Shelf Tutorials

Shelf

Delineation of the Continental Shelf

Geocap

Shelf (UNCLOS Art. 76) Tutorial

A. Introduction

About this Tutorial Collection

According to Article 76, coastal states can - under certain geological conditions - extend their juridical Continental Shelf and thus gain marine sovereignty rights beyond the 200-nautical-mile limit of the Exclusive Economic Zone (EEZ). Submissions for an extended Continental Shelf need to be filed directly with the Commission on the Limits of the Continental Shelf (CLCS) and must contain detailed marine geo-scientific data describing e. g. the shape of the continental margin, the location of the foot of the continental slope and the sediment thickness.

This tutorial collection will guide you through the process of analyzing different marine geo-scientific data and producing the documentation needed for an extended continental shelf submission. The order of the tutorials is more or less describing the suggested order in which a real submission would be done using the UNCLOS article 76 functionality in Geocap.

In this set of tutorials you will learn how to:

- Use Geocap in general
- Create a new project
- Import data
- Calculated different distance lines and constraint lines.
- Do analysis and set foot of slope points
- Calculate Hedberg points
- Interpret seismic data
- Depth convert interpretations and calculate sediment thickness
- Calculate Gardiner points
- Generate a final outer limit line
- +++

The Atlantis Project

The Atlantis project will be used as an example project in this tutorial, but you should also be able to follow the tutorial using your own data.The Kingdom of Atlantis is an imaginary state which has started to work on their extended continental shelf submission. Using this tutorial collection we are going to finalize the analysis and complete the Atlantis project using the functionality in Geocap Shelf.
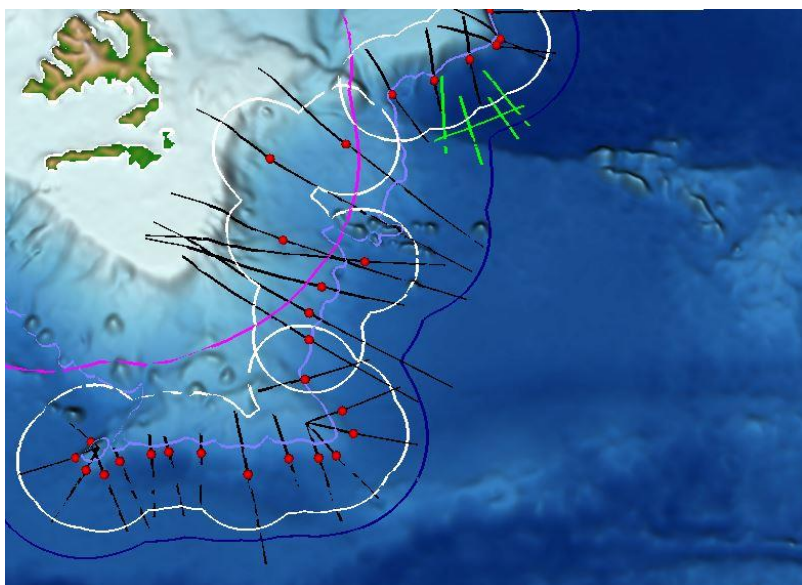
The continental shelf of Atlantis

-

UNCLOS Article 76

Below you will find the full definition of the continental shelf as stated in UNCLOS Article 76.

Definition of the continental shelf

1. The continental shelf of a coastal State comprises the seabed and subsoil of the submarine areas that extend beyond its territorial sea throughout the natural prolongation of its land territory to the outer edge of the continental margin, or to a distance of 200 nautical miles from the baselines from which the breadth of the territorial sea is measured where the outer edge of the continental margin does not extend up to that distance.

2. The continental shelf of a coastal State shall not extend beyond the limits provided for in paragraphs 4 to 6.

3. The continental margin comprises the submerged prolongation of the land mass of the coastal State, and consists of the seabed and subsoil of the shelf, the slope and the rise. It does not include the deep ocean floor with its oceanic ridges or the subsoil thereof.

4. (a) For the purposes of this Convention, the coastal State shall establish the outer edge of the continental margin wherever the margin extends beyond 200 nautical miles from the baselines from which the breadth of the territorial sea is measured, by either:
   i. a line delineated in accordance with paragraph 7 by reference to the outermost fixed points at each of which the thickness of sedimentary rocks is at least 1 per cent of the shortest distance from such point to the foot of the continental slope; or
   ii. a line delineated in accordance with paragraph 7 by reference to fixed points not more than 60 nautical miles from the foot of the continental slope.

   (b) In the absence of evidence to the contrary, the foot of the continental slope shall be determined as the point of maximum change in the gradient at its base.

5. The fixed points comprising the line of the outer limits of the continental shelf on the seabed, drawn in accordance with paragraph 4 (a)(i) and (ii), either shall not exceed 350 nautical miles from the baselines from which the breadth of the territorial sea is measured or shall not exceed 100 nautical miles from the 2,500 metre isobath, which is a line connecting the depth of 2,500 metres.

6. Notwithstanding the provisions of paragraph 5, on submarine ridges, the outer limit of the continental shelf shall not exceed 350 nautical miles from the baselines from which the breadth of the territorial sea is measured. This paragraph does not apply to submarine elevations that are natural components of the continental margin, such as its plateaux, rises, caps, banks and spurs.

7. The coastal State shall delineate the outer limits of its continental shelf, where that shelf extends beyond 200 nautical miles from the baselines from which the breadth of the territorial sea is measured, by straight lines not exceeding 60 nautical miles in length, connecting fixed points, defined by coordinates of latitude and longitude.

8. Information on the limits of the continental shelf beyond 200 nautical miles from the baselines from which the breadth of the territorial sea is measured shall be submitted by the coastal State to the Commission on the Limits of the Continental Shelf set up under Annex II on the basis of equitable geographical representation. The Commission shall make recommendations to coastal States on matters related to the establishment of the outer limits of their continental shelf. The limits of the shelf established by a coastal State on the basis of these recommendations shall be final and binding.

9. The provisions of this article are without prejudice to the question of delimitation of the continental shelf between States with opposite or adjacent coasts.

10. The coastal State shall deposit with the Secretary-General of the United Nations charts and relevant information, including geodetic data, permanently describing the outer limits of its continental shelf. The Secretary-General shall give due publicity thereto.

B. Getting Started with Shelf

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 Shelf/UNCLOS 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 Shelf module offers a range of commands with tailor made article 76 functionality.These commands are available in a Toolbox 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
- Start Geocap on Windows 8.1
- Set working directory and auto save
- Open the Atlantis project
- Explore the project folder structure
- Display data
- Navigate using the mouse

Starting Geocap

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

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

-

User Documentation

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

The user documentations consists of:

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

Initial Setup

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

Exercise

Set working directory and auto save

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

-

Geocap Project

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

Exercise

Open the Atlantis project

1. Click **File > Open > Project**
2. Browse to the location of the Atlantis project (*Geocap1/Projects/Atlantis_3/db*)
3. Select the **Atlantis.db** file and click **Open**.

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

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

In the next exercise we are going to explore how the Atlantis project is structured. Taking notice of how folders and datasets are organized. Datasets and folders can be cut, copied, pasted, renamed and deleted. This is performed from the popup menu which appears when right-clicking a dataset. The right click popup menu also contains **commands** which may be executed on the datasets.

Exercise

Explore the project folder structure

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

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

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

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

-

Displaying Data

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

---

Exercise

Display data

1. In the folder **2. Seabed / Grids** tick the checkbox next to the **atlantis** seabed surface.
2. Right click the **atlantis** seabed surface and select **Zoom to Data**. This will make the display window center the graphics window around the dataset.
3. In the folder **1. Maritime Lines / Coast lines** tick the checkbox next to **The Kingdom Of Atlantis** coast line.
4. In the folder **1. Maritime Lines / Base lines** tick the checkbox next to **Baseline_Atlantis** base line.
5. In the folder **1. Maritime Lines / 12 lines** tick the checkbox next to **Baseline_Atlantis + 12M** li mit line.
6. In the folder **2. Seabed / Bathymetric Profiles / North Sea** tick the checkbox next to all the bathymetric profiles

---

Note that by checking and unchecking the first level folder **1. Maritime Lines**, or **2. Seabed** 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. Geoc ap 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
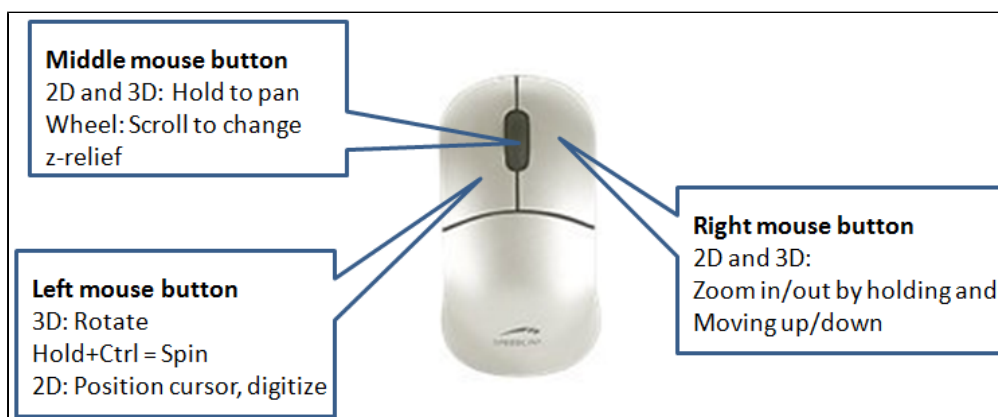
The mouse buttons

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.

