Seafloor Tutorials

for Geocap 6.3

Document generated: 16.01.2013

support@geocap.no

Geocap

Seafloor Tutorial

This tutorial collection is specific to the Seafloor functionality in Geocap, however a few chapters are of a general character and may be duplicated in other tutorials. Training data will be used as an example project, but other projects may be used instead when doing the exercises.

The order of the tutorials is more or less describing the suggested order in which a real project would be done. In this set of tutorials we will:

A. Getting Started with Geocap Seafloor

Introduction

Geocap is a software for visualization and manipulation of geodata. The core features of Geocap are:

- 2D/3D visualization of any geodata in the same graphics window
- Gridding
- Plotting
- 2D seismic and interpretation
- Geodetic conversion
- Image georeferencing
- Workflows
- GIS
- Scripting

On top of these features Geocap provides a set of plugins that fit perfectly in to your line of work:

- **Shelf** - for continental shelf delineation in accordance with United Nations Convention on the Law of the Sea, Article 76.
- **Seafloor** - for processing survey data from multibeam echo sounders,
- **Oil & Gas and GIM** - for oil exploration and reservoir modeling.

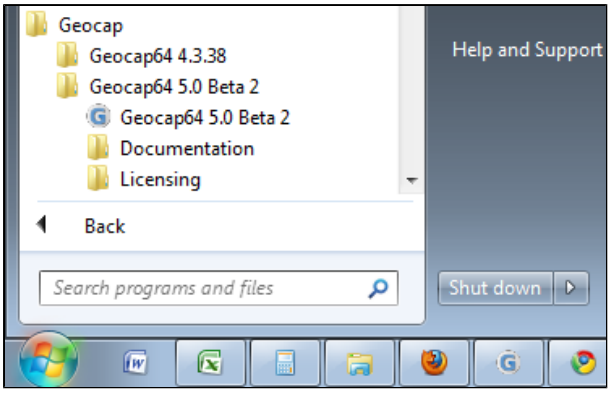
Your Geocap installation will consist of the basic Geocap platform and one or several of these plugins.

Exercises:

- Explore the different settings in the Option dialog

Start Geocap

Open Geocap from the main Windows **Start Menu > All Programs > Geocap**.



Starting Geocap from the startmenu

User Documentation

Parts of the user documentation is found under the Help pulldown. There rest is found here: http://www.geocap.no/doc. Read briefly through the documentation to get an understanding of what you can expect to find in the documents.

The user documentations consists of:

- **User Guide** with detailed descriptions of Geocap usage.
- **Reference manual** with syntax and details of the commands in the Geocap scripting language.
- **Installation Guide** with details on installation, dongle drivers etc.
- **Release Notes** contains incremental updates and bug fix descriptions as well as major releases.
- **Articles** is a collection of articles on various topics that still is not included in the documentation.
- **Tutorials** contains thematic tutorials on products. The General tutorials mainly contains topics that are put in a better context within the more specialized product tutorials.
- **FAQ** is a list of Frequently asked questions, with their corresponding answers.
- **Geocap Extensions** contains different scripts and add-on functionality for Geocap.

The Options dialog

The **Options** dialog lets you define what Geocap should do on startup. This means that you can predefine a background color, working directory, data window and automatic loading of plugins and scripts etc.

Exercise

Explore the different settings in the Option dialog

- Open the dialog by going to **Tools > Options**
- Go down section by section and make sure you understand the meaning of each of them
- Under **General**, set the **Working directory** to where your data and project is located.
- Specify your favorite background color in the **Graphics** section
- Look at the **Plug-ins** section to make sure that you have activated the right functionality
- Leave the **Projects > Sorting algorithm** on **Alphanumeric**, unless specifically requesting a numeric sorting.

B. Geocap Interface (Seafloor)

Introduction

Geocap has a very customable interface. The user may even program a new interface and develop new functionality. The concept of **commands** and **schemas** are key elements of understanding and operating Geocap.

Exercises

- Open the Atlantis (Mini) project
- Explore the project folder structure
- Display data
- Change the default display for seabed surfaces
- Navigate using the mouse and navigator panel
- Get familiar with schema commands
- Create a new command for custom display of limit lines.
- Display onto the Sticky Surface
- Commands: In panel Help
- Test keyboard shortcuts

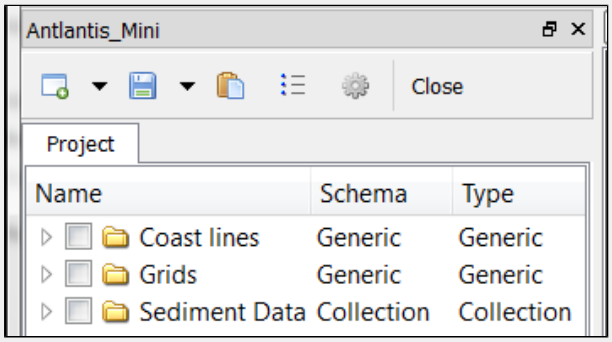
Geocap project

Geocap is operated through projects. The project "holds" the data in a folder-like structure, similar to Windows "File Explorer". All datasets are "children" of either a folder or another dataset. Geocap offers different project templates, giving you a pre-defined folder structure that fits your workflow for a specific type of work.

Exercise

Open the Atlantis (Mini) project

1. Click **File > Open > Project** and browse to the location of the Atlantis_Mini project.
2. Select the **Atlantis_Mini.db** file and click **Open**. A window similar to the one below will appear.



An open project

Tip
Next time you can open the project by using **File > Recent Projects**.

Pressing the small triangles (or '+' in older Windows versions) to the left of a folder will display the folder's contents. Datasets and folders are organized very similarly to a file tree structure. A folder can contain other folders, or datasets.

Exercise

Explore the project folder structure

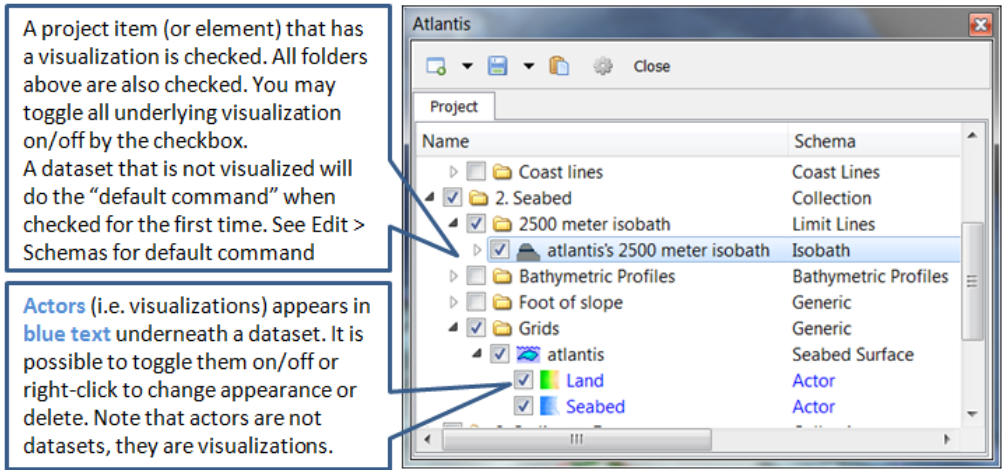
Navigate through the folder structure taking notice of how folders and datasets are organized. Datasets and folders can be cut, copied, pasted, renamed and deleted. This is performed from the popup menu which appears when right-clicking a dataset.

1. Expand the folders and observe the datasets and subfolders
2. Look at the right-click menus.

On folders, the **Multiple** will allow multiple folders to be selected for cutting, copying or deleting. The right click popup menu also contains **commands** which may be executed on the datasets. Notice the different icons of the different datasets. They correspond to the **schema** of the data set. The name of the schema is written in the second column.

Display data

Each dataset has a set of commands that can be used to display the dataset. You will find these commands in the **Toolbox** or on the right-click menu of a dataset. By executing a display command, the box next to the dataset will be checked. The dataset will also get a corresponding display actor. If you execute several display commands on the same dataset it will get one actor for each command. To see a datasets actor click the triangle next to the dataset.



Visualizations are called actors

Exercise

Display data

Display the various datasets in the project

1. Locate a grid dataset (structured points or seabed) e.g. **Grids/atlantis**
2. Right click the dataset and select **Zoom to Data**. This will make the display window center the graphics window around the dataset.
3. Right click the dataset again and try a display command e.g. **Display**, **Map Data**, **Map sea** or **Map land**.
4. Display some of the lines e.g. lines found under **Coastlines** by right-clicking and selecting the command **Display**
5. Check, and uncheck the checkboxes to the left of the displayed items and observe that this toggles displayed objects on/off. You can also check and uncheck the folders containing displayed items.
6. Items can also be displayed double clicking on one of the commands in the Toolbox (Item, Schema and Shared commands)

Exercise

Change the default display for seabed surfaces

The seabed surfaces has a default command **Map sea** or something similar.

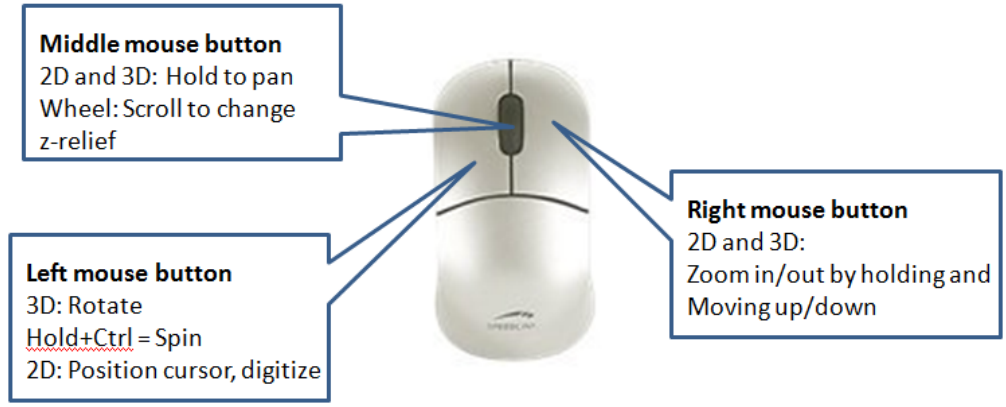
1. Go to **Edit > Schemas**. This will bring up a large panel.
2. Find the **Seabed** schema amongst other schemas in the left part of this panel.
3. When the **Seabed** schema is selected, the available commands are listed in the lower right of the panel. The default one is marked with a green ✔. Select another command (e.g. **Display Points**) and click the green ✔ icon above and observe that this command has now got the ✔.
4. Click **OK** on the panel an blank the screen by clicking the ✖ icon on the main toolbar.
5. Tick the checkbox in front of your Seabed surface. Observe that your default command is now performed.
6. You may want to go into **Edit > Schema** again to revert the default command or select a LOD command for this.

Navigating the display window

Note: Operating Geocap with a two-button mouse or using the Touch-pad is possible but not recommended. The recommendation is a three-button mouse with a wheel, see picture below.

You may use the computer mouse to move around in the display window. You do this by pressing one of the mouse buttons while the cursor is in the display window, and moving the mouse while keeping the button pressed.

- **Rotate** - Left mouse button
- **Pan** - Middle mouse button (or wheel) or Shift + left mouse button
- **Zoom** - Right mouse button
- **Scale Z** - Mouse wheel (scroll)
- **Spin** - Ctrl + left mouse button



The mouse buttons

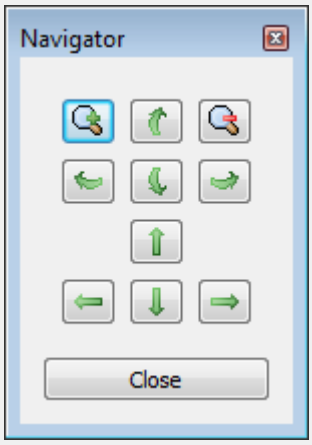
Scrolling the mouse wheel is one way to scale depth values of the dataset. The z values can also be scaled by clicking the **Actor Scale** button in the toolbar and dragging the z slider.

Tip
Set the focal point by positioning the mouse cursor on a desired point in the display window and push the **X** key on the keyboard. This focal point will be the center of the display and the point of rotation.

Exercise

Navigate using the mouse and navigator panel

- Use the mouse in the display window to zoom, pan, and rotate the data view in the graphics window.
- Instead of using the mouse for navigation, you may also use the **Navigator**. Click the  button in the main tool bar and the navigator panel will appear.
- Play around with the buttons in the navigator.



The navigator panel

Important Toolbar Buttons

See the full explanation of these buttons in the User Guide: **Geocap Main > Interface > Main Toolbar**

Basic Concepts in Geocap

The interface to specific features in Geocap are through the use of **schemas** on datasets and **commands**. The **commands** can be found in the **Toolbox** to the right in the Geocap interface or at the top of the menu which appear when you right click a dataset. Which commands are displayed in the right click menu depend on the **schema** of the dataset. A **Seabed Surface** will contain commands appropriate for the **Seabed Surface** schema, while a dataset with the schema **Structure Points** will contain different commands.

Schemas

Geocap uses schemas to classify a dataset. The Shelf Module i.e. contains several schemas. Some of the schemas used in the Shelf Module are **coast line**, **base line**, **limit line**, **sediment thickness**. You can define the schema of a dataset in the project by right-clicking it, and selecting **Set Schema** in the pop-up menu. The choice of a datasets schema controls which commands you see in the pop-up menu when you right-click the dataset. You may create your own schemas as well as edit existing schemas by selecting **Schemas** under **Edit** in the main menu. You can also edit the commands associated with the schemas.

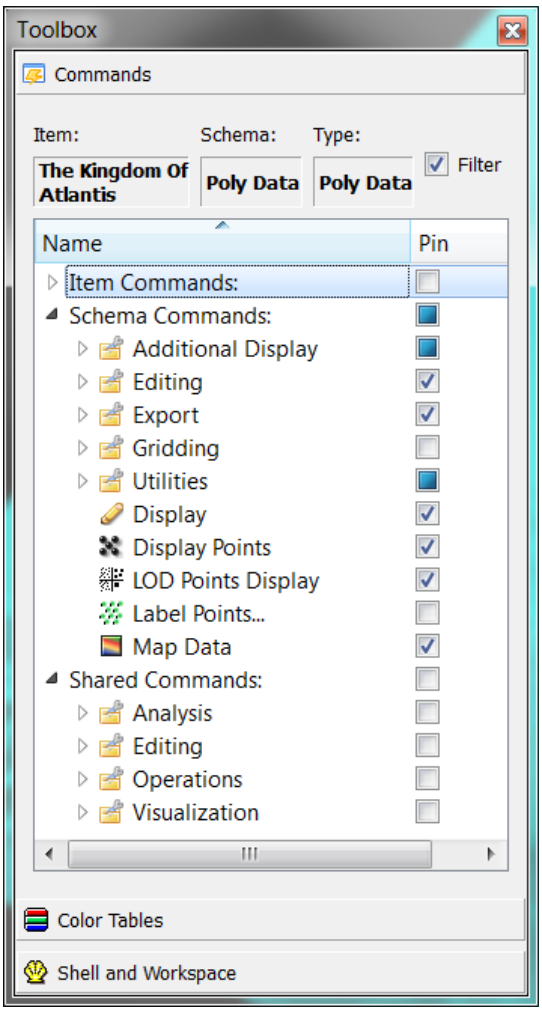
Commands

Commands are operations which can be performed on a dataset. Commands can for example be used to display a dataset in the display window, or to generate new datasets. You can even create your own scripted commands to cater to your specific needs. You execute a command by right-clicking the dataset or folder you want to run it on, and then selecting the relevant command in the pop-up menu. You can give different parameters to a command in the command editor. These parameters are stored with the command and used during execution.

Commands can be stored in three categories:

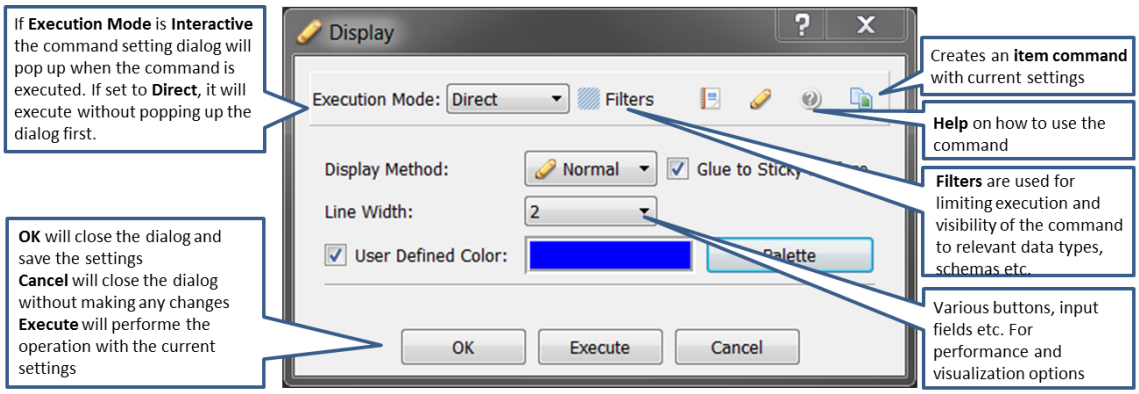
1. Item commands
2. Schema commands
3. Shared commands.

You will find the commands sorted into the different categories in the **Toolbox** (see illustration below) or on the right-click menu of a dataset or a folder. Commands are also put together in sequence in **Workflows** to perform visualizations or data operations, see chapter **R**.



The Commands section of the Toolbox

All commands have a front end panel, and most of them have settings that may be customized. You can open the front end panel by right-clicking the command and select **Edit**



An example of a command front end panel

Item Commands

A command can be stored at the level of a dataset or a folder. This is called an item command. This command is unique to this dataset or folder, it "belongs" to that dataset. You can see these commands on the top of the **Toolbox** or in a sub menu when you right-click a dataset and select **Item commands**. Most items in the project do not contain any item commands by default.

Schema Commands

A command stored at a schema level is called a schema command. All datasets or folders using the same schema share these commands, which also means that editing these commands will affect all the datasets using this schema. The schema commands of a dataset are listed on the top of the right-click menu.

Exercise

Get familiar with schema commands

- Right click the different datasets in the project, and see how the right click menu changes from schema to schema.

Shared Commands

Shared Commands are commands which are shared with all datasets and folders. The shared commands are listed in the **Toolbox** under **Shared commands**. If you cannot see the Toolbox, it can be opened from **View** on the main menu.

All commands have a command editor where you may change the properties, thus affecting the way it is executed.

Exercise

Create a new command for custom display of limit lines.

1. Click on one of the datasets in the **Coastline** folder
2. Right-click the **Display** command under **Schema commands** in the **Toolbox** and select **Copy**
3. Right-click **Schema commands** in the **Toolbox**, select **Paste** and observe that a copy of the selected command called **Display-1** will appear in the schema command list.
4. Right-click the new command in the command list, and select **Rename**
5. Name the command **Display Thick Yellow**
6. Right-click the command and select **Edit**. The settings dialog for the selected command will appear
7. Set **Line width** to **6**
8. Check the **User defined color** box and click the **Palette** button.
9. Select a yellow color and click **OK**
10. Click **OK** to close the **Display Thick Yellow** settings dialog.
11. Right-click the same data set, and observe that our new command is present in the right-click menu. This is because the **Pin** checkbox next to the **Display Thick Yellow** command in the **Toolbox** is checked.
12. Un-check the **Pin** checkbox next to the **Display Thick Yellow** command in the **Toolbox**.
13. Right-click the same dataset again, and observe that our new command is not present in the right-click menu.
14. Execute the command by double clicking **Display Thick Yellow** in the **Toolbox**. Observe that the line is displayed in yellow.
15. Check the **Pin** check box next to the **Display Thick Yellow** command in the **Toolbox** again. Now right click a different dataset with the same schema (Poly Data) and observe that the new command can be executed on this right-click menu as well.

Tip
The **Pin** check box lets you decide which commands should be available in the right click menu, so it is easy to keep organized. Try to experiment with this option to manipulate the right click menu.

The Sticky Surface

Geocap has a concept where any surface can be set to be **sticky**. When a surface is sticky, datasets like points, lines or images may be displayed onto that surface. This is mainly done by re-sampling lines and displaying them a little bit above the sticky surface.

When a surface is activated (or set) as a sticky surface, it is copied to workspace (visualized in the toolbox) under the name **sticky_surface**. If this dataset is removed from workspace, there is no sticky surface anymore.

