Seafloor Tutorials

Seafloor
Hydrographic toolbox

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 mapping and modeling geoscientific data, designed with focus on interactivity and flexible control of visualization and manipulation. 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. In this tutorial collection we are going to use the Seafloor project template.

Data visualization and manipulation is operated through commands. Commands can be used to display datasets in the graphics window, manipulate existing datasets or to generate new datasets. The Geocap Sefloor module offers a range of commands for processing hydrographic data.These commands are available from tthe Seafloor Main Menu or if you right click a dataset or folder in the project.

All the project data is visualized in the graphics window and by using the mouse it is possible to pan, zoom, rotate, spin, scale z-axis and so on.

This section will guide you through the basics of opening a project, explore and display the project data and navigate in the graphics window.

Exercises

- Start Geocap on Windows 7
- Start Geocap on Windows 8.1
- Set working directory and auto save
- Open the Atlantis (Mini) project
- Explore the project folder structure
- Display data
- Navigate using the mouse

Starting Geocap

Exercise
Start Geocap on Windows 7
On your start menu click **Start > All programs > Geocap > Geocap 7.x.x - 64 > Geocap 7.x.x - 64**

Exercise
Start Geocap on Windows 8.1
- Go to the **Start screen**.   - Near the lower-left corner of the Start screen, click the arrow down and go to the **Apps view**.   - In the search field in the upper right corner, enter the text **Geocap**.   - All the files installed by the installation wizard will be listed. (It is also possible to find the Geocap 7.x.x group without using the search field.)   - Click on the Geocap 7.x.x - 64 entry to start the program.

-

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.

Initial Setup

Geocap comes with a range of settings which can modified to make it easier to work with. One of these settings is the working directory which is the folder Geocap will open every time you browse for something. We will now set this directory and we will also turn on auto save.

Exercise

Set working directory and auto save

1. Click **Tools > Options**
2. Click the **browse** icon for the **Working Directory.**
3. Select the folder **Geocap1** and click **Select folder**
4. Click on **Projects**
5. Check the box **Automatically save all projects at regular time intervals**
6. Set the **Time interval** to **5**
7. Click **OK**

-

Geocap Project

The Atlantis project is built using the UNCLOS project template. This template consists of four main top level folders; Maritime lines, Seabed, Sediment data and Outer limit. The maritime lines folder will hold all the data that has no reference to depth or is located at the sea surface. The Seabed folder will hold all the data located at the seabed, like bathymetric profiles, foot of slope points and so on. The Sediment data folder will hold all the subsurface data like seismic, interpretations, velocities, sediment profiles etc. The Outer limit folder will hold all datasets which is used to construct the outer limit. We are now going to open the Atlantis project and look closer at this project structure.

Exercise

Open the Atlantis (Mini) project

1. Click **File > Open > Project**
2. Browse to the location of the Atlantis_Mini project. (*Geocap1/Projects/Atlantis_Mini/db*)
3. Select the **Atlantis_Mini.db** file and click **Open**. A window similar to the one below will appear.
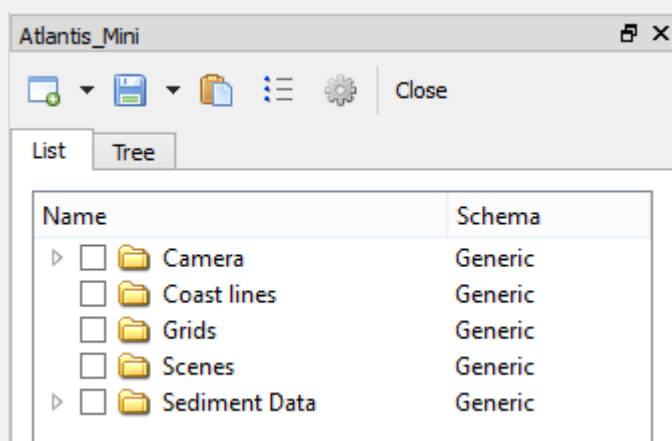
An open project

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

Pressing the small triangles (or '+' in other 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.
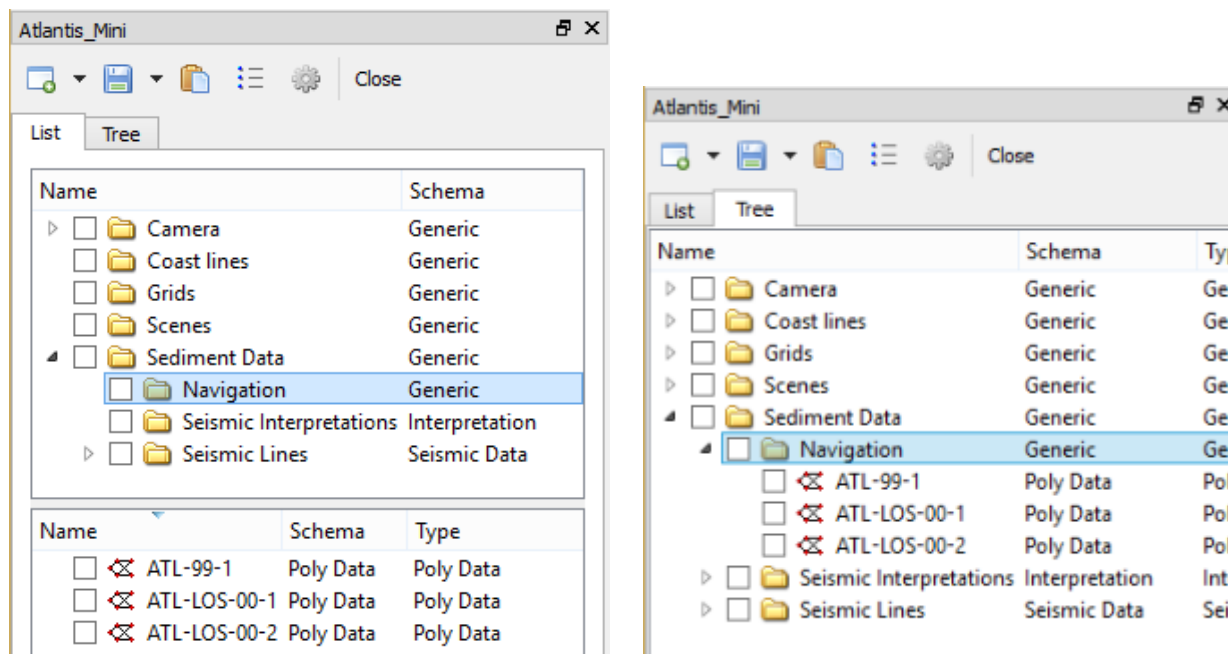
The Geocap Project structure and data items shown in List Mode (left) and in Tree mode (right).

In the next exercise we are going to explore how the Atlantis project is structured. 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. The right click popup menu also contains **commands** which may be executed on the datasets.

Exercise

Explore the project folder structure

1. Expand the different folders and observe the sub folders and datasets
2. Right click on folders and datasets and look at the right-click menus.

Note that there are two "modes" of the project view. Switch between these two by clicking on the tabs; **List** and **Tree**. **List** view sorts folder content into a separate window, while **Tree** view sorts everything into the tree itself.

A Geocap project is stored in a folder on your computer. The project consist of a **".db"** file, which contains information about the project tree, and several files which contain the data sets. You should let Geocap manage the files in the project folder and should not store other files in this folder.

If you want to make a backup of your projects, or if you want to exchange Geocap projects with other Geocap users you may zip the project folder with all the underlaying datasets into one file. This is convenient for sending e.g. to support@geocap.no.

Displaying Data

By right clicking a dataset a range of different visualization commands is made available. For each type of dataset Geocap will have a default command. This is the command which will be used when ticking the checkbox next to a dataset. We are now going to display the different datasets in the project using the default command.

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 **Coast lines** 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)

Note that by checking and unchecking the first level folder **Coast Lines**, or **Grids** all datasets displayed under this folder are shown and hidden.

-

Navigating in the 3D window

Now that we have displayed some of our data we can start navigating around the Atlantis continental shelf. Geocap is best operated using a three-button mouse with a wheel, see picture below. A two-button mouse or using the Touch-pad is possible but not recommended. Navigating in the 3D graphics window is done by pressing one of the mouse buttons while the cursor is in the graphics window, and moving the mouse while keeping the button pressed.

- **Rotate** - Press left mouse button + move
- **Pan** - Press middle mouse button (wheel) + move or Shift + left mouse button + move
- **Zoom** - Press right mouse button + move
- **Scale Z** - Scroll mouse wheel
- **Spin** - Ctrl + press left mouse button + move
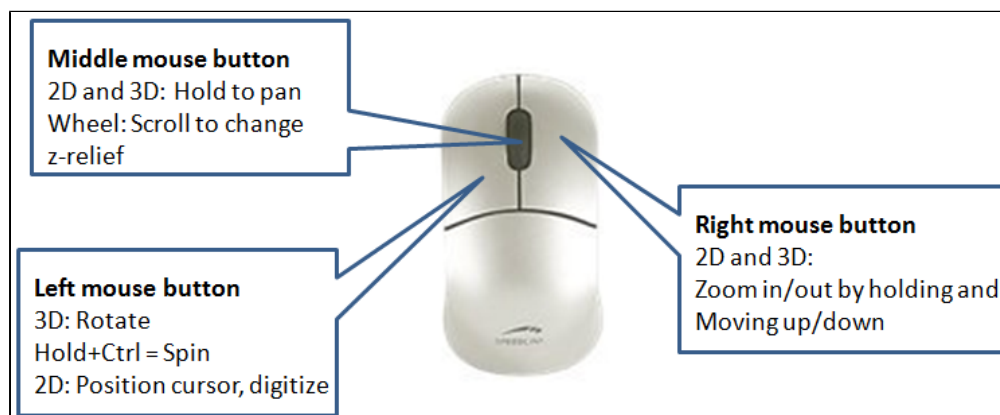
The mouse buttons

<table>
<tr><td colspan="2" align="center">Exercise</td></tr>
<tr><td colspan="2">Navigate using the mouse

1. Zoom in close to one of the bathymetric profiles displayed by press right mouse button and moving the mouse upwards.
2. Pan along the seabed surface by pressing the middle mouse button (wheel) and moving the mouse in any direction.
3. Zoom out again by press right mouse button and moving the mouse downwards.
4. Rotate the displayed data by pressing the left mouse button and moving the mouse.
5. Spin the data by holding down Ctrl on your keyboard, pressing the left mouse button and moving the mouse.
6. Now try to increase the z-scale by scrolling the mouse wheel upwards.
7. Move around the 3D model an practice these elements for a few minutes.