C. Generation of Distance Lines

Introduction

Calculating distances can be done in many different ways. Geocap uses algorithms that take into account the curvature of the Earth which provides for better accuracy and is in accordance with the scientific and technical guidelines established by the Commission on the Limits of the Continental Shelf (CLCS) for UNCLOS Article 76. Any distance can be calculated in Geocap but the main ones are:

- 12M Territorial Seas.
- 24M Contiguous Zones.
- 200M Exclusive Economic Zones
- 60M Formula Lines.
- 100M Constraint Lines.
- 350M Constraint Lines.

> Note that our Maritime Zones plugin extends the Shelf package with additional capabilities for calculating maritime zones. It works directly with ArcGIS feature classes and can calculate any maritime zone from both normal and straight baselines. The calculation also extracts critical points for each zone and store them in a feature class. Read more about the: Maritime Zones plugin

Exercises

- Generate the 200M line
- Display base point connection polygon
- Check Table View for the 200M line
- Generate 350M line
- Use the Distance Calculation tool

200M Line Generation

Exercise
Generate the 200M line

1. In the folder **1. Maritime lines / Base lines** right-click **Baseline_Atlantis** and select the command **Generate 200M line...**
2. Make sure that **Generate base point connection polygon** is checked. The menu should look exactly as the screenshot below.
3. Click **Execute**.
4. The new line should now be available under **1. Maritime Lines/200M Lines**. Click **OK**.
5. Click **OK** to close the menu and keep the settings.
6. In the folder **1. Maritime lines / 200M lines** tick the box next to the **Baseline_Atlantis + 200M** dataset to display it.
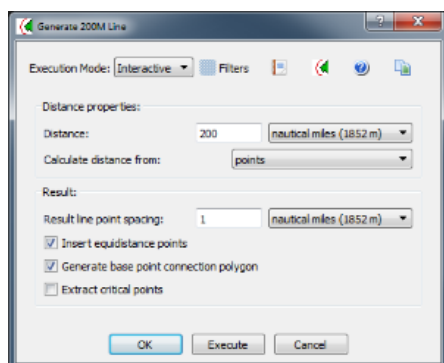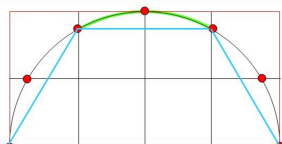
Generate 200M line menu

Information on point spacing

The arc on an "envelope of arcs" in reality consist of a series of short chords. The point spacing represents the length of these chords.
In the image below you will see that a larger area will be lost (outside blue chords) if we double the point spacing on the arc.


It is recommended that the chord lengths (point spacing) are determined such that the distance between the arc and the chord never exceeds 1 meter. Below you will find a table of distance lines and the corresponding point spacing which fulfills this recommendation.

Distance line (M)	Point spacing (M)
3 Nautical Miles	0.1 Nautical Miles
12 Nautical Miles	0.2 Nautical Miles
60 Nautical Miles	0.5 Nautical Miles
200 Nautical Miles	1.0 Nautical Miles
350 Nautical Miles	1.0 Nautical Miles

Note that not all points on the baseline will contribute to the generation of the **200M** line. To visualize which points on the baseline that contributes to the 200M we can display the **base point connection polygon**.

Exercise

Display base point connection polygon

1. In the folder **1. Maritime lines / Base lines** right-click **Baseline_Atlantis** and click **Display Points**
2. In the folder **1. Maritime lines / 200M lines** tick the box next to the **Baseline_Atlantis + 200M Connection Polygon** dataset to display it.
3. In the folder **1. Maritime lines / 200M lines** right click **Baseline_Atlantis + 200M Connection Polygon** and click **Display Points.**
4. Notice how the polygons points back to the critical points on your baseline (see image below).
5. Toggle the display between polygon mode and wire mode with the **W** and **S** keys.
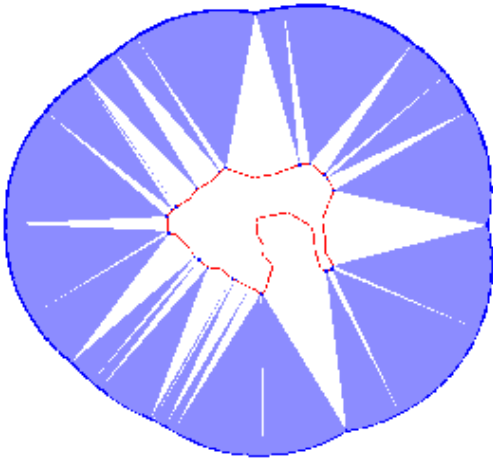
200M line with critical points on baseline in blue

> It is possible to store the points which contributes to the 200M line as a separate dataset in your project. This can be done by checking the **Extract critical points** on the **Generate 200M line...** menu

---

Exercise

Check Table View for the 200M line

1. In the folder **1. Maritime lines / 200M lines** right click **Baseline_Atlantis + 200M** and click **Table View.**
2. Observe the columns, **Base Line**, **Base Line Index** and **Second Base Line Index**. This tells you which baseline and which point on the baseline that contributed to the point on the 200M line.
3. Scroll down and select a point that has values in both **Base Line Index** and **Second Base Line Index.** If **Display marked points** is checked the point should be highlighted in the graphics window.
4. Observe that this is the crossing points between two arcs in an envelope of arcs calculation (equidistance point). That is why it is referring to two base points.
5. Click **Close** to close **Table View**

350M Line Generation

Exercise

Generate 350M line

1. In the folder **1. Maritime lines / Base lines** right-click **Baseline_Atlantis** and select the command **Generate 350M line...**
2. Keep the default settings and click **Execute**.
3. The new line should now be available under **1. Maritime Lines / 350M Lines**. Click **OK**.
4. Click **OK** to close the menu and keep the settings.
5. In the folder **1. Maritime lines / 350M lines** tick the box next to the **Baseline_Atlantis + 350M** dataset to display it.

-

To verify that the distances are correct we can use the **Distance Calculations** tool.

Exercise
Use the Distance Calculation tool

1. Open the tool from **Tools > Distance Calculations**.
2. Click **YES** to put Geocap in 2D
3. Select coordinate system **Mercator** (all values 0)
4. For distance calculation method, select **Vincenty calculation**. (**Flat earth** calculates in the metric values of the projection)
5. Select **Distance unit: nautical mile**
6. To measure the distance between two points, press **j** on the keyboard to snap to them**.**
7. The distance is displayed in the lower part of the panel. Notice that vertical, horizontal and total distance is measured and that lat/long of the select point is listed.
8. Click the **X** to close the menu
9. Click **3D** to get back into 3D view.

D. Bathymetry

Introduction

The use of bathymetry is a very central element when working with the determination of the extension of the continental shelf according to UNCLOS Article 76. It is relevant both for the depth constraint (2500m depth +100M) and the determination of the foot of the continental slope.

In Geocap bathymetry can have several different forms:

- Singlebeam data may be imported as bathymetric profiles.
- Multibeam soundings or other spread point data with depth measurements can also be imported. In order to use the foot of slope analysis tool on point data, the dataset must be gridded first. This can be done in Geocap.
- Grids may be imported directly into Geocap. Geocap supports import of several different grid file formats. Most Shelf projects will consist of a low resolution grid (i.e Etopo or SRTM) covering the whole area of interest and then smaller high resolution grids from multibeam data.

The Etopo or SRTM grids will **not** be accepted by the Commission as a basis for your foot of slope points. However once you have imported the grid you will be able to see the shape of the seafloor, and in particular the approximate position of the Foot of Slope or base of slope region.

Exercises

- Generate the 2500m isobath from a Seabed Surface
- Edit the generated 2500m isobath
- Generate 2500m isobath + 100M constraint line
- Generate a bathymetric profile from a grid
- Generate more bathymetric profiles in the East Sea area
- Group your bathymetric profiles into a folder
- Rename your bathymetric profiles

2500m isobath + 100M constraint

Computing the 2500m isobaths is necessary in order to determine the Depth constraint in UNCLOS Article 76, which is the 2500m isobaths plus 100 nautical miles. The 2500m isobaths can be generated based on bathymetry grids or on bathymetric profiles (from single beam bathymetry). The only ones that can be used for an actual submission to the CLCS are 2500m isobath points generated from real bathymetry data. For a grid, the resulting 2500m isobath will be a contour line. A bathymetric profile will produce one or more points wherever the profile value is 2500m.