Exercise

Display onto the Sticky Surface

1. Display the Seabed Surface under **Grids\atlantis**
2. Right-click on **Grids\atlantis** and select **Set as sticky surface**
3. Select a line, e.g. **Sediment Data\Navigation\ATL-99-1** in your project.
4. In the Toolbox under **Commands > Schema Commands** right-click and select **Edit** on the **Display** command.
5. Check the **Glue to Sticky Surface** and press **Execute** to display the line on the surface.
6. Uncheck the **Glue to Sticky Surface** and press **Execute** again and observe the difference.

Warning
Note that points and lines displayed onto a sticky surface are displayed without their original z-values, and this may not be what you intend to do when displaying i.e. a bathymetric line. Keep that in mind.

Exercise

Commands: In panel Help

1. In the **Toolbox** right-click the command "Display Points" and select **Edit**
2. Click the icon to open the in-panel help.
3. Read the information that pops up.

Keyboard shortcuts

Geocap has several **keyboard shortcuts** or **hotkeys**. Go Help > Keyboard shortcuts to bring up a list.
A selection of the most important keyboard shortcuts:

Key	Explanation	Key	Explanation
g	Fly and zoom towards the position you are pointing at. The cursor needs to point at graphics.	**+**	Zoom in
o	Toggle color code for last used map command on/off.	**-**	Zoom out
r	Rescale to all graphical elements. Note that r does not set the window.	**2**	Toggle graphics to 2D mode.
s	Turn graphics into *surface* mode.	**3**	Toggle graphics to 3D mode.
v	Value of surface depth (z coordinate) from graphics		
w	Turn graphics into *wire* mode.	**x**	Setting the focal point. The graphics will rotate around this point.
y	Cursor point is set at the surface of the graphical element.	**z**	Zoom by drawing a rubber band with leftmost mouse button. (2D mode only)

Exercise

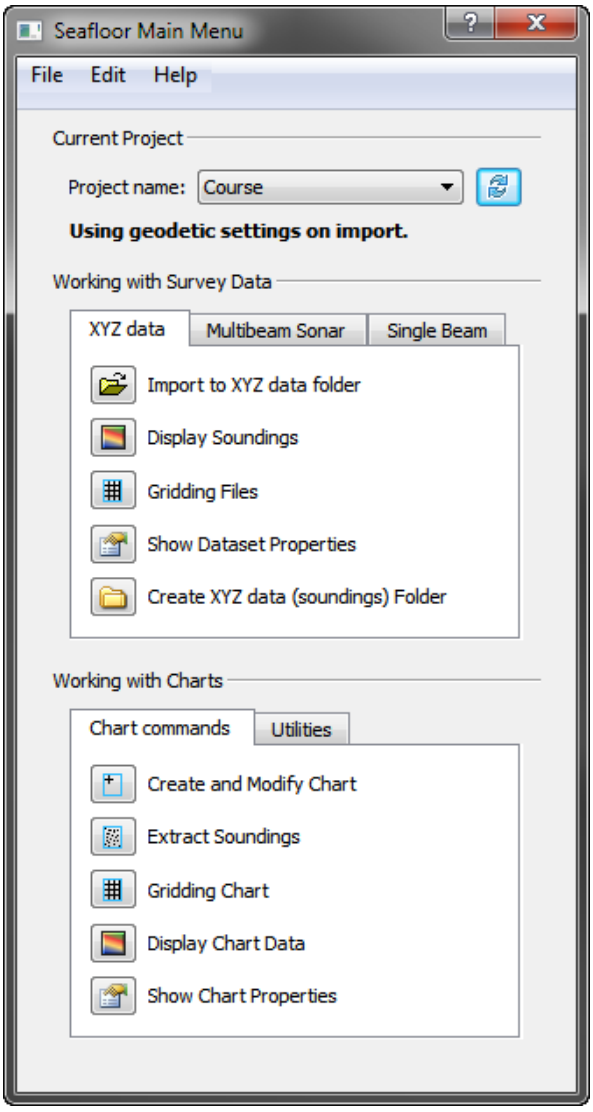
Test keyboard shortcuts

- Visualize a seabed surface and test all the above mentioned keyboard shortcuts.

C. Seafloor Main Menu (Extract from User Guide)

Introduction

The **Seafloor Main Menu** is a collection of commands and project properties in a Geocap Seafloor project. The most frequently used commands are accessible through this menu, which will make it easier for new users to operate the Seafloor user interface. However, these and more commands are also available from the project interface.

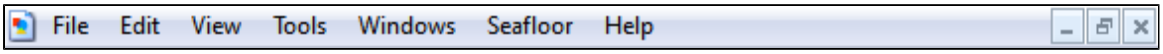


The Seafloor Main Menu

How to open the Seafloor Main Menu

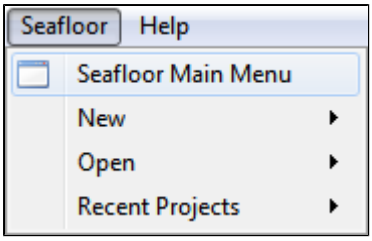
To open the **Seafloor Main Menu** it is required that the **Seafloor package** is loaded, and that requires that the license file contains one of the features for loading the package.
The Seafloor package is loaded either on startup of Geocap, or on the **Tools** menu, click the **Options** menu. In the menu that appears, click **Plug-ins**, and then select **Seafloor** in the right hand part of the menu.

When the **Seafloor package** is loaded, the menu bar in Geocap will contain a **Seafloor** pulldown menu.



The Seafloor pull down in the Geocap menu bar

On the **Seafloor** menu, click **Seafloor Main Menu**.



Open the menu

Then the **Seafloor Main Menu** is opened (see figure above).

For all actions in this menu it is required to load a project. If no project is loaded, the message "No project is currently loaded into Geocap!" will appear. Then a description on how to load a project follows.

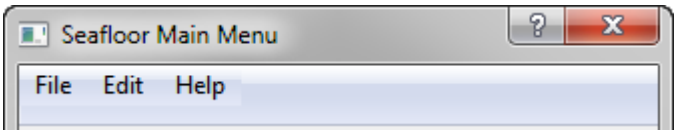
The Content of the Seafloor Main Menu

The **Seafloor Main Menu** is divided in four parts, the **menu bar**, the **Current project** part, the **Working with Survey Data** part, and the **Working with Charts** part.

- The **menu bar** has three options: File, Edit, Help
- The **Current project** part contains information about which project is active, and what coordinate system the project is using.
- The **Working with Survey Data** part is the interface to useful commands for processing survey data through the **Geocap Seafloor** package.
- The **Working with Charts** part is the interface to useful commands for processing chart data through the **Geocap Seafloor** package.

Menu bar options (File, Edit, Help)

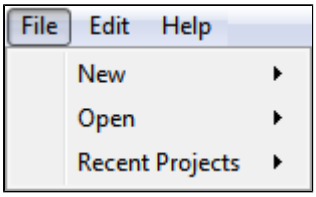
The upper part of the menu is a menu bar with the entries **File**, **Edit** and **Help**:



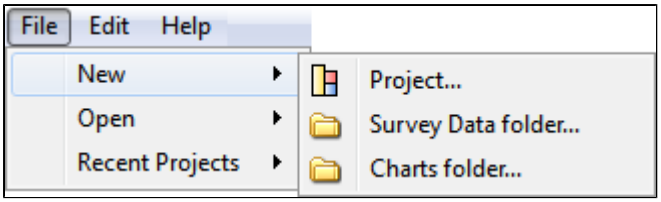
The Menu bar

The actions from menu bar entries:

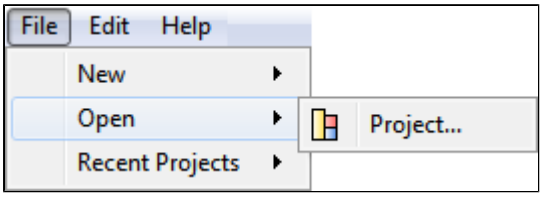
File:



- **New - Project...** will open the dialog to make a new Geocap (Seafloor) project.
  The dialog ends with a question about setting geodetic properties for the project.
- **New - Survey Data folder...** will create a folder of type **Survey** from top level in the project.
- **New - Charts Data folder...** will create a folder of type **Charts** from top level in the project.
  The last two commands are useful if the project does not have a folder of the actual type, or if a more than one folder of this type is required.

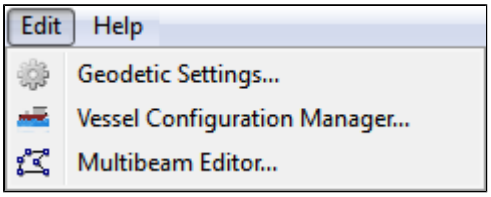


- **Open - Project...** will open any existing Geocap projects in the computer or network.



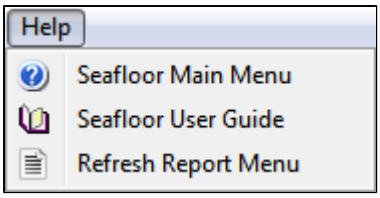
- **Recent Projects** will give a list of recently opened Geocap projects.

Edit:



- **Geodetic settings** will open the dialog for setting geodetic properties for the project.
- **Vessel Configuration Manager** will open the dialog for setting vessel properties for all open projects. This entry will have a meaning only when *Multibeam Sonar data* are loaded
- **Multibeam Editor** will open the 3D editing tool for multibeam points (Multibeam Editor). This entry will have a meaning only when *Multibeam Sonar data* are loaded

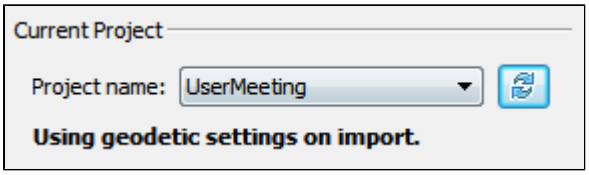
Help:



- **Seafloor Main Menu** will show the help dialog.
- **Seafloor User Guide** will open the Seafloor User Guide documentation as a pdf file.
- **Refresh Report Menu** will open the reports dialog containing the last processing results, if it is closed or hidden.

The Current Project part

The **Current Project** part shows the **active** project in the **Seafloor Main Menu**. In Geocap the user can load several projects at the time.



The Current Project part

All projects are shown in the pulldown menu behind the text *Project Name:*. The icon can be clicked to be sure that the project information is updated if a project is closed, or a new loaded from the Geocap main window.

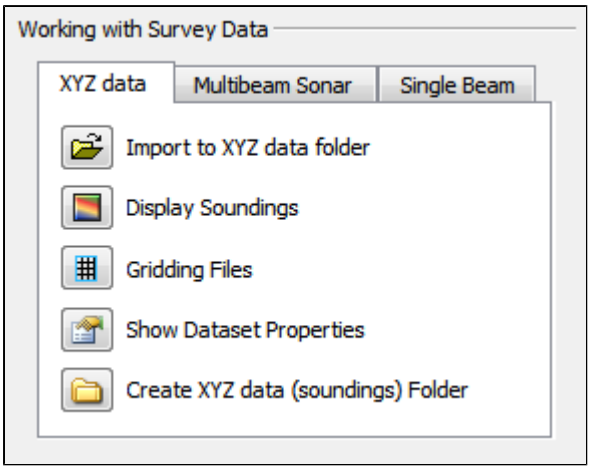
The text **Using geodetic settings on import** indicates that during import of any dataset into Geocap, the data will be transformed to the project coordinate system. This action is set up in the *Geodetic Settings* menu.

To change the coordinate system, find the **Edit** menu, and click **Geodetic Settings**.

The text **Not using geodetic settings on import** indicates that a coordinate system is defined, but no transformation will be performed during import.

The text **No geodetic settings is defined** indicates that the project doe not have any coordinate system.

The Working with Survey Data part

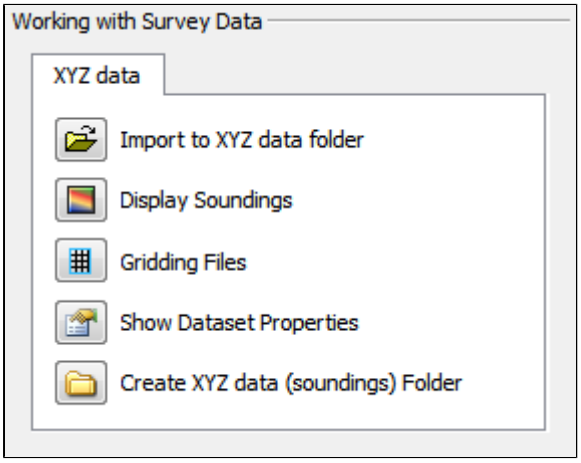


The Working with Survey Data part

This part contains tabs for import, display and work with data typically from surveys. The three tabs are:

- The XYZ data tab
- The Multibeam Sonar tab
- The Single Beam tab

The available tabs will be dependent on the features in the license file. For Geocap Shelf users the tab menu will only have the XYZ data part.

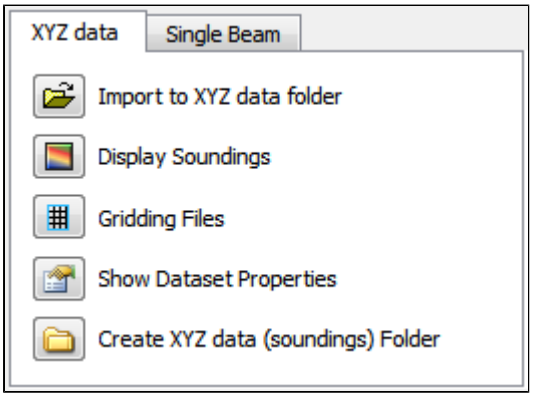


The tabs for Shelf

The XYZ data tab

The commands in the XYZ data tab are shortcuts to the commands on a folder of type **Soundings**. Many of the commands are **Item Commands**, which means that the settings in the menu is remembered for each folder.

If there are several folders of the same type in the project, a dialog will appear.



The XYZ data tab

Import to XYZ data folder: This command will import data to folders of type **Soundings**.

There is also a choice to use the ***Generic reader*** or the ***ASCII Column reader***.

Display Soundings: This command will display the data in folders of type **Soundings**.

Gridding Files: This command will make it possible to do gridding (terrain modelling) of the data in folders of type **Soundings**.

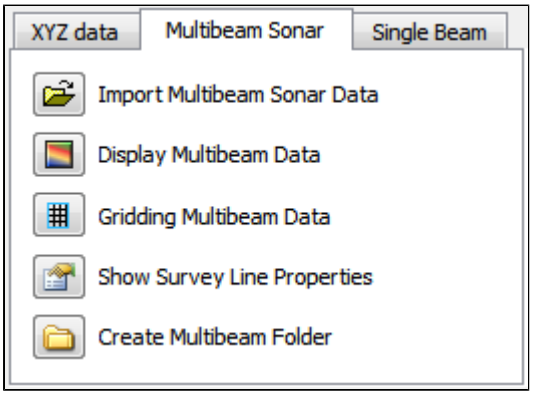
Show file properties: This command will show dataset name, bounding box and number of points of the data in folders of type **Soundings**.

Create XYZ data (soundings) Folder: This command will create a new folder of the type **Soundings** as a subfolder under a folder of type **Survey**.

The Multibeam Sonar tab

The commands in the Multibeam Sonar tab are shortcuts to the commands on a folder of type **Multibeam**. Many of the commands are **Item Commands**, which means that the settings in the menu is remembered for each folder.

If there are several folders of the same type in the project, a dialog will appear.



The Multibeam Sonar tab

Import Multibeam Sonar Data: This command will import data to folders of type **Multibeam**.

Display Multibeam Data: This command will display the data in folders of type **Multibeam**. Also data set name and navigation lines can be diplayed.

Gridding Multibeam Data: This command will make it possible to do gridding (terrain modelling) of the data in folders of type **Multibeam**.

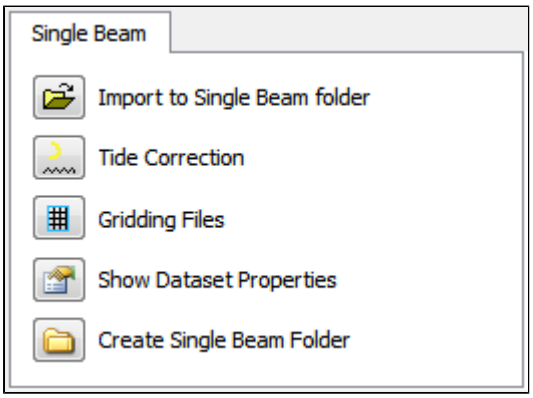
Show Survey Line properties: This command will show dataset name, bounding box and number of points of the data in folders of type **Multibeam**.

Create Multibeam Folder: This command will create a new folder of the type **Multibeam** as a subfolder under a folder of type **Survey**.

The Single Beam tab

The commands in the Single Beam tab are shortcuts to the commands on a folder of type **Single Beam**. Many of the commands are **Item Commands**, which means that the settings in the menu is remembered for each folder.

If there are several folders of the same type in the project, a dialog will appear.



The Single Beam tab

Import to Single Beam folder: This command will import data to folders of type **Single Beam**.

There is also a choice to use the ***Generic reader*** or the ***ASCII Column reader***.

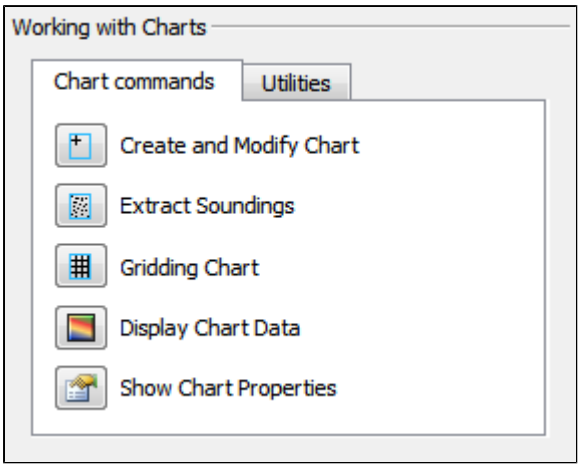
Tide Correction: This command will make it possible to perform tide correction of the poly data in a folder of type **Single Beam**. The requirement for this type of correction is that the data set contains a **DateTime** column.

Gridding Files: This command will make it possible to do gridding (terrain modelling) of the data in folders of type **Single Beam**.

Show file properties: This command will show dataset name, bounding box and number of points of the data in folders of type **Single Beam** .

Create Single Beam Folder: This command will create a new folder of the type **Single Beam** as a subfolder under a folder of type **Survey**.

The Working with Charts part



The Working with Charts part

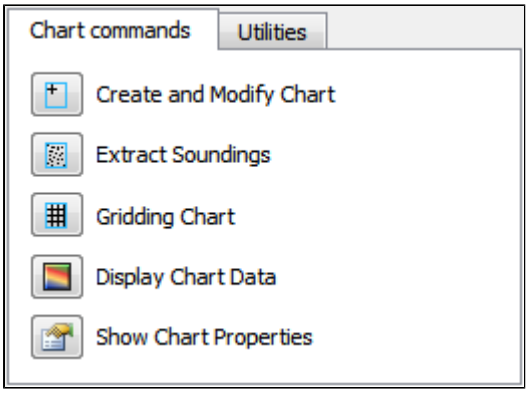
This part contains tabs for managing, display and work with data organized in charts (areas of interest). The two tabs are:

- The Chart commands tab
- The Utilities tab

The Chart commands tab

The commands in the Chart commands tab are shortcuts to the commands on folders of type **Charts** and on folders of type **Chart**. Many of the commands are **Item Commands**, which means that the settings in the menu is remembered for each folder.

If there are several folders of the same type in the project, a dialog will appear.



The Chart commands tab

Create and Modify Chart: This command will create or modify folders of type **Chart**.

Extract Soundings: This command will read data stored below a folder of type **Survey**, and save the result as a **Soundings** data set in folders of type **Chart**.

Gridding Chart: This command will work on folders of type **Chart**.
The command will read the **Soundings** dataset and create a digital terrain model (DTM). The result will be saved in the same folder as a **Seafloor** data set.

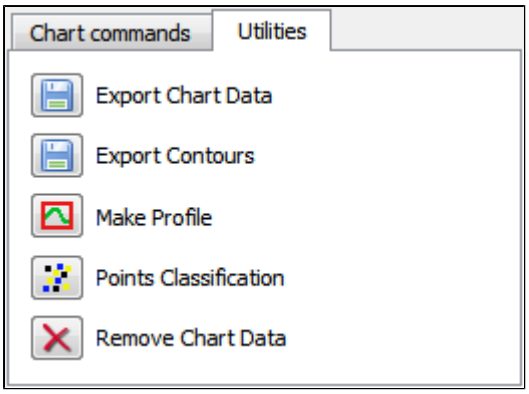
Display Chart Data: This command will display the different data stored in folders of type **Chart**. If there are several **Charts** folders in the project, a dialog will appear.

