Seafloor Tutorials

Seafloor
Hydrographic toolbox

Ge●cap

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 cover:

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 the 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 Seafloor_Demo 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**
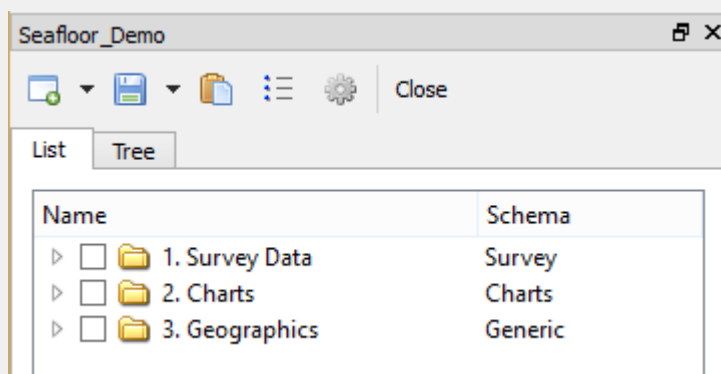6. Set the **Time interval** to **5**
7. Click **OK**

-

Geocap Project

The Seafloor_Demo project is built using the Seafloor project template. This template consists of three main top level folders; 1. Survey Data, 2. Charts, 3. Geographics. The number ahead of the folder name is to have a logical sorting in the project tree. The Survey Data folder contains the imported data, like multibeam data, xyz points (soundings) or single beam data. Within the Charts folder the survey data can be extracted and processed within certian areas of interest. We are now going to open the Seafloor_Demo project and look closer at this project structure.

Exercise

Open the Seafloor_Demo project

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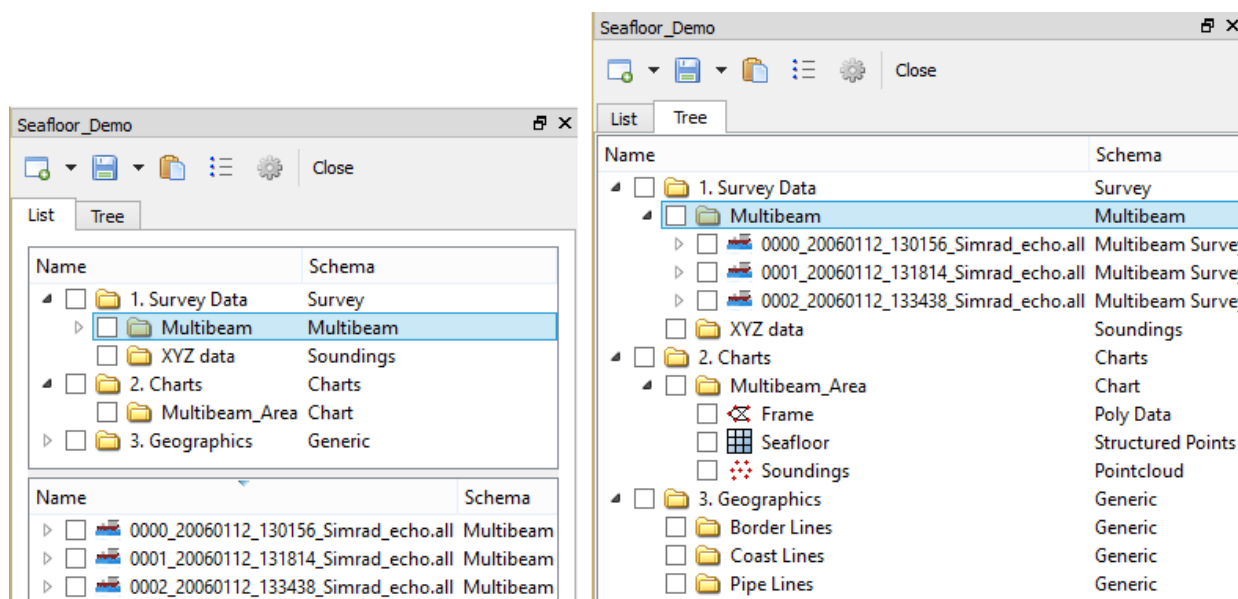3. Select the **Seafloor_Demo.db** file and click **Open**. A window similar to the one below will appear.

Seafloor_Demo		
List	Tree	
Name		**Schema**
▷ ☐ 📁 1. Survey Data		Survey
▷ ☐ 📁 2. Charts		Charts
▷ ☐ 📁 3. Geographics		Generic

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 datasets or other folders.

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 Seafloor 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
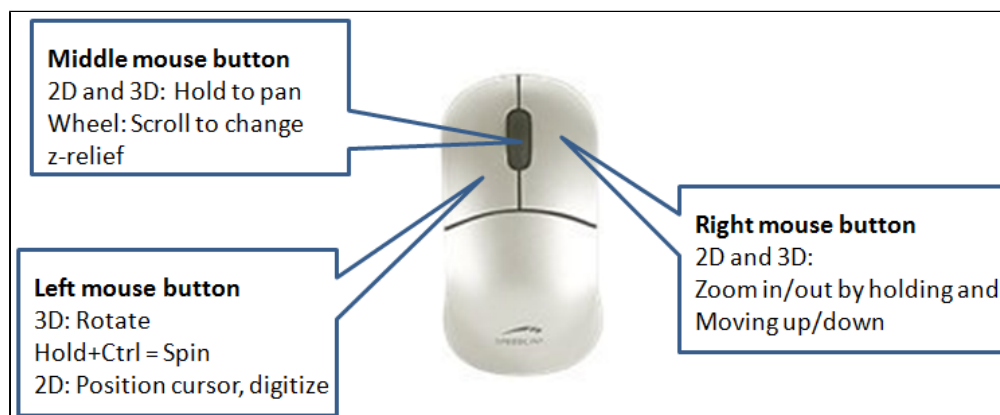
Display the various datasets in the project

1. Locate a grid dataset (structured points) e.g. **2. Charts/Multibeam_Area/Seafloor**
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**, **Lod Grid Display**.
4. Display some of the lines (Boundary and Frame) under the **Multibeam_Area** charts 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 upper level folder **2. Charts**, or **Multibeam_Area** 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

Exercise

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.
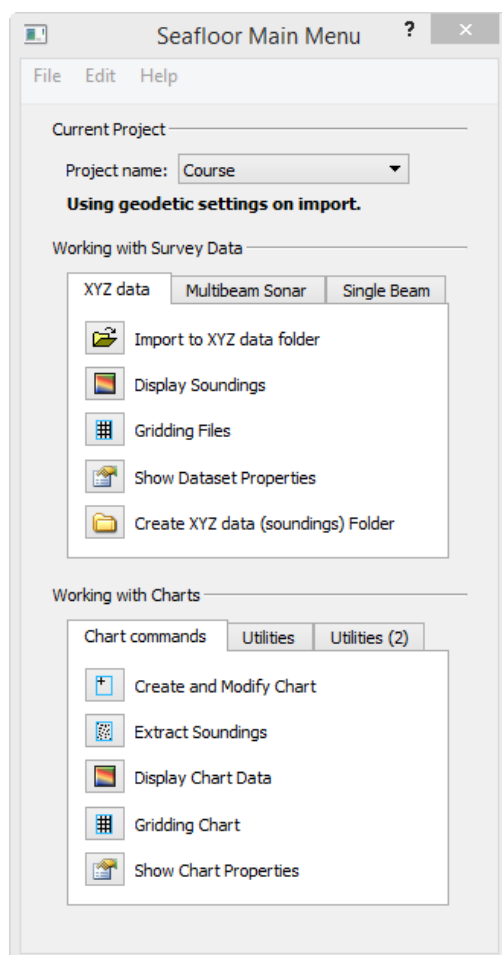
B. Seafloor Main Menu (Extract from User Guide)

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.
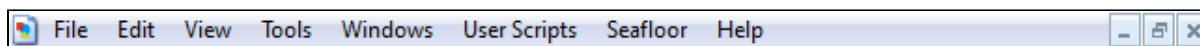
The Seafloor Main Menu.

-

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.
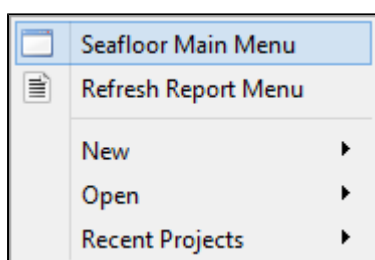The Seafloor package is loaded either on startup of Geocap, or on the **Tools** menu, click the **Options** menu. In the menu that appears, click **Plug-ins**, and then select **Seafloor** in the right hand part of the menu.

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.
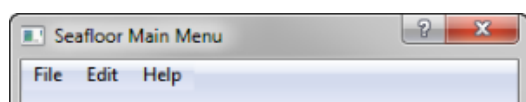
The Content of the Seafloor Main Menu

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.
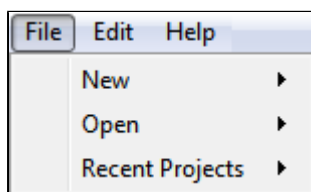
Menu bar options (File, Edit, Help)

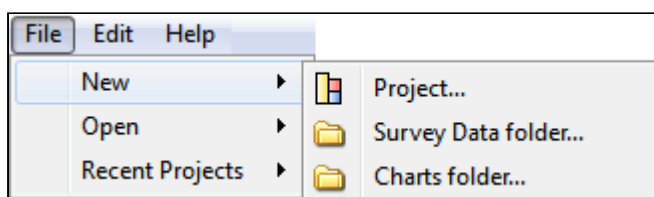The upper part of the menu is a menu bar with the entries **File**, **Edit** and **Help**:

The Menu bar

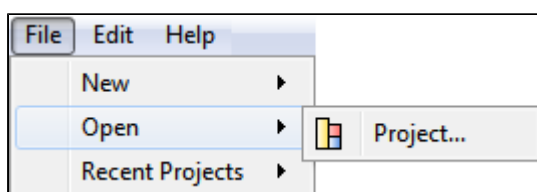For the actions from menu bar entries, see below.

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.
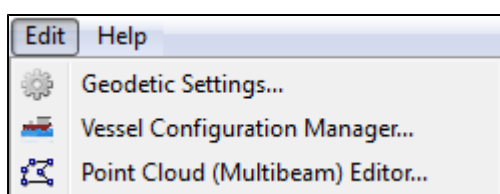

- **Open - Project...** will open any existing Geocap projects in the computer or network.
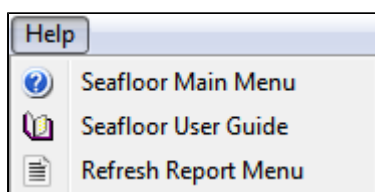

- **Recent Projects** will give a list of recently opened Geocap projects.

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.
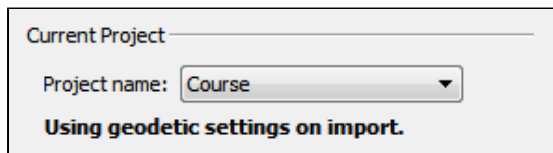
Help

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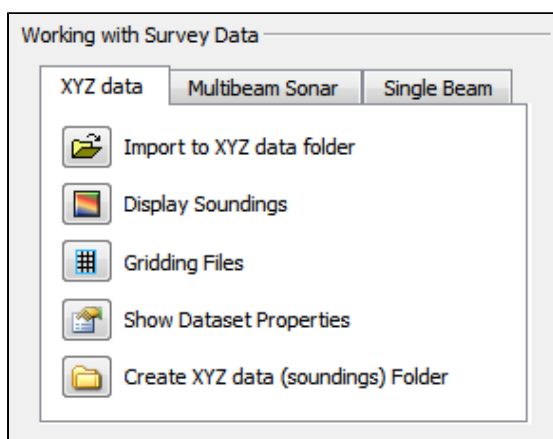
The text **Using geodetic settings on import** indicates that during import of any dataset into Geocap, the data will be transformed to the project coordinate system. This action is set up in the *Geodetic Settings* menu.