Tip
The focal point is the center of the display and the point of rotation. Change the focal point by positioning the mouse cursor on a desired point in the display window and press the X key on the keyboard.
</td></tr>
</table>

B. Seafloor Main Menu (Extract from User Guide)

In this section:

- Introduction
- How to open the Seafloor Main Menu
- The Content of the Seafloor Main Menu
- Menu bar options (File, Edit, Help)
- The Current Project part
- The Working with Survey Data part
- The Working with Charts part

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.
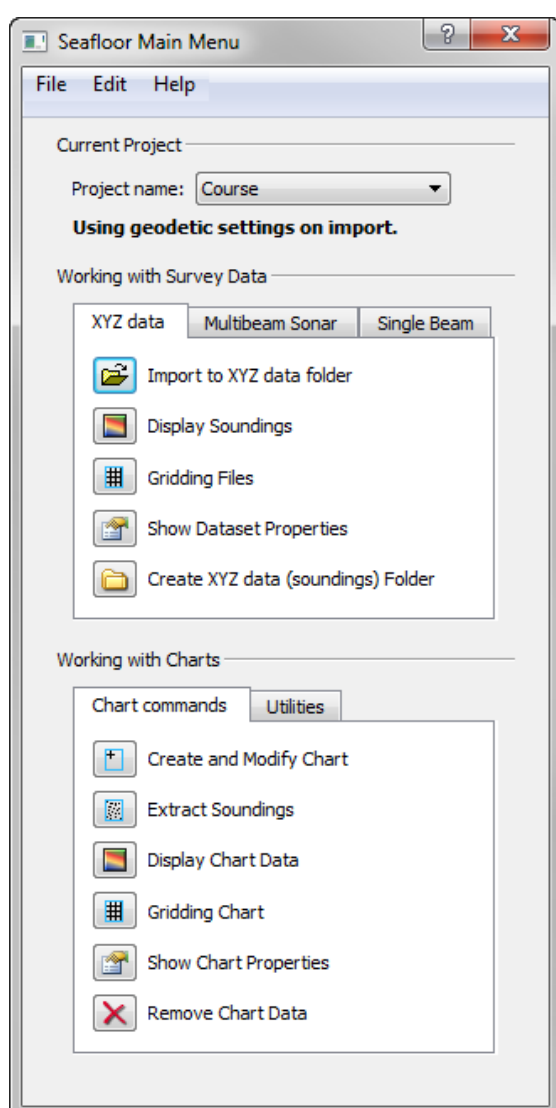
The Seafloor Main Menu.

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.
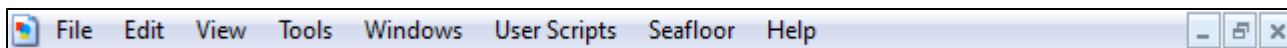
The Seafloor pull down in the Geocap menu bar

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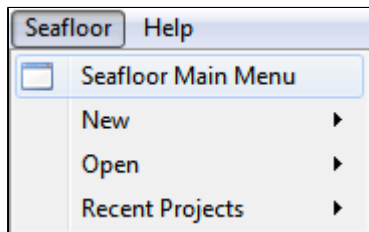
Open the menu

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)

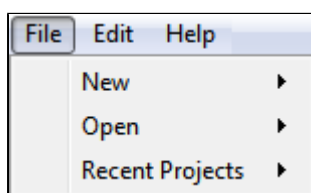The upper part of the menu is a menu bar with the entries **File**, **Edit** and **Help**:
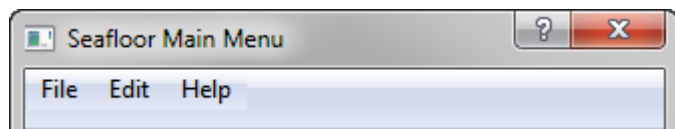

The Menu bar

For the actions from menu bar entries, see below.

-

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.
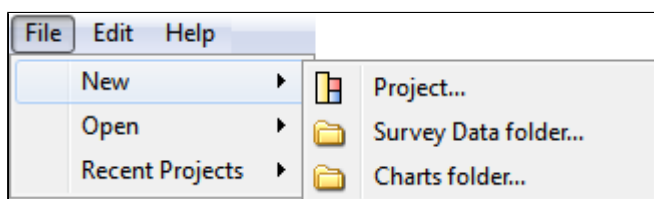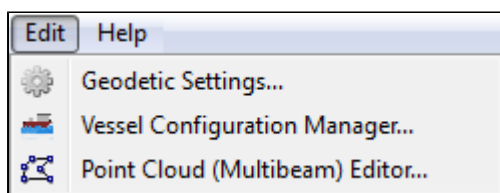

- **Open - Project...** will open any existing Geocap projects in the computer or network.
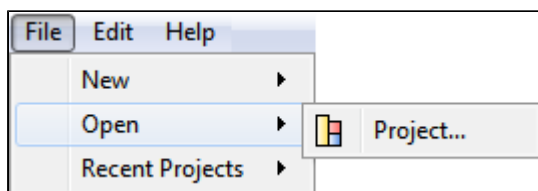
- **Recent Projects** will give a list of recently opened Geocap projects.

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.
- **Point Cloud (Multibeam) Editor** will open the 3D editing tool for point cloud data (multibeam data) (Multi beam Editor). This entry will work both when *Point Cloud data* and *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.
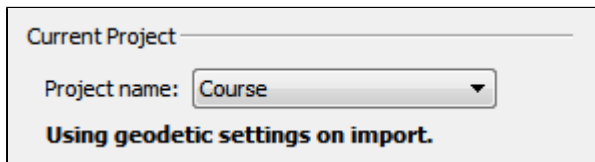
The Current Project part

All projects are shown in the pulldown menu behind the text *Project Name*. The pulldown and the the project information is is automatically updated if a project is closed or a new project loaded into Geocap.

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
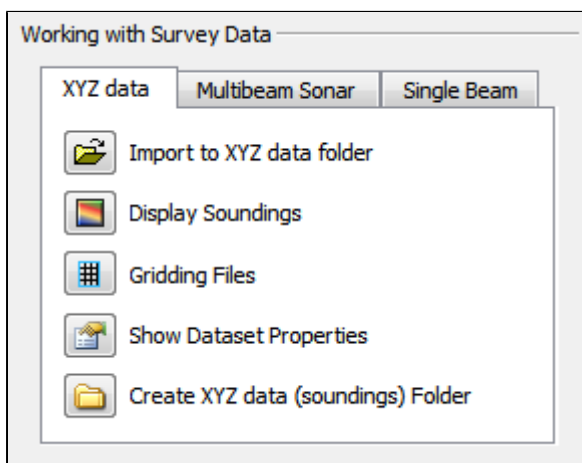
*The Working with Survey Data part*

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 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.
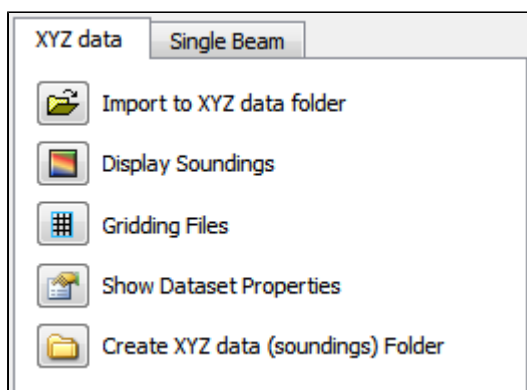
The XYZ data tab

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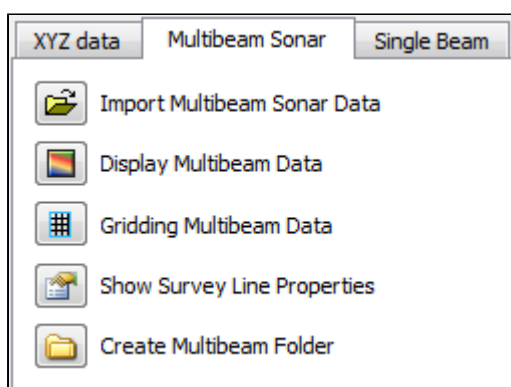
Show file properties: This command will show dataset name, bounding box and number of points of the data in folders of type **Soundings**.

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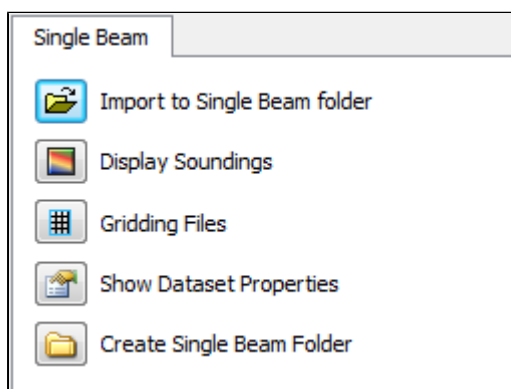
Show Survey Line properties: This command will show dataset name, bounding box and number of points of the data in folders of type **Multibeam**.

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*.

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

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
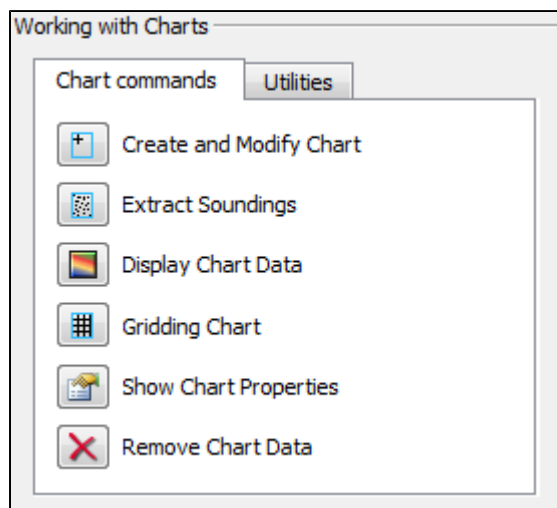

The Working with Charts part

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.
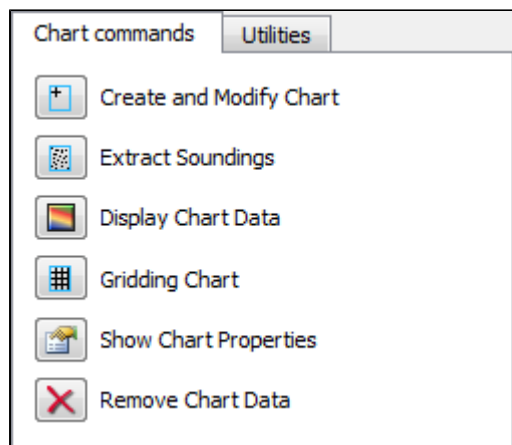
The Chart commands tab

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**.

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.

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.

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

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**.

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.
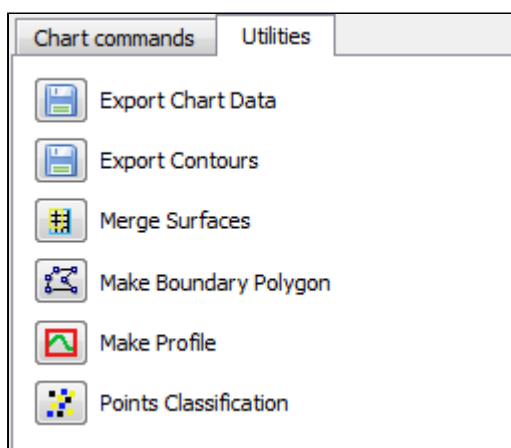
The Utilities tab

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.

Merge Surfaces: This command will create a new **Seafloor** surface (DTM) for a chart by merging surfaces from other charts in folders of type **Chart**.

Make Boundary Polygon: This command will create the boundary (outline) for a chart**.** The boundary can be created from the **Seafloor** (DTM) dataset or from the **Soundings** (points) dataset. (Not enabled for Shelf.)

Make Profile: This command will 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.)

C. 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

Exercises

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

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.