Show Chart properties: This command will show properties of the content of folders with type **Chart**.

The Utilities tab

This part contains buttons with commands to show properties, display and export data in a **Seafloor** project.
All commands work on folders on type **Charts** and on folders of type **Chart**.

If there are several folders of the same type in the project, a dialog will appear.



The Utilities tab

Export Chart Data: This command will export different type of data from folders of type **Chart**.

Typical datasets to export are **Frame** and **Boundary** polygons, **Soundings** (points) and **Surface** (DTM).

Export Contours: This command will export **Contours** data from the **Seafloor** (DTM) data set in folders of type **Chart**. An option is to export as **Contour Areas** instead.

Make Profile: This command will export create a continuous depth profile (intersection) through the Seafloor (DTM) data set in folders of type Chart. (Not enabled for Shelf.)

Points Classification: This command will anaylyze the Seafloor (DTM) data set in folders of typeChart. The commands will create new points data sets of types **Shoals**, **Deeps**, **Ridges**, **Saddles** and **Valleys**. (Not enabled for Shelf.)

Remove Chart Data: With this commands it is possible to remove specific data objects from folders of type **Chart**. The command can be specified to **Remove entire chart folder and data**, **Remove specified chart folder item(s)**, or **Remove empty chart**.

D. Create a New Seafloor Project

Introduction

Geocap provide empty folder structure for various types of projects. This will give you a starting point to get organized with your own project. A project template either gives you a ready made folder structure, or it gives you a suite of folders to choose from.

An empty project structure may be used for communicating relevant data to colleagues, or for analysis and trouble shooting by us. To send parts of a project to us in Geocap Support, even only a single dataset, you may copy the dataset (or a folder) from your main project and paste it into this empty project. Then zip and send the disk folder *projectname*.zip to support@geocap.no.

The Seafloor project template

Geocap can provide an empty folder structure for Seafloor projects. This will give you a starting point to get organized with your own Seafloor project. The default folder structure holds empty folders for most of the data types you will need. If you do not find a suitable folder you can create a new folder for that data. It is also a good idea to create sub-folders if you have a lot of data, for instance a sub-folder for each survey or for each region.

Exercises

- Generate a new and empty Seafloor project
- Explore the project settings

Exercise

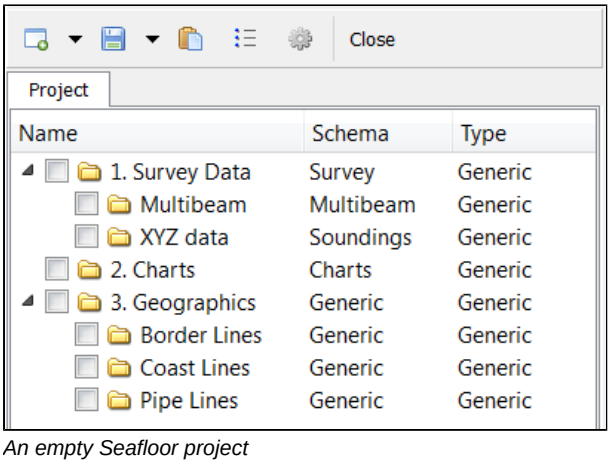
Generate a new and empty Seafloor project

1. In the main menu of Geocap, click **Seafloor > Seafloor Main Menu**
2. In the Seafloor Main Menu click **File > New > Project**. A new dialog will pop up.
3. Select the **Seafloor** project template
4. Type in the name of your project in the **Name** field. The name may consist of letters, numbers and spaces, but special characters like [, æ,ø, å, &, /, % ... should be avoided.
5. Click the **Browse** button, and select where you want to store your project on your hard disk.
6. Click **Finish**.

You will now be prompted with a window asking you if you want to define the geodetic setting of the project.

1. Click **Yes**
2. Check the **Use geodetic settings for project** box and select **WGS84** as datum and **UTM Zone 32 N** as coordinate system.
3. Click **OK**

When enabling geodetic settings all data will be converted to these settings on import.



An empty Seafloor project

Your project will now look like the image above. The idea is that this basic structure is kept. Folders may be added and data imported, but the original folders should not be renamed or moved, and their schemas should not be changed. This is because this folder structure is used when new datasets are generated. If an original folder is not present, it will be recreated.

Exercise

Explore the project settings

- Click the  icon on the project toolbar to look at the settings. Pay particular attention to **Geodetics**. You can see that the settings you previously entered are stored here. You can also access these from the Seafloor main menu **Edit > Geodetic Settings**

E. Import Data

Introduction

Geocap supports a vast amount of Multibeam Sonar Data, in addition to Single Beam and XYZ data. These types of datasets are imported in different folders in the **1. Survey Data** folder.

Geocap also supports import of sensor files in ASCII format, like Tide, Navigation, Velocity, Gyro, Heave, Roll and Pitch.

Exercises

- Import XYZ data
- Import Multibeam Sonar data
- Show properties for the Multibeam data
- Show properties for the XYZ data

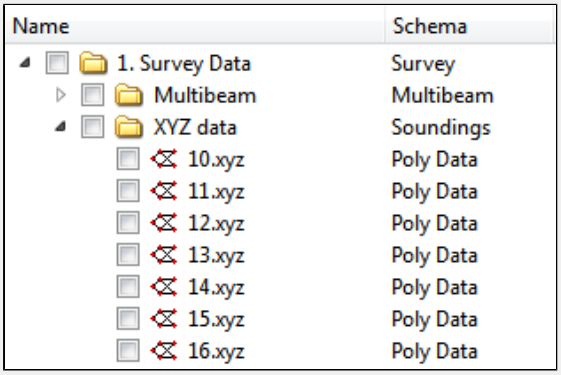
Importing

Exercise

Import XYZ data

1. Click the **XYZ data** tab in the **Seafloor Main Menu**
2. Click **Import to XYZ data folder**
3. Click **Generic reader**
4. Click the  icon and browse to the folder **..\Seafloor_Course\XYZ\Simrad_xyz** on disk.
5. Select all files and click **Open**
6. Observe that the **File header/information** looks ok.
7. Keep the **Schema** as is.
8. Click **Execute** and the import will start.
9. After import a dialog will pop up, with information about the import. Click **OK** on this.

You should now be able to see your xyz data files in the project folder **1. Survey Data / XYZ data**



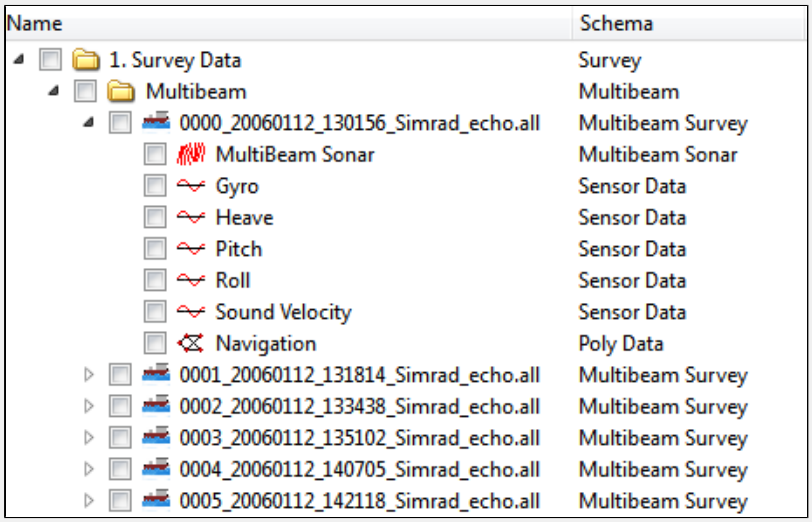
Imported XYZ files

Exercise

Import Multibeam Sonar data

1. Click the **Multibeam Sonar** tab in the **Seafloor Main Menu**
2. Click **Import Multibeam Sonar Data**
3. Click the icon and browse to the folder **..\Seafloor_Course\Multibeam Sensor Data\EM_710** on disk.
4. Select all files and click **Open**
5. Keep the **Import Options** as is.
6. Click **Execute** and the import will start.
7. After import a dialog will pop up, with information about the import. This report will also be available in the folder **1. Survey Data / Multibeam /Reports**.

You should now be able to see your multibeam files in the project folder **1. Survey Data / Multibeam**



Imported Multibeam Sonar data

Read more about the import options by clicking the icon on the import menu.

The different import commands are also available on the right-click menus of the **Multibeam** folder and **XYZ data** folder.

Data properties

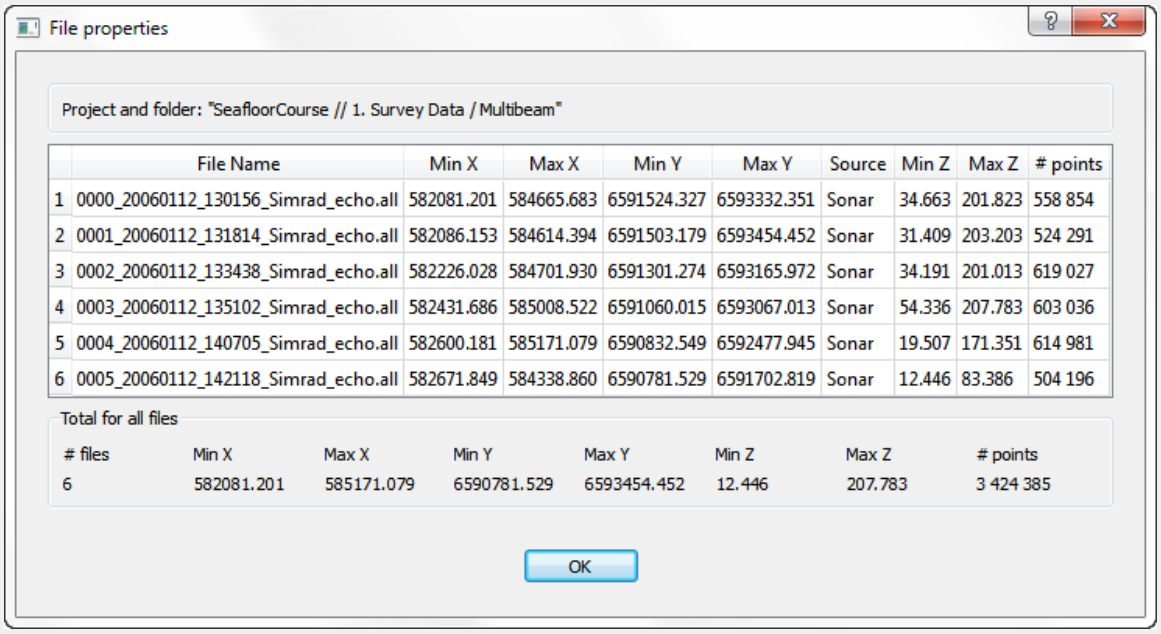
After importing it might be valuable to take a look at the properties for the files you just imported, to check the range, number of points etc.

Exercise

Show properties for the Multibeam data

1. Click the **Multibeam Sonar** tab in the **Seafloor Main Menu**.
2. Click **Show Survey Line Properties**. A window containing the properties will pop up.

Observe that the list contains a row for each file, with X-, Y-, Z-range and number of points. At the bottom of the window you will see the total range and number of points for all files.



	File Name	Min X	Max X	Min Y	Max Y	Source	Min Z	Max Z	# points
1	0000_20060112_130156_Simrad_echo.all	582081.201	584665.683	6591524.327	6593332.351	Sonar	34.663	201.823	558 854
2	0001_20060112_131814_Simrad_echo.all	582086.153	584614.394	6591503.179	6593454.452	Sonar	31.409	203.203	524 291
3	0002_20060112_133438_Simrad_echo.all	582226.028	584701.930	6591301.274	6593165.972	Sonar	34.191	201.013	619 027
4	0003_20060112_135102_Simrad_echo.all	582431.686	585008.522	6591060.015	6593067.013	Sonar	54.336	207.783	603 036
5	0004_20060112_140705_Simrad_echo.all	582600.181	585171.079	6590832.549	6592477.945	Sonar	19.507	171.351	614 981
6	0005_20060112_142118_Simrad_echo.all	582671.849	584338.860	6590781.529	6591702.819	Sonar	12.446	83.386	504 196

Total for all files

# files	Min X	Max X	Min Y	Max Y	Min Z	Max Z	# points
6	582081.201	585171.079	6590781.529	6593454.452	12.446	207.783	3 424 385

Showing Survey Line Properties

You can copy these properties straight into e.g. Excel. Just select a row, column or the whole table (Ctrl+A), press Ctrl+C on your keyboard and den Ctrl+V in Excel.

Exercise

Show properties for the XYZ data

1. Click the **XYZ data** tab in the **Seafloor Main Menu**.
2. Click **Show Dataset Properties**. A window containing the properties will pop up.

Observe that the list contains a row for each file, with X-, Y-, Z-range and number of points. At the bottom of the window you will see the total range and number of points for all files.

These commands are also available on the **Multibeam** and **XYZ data** folders in the project. You can also display the properties from within different command menus, i.e. **Display Multibeam Data**

F. Display Data

Introduction

There are several ways to display multibeam soundings and XYZ data.
XYZ datasets has the following display commands available on their right click menu;

- **Display**
- **Display Points**
- **LOD Points Display**
- **Map Data**

Multibeam Survey files has the following display commands available on their right click menu;

- **Display**
- **LOD Points Display**
- **Map Depth**
- **Map Amplitude**

In many cases you want to display several datasets together, showing the same range for all the files. The **Display Soundings** and **Display Multibeam Data** commands in the **Seafloor Main Menu** offers an easy interface for displaying data objects together in different ways. These commands are also available on the right-click on the **XYZ data** folder and **Multibeam** folder respectively.

Level Of Detail (LOD) displays points and grids based on the current zoom level. This means that the resolution increases automatically when you zoom closer, which makes the display of large point clouds or grid a lot faster.

Exercises

- Display XYZ data (Soundings)
- Display Multibeam data
- Experiment with different display settings
- Display filtered data

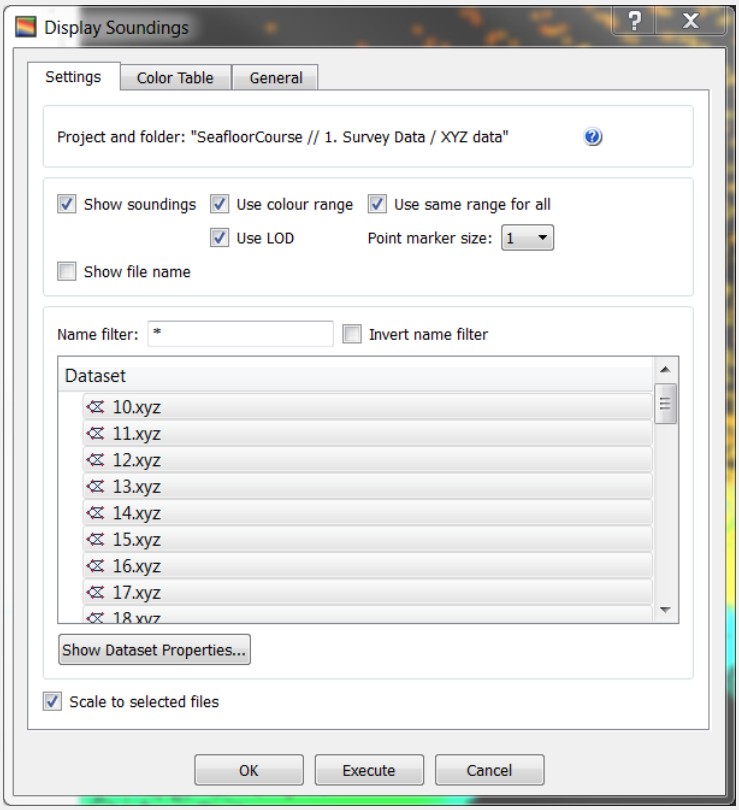
XYZ Data

Exercise

Display XYZ data (Soundings)

1. Select the **XYZ data** tab on the **Seafloor Main Menu**
2. Click **Display Soundings**
3. In the list of datasets, select the XYZ datasets you want to display and make sure the settings are equal to the image below

> Selecting datasets works the same way as in a Windows file browser, which means you can hold down *shift* or *ctrl* to select several datasets, or use *ctrl + a* to select all.



4. Click **Execute**

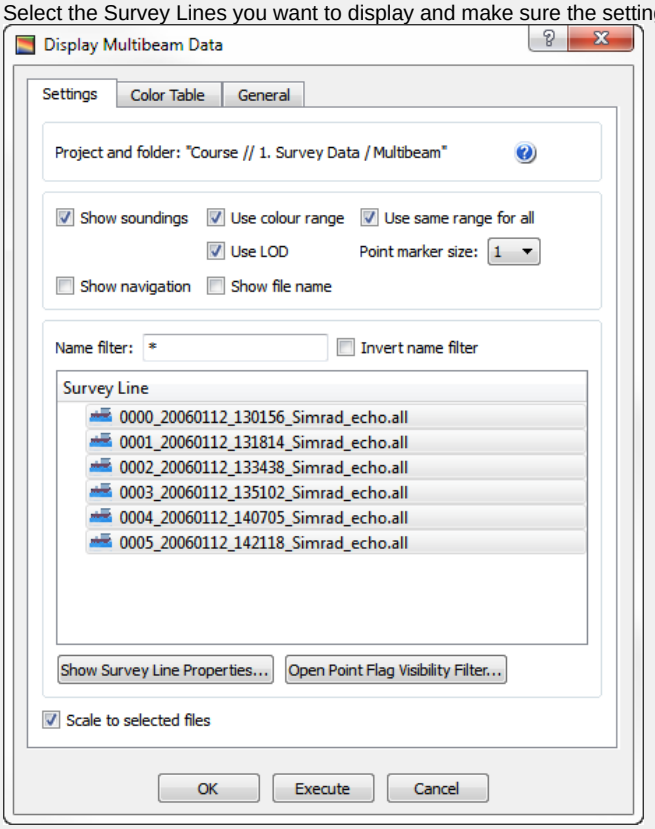
You should now be able to see the soundings in the graphics window.

Multibeam Data

Exercise

Display Multibeam data

1. Select the **Multibeam Sonar** tab on the **Seafloor Main Menu**
2. Click **Display Multibeam Data**
3. Select the Survey Lines you want to display and make sure the settings are equal to the image below



4. Click **Execute**

You should now be able to see the soundings in the graphics window.

Exercise

Experiment with different display settings

Make sure the **Display Mulitibeam Data** menu is open and that all survey lines has been selected.
Try this:

- Un-check **Use color range** and click **Execute** - Observe that the soundings from each survey line will have different colors
- Set the **Point marker size** to **5** and click **Execute** - Observe that the points size increased.
- Check the **Show navigation** box and click **Execute** - Observe that the navigation line is now displayed.

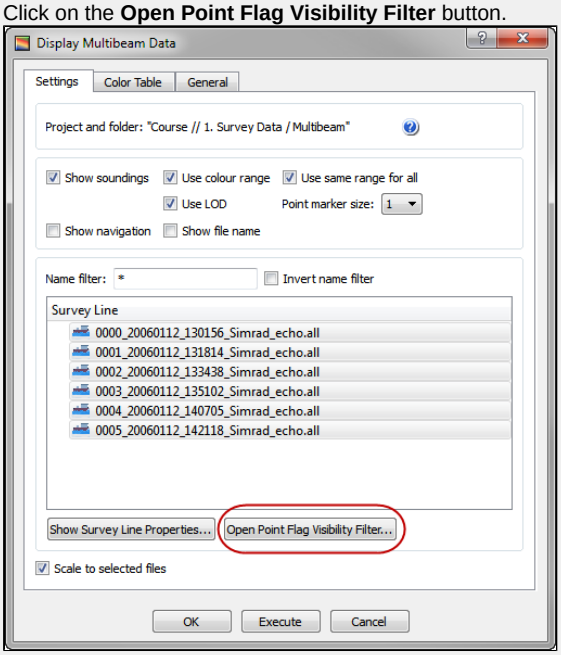
1. Check the **Use color range** box again and go to the **Color Table** tab.
2. Check the **Use Color Table** box, select **Absolute range** and type in the values **50** and **200**.
3. Click **Execute** and observe that the mapping range is between 50m and 200m.

Exercise

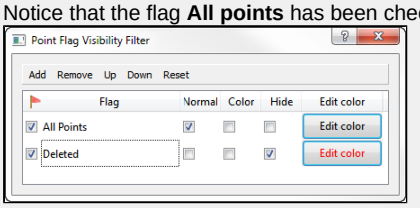
Display filtered data

Make sure the **Display Mulitibeam Data** menu is open and that all survey lines has been displayed.