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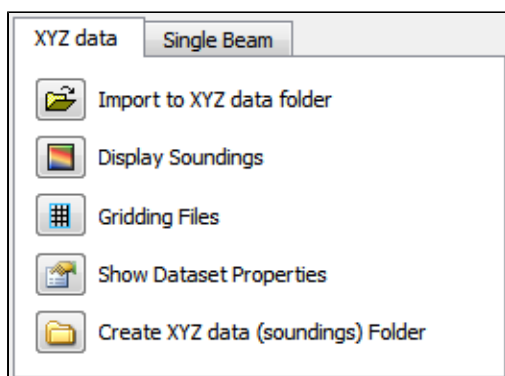
- The XYZ data tab
- The Multibeam Sonar tab
- The Single Beam tab

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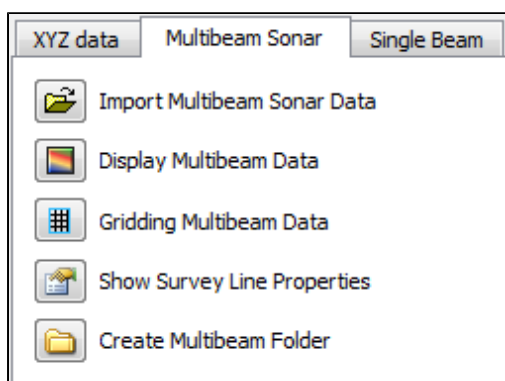
Show file properties: This command will show dataset name, bounding box and number of points of the data in folders of type **Soundings**.

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
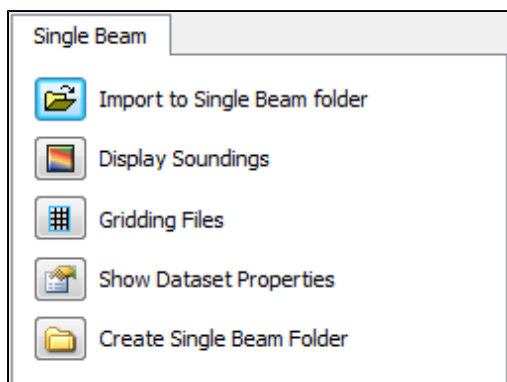
folders of type **Multibeam**.

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**.

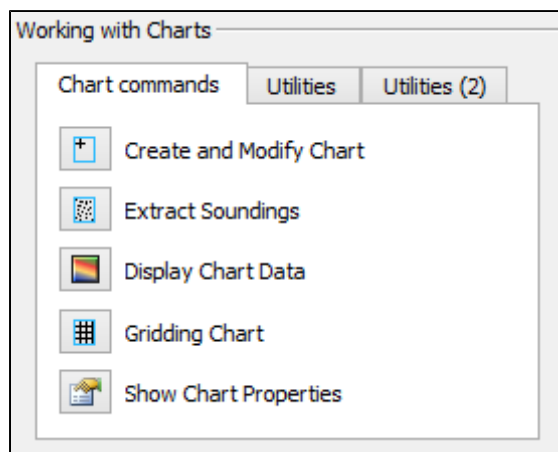**Gridding Files**: This command will make it possible to do gridding (terrain modelling) of the data in folders of type **Single Beam**.

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
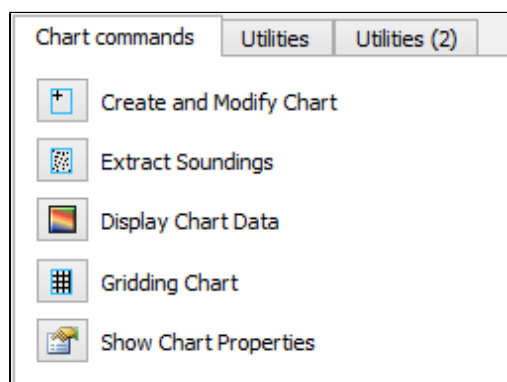

The Working with Charts part

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

- The **Chart commands** tab
- The **Utilities** tab
- The **Utilities (2)** tab (Not enabled for Shelf)

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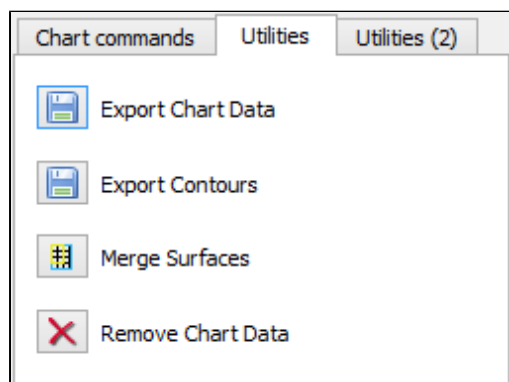
Show Chart properties: This command will show properties of the content of folders with type **Chart**.

The Utilities tab

This part contains buttons with commands for analysis of data in a **Seafloor** project. All commands work on folders on type **Charts** and on folders of type **Chart**.
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**.

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 (2) tab

This part contains buttons with commands for analysis of data in a **Seafloor** project. All commands work on folders on type **Charts** and on folders of type **Chart**. This utility is not enabled for Shelf,

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

The Utilities (2) tab

Make Outline and Holes: This command will create the outline (boundary) and interior hole polygons for a chart **.** The polygons can be created from the **Seafloor** (DTM) dataset or from the **Soundings** (points) dataset.

Make Profile: This command will create a continuous depth profile (intersection) through the Seafloor (DTM) data set in folders of type Chart.

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**.

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**.

Statistical Point Reduction: This command will reduce the amount of points from one or more input datasets. The selection method can for instance be priority of the **Minimum** or**Maximum** depth values. The selection area can be by looking at points inside a **Rectangle** or **Circle**. The selected points are always with the original position and not snapped into any grid structure.

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

Exercise

Import XYZ data

1. Click the **XYZ data** tab in the **Seafloor Main Menu**.
2. Click **Import to XYZ data folder**.
3. Click **Generic reader**.
4. Click the browse 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 browse 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.

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

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

Total for all files

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

Showing Survey Line Properties

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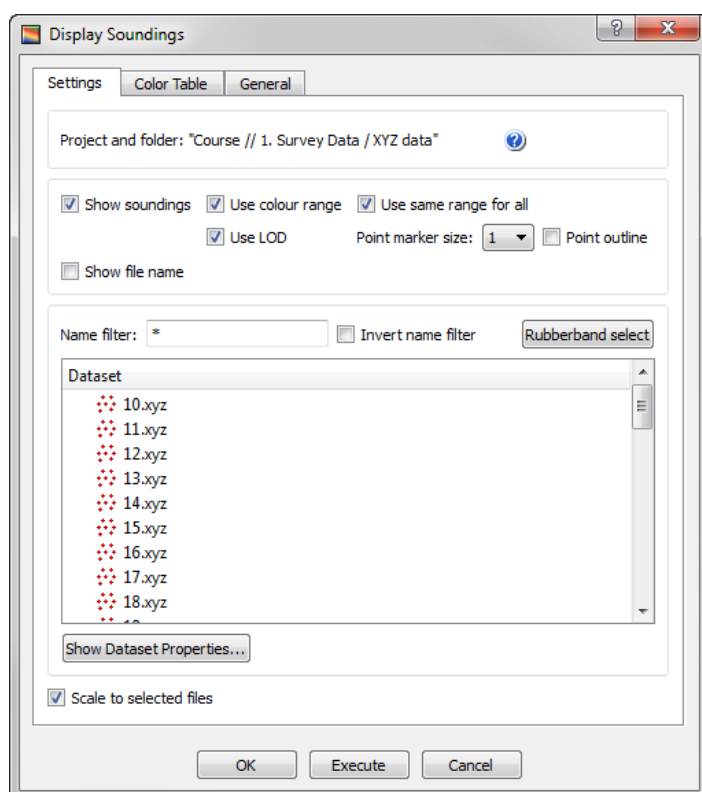- Display filtered data

-

XYZ Data

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.
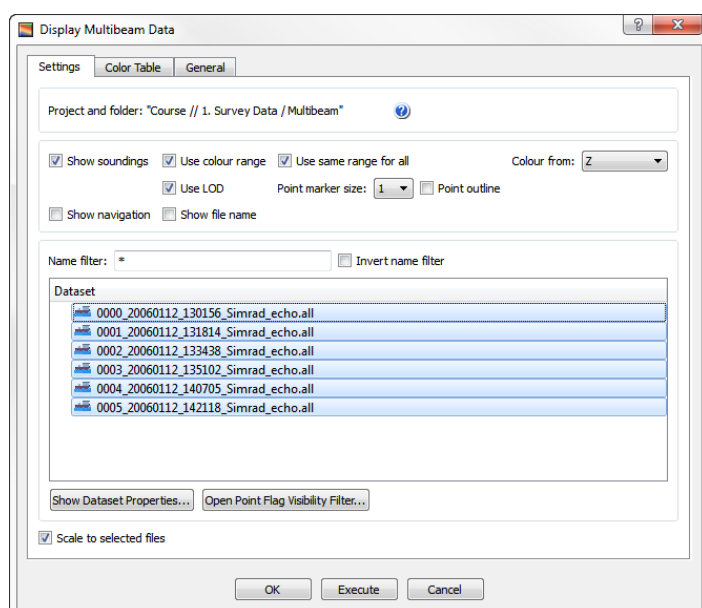
-

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 **actors**. **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 Geocap modules contains several schemas. For example some of the schemas used in the Shelf Module are **coast line**, **base 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 **In teractive** 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

Icons	Description of Main Toolbar icons
	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.
2D	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.
3D	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 each chart.

The **Chart operations** are commands executed on folders of schema type **Chart**, located on folders of schema type **Charts**.

The **Charts** folder contains several menu items on the right click menu. The tools for charts are located in the upper part of the menu and listed in the graphic below.

	Create and Modify Chart...
	Display Chart Data...
	Export Chart Data...
	Export Contours...
	Extract Soundings...
	Gridding Chart...
	Make Outline and Holes...
	Make Profile...
	Merge Surfaces...
	Points Classification...
	Remove Chart Data...
	Statistical Point Reduction...
	Show Chart Properties

The Charts folder menu

In this excercise we will look at how to **create**, **display** and **modify** charts.

Exercises

- Create a single chart from the Multibeam data
- Create a single chart from the XYZ data
- Create sub-charts as matrix of charts
- Populate charts with data
- Show properties for each chart
- Display chart frames
- Display chart soundings
- How to modify chart definitions
- How to remove a chart with the Create and Modify Charts tool

-

Create a single chart from the Multibeam data

Exercise

Create a single chart from the Multibeam data

1. Click the **Chart commands** tab in the **Seafloor Main Menu**.
2. Click the **Create and Modify Chart** button.
3. In the **Create Single Chart** tab, type *MainChart* 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.

Example of chart frame types: Single Chart, Matrix of Charts and Charts along Line.

-

Create a single chart from the XYZ data

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

Create a single chart from the XYZ data

1. In the **Create Single Chart** type in *XYZ_data* as **Chart name**.
2. In the **Create Single Chart** tab click **Scale to Survey Data**.
3. In the dialog that pops up, tick the **XYZ data** folder checkbox and click **OK**. The menu should now look similar to the one below:

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

</td></tr>
</table>

It is also possible to create charts by the methods:

- Get coordinates from the active window (Scale to active window).
- Specifying lower left and upper right coordinates.
- Specifying lower left corner coordinates, width and height.
- Drawing a rectangle in the graphics window (Rubberband select).
- Create Matrix of Charts.
- Create Charts along a line.