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 **Geo detics**. 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**

D. 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
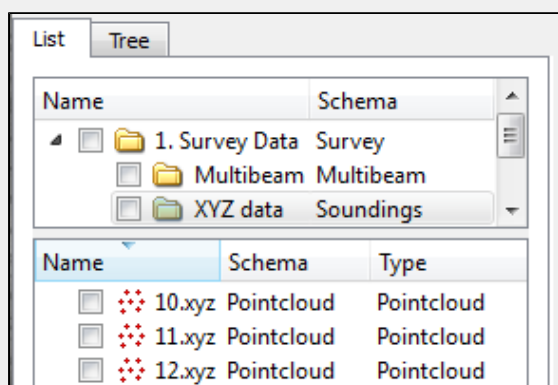- Show properties for the Multibeam data
- Show properties for the XYZ data

Importing

<table>
<tr><td align="center">Exercise</td></tr>
</table>

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**

Name	Schema
▲ ☐ 📁 1. Survey Data	Survey
▲ ☐ 📁 Multibeam	Multibeam
▲ ☐ 📖 0000_20060112_130156_Simrad_echo.all	Multibeam Survey
☐ 〰 MultiBeam Sonar	Multibeam Sonar
☐ 〰 Gyro	Sensor Data
☐ 〰 Heave	Sensor Data
☐ 〰 Pitch	Sensor Data
☐ 〰 Roll	Sensor Data
☐ 〰 Sound Velocity	Sensor Data
☐ ✗ Navigation	Poly Data
▷ ☐ 📖 0001_20060112_131814_Simrad_echo.all	Multibeam Survey
▷ ☐ 📖 0002_20060112_133438_Simrad_echo.all	Multibeam Survey
▷ ☐ 📖 0003_20060112_135102_Simrad_echo.all	Multibeam Survey
▷ ☐ 📖 0004_20060112_140705_Simrad_echo.all	Multibeam Survey
▷ ☐ 📖 0005_20060112_142118_Simrad_echo.all	Multibeam Survey

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

Project and folder: "SeafloorCourse // 1. Survey Data / Multibeam"

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 then 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**

E. 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
4. Click **Execute**

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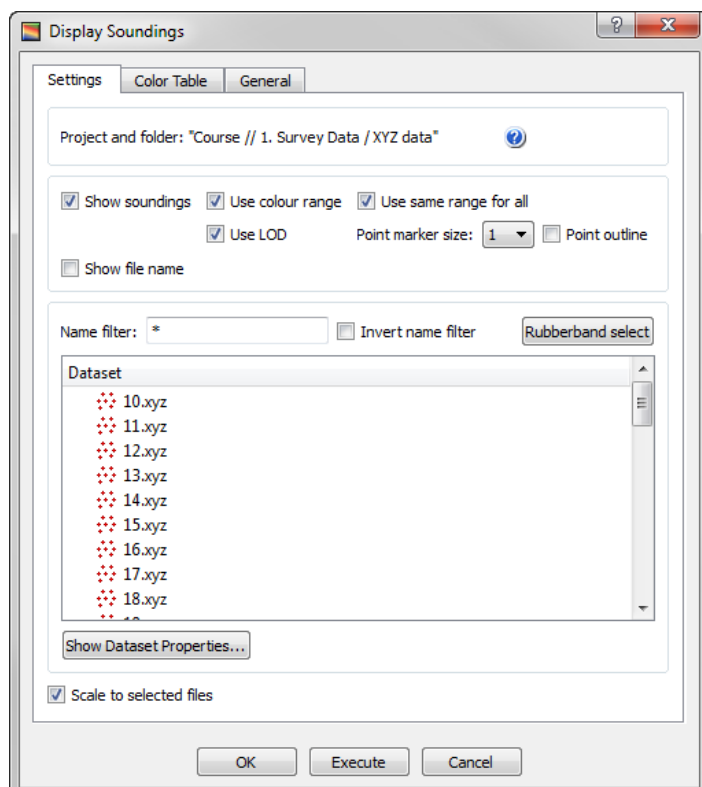
Displaying XYZ data

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.

-

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.
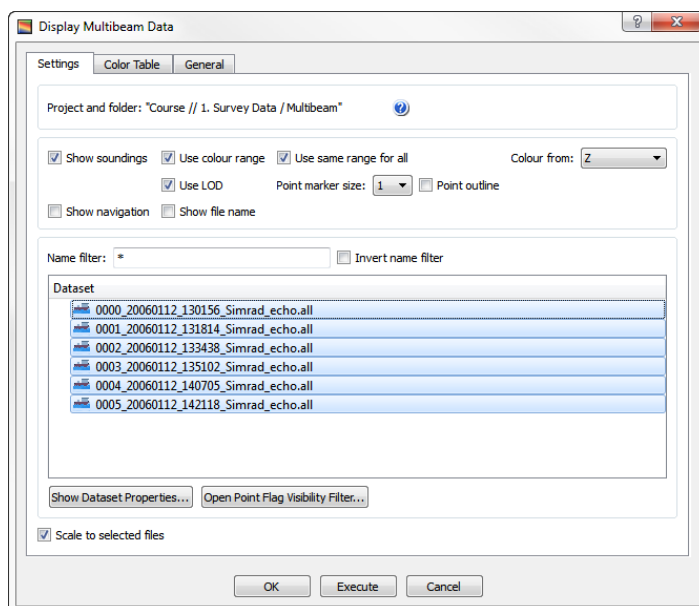
Displaying multibeam data

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.
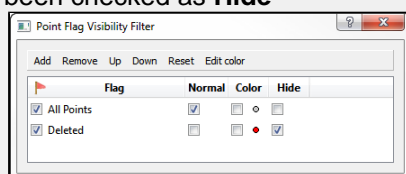

Exercise

Display filtered data

Make sure the **Display Mulitibeam Data** menu is open and that all survey lines has been displayed.
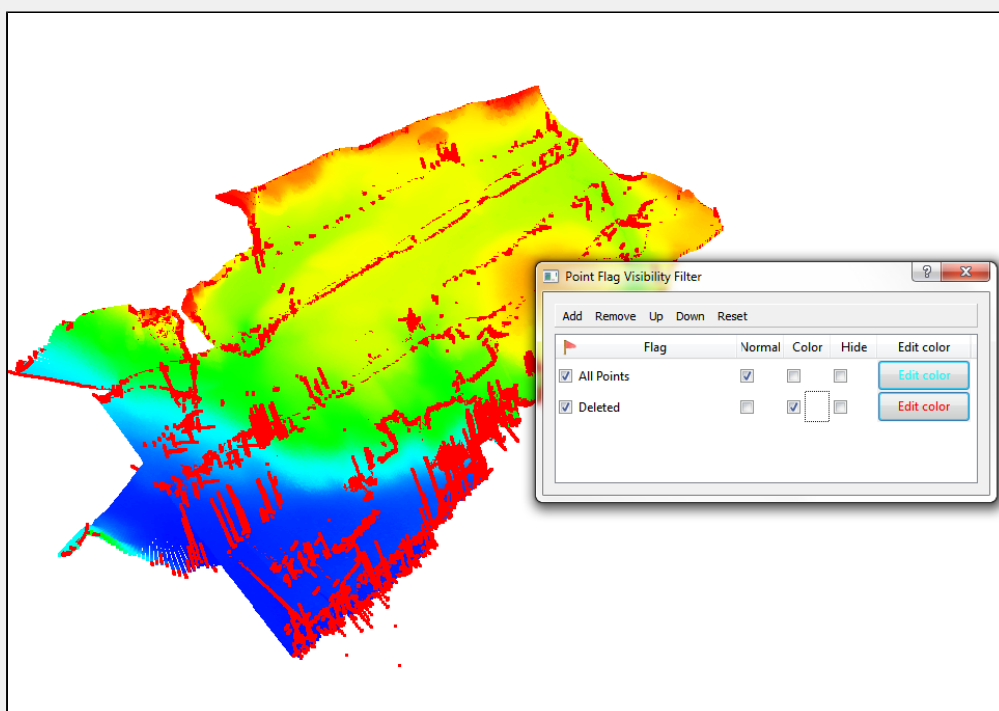
1. Click on the **Open Point Flag Visibility Filter** button.
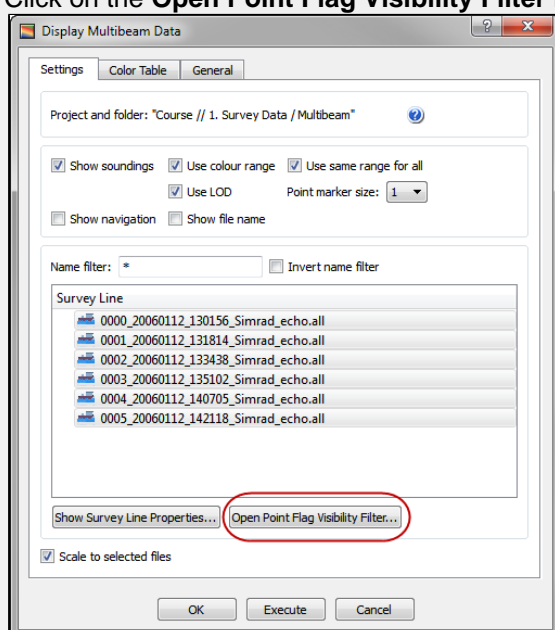
2. Notice that the flag **All points** has been checked as **Normal** and that the flag **Deleted** has been checked as **Hide**

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.

F. Basic Concepts in Geocap

Introduction

There are a few concepts in Geocap which are important to understand in order to work efficiently with the software. The main concepts are **Actors**, **Schemas, Geodetic Settings** and **Commands.** These concepts will be explained in this section. In addition to this section also talks about toolbars, color tables and keyboard short cuts.

Exercises

- Get familiar with schema commands
- Change the default display for seabed surfaces
- View the geodetic settings of a dataset
- Change the color table on a seabed surface
- Display data on the Sticky Surface
- Try out the different tools on the toolbar
- Try the different keyboard shortcuts

Actors

In display context an **actor** is a graphical term for a visualization unit. A complex display is created by a set of **ac tors**. **Actors** are shown as child's of datasets and are shown in blue text. In Geocap the basic principle is that a display **command** creates one or several **actors**. It is possible to remove the **actors** and the associated display by right clicking and selecting **Delete**

Visualizations are called actors

Schemas

By default datasets are organized into data types, such as points and lines (polydata), grids (structured points) etc. These are basic geometric types that contain little information about the domain in which they operate. E.g. coast lines, bathymetric profiles and boundary polygons are all lines (polydata), but the way we work with these types of data are very different and so it is only natural to classify them into different categories. This higher level of classification is made possible by **schemas**.

Notice that the dataset types are the same, while the schemas are different

The Shelf Module contains several schemas. Some of the schemas used in the Shelf Module are **coast line**, **ba se line**, **limit line**, **seabed surface**, **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 dataset´s 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.
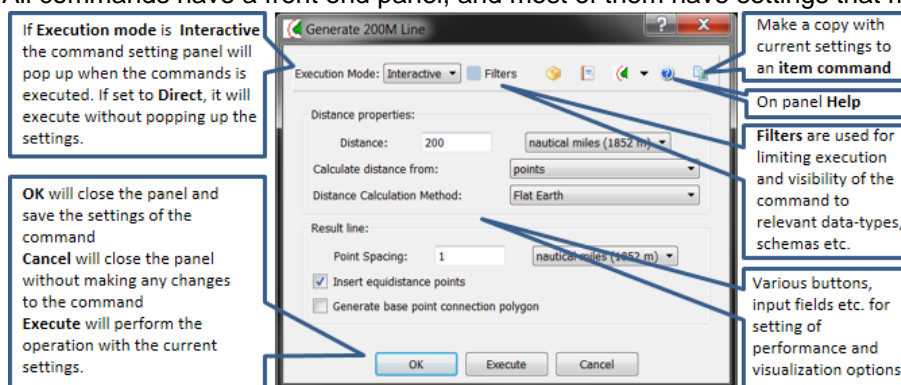
Commands

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.