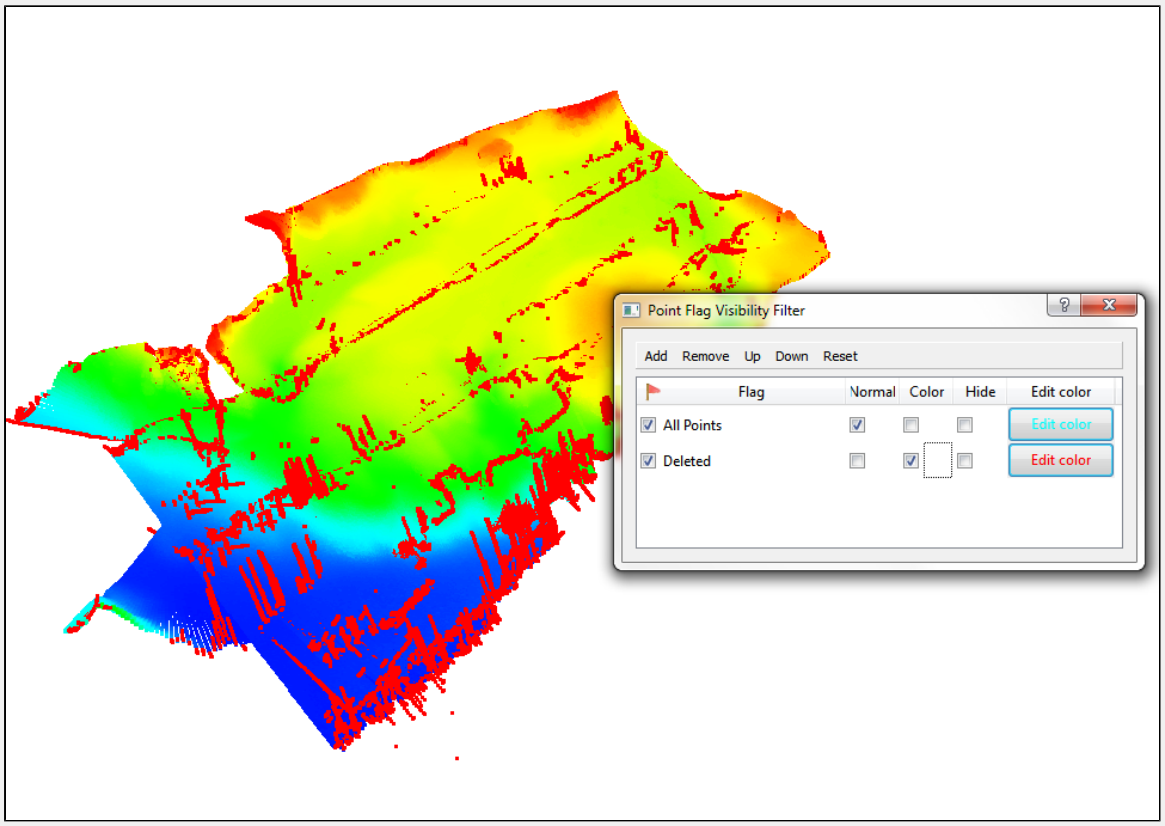
1. Click on the **Open Point Flag Visibility Filter** button.



2. Notice that the flag **All points** has been checked as **Normal** and that the flag **Deleted** has been checked as **Hide**



3. Check the **Color** box for the **Deleted** flag.



The red points that are now displayed are points that were filtered away during import.
This could be because of missing navigation or that they were flagged as not valid by the multibeam echosounder.

G. Charts

Introduction

Working with large amounts of data can be cumbersome. Thus, Seafloor lets you divide your survey area into smaller areas called charts. These charts are populated with multibeam soundings and you can then perform operations like gridding and points classification on each individual chart. You can later merge these charts together again.

Exercises

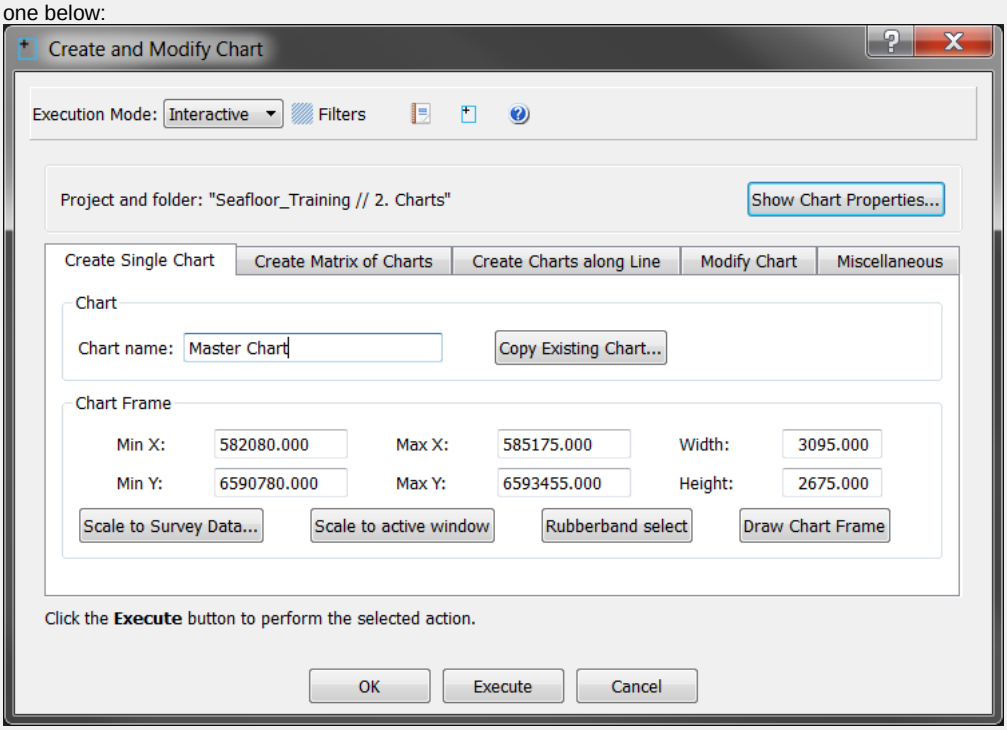
- Create a master chart
- Create sub-charts
- Populate charts with soundings
- Show properties for each chart
- Display chart frames
- Display chart soundings

Creating and populating charts

Exercise

Create a master chart

1. Click the **Chart commands** tab in the **Seafloor Main Menu**
2. Click **Create and Modify Chart**
3. In the **Create Single Chart** type in *Master Chart* as **Chart name**
4. In the **Create Single Chart** tab click **Scale to Survey Data**
5. In the dialog that pops up, tick the **Multibeam** folder checkbox and click **OK**. The menu should now look similar to the one below:



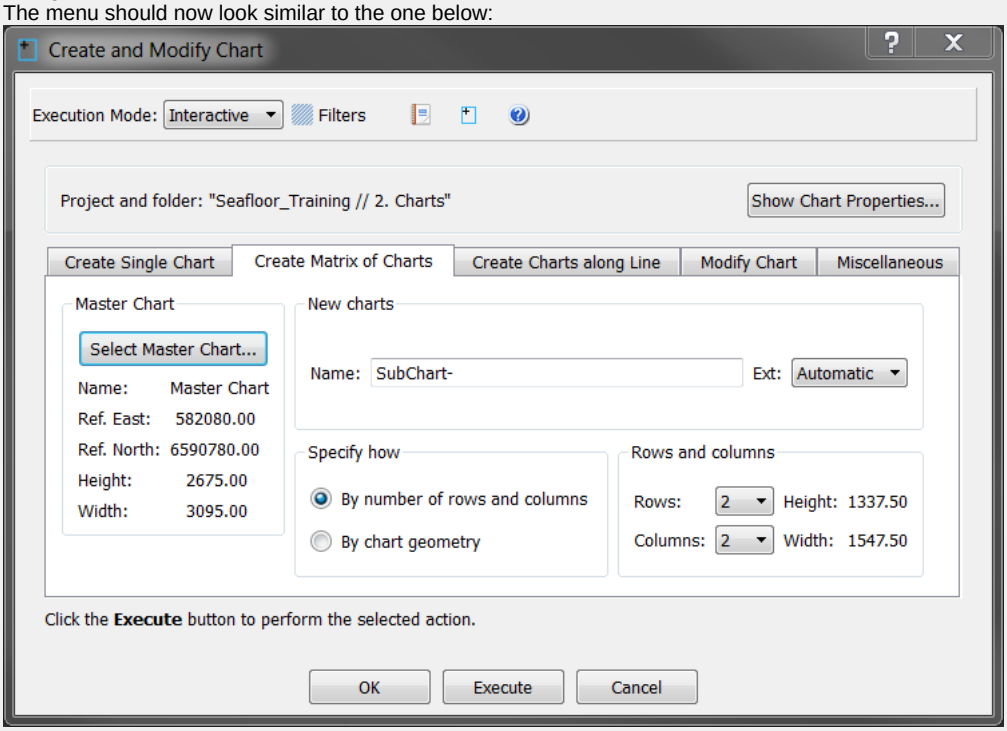
6. Click **Execute** to create the chart and click **OK** in the dialog that pops up.

You have now created a master chart for the whole area of your survey, and we are now going to create sub-charts for this master chart.

Exercise

Create sub-charts

1. Click the **Create Matrix of Charts** tab.
2. Make sure that your master chart has been selected in the **Master Chart** section of the menu.
3. Keep the name **SubChart-** with extension set to **Automatic**. This will give the chart name **SubChart-1** and so on.
4. Make sure that **By number of rows and columns** has been selected
5. Set **Rows** and **Columns** to **2**. Note that the **Height** and **Width** of the charts will automatically change when you change the amount of rows and columns.
6. The menu should now look similar to the one below:



7. Click **Execute**
8. You will be asked if you want to create 4 (2x2) charts. Click **Yes**
9. Click **OK** in the dialog that pops up.
10. Click **OK** on the **Create and Modify Chart** menu

You now have one **Master Chart** folder and several **SubChart-X** folders inside the **2. Charts** folder.

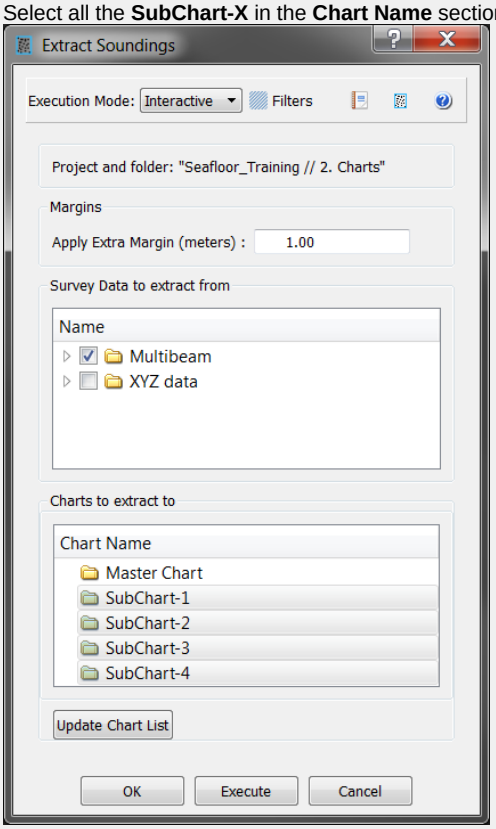
It is also possible to generate charts along a line, by drawing a rectangle, by specifying a certain size or by specifying coordinates.

After the you have created the charts the folders will only contain a frame, thus we need to extract the multibeam soundings and populate them into the frames.

Exercise

Populate charts with soundings

1. Click **Extract Soundings** on the **Chart commands** tab in the **Seafloor Main Menu**
2. In **Margins** type in **5** m to avoid any loss of data between the charts. (Dependent on the gridding increments.)
3. Tick the **Multibeam** checkbox.
4. Select all the **SubChart-X** in the **Chart Name** section.



5. Click **Execute** and the extraction process will start.
6. Click **OK** in the dialog the pops up.
7. Click **OK** on the **Extract Soundings** menu.

Each sub-chart will now contain a dataset named **Soundings** in addition to the **Frame** dataset

Chart properties

Exercise

Show properties for each chart

- Click **Show Chart Properties** on the **Chart commands** tab in the **Seafloor Main Menu**.

Observe that the list contains a row for each chart. Take a closer look at the different columns.

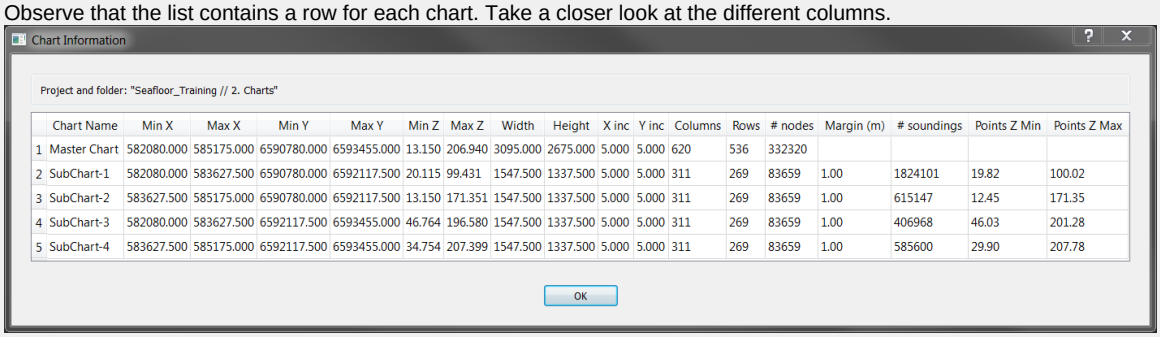


	Chart Name	Min X	Max X	Min Y	Max Y	Min Z	Max Z	Width	Height	X inc	Y inc	Columns	Rows	# nodes	Margin (m)	# soundings	Points Z Min	Points Z Max
1	Master Chart	582080.000	585175.000	6590780.000	6593455.000	13.150	206.940	3095.000	2675.000	5.000	5.000	620	536	332320				
2	SubChart-1	582080.000	583627.500	6590780.000	6592117.500	20.115	99.431	1547.500	1337.500	5.000	5.000	311	269	83659	1.00	1824101	19.82	100.02
3	SubChart-2	583627.500	585175.000	6590780.000	6592117.500	13.150	171.351	1547.500	1337.500	5.000	5.000	311	269	83659	1.00	615147	12.45	171.35
4	SubChart-3	582080.000	583627.500	6592117.500	6593455.000	46.764	196.580	1547.500	1337.500	5.000	5.000	311	269	83659	1.00	406968	46.03	201.28
5	SubChart-4	583627.500	585175.000	6592117.500	6593455.000	34.754	207.399	1547.500	1337.500	5.000	5.000	311	269	83659	1.00	585600	29.90	207.78

Showing Chart Properties

Displaying chart data

Exercise

Display chart frames

1. Click **Display Chart Data** on the **Chart commands** tab in the **Seafloor Main Menu**.
2. Make sure that only **Frame** is checked in **Select Data Object**
3. Select all the sub charts.
4. Make sure that **Scale to selected charts** is checked.
5. Click **Execute**
6. Check the **Chart Name** box to see the name of the charts.

Exercise

Display chart soundings

1. Open the **Display Chart Data** menu again
2. Check the following boxes; **Frame**, **Soundings**, **Use colour range**, **Use same range for all** and **Use LOD**
3. Select all the sub charts
4. Click **Execute**

H. Gridding Files and Charts

Introduction

Gridding can be performed directly on processed multibeam data or on separate charts. Seafloor offers several gridding algorithms, filtering and smoothing. Filtered points can be saved in the project.

Exercises

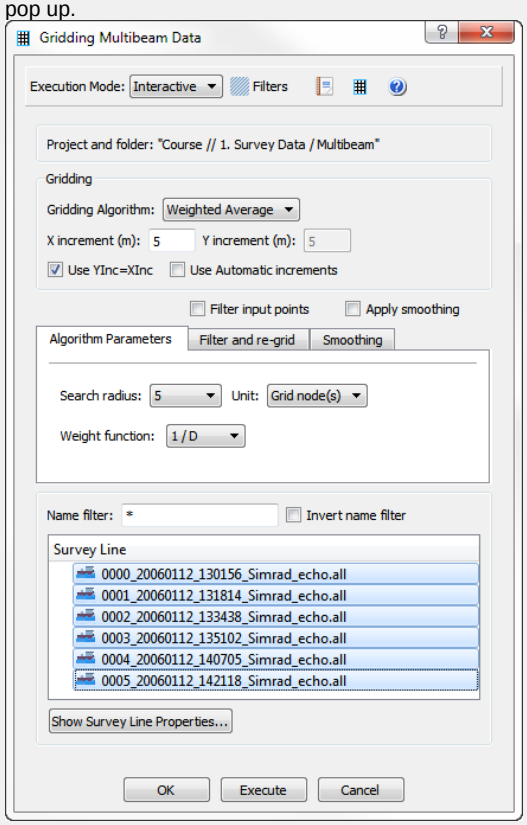
- Gridding multibeam data
- Displaying the result grid
- Gridding charts
- Display Chart grids
- Display rejected soundings
- Merge Chart surfaces

Multibeam

Exercise

Gridding multibeam data

1. Right click the **Multibeam** folder under **1.Survey Data**, select **Gridding Multibeam Data...** and the gridding menu will pop up.



Multibeam gridding menu

2. Keep **Weighted Average** as the gridding algorithm.
3. Change **X increment** to **3** and keep the **Use YInc=XInc** option.
4. In the **Algorithm Parameters** tab set **Search radius** to **5** and change the **Unit** to **Metric**.
5. Keep the **Weight function** as **1/D**
6. Select all the survey lines (click one and use ctrl + a).
7. Click **Execute** and the gridding process will start.
8. When the gridding has finished a report window will pop up, showing the details of the gridding process
9. Click **OK**

The grid is saved as **Surface_WeightedAverage** in the **1. Survey Data / Multibeam** folder

Exercise

Displaying the result grid

1. Right-click the **Surface_WeightedAverage** grid in the **1. Survey Data / Multibeam** folder and select **Zoom to Data**
2. Right-click the **Surface_WeightedAverage** grid again and select **LOD Grid Display**

Charts

Exercise

Gridding charts

1. Select the **Chart commands** tab in the **Seafloor Main Menu**
2. Click **Gridding Chart**
3. Keep **Weighted Average** as the gridding algorithm.
4. Change **X increment** to **3** and keep the **Use YInc=XInc** option.
5. In the **Algorithm Parameters** tab set **Search radius** to **5** and change the **Unit** to **Metric**.
6. Keep the **Weight function** as **1/D**
7. Check the **Filter input points** box
8. Click the **Filter and re-grid** tab
9. Check the **Save filtered points** box, set **Filter type** to **Standard deviation** and set **Shallow-** and **Deep factor** to **3**.
10. Select all the **SubCharts-X** (click first and use ctrl + click on last).
11. Click **Execute** and the gridding process will start.

When the gridding has finished, each sub chart folder will have four new datasets: **Seafloor**, **Soundings**, **Soundings_Accepted** and **Soundings_Rejected**.

