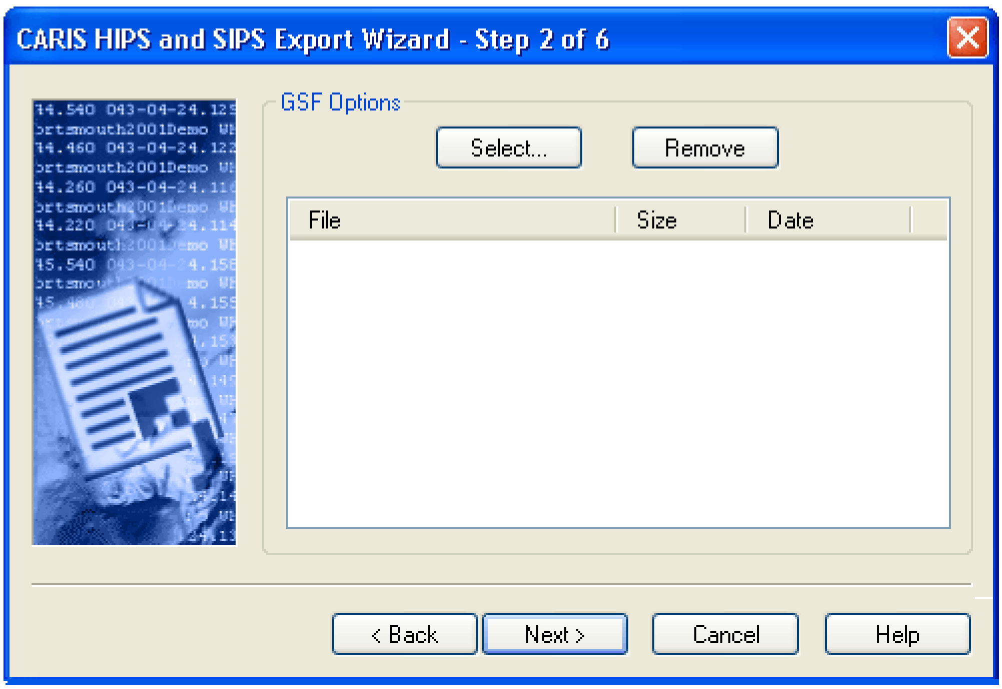
GeoBaR to Image

Please see "SURFACE TO IMAGE" ON PAGE 501 for the description of this export process.

GSF to CARIS Map

The Generic Sensor Format (GSF) to CARIS function loads soundings and track lines from one or more GSF files into a CARIS map. GSF data does not have to be converted to HIPS data prior to using this utility. For exporting to HOB, see "GSF to HOB" on page 514.

Select GSF Files

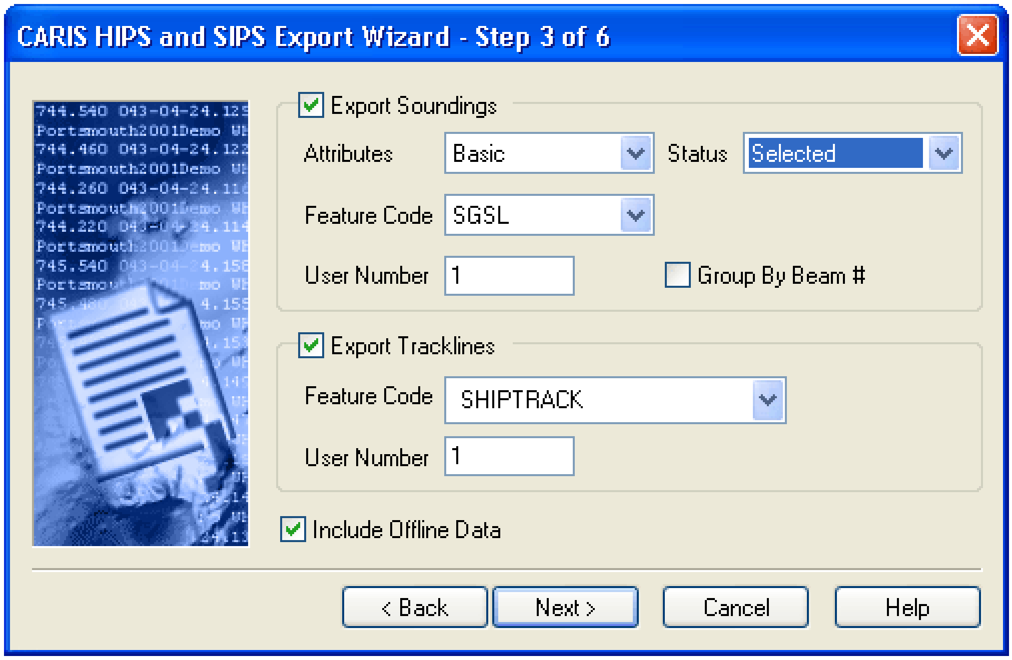


In this dialog box you can list the file path to the GSF file(s) you want to export.

1. Click **Select** to choose the GSF files to be exported.
2. Hold the <**Ctrl**> key while selecting to add more than one file to the list.
3. To remove a file from the list, highlight the file and click **Remove**.
4. Click **Next**.

Export Soundings and Track lines

Data can be exported either in the form of soundings or track lines:



- **Soundings**. Export all soundings that fit within the boundaries of the work file (subject to the HIPS Sounding Status and Data Thinning settings).
- **Track Lines**. Export a line that connects the centre of each swath for each survey line, thus showing the track of the vessel along each survey line. It is not necessary to export soundings to use this option.

1. Select either *Export Soundings* or *Export track lines* or both.
2. If you selected *Export Soundings*, complete any or all of the following fields:
   - *Attributes*: Select *Basic* or *Advanced* from the drop-down list.

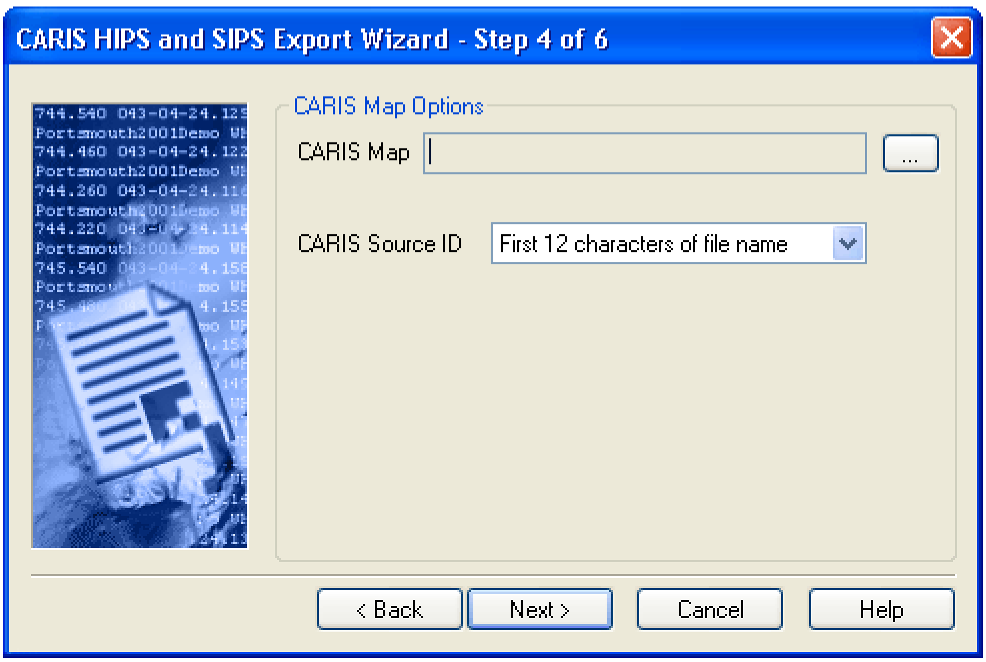
     The *Basic Attributes* option creates mandatory CARIS feature attributes for the map file, such as feature code, user number and source ID.

     The *Advanced Attributes* option creates additional CARIS feature attributes for soundings such as time stamps, launch identifier and tidal correction.
   - *Status*: Select soundings flagged as Selected, Accepted or Accepted and Rejected.
   - *Feature Code*: Select a feature code for the soundings from the drop-down list.
   - *User Number*: Type a number value for layering data in the CARIS map or HOB file.
   - *Group By Beam*: Load soundings into user numbers according to sonar beam number. For example, all soundings from beam 37 are loaded to user number 37 in the CARIS map file or HOB file
3. If you selected *Export Tracklines*, complete any or all of the following fields.

- *Feature Code*: Select a feature code for the track lines object from the drop-down list.
- *User Number*: Type a number value for layering data in the CARIS map.

4. Select the *Include Offline Data* check box to include data recorded during turns between track lines.
5. Click **Next**.

CARIS Map Options



Use this dialog box to set the destination of the exported data.

1. Click the Browse button [**...**] to select CARIS map (*.des file) for data export, or to create a new map file.

The map file path and name are displayed in the *CARIS Map* field.

The *CARIS Source ID* is a 12-character alphanumeric attribute in CARIS maps. The ID can be the first or last 12 characters in a GSF project name.

2. Choose either the first 12 characters or the last 12 characters for the *CARIS Source ID* from the drop-down list.
3. Click **Next** to select the projection for the exported data.

GSF to HOB

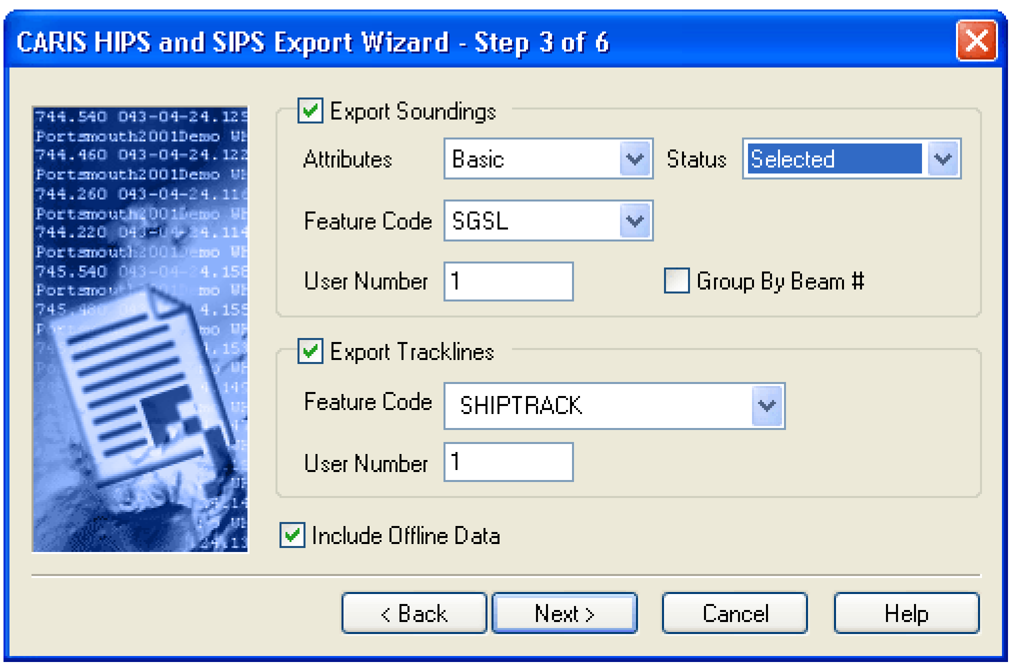
The Generic Sensor Format (GSF) to CARIS function loads soundings and track lines from one or more GSF files into a HOB file. GSF data does not have to be converted to HIPS data prior to using this utility. For information on converting GSF to CARIS map, see "GSF to CARIS Map" on page 511.

GSF Files to Export

In this dialog box you can list the file path to the GSF file(s) you want to export.

1. Click **Select** to choose the GSF files to be exported.
2. Hold the <**Ctrl**> key while selecting to add more than one file to the list.
3. To remove a file from the list, highlight the file and click **Remove**.
4. Click **Next**.