Commands have two execution modes: **Direct** and **Interactive**. If the execution mode is set to **Interactive** the commands editor with the front menu will be displayed when you execute the command. This allows you to adjust various settings that define how the command works. Depending on the command, the menu will consist of different input settings from simple display settings to complex manipulation options. If the execution mode is set to **Direct** the command will execute with the last active input settings for the command. It is possible to see the underlying settings for any command by right clicking it in the **Toolbox** and selecting **Edit**.Note that some commands does not require input parameters and will therefore display the underlying code when running it in **Interactive** mode.

> To see if a command is set to **Direct** or **Interactive** mode, look behind the command name. If there are ellipsis (**...**) behind the name, the command is set to **Interactive** mode.

All commands have a front end panel, and most of them have settings that may be customized.

An example of a command front end panel

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** or on the right-click menu of a dataset or a folder. Commands can also be put together in sequence in **Workflows** to perform visualizations or data operations.

Item commands

An item command is associated with a single dataset or folder in the project. Item commands are stored along with the project. Therefore, if a project is transferred the item commands will transferred as well. Item commands typically contain settings that pertain specifically to the dataset to which it belongs.

Schema Commands

A schema command is stored with a schema. 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 are listed on the right click menu and in the **Toolbox** under **Schema commands**.

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.

Tip

The **Pin to Menu** 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.

Shared Commands

Shared Commands are commands which are shared with all datasets and folders. These commands are always available and listed in the **Toolbox** under **Shared commands**.

Default Commands

The **default command** is the command that is executed when you tick the box next to a dataset in the project. By default a dataset will have one of the schema commands as a default command. This can however be changed to any command.

Exercise

Change the default display for seabed surfaces

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

1. Click on the **Seabed** datasets in the **Grids** folder.
2. In the **Toolbox** right-click another command (i.e **LOD Grid Display**) and select **Set as default command**
3. Tick the checkbox next to the **Seabed** dataset and notice that the new command is executed.

-

Geodetic settings

In Geocap it is the responsibility of the user to secure that data have the correct **Datum** and **Coordinate system**, also called a **Projection** or **Geodetic settings**. Datasets with the same geographical location but with different coordinate system will not be displayed in the same location in the graphical window. Thus, you will need to convert one of the datasets to the other coordinate system.

Exercise
View the geodetic settings of a dataset
1. Right click any dataset in your project and click **Properties**   2. Click on the **Geodetics** tab   3. Observe the Geodetic settings of the dataset   4. Read the warning message   5. Click **Close**

You will learn how to convert datasets to other projectons later in this tutorial.

Color Tables

Geocap comes with a set of predefined color tables which can be seen in the toolbox. These color tables are of course customizable, or you can create your own color table from scratch. By default the color tables shown in the lower right corner of Geocap will be used to display a dataset. You can change these color tables by selecting one of them, right-click another color table in the toolbox and select **Activate**. You can also drag color tables from the toolbox and drop it onto a displayed dataset in the graphical window.

Exercise
Change the color table on a seabed surface
1. Visualize a seabed surface   2. Click on the **Color Tables** tab in the **Toolbox**   3. **Click**, **drag** and **drop** one of the color tables on the seabed surface

Sticky Surface

Geocap has a concept where any surface can be set to be **sticky**. When a surface is sticky, data 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 under the name **sticky_surface**.

Exercise
Display data on the Sticky Surface
1. Right-click on a Seabed surface dataset and select **Set as sticky surface**   2. Select a line, e.g. **Sedimen Data > ATL-99-1** in your project.   3. In the Toolbox under **Commands > Schema commands** right-click **Display** and click **Edit**.   4. Check the **Glue to Sticky Surface** and press **Execute**.   5. Investigate the line display   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 a foot of slope point or a bathymetric line.
>
> Keep that in mind.

Toolbar

- Will cancel any mouse action that is started but is regretted and should be ended without any operation.

- Start a mouse action that when clicked on a graphical object will delete that object.

- Start a mouse action that when clicked on a graphical object can set **appearance** parameters for that object. The **appearence** parameters are: **Opacity, Reflection, Diffusion, Ambience**.

- Start a mouse action where the user is supposed to click on a surface that is map with **scalar** values. The mapping range of the scalar values according to the color table can then be adjusted.

- Start a mouse action that when clicked on a graphical object will highlight the corresponding data object in the project.

- Start a mouse action where the user can click and draw a rectangle around displayed data and Geocap will list all dataset names and locations.

- Will delete all graphics in the current viewport or window (if the window holds only one viewport). The corresponding references to actors in the project are deleted. Frequently used in free interactive work.

- Will delete the last graphical object (also called and actor) that is displayed. Frequently used to erase a graphical object that shall be removed. **Note:** Redisplaying data from the project using the same command will erase the previous corresponding actor.

- Will set the graphical window in 2D mode; i.e. only **pan** and **zoom** is allowed. The view direction (x, y or z) depends on the corresponding setting. In 2D mode the left button on the mouse is used for immediate cursor response for instance in digitizing

- Will set the graphical window in 3D mode; **pan**, **zoom** and **3D rolling** is enabled. The default 3D mode renders in **perspective view**; i.e. parallel lines are not parallel on the screen, but shows a perspective.

- Will turn the z direction of the graphical window upwards; i.e. the positive direction of the z axis points upwards.

- Will turn the z direction of the graphical window downwards; i.e. the positive direction of the z axis points downwards. This is the default case because most surfaces are below the zero level, but still have positive z values.

- Will show the graphical window as a frame box. It is important to use this icon to check the graphical window or whenever some display comes out weird if the display algorithm uses the graphical window to set display parameters.

- Will draw default axes for all visible axes directions. No tick marks are display for simplicity, but the exact location of an annotation position starts at the beginning of the annotation.

- Will open a **Navigator** which is a convenient menu for navigating a 3D graphical scene. To some (especially newbies) it can be difficult to use the mouse buttons to orientate the graphics. Be aware that rotation is around the **focal point** which also can be set by pointing the cursor mouse at a any location on a solid object and push keyboard **x**.

- Will zoom in towards the **focal point**. The graphical window is not changed, although the graphical frame may lie out the visible part of the screen.

- Will zoom out from the **focal point**.

- Will **View from above**. This icon also contains sub-icons for viewing in other directions.

- If several windows are created on the same **Tab**, this icon connects the selected windows to the same visual camera. Very useful when different surfaces that shall be compared are displayed in separate windows that are connected and show the same under all graphical movements.

- Will show the **viewport menu** that allows for a quick way to create viewports inside a window. The number and layout of the viewports are determined by just double clicking on the lower right frame in the veiwport menu. If the viewports are connected they can also by used for simultaneously display of surfaces or features that shall be compared.

- Enables the light source to be moved around in the graphical scene. Will create shadows and highlight features. Usefull for presentation graphics and detailed studies of special structures.

- Will scale the graphical scene with all its actor up or down. Very important for selecting a good view. The scaling should also preferable be set in the **Project Settings** so that the preferred view comes up when loading a project.

- Will show up a 2D compass in upper right that follows the rotation of the graphical window. Another click on the icon will remove the compass.

Exercise

Try out the different tools on the toolbar

- Try out the different tools.

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
j	Snap to any point on a displayed line.	+	Zoom in.
o	Toggle color code for last used map command on/off.	-	Zoom out.
r	Rescale to all graphical elements.	2	Toggle graphics to 2d mode.
s	Turn graphics into surface mode.	3	Toggle graphics to 3d mode.
v	Value of height/depth (z coordinate) from graphics.	3	Toggle stereo view on/off when stereo is activated.
w	Turn graphics into wire mode.	y	Cursor point is set at the surface of the graphical element.
x	Setting the focal point. The graphics will rotate around this point.	z	Zoom by drawing a rubber band with left button on the mouse.

When using **j** or **y** to snap to lines or surfaces, Geocap will report what you have snapped to in the lower left corner.

Exercise

Try the different keyboard shortcuts

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

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 ea
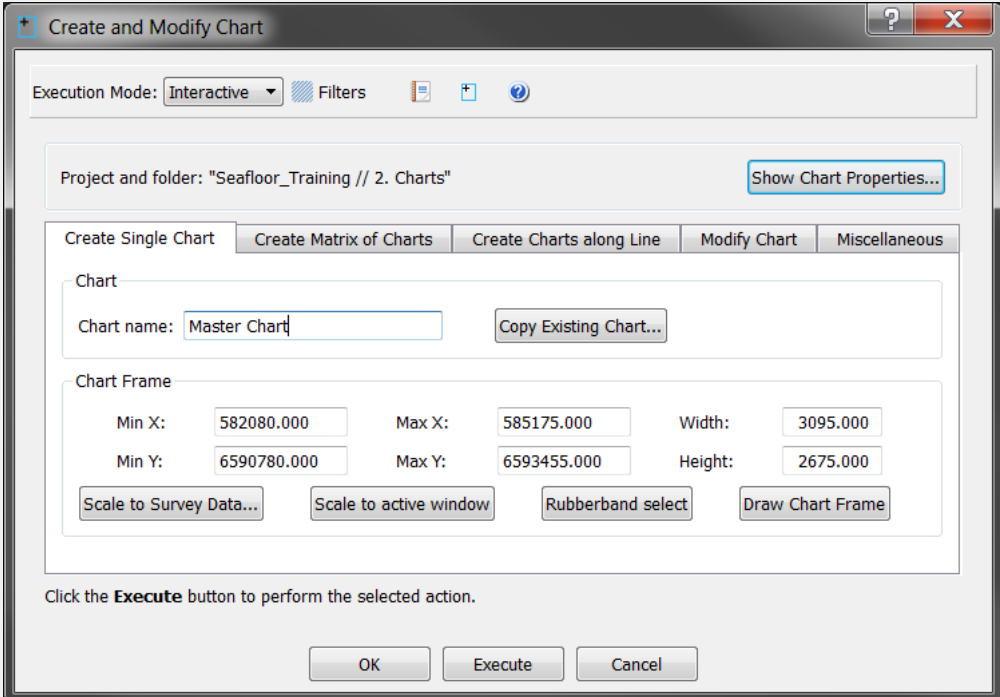
Exercises