Exercise

Display Chart grids

1. In the **Chart commands** tab in the **Seafloor Main Menu**, click **Display Chart Data**
2. Make sure **Frame** box is checked
3. Check the **Seafloor** box and make sure **Use same range for all** and **Use LOD** is checked.
4. Make sure the **Scale to selected charts** is checked.
5. Select all the **SubCharts-X** (click first and use ctrl + click on last).
6. Click **Execute**

Exercise

Display rejected soundings

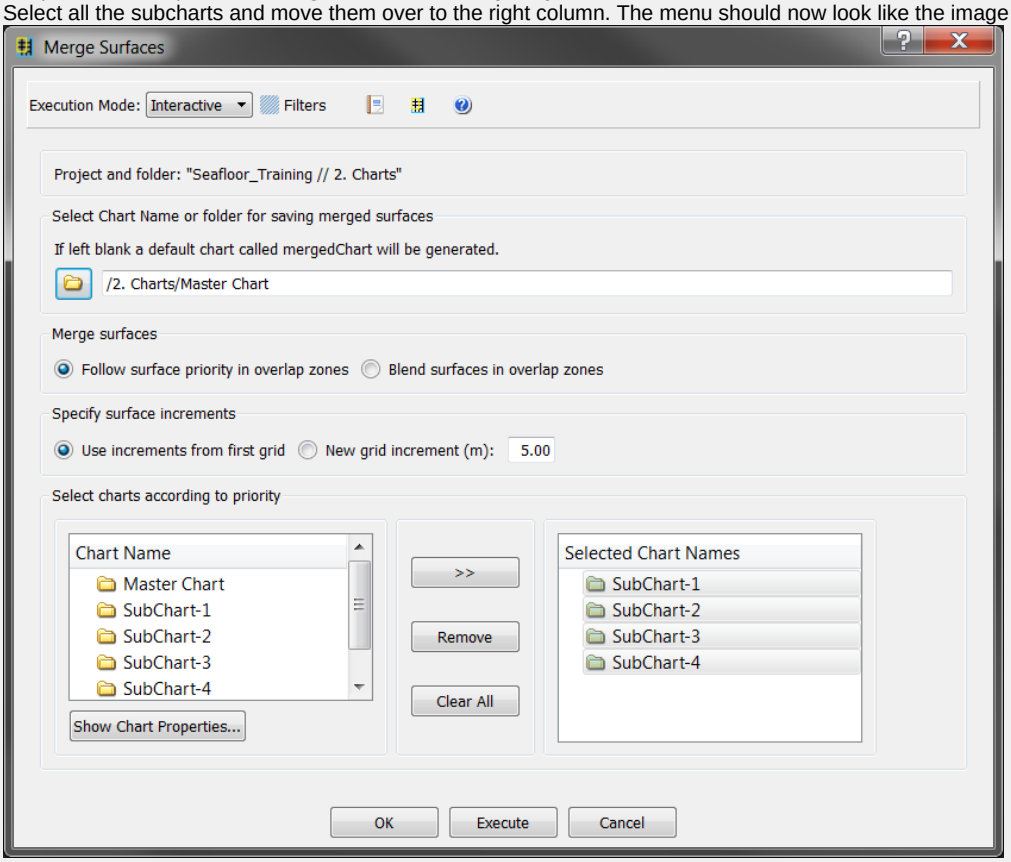
1. Click the  icon on the main toolbar to remove all graphics
2. Go to the folder **SubChart-1** in the project
3. Right click the **Seafloor** dataset and select **Zoom to Data**
4. Right click the **Seafloor** dataset again and select **LOD Grid Display**
5. Right click **Sounding_Rejected** and select **Display Points**

You will now see the points that were filtered away with the standard deviation filter during the gridding process.

Exercise

Merge Chart surfaces

1. Right-click the **2. Charts** folder and select **Merge Surfaces**
2. Click the  button and select the **2. Charts / Master Chart** folder to save the merged surface in.
3. Keep teh default options for **Mergfe surfaces** and **Specify surface increments**
4. Select all the subcharts and move them over to the right column. The menu should now look like the image below:



5. Click **Execute** and a dialog will pop up asking if you want to merge 4 charts.
6. Click **Yes**

A report window will pop up when the merging has been performed. The merged surface will be saved in the **2. Charts / Master Chart** folder.

I. Check Calibration of Survey Lines

Introduction

Geocap Seafloor can compare data from different datasets by viewing the data in a cross section view. By changing the visualisation attributes it is possible to identify problems in the data. In this exercise the data will be visualised in both 3D and 2D.

Exercises

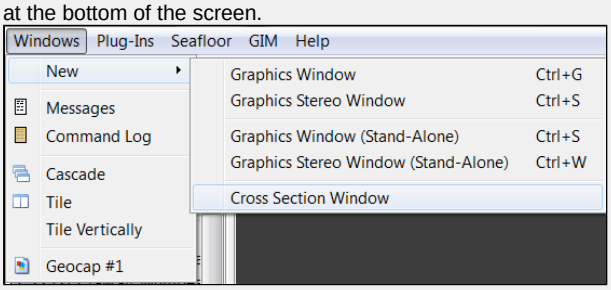
- Compare two overlapping survey lines against each other
- Edit the color and size of points in the cross section window
- Edit the size, direction and position of the profile

Exercise

Compare two overlapping survey lines against each other

Make sure you have displayed two overlapping survey lines in the graphics window, using the same method as in one of the previous exercises. For instance, display the two lines *0000_20060112_130156_Simrad_echo.all* and *0001_20060112_131814_Simrad_echo.all*.

1. In the Geocap main window, click **Windows > New > Cross Section Window**. The cross section window will appear at the bottom of the screen.



2. In the cross section window click the arrow to right of the  icon, and open the menu.
3. Click the  **Digitize cross section** entry in menu.
4. Create a profile over the overlapping area, by digitizing two points in the graphics window using your left mouse button.

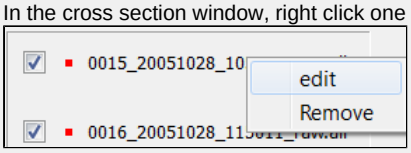
You should now see the multibeam soundings along the profile in the cross section window. You can zoom using your right mouse button, use the left mouse button to pan and the mouse wheel to scale the z-direction.

Points at a certain distance on both sides of the cross section are projected on to the cross section window. You can decide this distance by editing the **Point projection distance**.

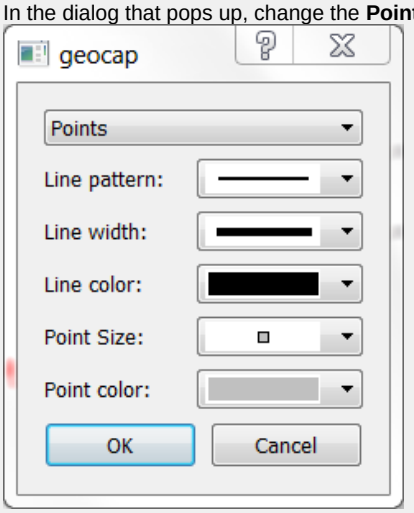
Exercise

Edit the color and size of points in the cross section window

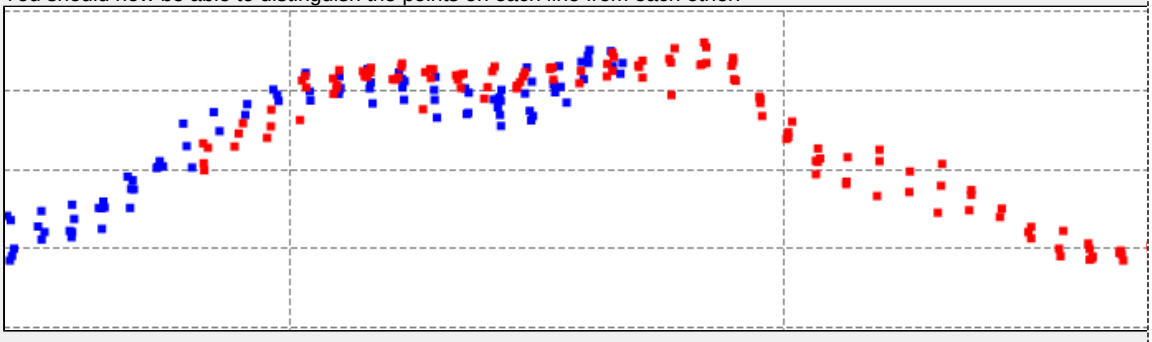
1. In the cross section window, right click one the name of one of the survey line and select **Edit**.



2. In the dialog that pops up, change the **Point color** to another color than the default.



3. You should now be able to distinguish the points on each line from each other.

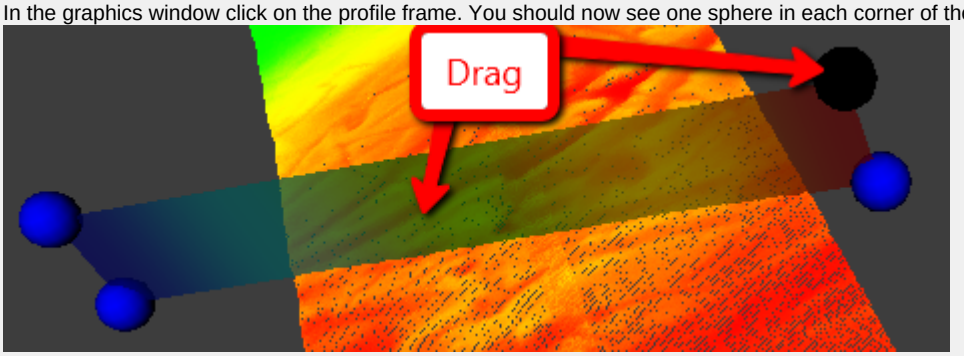


Try out different display options in the **edit** dialog.

Exercise

Edit the size, direction and position of the profile

1. In the graphics window click on the profile frame. You should now see one sphere in each corner of the profile.



2. Click and drag one of the spheres to re-size or change the angle of the profile.
3. Click in the middle of the profile and drag in order to move the whole profile.

J. Vessel Configuration Manager

Introduction

Multibeam data are imported by the command **Import Multibeam Sonar Data**. As a part of the import the make and vendor of the multibeam echosounder is registered. Vital information like the mount parametres are read from the datafiles and will be saved in a Vessel configuration file in the project. Multibeam files with different installation parameters will have reference to different vessel configuration setup.

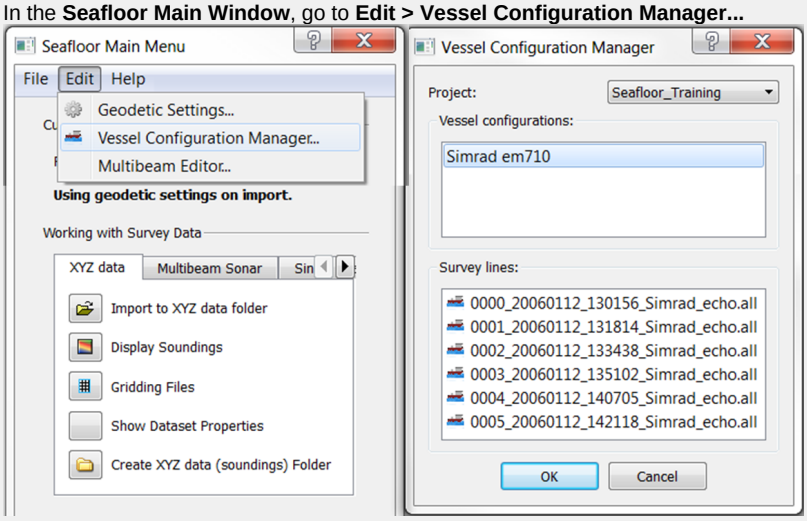
Exercises

- Open the Vessel Configuration Manager
- Check all parameters
- Change Roll parameters

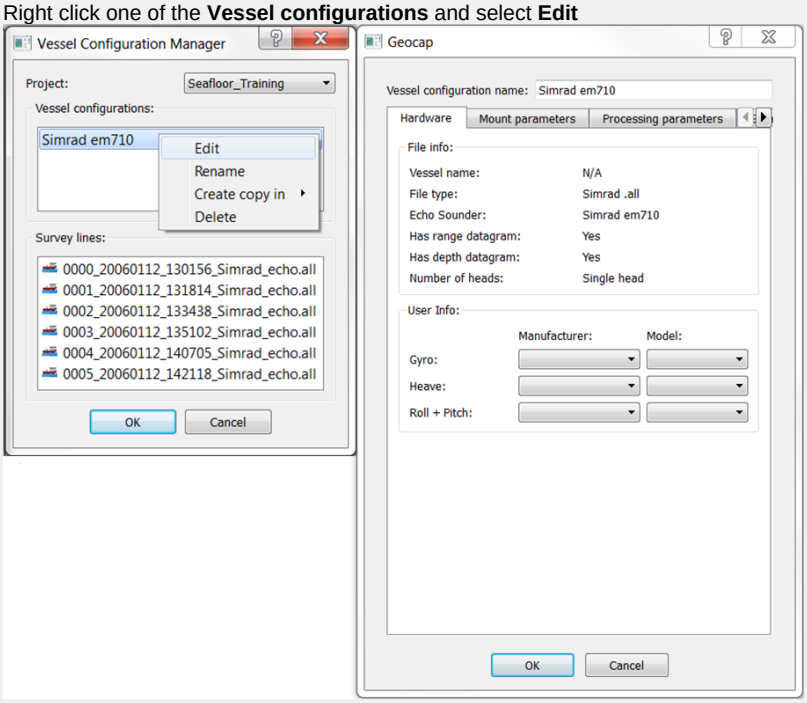
Exercise

Open the Vessel Configuration Manager

1. In the **Seafloor Main Window**, go to **Edit > Vessel Configuration Manager...**



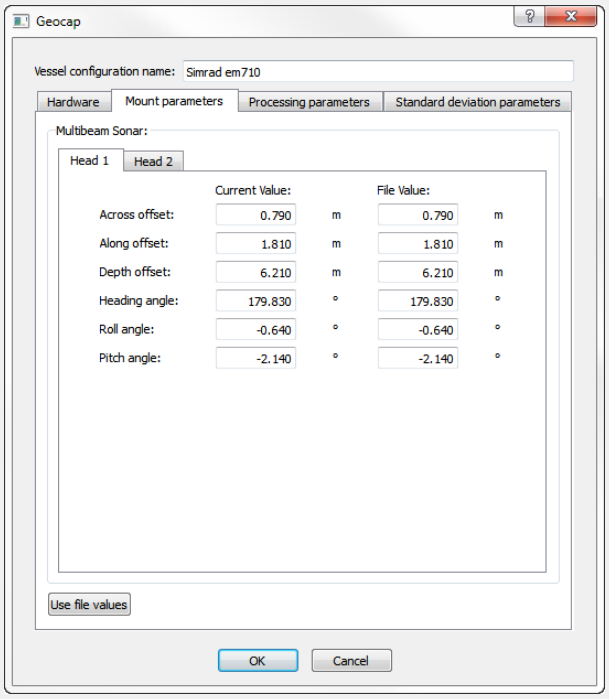
2. Right click one of the **Vessel configurations** and select **Edit**



Exercise

Check all parameters

- Click the **Mount parameters** tab to see the information.
- Click the **Processing parameters** tab to see the information.
- Click the **Standard deviation parameters** tab to see the information.



Exercise

Change Roll parameters

1. Click the **Mount parameters** tab.
2. See the Roll angle value, **-0.640**.
3. Change the value to **-2.640**.
4. Click **OK**
5. Also click **OK** in the Vessel Configuration Manager dialog.
6. Redisplay the survey lines *0000_20060112_130156_Simrad_echo.all* and *0001_20060112_131814_Simrad_echo.all*.
7. Verify the changes in the roll.
8. Again, open the Vessel Configuration Manager dialog.
9. Click the **Mount parameters** tab.
10. Click the **Use file values** button and verify that the Roll angle is reset back to **-0.640**.
11. Click **OK**
12. Also click **OK** in the Vessel Configuration Manager dialog.
13. Redisplay the survey lines *0000_20060112_130156_Simrad_echo.all* and *0001_20060112_131814_Simrad_echo.all*.

K. Import Sensor data

Introduction

Seafloor can import and replace the sensor data of the selected multibeam survey lines from ASCII formats.
Sensor types which can be import replaced:

- **Navigation**
- **Tide**
- **Velocity**
- **Heave**
- **Roll**
- **Pitch**

Exercises

- Check available ASCII formats
- Import Tide file
- Import Navigation file

Exercise

Check available ASCII formats

1. Right click the **Multibeam** folder under **1. Survey Data** and select **Import ASCII Sensor...**
2. Click the icon and open the help menu.
3. Check the available **Sensor type**.
4. Click **OK**

Exercise

Import Tide file

1. Right click the **Multibeam** folder under **1. Survey Data** and select **Import ASCII Sensor...**
2. Click the icon and browse for the tide file.
3. Make sure that **Tide** is selected in under **Sensor type**.
4. Select one or several of the Survey lines.
5. Click **Execute**

Exercise

Import Navigation file

1. Right click the **Multibeam** folder under **1. Survey Data** and select **Import ASCII Sensor...**
2. Click the icon and browse for the navigation file.
3. Make sure that **Navigation** is selected in under **Sensor type**.
4. Select one or several of the Survey lines.
5. Click **Execute**

L. Navigation Processing and Editing

Introduction

Seafloor enables you to perform both automatic processing and manual editing of navigation data. The automatic navigation processing offers two methods:

1. A spike filter which will remove points which seems to have a speed faster than a certain value.
2. Moving average filter which smooths data according to a given "point window" length.

Manual editing is possible by using the **Table View** menu, which is a general menu who combines data tables, 2D view and 3D view.

The delete methods will actually remove the points from the navigation line, while the smoothing operation will save the original positions, which means that you can return to the unsmoothed version at a later stage.

Exercises

- Automatic navigation processing
- Manual editing of navigation data

Exercise

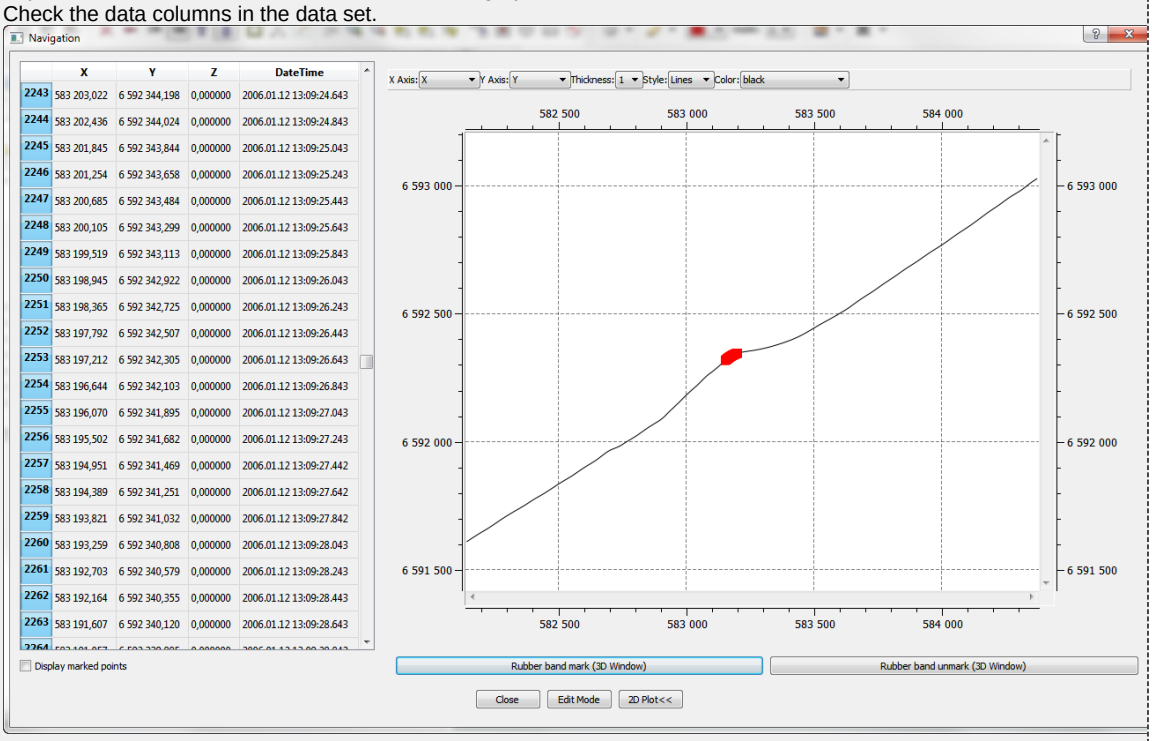
Automatic navigation processing

1. Right click the **Multibeam** folder inside **1. Survey Data** and select **Navigation Processing**.
   Unable to render embedded object: File (Navigation_Processing_menu.png) not found.
2. In the Navigation Processing dialog check the **Smooth** box
3. Set Window length to **5**, and iterations to **1**.
4. Check that the smoothing mode is **Apply**.
5. Select one or several of the survey lines.
6. Click **Execute**

Exercise

Manual editing of navigation data

1. In the project click on the small triangle next to a Multibeam Survey dataset to expand all its elements.
2. Right click **Navigation** under the Multibeam Survey dataset and select **Table View**.
3. Right click the **Navigation** data object and select **Display**.
4. In the **Navigation** table view menu, click the **2D Plot>>** button.
5. Check the **Display marked points** box.
6. Expand the window to better see the table and the graphics.
7. Check the data columns in the data set.