Export Soundings and Track lines



Data can be exported either in the form of soundings or track lines:

- **Soundings**. Export all soundings that fit within the boundaries of the work file (subject to the HIPS Sounding Status and Data Thinning settings).

- **Track Lines**. Export a line that connects the centre of each swath for each survey line, thus showing the track of the vessel along each survey line. It is not necessary to export soundings to use this option.

1. Select either *Export Soundings* or *Export track lines* or both.
2. If you selected *Export Soundings*, complete any or all of the following fields:
   - *Attributes*: Select *Basic* or *Advanced* from the drop-down list.

     The *Basic Attributes* option creates mandatory CARIS feature attributes for the map file, such as feature code, user number and source ID.

     The *Advanced Attributes* option creates additional CARIS feature attributes for soundings such as time stamps, launch identifier and tidal correction.
   - *Status*: Select soundings flagged as Selected, Accepted or Accepted and Rejected.
   - *Feature Code*: Select a feature code for the soundings from the drop-down list.
   - *User Number*: Type a number value for layering data in the CARIS map or HOB file.
   - *Group By Beam*: Load soundings into user numbers according to sonar beam number. For example, all soundings from beam 37 are loaded to user number 37 in the CARIS map file or HOB file
3. If you selected *Export Tracklines*, complete any or all of the following fields.
   - *Feature Code*: Select a feature code for the track lines object from the drop-down list.
   - *User Number*: Type a number value for layering data in the CARIS map.
4. Select the *Include Offline Data* check box to include data recorded during turns between track lines.
5. Click **Next**.

GSF to HOB (Step 4)

Use this dialog box to set the location of HOB file to contain the exported GSF data.

1. Click the Browse button [**...**] to select the HOB file for data export, or to create a new file.

The file path and name are displayed in the *HOB file* field.

2. Click **Next**.

"Export" on page 542

HIPS Tide to ASCII

Export HIPS tide data from selected track lines to a TID file. You can create TID files from single or multiple track lines in a project. TID files can be viewed in a text editor.

HIPS Tide to ASCII (Step 3)

The Output Options dialog box creates the output tide file.

1. Click **Browse** and use the standard Windows Open dialog box to select a path and name for the output TID file.

The path and name of the tide file is displayed in the *Output File Name* field.

2. Select the *Include Rejected Records* check box to include tide records that have been rejected.
3. Select *GPS Tide* to use tides calculated from a GPS ellipsoid height.
4. Select *Smooth data* to use tide data that has been smoothed for localized variability.

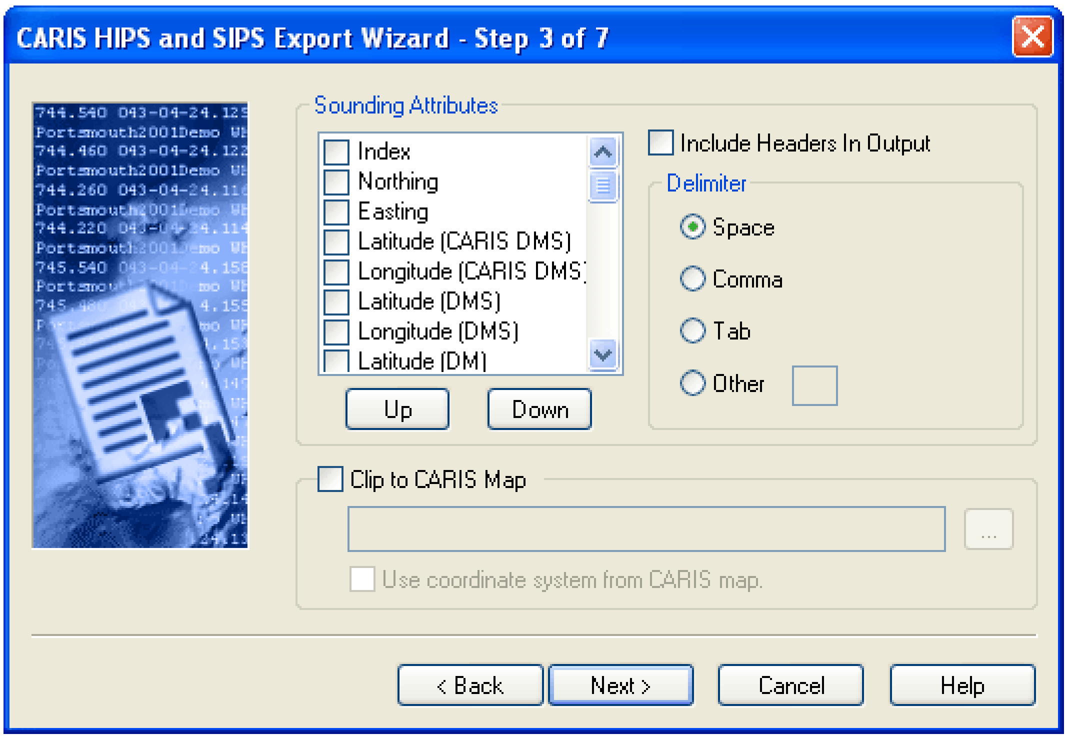
"EXPORT" ON PAGE 542

HIPS To ASCII

The HIPS to ASCII function creates a customized text listing of soundings that you can load into other software systems.

HIPS to ASCII (Step 3)

This dialog box lists the attributes that can be exported to the ASCII file and the delimiters used to separate data in the file.



The attributes that are available to export will vary with the type of data being exported, for example, if single beam data is being exported, Primary Depth and Secondary Depth are listed with other attributes in the Available list.

1. Select the sounding attributes to be included in the ASCII file.
   - To add an attribute, select the attribute in the *Available* list and click **Add** to move it to the *Active* list.
   - To remove an attribute, select the attribute in the *Active* list and click **Remove** to move it to the *Available* list.
2. To change the order that attributes are exported, select the attribute in the *Active* list and click the up or down arrow buttons.
3. Select the *Headers* check box to create print attribute names at the top of the attribute rows.

4. Select either the *Space*, *Comma*, or *Tab* check boxes to use these characters as delimiters, or select the *Other* check box and type a character to use as a delimiter.

You can define the boundaries of the ASCII map data from the boundaries of an existing CARIS map. If you do not select this option, this is defined from the ASCII data.

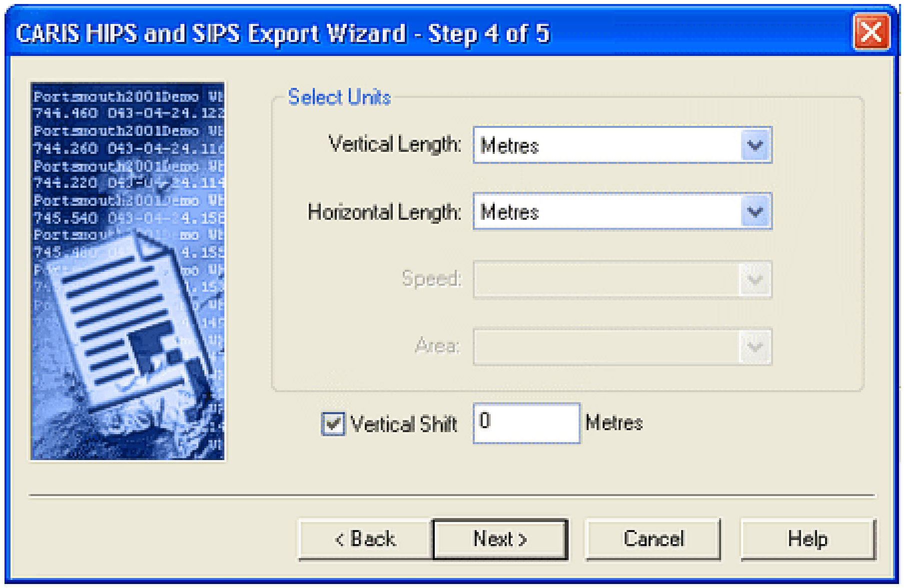
5. Select the *Clip to CARIS Map* check box to use a CARIS map to define the boundaries of the ASCII map data.
6. Click **Browse** to select an existing CARIS map.
7. To use the coordinate system from the selected CARIS map for the ASCII data, click the check box.

If you selected the *Use Coordinate System From CARIS Map* option, the next dialog box prompts you to export the data. If you did not select this option, the next dialog box prompts you to select a coordinate system.

8. Click **Next**.

Select Export Units

This dialog box sets the unit type for recording data. The fields that are active in this dialog are determined by the attributes selected in the previous dialog box.



Available units are:

- metres and kilometres
- fathoms (US and international)
- feet (US and international)
- yards (US and international)
- miles (US, international, and nautical)

1. In the *Vertical Length* field, select the units for depth and uncertainty values.

The *Horizontal Length* field is only active if you selected Ground in the *Position Units* field in the previous dialog box.

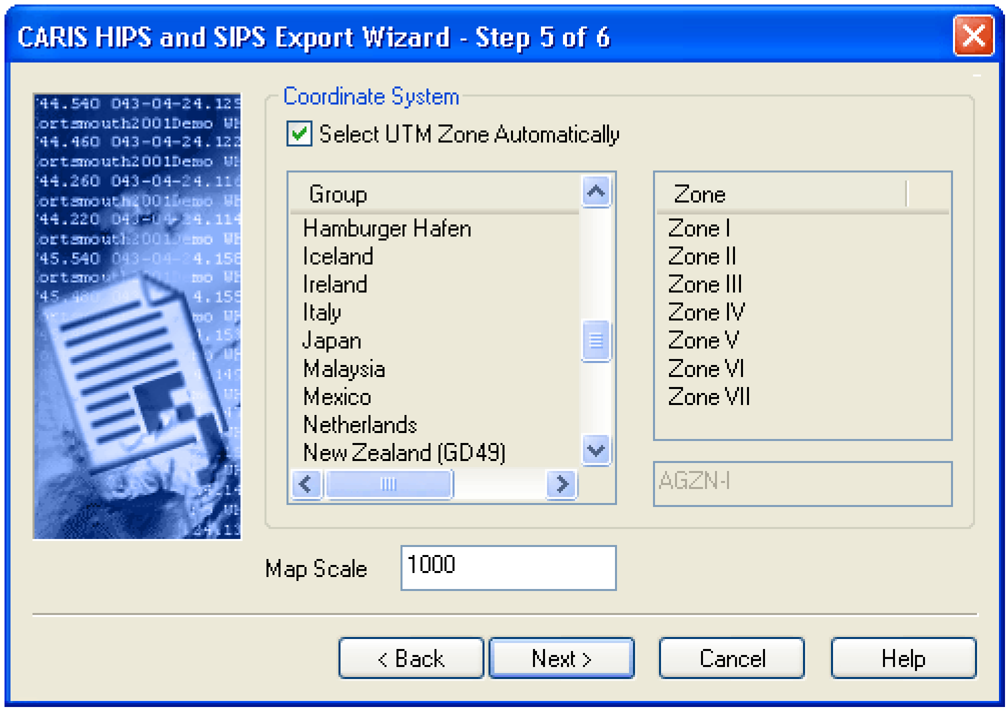
2. Select the units for position data.

To apply a static vertical shift to data during export:

3. Select the *Vertical Shift* check box and type the value for the shift (in metres).
4. The shift will be applied to the depth value (also to the primary and secondary depths for single beam data).
5. Click **Next** to set the coordinate system for the data.