- 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:
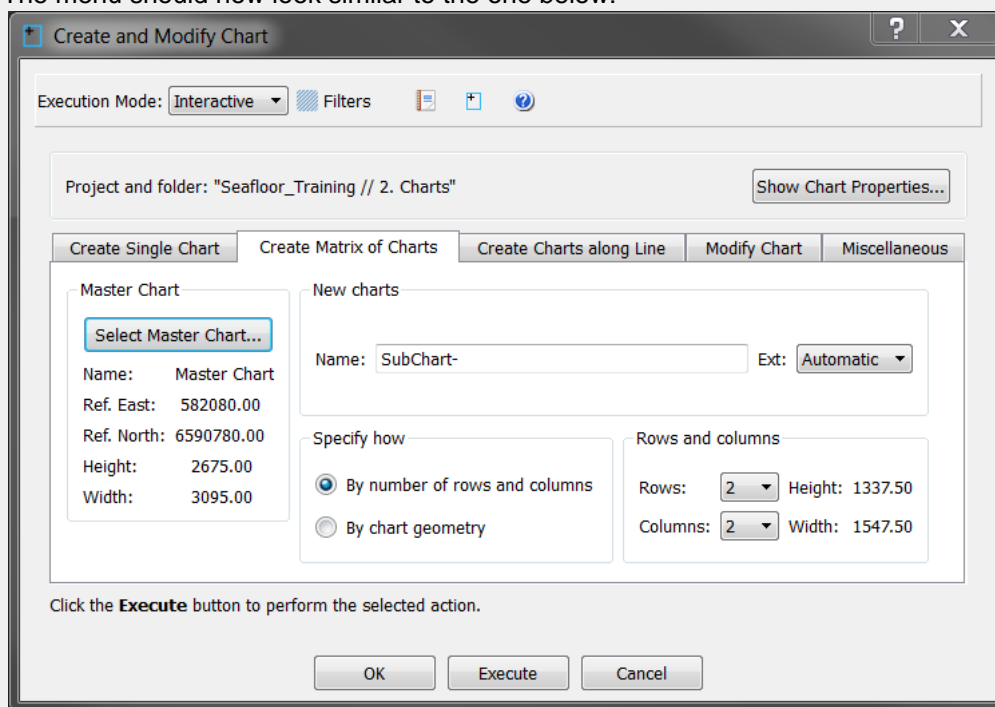

6. Click **Execute** to create the chart and click **OK** in the dialog that pops up.

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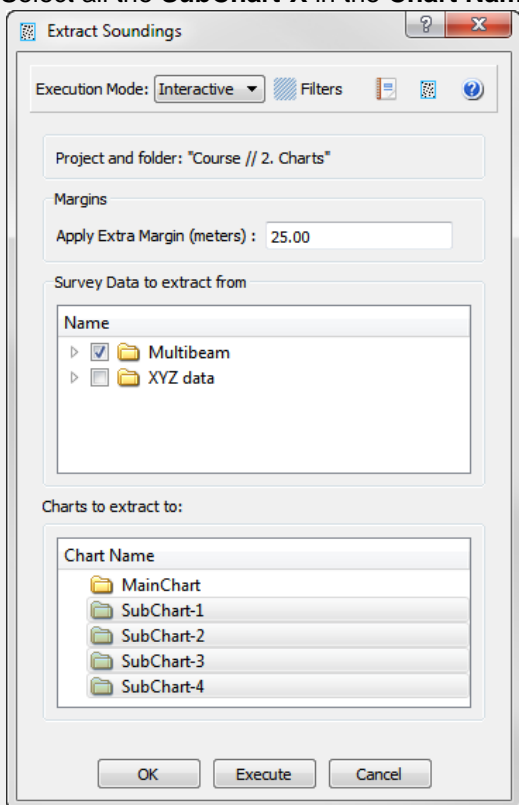
5. Click **Execute** and the extraction process will start.
6. Click **OK** in the dialog the pops up.
7. Click **OK** on the **Extract Soundings** menu.

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

-

Chart properties

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 Information

Project and folder: "Seafloor_Training // 2. Charts"

	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
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
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
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
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

OK

Showing Chart Properties

Displaying chart data

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.

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** a nd **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
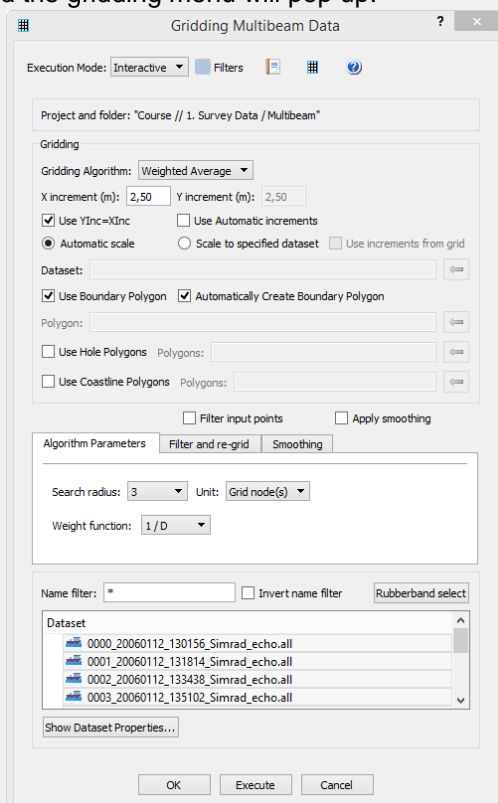- Display rejected soundings
- Merge Chart surfaces

-

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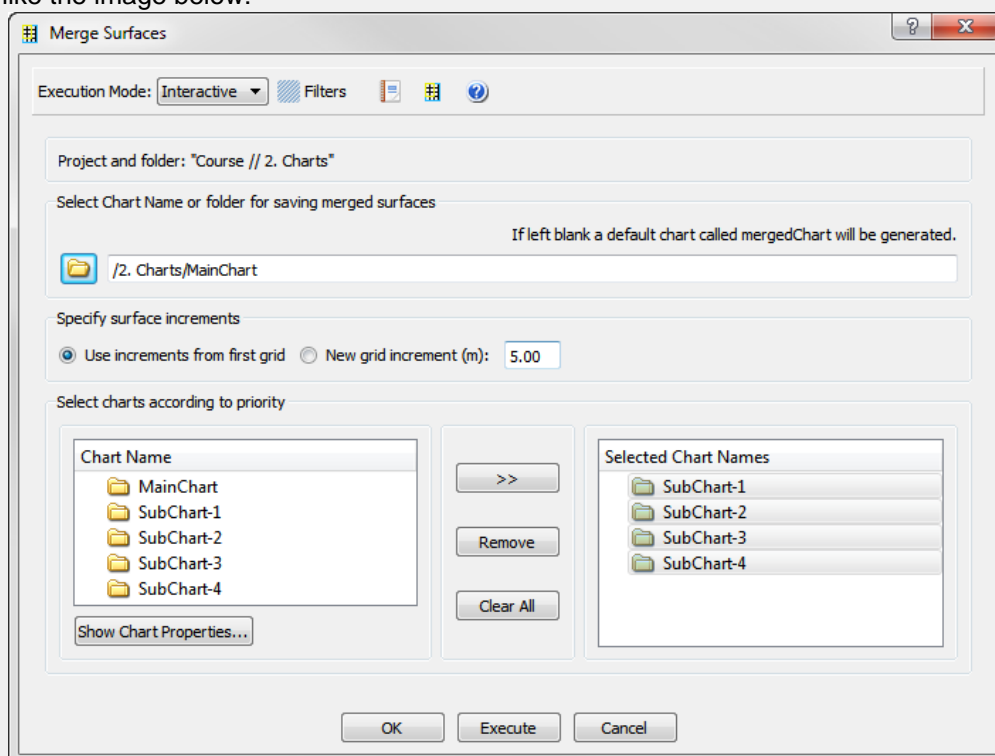
5. Click **Execute** and a dialog will pop up asking if you want to merge 4 charts.
6. Click **Yes**

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. Multibeam Depth Editing

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.

The **manual editing** is performed by the **Point Cloud Editor** (Multibeam Editor).

> 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.

> The editing methods described in this section are also applicable on Point Cloud (XYZ) data.

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.
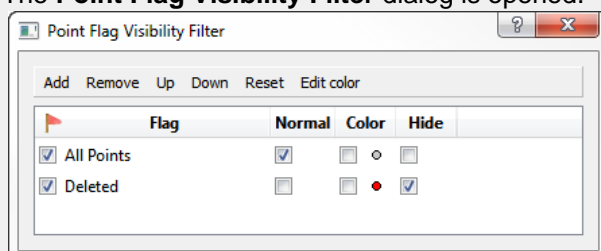

Exercise

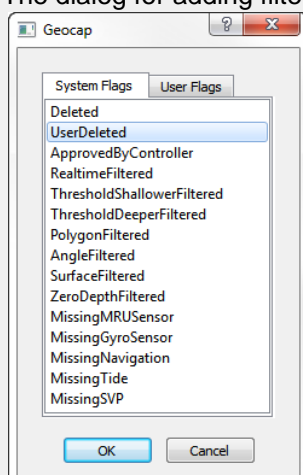**View the result from the automatic Threshold filtering**

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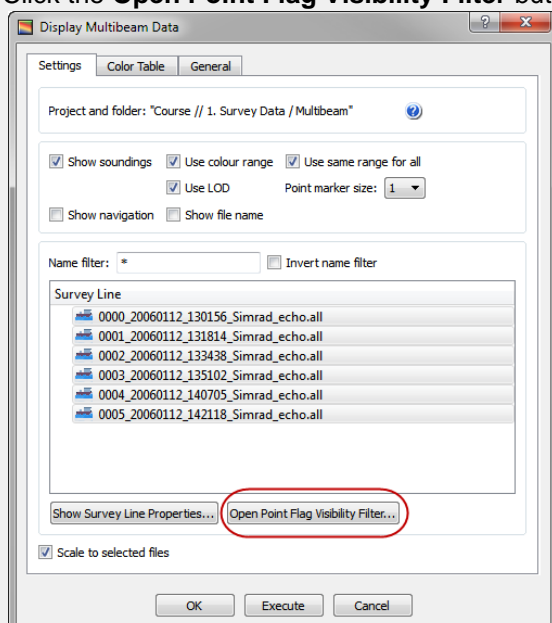
4. The **Point Flag Visibility Filter** dialog is opened.

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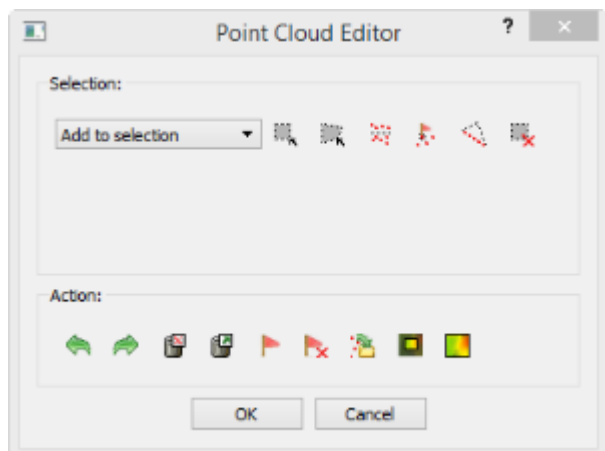
The Point Cloud Editor menu

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 five 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.
- Expand selection to ping - First use one of the other tools to select points. Then click the this tool to select all the other points in the corresponding ping(s).
- 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.