-

Create a matrix of charts

We will now use the **MainChart** as a master chart for the whole area of your multibeam survey. We are now going to create sub-charts from this chart.

> The amount of points we work with in this section is not very huge, so for this amount of data we would not made the matrix of charts. We will recommend this method when the amount of points in the project is several hundreds of million, also dependent on the Geocap version (32 bit or 64 bit) and the computer's memory configuration.

Exercise

Create sub-charts as matrix of charts

1. Click the **Create Matrix of Charts** tab.
2. Make sure that your **MainChart** 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. Note that the **Matrix specification** is **By number of columns and rows** has been selected with number of **Columns** and **Rows** set to **2**, the corresponding **Width** is **1547,50** and the **Height** is **1337,50**.
5. To have nicer chart sizes change the **Matrix specification** to **By chart geometry**, and set the **Width** to **1550** and **Height** to **1340**. Note that the number of **Rows** and **Columns** of charts automatically will change when you change the height and width.
6. The menu should now look similar to the image 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 **MainChart** folder, one **XYZ_data** folder and four **SubChart-x** folders inside the **2. Charts** folder.

-

Populate charts with data

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 data
- Click **Extract Soundings** on the **Chart commands** tab in the **Seafloor Main Menu**.   - In **Margins** type in **5** m to avoid any loss of data between the charts. (Dependent on the gridding increments.)   - Tick both the **Multibeam** checkbox and the **XYZ data** checkbox.   - Select all the **SubChart-x** in the **Chart Name** section. Also select the **XYZ_data** chart.

<ol start="5"> - Click **Execute** and the extraction process will start.   - Click **OK** in the dialog the pops up.   - Click **OK** on the **Extract Soundings** menu.
Each sub-chart will now contain a dataset named **Soundings** in addition to the **Frame** dataset.

-

Chart properties

Exercise

Show properties for each chart

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

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

Project and folder: "Course://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 Z Min	Points Z Max
1	MainChart	582 080,000	585 175,000	6 590 780,000	6 593 455,000			3 095,000	2 675,000									
2	SubChart-1	582 080,000	583 630,000	6 590 780,000	6 592 120,000			1 550,000	1 340,000						5,00	1 840 179	19,82	100,07
3	SubChart-2	583 630,000	585 180,000	6 590 780,000	6 592 120,000			1 550,000	1 340,000						5,00	605 570	12,45	171,34
4	SubChart-3	582 080,000	583 630,000	6 592 120,000	6 593 460,000			1 550,000	1 340,000						5,00	420 787	39,49	201,27
5	SubChart-4	583 630,000	585 180,000	6 592 120,000	6 593 460,000			1 550,000	1 340,000						5,00	564 116	2,99	260,57
6	XYZ_data	600 590,000	606 590,000	6 446 150,000	6 452 060,000			6 000,000	5 910,000						5,00	3 531 785	169,18	319,01

Total number of charts: 6

OK

Showing Chart Properties

Display chart data

Exercise

Display chart frames

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

Exercise

Display chart soundings

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

-

Modify chart

The Modify Chart tab can be used to rename the chart or change the definitions of the chart frame.

Exercise

How to modify chart definitions

1. Click the **Chart commands** tab in the **Seafloor Main Menu**.
2. Click the **Create and Modify Chart** button.
3. Open the **Modify Chart** tab and look at the content.
4. Click the **Chart to modify** button and select the **XYZ_data** chart.

5. In the **Chart Frame** part, change the **Height** from 5910 to **5900** (meters).
6. Click **Execute**.
7. Click **Yes** on **Do you really want to modify chart "XYZ_data"?** and **OK** in the information box which appears next.
8. Click the **Show Chart Properties** button up to the right for all the tabs to verify the changes.

When the chart frame is changed it can be necessary to repeat the **Extract Soundings** operation. In the example above the change is so small that we do not want perform a new extraction of points. We have also made the area smaller so we will not need to get more points for filling into the chart frame.

If we had been through the gridding of the chart we should consider to redo the gridding operation within the new chart frame.

-

Manipulate charts in the Miscellaneous tab

In the Miscellaneous tab there are som additional tools for the charts:

- **Rename charts**: Will rename the selected charts (chart series) to new charts series name and number.
- **Export chart definitions**: The chart names and chart extensions (frame) can be exported to an ascii file.
- **Import chart definitions**: The chart names and chart extensions can be imported from an ascii file.
- **Delete charts**: The selected chart(s) will be removed from the project.

Exercise

How to remove a chart with the Create and Modify Charts tool

1. Click the **Chart commands** tab in the **Seafloor Main Menu**.
2. Click the **Create and Modify Chart** button.
3. Open the **Miscellaneous** tab and look at the content.
4. In the **Mode** part, select the **Delete charts** option.
5. In the **Chart Name** list, select the **XYZ_data** chart:

6. Click **Execute**.
7. Click **Yes** on **Do you want to delete 1 chart?** and **OK** in the information box which appears next.
8. Notice that the **Chart Name** list is updated and the XYZ_data chart has disappeared.

The command **Remove Chart Data** have some other options for the charts. The command can remove specific chart items like Boundary, Holes, Seafloor, Soundings etc.

The command can also remove **Empty** charts, i.e. charts which are missing **Soundings** after **Extract Soundings.**

The command can also remove the entire chart(s), like the **Delete charts** option described in the exercise above.

It is also possible to remove charts simply by deleting the chart's folder from the **2. Charts** main folder.

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

-

Gridding Multibeam Data

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 **2.5** and keep the **Use YInc=XInc** option.
4. In the **Algorithm Parameters** tab set **Search radius** to **5** and verify that the **Unit** is set to **Grid node(s)**.
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**.

-

Gridding data in 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 **2.5** and keep the **Use YInc=XInc** option.
5. In the **Algorithm Parameters** tab set **Search radius** to **5** and verify that the **Unit** is set to **Grid node(s)**.
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 factor** and **Deep factor** to **3**.
10. Select all the **SubChart-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 three new datasets: **Seafloor**, **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 **SubChart-x** (click first and use shift + click on last).
6. Click **Execute**.

Exercise

Display rejected soundings

1. Click the **X** 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 **LOD Points Display**.

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 browse button and select the **2. Charts / MainChart** folder to save the merged surface in.
3. Keep the default option for **Specify surface increments** (Use increments from first grid).
4. Select all the subcharts and move them over to the right column by clicing the **>>** button. 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 / MainChart** 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 browse 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 **Surfa ceFiltered**.
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.
- Un-delete them.
- Flag them according to a user defined 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 **X** 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**.

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

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

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

Exercise

Edit the size, direction and position of the profile

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

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

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
- Create new project
- Load multibeam sensor data
- 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**.

Preparations

Before we can load the ASCII sensor data we need to have a project with multibeam data.

Exercise
Create new 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 the project name **EM2040_WGS84_UTM_60S** into the **Name** field.   5. Click the **Browse** button, and select where you want to store your project on your hard disk.   6. Click **Finish**.   7. You are now be prompted with a window asking you if you want to define the geodetic setting of the project.   8. Click **Yes**.   9. Check the **Use geodetic settings for project** box and select **WGS84** as datum and **UTM Zone 60 S** as coordinate system.   10. Click **OK**.

Exercise

Load multibeam sensor data

1. Click the **Multibeam Sonar** tab in the **Seafloor Main Menu**.
2. Click **Import Multibeam Sonar Data**.
3. Click the browse icon and browse to the folder **...\Seafloor_Course\Multibeam Sensor Data\EM_2040\South_seas_wreck_closeup**.
4. Select all 13 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**.
8. Display the multibeam survey lines by the command **Display Multibeam Data**.

Import Tide file

Exercise

Import Tide file

1. Right click the **Multibeam** folder under **1. Survey Data** and select **Import ASCII Sensor...**
2. Click the browse icon and browse for the file **predictedtide.tide** in the folder **...\Seafloor_Course\Multibeam Sensor Data\EM_2040\tide**.
3. Make sure that **Tide** is selected in under **Sensor type**. Let the option **Reverse sign** be un-checked.
4. Select all the Survey lines.
5. Click **Execute**.
6. Re-display by the command **Display Multibeam Data**. Do you see any differences?
7. Import the tide file predictedtide.tide once more on all the files. This time set the option **Reverse sign** checked.
8. Re-display by the command **Display Multibeam Data**. Do you see any differences?

Import Navigation file

Exercise

Import Navigation file

1. Right click the **Multibeam** folder under **1. Survey Data** and select **Import ASCII Sensor...**
2. Click the browse icon and browse for the navigation file **merged_navigation_south_seas_wreck_closeup.txt** in the folder **...\Seafloor_Course\Multibeam Sensor Data\EM_2040\navigation**.
3. Make sure that **Navigation** is selected in under **Sensor type**.
4. Select all the Survey lines.
5. Click **Execute**.
6. Re-display by the command **Display Multibeam Data**. Do you see any differences?

N. Export Multibeam Data

Introduction

Seafloor lets you export multibeam survey data as Points and Navigation.

There is a wide range of export possibilities and formats such as, VTK, Esri Shapefile, ASCII Column, VTK ASCII and Dxf ASCII.

Exercises

- Export Multibeam Points
- Export Navigation data

Export Multibeam Points

Exercise
Export Multibeam Points

1. Right click the **Multibeam** folder under **1. Survey Data** and select Export **Export Multibeam Data Sensor...**
2. Click the **?** icon and read the help text.
3. Click the **Export Points** tab.
4. Select the two first survey lines in the **Survey Line** part.
5. In the **Export Format** part, select the **ASCII Column** format.
6. Click the **Ascii export settings** button to open the format selection dialog. Click the column datatypes (**X**, **Y** and **Z**) too see the choices. Then close the dialog with **Cancel**.
7. In the **Output file** part select the folder name for the output.
8. Click **Execute**.
9. Check the result in the selected output folder.

Export Navigation data

Exercise
Export Navigation data

1. Click the **Export Navigation** tab.
2. Select the two first survey lines in the **Survey Line** part.
3. In the **Export Format** part, select the **ASCII Column** format.
4. Click the **Ascii export settings** button to open the format selection dialog. Note the column datatypes **DateTime**, **X**, **Y** and **Z**. Close the dialog with **Cancel**.
5. In the **Output file** part select the folder name for the output.
6. Click **Execute**.
7. Check the result in the selected output folder.

O. Chart Utilities

Introduction

In Seafloor there is a lot of functionality to create more results and analysis. Some of the functionality mentioned in this section is started from the **Seafloor Main Menu**, or from the **Charts** folder, or from a specific data type, like **Structured Points**.

Exercises

- Make Outline and Holes
- Quick digitizing
- Make Profile

Make Outline and Holes

This command will create outline polygons (boundary) and/or interior holes from either points or surfaces in the selected chart(s). The command has options to make the polygons from **Soundings** or **Seafloor** data objects in the charts.

Exercise
Make Outline and Holes
1. In the **Utilities (2)** tab in the **Seafloor Main Menu**, click **Make Outline and Holes**.   2. Inside the frame named "Make polygons from", click the option **Seafloor**.   3. Check the box ahead of **Outline**, and use the option **Keep all outline polygons**.   4. Check the box ahead of **Holes**, and use the option **Keep all holes polygons**.   5. In the Chart Name part, select the MainChart (or another chart).   6. Click **Execute**.   7. Verify the new data sets **Boundary** and **Holes** in the chart folder.

Repeat the exercise by trying the option **Remove small outline poygons** and turning **off** the option to make the Holes polygons.

Also make the outline from the **Soundings** dataset.