Coordinate System

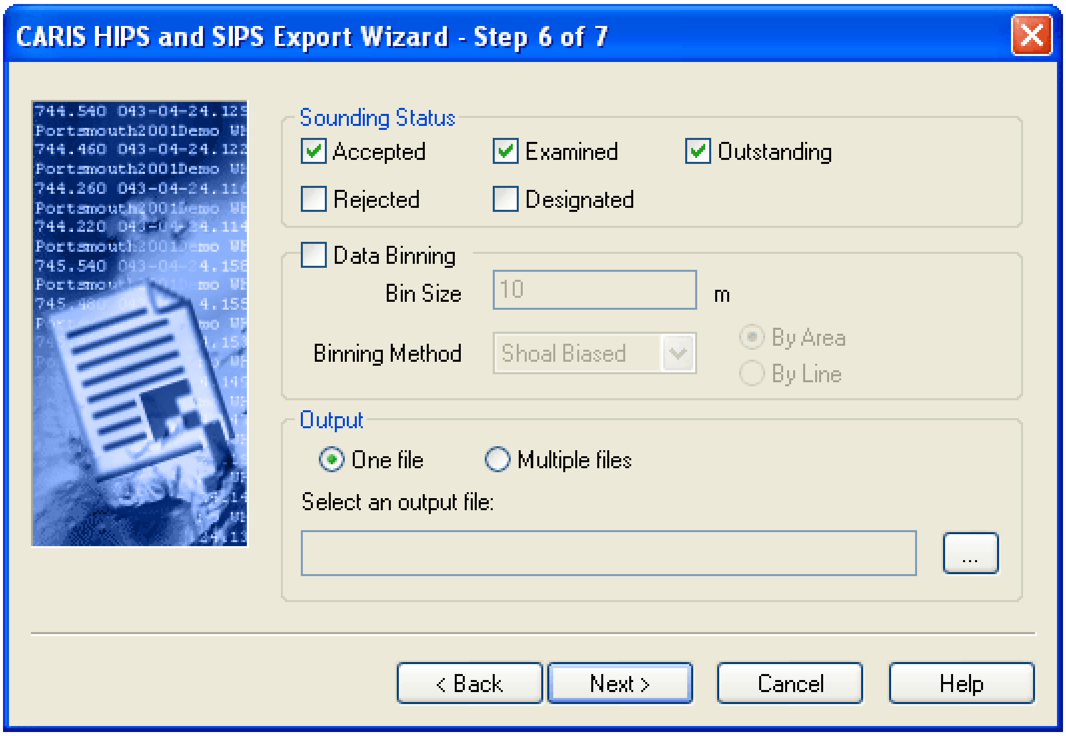
All sounding data is stored in HIPS and SIPS as un-projected longitude and latitude. This dialog selects the appropriate coordinate system to reference the soundings. All soundings will be transformed to this coordinate system as they are exported.



1. Choose a coordinate system by selecting a Projection Group from the list.
2. Select a Zone associated with the Projection Group (the Key field is filled when the Zone is selected).

Click **Next**.

HIPS to ASCII - Step 6



Each sounding in a cleaned HIPS data set are stored with bits that determine the status of that sounding. You can choose to restrict the export process to soundings with specific status flags.

1. To export soundings with a specific status flag, select the appropriate *Sounding Status* check boxes.
2. To apply a data thinning function, select the *Data Binning* check box.

The *Data Binning* option divides the survey area into a grid and selects either the shoal- or deep-biased soundings from each cell for export. The *Bin Size* option sets the size of the grid cells.

3. Type the size for the grid cell in the *Bin Size* field. The units used here can be changed in the Tools > Options > Display >Units dialog box.
4. Select either the *Shoal-Biased* or *Deep-Biased* options to export only the shoalest or deepest soundings, respectively, to the ASCII file.
5. Select the *By Area* option to export one sounding per bin.
6. Select the *By Line* option to process each line, and export a sounding for each line per bin. Where line data overlaps this method can give multiple soundings per bin.
7. Select an output location for the data. Select *One File* to export all data to a single file or select *Multiple Files* to export data to separate files for each survey line.

- If you select *One File*, type a name and file path for the ASCII file, or click **Browse** to select a location and name for the file.
- If you select *Multiple Files*, click **Browse** to select a folder to save the files. The files are automatically named according to Project_Vessel_Date_SurveyLine.

8. Click **Next**.

"Export" on page 542

HIPS To CARIS Map

Export data to a CARIS map.

HIPS to CARIS Map (Step 3)

You can select which soundings are to be exported and how they will be stored in the CARIS map.

The *Export Soundings* option exports all soundings (subject to the sounding status and data binning settings).

1. Select the *Export Soundings* check box if you want to export soundings to a CARIS map. (See the next dialog box for the *Export Tracklines* and *Export Swaths* options).

There are four methods for exporting sounding attributes to a CARIS map.

- *Basic*: The basic sounding attributes in the CARIS map. These include user number, feature code, and source ID. This information is adequate to generate products such as sounding plots, contour plots, and DTMs.
- *Extended*. Basic attributes plus time stamps, launch identifier (first four letters of the vessel name), and tidal correction.
- *Extended with Key*: The extended attributes set, plus the profile number and beam number as the CARIS Key. The profile number and the beam number, in addition to the other attributes, allow each sounding to be identified uniquely with respect to the original data set.

- *Extended with Unique Key*: The extended attributes set, plus every object is given a unique key.

This additional information allows more flexibility for visualizing and manipulating CARIS files. For example, if time stamps are associated with each sounding, you can limit the display of soundings based on a time range.

However, each extra attribute adds to the storage requirements. More disk space is required to store the work file, and any HIPS program that processes the file must do more work and use more time and CPU resources.

2. Select from the four options in the *Attributes* drop-down list.
3. Select a new *Feature Code* for the soundings, if needed.

The *Group by Beam Number* option layers soundings according to a beam number in the CARIS file. Soundings belonging to a beam are assigned to the same user number in the map. User numbers in CARIS file are similar to layering. This option is used when exporting soundings from checklines into a separate checkline map so that a Quality Control Report can be generated. If you do not select this option, then a single user number will be used to hold all the exported soundings.

4. Select the *Group by Beam Number* check box to layer soundings according to beam numbers in the CARIS map.

The *User Number* option is available only if you do not select the *Group by Beam Number* option.

5. [Optional] Type a *User Number* to which all soundings are to be assigned.

The *Sounding Status* option enables you to export only soundings with a specific status flag.

6. To export soundings with a specific status flag, select any of the *Sounding Status* check boxes.

The *Data Binning* option divides the survey area into a grid for sounding selection. The *Bin Size* sets the size of the grid cells.

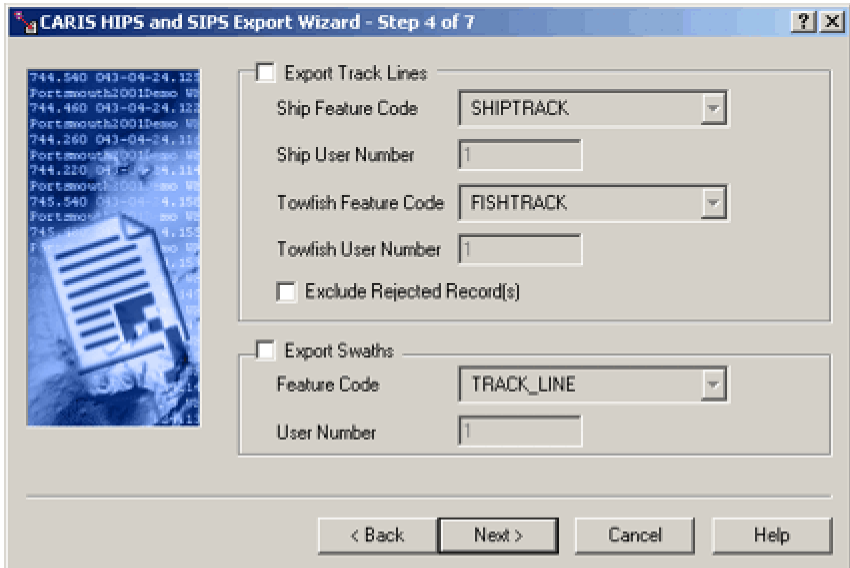
7. To apply a data binning function on export, select the *Data Binning* check box.
8. Type the size for the grid cell in the *Bin Size* field and select either *Metres* or *Feet*.
9. Select either *Shoal-Biased* or *Deep-Biased* to export only the shoalest or deepest soundings from each cell, respectively, to the CARIS file.

Soundings can be selected for export from the total survey area or by processing each track line in sequence.

10. Select either the *By Area* or *By Line* options.
11. Click **Next**.

HIPS to CARIS Map (Step 4)

This dialog box sets the track line and swath export options.



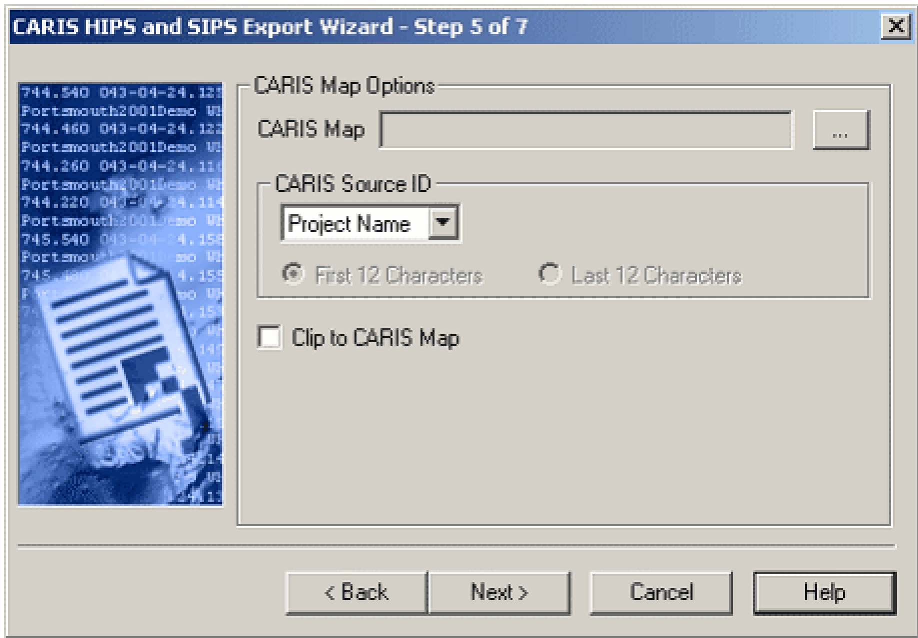
The *Export Track Lines* option connects a line along the centre of each swath, thus showing the track of the vessel along each survey line. It is not necessary to import soundings to use this option.

The *Export Swaths* command connects a set of lines (one per swath) along the outermost port and starboard beams, thus showing the coverage of each ping. It is not necessary to import soundings to use this option.

1. Select the *Export Track Lines* check box to export a line representing the track of the vessel along the survey area.
2. Type a new feature code for the vessel, if needed.
3. Type a *User Number* for the vessel so it is distinguished from other vessels in the same CARIS map.
4. Select a new *Towfish Feature Code*, if needed.
5. Type a *User Number* for the towfish so it is distinguished from other vessels in the same CARIS map.
6. Select the *Exclude Rejected Record(s)* check box to not include this data.
7. Select the *Export Swaths* check box to export a set of lines that represents the swaths along the survey.
8. Type a *User Number* for the swath so it is distinguished from other swaths in the same CARIS map.
9. Click **Next**.

HIPS to CARIS Map - Options

1. Click **Browse** to select a CARIS map for data export, or type the name of a new map.



The map file path and name are displayed in the *CARIS Map* field.

The Source ID is a 12-character alphanumeric attribute in CARIS maps. The ID can be the first or last 12 characters in a project name or line name.

2. Select either *Project Name* or *Line Name* to use as a Source ID.
3. Choose either the first 12 characters or the last 12 characters for the Source ID.

The *Clip to CARIS Map* option cuts the HIPS data at the boundaries of an existing CARIS map. If this option is not selected, the geographic extent of the map are extended to accommodate the HIPS data.