J. 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**.
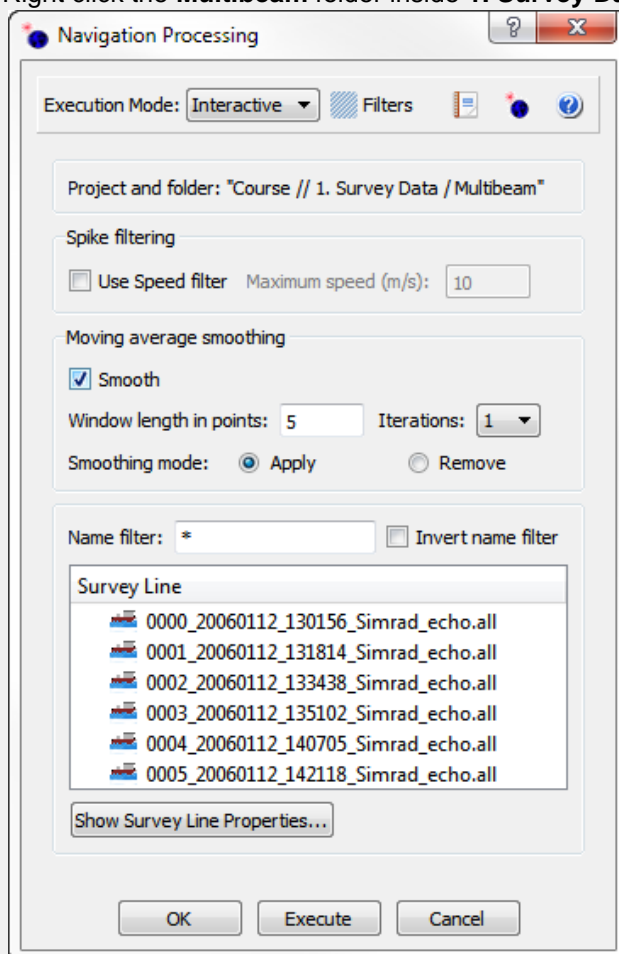
2. In the Navigation Processing dialog check the **Smooth** box
3. Set Window length to **5**, and iterations to **1**.
4. Check that the smoothing mode is **Apply**.
5. Select one or several of the survey lines.
6. Click **Execute**

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.
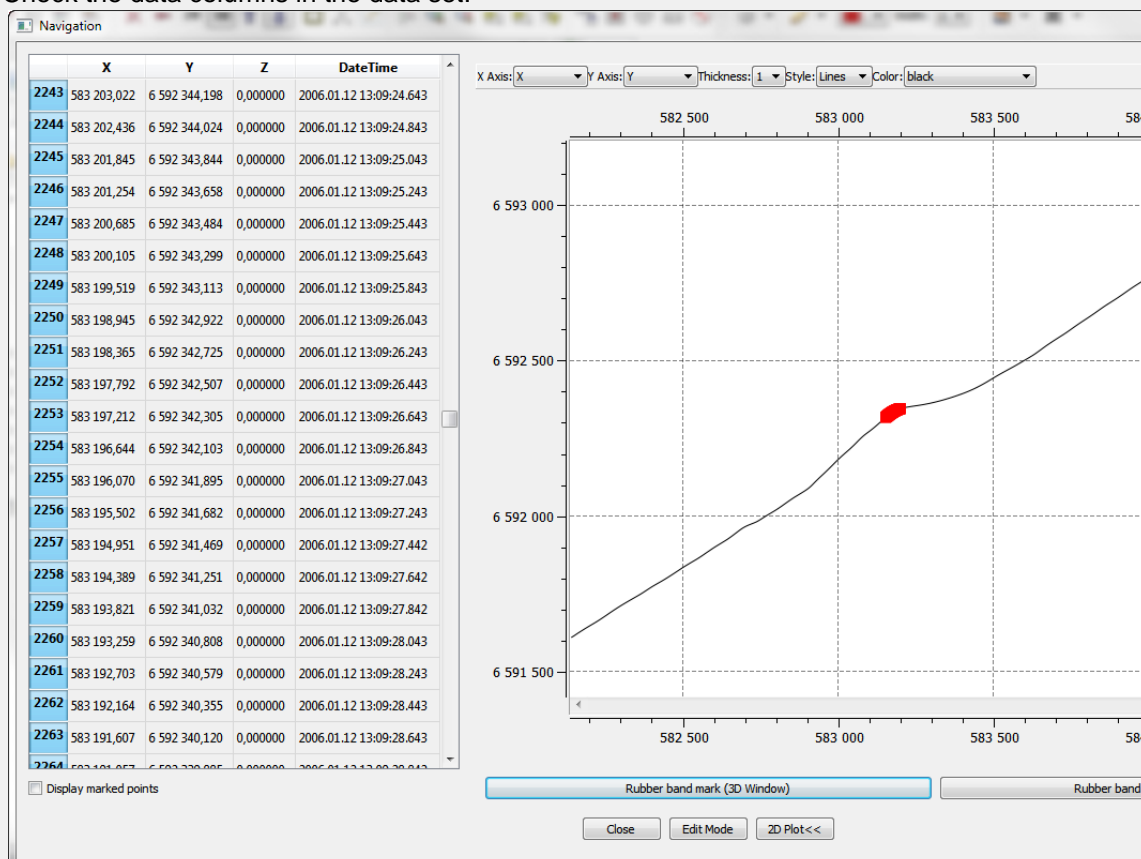
8. In the graphics window, hold the **Shift** button and click and drag the rubber band around the points you want to investigate.
9. Notice that the selected points are displayed both in the table and in the 3D graphics window.
10. Use the buttons **Rubber band mark (3D window)** and **Rubber band unmark (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.

K. The Cross Section Window

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.

The Cross Section Window can be used to check calibration of the multibeam survey lines.

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_1301 56_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.
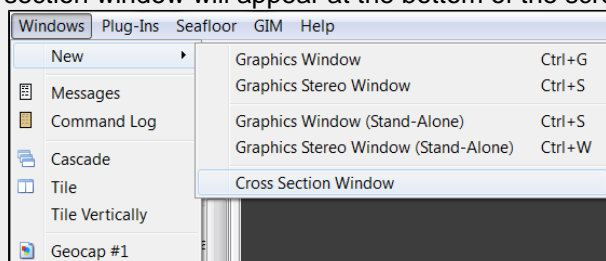
2. In the cross section window click the arrow to right of the icon, and open the menu.
3. Click the **Digitize cross section** entry in menu.
4. Create a profile over the overlapping area, by digitizing two points in the graphics window using your left mouse button.

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**.
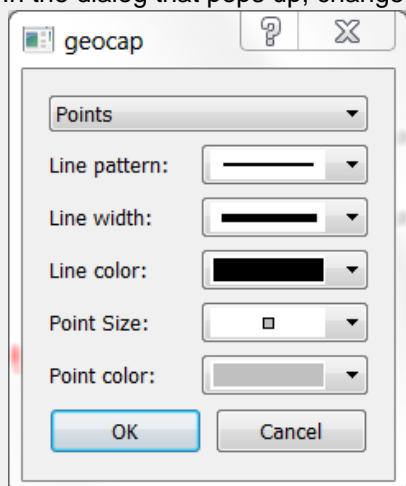

Exercise

**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**.

   ☑ ▪ 0015_20051028_10
   edit
   Remove
   ☑ ▪ 0016_20051028_11...
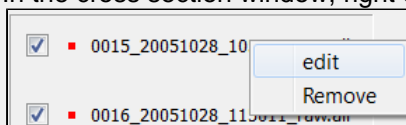
2. In the dialog that pops up, change the **Point color** to another color than the default.

   geocap

   Points

   Line pattern: ──────
   Line width: ──────
   Line color: ████
   Point Size: □
   Point color: ████

   OK   Cancel

3. You should now be able to distinguish the points on each line from each other.
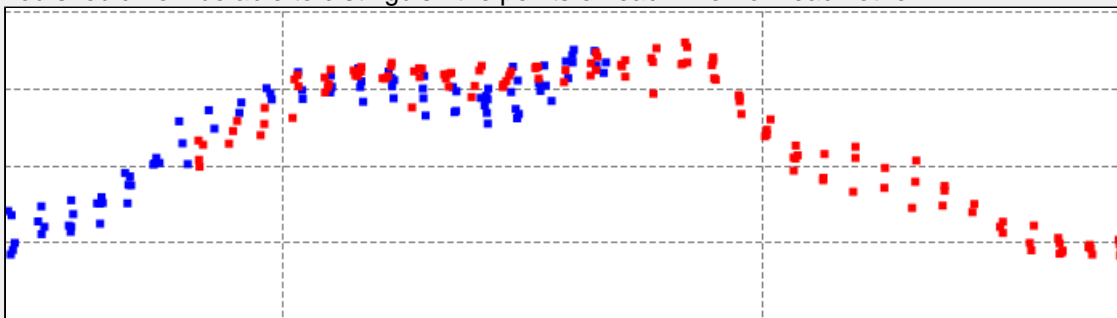
Try out different display options in the **edit** dialog.

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.
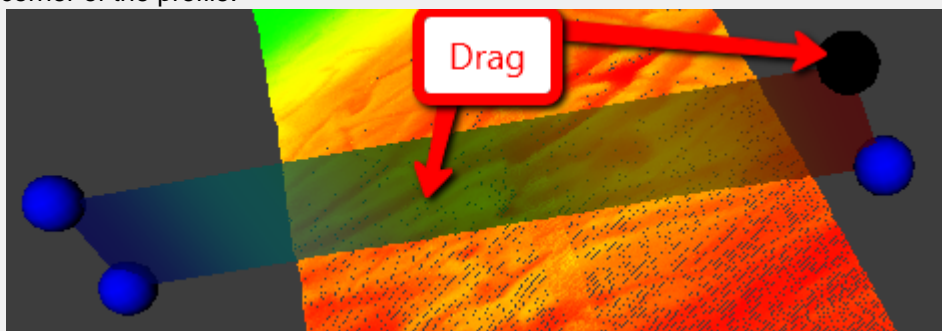
   Drag

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.

L. 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.
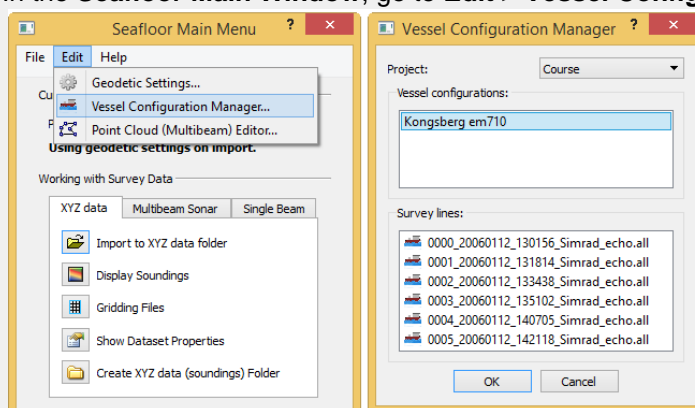
Exercises

- 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**
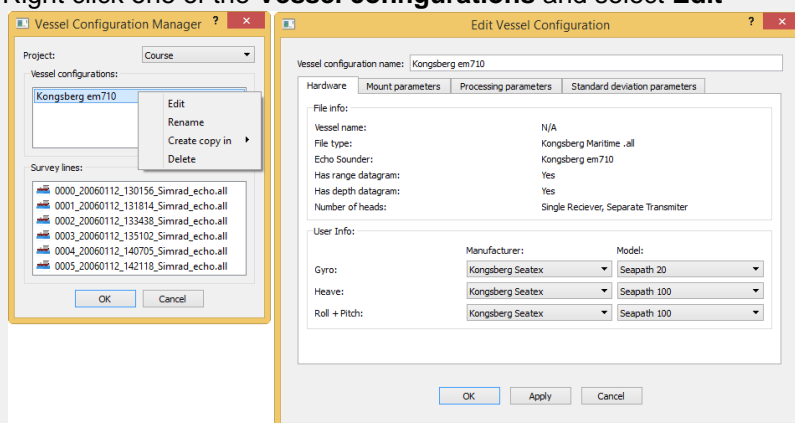
Exercise

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.
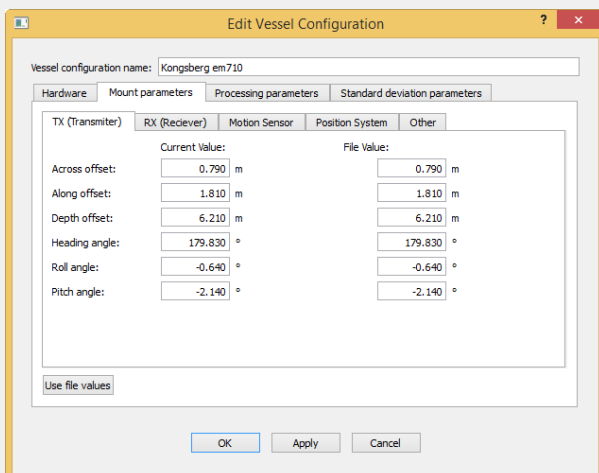
Exercise

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_13 1814_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_13 1814_Simrad_echo.all*.
14. Also try the Roll angle **-0.300**.