Exercise

Generate the 2500m isobath from a Seabed Surface

1. In the folder **2. Seabed / Grids** right click the **atlantis** dataset and select **Generate 2500m Isobath.**
2. The command will generate a new dataset. Click **OK.**
3. In the folder **2. Seabed / 2500 meter isobath** check the box next to the dataset **atlantis's 2500 meter isobath.**

Notice how parts of the 2500m isobath are smaller contour lines around sea mounts etc. We want to remove these before we generate the 2500m + 100M constraint line.

Exercise

Edit the generated 2500m isobath

1. Clear the display window by clicking the **X** button in the main toolbar.
2. In the folder **2. Seabed / 2500 meter Isobath** right click **atlantis's 2500 meter isobath** and select **Edit points and lines..**.
3. Click the **Display** button near the bottom of the **Edit points and lines** dialogue box in order to see the isobaths.
4. Select the **Delete** tab.
5. Inside the **Delete** tab, select **By closed line** tab.
6. Click the **Start** button, then digitize a closed polygon around the part you want to delete by clicking with the left mouse button.
7. Click **Connect to Start and End** in order to connect the last digitized point with the start point.
8. Click **Delete points INSIDE.** The part inside the closed polygon is now deleted.
9. Change the drop down menu from **Save as an extra copy** to **Overwrite input dataset.**
10. In order to store the changes in the **Edit points and lines** menu, click the **Execute** button at the bottom.
11. Click **Cancel** to close the **Edit points and Lines** menu.
12. Clear the display window by clicking the **X** button in the main toolbar.
13. In the folder **2. Seabed / 2500 meter Isobath** right click **atlantis's 2500 meter isobath** and select the **Display** and confirm that the unwanted parts have been removed.
14. The **Edit points and lines** command stored the original 2500m isobath as a backup. Right click this dataset and select **Delete.**
15. Click **OK**.

We can now generate the last constraint line.

Exercise

Generate 2500m isobath + 100M constraint line

1. In the folder **2. Seabed / 2500 meter Isobath** right click **atlantis's 2500 meter isobath** and select **Generate 100M Line.**
2. Keep the default settings and click **Execute.**
3. The new line should now be available under **1. Maritime Lines / 100M Lines**. Click **OK.**
4. Click **OK** to close the menu.
5. In the folder 1. **Maritime Lines / 100M Lines** display **atlantis's 2500 meter isobath + 100M.**

-

Bathymetric Profiles

To determine the foot of the continental slope we need bathymetric profiles. These profiles will be used as input to the foot of slope analysis tool in the next chapter. Bathymetric profiles can be imported from single beam or generated from bathymetric grids. In this tutorial we will use the atlantis seabed surface grid (similar to Etopo1) to generate bathymetric profiles. The project already contains several bathymetric profiles grouped into different areas. Our task is to complete the foot of slope analysis in region called East Sea. In this section we will generate the bathymetric profiles, give them a proper name and group them into a folder.
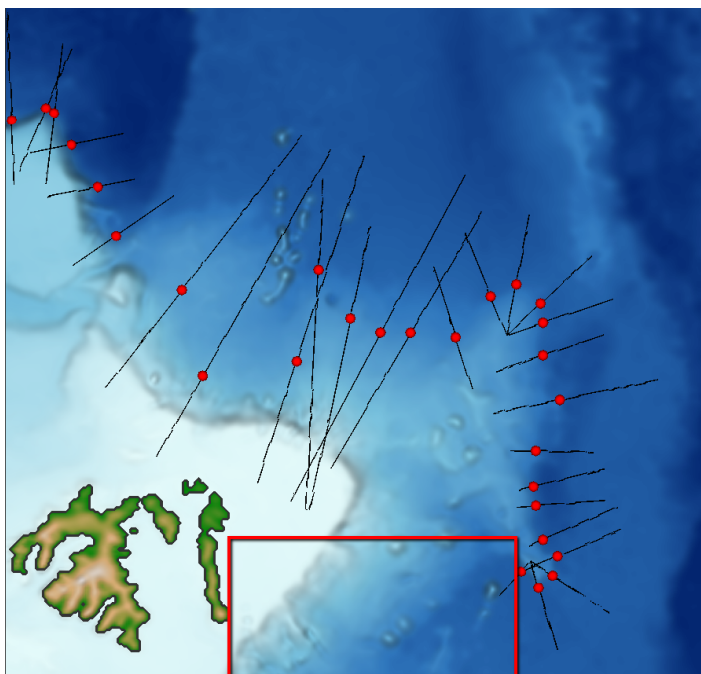
East Sea is the area in red

To ensure the most accurate calculation of the foot of slope the bathymetric profiles should run perpendicular to the seabed surface contours.

Exercise

Generate a bathymetric profile from a grid

1. Display the **atlantis** seabed surface dataset in **2. Seabed / Grids** and zoom in to the East Sea area.
2. Right click the **atlantis** seabed surface dataset again and select the command **Generate Bathymetric Profile**. A menu will appear.
3. Click the **Start** button, then digitize the first point of the profile by clicking with the left mouse button where you want the profile to start on the seafloor in the display window.
4. Digitize the last point of the bathymetric profile by clicking with the left mouse button where you want the profile to stop on the seafloor.
5. Click **Stop**
6. You will be prompted with a dialog asking you to provide a name for the profile. Keep the default name and click **OK**.
7. Geocap will notify you where the resulting Bathymetric profile has been stored in the project. Click **OK**.
8. Locate the new profile in **2. Seabed / Bathymetric Profiles.** This is the default position of bathymetric profiles.
9. Display the profile.

> It is possible to generate a bathymetric profile following a specific path or feature by digitizing more than two points.

Exercise

Generate more bathymetric profiles in the East Sea area

Repeat the steps in the previous exercise and generate a few more profiles in the same area. You should have 3-5 profiles in the area.

We should now group our bathymetric profiles into a folder and give them a proper name.

Exercise

Group your bathymetric profiles into a folder

1. Right click the folder **2. Seabed / Bathymetric Profiles** and select **New > Folder**.
2. Select **Generic** and type in the name **East Sea.**
3. Click **OK**.
4. Right click the **East Sea** folder and select **Set Schema > Bathymetric Profiles**
5. Locate the profiles you generated in **2. Seabed / Bathymetric Profiles.**
6. Select all the profiles, right click and select **Cut**
7. Right click the new **East Sea** folder and select **Paste.** The profiles are now moved into the new folder.

Exercise

Rename your bathymetric profiles

1. Right click one of the profiles in the **East Sea** folder and select **Rename.**
2. Type in the name **East Sea 1.**
3. Click **OK**
4. Continue to rename the other profiles in the same manner. **East Sea 2** etc.

> It is advisable to name the profiles with numbers increasing from south to north.

E. Foot of Slope Analysis & Hedberg points

Introduction

UNCLOS Article 76 states that *"the foot of the continental slope shall be delineated as the point of maximum change in the gradient at its base".* FOS analysis is crucial because both formula lines (Gardiner and Hedberg) are calculated using FOS positions. In Geocap there are several tools for analyzing the gradient of the slope, based on different data sets. The **Gradient Band Analysis** is often used for identifying the approximate location of the base of slope region, based on bathymetric grids. To calculate the accurate location of a foot of slope (FOS) point the **Bathymetric Profile Analysis** panel is used.

In this section we will analyze one foot of slope point for each of the bathymetric profiles we created in the previous section. Then we will proceed to generate 60M lines (Hedberg) from these FOS points.

Exercises

- Visualizing gradient bands
- Analyze a bathymetric profile and create a FOS point
- Analyze and create FOS points for the remaining bathymetric profiles in the East Sea area
- Change an analysis and FOS point
- Display FOS points
- Investigate the FOS point table view
- Joining FOS points into one FOS group dataset
- Generating 60M line to the FOS points
- Investigate the FOS + 60M table view
- Import a bathymetric profile
- Save an adjusted bathymetric profile as a new bathymetric profile
- Smooth the bathymetry

Base of Slope

Paragraph 3 of article 76 states that *"The continental margin comprises the submerged prolongation of the land mass of the coastal State, and consists of the sea-bed and subsoil of the shelf, the slope and the rise. It does not include the deep ocean floor with its oceanic ridges or the subsoil thereof".* This means that the base of slope region is the transition region from the rise to the deep ocean floor. We can try to analyze and visualize this transition zone using the **Gradient Band Analysis.**

The **Gradient Band Analysis** command will divide a grid into different areas depending on the steepness of the grid. The areas will be display as color bands. It is possible to adjust the boundaries between the different areas. The boundaries are set in degrees.