	X	Y	Z	DateTime
2243	583 203,022	6 592 344,198	0,000000	2006.01.12 13:09:24.643
2244	583 202,436	6 592 344,024	0,000000	2006.01.12 13:09:24.843
2245	583 201,845	6 592 343,844	0,000000	2006.01.12 13:09:25.043
2246	583 201,254	6 592 343,658	0,000000	2006.01.12 13:09:25.243
2247	583 200,685	6 592 343,484	0,000000	2006.01.12 13:09:25.443
2248	583 200,105	6 592 343,299	0,000000	2006.01.12 13:09:25.643
2249	583 199,519	6 592 343,113	0,000000	2006.01.12 13:09:25.843
2250	583 198,945	6 592 342,922	0,000000	2006.01.12 13:09:26.043
2251	583 198,365	6 592 342,725	0,000000	2006.01.12 13:09:26.243
2252	583 197,792	6 592 342,507	0,000000	2006.01.12 13:09:26.443
2253	583 197,212	6 592 342,305	0,000000	2006.01.12 13:09:26.643
2254	583 196,644	6 592 342,103	0,000000	2006.01.12 13:09:26.843
2255	583 196,070	6 592 341,895	0,000000	2006.01.12 13:09:27.043
2256	583 195,502	6 592 341,682	0,000000	2006.01.12 13:09:27.243
2257	583 194,951	6 592 341,469	0,000000	2006.01.12 13:09:27.442
2258	583 194,389	6 592 341,251	0,000000	2006.01.12 13:09:27.642
2259	583 193,821	6 592 341,032	0,000000	2006.01.12 13:09:27.842
2260	583 193,259	6 592 340,808	0,000000	2006.01.12 13:09:28.043
2261	583 192,703	6 592 340,579	0,000000	2006.01.12 13:09:28.243
2262	583 192,164	6 592 340,355	0,000000	2006.01.12 13:09:28.443
2263	583 191,607	6 592 340,120	0,000000	2006.01.12 13:09:28.643

8. In the graphics window, hold the **Shift** button and click and drag the rubber band around the points you want to investigate.
9. Notice that the selected points are displayed both in the table and in the 3D graphics window.
10. Use the buttons **Rubber band mark (3D window)** and **Rubber band unmark (3D window)** to select and de-select points.
11. To enable for deleting points you must click the **Edit Mode** button.
12. Click **Close** to close the menu.

M. Multibeam Depth Filtering

Introduction

Seafloor enables you to perform both automatic depth filtering and manual editing of sonar data. The automatic depth filtering offers three filters:

1. A threshold filter which will remove all points shallower or deeper than the threshold.
2. An angle filter which measure the angle between neighbour points on the same ping and eliminate those exceeding a certain value.
3. A surface filter which can calculate the standard deviation for all points and remove points outside, e.g. 2 times the standard deviation.

Note that none of the methods will actually delete the points. They will only be flagged as deleted, which means you can go back to the original, unfiltered version at a later stage.

Exercises

- Automatic Threshold filtering
- View the result from the automatic Threshold filtering
- Automatic Surface filtering
- View the result from the automatic Surface filtering
- Revert flagged points from automatic filtering
- Manually edit a single multibeam survey line
- Display removed points and undelete them again
- Manually edit multiple multibeam survey lines

Automatic Filtering

Exercise

Automatic Threshold filtering

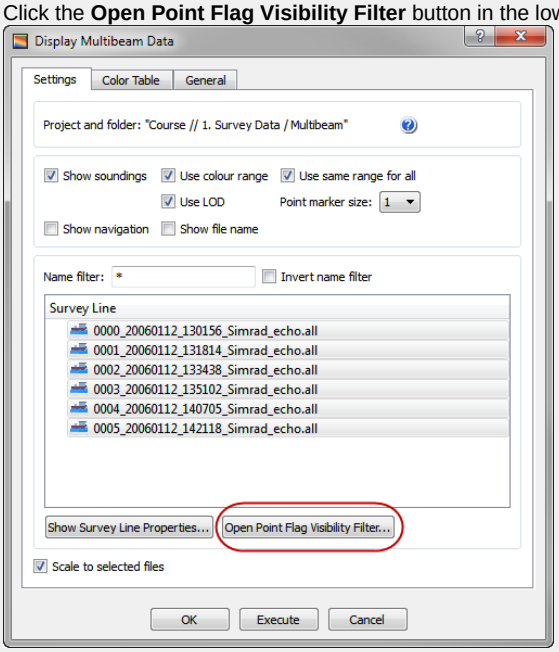
1. Right click the **Multibeam** folder inside **1. Survey Data** and select **Multibeam Depth Filtering**
2. In the menu that pops up, check the **Treshold filter** box
3. To find the min and max, click the **Show Survey line properties...** button.
4. Check both **Remove shallower than depth** and **Remove deeper than depth** and type in the min and max value.
5. Leave the **Angle filter** box unchecked for now.
6. Leave the **Surface filter** box unchecked for now.
7. Select one or several of the survey lines.
8. Click **Execute**

In the report that appears you can see how many points that were flagged in each multibeam file.

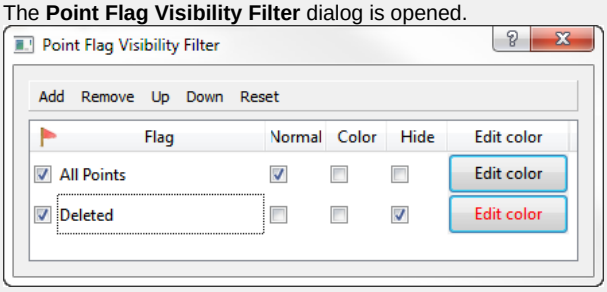
Exercise

View the result from the automatic Threshold filtering

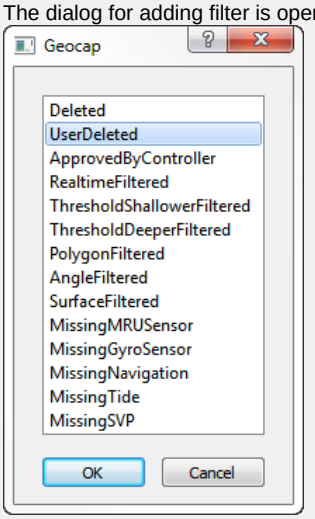
1. Open the **Display Multibeam Data** dialog.
2. Display (some of) the lines which have been filtered.
3. Click the **Open Point Flag Visibility Filter** button in the lower part of the window.



4. The **Point Flag Visibility Filter** dialog is opened.



5. Click the **Add** button on the menu bar.
6. The dialog for adding filter is opened.



7. Select, for instance, the **ThresholdShallowerFiltered** entry and click **OK**.
8. Repeat clicking the **Add** button. This time, select the **ThresholdDeeperFiltered** entry and click **OK**.
9. In the **Point Flag Visibility Filter** dialog, tick the **Color** checkbox for both filters.
10. Click the **Edit Color** button and set individual colors for the filters.

The Threshold filtered points are now displayed together with the accepted points.

To remove the accepted points, tick the **Hide** checkbox for **All Points**.

Exercise

Automatic Surface filtering

1. Right-click the **Multibeam** folder inside **1. Survey Data** and select **Multibeam Depth Filtering**
2. In the menu that pops up, check the **Surface filter** box
3. Click the  icon and browse in the surface you generated in the Gridding chapter.
4. Select **Standard Deviation** as **Filter type**
5. Select a **Shallow factor** and a **Deep factor**
6. Select one or several of the survey lines.
7. Click **Execute**

Exercise

View the result from the automatic Surface filtering

To view the result from the automatic Surface filtering, the method is the same as for Threshold filtering.

1. Open the **Display Multibeam Data** dialog.
2. Display (some of) the lines which have been filtered.
3. Click the **Open Point Flag Visibility Filter** button in the lower part of the window.
4. Click the **Add** button on the **Point Flag Visibility Filter** dialog menu bar, and select the **SurfaceFiltered**.
5. In the **Point Flag Visibility Filter** dialog, tick the **Hilight** checkbox for **SurfaceFiltered** (and also tick **Hide** for the threshold filters).

The points that are displayed are the ones that got flagged by the surface filter.

Exercise

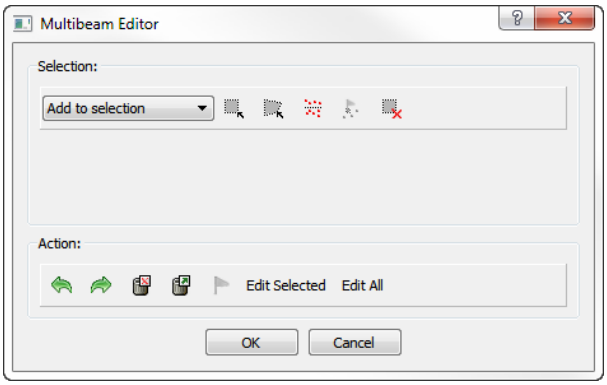
Revert flagged points from automatic filtering

1. Right click the **Multibeam** folder in your project and select **Revert flagged points**
2. Check the flags you want to revert
3. Select the **Survey Lines** you want to revert from
4. Click **Execute**
5. Click **Yes** when asked about reverting flags for 6 files

After the processed has finished you are presented with a report telling you how many points were reverted for each flag for each survey line.

Manual Editing

The **Multibeam Editor** lets you manually edit points in both 3D and 2D. Removed points will be flagged as deleted, but can later be retrieved.



The Multibeam Editor menu

The menu has two parts. A **Selection** part and an **Action** part.

Selection
You have to select a set of points before you can perform an action on them. There are four different ways to select points:

- Rubber band select (shortcut key **R**) - Click and drag a rectangle
- Lasso select (shortcut key **L**) - Digitize a polygon (double click to close)
- Threshold select - Select a **Deeper** or **Shallower** value.
- Flag select - Select by flag type

> Note that by clicking **Rubber band select** (**R**) or **Lasso select** (**L**) the 3D window will be locked until you have finished the selection. This means that you will not be able to rotate or pan during the selection process.

After selection the points will be visualized in white.

> It is possible to deselect parts of a selection by changing the drop down list to **Remove from selection** and then perform another selection within the previously selected points.
> You can also use the **Clear selection** icon to remove the selection.

Action
After a set of points have been selected it is possible to perform several actions on them:

- Delete them (shortcut key **Delete**) - The points are only flagged as deleted
- Undelete them
- Flag them according to a userndefined flag
- Edit on the selected points

By clicking **Edit Selected** it is possible to make a new selection only within the previous selection. This makes it easier to narrow in on points/noise you want to delete.

The green arrows allows you to redo or undo previous step, while **Edit All** will allow you to make all the points available for editing.

Exercise

Manually edit a single multibeam survey line

1. In the **Multibeam** folder, right click on of the Multibeam Survey datasets and select **LOD Points Display**
2. Click on the small triangle next to the Multibeam Survey dataset to expand all its elements.
3. Right click **Multibeam Sonar** under the Multibeam Survey dataset and select **Edit**
4. Click the **Rubberband Select** icon
5. In the graphics window, click and drag the rubber band around an area where there are spikes you want to remove.
6. Notice that the selected points are displayed in white.
7. Click **Delete** and notice that the points are removed.
8. Continue to select and flag points until you have removed all the spikes.
9. Click **OK**

Exercise

Display removed points and undelete them again

1. On the **Multibeam Sonar** tab in the **Seafloor Main Menu** click **Display Multibeam Data**.
2. Click **Open Point Flag Visibility Filter**.
3. Click **Add**.
4. Select **UserDeleted** and click **OK**.
5. Tick the **Highlight** box for the **UserDeleted** flag and observe that the points you deleted are now displayed in red.
6. Open the **Multibeam Editor** again.
7. Press **R** on your keyboard and draw a rectangle around all (or some) of the red points that you want to undelete.
8. Click **Undelete** and observe that the points are now display normal again.

Exercise

Manually edit multiple multibeam survey lines

1. Click the  on the main toolbar to clear the screen.
2. On the **Multibeam Sonar** tab in the **Seafloor Main Menu** click **Display Multibeam Data**.
3. Select all survey lines and click **Execute**
4. On the **Seafloor Main Menu** click **Edit** and select **Multibeam Editor**.
5. Click on **Rubberband Select** and create a large selection area around some spikes you want to remove.
6. Click **Edit Selected**. Notice how all points outside the selection are are now transparent while the selection area is displayed in "normal" colors again.
7. Click **Lasso Select** and make sure to select an area containing both transparent and "normal" points. Notice that only the "normal" points are selected.
8. Repeat the last two steps until you have narrowed into the spikes you want to remove.

N. Points Classification

Introduction

Seafloor can automatically classify points into the following categories:

- Shoals
- Deeps
- Ridges
- Saddles
- Valleys

The user can specify the search radius and angle, where applicable. The points will be saved as separate datasets in a **classification** folder under each chart.

> ⚠ This excercise expects that the project already contains at least one chart and a corresponding **Seafloor** surface.

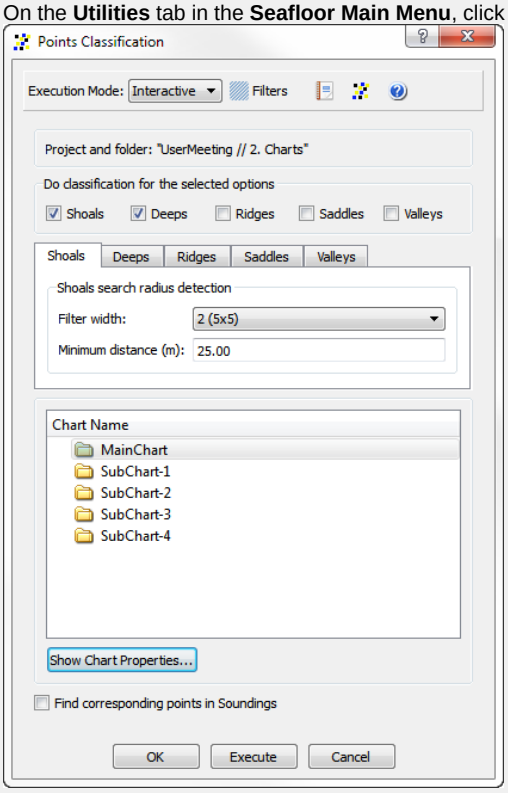
Exercises

- Classify points
- Display classified points

Exercise

Classify points

1. On the **Utilities** tab in the **Seafloor Main Menu**, click **Point Classification**.



2. Check the box for **Shoals** and the box for **Deeps**.
3. Note all the boxes for the different classification types.
4. Specify the search radius, for instance **50**, in the corresponding tabs for the two data types.
5. Select the main chart.
6. Click **Execute**

The resulting datasets can be found in the **classification** folder under the chart.

Exercise

Display classified points

1. Browse to the **classification** folder for the actual chart.
2. Display the dataset **shoals_50** by right click and select **Display Points**
3. Display the dataset **deeps_50** by right click and select **Display Points**

Repeat the last to excercise by changing the search radius. Also try to check the box **Find corresponding points in Soundings**.

O. Export Data

Introduction

Seafloor lets you export Polygons, Points, Surfaces, Hill shades, Contours.

There is a wide range of export possibilities and formats such as, Shapefiles, ASCII-formats, GeoTiff, Irap-formats and VTK.

Exercises

- Export surfaces as Irap ASCII
- Export surfaces as points
- Try different export formats for surface export
- Export all soundings
- Export contour lines to ASCII
- Export contour lines to KML
- Export contour areas
- Export Hill Shade as GeoTiff

Surfaces

Exercise

Export surfaces as Irap ASCII

1. In the **Utilities** tab in the **Seafloor Main Menu**, click **Export Chart Data**
2. Click the **Surfaces** tab
3. Select all the sub-charts
4. Select **Irap ASCII** as the **Export Format**
5. Click the  icon and select a directory to save the files.
6. Leave the **File name** blank. This will cause your files to be named *"Seafloor_SubChart-1.ifgr"* and so on.
7. Click **Execute**

Exercise

Export surfaces as points

1. In the **Utilities** tab in the **Seafloor Main Menu**, click **Export Chart Data**
2. Click the **Points** tab
3. Check the **Seafloor surfaces as points** box
4. Select all the sub-charts
5. Select **ASCII Column** as the **Export Format**
6. Click the  icon and select a directory to save the files.
7. Leave the **File name** blank. This will cause your files to be named *"Seafloor_SubChart-1.asc"* and so on.
8. Click **Execute**

Exercise

Try different export formats for surface export

1. Repeat the previous exercise, but this time use **KML** and/or **Shape** as export format.
2. Try opening the files in another software that can read these files e.g. Google Earth, ArcGIS, QGIS etc.

Note that Google Earth will have problem loading files with large amounts of points. You should try to limit the amount of points to less than 50 000. Right-click a grid and select **Properties** to see the amounts of points in a grid.
To reduce the amount of points you need to create grids with coarser resolution or create smaller sub charts.

Geocap exports KML-files with default settings (yellow pin for points). If you want to change the visual settings for the KML file, you can use text editors like Emacs or Notepad ++. Read more about KML.

.

Soundings

Exercise

Export all soundings

1. In the **Utilities** tab in the **Seafloor Main Menu**, click **Export Chart Data**
2. Click the **Points** tab
3. Check the boxes for **Soundings**, **Soundings_Accepted** and **Soundings_Rejected**
4. Select all the sub-charts
5. Select **ASCII Column** as the **Export Format**
6. Click the  icon and select a directory to save the files.
7. Leave the **File name** blank.
8. Click **Execute**

This will export three datasets for each sub chart.

Contours

Exercise

Export contour lines to ASCII

1. On the **Utilities** tab in the **Seafloor Main Menu**, click **Export Contours**
2. Check that the **Export as: Contour Lines** radio button is ticked.
3. Select the **Master Chart**.
4. Set **Contouring method** to **Incerement**.
5. Set **Increment (m)** to **1.00**.
6. Also consider the parameters **Use min level**, **Use max level**, and **Invert Z**
7. Check the **Save to** option **File and project**.
8. Select **ASCII Column** as **Export Format**.
9. Click the  icon and select a directory to save the files.
10. Enter ***1m*** as a prefix for for the file names.
11. Click **Execute**
12. Verify that the files are saved on the prefered folder on the disk.
13. Verify that the selected chart now have a **Contours** dataset saved in the charts' folder.

Exercise

Export contour lines to KML

1. Repeat the exercise above, but this time select **KML** as the export format.
2. Try to open the exported file in Google Earth

Exercise

Export contour areas

1. On the **Utilities** tab in the **Seafloor Main Menu**, click **Export Contours**
2. Check that the **Export as: Contour Areas** radio button is ticked.
3. Select the **Master Chart**.
4. Set **Contouring method** to **Incerement**.
5. Set **Increment (m)** to **5.00**.
6. Check the **Save to** option **File only**.
7. Select **Shape** as the **Export Format**.
8. Click the  icon and select a directory to save the files.
9. Enter ***5m*** as a prefix for for the file names.
10. Click **Execute**
11. Verify that the files are saved on the prefered folder on the disk.

Hillshade

Exercise

Export Hill Shade as GeoTiff

1. On the **Utilities** tab in the **Seafloor Main Menu**, click **Export Chart Data**
2. Click the **Hill Shade** tab
3. Set **Altitude deg:** to **45**
4. Set **Azimuth deg:** to **SouthWest**
5. Set **Z Scale:** to **10**
6. Select the **Master Chart**
7. Check the **Save to** option **File and project**.
8. Select **GeoTiff** as the **Export Format**.
9. Click the  icon and select a directory to save the files.
10. Enter **45deg_SoutWest_Zx10** as a prefix for for the file name.
11. Click **Execute**
12. Verify that the file is saved on the prefered folder on the disk.
13. Right-click the **HillShade** dataset in the **Master Chart** folder and select **Display**

P. Image Georeferencing

Introduction

This tutorial will guide you through the process of georeferencing a map. To georeference a map or image you need at least 3 tie points. These points should cover as much of the map as possible. In this tutorial we will use a sea chart from the Oslofjord. This chart has some marked positions referenced in WGS84, UTM Zone 33, as tie points.