M. 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**

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.
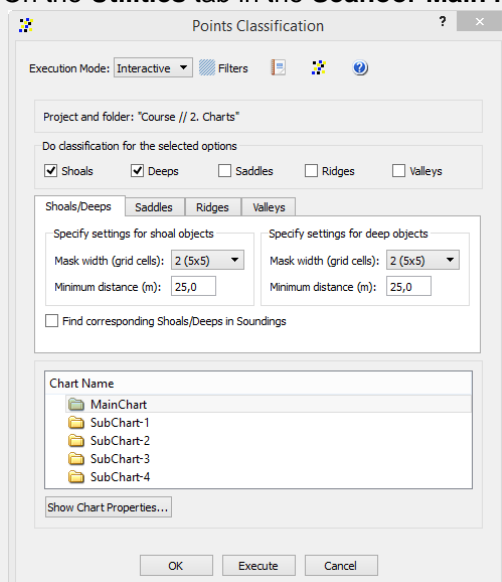
Exercises

- 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 **Increment**.
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 **Increment**.
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
- Convert an image to another UTM zone

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**.
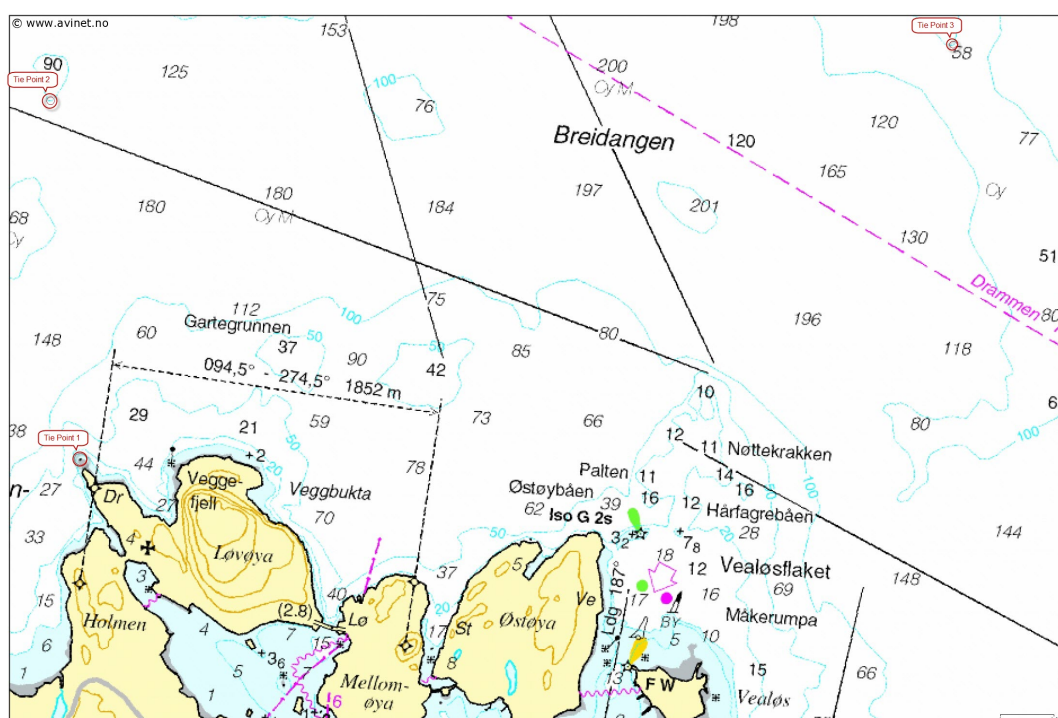
A map about to be georeferenced.