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

1. In the folder **2. Seabed / Grids** right click **atlantis** and select **Advanced Display > Gradient Band Analysis**.
2. Set the **Smoothing steps** to **5**.
3. Keep the rest of the settings as default and click **Execute.**
4. Investigate the color bands and the steepness. Are the boundaries reasonable?
5. Try to change the angles and click **Execute.**
6. Can you find values that could represent the shelf, slope, rise and the deep ocean floor in the East Sea region?
</td></tr>
</table>
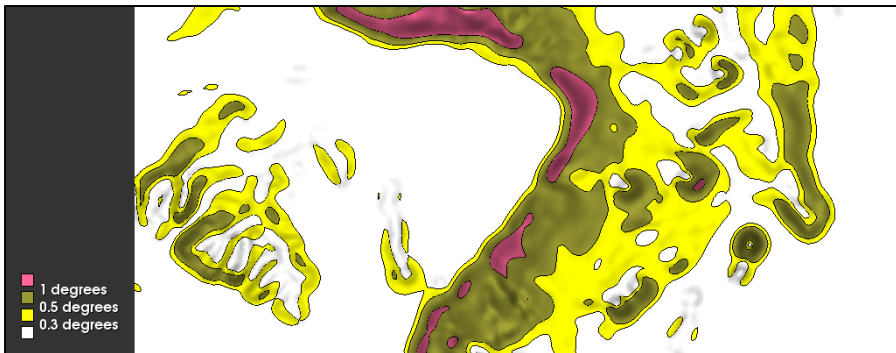
Gradient Band Analysis

> Note that the Gradient Band Analysis only takes morphological data into account. Geological and geophysical data can also be used to identify the base of slope region.
> For more information refer to the Scientific and Technical Guidelines of the Commission on the Limits of the Continental Shelf found here: http://www.un.org/Depts/los/clcs_new/commission_guidelines.htm

Foot of Slope Analysis

The Analysis Panel is one of the most important tools in Geocap because it allows you to determine the Foot of the Continental Slope in accordance with UNCLOS Article 76.
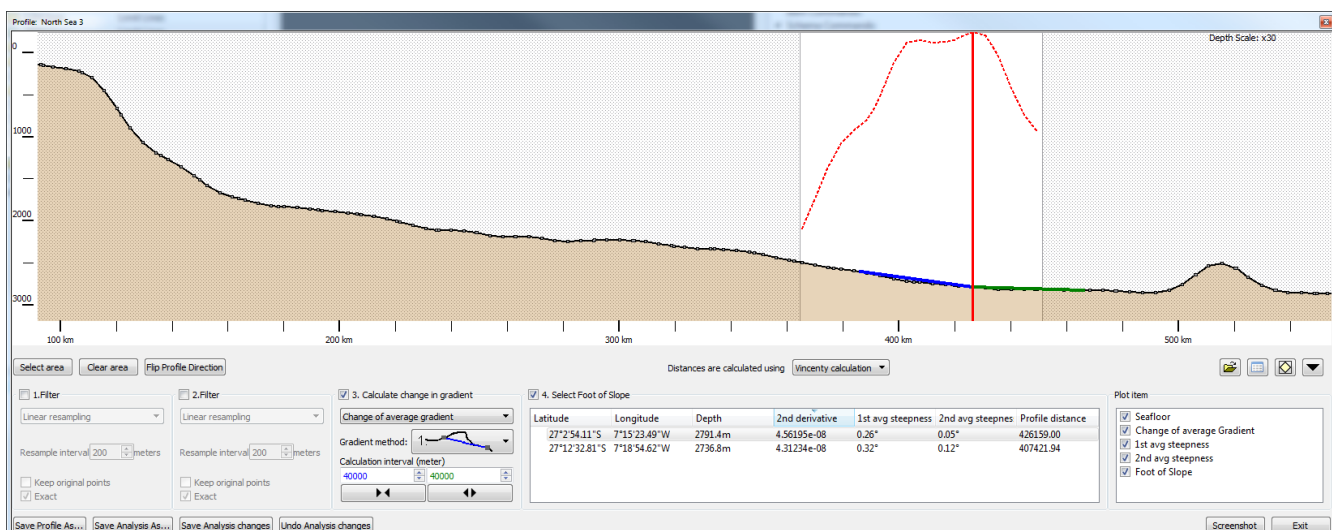
The elements of the FOS analysis panel

Important elements in the FOS analysis panel:

- The **Select area** button lets you select the area you are interested in analyzing. The FOS point will end up inside this area.
- The **Calculate change in gradient** is where you set up the mathematical methods and variables that go into the FOS point calculation.
- The **Select Foot of Slope** lists the possible FOS points in the area. The point with the larges change in gradient will be listed at the top and selected by default.
- The **Plot item** list the different elements in the plot. Elements can be turned on and off or edited. It is also possible to add additional elements.
- The **Save Analysis As...** button lets you save an analysis and the corresponding FOS point.

A typical workflow for analyzing a foot of slope point would be:

1. Select an area of interest - This should represent your base of slope region.
2. Select a method for calculating the gradient - Observe the difference for the three methods.
3. Select a calculation interval - These values should represent the general trend in the area.
4. Save the Analysis and FOS point.

The analysis and FOS point will be stored under the bathymetric profile

Profile, Analysis and FOS point

Exercise

Analyze a bathymetric profile and create a FOS point

1. In the folder **2. Seabed / Bathymetric Profiles / East Sea** right click the **East Sea 1** profile the and select **Analyze Profile.** The profile analysis tool will appear at the bottom of your screen. The same profile is also visible in the 3D window above the analysis panel where you will notice the red vertical line marking the maximum change of gradient.
2. Click the **Select area** button.
3. Press the left mouse button in the profile window, drag the mouse to the side, and release the button. You have now selected the area where the software will find the FOS point.
4. Try increasing the **Calculation interval** on both sides (blue and green) by typing in a higher value (i.e 25 000).
5. When you think you have identified the FOS point. Click the **Save Analysis as..** button.
6. Keep the default name for the Analysis and click **OK**.
7. Type in the FOS name **ES 1** and click **OK**.
8. The FOS point is now saved. Click **OK**.
9. Click the triangle (or plus sign) next to the **East Sea 1** profile in the project and you will see that the FOS analysis is stored as a child of the profile in the tree.
10. Click the triangle (or plus sign) next to the **FOS analysis** dataset, and you will see the **ES 1** fo ot of slope dataset. This is the data set holding the foot of slope point we just made.
11. Click the **Exit** button in the Profile Analysis Panel to close it.

It is possible to display Base of slope (BOS) polygon or other lines which crosses the bathymetric profile in the profile window. Click the browse icon on the analysis panel, browse in the line or polygon and it wil l be displayed as a vertical line.

Exercise

Analyze and create FOS points for the remaining bathymetric profiles in the East Sea area

1. In the folder **2. Seabed / Bathymetric Profiles / East Sea** right click the next **East Sea** profile the and select **Analyze Profile.**
2. Use the analysis panel to find the FOS point and click **Save Analysis as..**
3. Keep the analysis name and type in the FOS name **ES *<number corresponding to bathymetric profile>*** and click **OK**.
4. Click the **Exit** button in the Profile Analysis Panel to close it.
5. Follow the steps above until you have an analysis and FOS point for all bathymetric profiles in the East Sea area.

-

In many cases you want to go back and look at the analysis you did for a specific FOS point and maybe change the analysis and update the FOS point. In the next exercise we will change the FOS point for an existing analysis.

Exercise

Change an analysis and FOS point

1. In the folder **2. Seabed / Bathymetric Profiles / East Sea,** expand the **East Sea 1** profile so you can see the FOS analysis dataset.
2. Right click the **FOS analysis** dataset and click **Analyze.** This will open the profile with the settings you used for this analysis.
3. Change some of the settings in the analysis in order to move/change the FOS point. (i.e selected area, calculation interval, method).
4. Click **Save Analysis changes**.
5. This will overwrite the previous settings. Click **OK**.
6. Click **Exit** to close the analysis panel.

Exercise

Display FOS points

1. In the folder **2. Seabed / Bathymetric Profiles / East Sea** expand the **East Sea 1** profile and analysis so you can see the **ES 1** dataset.
2. Right click **ES 1** and select **Display FOS points.**
3. Notice that the FOS point is displayed as a sphere in the graphics window.
4. Repeat the steps above to display all your FOS points in the East Sea area.

Exercise

Investigate the FOS point table view

1. In the folder **2. Seabed / Bathymetric Profiles / East Sea** expand the **East Sea 1** profile and analysis so you can see the **ES 1** dataset.
2. Right click **ES 1** and select **Table View.**
3. Notice that the various field data contains the information from the analysis panel.
4. Click **Close**.

-

Generating Hedberg points (FOS + 60M)

Once we have our FOS points we can generate the 60M Hedberg points which is one of the formula lines in article 76. It is possible to generate a 60M circle around each FOS point but it is often useful to work with a group of FOS points instead of working with them individually. The next exercises will show you how to collect many FOS points into one foot of slope dataset and then generate 60M Hedberg points around those FOS points.

Exercise

Joining FOS points into one FOS group dataset

1. In the folder **2. Seabed / Bathymetric Profiles / East Sea** expand the **East Sea 1** profile and analysis so you can see the **ES 1** dataset.
2. Right click **ES 1** and select **Merge Foot of Slope Points.** This will open a menu with a list containing your first FOS point. We are now going to populate this list with the rest of the FOS points.
3. Keep the menu open and select the **ES 2** dataset in your project. Click the

   icon in the merge menu. This will add the FOS point to your list.
4. Repeat the previous step to add all your FOS points in the East Sea area to the list.
5. In the folder **2. Seabed / Foot of slope** select the folder **Region Groups** and click the

   icon in the merge menu to select the **Result** folder.
6. Click **Execute**.
7. Type in the name **East Sea** and click **OK.** This will store your FOS group in the **Region Groups** folder.
8. Click **Cancel** to close the menu.
9. In the folder **2. Seabed / Foot of slope / Region groups** right click **East Sea** and click **Display FOS points.**

We are now going to create the Hedberg lines from these FOS points.