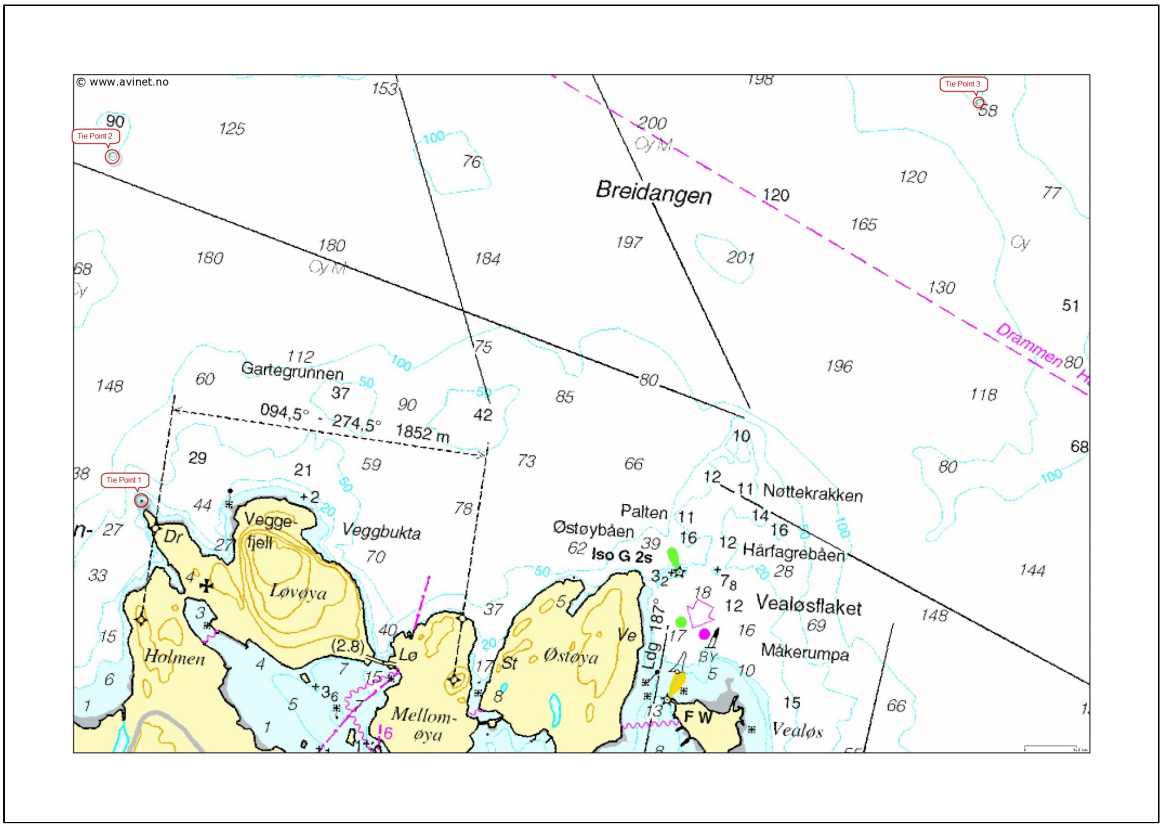
Exercises

- Import an image into your project
- Georeference an image

Exercise

Import an image into your project

1. Create a **Generic** folder (i.e. a folder with schema Generic) and rename that folder to **My Maps**
2. Right-click the **My Maps** folder and go to **Import > Generic**. Browse for the image **sjokart.png** (a *.jpg or a *.png).
3. Make sure the schema of the image set to **Image Data**.



A map about to be georeferenced. It is important to understand what coordinate system and datum the map is in - or make a qualified guess. Note also that you should be able to determine the coordinates of your tie-points, here encircled in red.

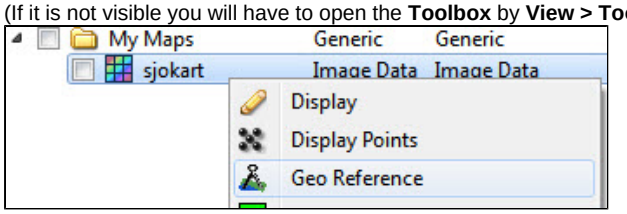
*The tie-points can be find in the text file **coordinates.txt**:*

```
Coordinate system: WGS84, UTM zone 33

Lower left:  East: 241073.89 North: 6599354.87

Upper left:  East: 240904.64 North: 6601351.49

Upper right: East: 245955.69 North: 6601663.55
```

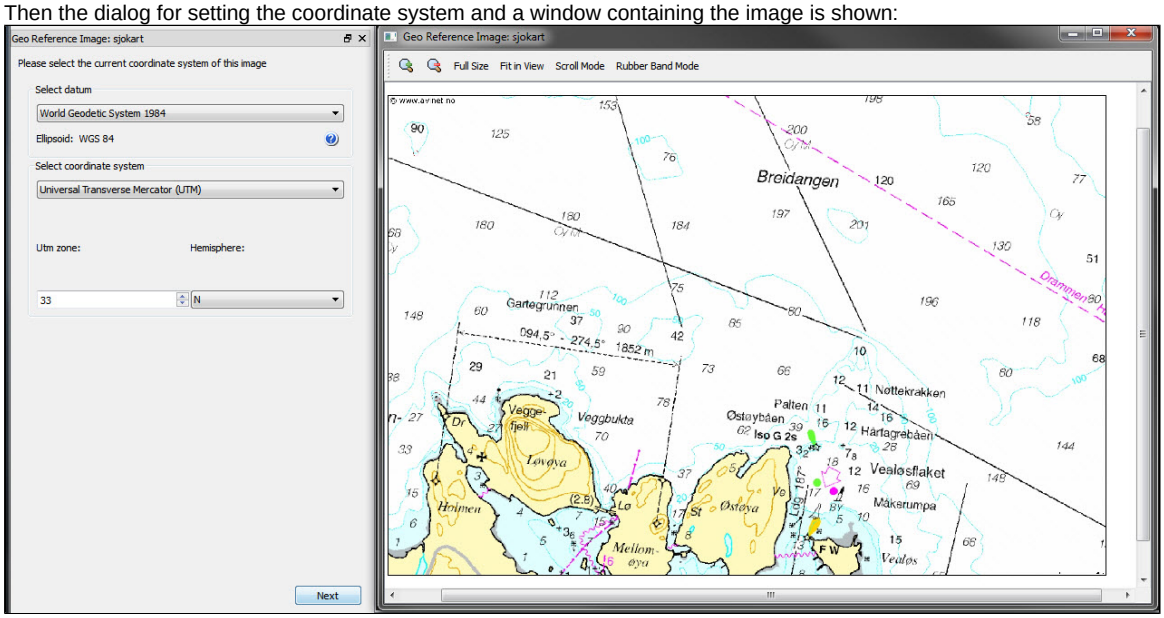
Exercise
Georeference an image Use the Georeferencing wizard to georeference an image. Follow the step-by-step procedure below.

Georeferencing step by step

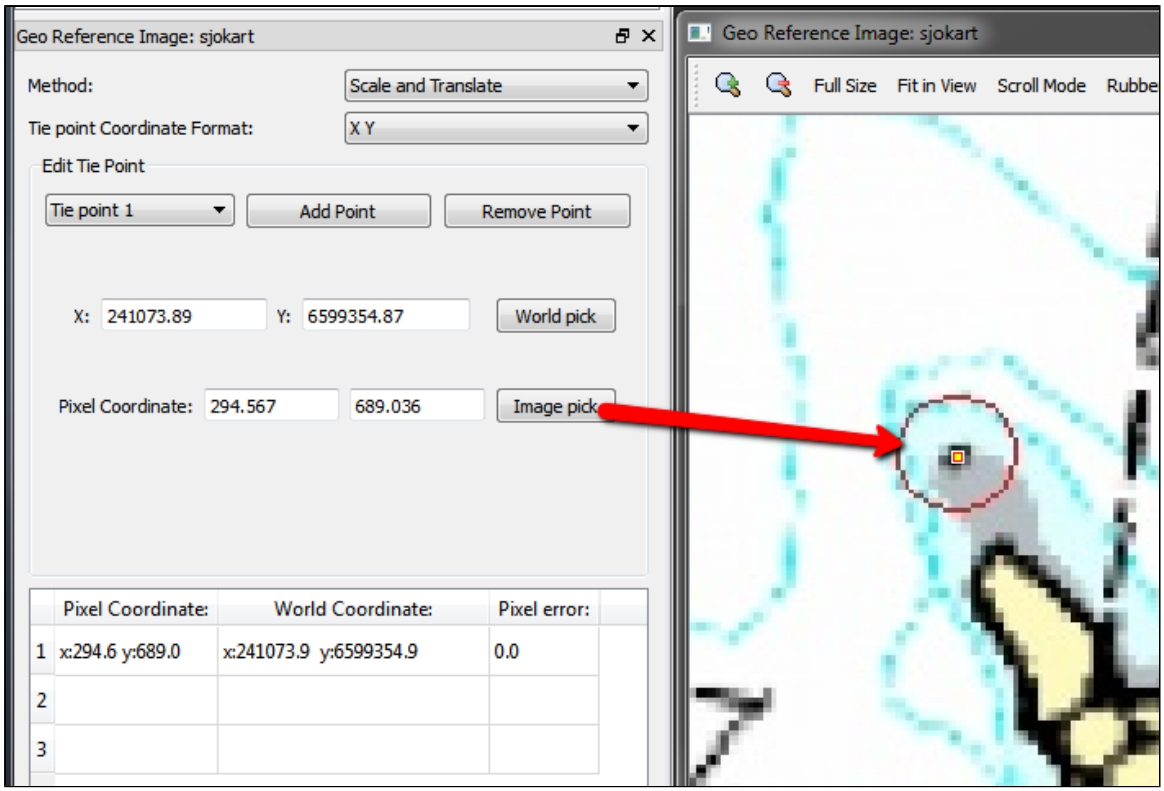
- Make sure the image is selected in the project and right-click and select the **Geo Reference** command in the pop-up. (If it is not visible you will have to open the **Toolbox** by **View > Toolbox**).



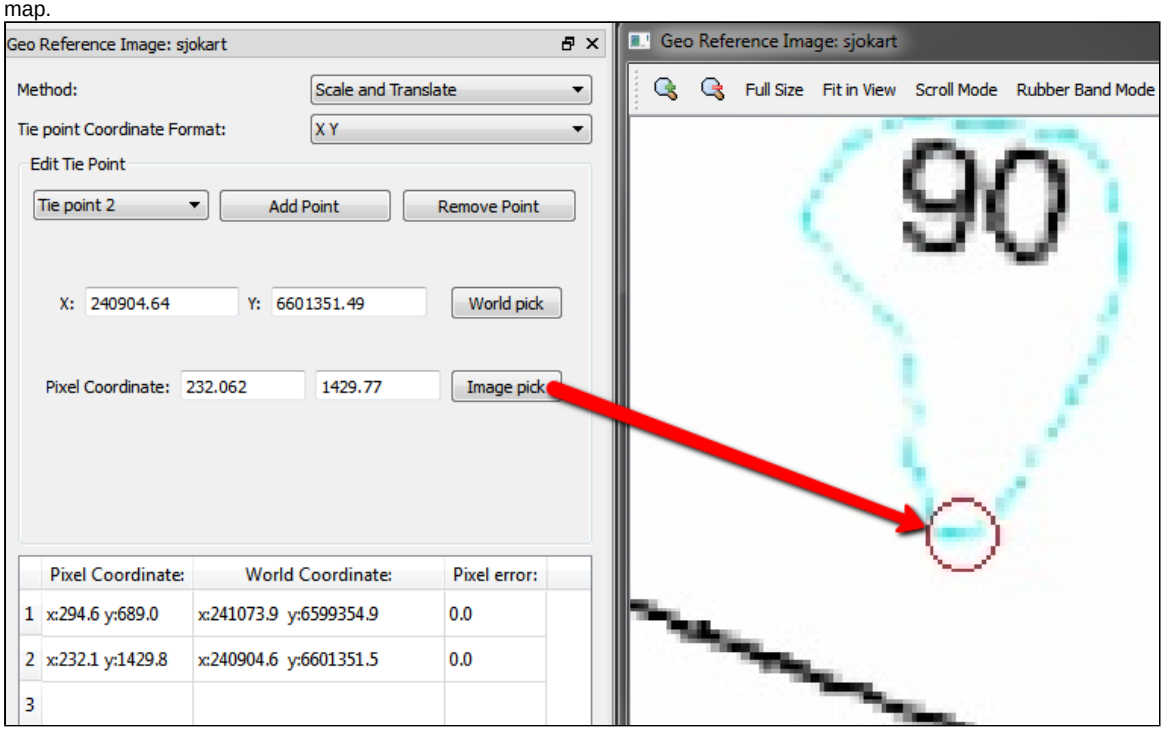
- Then the dialog for setting the coordinate system and a window containing the image is shown:



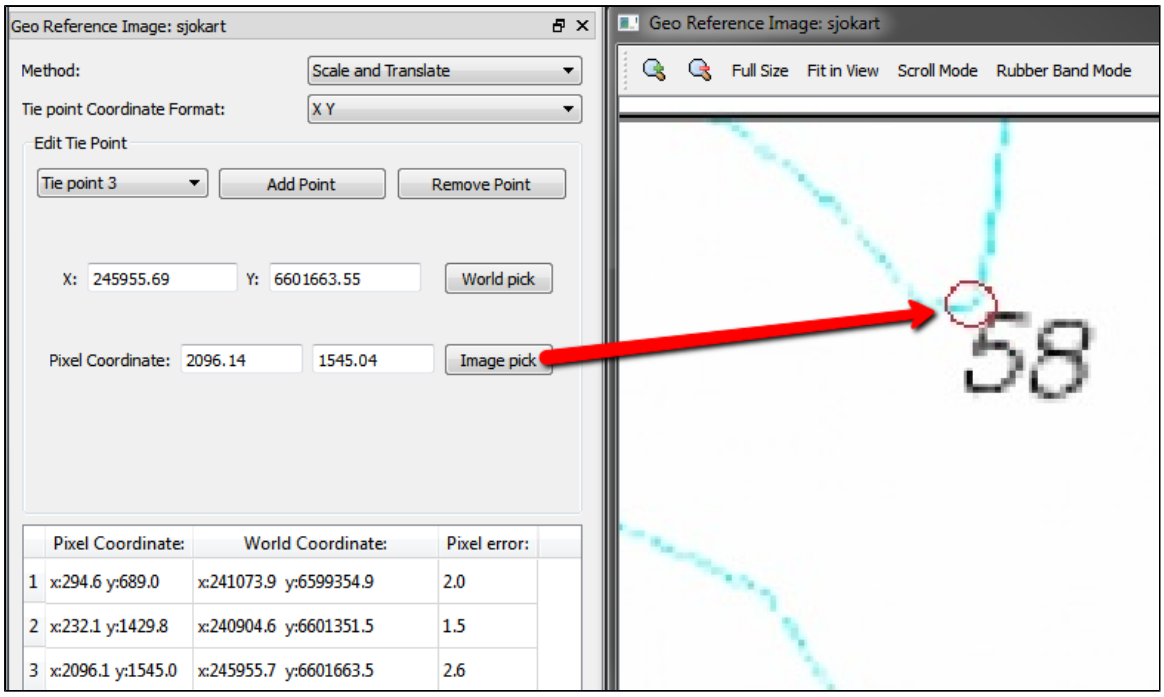
- Set geodetic properties of the chart: **WGS84, UTM Zone 33 N**
- Click **Next**.
- Set **Method:** to **Scale and Translate**.
- Set **Tie point Coordinate Format:** to XY.
- Type in the **X** = **241073.89** and **Y** = **6599354.87** for the first tie point, click **Image pick** and click on the corresponding point in the map.



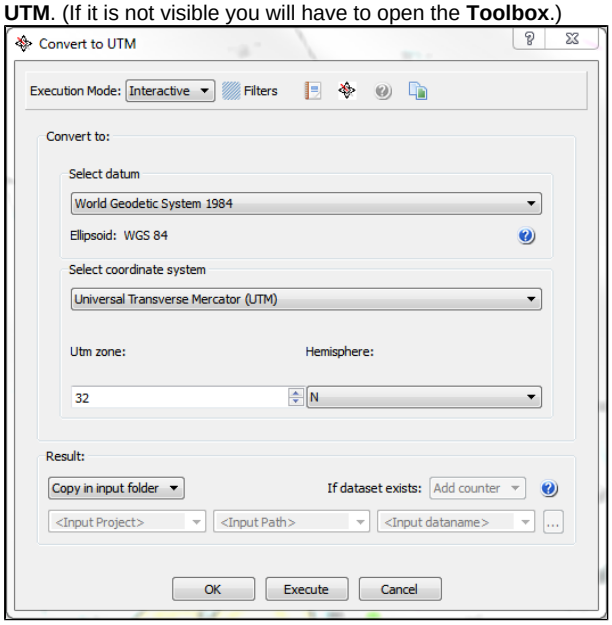
- Change to **Tie point 2**.
- Type in the **X** = **240904.64** and **Y** = **6601351.49** for the second tie point, click **Image pick** and click on the corresponding point in the map.



- Change to **Tie point 3**.
- Type in the **X** = **245955.69** and **Y** = **6601663.55** for the third tie point, click **Image pick** and click on the corresponding point in the map.



- Evaluate the result. The **Pixel error** represents the accuracy of the georeferencing. The value should not exceed more than 1 pixel on each point.
- Click **Finish**
- Now the image must be transformed from *UTM Zone 33* to *UTM Zone 32*.
  From the image **sjokart**, right-click and in the pop-up select **All Commands > Shared Commands > Operations > Convert to UTM**. (If it is not visible you will have to open the **Toolbox**.)



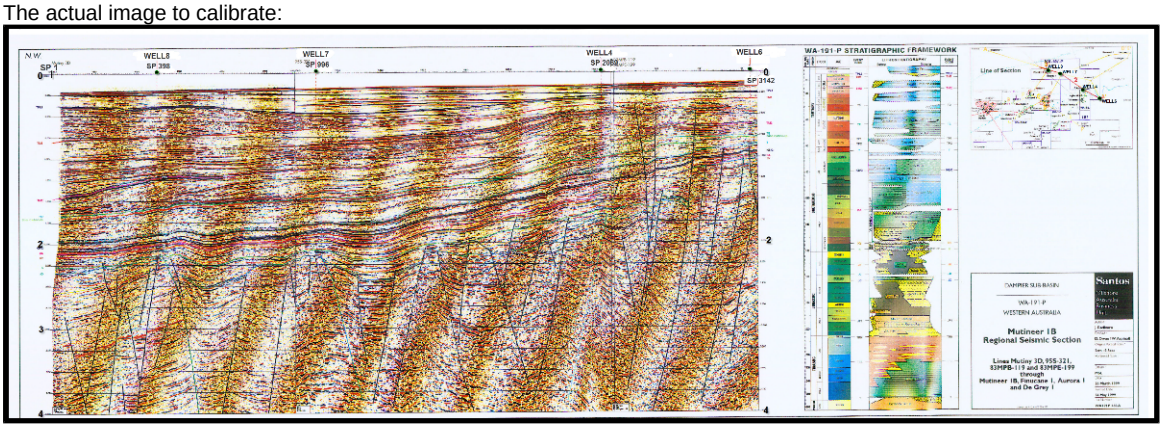
- Set the new geodetic properties for the image: **WGS84, UTM Zone 32 N**
  Specify how to save the new image for e.g. **Replace**, **Copy in input folder** or **Specify** name and where to store it.
  Then click **Execute**.
- Display the new image in the main graphic window, and also display teh multibeam data or the surface created from the multibeam data.