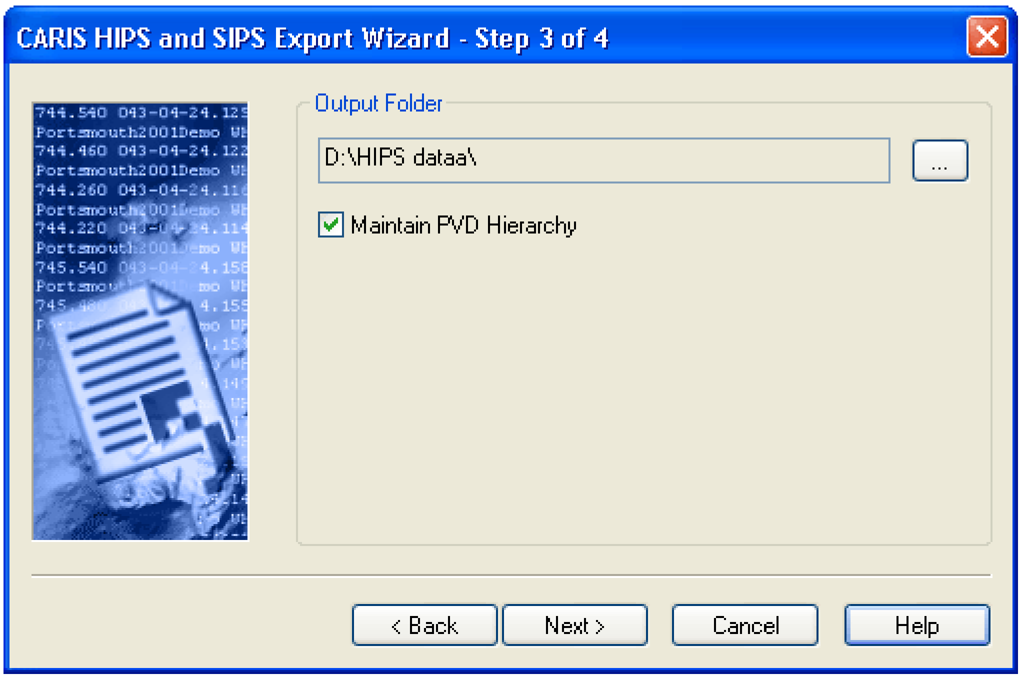
4. Select the *Clip to CARIS Map* check box to implement this option.
5. Click **Next**.

If you created a new CARIS map in Step 1, a dialog box for selecting a projection is displayed. Otherwise you can begin export of the data in the next dialog box.

HIPS to FAU

Export processed HIPS data, with optional heave/pitch/roll/quality/amplitude attributes, to the FAU format. The process will create multiple files with the extension *.fauTC designating the data as "tide corrected". The names of the output files will match the HIPS line names.

After you have selected the track lines to export, the next step is to designate an output folder.



1. Type a path and name for the output file, or click the Browse button (**...**) to select a folder and then type the file name.
2. Select *Maintain PVD Hierarchy* to create output directories that correspond to the HIPS data being exported.
3. Click **Next** to set the coordinate system.

"Coordinate System" on page 541

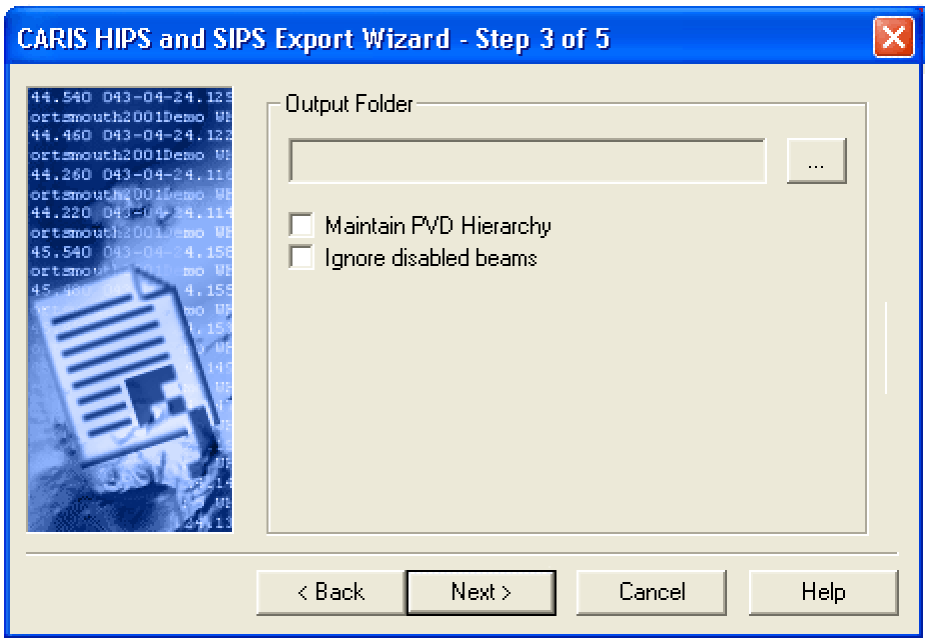
"Export" on page 542

HIPS To GSF

The HIPS to GSF function exports processed HIPS data to a GSF format. A new GSF file is created for each track line that is exported from HIPS.

Conversely, when HIPS data is created from GSF data, a copy of the original GSF file is maintained inside the HIPS directory structure. The Export to GSF function updates these GSF files.

Output Folder



1. Click the **Browse** button (**...**) to select a location for exported GSF files.

The Browse for Folder dialog box is displayed. The stored file will have the same name as the track line.

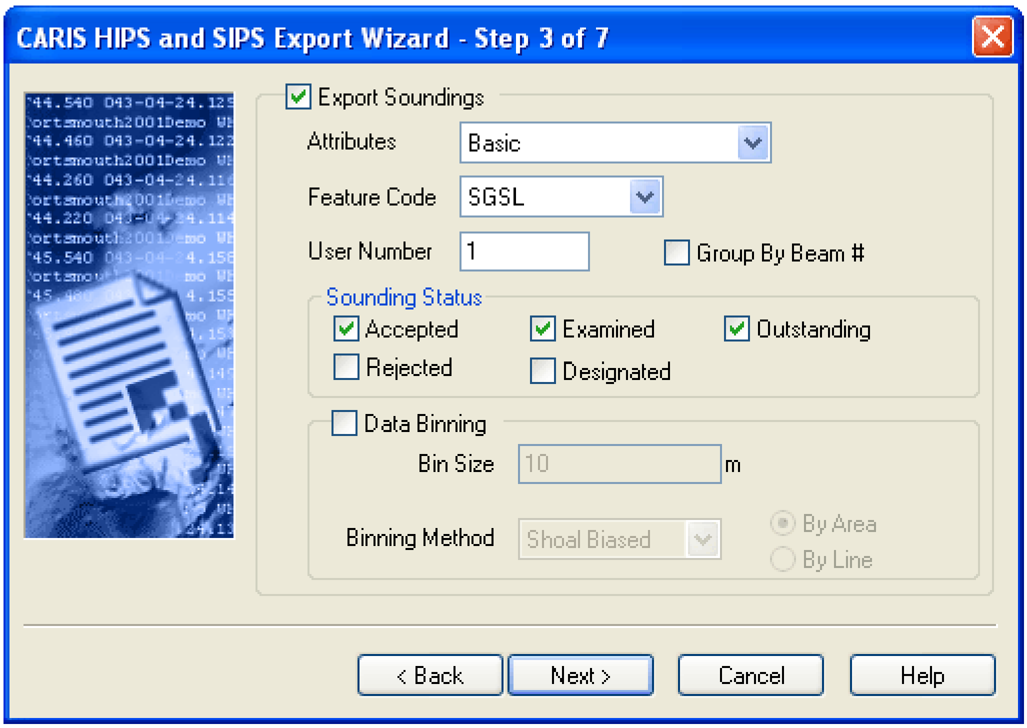
2. To export entire projects and maintain the HIPS Project/Vessel/Day/Line structure in the output, click the *Maintain PVD Hierarchy* check box.
3. Select the *Ignore Disabled Beams* option to exclude any soundings which have been rejected (because of disabled beams) from your export file.
4. Click **Next**.

"EXPORT" ON PAGE 542

HIPS To HOB

The Hydrographic Object Binary (HOB) file stores feature objects and associated attribute data. The HOB file can contain links to CARIS spatial objects.

HIPS to HOB (Step 3)



The *Export Soundings* option exports all soundings (subject to the sounding status and data binning settings).

1. Select the *Export Soundings* check box to export soundings to a CARIS map.
2. ATTRIBUTES
3. FEATURE CODE
4. USER NUMBER
5. To export soundings with a specific status flag, select any of the *Sounding Status* check boxes.

The *Data Binning* option divides the survey area into a grid for sounding selection. The *Bin Size* sets the size of the grid cells.

6. To apply data binning, select the *Data Binning* check box.
7. Type the size for the grid cell in the *Bin Size* field and select either *Metres* or *Feet*.

8. Select either *Shoal-* or *Deep-Biased* to export only the shoalest or deepest soundings, respectively, to the CARIS file.

Soundings can be selected for export from the total survey area or by processing each track line in sequence.

9. Select either the *By Area* or *By Line* options.

The *Export Track Lines* option connects a line along the centre of each swath, thus showing the track of the vessel along each survey line. It is not necessary to export soundings to use this option.

The *Export Swaths* command connects a set of lines (one per swath) along outermost port and starboard beams, thus showing the coverage of each ping. It is not necessary to export soundings to use this option.

10. Select the *Export Track Lines* and *Export Swaths* options.
11. Click **Next**.

HIPS to HOB (Step 4)

1. Click the **Browse** button to select a location for the HOB files.

The Browse for Folder dialog box is displayed.

2. Select a directory to store the HOB data and click **OK** to close the dialog box.

The file path is displayed in the *Output File* field.

3. Click **Next**.

"Export" on page 542

HIPS To HTF

The HIPS and SIPS Export Wizard converts field sheet layer data and soundings to a Hydrographic Transfer Format (HTF) file.

Hydrographic Transfer Format

The HTF file is an ASCII file that can be opened in any text editor (such as Notepad). It consists of two sections: a header section and a sounding data record.

- The header contains general information about the survey.
- The sounding data record contains the depths and other relevant information.

The total number of fields generated in the sounding data record depends on the survey. At the start of the sounding data record is a Field Population Key that gives each field a 1 bit to show that a field is included, or a 0 bit to show if a field is not included.

For example: `[00001111100000000000]`

In the above Field Population Key, fields 5 to 9 contain data (hence the 1 bits), while fields 1 to 4 and 10 to 20 contain no data (hence the 0 bits). Fields without data are not written to the HTF file when it is generated. There are a total of 20 HTF fields in the sounding data record. Two additional fields are added by the Export Wizard for beam and profile number. Extra fields can be added, if needed. The HTF export automatically exports TPU depth and TPU position values if these are included in the track line.

The HTF file contains an ordered list of all possible fields and a description of each field.

Hydrographic Transfer Format Template

Sometimes the same header information can be used for multiple HTF files. The export wizard gives you the option of saving a header for use in another file. Headers are saved as HTFT files (Hydrographic Transfer Format Template). The file is in XML format that can be viewed in a text editor or an XML-compliant Internet browser.

Select Input Data

When exporting HIPS to HTF the Input Options dialog box displays either a list of projects and associated field sheets or the Project/Vessel/Day/Line data structure depending on which export option you select.

1. Select the *Field Sheet Product* option and select a product layer associated with a field sheet,

   or

2. Select the *BASE surface* option and select one or more BASE surfaces,

   or

3. Select the *Raw Data* option to export sounding data.

This will refresh the file tree to a Project/Vessel/Day/Line directory structure.