> The command can also be executed from any folder of type **Generic**, **Soundings**, **Single Beam** or **Mult ibeam**. Select either **Points** or **Surface** to make polygons from. The command will not merge the input data but process each input dataset individually.
>
> The command can also be executed from any **Structured Points** or **Poly Data/Pointcloud** dataset.

-

Digitizing

Digitizing is one of the most important ways the user can interact with Geocap to create new dataset. In contrast to typing in numbers in an editor, the digitizer gives the user the ability to interact with graphically displayed data.

Exercise

Quick digitizing

1. Make sure you have displayed all relevant data before you start the digitizing. Also check if the graphics window is in **2D** mode.
2. The **Quick digitizer** is started from **Tools > Quick Digitizing**.
3. Check the box ahead of **Use rubber line** in the lower part of the menu. The rubber line can be turned on and off during the digitizing by pressing the **b** key on the keyboard.
4. Click **Start digitizing** and use the left mouse button to digitize.
5. Click **Connect to start** if you want to close the polygon, or press the **h** key (=home) on the keyboard.
6. To start a new line segment, press the **f** key (=insert flag) on the keyboard.
7. Click **Stop digitizing** to save the line/polygon.
8. The digitized_line is now in the workspace. To save the data set in the project, select a folder and right click and select **New > Workspace Data > digitized_line**.

A digitized line is saved in the workspace by default. Use the **Save digitized line also in folder** option to save it directly in the project. Just tick the check box and browse in the folder you would like to save it to.

-

Make Profile

This command will extract continuous profile data based on a line crossing the selected chart(s). The command has two options, make one **Along profile** or multiple **Across profiles**. The profiles will be seamless over the edges of the input charts.

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

Make Profile

1. First use the **Quick Digitizer** to create a profile line over the actual surface(s). It is also possible to import an existing line from a disk file.
2. In the **Utilities (2)** tab in the **Seafloor Main Menu**, click **Make Profile**.
3. In the project folder, locate the profile line (for example *digitized_line*). Then, in the Profile Data part, click the green arrow to the left of the text field to select the line.
4. Select the Mode **Along Profile**. and set the parameters **Point spacing (m)** to **1,00** and **Start value (m)** to **0,00**.
5. Select all the sub-charts. in the Chart Name part.
6. Click **Execute**
7. This will create the dataset **digitized_line_alongProfile** in the same folder as the XY profile is located.
8. Change the mode to Across Profiles. Set the parameters **Profile spacing (m)** to **10,00**, **Profile length (m)** to **50,00**, **Point spacing (m)** to **1,00**, and **Start value (m)** to **0,00**.
9. When clicking **Execute** the dataset **digitized_line_acrossProfile**.
10. Display both lines in the 3D window.
11. Also open the Table View for each of the new datasets to look at the data.

</td></tr>
</table>

Along and Across profiles shown on the surface.

P. 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 exercise 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 (2)** 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 excercise by changing the search radius. Also try to check the box **Find corresponding points in Soundings**.

Q. Statistical Point Reduction

Introduction

This command will reduce the amount of points from one or more input data sets. The selection method can for instance be priority of the **Minimum** or **Maximum** depth values. The selection area can be by looking at points inside a **Rectangle** or **Circle**. The selected points are always with the original position and not snapped into any grid structure.

The output will be a new point data set which in Geocap will have a schema of type **Pointcloud**. The command is a folder command and is implemented for folders with schema types **Charts**, **Multibeam**, **Soundings** and **Generic**.

Seafloor can select the points within the following categories:

- Minimum
- Maximum
- Shallower than Mean
- Deeper than Mean
- Closest to Mean

Selection of shallowest point within 250 m radii.

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

Exercises

- Select points within Circle
- Display selected points

Exercise

Select points within Circle

1. On the **Utilities (2)** tab in the **Seafloor Main Menu**, click **Statistical Point Reduction**.

2. Check the selection area **Circle**.
3. Set radius **25** meters.
4. Check the Point selection methods **Minimum** and **Maximum**.
5. Note that the uncheckable boxes are for the Rectangle selection area.
6. Select the main chart.
7. Click **Execute**.

The resulting datasets can be found in the specified chart folder a with the name **Soundings_<method>_Radius_<radius>**.

Exercise

Display selected points

1. Browse to the folder for the actual chart.
2. Display the dataset **Soundings_Minimum_Values_Radius_25** by right click and select **LOD Points Display**.
3. Display the dataset **Soundings_Maximum_Values_Radius_25** by right click and select **LOD Points Display**.

Repeat the last excercise by changing the radius. Also try to run the command with the selection area **Rectangle**.

R. Surface Volumetrics

Introduction

In Seafloor there is a lot of functionality to create more results and analysis. Some of the functionality mentioned in this section is started from the **Seafloor Main Menu**, or from the **Charts** folder, or from a specific data type, like **Structured Points**.

Exercises

- Surface Volumetrics

Surface Volumetrics

The command **Surface cut/fill volumetrics** calculates the cut and fill volumes between two surfaces or between a surface and a polygon data set representing the theoretical terrain.
The cut volume can be seen as the volume to be dredged (positive volume), whereas the fill volume is the volume to be filled (negative volume).

The command reports the volume and the area (horizontal projection) of the cut and fill volume in a text item.

In addition this command can produce the following products:

- **Cut surface**
- **Difference surface**
- **Terrain with cut**
- **Terrain with fill**
- **Terrain with cut and fill**
- **Cut area**
- **Fill area**

Exercise

Surface Volumetrics

1. First use the **Quick Digitizer** to create a closed polygon where the volume is to be calculated. It is also possible to import the polygon from a disk file. Use the underwater shoal in the MainChart as the feature to volume calculate.
2. Create the polygon in the MainChart folder by **New > Workspace Data > digitized_line**. Rename the polygon to the name **digitized_closed_polygon**.
3. It is possible to diqitize the polygon with a depth value, but the depth value can also be assigned by the **Polydata Function** command. From the item **digitized_closed_polygon,** select **All Commands > Shared Commands > Operations > Polydata Function...** . Set function **z = 60** and click Execute.
4. Rename the new polygon to **digitized_closed_polygon (60m)**.
5. From the **Seafloor** icon in the MainChart folder, click **Surface cut/fill volumetrics**.
6. In the **Surface cut/fill volumetrics** dialog, insert the **digitized_closed_polygon (60m)** into the Cut surface/polygon input field. (Mark in the project and then use the green arrow to the right.)
7. Turn on the options **Create cut surface**, **Create terrain with cut** and **Create terrain with cut and fill**.
8. Click **Execute**.
9. The results from the volume calculation is presented in the Report menu. In the MainChart folder a new folder **Volumetrics** is created containing the produced data sets, **CutSurface**, **TerrainwithCut** and **TerrainWithCutAndFill**.
10. Display the datasets and the polygon to see the difference.
11. Change the depth level for the polygon to 70, as in the steps **3** and **4**, and repeat the steps **6 to 10**.

The actual surface and the digitized closed polygon.

The cut surface and the digitized closed polygon at level 60 m.

The terrain with cut and fill and the closed polygon.

S. Export Data from Charts

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 browse 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 browse 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

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

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 browse 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.
</td></tr>
</table>

Contours

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

Export contour lines to ASCII

1. On the **Utilities** tab in the **Seafloor Main Menu**, click **Export Contours**.
2. Check that the **Export as: Contour Lines** radio button is ticked.
3. Select the **Master Chart**.
4. Set **Contouring method** to **Incerement**.
5. Set **Increment (m)** to **1.00**.
6. Also consider the parameters **Use min level**, **Use max level**, and **Invert Z**
7. Check the **Save to** option **File and project**.
8. Select **ASCII Column** as **Export Format**.
9. Click the browse 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.
</td></tr>
</table>

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

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
</td></tr>
</table>

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 browse 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 browse 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**.