Exercise

Generating 60M line to the FOS points

1. In the folder **2. Seabed / Foot of slope / Region groups** right click **East Sea** and click **Generate 60M Line.**
2. Keep the default settings and click **Execute**.
3. The new line will be stored in **1. Maritime Lines / 60M Lines.** Click **OK.**
4. Click **OK** to close the menu.
5. In the folder **1. Maritime Lines / 60M Lines** right click **East Sea + 60M** and click **White (FOS+60).**

The Table View contains important metadata about the Hedberg points.

Exercise

Investigate the FOS + 60M table view

1. In the folder **1. Maritime Lines / 60M Lines** right click **East Sea + 60M** and click **Table View.**
2. Notice how the **FOS Point Name** changes as you scroll down the list of points.
3. Click **Close**

-

Real Bathymetric Profiles

Real bathymetric profiles are measured, usually by sonar. They are not generated by extracting values from a seabed grid or from any other data derived mainly from satellite altimetry, such as ETOPO1. An example of real bathymetry is "Corrected Depth Bathymetry" from the U.S. National Oceanic and Atmospheric Administration's (NOAA) Geophysical Data System (GEODAS). Bathymetric profiles often come in ASCII formats with X Y Z or Lat Lon Z values. These type of files can be imported using the ASCII importer in Geocap. In the next exercise we are going to import a bathymetric profile using this import menu.

Exercise

Import a bathymetric profile

1. In the folder **2. Seabed / Bathymetric Profiles** right click **Singlebeam** and click **Import > ASCII Column.**
2. Click the browse button and browse for the dataset called **Cruise-12** (*Data/Atlantis/Bathymetry/Singlebeam/Cruise-12.txt*)**.** Click **Open**
3. Make sure **Adjust header** is set to **0**.
4. Make sure **Coordinate format** is set to **XY**.
5. Make sure **Z** is checked.
6. Make sure **Scalars** is unchecked.
7. Make sure that **File column X**=**1**, **Y**=**2**, **Z**=**3**
8. Make sure that **Cell type** is set to **Line**
9. Make sure that **Schema** is set to **Bathymetric Profile**
10. Click **Execute**.
11. You will be prompted with a dialog asking you for Coordinate System and Datum. Select **World Geodetic System 1984** and **Mercator** (all values 0).
12. Click **OK**.
13. The file is now imported**.** Click **OK**.
14. Click **OK** to close the import menu.

The GEODAS bathymetric profiles can sometimes be very long, covering areas that you are not interested in. In addition to this they can contain noise etc. It is possible to adjust the length of the profiles in Geocap, so that you will only analyse the part you need. It is also possible to filter/smoothen the data. We are now going to adjust the length of the profile we just imported and then use filters to smoothen it.

Exercise

Save an adjusted bathymetric profile as a new bathymetric profile

1. In the folder **2. Seabed / Bathymetric Profiles / Single Beam** right click **Cruise-12** and click **Analyze Profile**.
2. In the 3D graphics window find the white spheres at the end of the profile and adjust them by dragging the spheres to the desired position.
3. In the bottom left corner of the analysis panel click on **Save profile As...**
4. Type in the name **Cruise-12-East-Sea** and click **OK**.
5. The new profile will be saved in the same folder as the original. Click **OK**.
6. Click **Exit to close the** analysis panel.
7. Right-click the new bathymetric profile in your project and click **Analyze Profile.** Notice the new bathymetric profile is now a sub-set of the original profile.

> **Tip**
> In some instances the white spheres may be very small when the analysis panel is first opened. It may be necessary to zoom in closer to find one of the ends of the profile. Once the white sphere is clicked on once, the white spheres on both ends will become larger and easier to see when dragging them to the desired position.

The bathymetry in the Cruise-12 singlebeam dataset is very "choppy" for some reason, and we want to smoothen the dataset,

> As a general rule we do not recommend filtering and smoothing of bathymetric profiles unless it is absolutely necessary. If you do use filtering in an FOS analysis the Commission might request full disclosure of the original data, the mathematical details of the smoothing algorithm and the output data. For more information refer to section 5.3 in the Scientific and Technical Guidelines of the Commission on the Limits of the Continental Shelf found here: http://www.un.org/Depts/los/clcs_new/commission_guidelines.htm

---

Exercise

Smooth the bathymetry

1. In the analysis panel check the box in front of **1.Filter.** Notice that a thin yellow line is shown on top of your seabed in the analysis window.
2. Change the filter to e.g. **Moving Average**.
3. Try changing the **Calculation point radius** number and observed the difference in the filtered line.
4. Try to find a FOS point you think you can justify and save the analysis by clicking **Save Analysis As**
5. Give the FOS point a proper name and click **OK**.
6. Click **OK** again.
7. Click **Exit** to close the analysis window.

> It is possible to store the filtered (e.g. smoothed) bathymetry as a separate dataset. Click the **Tables of values** button to the right in the analysis panel. The filtered result is then available in a tab, and may be extracted as a project dataset.

F. Sediment Thickness Grid and 1% distance to FOS

Introduction

Sediment thickness plays an important role in UNCLOS article 76, since it is used to define the Gardiner formula points. These are points where the thickness of sedimentary rocks is at least 1 per cent of the shortest distance to the foot of the continental slope.

NOAA/NGDC provides a sediment thickness grid of the Worlds Oceans here: http://www.ngdc.noaa.gov/mgg/sedthick/index.html. This dataset is very coarse and will not be accepted by the Commission. It is however a dataset which can provide you with an overview of where you might potentially have enough sediments to use the Gardiner formula.

In this section we will use a sediment thickness grid to identify areas where there might be enough sediments to identify Gardiner points. Then in the next section we will take a closer look at seismic data in order to calculate the sediment thickness more accurately.

Exercises

- Display Sediment Thickness Grid
- Calculate 1% of the distance to the closest FOS point

Sediment Thickness Grid

Exercise

Display Sediment Thickness Grid

1. In the folder **3. Sediment Data / Grids** right click **Sediment Thickness** and click **Map Data**
2. The grid displayed shows thickness in meter and has no reference to depth. Blue is the thickest part while red is the thinnest part.
3. In **1. Maritime lines / 60M lines** display all your Hedberg points.
4. Try to identify an area where you might have enough sediments.

Calculate 1% distance to FOS

In the next exercises we are going to identify the areas of the grid where the thickness is more than 1% of the distance back to the closest foot of slope point. Geocap can calculate this in one operation taking all FOS points into account.

Exercise

Calculate 1% of the distance to the closest FOS point

1. In the folder **3. Sediment Data / Grids** right click **Sediment Thickness** and click **Generate >1% dist to fos Area**
2. In the menu that pops up, click the **Browse** icon and browse for the **Northern Plateau** dataset in the **2. Seabed / Foot of slope / Region Groups** folder.
3. Make sure that **Store in project** is **unchecked.**
4. Click **Execute**
5. Click **OK** to close the menu
6. The area displayed has a thickness which is more than 1% of the distance back to one or more of the foot of slope points in the **Northern Plateau** dataset**.**

In the next section we will take a closer look at the area in the north.

G. Seismic Interpretation

Introduction

In order to accurately calculate the sediment thickness we have to interpret the seabed surface and the base of sediments on seismic lines. In Geocap these two layers are called horizons. The distance between the seabed and basement horizons is the sediment thickness.

When collecting seismic data a seismic wave travels from its source to a given reflector and returns to a receiver at the sea surface. The elapsed time is called two way travel time. In most cases this will be the depth unit (in milliseconds) of your seismic. Since the seismic is in time (milliseconds) and we need to calculate the sediment thickness in meters we need to do a conversion. This is called a depth conversion. The formula itself is simple: Distance = Speed * Time. The difficult part of the formula is finding the correct speed/velocity model, since the velocity changes through the different layers in the subsurface. In Geocap the interpretation is normally done in the time domain. This gives us the flexibility to change the velocity model and just update the sediment thickness calculation accordingly. We will have a look at different velocity models in the next section.
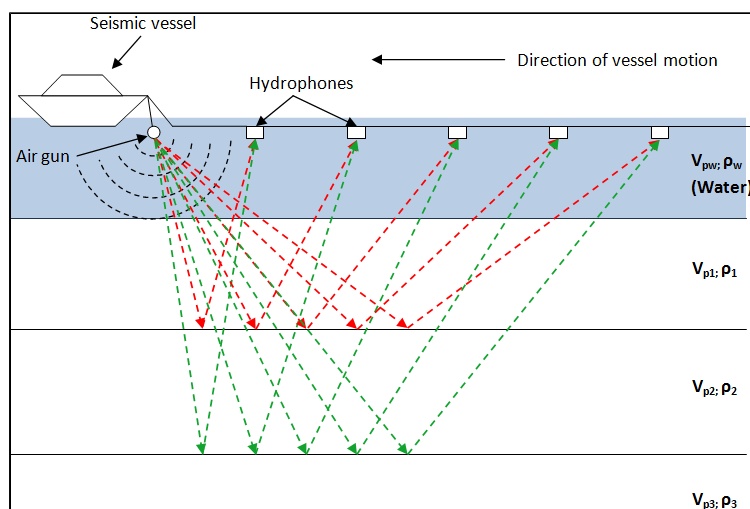
Illustration of a marine seismic survey using a towed streamer

In this section we will visualize seismic data, import seismic and interpret seismic.