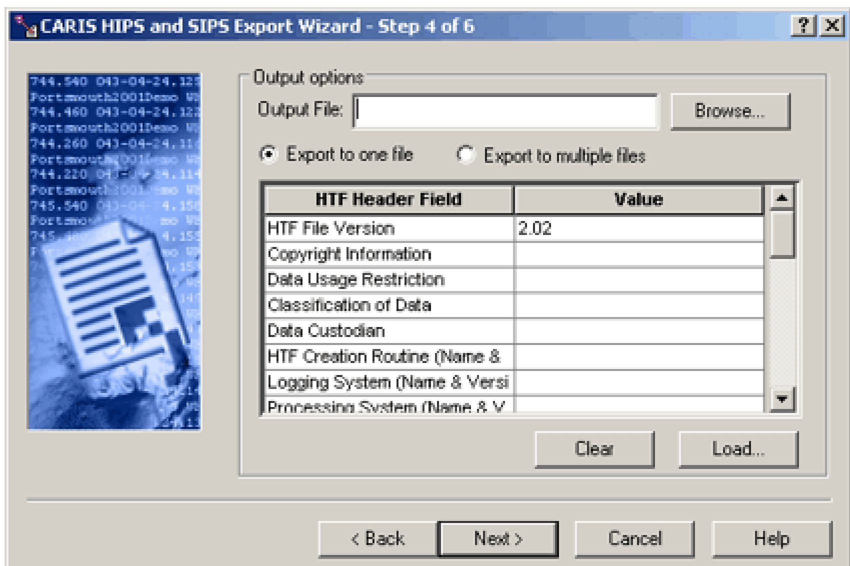
4. Open a project or session file tree and select a layer or trackline data.

If you select the *Raw Data* option, the *Include Rejected* option is active. This option includes rejected soundings in the export process.

5. [Optional] Select the *Include Rejected* option to export rejected data.

Click **Next**.

HIPS to HTF - Output options



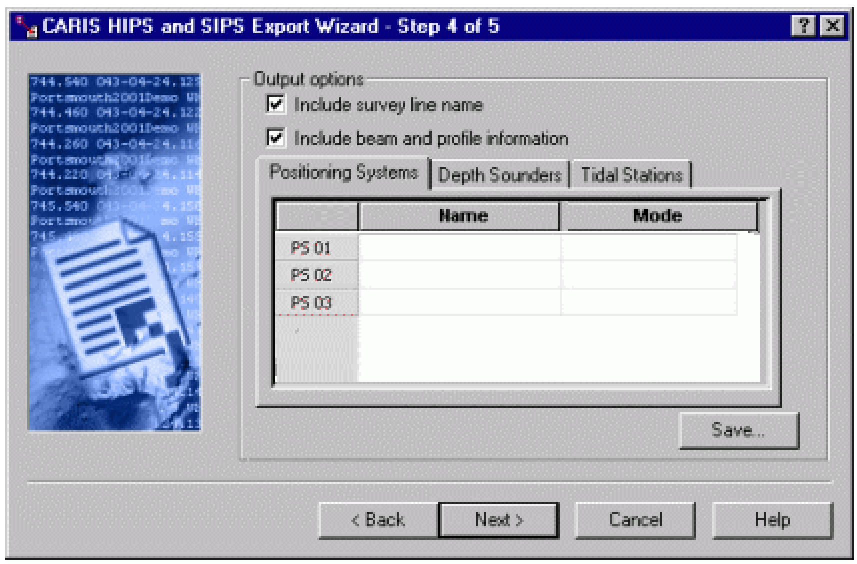
Enter the necessary header information. You can directly enter the data or load it from an HTFT (Hydrographic Transfer Format Template) file.

If you are exporting soundings, you can export all data to a single file or export data from each survey line to its own file. This option is not available for field sheet products.

1. Select an output location for the HTF file(s).
   - If you choose *Export to One File*, type a name and file path for the HTF file, or click **Browse** to select a location and name for the file.
   - If you choose *Export to Multiple Files*, click **Browse** to select a folder to save the HTF files. The files are automatically named according to Project_Vessel_Date_SurveyLine.

2. To enter new information, for example, the number of depth sounders or tidal stations, click in the *Value* field next to the appropriate *HTF Header Field* and type the data, or
3. [Optional] Click **Load** to open data from an existing HTFT file into the *Value* fields.
4. Click **Clear** to remove data from all fields, if needed.
5. Click **Next**.

HIPS to HTF - Output options page 2



1. Select the *Include Survey Line Name* check box to include this information in the HTF file.
2. Select the *Include Beam and Profile Information* check box to include the beam and profile numbers in the HTF file.

If you entered values in the depth sounder, positioning system, and tidal station fields in the previous dialog box then the respective fields in these tabs are enabled. If you did not enter values, click **Back** to fill in these fields.

The Positioning Systems tab is for entering navigation systems information.

3. Type the name and model (and/or any additional information) of the system in the *Name* field.
4. Type the system used on the survey (GPS, DGPS, etc.).

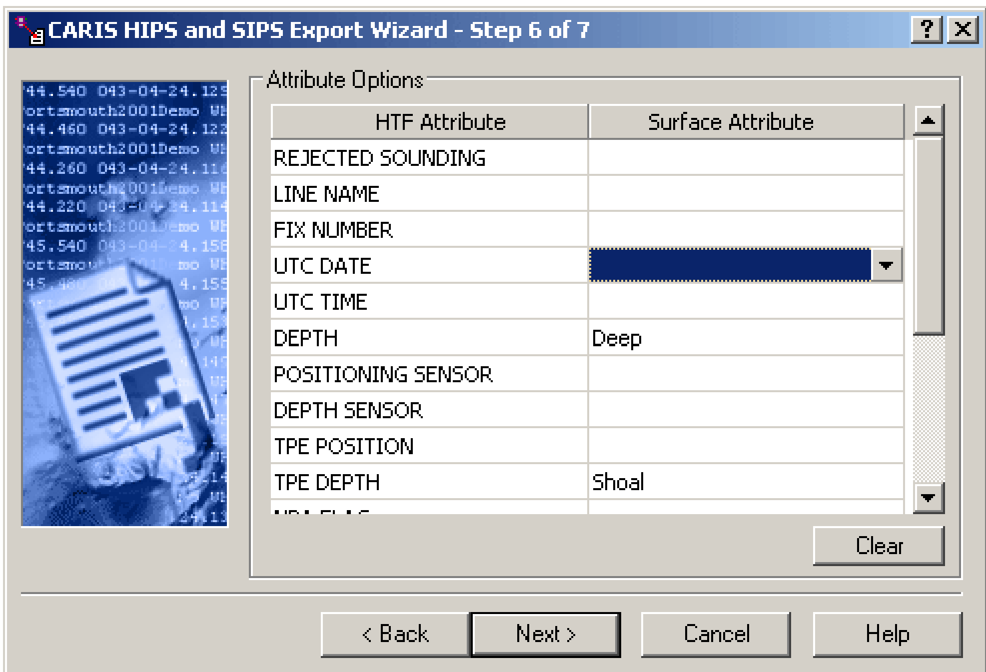
The Depth Sounders tab is for entering sonar information.

5. Type the name and model of the sonar system used on the survey.
6. Type the mode of sonar (multibeam, single beam, multi transducer).

The Tidal Stations tab is for entering tide information.

7. Type the name of a tidal station.
8. Type the location.
9. Type the method used to acquire data (predicted, observed, etc.)
10. Click **Save** to save information to a new or existing HTFT file.
11. Click **Next**.

HIPS to HTF - Attributes Options



If you are exporting a BASE surface you must map at least one BASE surface attributes to a HTF attribute.

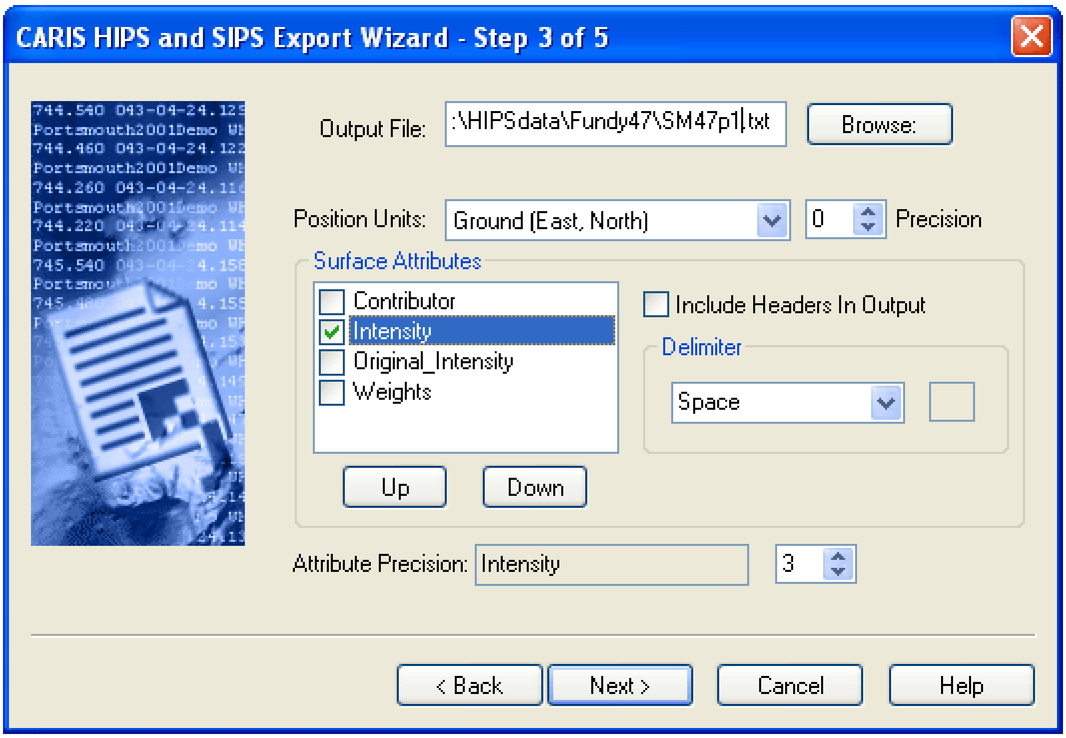
1. Click in a surface attribute field to access a drop-down box listing all the attributes available in the data file.
2. Select an attribute from the list.
3. Click **Clear** to remove all values from the Surface Attributes fields.
4. Click **Next**.

"Export" on page 542

Mosaic to ASCII

Export Mosaic attributes to an ASCII text file.

Mosaic to ASCII Output Options



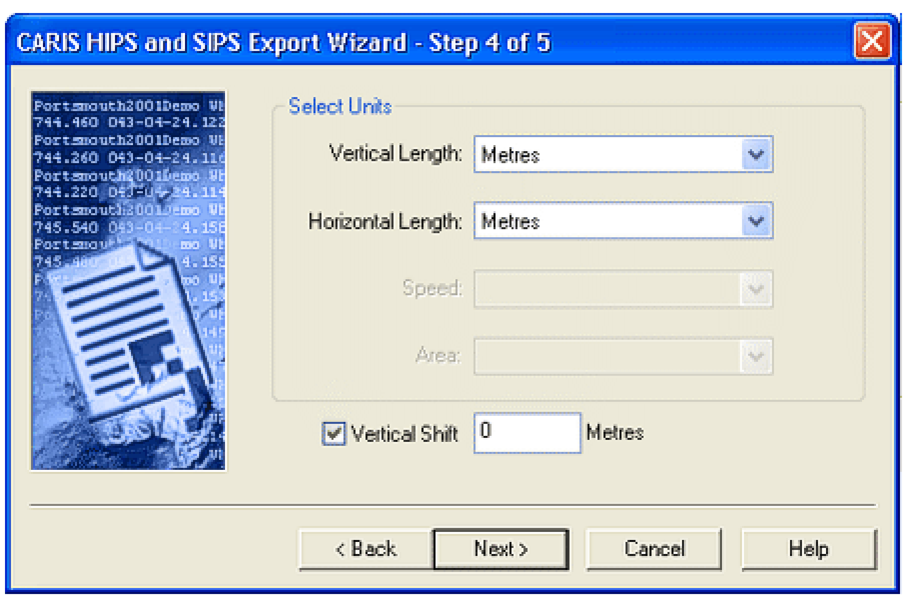
1. Type a path and name for the output file, or click **Browse** to select a folder and then type the file name.
2. Select the *Position Units* for data from the drop-down list:
   - Ground (East, North): position displayed as Easting, Northing to the precision defined (for example:`360425.000, 4770355.000`)
   - Geographic (DMS): position displayed as Latitude, Longitude in degrees, minutes, seconds to the precision defined (for example: `43-04-23.03N, 070-42-51.89W`
   - Geographic (Unformatted DMS): displays the position without using hyphens or direction indicators, for example, `43 04 23.03, -070 42 51.89.`
   - Geographic (Decimal Degrees): position displayed as Latitude, Longitude, in decimal degrees to the precision set, for example, 43.073 -070.714.
   - Geographic (Degree Minutes): position displayed as Latitude, Longitude, in degrees and minutes, to the precision set, for example, 34-23.610N,119-52.585W.
   - Note that for positions in the northern hemisphere, longitude West values are negative; in the southern

hemisphere, the South latitude values are negative. For example, a position near Portsmouth NH, USA is `43 04 23.03, -070 42 51.89`. A position near Sydney Australia is `-33 51 00.84 151 11 43.7`

3. Specify the precision for the display of position values, by using the *Precision* up or down arrow buttons to set the number of digits to be displayed to the right of the decimal point.
4. Select the BASE Surface attribute values to be recorded in the ASCII file by selecting the attribute in the *Available* column and clicking **Add** to move it to the *Active* column.
5. To remove an attribute, select the attribute from the *Active* column and click **Remove** to move it back to the *Available* column.
6. Choose a *Attribute Precision* value (number of digits to the right of decimal point) for an attribute in the *Active* column by selecting an attribute and clicking the up or down arrow buttons.
7. Select the *Headers* check box to print attribute names at the top of the attribute rows in the output file.
8. Select either Space, Comma, or Tab from the list to use these characters to separate fields in the output file. If you select Other, type a character to use a delimiter.
9. Click **Next**.