T. 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
```

Exercise

Georeference an image

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

Georeferencing step by step

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

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

- Set geodetic properties of the chart: **WGS84, UTM Zone 33 N**.

- Click **Next**.

- Set **Method:** to **Scale and Translate**.

- Set **Tie point Coordinate Format:** to XY.

- Type in the **X = 241073.89** and **Y = 6599354.87** for the first tie point, click **Image pick** and click on the corresponding point in the map.

- Change to **Tie point 2**.

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

- Change to **Tie point 3**.

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

- Evaluate the result. The **Pixel error** represents the accuracy of the georeferencing. The value should not exceed more than 1 pixel on each point.

- Click **Finish.**

- Now the image must be transformed from *UTM Zone 33* to *UTM Zone 32*.

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** na me 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.

U. 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
- Build the workflow
- Create a flight tour

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**.

Workflows are saved as single files on disk with file extension *.**gwf**, (Geocap Work Flow). It is a good idea to store them in a folder called **workflows** next to the **db** folder on disk. In the next exercises we will create a new empty workflow.

Exercise

Create a new workflow

1. Click **File > New > Workflow**.
2. Browse for the **workflow** folder or create a new one.
3. Give your workflow a name and click **Save.**
4. The new empty workflow window will pop up below the project.

We are now going to add different commands to our workflow. There are a number of ways to do this. You can turn on the recorder and start executing commands and they will end up in the workflow or you can copy commands from the Toolbox and paste them into the workflow. You can also add elements from the workflow toolbar.

Exercise

Build the workflow

1. Press the **New Workflow Element** icon to create the first entry. That may also be achieved by a right-click **New Element**.
2. Select the **Display** command found in the **Commands** folder and click **OK**.
3. Right-click the **Display** command in the workflow and select **Edit**.
4. 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).
5. In your project, select a line and click the green arrow in the Display edit menu.
6. Choose the display settings you like (color, glue to surface or not, etc.) and click **OK** again.
7. Right click the element in the workflow and choose **Execute**. Alternatively, click the **Execute** icon.
8. 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**.
9. 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**.
10. Continue to add different commands to your workflow.
11. Remember to save the workflow by clicking the **Save Workflow** icon.

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

A flight tour collects a series of camera positions and moves the camera smoothly through these positions. In order to create a camera position all you need to do is right click a folder and select **New > Camera Position.**

Exercise

Create a flight tour

1. In your project, create a new generic folder called **Cameras**
2. Zoom to an area you want to save as a camera position.
3. Right click the folder **Cameras** and select **New > Camera Position**
4. Repeat the last two steps to create more camera positions.
5. Select the **Cameras** folder in your project.
6. In the **Toolbox** go to **Shared Commands > Visualization**, right click **Flight Tour** and select **Copy.**
7. **Paste** the command into your workflow. The command will open and it should have selected the **Cameras** folder as input.
8. Set the flight time between every camera position to **5** seconds and make sure **Collect cameras in folder to smooth flight tour** and **Single flight tour** is checked
9. Click **OK**
10. Select the **Flight Tour** command in the workflow and click the **Execute next element** icon to run the flight tour.

To update a camera position, set the view in the graphics window, right click the camera position and select **Assign View.**

It is possible to group elements in a folder. Add folders by clicking **New workflow element** on the workflow toolbar and selecting **Group** under **General .**

V. 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.

W. How to make charts in ArcMap

Introduction

This document is a description on how to print maps or charts in ArcMap Basic. The examples are typical data exported from Geocap Seafloor.

This document is not meant to replace any training courses in using ArcMap. The intention is to have a short guide on how to create map layouts, import data and define the symbology (graphic attributes) for rasters, points, lines and polygons. Additional knowledge about using ArcMap is recommended.

Some of the texts and images in this document are ArcGIS Desktop 10.3 Help texts.

Seafloor
<VALUE>

Color	Range
	0 - 25
	25 - 50
	50 - 75
	75 - 100
	100 - 125
	125 - 150
	150 - 175
	175 - 200
	200 - 225

Title:	VERTICAL REFERENCE:
<Title of Survey>	Mean Sea Level
	COORDINATE SYSTEM: WGS 1984
<Name of survey vessel>	PROJECTION: WGS 1984 UTM Zone 32N
Work area: **Scale: 1:15 000**	DEVICE FOR MEASURING THE DEPTH :
<Name of work area>	XXXXXX Multibeam echo sounder
	DEVICE FOR MEASURING THE POSITION:
	<name of device>
Chart number: **xx / 2014**	SURVEY TECHNICIAN: <operator name>
	<operator title>
Survey date: **dd.mm.2014**	AUTHOR: <name of chart author>
Revision date: **dd.mm.2014**	<title of chart author>

How to create map documents

A map document is containing the layout for a map to be distributed or printed. The document is saved as an .mxd file type.
The document contains data frames, text boxes, static or dynamic texts, legends, north arrows etc.
The data frames contains the displayed geographical data symbolized as lines, points, symbols and labels. The data storage for the displayed data can be files on a disk or from one or more geodatabases.

Open a new map document

To create a new map document, open the **File** menu and select **New…** to open the **New Document** dialog.
Under My Templates there will be one template named Blank Map.
Under **Standard Page Sizes > ISO (A) Page Sizes** several templates can be found. For instance, select ISO A3 Landscape and click OK to close the New Document dialog.
The document will be opened with one data frame named Layers in the Table Of Contents.
The new map document can be saved first time to the disk by File > Save. Browse to the correct folder and specify a name for the map document and click Save. The file type will be .mxd.

Layout View and Data View

The map document can be viewed in two modes. In the **View** menu, find the **Layout View** and the **Data View**:

- **Layout View** is showing the whole document where all map elements and map data are available.
- **Data View** shows the data in the data frame only and not the entire map content. If the map document contains several data frames, the contents of the active data frame will be shown.

Modify the main data frame

When creating the new map document there will be a default data frame named Layers.
By double clicking the text **Layers** the name of the data frame can be modified.
Another method is to right-click the Layers and select Properties. In the Data Frame Properties click the **General** tab. There the name of the data frame can be modified. For instance, type in the text **Main Data Frame** and click Apply.

Add and display data in the data frame

To add data in the data frame right-click the data frame and select Add Data… to open the Add Data dialog.
Browse and select the data from disk or from a geodatabase. It is also convenient to open the Catalog window and locate the data objects directly and drag the data item and drop either into the data frame in the graphics or into the Table Of Contents.
The symbolizing of the data can be set in the Properties dialog for the layer.

Set coordinate system

If no coordinate system is defined before adding data the data frame will get the same system like the first data set. If the coordinates are geographic the data frame will select the same geographic (e.g. WGS84), and if they are in a projection the data's datum and projection will be used (e.g. UTM 34N, WGS84).
To ensure the correct coordinate system, open the **Data Frame Properties** dialog (right-click and select Properties…) and select the **Coordinate System** tab. To set a projected coordinate system, click Projected Coordinate Systems + UTM + WGS 1984 + Northern Hemisphere, and select the item **WGS 1984 UTM Zone 34N**. Click Apply to set the coordinate system. Before leaving the Coordinate System tab it is possible to right-click the WGS 1984 UTM Zone 34N system and select Add To Favorites. Then the system can be found in the Favorites folder over the Geographic Coordinate Systems folder.

Set position and scale

The position of the data frame can be set in the **Size and Position** tab. Enter the position of the Anchor Point, given in mm from the lower left corner of the map document.
The size of the frame is defined in the Size part of the tab. Set width and height in mm.

The scale of the data frame is set in the **Data Frame** tab. The Extent can be defined as **Automatic**, **Fixed Scale** or **Fixed Extent**.

- **Automatic** will set the scale and extent from zooming and panning inside the map frame.
- **Fixed Scale** will lock the data frame to the given scale, for instance 1:1000. Select the scale value from the dropdown menu, or write it as text in the Scale field. The part "1:" can be omitted. When using Fixed Scale it is not possible to zoom in and out of the map frame.
- **Fixed Extent** means specify the coordinates for the lower left and upper right corners. The scale is calculated from the given coordinates and the size of the data frame in mm. When using Fixed Extent it is not possible to zoom and pan.

Geographic graticule and Cartesian grid

To apply geographic graticule and/or a projection grid is done in the **Grids** tab in **Data Frame Properties** dialog. The final layout of both the graticule and the grid can be changed after the initial setup.

Geographic graticule

Click the New Grid… button and open the Grids and Graticules Wizard.
Select Graticule and click Next.
In the Create graticule dialog, select, for instance, Graticule and labels, and specify Intervals for parallels and meridians and click Next.
In the Axes and labels dialog click text style button for selecting font size and style. After closing the Symbol Selector dialog, click Next.
In the Create a graticule dialog select Place a calibrated border at edge of graticule. Specify the Properties, for instance Fill color is black, Void color is white, Width is 5 points, Interval 72 points and Use double alternating border. Check the Graticule Properties part and use the Store as a fixed grid that updates with changes to the data frame. Click Finish.
To view the Graticule grid click Apply in the Data Frame Properties dialog.
To edit the Properties of the Graticule, highlight the Graticule in the Grids tab and click the Properties... button. Check the tabs Axes, Labels, Lines and Intervals.

Cartesian grid (UTM grid)

Click the New Grid… button and open the Grids and Graticules Wizard.
Select Measured Grid and click Next.
In the Create a measured grid dialog, select, for instance, Grid and labels, and specify Intervals for X Axis and Y Axis and click Next.
In the Axes and labels dialog click text style button for selecting font size and style. After closing the Symbol Selector dialog, click Next.
In the Create a measured grid dialog select Place a border outside the grid. Check the Grid Properties part and use the Store as a fixed grid that updates with changes to the data frame. Click Finish.
To view the Measured Grid click Apply in the Data Frame Properties dialog.
To edit the Properties of the Cartesian grid, highlight the Measured Grid in the Grids tab and click the Properties... button. Check the tabs Axes, Labels, Lines and Intervals.

Create an Index Map

The Index Map is just another data frame. The new frame is created by opening the **Insert** menu and select **Data Frame**. The New Data Frame is positioned in the middle of the map layout. By using the mouse the frame can be selected and resized.
Detailed specification for the size, position, coordinate system and scale is set in the Data Frame Properties dialog as discussed above.

Make non geographic boxes in the map

In the map we need to create boxes around texts and other content, like legends etc.
In the **Insert** menu select the **Neatline** option.
In the Neatline dialog click Place inside margins. Click the Border and select the style and thickness, for instance 0.5 points. If the *i* icon is clicked the thickness can be changed to 0.1 points if wanted.

When clicking OK the neatline (box) will be created along the edge of the page area. The box can be resized and moved with the mouse.

When right-clicking the selected frame, the Properties for the neatline (box) can be opened. In the tabs **Area**, **Frame** and **Size and Position**, the properties can be changed. Set the position of the anchor point (default is lower left corner) and the width and height of the box.

When making several identical boxes it is possible to copy and paste the box, and then set the correct position of the anchor point of the new box.

It is also possible to select several boxes and set the lower left corner for the group of boxes.

Insert texts

In the **Insert** menu select the **Text** option.

The text is created somewhere in the middle of the map. Locate the Text and move to the wanted position.

When right-clicking the selected text, the Properties for the text can be opened. In the tabs **Text** and **Size and Position**, the properties can be changed. Set the position of the anchor point (default is lower left corner) for the text can be specified here.

In the Text tab the text string can be entered. To set the font size and type, click the Change Symbol… button.

When making several texts with identical style is possible by copy and paste the text. Open the Properties by double-clicking the new text or right-click and select Properties. Set the new text string and the correct position for the new text. The position can also be set by moving the text with the mouse.

Dynamic texts

In the **Insert** menu select the **Dynamic Text** option, and select between different text content.

The Dynamic text is using information from the map document, like a data frame's scale and coordinate system, current data and time, last saved time etc.

The changing of properties is equal as for a static text.

How to export data from Geocap

From Geocap we have two methods for exporting data to ArcMap and other software.

1. Export as shapefiles or other formats.
2. Use the DataLink for ArcGIS to copy directly to an ArcGIS file geodatabase when using the 32 bit version of Geocap.

Export to files

Each dataset also have an **Export** entry on the right-click menu where different formats can be selected.

In Geocap Seafloor the data can be exported from the *2. Charts* folder by the commands **Export Chart Data** an d **Export Contours**.

Export Chart Data

The Export Chart Data command has the four tabs **Polygons**, **Points**, **Surfaces** and **Hill Shade**.
In each of the tabs, select which charts to export. For some of the formats there is an option to check the Lat/Lon box when the coordinates should be exported as geographic.

- From the **Polygons** tab export the Frame and Boundary polygons for the selected chart(s). The best output format is **Esri Shapefile Polygons** (as Polygon Feature).
- From the **Points** tab export the Seafloor surface as points for the selected chart(s). The best output format is **Esri Shapefile Points** (as Point Feature).
- From the **Surfaces** tab export the Seafloor surface for the selected chart(s). The best output formats are **Bathymetry Attributed Grid (BAG)** and **Geo TIFF**.
- From the **Hill Shade** tab export the Seafloor surface as a hill shade image for the selected chart(s). The output format is **Geo TIFF**.

- Specify the output directory for the files by clicking the browse button and select folder.
- The file name will (for instance) be **Surfacepoints_<Chart Name>.shp**. If the file already exist it can be overwritten or a new version can be created. A prefix to the filename can be specified in the input field.

Export Contours

The Export Contours command has the two modes, export as **Contour Lines** or **Contour Areas**. For ordinary contour lines, select the first mode.

- In the Chart Name part, select from which charts to create the contours.
- Select the contouring method (Increment) and the increment value.
- Check the Lat/Lon box when the coordinates should be exported as geographic. The option is not available for all format types.
- Select if the contours should be saved to a file, to the project, or both. The project object will get the name Contours.
- Select Export Format. The best output format is **Esri Shapefile Contours** (as Line Feature).
- Specify the output directory for the files by clicking the browse button and select folder.
- The file name will (for instance) be **Contours_<Chart Name>.shp**. If the file already exist it can be overwritten or a new version can be created. A prefix to the filename can be specified in the input field.
- The contours in the shape file will have the same layer name for all contour line levels. If some contours (for instance every 5th) should be displayed with different line width, it is necessary to do the export 2 times with the different increments, for instance 1 meter and 5 meter increments. When later creating annotation on the contour lines, the operator can decide which contour data set to use, if all contours should have annotation or just every 5th.

Export to ArcGIS file geodatabase using the DataLink

The export to the ArcGIS file geodatabase requires a license of the module **Geocap DataLink for ArcGIS**. The feature in the license file will have the name "gis".

On the computer the ArcMap (ArcGIS) program must be installed and have a valid license.

To run the module the 32-bit version of Geocap must be started, since ArcGIS is a 32-bit application.

Activate the Geocap DataLink for ArcGIS

From the Geocap main window select Tools > Options …

In the **Options** menu select **Plug-ins** in the left part. In the Plug-ins part, check the box for **DataLink for ArcGIS** .

Below the Plug-ins part the ArcGIS version must be selected. The latest version is 10.3.

Then click the Apply or OK buttons to activate the module. Next time Geocap is started after exit, the module will be automatically loaded. If the 64-bit version is started, the module will just be ignored.
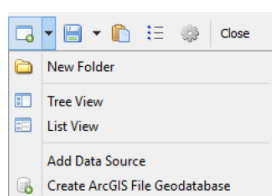
Create datasets for copying

One of the preparations before the copying is to create the datasets which should be presented in the map.

- **Boundary**: Execute the **Make Boundary and Holes** command to create the Boundary dataset (outline) around the input points or the generated surface (Seafloor).

- **Contours**: Execute the **Export Contours** command to create a Contours dataset for the chart. Note that if you want to have two contour data sets (see the text in section **Export Contours**) the command must be executed two times, and the first Contours data set must be renamed before executing the command the second time.

- **Surfacepoints (Seafloor surface as points)**: Execute the command Export Chart Data and go to the Points tab. Select the option Seafloor surface as points, and also select Save in project only. When clicking Execute the Surfacepoints data set is created for the selected chart(s).

Create or open an ArcGIS file geodatabase

The ArcGIS file geodatabase can be opened as an object in a Geocap project. So before you can open a geodatabase the Geocap project must be loaded.

Click the arrow next to the ⬚ icon in the project toolbar, and open the menu:

- If the ArcGIS file geodatabase already exist, click Add Data Source in the menu. Set Datasource to File Geodatabase and click the Browse button for the correct database (folder). When clicking the Connect button the database is opened in the project tree.
- To create a new file geodatabase, click Create ArcGIS File Geodatabase. In the dialog which appears select **In project** or **Outside project**. If the database is to be shared, it is recommended to create it outside the Geocap project. Browse to the proper folder. Then, in the Name field, enter the name of the database (for instance myChart). Click OK to create the database.

Copy data to the ArcGIS file geodatabase

To copy data from a Geocap project to an ArcGIS file geodatabase is done from the right-click menu option **Copy** on the selected data items. It is possible with multiple select and also different data types. In addition to select data in the list view, it is also possible to use the Multiple > Copy from a folder in the project.

Click on the file geodatabase (for instance myChart) in the project tree, right-click and select **Paste**.

Then a menu opens for setting parameters for the different datatypes Points, Lines and Surfaces.

Copy Point data as:

- Export as **Point Feature** or **Multipoint Feature**. To copy as Multipoint Feature it will be faster to copy the data, but if the each points will be labeled with depth value, the **Point Feature** class is required.
- **Add Z value of points as fields in feature class**: To be able to print the depth value in a map, the Z value must be a column in the attribute table.

Copy Line data as:

- Export as **Line Feature**, **Contours (Line Feature)** or **Polygon Feature**.
  In ArcMap line data are defined in two types of features: Lines Features and Polygon Features. Lines (polylines) are open-ended lines, and Polygon Features are closed lines defining an area. A polygon is always connecting the last point to the first point in the line. Geocap not differ on line data, and do not check if it the lines are open or closed.
  The selected feature type is dependent on the data to copy. For instance, the **Frame** and **Boundary** data sets should be copied as **Polygon Features**, while the contour lines should be saved as **Contours (Line Feature)**.
  The **Contours (Line Feature)** option stores the lines as line features even if there are closed contours in the data set. The contours are sorted on increasing depth to easier check the depth levels in the dataset. Each line element will also be saved as a row in the ArcGIS data table. This makes it possible to use the Editor tool in ArcMap to select and edit or delete small contour lines.

- **Also export to Point Feature class**: When turning on this option the lines data can also be copied as a point data set with the postfix *_points* in the feature name. This is necessary if any attributes of points in the line should be used in the map document.

The next options are appropriate for all data types, both **Points**, **Lines**, **Polygons** and **Rasters**.

Vertical Options:

- **Flip Z values** means that the z values in the data geometry is multiplied with -1 to convert to elevation (height) with positive values upwards, instead of depths with positive value downwards.
- **Set Vertical Coordinate system**: When vertical reference system is known it can be defined by using this option. The system can be an elevation or a depth system. First select the region (Africa, Asia,

Europe, World etc), and then a system in the region.
If a depth system is selected the Z values do not need to be flipped.
Note: The option only sets the actual vertical reference system and no shift between the systems is performed.

Target database: The name and path to the database selected for copying to.

Output Spatial Reference:

- Set the horizontal coordinate system for the data. The information box will contain the coordinate system of the first selected data object which is copied. The Geocap geodetic properties are translated into the ArcGIS definition. If a vertical reference system is defined it will appear after the horizontal system.
- Click the Browse button to open the dialog to set the spatial reference.

Note: The selected vertical and spatial reference will be set on all the selected data sets. If the selected data items have different coordinate systems, for instance projected and geographic, the copying should be done in two operations. In ArcMap the data will automatically be transformed to the projection in the data frame during display and plotting, so it is no problem to have data with different coordinate systems in the same database. Click **OK** to start the copying. Large point data sets copied as Point Feature may take some time.

How to add layers to a map (in ArcMap)

Adding a dataset

To create a new map layer, simply add a dataset to your map, globe, or 3D scene. There are a few ways to add datasets:

- **Using the Add Data button** — Click the **Add Data** button ![icon] and navigate to the desired dataset. Then select and add it to your map.

- **Copying or dragging a layer** — You can move layers between data frames or maps by copying and pasting or dragging the layer from one data frame to another.

- **Dragging a dataset from the Catalog window** — You can navigate to datasets and add them directly in ArcMap. Using the **Catalog** window, navigate to the desired dataset. Drag the dataset into the map's data frame.

All datasets which are recognized by ArcMap can be copied or dragged into the active Data Frame this way. If the dataset is an excel file it will appear as a table in ArcMap. Also, if the dataset is an ascii file it will also appear as a table when dragged into ArcMap.

Adding data from Geocap Seafloor

1. From shapefiles

In the Catalogue window, browse to the folder where the exported shapefiles are located. See the files with Surfacepoints, Contours, Frame, Boundary and Seafloor etc, and drag the files into the active data frame.

2. From file geodatabase

In the Catalogue window, browse to the folder where the database myChart.gdb is located. Open the database and see the data items copied into the database. To be sure to that all data items are shown, click the How to symbolize the data layers will be discussed later in the document.

Adding an ASCII or text file table

In ArcGIS, you can directly access data in delimited text files and work with them as tables. ArcCatalog and the **Add Data** dialog box in ArcMap list files with .txt, .asc, .csv, or .tab extensions and assign them a file type of text file.

Files with a .txt, .asc, or .csv extension are interpreted as comma (or semi-colon) delimited, while files with a .tab extension are interpreted as tab delimited by default. Any file with one of these extensions will be interpreted as a text file table even if it doesn't contain tabular data. If you attempt to display a text file that doesn't contain tabular data, the software will either produce an error or attempt to display the data as a table. To avoid this problem, give your delimited text files a .csv or a .tab extension. This will help differentiate text files with delimited data from unformatted text files.

The first row of your text file can contain the column headings. The following rows can contain coordinates and attributes. Remember to use commas or tabs to distinguish the columns. The following is an example of a comma-delimited text file:

```
x,y,ID,color
8.6,5.6,001,blue
99.3,77.0,002,blue and red
8.01,44.3,003,orange
```

This can be previewed in ArcCatalog or added as data in ArcMap and will appear as shown:

To convert the table to a geographic layer you will need to run a geoprocessing tool.

Convert data with the Make XY event layer tool

Having a data in a text file (ascii) the best way to import the information is from a semi-colon separated file with file type **.csv**. The coordinate system can be in geographic or in a projection.

The .csv file must have a header row for specifying the columns, for instance "Latitude; Longitude; Label":

```
Latitude; Longitude; Label
54.38228614; 18.65559966; 1
54.38222022; 18.65523383; 3
54.38215409; 18.65486909; 5
54.38208797; 18.65449803; 7
54.38202205; 18.65413099; 9
```

In any ArcGIS for Desktop application, you open the ArcToolbox with the ArcToolbox Window button 🔴 found on the standard toolbar or by clicking **Geoprocessing** > **ArcToolbox**.

To import the points and the labels we will use the **Make XY Event Layer** tool. If you have more than one data frame in your map document, check which data frame is active, if it is the Index map or the main map.

The tool is opened from the ArcToolbox window:

> **Data Management Tools > Layers and Table views > Make XY Event Layer**

The menu will have these input fields:

> **XY Table**: Browse to the correct folder and select the .csv file containing the points and labels.

> **X Field**: Latitude (or X if the file has Cartesian coordinates.)

> **Y Field**: Longitude (or Y)

> **Z Field (optional)**: Set blank if the file does not have a Z column.

> **Layer Name or Table View**: The name of the output point event layer. Will by default be the data file name without .csv.

> **Spatial Reference (optional)**: The spatial reference of the coordinates in the X and Y Fields defined above. This will be the output event layer's spatial reference. It is recommended to set the spatial reference.

Click **OK** to read the input file into the map document.

How to symbolize data in the map

The layers in the data frame can be symbolized and presented in different ways according to the geometry type. Lines can be symbolized as lines only, polygons (areas) as closed lines and colourfill/hatching, points with symbols and labels. Rasters (grids and images) has another type of symbology.

Points with symbols and text labels

Some labels in the map will contain symbols which represents the point position, and a text numeric label which should be annotated close to the symbol.

The simplest is to import the information from an ascii file, and the best way to import the information is from a semi-colon separated file with file type **.csv**. The coordinate system can be in geographic or in a projection.

The .csv file must have a header row for specifying the columns, for instance "Latitude; Longitude; Label":

```
Latitude; Longitude; Label
54.38228614; 18.65559966; 1
54.38222022; 18.65523383; 3
54.38215409; 18.65486909; 5
54.38208797; 18.65449803; 7
54.38202205; 18.65413099; 9
```

The data points can be imported to the map document by the **Make XY Event Layer** tool.

1. Apply a symbol

Find the new layer and open the Properties dialog for the layer. In the Layer Properties dialog, click the Symbology tab and click the Symbol icon. In the Symbol Selector dialog, click Edit Symbol to open the Symbol Property Editor.

In the Symbol Property Editor the properties type should be **Character Marker Symbol**. In the Character Marker tab below there are a lot of ESRI font sets. Some useful symbols can be found in the **ESRI Default Marker** font set (for instance the symbol P). Many hydrographic symbols can be found in the font set **ESRI NIMA DNC PT** (digital nautical chart, points). Another useful font set is **ESRI Geometric Symbols**.

It is possible to create a new symbol by combining two or more symbols from different font sets.

When clicking Apply in the Layer Properties dialog, the symbol plotting will start.

2. Apply the Label

In the Layer Properties dialog, click the Labels tab.

In the Text String box, set the Label Field to the name from the third column in the input file: Label.

In the Text Symbol box, set the font type, font style, color and size.

Click the Placement Properties button and open the dialog. In the Placement tab, select the option **Offset label horizontally around the point**. This will plot the label with the on the specified side of the point's position. Click the **Change Location** button to change the labels' position around the points.

Some optional locations:

Prefer Top Left, then Left then Top   Prefer Top Right, all allowed   Prefer Top Center, all allowed

In the Conflict Detection tab the parameters for overplot removal can be adjusted. Check the Label weight and the Feature weight options. Click OK to close the Placement Properties dialog.

In the Labels tab in the Layer Properties dialog, check the box ahead of Label Features in this layer. When clicking Apply or OK the label plotting will start.

Note: If the Placement Properties dialog has more the two tabs, we have to change the settings for the data frame. Enter the properties for the data frame and click the General tab. In the lower part there is a selection of Label Engine. Select the Standard Label Engine and not the Maplex Label Engine.

Hydrographic Depth Annotation

Add the Surfacepoints dataset from the geodatabase to the active data frame.

The default symbolization is a coloured filled point symbol with outline with size 4 points.

Open the Surfacepoint's **Layer Properties** dialog (double-click or right-click and select Properties).

Click the **Symbology** tab and then double-click on the symbol. In the Symbol Selector dialog, select the symbol named Circle 1, and set the Color to No Color. Click OK to finish the dialog.

Click the **Labels** tab and do the following:

- In the Text String box, set the Label Field to **Z**. (If Z is missing the export from Geocap or the import to ArcMap has not been properly).

- Click the Placement Properties button and open the dialog. In the Placement tab, select the option **Place label on top of the point**. This will plot the label with the geometric center of the point's position. In the Conflict Detection tab the parameters for overplot removal can be adjusted. Click OK to close the Placement Properties dialog.

  Note: If the Placement Properties dialog has more the two tabs we have to change the settings for the data frame. Enter the properties for the data frame and click the General tab. In the lower part there is a selection of Label Engine. Select the Standard Label Engine and <u>not</u> the Maplex Label Engine.

- To have the Hydrographic Annotation we have to enter a formula:
    - Click the Expression button and open the Label Expression dialog.
    - In the lower part, click the Load button. Browse to the file Hydrographic_Z_labels_0_21_31.lxp and click open. The file contains this expression:

```
Function FindLabel ( [Z] )
' Generate depth labels according to IHO CHART SPECIFICATIONS,
' Edition 4.4.0 – September 2013:
' Soundings must be rounded down, if necessary. The rounding
should be:
' · to the nearest decimetre between 0,1 and 21m,
' · to the nearest half metre from 21 to 31m and,
' · thereafter, to the nearest metre.
dim decimeters
dim meters
dim decimal
decimeters = int( [Z] * 10 )
meters = int( decimeters / 10 )
' ** Deeper than 31 meters, label the meter values only **
If meters >= 31 Then
FindLabel = meters
' ** Generate label with meters only **
FindLabel ="<FNT name = 'Arial' size = '7' > " & meters & "
</FNT>"
Else
' ** Between 21 and 31 meters, cut decimal to 0.5 meter,
shallower **
' ** Between 0 and 21 meters, cut decimal to 0.1 meter,
shallower **
decimal = int ( decimeters - meters*10 )
If meters >= 21 Then
If decimal>=5 Then
decimal = 5
Else
decimal = 0
End If
End If
' ** Generate label with meters and decimals. If decimal is
zero it shall not be shown
If decimal > 0 Then
FindLabel ="<FNT name ='Arial' size = '7'> " & meters &
"</FNT><FNT name ='Arial' size = '9'><SUB>" & decimal &
"</SUB></FNT>"
Else
FindLabel ="<FNT name ='Arial' size = '7' > " & meters & "
</FNT>"
End If
End If
End Function
```

- The expression can be modified and saved to a new file by the Save button.
- Note that the font type and font size is defined in the expression. The font is Arial and the size is 7. The lowered decimal is printed as a subscript font, but with size 9 pts to make it not too small.
- Click OK to finish the Label Expression dialog.

- In the Labels tab in the Layer Properties dialog, check the box ahead of Label Features in this layer. When clicking Apply or OK the label plotting will start.

- The labeling of the **Surfacepoint** layer can also be turned on and off by the checkbox ahead of the layer name. Also right-click the layer and turn on and off Label Features.

Points with Hydrographic Symbols

Plotting Hydrographic symbols can defined by opening the Properties dialog for the layer. For example the buoy symbol IHO_Q5_B can be set up by combining two symbols (fonts).

In the Layer Properties dialog, click the Symbology tab and then click the Symbol icon. In the Symbol Selector dialog, click Edit Symbol to open the Symbol Property Editor.

In the **Symbol Property Editor** the properties type should be **Character Marker Symbol**. In the Character Marker tab below select the font **ESRI NIMA DNC PT** (fonts for Digital Nautical Chart, points).

Select the isymbol and then add the s symbol. By combining the size and offset of the two fonts, the result can

be the hydrographic symbol The dotted cross in the preview shows where the position of the symbol will be.

When clicking Apply or OK in the Symbol Property Editor, the Symbol Selector dialog, and then the Layer Properties dialog, the symbol plotting will start.

Save and re-use a symbology

To save the symbol for use on other data sets later, close the Layer Properties dialog and right-click the layer (data set) name and select Save As Layer File. This will save the layer definition and not the data as a .lyr file.

In the Save Layer dialog enter the file name, for instance buoys_IHO_Q5_B.lyr, and browse to a folder where to save the definitions.

Next time the same symbol is to be used for a data set (layer), open the Layer Properties dialog, and click the Symbology tab. To re-use the symbol, click the Import button up to the right. Click the browse button and add the saved .lyr file to the Layer pulldown menu. Click OK to select the symbol definition.

Use the same method for all other point, lines and polygons symbology. Note that labels will not be set from the layer definition. Enter the Labels tab to define the label style.

Contours

1. Create contour annotation with the Contour Annotation tool

To create the contour annotation and the gap in the contour lines, we will use the **Contour Annotation** tool. If you have more than one data frame in your map document, check which data frame is active, if it is the Index map or the main map.

The tool is opened from the ArcToolbox window:

> **Cartography Tools > Annotation > Contour Annotation**

The menu will have these input fields:

> **Input Features**: Contours (when copied from the file geodatabase)
>
> **Output Geodatabase**: C:\ArcGIS\MyChart\myChart.gdb
>
> **Contour Label Field**: Z
>
> **Reference Scale**: 1000 (the scale for the plot)
>
> **Output Layer Name**: Contours_layer
>
> **Contour and Label color**: BLACK
>
> **Contour Type Field (optional)**: (no value)
>
> **Contour Alignment (optional)**: PAGE (or UPHILL)

Enable Laddering (optional): Yes (Check this box)

Click **OK** to create the contour annotation features into the map document.

2. Set drawing properties on contours and annotation

Use **List By Drawing Order** 🗇 in the table of contents to author the contents of your map, such as to change the display order of layers on the map, rename or remove layers, and create or manage group layers.

A new group layer, Contours_layer, is created containing these feature layers:

- Contour Features: Contains contour lines. Set the line properties by double-clicking the automatic generated line symbol.

  Note: This dataset is not saved in the output geodatabase. Can be saved by right-clicking, select Data > Export Data… Click the browse button to select which folder or file geodatabase to save the data layer, and specify the name.

- Contour_FeaturesAnno: Contains the annotation labels. The properties can be changed by using the **Edit or** tool. The dataset is saved in the output file geodatabase.

- Contour_FeaturesAnnoMask: Contains the mask to clip holes in the contour lines. Change the Symbology (on the Layer Properties) to be white text without line around. The dataset is saved in the output file geodatabase.

The drawing order must be changed to:

1. Contour_FeaturesAnno
2. Contour_FeaturesAnnoMask
3. Contour Features

3. Change font size for annotation labels

The font style for the annotation labels is changed with the Editor tool. The Editor should not be active before starting.

From the **Contour_FeaturesAnno** layer, right-click and select **Selection > Select All.**
Then again from the **Contour_FeaturesAnno** layer, right-click and select **Edit Features > Start Editing.** Often a dialog with an error message appears, just click the Continue button.

Then locate the **Editor** toolbar and click the menu arrow and select **Editing Windows > Attributes**. The attributes window appears on the right side of ArcMap. The attributes window contains the name of the layer, Contour_FeaturesAnno, and the list of the annotation labels. To change the style for all labels, mark the top text (Contour_FeaturesAnno). Click the Annotation tab below, and in the lower part the font style can be changed. Click Apply to set the new style.

When finished, locate the **Editor** toolbar, click the menu arrow and select **Stop Editing**. Confirm that the editing is saved.

If the Editor toolbar is not visible, click **Customize > Toolbars**, and check the box for the Editor toolbar.

Lines and polygons

Data sets of type Lines and Polygons can change symbology by opening the Properties dialog form the data layer.

Lines can be symbolized by the line color, thickness and pattern.

Polygons can be symbolized with different types of fill patterns and colors, and with or without the outline.

Rasters (surfaces)

Raster data s can change the symbology by opening the Properties dialog from the data layer. As default the raster is displayed using a greyscale color table. A change of the color table can be done by double-clicking the table in the Table Of Contents. Then, in the Select Color Ramp dialog the color table can be changed.

To do some more detailed changes, open the Layer Properties dialog.

First, open the **Display** tab. Change the option "**Resample during display using:**" from Nearest Neighbor to **Bil inear Interpolation** (or Cubic Convolution). This will make the color coding smoother for continuous data. Also check that the option "**Display Quality:**" is set to **Normal**.

Then, open the **Symbology** tab. The default display method is **Stretched**. If that is ok, the color table (**Color Ramp**) can be changed also here.

To make color levels with fixed increment, change the Show method from Stretched to **Classified**. First time the Compute Histogram dialog appears. Click the Yes button to compute the histogram. By default 5 classes are created. Click the **Classify…** button.

In the Classification dialog, change the Classification Method to **Defined Interval**. Enter the interval size (increment) to (for instance) 1 m. On the right hand side of the histogram the break values appear. Click the OK button when they seems correct. Back in the Symbology tab in the Layer Properties the color legend is shown. In the Label part the text for the labels in the color table can be edited. The colors are changed when changing the color ramp (color table), and the can also be changed individually by double-clicking the colored rectangle. Click Apply to see the changes in the Table Of Contents and to see the color code raster.

If the message "Classification failed. Please choose different data or classification method" appears, the problem is that the difference between the minimum and maximum value is too small for the default settings. If 1 meter interval size do not work, try 0,5 m. To force the classification to be with the correct number of intervals, select the method **Equal Interval**, for instance 2. In the list of the break values, change the values to be the exact level for the color change. If the depth values in the raster are from 10,03 meters to 11,86 meters, change the break values to be 11 and 12. Click OK in the Classification dialog and see the color table in the Symbology tab of the Layer Properties. Check the color symbols and labels before clicking Apply or OK in the Layer Properties dialog.

Create circles in the map

Creating new datasets based on other geographic element can be done using the Geoprocessing tools. For instance, to create a circle around a given position, use the Buffer tool.

As an example a circle with radius 50m should be created around the point "54.38161998, 18.65444449". The coordinates are given in WGS84 as longitude/latitude, while the map projection is WGS84, UTM Zone 34N.

To import the circle center, create a .csv file named circle_point.csv:

```
Latitude; Longitude
54.38161998; 18.65444449
```

Verify that the correct data frame is active, and use the **Make XY Event Layer** tool to import the data and create a new layer named **circle_point** in the map.

To run the Geoprocessing tool, the circle_point must be converted to a Feature Class in by exporting the data from the layer to a geodatabase:

- From the new circle_point layer in the map document, right-click and select **Data > Export Data…**

- In the Export Data dialog set "Use the same coordinate system as: this layer's source data.

- Set Output feature class to, for instance: C:\ArcGIS\MyChart**myGeodatabase.gdb\circle_point** and click **OK**. Now the feature class circle_point is created in the database. Say yes to create a layer in the map document.

To create the circle around the point, click **Geoprocessing** and then **Buffer** tool:

The menu will have these input fields:

Input Features: C:\ArcGIS\MyChart\myGeodatabase.gdb\circle_point

Output Feature Class: C:\ArcGIS\MyChart\myGeodatabase.gdb\ circle_buffer

Distance [value or field]: Use **Linear unit** , 50 meters.

Side Type (optional)

End Type (optional)

Method: PLANAR (option GEODESIC for very large radius only)

Dissolve Type (optional): NONE

Dissolve Field(s) (optional)

Click **OK** to create the circle and save the circle_buffer in the file geodatabase myGeodatabase.gdb. In the map document also the layer circle_buffer is created. Open the Layer Properties to set and change the symbolization fir this circle.

Transform a point from geographic to a projection

The coordinates of the circle_point above are Geographic (WGS84). To transform to a new feature class in WGS84, UTM Zone 34N, we can use the tool **Project**:

The tool is opened from the ArcToolbox window:

Data Management Tools > Projections and Tranformations > Project

The menu will have these input fields:

Input Dataset or Feature Class: C:\ArcGIS\MyChart\myGeodatabase.gdb\circle_point

Input Coordinate System (optional) (Use the information from the dataset)

Output Dataset or Feature Class:

C:\ArcGIS\MyChart\myGeodatabase.gdb\circle_point_utm34n

Output Coordinate System: WGS_1984_UTM_Zone_34N (use the browse button)

Geographic Transfomation (optional)

Click **OK** to create the new feature class.

In the database C:\ArcGIS\MyChart\myGeodatabase.gdb, locate the feature class circle_point_utm34n, right-click and select Properties. In the Feature Class Properties dialog, select the **Feature Extent** tab. Since the feature class contains one point only, the extent will be equal to the coordinates of the point: X: 347673.2274, Y: 6028517.4613.

Digitize new lines in the map

Before starting digitizing new data into the map and a file geodatabase a new feature class must be created.

Locate the actual database in the Catalog window, for instance the myGeodatabase.gdb.

Right-click and select **New > Feature Class…**

In the New Feature Class wizard, specify the

Name: new_line_feature (as an example)

Alias: (leave empty)

Type: Line Features (if the feature class should be a line)

Geomentry properties: (use default)

Click the **Next>** button :

> The wizard continues with the coordinate system for the feature class. Set for instance the same as the map projection, for instance WGS84, UTM Zone 34N. Note that the Layers folder in the wizard shows the coordinate systems for the layer in the map document.

Click the **Next>** button :

> Verify the XY Tolerance.

Click the **Next>** button :

> Verify the database storage configuration.

Click the **Next>** button :

> Verify the database attribute table.

Click the **Finish** button.

The layer **new_line_feature** is created as an empty layer in the document and also as a feature class in the database.

For instance, to create a line showing the diameter in the circle created above, we need to digitize one point on the circle circumference on each side of the circle center with coordinates X=347673.23, Y=6028517.46. The new points should be: 1) X=347623.23, Y=6028517.46 2) X=347723.23, Y=6028517.46.

Locate the new_line_feature in the map document layers, right-click and select **Edit Features > Start Editing…** Often a dialog with an error message appears, just click the Continue button.

Then the Create Features window appears to the right of the graphics window. (Or locate the **Editor** toolbar and click the menu arrow and select **Editing Windows > Create Features**.)

Select the new_line_feature in the table of features, and then, under the **Construction Tools**, select the **Line** tool. When moving the cursor to the graphics window, the cursor changes to a cross for digitizing mode. Using the leftmost mouse button the coordinates can be taken from the mouse position. When right-clicking the coordinate can be snapped to another element or the absolute X,Y coordinate. Select **Absolute X,Y…** and the Absolute X, Y window opens with the coordinates from the cursor.

Enter the coordinates (X: 347623,23, Y: 6028517,46) and click the Enter key on the keyboard.

Again, select **Absolute X,Y…** and the Absolute X, Y window opens with the coordinates from the cursor.

Enter the coordinates (X: 347723,23, Y: 6028517,46) and click the Enter key on the keyboard. (Note: It is also possible to use Delta X,Y values from the first point instead of absolute X,Y).

To end the digitizing session, right-click and select Finish Part. Then a new line can be created.

To finish all the digitizing, right-click and select **Finish Sketch**.

Then locate the **Editor** toolbar and click the menu arrow and select **Stop Editing**, and verify that the Edits should be saved. Now the digitized coordinates are saved in the file geodatabase.

The symbolization of the layer in the map document can be changed in the Layer Properties dialogue.

Essential ArcMap vocabulary
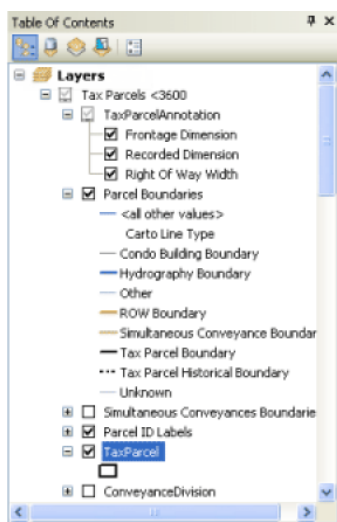
Map document (.mxd)

A map used in ArcMap that is stored as a file on disk. Each map document contains the specifications for the map layers, the page layout, and all other map properties. Map documents make it easy to save, reuse, and share your work in ArcMap. Double-clicking a map document opens it as a new ArcMap session.

Layer

A map layer defines how a GIS dataset is symbolized and labeled (i.e., portrayed) in your map views. Each layer represents geographic data in ArcMap such as a particular theme of data. Example map layers include streams and lakes, terrain, roads, political boundaries, parcels, building footprints, utility lines, and orthophoto imagery.
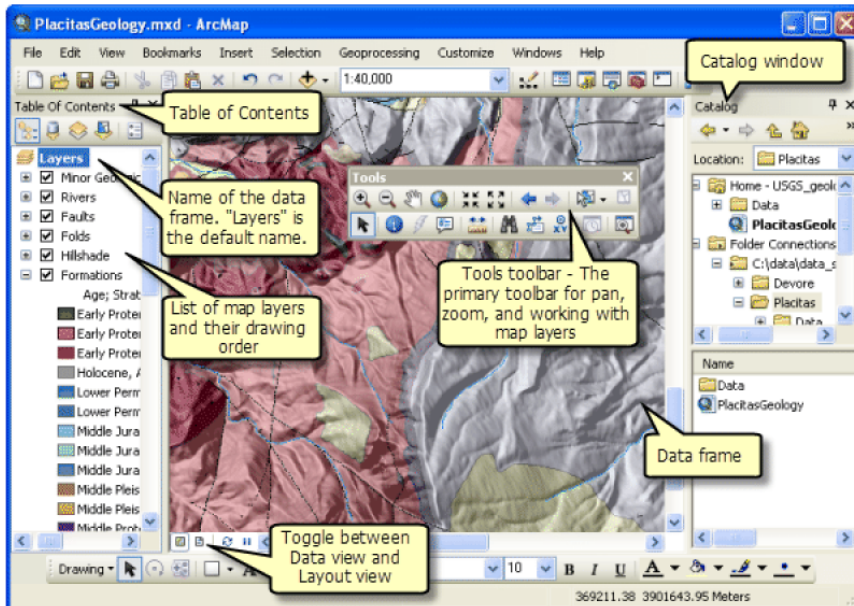
Table of contents

The table of contents lists all the layers on the map and shows what the features in each layer represent. The check box next to each layer indicates whether its display is currently turned on or off. The order of layers within the table of contents specifies their drawing order in the data frame from bottom to top.

The map's table of contents helps you manage the display order of map layers and symbol assignment, as well as to set the display and other properties of each map layer.
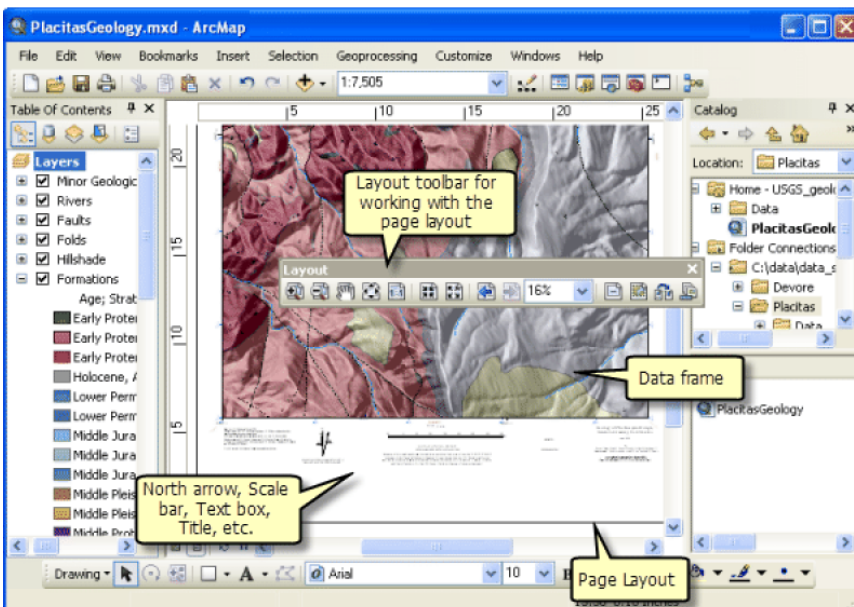
Data frame

The data frame displays a collection of layers drawn in a particular order for a given map extent and map projection. The table of contents on the left side of the map window shows the list of layers in the data frame.
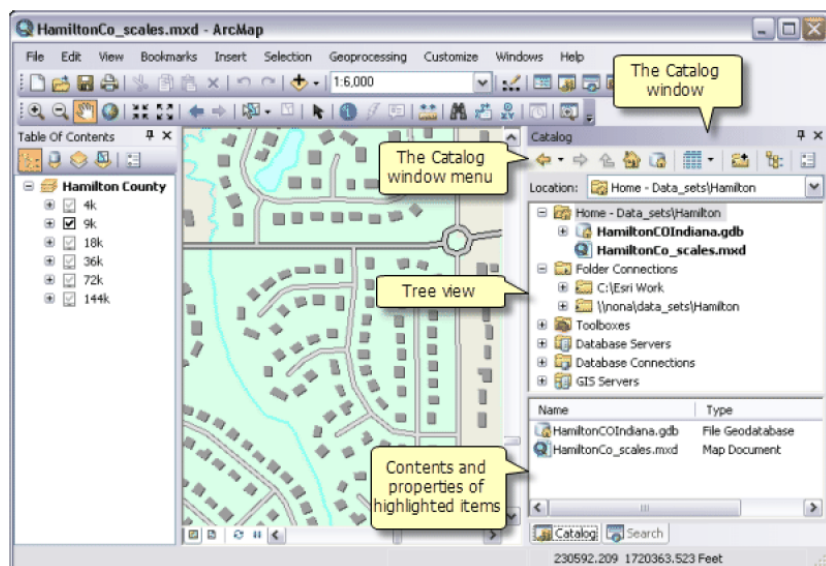

Page layouts

A layout is a collection of map elements laid out and organized on a page. Common map elements include one or more data frames (each containing an ordered set of map layers), a scale bar, north arrow, map title, descriptive text, and a symbol legend.

The Catalog Window

ArcMap, ArcGlobe, and ArcScene include a **Catalog** window that is used to organize and manage various types of geographic information as logical collections— for example, the data, maps, and results of your current GIS projects that you work with in ArcGIS.

The **Catalog** window provides a tree view of file folders and geodatabases. File folders are used to organize your ArcGIS documents and files. Geodatabases are used to organize your GIS datasets.

Labels

Labels are text strings that are used to label features within map layers. Properties define the attribute column used as the source of the text string and how the label is portrayed in your map. Labels are dynamic — Label display is recomputed each time the map is redrawn (e.g., as you pan and zoom the map).

Annotation

Annotation is used to represent feature labels that are saved as graphic feature locations in the geodatabase. The text location is saved along with other text properties for each annotation feature. Annotation differs from labels because each annotation location and depiction is only computed once and saved. These are reused each time you redraw your map. Since the annotation position is preset, no label computation need be done each time the map is redrawn.

Symbols

Symbols are graphic elements that are used in map displays. There are a number of symbol types, such as:

- Markers which are primarily used to display point locations
- Line symbols used to display linear features and boundaries
- Fill symbols used to fill in polygons
- Text symbols used to set the font, size, color, and other text properties.

Styles

A style is a collection of symbols, colors, and map elements that match a theme or application domain—for example, a style set for transportation maps or geology maps.

Basemap layers

A basemap is used for locational reference and provides a framework on which users overlay or mashup their operational layers, perform tasks, and visualize geographic information. In ArcMap, a basemap layer can be used to hold map layers that are more static and thus can be used to support high performance, dynamic map display.

Notes