Exercises

- Display a seismic line
- Display the interpretation
- Interpret the Seabed
- Interpret the Sediment Base
- Import SEG-Y files
- Auto track the Seabed
- Complete the Sediment Base interpretation

Display Seismic Lines and Interpretation

Exercise
Display a seismic line

1. In the folder **3. Sediment Data / Seismic Data / ATL-99** right click the seismic line **ATL-99-1** and click **Seismic Display**
2. In the folder **3. Sediment Data / Seismic Data / ATL-LOS-00** right click the seismic line **ATL-LOS-00-1** and click **Seismic Display**
3. Investigate the seismic lines. Can you identify the Seabed and Sediment Base?

Exercise

Display the interpretation

1. In the folder **3. Sediment Data / Seismic Interpretations** right click the Ihorizon **Seabed** and click **IHorizon Display.**
2. In the folder **3. Sediment Data / Seismic Interpretations** right click the Ihorizon **Sediment Base** and click **IHorizon Display.**
3. Investigate the interpreted horizons. Notice that there is only interpretation for the first line.

Interpret Seismic Lines

In the next exercises we are going to complete the interpretation of the Seabed and Sediment Base for the seismic line ATL-LOS-00-1.

Exercise

Interpret the Seabed

1. Go to **Tools > Seismic Interpretation**. This is the interpretation system in Geocap
2. Select **Interpetation** folder: Click the browse button, select the folder **Seismic Interpretations** and click **OK**
3. Select **Ihorizon:** Click the browse button, select **Seabed** and click **OK**
4. Select the **Seismic**: Click the

    Select

   button and click on the seismic line ATL-LOS-00-1 in the display window (with your left mouse button)
5. Keep the rest of the settings as default and click the

   button.
6. Zoom to where the yellow line ends on the previous line and start to pick points on the seismic line by pointing the mouse cursor on the seabed and pressing the **<space>** key on the keyboard. Press **d** to undo a pick.
7. When you want to stop, click the

   button. This will save the interpretation. You can continue to interpret the seabed by clicking the

   button again.
8. When you are done with the seabed, click the

   button.

You do not have to delete parts of the interpretation if it is wrong. Just re-interpret the part that is wrong and Geocap will automatically update the Ihorizon when you click the stop button.

We are now going to interpret the Sediment Base but before we can do this we need to change the Ihorizon we are interpreting on.

Exercise

Interpret the Sediment Base

1. Change **Ihorizon:** Click the browse button, select **Sediment Base** and click **OK**
2. Keep the rest of the settings and click the
   ⮕
   button.
3. Zoom to where the blue line ends on the previous line and start to pick points on the seismic line by pointing the mouse cursor on the sediment base and pressing the **<space>** key on the keyboard. Press **d** to undo a pick.
4. When you want to stop, click the
   🛑
   button. This will save the interpretation. You can continue to interpret the sediment base by clicking the
   ⮕
   button again.
5. When you are done with the sediment base, click the
   🛑
   button.
6. Click **Close** on the Seismic Interpretation menu.

Your interpretation should look something like this when you are finished:
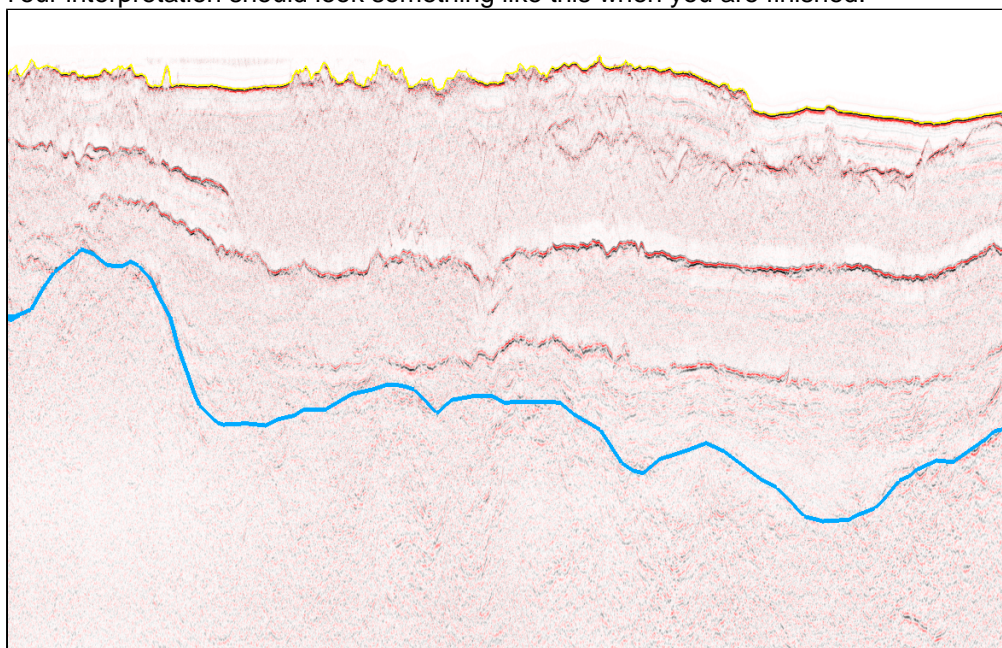
Seabed and Sediment Base interpretations

Import Seismic Lines

There is one more seismic line in this area but it has not been imported yet. In the next exercise we will import this line and then continue to interpret it.

The format for seismic lines is SEG-Y. This format was developed by the Society of Exploration Geophysicists (SEG) in 1973. Since then format has evolved in to many different "flavors", which sometimes makes it difficult to import. When importing a SEG-Y file it is important to set the byte positions correctly. If you are lucky there might be some information about the byte positions in the EBCDIC header, and if you are even luckier the default settings in Geocap will be correct.

This exercise will not go into details on seismic/SEG-Y import. It will cover the simple case for a plain SEG-Y file which follows the SEG-Y standard and has navigation stored in the trace header. If you have problems importing a SEG-Y file at a later stage, please contact support@geocap.no


Exercise

Import SEG-Y files

1. In the folder **3. Sediment Data/Seismic Data** right click the folder **ATL-LOS-00** and click **Import > SEG-Y 2D**. The SEG-Y import menu will appear.
2. Click the **Defaults** button in order to reset the menu to the default options.
3. Click the browse button and select the **ATL-LOS-00-2.sgy**-file located in the Atlantis folder structure on disk (*Data/Atlantis/Sediment Data/ATL-LOS-00-2.sgy*). Click **Open**.
4. Look at the information in the **EBCDIC header** and see if the information matches the import settings.
5. Click **Scan.** Geocap will investigate the SEG-Y file.
6. Click the **Scan results** tab. Investigate the range of the file and that the coordinates, shot point range etc. looks correct.
7. Click **Execute**
8. You will be prompted with a dialog asking you for **datum** and **projection** for the seismic line. Select **WGS 84** and **Mercator 0** and click **OK**
9. Click **OK** again
10. Click **OK** in order to close the SEG-Y import command.
11. In the folder **3. Sediment Data / Seismic Data / ATL-LOS-00** right click the seismic line **ATL-LOS-00-2** and click **Seismic Display**

Interpret Seismic Lines continued

We are now going to complete the interpretation of the Seabed and Sediment Base on the seismic line ATL-LOS-00-2. This seabed on this seismic line is very flat and has a strong continuous reflector. This lets us use the auto tracker to automatically interpret the seabed.


Exercise

Auto track the Seabed

1. Go to **Tools > Seismic Interpretation**. This is the interpretation system in Geocap.
2. Select **Interpetation** folder: Click the browse button,select the folder **Seismic Interpretation** and click **OK.**
3. Select **Ihorizon:** Click the browse button, select **Seabed** and click **OK**
4. Select the **Seismic**: Click the

   [Select]

   button and click on the seismic line ATL-LOS-00-2 in the display window (with your left mouse button)
5. Change the **Snap mode** to **Min**
6. Change the **Track mode** to **Auto**
7. Keep the rest of the settings and click the

   [→]

   button.
8. Zoom in close to the seabed, point the mouse cursor anywhere on the seabed and press the **<space>** key. This will automatically interpret the seabed.
9. Zoom out and see if the whole seabed was interpreted. If not, continue to press the **<space>** key where it is missing interpretation.
10. When the whole seabed has been interpreted, click the

    [STOP]

    button.

We are now going to interpret the Sediment Base on the seismic line ATL-LOS-00-2.