Select Export Units

This dialog box sets the unit type for recording data. The fields that are active in this dialog are determined by the attributes selected in the previous dialog box.



Available units are:

- metres and kilometres

- fathoms (US and international)
- feet (US and international)
- yards (US and international)
- miles (US, international, and nautical)

1. In the *Vertical Length* field, select the units for depth and uncertainty values.

The *Horizontal Length* field is only active if you selected Ground in the *Position Units* field in the previous dialog box.

2. Select the units for position data.
3. Click **Next**.

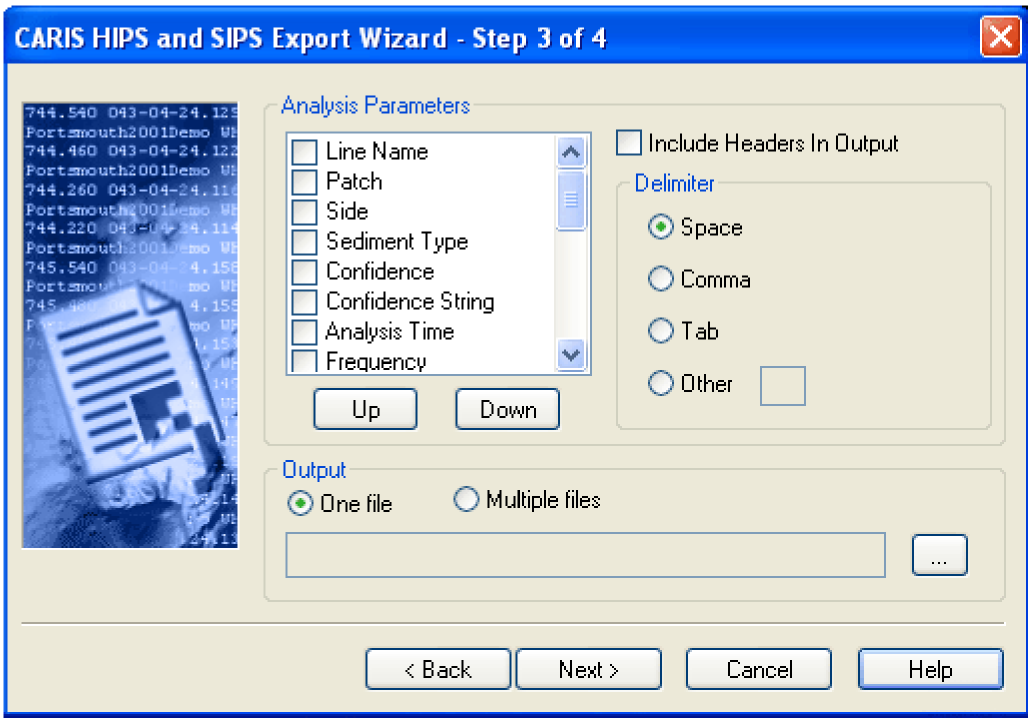
“Export” on page 542

Mosaic To Image

Please see "Surface To Image" on page 501 for the description of this export process.

Sediment Analysis to ASCII

Export the results of Sediment Analysis done in Mosaic Editor. You can select which parameters to export, and the order they will be displayed, as well the type of delimiter to be used in the ASCII file. Results can be exported for one line or for multiple lines.



1. To include a parameter in the ASCII output, select its check box in the *Analysis Parameters* list. You can select more than one parameter at a time.
2. Use the **Move Up** and **Move Down** buttons to change the order in which the data will be shown in the output file.
3. If you want to include the parameter names as column headers in your output file, select the *Include Headers in Output* check box.
4. Select the type of delimiter you want to use to space the data in your output file. If you select *Other*, type the character you want to use.

If you are exporting the sediment analysis data from more than one line you can export the data to a single file or to a file for each line.

5. Select *One file* or *Multiple files*.
6. Click **Browse [...]** and select the destination folder for the output file(s).
7. Type the name of the file if output is going to a single file.
8. Click **Next**.

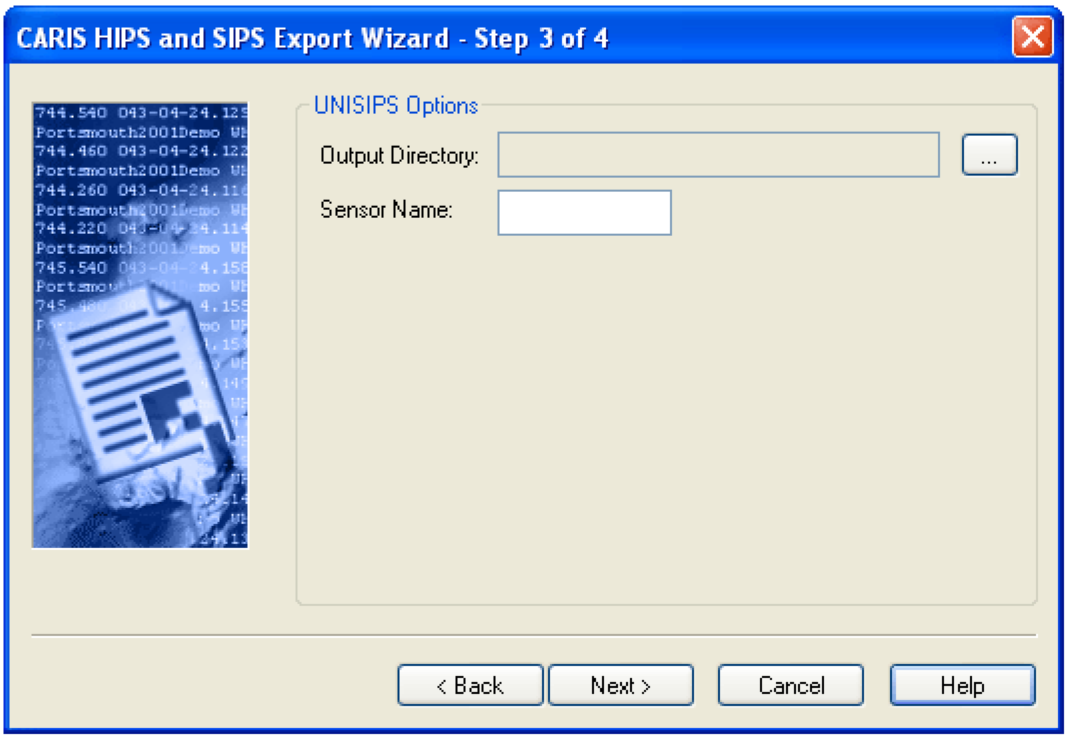
"Export" on page 542

SIPS To UNISIPS

Export SIPS data to a Unified Sonar Imaging Processing System (UNISIPS) format.

SIPS TO UNISIPS (Step 3)

Select the location to store the output data.



1. Click **Browse** to select a location for the UNISIPS files.

The Browse for Folder dialog box is displayed.

2. Select a directory to store the UNISIPS data and click **OK** to close the dialog box. (The UNISIPS file will have the same name as the track line with an .u file extension.)
3. Type a *Sensor Name* in the field (up to 10 characters).

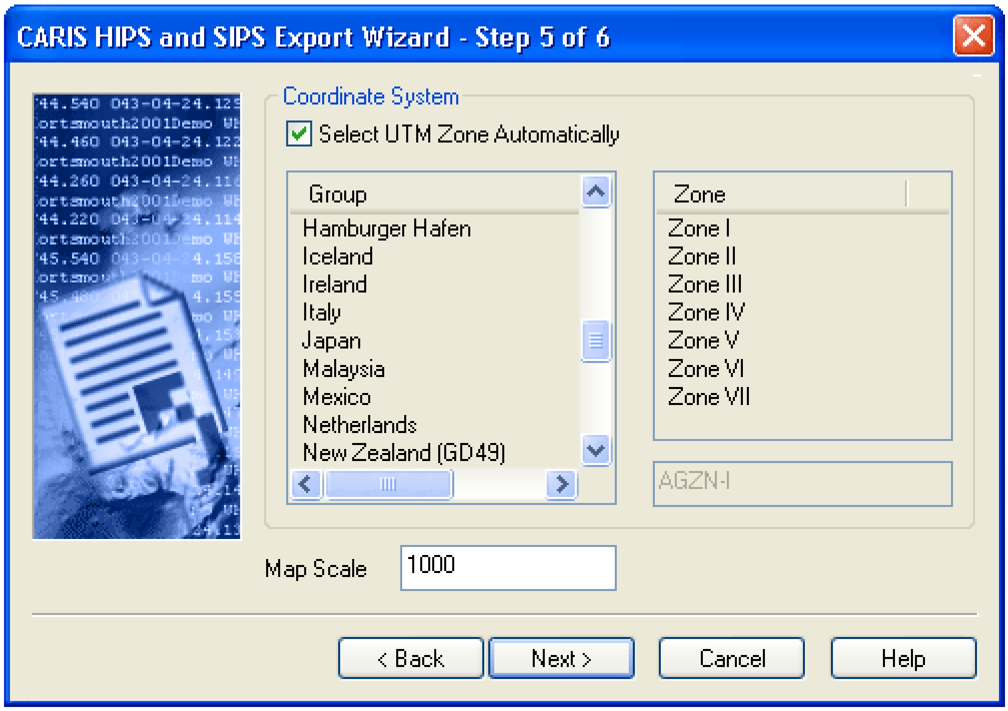
The file path is displayed in the *Output File* field.

4. Click **Next.**

"Export" on page 542

Coordinate System

All sounding data is stored in HIPS and SIPS as un-projected longitude and latitude. This dialog selects the appropriate coordinate system to reference the soundings. All soundings will be transformed to this coordinate system as they are exported.