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
```
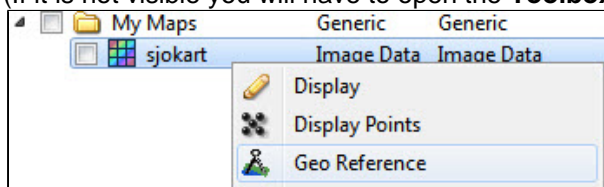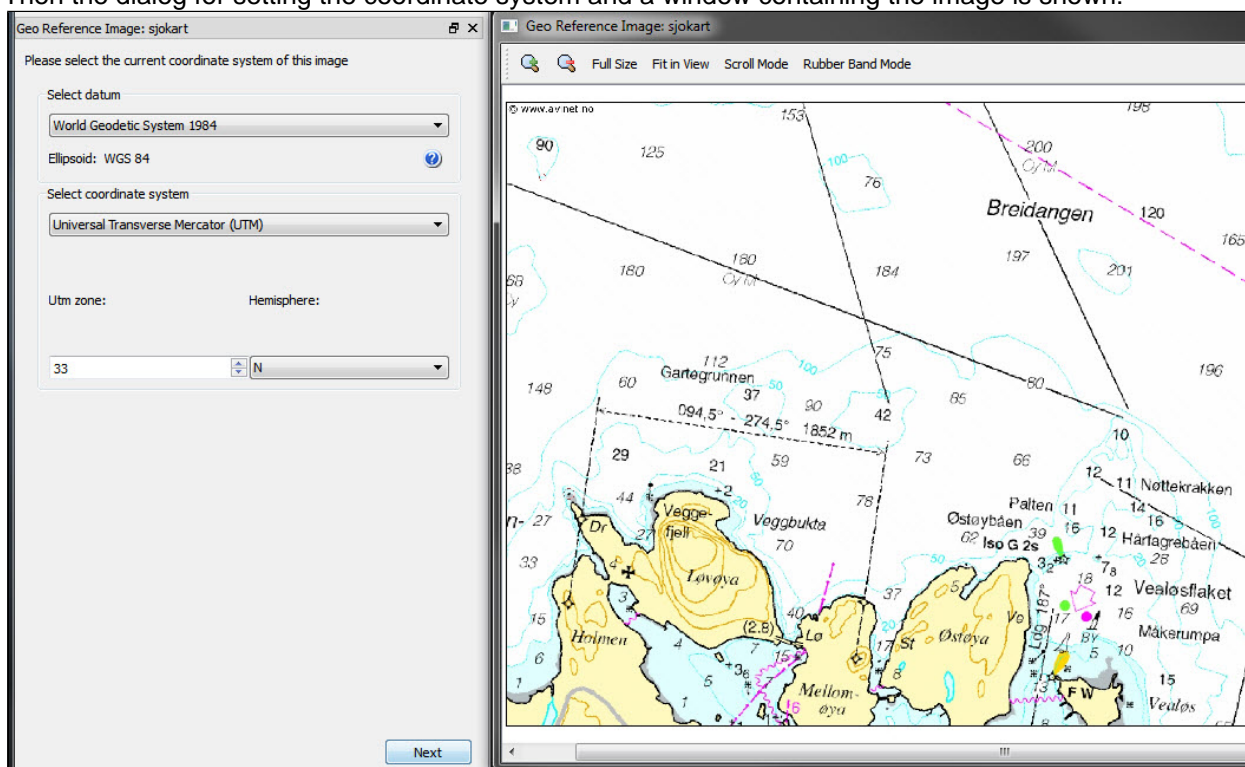
Georeferencing step by step

- 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.
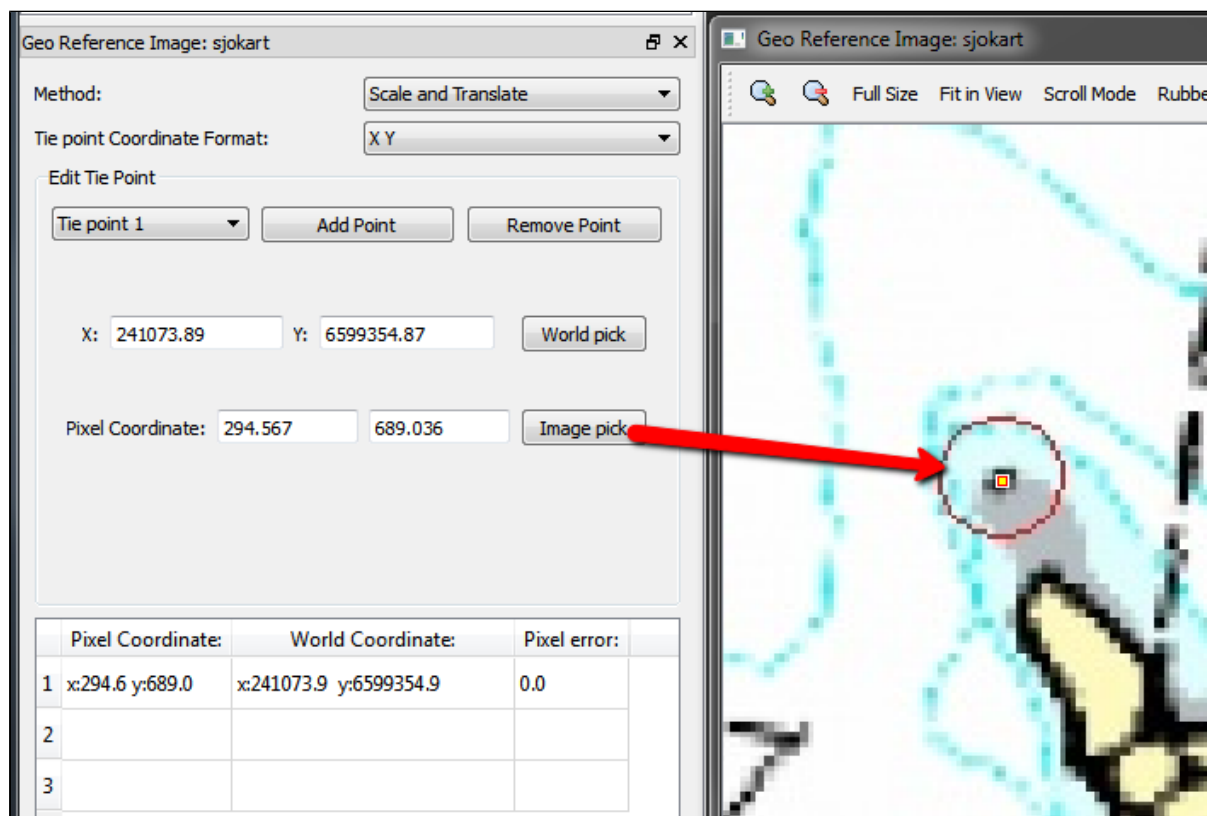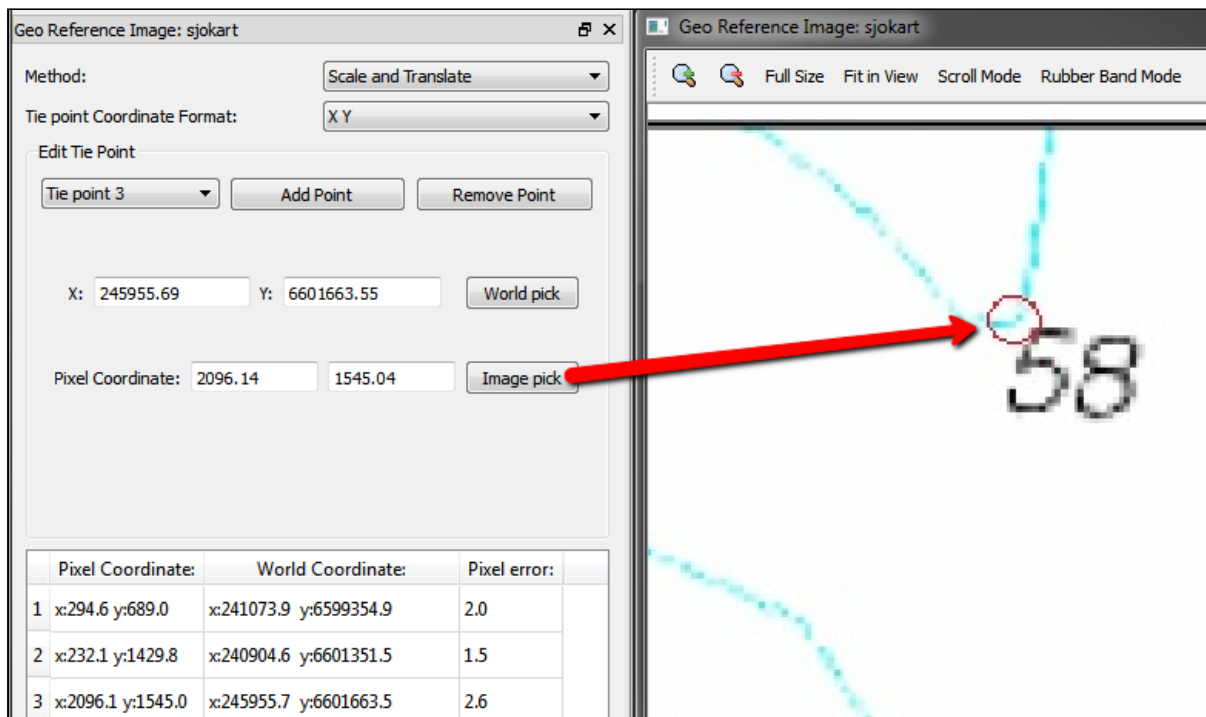

- Change to **Tie point 2**.

- 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.
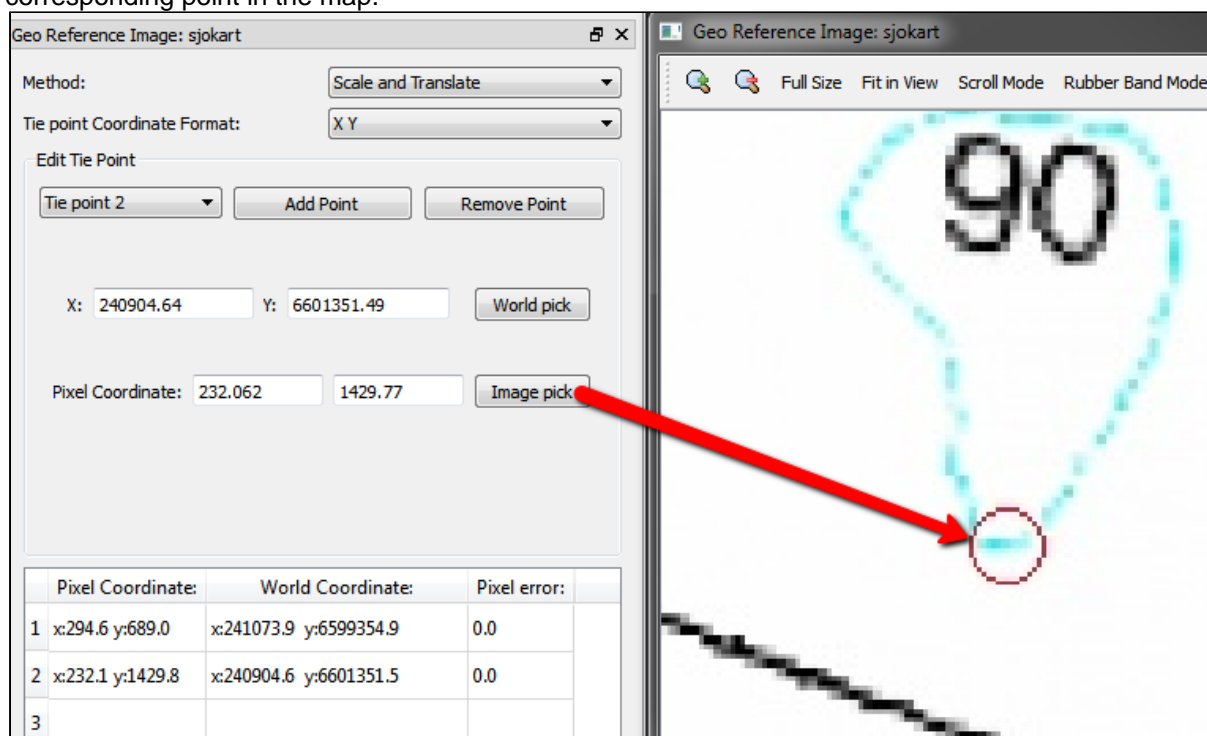
- Change to **Tie point 3**.

- 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*.

Exercise

Convert an image to another UTM zone

1. 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**.)
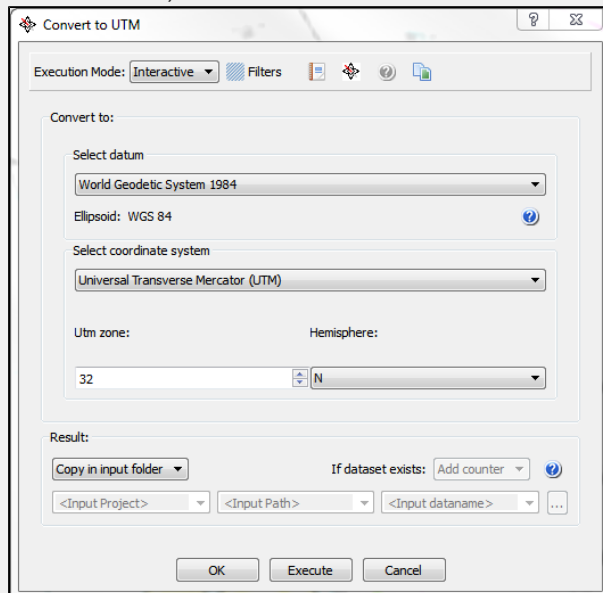
2. Set the new geodetic properties for the image: **WGS84, UTM Zone 32 N**
3. Specify how to save the new image for e.g. **Replace**, **Copy in input folder** or **Specify** name and where to store it.
4. Then click **Execute**.
5. Display the new image in the main graphic window, and also display the multibeam data or the surface created from the multibeam data.

Q. 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** ic on.
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** a nd pasted into the workflow.

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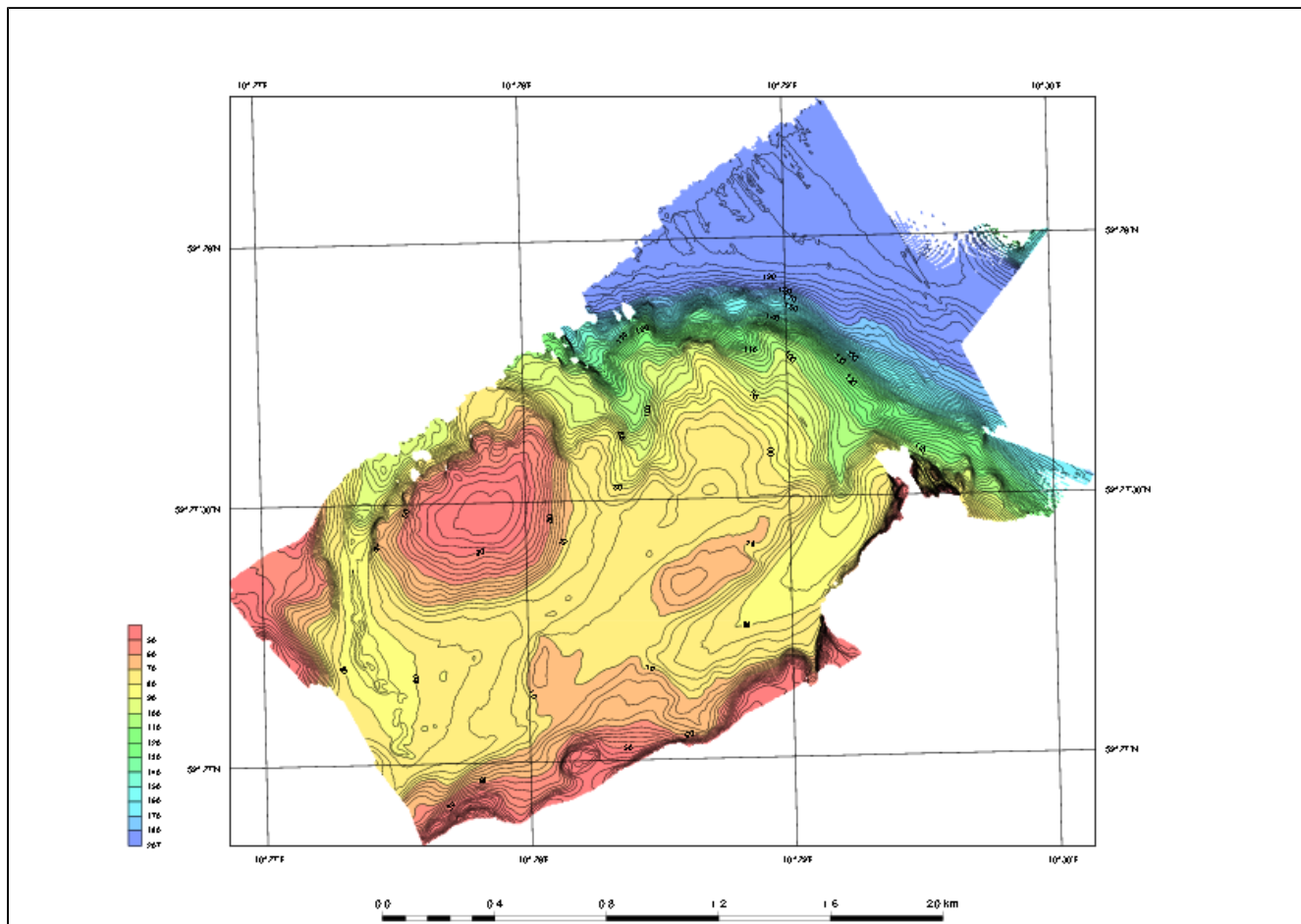
Note that you will have to install Ghostview and Ghostscript in order to make the plotting system work.

Exercises

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

Open and run a saveset in the plotting system

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.

Explore the different plotting options

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.

Notes

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____