Exercise

Complete the Sediment Base interpretation

1. Change **Ihorizon:** Click the browse button, select **Sediment Base** and click **OK**
2. Change the **Snap mode** to **Off**
3. Click the

   ➡️

   button.
4. Complete the sediment base interpretation and click

   🛑
5. **Close** the seismic interpretation menu when you are done.

The seabed and sediement base interpretation for all lines should look something like this:
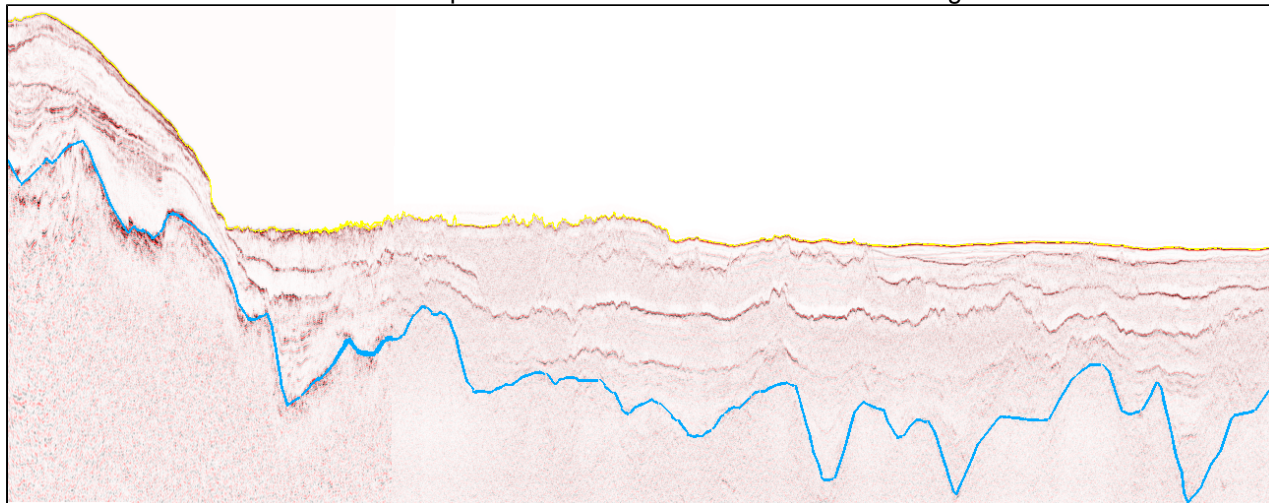
Seabed and Sediment Base interpretations

H. Depth Conversion & Thickness Calculations

Introduction

In order to use the sediment formula criterion in UNCLOS Article 76, we need a data set containing the sediment thickness; a sediment profile. The seismic lines interpreted in the previous section is stored with milliseconds as a z coordinate, not meters. In order to compute a sediment thickness in meters, we need to do a depth conversion. In this section we will cover the basics for working with velocities, depth conversion and sediment thickness calculation.

Exercises

- Import a velocity data set
- Assign navigation to the velocity datasets
- Display the Stacking Velocities
- Convert stacking velocities to interval velocities
- Generate velocity profile from seismic interpretation using interval velocity dataset
- Generate a sediment profile
- Display the sediment profile
- Inspect the table view of the sediment profile

Velocities

Velocities are often provided as ASCII column files. The file should contain the shot point, time, and velocity. The file may also contain coordinates. In our training data set, there are no coordinates, therefore we need to assign navigation from a navigation data set. If the file contains velocities from more than one line, it should also have a column with the line name.

Exercise
Import a velocity data set

1. In the folder **3. Sediment Data / Velocities** right click the folder **Stacking Velocities** and click **Import > Stacking Velocities (no navigation)**
2. Click the browse button and browse for the data set called **ATL-LOS-00.vel** (*Data/Atlantis/Sediment data/ATL-LOS-00.vel*). Click **Open.**
3. Adjust the columns according to the screen shot below by clicking up or down in the spin box.
4. Click **Execute.**
5. You will be prompted with a dialog asking you to specify **Datum** and **Coordinate System.** Click **All Undefined.**
6. The dataset has been imported. Click **OK**
7. Click **OK** on the **Stacking Velocity** import menu.
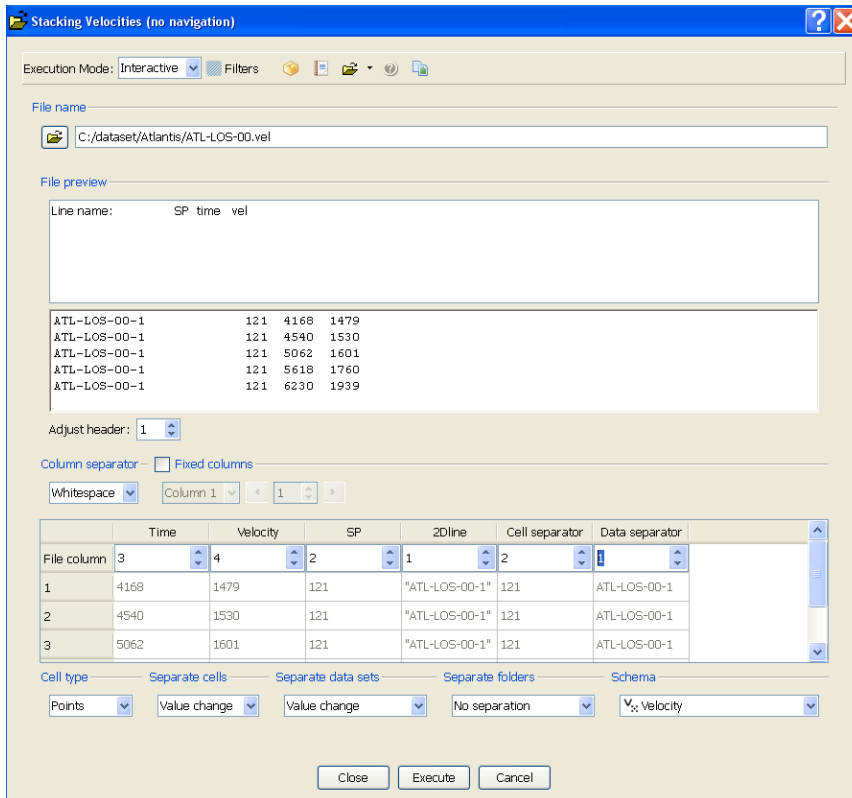
Stacking Velocity import menu

The data sets have now been imported into your project. However they do not yet hold any coordinates. In order to visualize the stacking velocities with the seismic data, we need to assign navigation to the velocities. Geocap will use the shot point number in the velocity data set and match them with the shot points in a navigation file. Either a separate navigation file, or the actual seismic line can be used as source of navigation.

Exercise

Assign navigation to the velocity datasets

1. In the folder **3. Sediment Data / Velocities / Stacking Velocities** right click the velocity dataset **ATL-LOS-00-1** and click **Assign Navigation.**
2. Browse for the seismic line **ATL-LOS-00-1** in the **Seismic Data folder** and click **OK.**
3. In the folder **3. Sediment Data / Velocities / Stacking Velocities** right click the velocity dataset **ATL-LOS-00-2** and click **Assign Navigation.**
4. Browse for the seismic line **ATL-LOS-00-2** in the **Seismic Data folder** and click **OK.**

The velocity data sets are now in the right location and can be displayed together with the seismic data.

Exercise

Display the Stacking Velocities

1. In the folder **3. Sediment Data / Velocities / Stacking Velocities** right click the velocity dataset **ATL-LOS-00-1** and click **Map Data.**
2. In the folder **3. Sediment Data / Velocities / Stacking Velocities** right click the velocity dataset **ATL-LOS-00-2** and click **Map Data.**
3. Display the two seismic lines in the folder **3. Sediment Data / Seismic Data / ATL-LOS-00.**
4. Investigate the velocity data together with the seismic data.

If you have imported stacking velocities or RMS velocities, we need to convert them to interval velocities using

Dix formula before we can use them to generate velocity profiles. Dix's formula will use the stacking velocities as an approximation for RMS velocities and produce interval velocities.

Exercise

Convert stacking velocities to interval velocities

1. In the folder **3. Sediment Data / Velocities** right click the folder **Stacking Velocities** and click **Compute Interval Velocities (Dix Formula)**
2. The result will be stored in the folder **3. Sediment Data / Velocities / Interval Velocities.** Click **OK.**

Velocity Profiles

We now need to calculate the average velocity from the seabed down to the sediment base, along the seismic lines. To do this we have to generate velocity profiles.

Exercise

Generate velocity profile from seismic interpretation using interval velocity dataset

1. In the folder **3. Sediment Data / Velocities** right click the folder **Velocity Profiles** and click **Generate Velocity Profile.**
2. Click the first browse icon and browse in the seismic data folder **3. Sediment Data / Seismic Data / ATL-LOS-00** and click **OK.**
3. Click the second browse icon, browse in the folder **Seismic Interpretations** and click **OK.**
4. Make sure **Match points by:** is set to **Shot Point.**
5. Make sure **Use interval velocities from:** is set to **Dataset in Project.**
6. Click the **Interval Velocities** browse icon, browse in the **Interval Velocities** folder and click **OK.**
7. Leave the rest of the settings as default and click **Execute.**
8. In the menu that pops up make sure that both lines are checked and click **OK**
9. The result is stored in the folder **3. Sediment Data / Velocities / Velocity Profiles**
10. Click **Cancel** to close the menu.