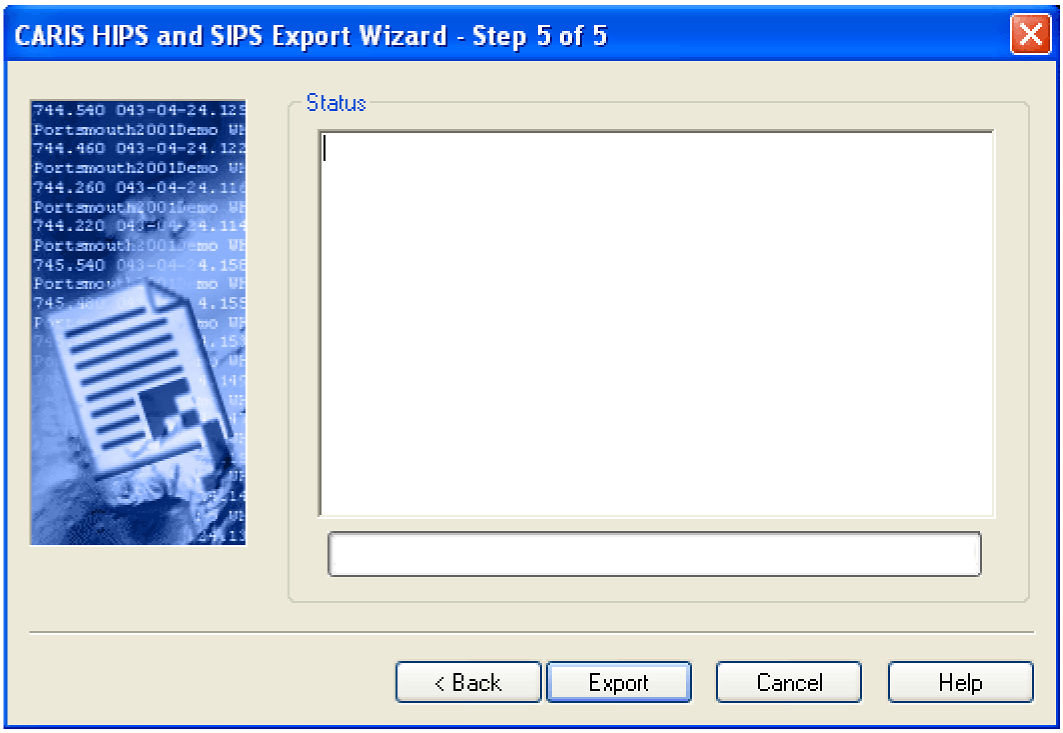


By default, the *Select UTM Zone Automatically* option is selected. To set the projection manually:

1. Select a projection from the *Group* list.
2. Select a Zone associated with the projection group.
3. Click **Next**.

Export

At this step you execute the export of the data with the settings you have chosen. If a large amount of data is being exported, the dialog box will display the percentage of data exported and the status of the export process.



1. Click **Back** to review your settings.
2. Click **Export** to export the data with the settings you've selected.
3. Click **Cancel** to call off the process.

A progress indicator shows the percentage of files converted to the selected format.

When the export is completed, the **Restart** button is displayed to enable you to go back and export another file.

Index

Symbols

*.tid 146
*.zdf 147

A

across-track display
...of water column data 462
acrosstrack/altitude ratio 424
Active Field Sheet 180
active field sheet 179
active sensors in Vessel Editor 38
add a GeoBaR to mosaic 441
add layer to BASE surface 219
Add Session, see Open session 21
Add to BASE surface 218
along-track display
...of water column data 462
altitude 424
Analyze selected patch, in sediment analysis 448
annotations
...in Projection gird 184
Annotations tab 184
Apply Roll, in Refraction Editor 335
Apply TPU filter 161
Apply True Heave 138
archive project 74
ASCII
...export BASE surface to 498
...export data to 517
...export GeoBaR to 507
...export HIPS tide 516
...export mosaic to 535
...export profile to 377
...export sediment analysis 539
...export surface statistics to 237
Atlas format, conversion options for 97
attitude data
...load Applanix files 291
...reset rejected data 305
...view rejected soundings 295
Attitude filter 286
attribute layer
...sun position 210
...transparency 210
attributes
...in Subset tiles 345
Auto Analyze sediment 447
auto cursor mode 328
Auto Populate SORIND and RECDAT 391
AutoCAD, export to 487
automatic depth filtering 327
automatic filtering in Swath Editor 325
Automatically mark as complete
...option for tracking in subsets 348
Auto-Seam
...use to resolve overlapping GeoBaRs 436
Available Vessels 59
azimuth, of sun postion 210

B

BAG
...templates 484
BAG (Bathymetric Attributed Grid), export data to 482
BASE Surface
...export to ASCII 498
...export to image 501
BASE surface
...add legend 208
...add survey data 218
...attribute display properties 210
...close 218
...combine surfaces 226
...compute statistics 232
...create 193
...create holiday layer 238
...custom layer 219
...delete 218
...difference surface 229
...filter display 209
...finalized surface 224
...group surfaces 242
...holiday layer 238
...Interpolate 222
...open 218
...open in Subset Editor 350
...properties 204
...quality control 307
...quality report 238
...range weight 190
...recompute 221
...remove survey lines 219
...swath angle weight 190
...uncertainy weight 189
...view 203
Beam Pattern Correction
...in Mosaic Editor 426
Benthos C3D data 128
binning
...new bin layer 401
...overview 397
bounding box, to create field sheet 177

C

CARIS map
...export HIPS data to 523
...output soundings to 388
CARIS Spatial ARchive (CSAR) 188
Carry Over Raw Data Files 84
carry over raw data files 84
CATZOC 159
check Load Tide status 145
classify lines 293
close
...field sheet 180
CMAX format, conversion options for 98
Coda format, conversion options for 98
Colour by SV Profile 140
Colour Table Editor 394
compare BASE surfaces 229
components of mosaic
...edit 440
Compute GPS Tide 149
...options 150
Compute layer in BASE surface 219
Compute Statistics
...Mosaic attribute 441
Compute Surface Statistics 232
Compute TPU 154
configuration options
...set in Vessel Wizard 32
conflict resolution
...sounding selection 385
conflict resolution methods, selecting GeoBaR 445
Conflict Resolution, Mosaic Editor 444
Connect To 59, 86
connect to 64
connect to data repository 59
contacts
...export 504
contour to field sheet 364
contour to HOB file 365
contours
...depth list file 362
...labels 365
...masking lines behind labels 366
...overview 361
Control Coverage by Surface
...in new subset tiles 344
conversion
...Atlas 97
...carry over raw data 84
...CMAX 98
...Coda 98
...EdgeTech 99
...EIVA 101
...Elac 102
...Furuno 103
...GSF 107
...Hawkeye 108
...Hypack 109
...Imagenex 112
...LADS 113
...LAS 114
...layback 131
...MarineSonics 114
...Navitronics 114
...Qmips 115
...Reson PDS 116
...SDF 120
...Seabeam 121
...Seafalcon 121
...SEGY 121
...SHOALS 122
...Simrad 122
...Spawar 124
...SWATHPlus 124
...tow cable 131
...UNB (Reson) 124
...Winfrog 124
...XTF 126
Conversion Wizard 78– 94
conversion, ship navigation source in XTF 126
convert hidden or non-logged Reson data 118
convert layback 131
Coordinate definitions
...for HVF 53
coordinate system
...in vessel file 53
...set during export 541
create a product surface 358
create contours 361
create GeoBaR
...advanced options with Geocoder engine 420
...with Geocoder Engine 419
...with the SIPS Engine 422
create layer of soundings 383
create mosaic 434
...intensity options 437
...overlapping GeoBaRs 436
...resolution 436
...set colours 437
create QTC tiles 402
create vessel file,workflow 27
Critical soundings
...protective radius 304
critical soundings
...outdated layer 303
...query 303
Critical Soundings Detection
...process 298
critical soundings detection 298
CSAR 188
CSST 385
CUBE
...create 251
...hypothesis cleaning 266
...hypothesis editing 259
...nominate hypotheses 263
...properties in Subset Editor 259

D

data conversion 78
Data QC
...interpolation methods 282
data repository 59
decimate imagery, converting Coda to HIPS 98
default ellipsoid 55
default options in GeoBaR creation
...Geocoder engine 419
defocusing 356
delete 437
...GeoBaR 433
delete mosaic 437
delta draft
...loading 172
density
...creating surface cleaning tiles 315
depth list file 362
depth range files 362, 409
...tiling by 400
designate critical soundings 298
designate shoalest soundings 342
Designate soundings 342
Designated soundings 297, 297
Despeckle in Mosaic Editor 428
Detailed Line Query 274
...SBET loaded status 165
Difference Surfaces 229
digitize a profile 367
digitize profile line 368
disambiguation 248
Display
...SVP as backgorund 140
display colours
...set when creating mosaic 437
display options
...sunset tiles 346
Display properties
...attribute layer 210
display tide zones 147
double buffering 356
DpTPU 156
DXF, export to 487
dynamic draft data in vessel file 46

E

EdgeTech format, conversion options for 99
Edit
...GeoBaR 429
...mosaic components 440
Edit mosaic 438
Edit profile line 375
edit sensor configuration 38
editing data 272
EIVA 101
Elac 102
ellipsoid in vessel file 55
elves 476
engineer soundings 388
examining and editing data 272
Expor5t
...to S-57 480
Export
...BAG metadata 497
...BAG to ASCII 497
...BASE surface to ASCII 498
...formats exported 492
...GeoBaR to ASCII 507
...GeoBaR to Image, see Surface to Image 510
...HIPS data to ASCII 517
...HIPS tide to ASCII 516
...HIPS to CARIS map 523
...mosaic to ASCII 535
...multiple GeoBaRs 502
...sedimewnt analysis to ASCII 539
...select data 495
...select format 494
...soundings to HOB 529
...surface to image 501
...to DXF 487
...urface to BAG 482
export
...contacts 504
...multiple GeoBaRs 495
Export a surface
...select 495
Export contacts or line data
...select 496
Export GSF to CARIS map 511
Export HIPS
...to FAU 527
...to GSF 528
...to HOB 529
...to HTF 531
Export Histogram to Image 234
export profile 377
export SIPS to UNISIPS 540
Export surface metadata 481
Export to DXF
...contours, soundings or bounding polygon 487
...lines or Critical Soundings 487
Export Wizard
...open 494

F

field sheet
...active 180
...contour to 364
...create 174
...open 179
...projection grid 182
...properties, viewing 181
...set as active 179
...turn off 180

field sheets
...overview 173
file formats exported 492
filter
...BASE surface attribute display 209
Filter Slice 353
filtering
...Benthos 128
...by Special Order 159
...QINSy 128
...run TPU filter 161
filtering S-44 159
filters
...attitude data 285– 288
...protect crtitical soundings 304
...swath 325
...Total Propagated Uncertainty 159
finalized surface 224
Find
...search for sounding 328
Find and Designate 342
find soundings 328
fixed size points
...WCI data 471
flag
...designated soundings 333
flag shoalest soundings 342
formatting tags for GeoBaR names 418
Full Blend
...use to resolve overlapping GeoBaRs 436

G

Generalized layer 359
Generic Data Parser 24
GeoBaR 412
...acronym explained 412
...conflict resolution methods when selecting 445
...create 417
...define 412
...delete 433
...edit 429
...export many 502
...naming 418
...remove from mosaic 441
...using variables to name 418
GeoBaR conflict 444
GeoBar layer
...statistics 441
GeoBaR to Image, see Surface to Image 510
GeoBaRs, export multiple 495
Geocoder
...workflow for multibeam data 415
Geocoder engine 414
...default options in GeoBaR creation 419
grain size 446
Grain Size table 451
graph, sediment analysis 453
grid
...projection grid in field sheet 182
grid lines
...in projection grid 182
Group surfaces 242
GSF
...export to CARIS map 511
...export to HOB 514
GSF (Generic Sensor Format) 511, 514
...conversion options for 107
...export HIPS data to 528
gyro sensor data in vessel file 43

H

Hawkeye converter options 108
heave sensor data in vessel file 43
height source
...used in profile 368
HIPS tide
...export to ASCII 516
HOB (Hydrographic Object Binary) files
...export GSF to 514
...export to 529
HOB file, contour to 365
holiday (data gaps)
...identify in BASE surface 238
holiday layer
...create in surface 238
holiday search radius 238, 240
HTF (Hydrographic Transfer Format), export layer data to 531
HVF
...create new vessel file 27
...edit sensor configuration 38
...location 26
...sections 26
HVF (HIPS Vessel File) 26
HVF location 28
Hypack format, conversion options for 109
hypothesis editing 259
HzTPU 156

I

image classification 431
Image Export
...of BASEsurface statistics 234
...of profile 380
image, export surface to 501
Imagenex, conversion options for 112
imagery processing workflow 413
Import Profile 373
Index Contours 363
initialization, in CUBE surface creation 248
Intensity 417
intensity options
...creating mosaic 437