Q. Vertical Image Calibration

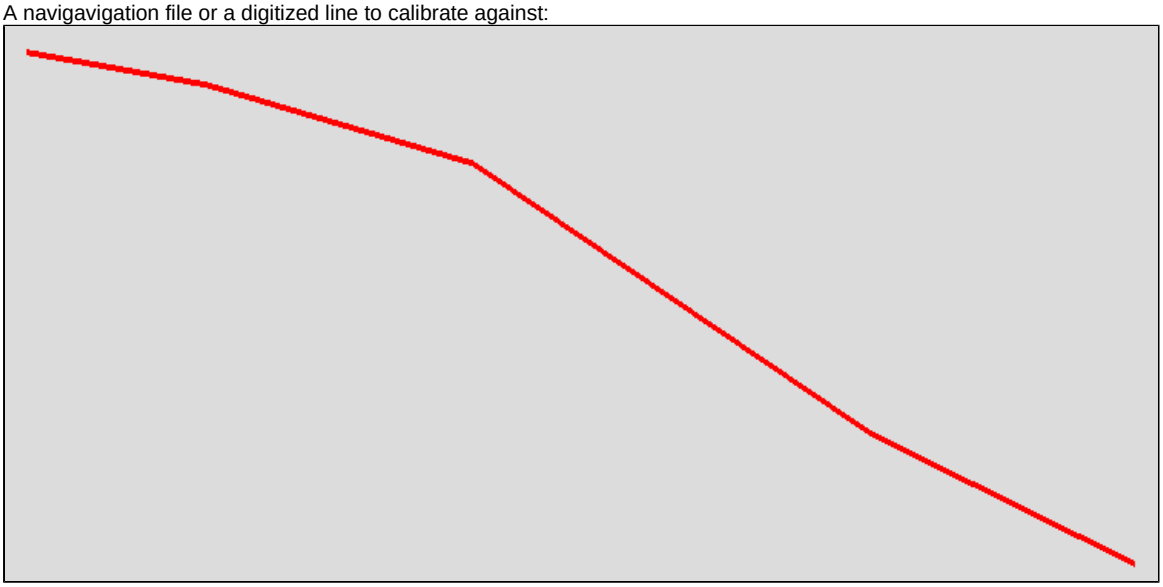
Introduction

This tutorial will guide you through creating a vertical calibrated image (VCI) from an old scanned seismic image. The process can however be performed on any image. All you need are two datasets:

The actual image to calibrate:



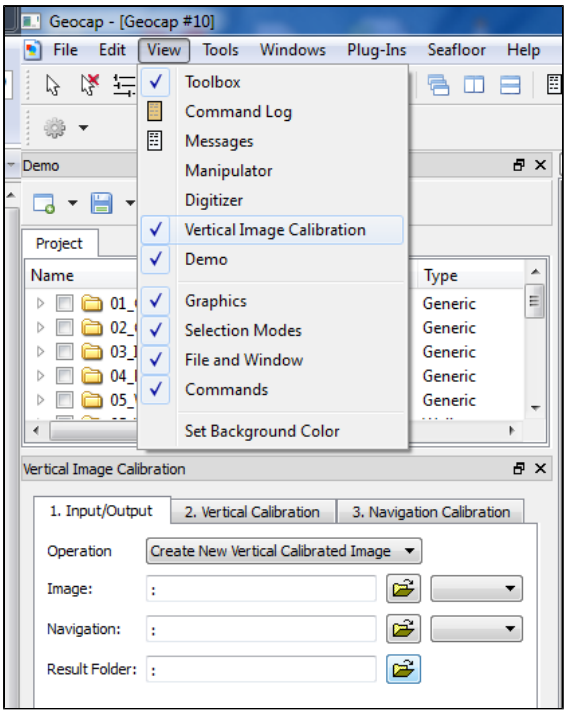
A navigavigation file or a digitized line to calibrate against:



We assume you have already imported the image and navigation (or digitized a navigation line).

Open VCI menu

The VCI menu is opened from **View > Vertical Image Calibration** *(XCal in older Geocap versions)*. The menu should appear in the same panel as your project, like the screen shot below:



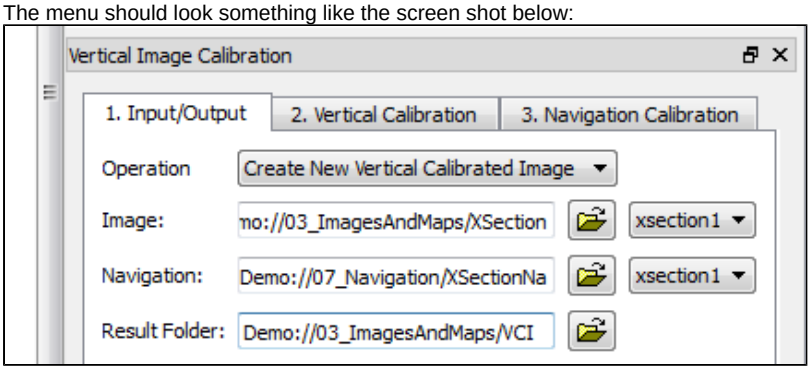
Opening the VCI menu

As you can see this menu consists of 3 steps (tabs) that you need to complete in order to complete the vertical calibration.

Input/Output

1. Click the **Image** browse button and select the **folder** where your image resides.
2. Select the image you want to vertically calibrate in the drop down box.
3. Click the **Navigation** browse button and select the **folder** where your navigation/digitized line resides.
4. Select the navigation/digitized line you want to vertically calibrate against in the drop down box.
5. Click the **Result folder** browse button and select the **folder** where you want your vertically calibrated image to be stored.

The menu should look something like the screen shot below:

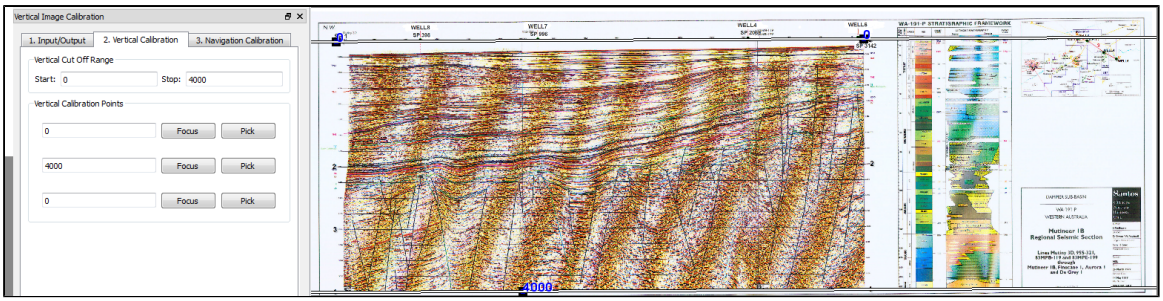


Vertical Calibration

The **vertical calibration** uses a cut off range and 3 calibration points. The cut off range usually starts at 0 and stops at the max depth/time of your seismic image. Everything above or below this cut off range will be eliminated from the completed VCI. The 3 calibration points specify the orientation and scale in vertical direction of the image.

1. Click the **2. Vertical Calibration** tab to start the vertical calibration.
2. Set the **Stop** value in the **Vertical Cut Off Range** to the max time/depth in your image.
3. Set the first calibration point to **0**, click the **Pick** button and click on the corresponding value to the left in your image.
4. Set the second calibration point to the stop value in your cut off range, click the **Pick** button and click on the corresponding value in the middle of your image.
5. Set the third calibration point to **0**, click the **Pick** button and click on the corresponding value to the right in your image.
6. Use the **Focus** buttons to zoom closer to each point and re-pick them for a more accurate selection.

You should now see two white horizontal lines across your image. One at 0 depth and one at max depth.

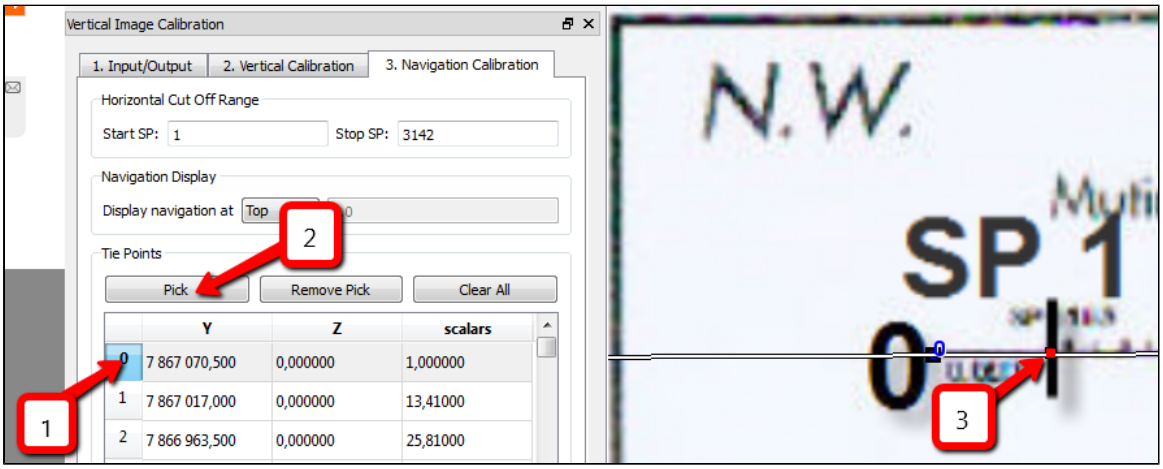


Vertical calibrated

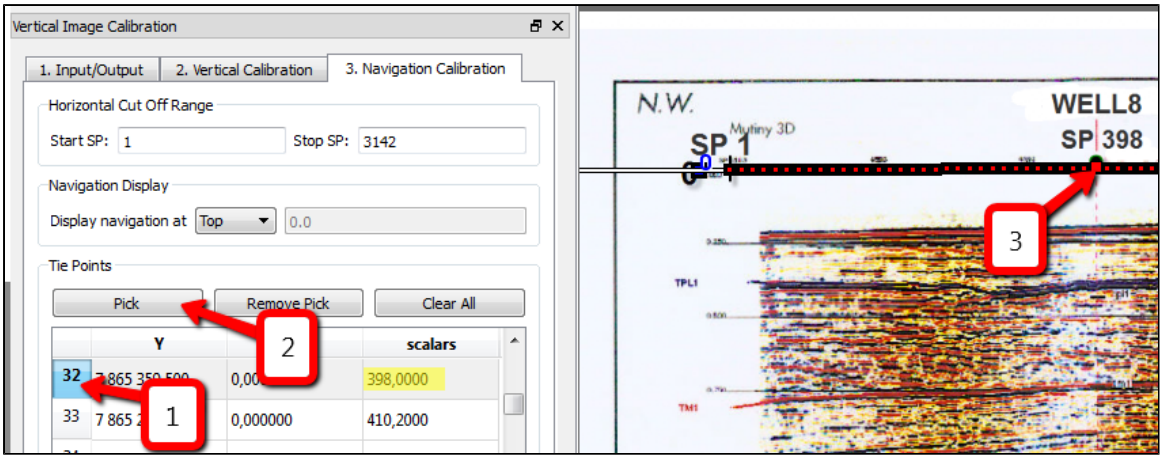
If you want to keep parts of the image above 0 or below the max depth of the seismic image you can do it by adjusting the cut off range after you have set the calibration points. In the example above we could set the cut off start to -100. This would preserve the well-information at the top of the image.

Navigation Calibration

- Click the **3. Navigation Calibration** tab to start the vertical calibration.
- Locate the first shot point in your image and find the corresponding shot point in the **Tie Points** list in the menu.
- Click on the **Tie Point** in the list, click the **Pick** button and click on the shot point in the image.

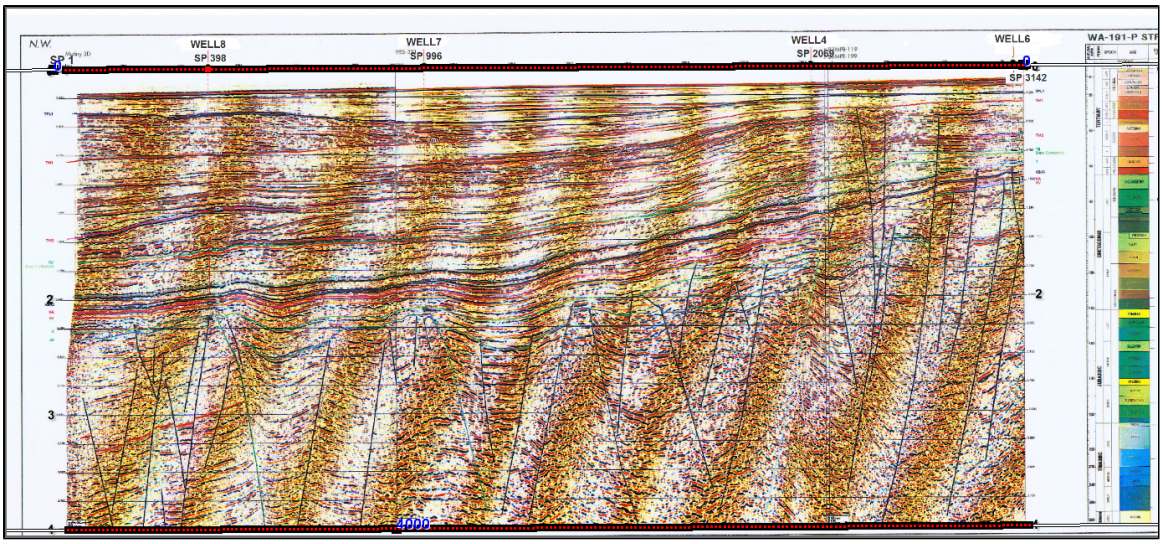


- Locate the next shot point in your image and find the corresponding shot point in the **Tie Points** list in the menu.
- Click on the **Tie Point** in the list, click the **Pick** button and click on the shot point in the image.



- You should now see a line of points along the shot point line. Continue this process until you have reached the end of the image.

The finished navigation calibration should look something like the screen shot below:

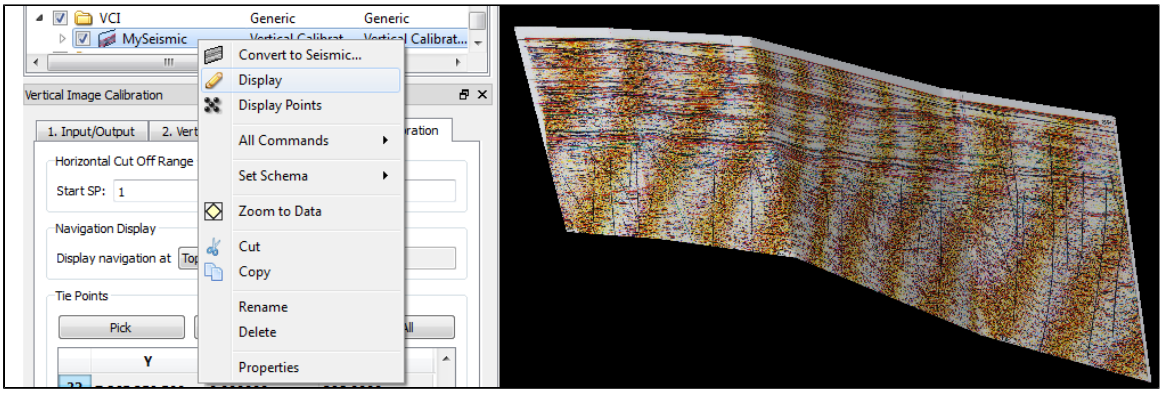


Everything to the left and right of the dotted line and everything above and below the white lines will be cut off on the completed VCI.

- Click the **Save** button in order to create the VCI

Display

1. Locate your VCI in your result folder.
2. Right-click the VCI dataset and select **Display**



It is possible to do seismic interpretation on the vertical calibrated image. Just open the Tools > Seismic Interpretation panel and pick the Vertical Calibrated Image instead of Seismic.

It is possible to convert a VCI to a seismic by "Convert to Seismic". Then the result may be exported as a SEG-Y.

R. Workflows

Introduction

A workflow in Geocap is a way of combining datasets and actions or visualizations in sequence to produce visualizations, movements and powerpoint style 3D presentations. The Atlantis Project is used throughout this tutorial.

Exercises

- Examine the workflows provided in the Atlantis_Mini project.
- Create a new workflow

Exercise

Examine the workflows provided in the Atlantis_Mini project.

1. In Geocap, click **File > Open > File**, and browse to the Atlantis_Mini folder, then open the the sub-folder called **Workflows**.
2. Double click on the file called **sample_workflow.gwf** to open it, or single click to highlight it and then click, **Open**. This will open the workflow below the project.
3. Examine the contents of the workflow items by clicking on the triangles next to the root folders.
4. Click on the **Go to first executable element** icon and then click on the **Execute next element** icon. This will display the seabed and land.
5. Click on the **Execute next element** icon again and notice that the 2500m isobath is displayed.
6. Continue clicking on the **Execute next element** icon until you reach the end of the workflow and observe how different elements of the Atlantis_Mini project are displayed.

Note that the user may also execute each element one at a time by right clicking on the command and choosing **Execute**.

Exercise

Create a new workflow

Workflows are saved as single files on disk with file extension ***.gwf**, (Geocap Workflow). A good idea may be to create a folder in your project called **Workflows** and put the workflow there.

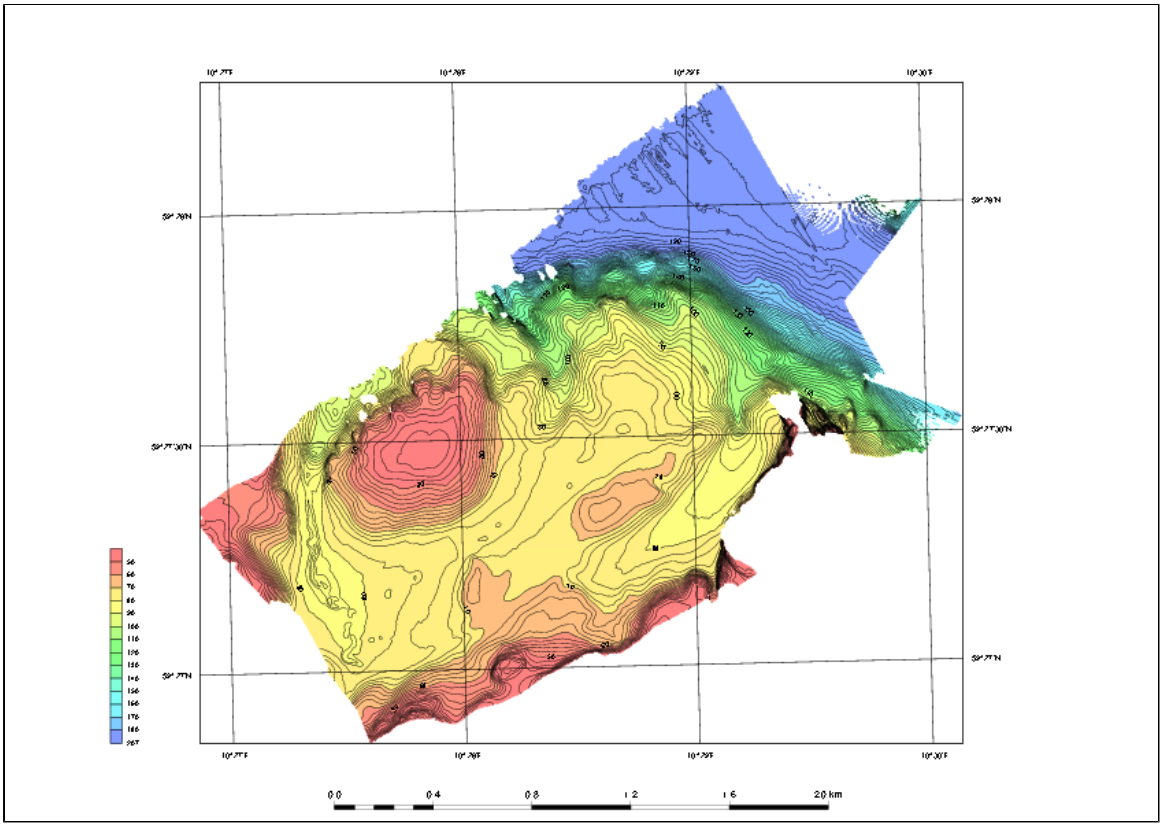
1. Click **File > New > Workflow**. Then a browser pops up.
2. Press the **New Element** icon to create the first entry. That may also be achieved by a right-click **New Element**.
3. Select the **Display** command found in the **Commands** folder and click **OK**.
4. Right-click the **Display** command in the workflow and select **Edit**.
5. In a workflow, you will need to select a dataset from the project. The command usually requires a dataset to do an action on (such as a type of display).
6. In your project, select a line and click the green arrow in the Display edit menu.
7. Choose the display settings you like (color, glue to surface or not, etc.) and click **OK** again.
8. Right click the element in the workflow and choose **Execute**. Alternatively, click the **Execute** icon.
9. Examine how your display looks and edit the command if necessary to achieve the results you want by right clicking on the command and choosing **edit**.
10. In order to keep your workflow organized it is recommended to change the name of the newly added commands from **Display** to something more descriptive like, **Display Coastline Yellow** which is done by right clicking the command and choosing **Rename**.
11. Continue to add different commands to your workflow
12. Remember to save the workflow by clicking

Commands may also be copied from **Shared commands**, **Item commands** and **Schema commands** and pasted into the workflow.

S. PostScript Plotting

Introduction

Geocap has a plotting system utilizing the PostScript language. The plotting system enables you to create scaled maps containing axes, scale bars, contour maps, lines, points, legends and anything else you might need.



Note that you will have to install Ghostview and Ghostscript in order tho make the plotting system work.

Exercises

- Open and run a saveset in the plotting system
- Explore the different plotting options

Exercise

Open and run a saveset in the plotting system

1. To open the PostScript plotting panel go to *Tools > PostScript Plotting
2. Click **File > Read saveset**
3. Browse for the saveset in the plotting folder under the **Seafloor_Training** project on your disk.
4. Click **Generate Draft**
5. When asked if you want to overwrite existing plot, click **Continue**
6. When the plotting has finished click **File > View Plot in Ghostview**

Ghostview should pop up and show your plot.

Exercise

Explore the different plotting options

1. Try clicking on the different check boxes to open the settings menu for each option.
2. Try to change some values and re-run the plot to see the changes.