Interpolate mosaic 442
Interpolate surface, of mosaic 442
interpolation
...in Refraction Editor 336
...set for navigation data 283
interpolation methods, Navigation Editor 282

L

label contours 365
LADS format, conversion options for 113
LAS data format, conversion options for 114
layback
...convert 131
lidar
...LADS conversion options 113
...Shoals conversion options 122, 122
Line Report 275
Load
...Delta Draft 172
Load Attitude/Navigation Data 291
Load Delayed Heave 289
Load Tide 144
...tide applied 145
load tide
...from multiple tide stations 144
load ZDF 144
Lock aspect ratio
...in profile graph 372

M

manual blending of GeoBaRs in edit 441
Manual cleaning
...in Swath Editor 328
map BASE surface attributes to S-57 391
MarineSonics data format, conversion options for 114
masking contour lines 366
Merge
...process description 168
metadata
...surface export 481
Mosaic
...export to image 538
mosaic 437
...add GeoBaR to 441
...create 434
...edit 438
...image classification 431
...interpolate surface 442
...open early version 438
...remove GeoBaR from 441
...select for edit 438
...unique filename 435
mosaic components
...edit 440
Mosaic Editor
...GrainSize table 451
...open 416
...sediment analysis 446
...Sediment Analysis Graph 453
...workflow diagram 413
...workflow overview 412
Mosaic Editor, conflict resolution 444
mosaic statistics 441
mtions sensors, in HVF 32
Multibeam Filtering, in XTF conversion 128
multibeam workflow 17
multiple data paths
...connect to 64
multi-transducer
...settings in HVF 30

N

name
...mosaic 435
naming GeoBaRs 418
Navigation
...interpolation methods 282
navigation
...antenna data in vessel file 42
...bezier interpolation 281
...linear interpolation 281
...load Applanix files 291
...reject-break interpolation 281
...reject-with interpolation 281
...reset rejected data 305
...set interpolation 283
...speed jumps 279
...time jumps 279
...view rejected soundings 295
navigation time overlap 84
Navitronics data format, conversion options for 114
new data respository 64
new project, create 58
nominate hypotheses 263

O

offsets for sensors
...set reference point 35
Open
...Attitude Editor 285
...BASE surface 218
...BASE surface in Subset Editor 350
...Export Wizard 494
...field sheet 179
...Mosaic Editor 416
...Plot Composer 476
...project 64
...Refraction Editor 334
...session 21
...Vessel Editor 27
open legacy mosaic 438
Open project with shortcut 66

Options
...sediment analysis 447
original intensity 417
Outdated line 171, 294
output soundings to CARIS map 388
overlap time stamp 84
overlapping GeoBaRs in mosiac 436
Overplot Removal, sounding selection 385, 387
Overwrite
...use to resolve overlapping GeoBaRs 436

P

Patch Size, set in sediemtn analysis 447
patch test, for beam pattern correction 426
PDS2000 data 118
PFree Heave 289
pitch data in vessel file 44
playback WCI 464
Plot Composer 476, 476
primary sounding. start point for tiling 401
product surface 356
product surface, create 358
profile
...edit line 375
...export 377
...import 373
...line colour 370
...options 370
...remove 377
...resize window 373
Profile by Digitizing 367
Profile by Superselection 369
profile line
...digitize 368
profile settings 367
profiles
...overview 367
Project
...create new 58
project
...archive 74
...open 64
project, Connect To 59, 86
projection grid, field sheet 182
projects
...convert raw data 78
...rename folders 73
...session files 21
Properties
...project 67
properties
...BASE Surface 204
...field sheets 181
...tiles 404
...tiles display options 346
...track lines 71
protect critical soundings 304
protective radius
...use in surface cleaning 316

Q

QC report 307
...template file 307
Qmips format, conversion options for 115
QPS 128
QTC tiles 397
quality report for BASE surface 238
Query
...critical soundings 303
Query command 285
query data 273
Query data in editors 274
Query Line 275
query SVC 140
query tiles, see View tile attributes 406

R

Radius, sounding selection 385, 385
range weight 190
raster legend properties 208
RECDAT 391
Recompute BASE surface 221
Reference Point
...set in vessel file 35
Reference Point for sensor equipment 53
Reference Surface properties, Subset Editor 350
refraction artifacts 334
Refraction Editor 334
regenerate critical soundings layer 303
reject hypothesis 266
Reject Line 294
rejected soundings, view 295
rejecting hypotheses 266
remove a GeoBaR from a mosaic 441
remove profile 377
remove rejected soundings from backscatter 332
rename folders 73
residual in surface cleaning 321
resolution
...set in new mosaic 436
Reson data
...convert hidden or non-logged 118
Reson PDS format, conversion options for 116
restart cleaning 305
reverse colours 395
roll data in vessel file 45

S

S-44
...quality control 307
...TPU filtering 159

S-44 filtering 159
S-57
...export to 480
S7K data
...convert 118
save a session 21
SBET loaded? 165
SDF format, conversion options for 120
Seabeam format, conversion options for 103, 121
Seafalcon format, conversion options for 121
search by beam or profile 328
search by depth profile or beam number 328
Search directories for files with temporal overlap 85
Search for files with temporal overlap 84
search for sounding 328
sediment analysis 446
...auto analyze 447
...export to ASCII 539
...graph 453
...set patch size 448
Sediment analysis options 447
sediment analysis overlay
...view in Mosaic Editor 455
select contacts or line data to export 496
Select files to export 495
Select Format to export 494
select surface to export 495
selected soundings 383
...display 388
...output 388
Selection tab
...view tile attributes 406
sensor configuration 38
sensor data in vessel file 38
session files 21
Set Interpolation 283
set line classification 293
set vessel measurements 35
settings
...bottom detection colour 471
settings to create profile 367
Shine through
...use to resolve overlapping GeoBaRs 436
Ship Navigation data source in conversion 126
shoal-biased product surface 356
Shoalest Depth True Position
...about 191
shoalest soundings, flag 342
SHOALS format, conversion options for 122
shortcut to project 66
Show Alternative Hypotheses 263
Show Rejected 330, 330, 339
Side Scan Editor
...open in Mosaic Editor 416
Simrad format, conversion options for 122
single beam data
...restart cleaning 305
...view rejected soundings 295
SIPS engine 414
slant range filter
...in water column data 469
slanted soundings 388
smooth
...profile line 371
Smooth data, with attitude filter 286
SORIND 391
sound velocity correction
...nearest in distance option 134
...nearest in time option 134
...previous in time option 134
Sound Velocity Corrections
...parameters in HVF 33
sound velocity profile options 134
sounding radius table 386
sounding selection
...conflict resolution 385
...criteria 384
...filter 390
...new layer 383
Sounding Selection Template 385
soundings
...Designated 297
...export to HOB 529
...set radius for critical 304
...set status in Subset Editor 339
Special order 159
Stacked view of WCI 462
Starting Level Tile Set 312, 315
Status Flag
...LADS conversion to HIPS 113
...Reject Line 294
Status Flags
...Navigation data 281
...used in BASE surface 197, 199
subset
...defined 338
Subset Editor
...accept data 340
...designate shoal soundings 342
...examined status flag 340
...outstanding status flag 340
...query 340
...reference surface 350
...reject data 339
...set sounding status 339
...subset tiles 343
...subset tiles flagged 349
...track cleaning status 349
...tracking layer options 347
...workflow 338
Subset tile properties 345
Subset tiles
...attribute status 345
subsets
...and BASE surfaces 350
...CUBE Surfaces 259
...designated soundings 350
...reset rejected data 305

...surface filtering 350, 352
...view rejected soundings 295
sun
...position 210
superselect area for beam pattern correction 427
surface cleaning
...histogram 319
...overview 312
...use tiles 315
surface statistics, compute 232
SVC
...query line 140
SVP
...display by colour 140
...display location of profile 140
...open file 140
...pole offsets in the vessel file 47
swath angle weight 190
swath cleaning 325
Swath Editor
...accept data 330
...across-track WCI view 462
...along-track view WCI 462
...auto cursor mode 328
...query data 331
...Refraction Editor 334
...reject data 330
...reject swaths 330
...reset rejected data 305
...view rejected soundings 295
...view water column data 462
swath filters 325
swath transducer location in vessel file 41
Swath, clean manually 328
SWATHPlus format, conversion options for 124
sweep transducer location in vessel file 48
symbology, track line 71
synchronize reference time stamps 84

T

Teledyne 124
template file for QC report 307
tide
... *see also* Tide Editor
...GPS tide 149
...load tide 144
...weighted average option 148
Tide stations 148
Tide Zone files 146
tile by depth range interval 399
Tile Display Query 407
Tile Overlap
...Subset tiles 348
Tiled image output, example 503
tiles
...binning 397
...close 404
...create QTC 402
...display options 405
...display properties 404
...new tile layer 397
...open 404
...view attributes 406
tiles display options 346
tiles, QTC 397
tiling 399
...by density 315
...by size in surface cleaning 315
...in Surface Cleaning 315
...primary sounding 401
...same size for all tiles 398
...tile by depth range file 400
...tile by depth tolerance 399
tiling a field sheet 396
Tiling Wizard 397
Time Correction 40
time stamp 39
time stamp overlap 84
title
...of raster legend 208
Total Propagated Uncertainty 154
tow cable
...convert 131
towed sensors location data in vessel file 49
TPE, see TPU 154
TPU 159
...apply filter 161
...compute 154
...filter 159
TPU (Total Propagated Uncertainty)
...vessel file values 51
TPU status
...field in sensor configuration 49
track lines
...classify lines 293
...colour 71
...opening a single line 66
...properties 71
...symbology 71
tracking cleaning in subsets 349
tracking layer options
...Subset Editor 347
Transducer Status, in vessel file 48
TrueHeave 289
...apply in SVC 138

U

UNB 124
UNB (Reson) format, conversion options for 124
Underlay
...use to resolve overlapping GeoBaRs 436
UNISIPS, export data to 540
units, for export 500
Unmask contours 366

update backscatter imagery, after swath cleaning 332
Update Coverage 205
Update existing survey lines 83
UTC, no distinction made for 39, 39, 39

V

VCF (Vessel Configuration File), see HVF 26
vertical axis
...in profile 370
Vertical Reference System Editor 215
vertical uncertainty 224
Vessel Configuration file 26
Vessel Editor
...coordinate system 53
...dynamic draft 46
...ellipsoid 55
...gyro 43
...heave 43
...navigation 42
...pitch 44
...reference position 53
...roll 45
...swath transducer location 41
...sweep transducers location 48
...time error 40
...time stamp 39
...towed sensors location 49
...TPU 51
...vessel outline 35
...waterline data 50
vessel file
...create new 27
...location 28
...motion sensors 32
...multibeam settings 29
...multi-transducer settings 30
vessel measurements 35
Vessel Properties 70
vessel shape 35
View
...sediment analysis overlay 455
view critical soundings
...in an Editor 301
...in Display window 302
view data 272
View Sediment Patches 454
view WCI data 463

W

Water Column Data in HIPS 460
water column data in Swath Editor 462
Water Column Data toolbar 465
water column imagery 459
water column imagery in stacked view 462
waterfall view
...in Mosaic Editor 416
waterline data in vessel file 50
WCI
...apply slant range filter 469
...bottom detection 471
...colour attributes 471
...display options 471
...size of points 471
...view across-track 462
...view along-track 462
...view stacked 462
WCI controls
...in Subset Editor 469
...in Swath Editor 466
WCI playback 464
WCI, see water column imagery 459
Windows shortcut to link to data 66
Winfrog format, conversion options for 124
Wireframe Hypotheses 261
workflow
...data cleaning in Subset Editor 338
...imagery processing 413
...to create vessel file 27
workflow diagram 16
workflow, multibeam 17
workflow, side scan data 412

X

XTF format
...conversion options for 126
...ship navigation source 126

Z

ZDF, load 144
zerotide 146
Zone Definition File 147