If you do not have velocity data available it is possible to use a velocity look-up table or formula. See documentation for more information.

-

Depth conversion and thickness calculation

The next step is to perform the depth conversion and calculate the sediment thickness. In Geocap this can be done in one operation by generating a sediment profile. The velocity profile holds all the data necessary to do this.

Exercise

Generate a sediment profile

1. In the folder **3. Sediment Data / Velocities** right click the folder **Velocity Profiles** and click **Calculate Sediment Thickness.**
2. In the menu that pops up make sure that **Calculate Sediment Thickness** as been selected.
3. Click **Execute.**
4. The result will be stored in in **3. Sediment Data / Sediment profiles.** Click **OK.**
5. Click **OK** to close the menu

Exercise

Display the sediment profile

1. In the folder **3. Sediment Data / Sediment profiles** right click the sediment thickness profile **ATL-LOS-00-1** and click **Display profile as Cross Section**
2. In the folder **3. Sediment Data / Sediment profiles** right click the sediment thickness profile **ATL-LOS-00-2** and click **Display profile as Cross Section**
3. Try displaying the **atlantis** seabed grid also and compare it with the sediment profiles.

Exercise

Inspect the table view of the sediment profile

1. In the folder **3. Sediment Data / Sediment profiles** right click the sediment thickness profile **ATL-LOS-00-1** and click **Table View**
2. Inspect the different columns. The **X**,**Y** and **Z** is the position and the seabed depth at that point. The **Scalar** value is the sediment thickness in meters. The Shot point column is the points corresponding shot point from the seismic line.
3. Click **Close** to close the **Table View**

I. Gardiner Points

Introduction

In the previous section we calculated sediment thicknesses. In order to apply the sediment thickness formula (Gardiner) in UNCLOS Article 76, we need to compare the sediment thickness with the distance to the foot of slope. When working with 1% sediment thickness points it is often useful to experiment with different datasets and settings. You might for example change the velocity model and see how that effects the final Gardiner points, or you might try to change the FOS points to see if that has any effect on the final result. In this section we are going to experiment with different percentages, instead of looking just at 1%, and see how much this affects the calculations.

Exercises

- Display sediment thickness points > 1% of distance to FOS
- Use different percentages to calculate x% sediment thickness points
- Store the 1% sediment thickness (Gardiner) points
- Examine the metadata for sediment thickness points

Visualizing 1% sediment thickness points

The **Generate > 1% of distance to FOS Area** command uses a FOS dataset as input to the calculation. This can be a single FOS point or a group of many FOS points. Geocap will automatically find the FOS point which is closest to any sediment thickness point and then do the 1% calculation. Since all our profiles are located in the Northern Plateau we know that our **Northern Plateau** FOS points will be the closest, so we will use this dataset.

Exercise

Display sediment thickness points > 1% of distance to FOS

1. In the folder **3. Sediment Data / Sediment Profiles** right click the profile **ATL-LOS-00-2** and click **Generate > 1% of distance to FOS Area.**
2. Click the browse icon, browse for the **Northern Plateau** dataset and click **OK.**
3. Make sure the **Display as cross section** is checked.
4. Uncheck the **Store in project** box.
5. Click **Execute.**
6. Examine the result. Points thicker than 1% of the distance to the nearest FOS point will display the sediment profile in green. Points less than 1% distance to FOS is displayed in red.
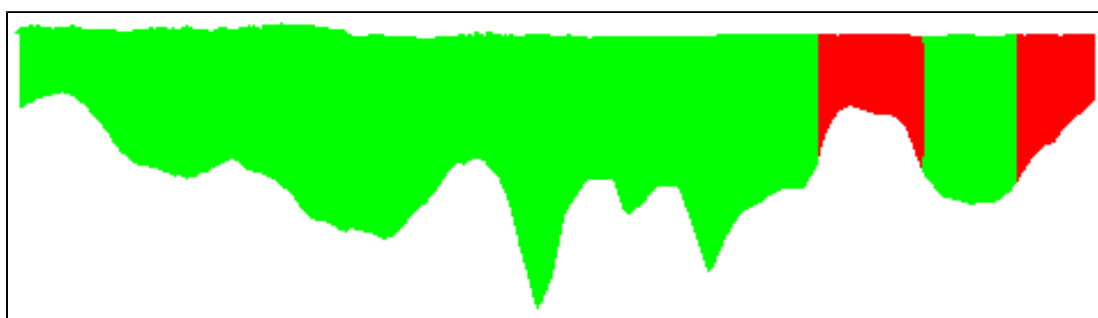
Sediment profile showing thickness more (green) and less (red) than 1% of the distance back to FOS

You now have a set of Gardiner points, but what would happen if you changed your interpretation, velocity model or FOS points? What is the uncertainty of the sediment thickness? An easy way of visualizing how "stable" your Gardiner points are is to change the percentage of distance to FOS.

Exercise

Use different percentages to calculate x% sediment thickness points

1. Change the **Percentage of distance to FOS** to **1.1%**.
2. Keep the rest of the settings and click **Execute.**
3. Investigate the result. Did it change? In which direction? Why?
4. Change the **Percentage of distance to FOS** to **0.9%**
5. Keep the rest of the settings and click **Execute.**
6. Investigate the result. Did it change? In which direction? Why?

It is also possible to look at the uncertainty of your sediment thickness by increasing or decreasing it with i.e 10%. This can be done using the **Function** command and multiplying the sediment thickness points with **1.1** or **0.9.**

Saving 1% sediment thickness points

We now want to store our Gardiner points as a separate dataset so we can continue to work with them.

Exercise

Store the 1% sediment thickness (Gardiner) points

1. Change the **Percentage of distance to FOS** back to **1%**.
2. Check the box **Store in project**.
3. Make sure **Display as cross section** is checked.
4. Click **Execute**.
5. A new data set will be created in **3. Sediment Data / Sediment Thickness Points.** Click **OK**.
6. Repeat the process for all sediment profiles.
7. All profiles should now be displayed with green and/or red colors representing more than 1% sediment thickness and less than 1% sediment thickness.

The 1% sediment thickness (Gardiner) points datset contains a lot of relevant metadata, like which FOS point was used in the calculation, the exact percentage, the shot point, the distance in meters etc. This is information that should be documented in the final submission.

Exercise

Examine the metadata for sediment thickness points

1. In the folder **3. Sediment Data / Sediment Thickness Points** right click the **ATL-LOS-00-2** G ardiner points and click **Table View**.
2. Observe that that each 1% sediment thickness point refers back to a bathymetric profile and a FOS point. It also reports the thickness, shot point, distance in meters and percent of distance to FOS.
3. Identify the Gardiner point furthest away from the coastline, it should be the last one in the table. Tip: Check the **Display marked points** box and select the last point in the table.
4. Which FOS point contributed to this Gardiner point? What is the percentage of the distance back to this FOS point?

J. Final Outer Limits

Introduction

UNCLOS Article 76 states that *"The coastal State shall delineate the outer limits of its continental shelf, where that shelf extends beyond 200 nautical miles from the baselines from which the breadth of the territorial sea is measured, by straight lines not exceeding 60 nautical miles in length, connecting fixed points, defined by coordinates of latitude and longitude."*

At this point we have created all the formula lines and constraint lines and we can proceed to construct the final outer limit. The construction of outer limits is a complex operation but the **Generate Outer Limit** tool is designed to assist you in the best possible way. The tool lets you digitize and combine line segments and keep distances between points less than 60M. Sometimes it is best to do this area by area and in multiple steps, then combining points into one or more final outer limits. In many cases the best approach is to construct and outer edge line consisting of only Hedberg points, Gardiner points and the 200M line, and then construct a constraint line consisting of the 350M line and the 2500m + 100M isobath. These two lines can then be combined into one final outer limit.

In the exercises in this section we will try to produce a final outer limit using the **Generate Outer Limit** tool.

Exercises

- Copy your data to the Outer limit folder
- Getting familiar with the Outer Limit Menu
- Digitize the outer edge
- Investigate the Outer Edge Limit line metadata
- Optimize the Outer Edge Limit
- Digitize the constraint line
- Produce Final Outer Limit

The Outer Limit Menu

Before we can use the **Generate Outer Limit** tool we need to populate the **4. Outer limit** folder with the necessary data. If you look at the folders inside the outer limit folder you will see three folders called; **Constraints, Formulae** and **Outer Limit.** We need to copy our distance lines, Hedberg points and Gardiner points into these folders.

Exercise

Copy your data to the Outer limit folder

1. In the folder **1. Martime lines / 200M lines** right click **Baseline_Atlantis + 200M** and click **Copy**.
2. In the folder **4. Outer limit** right click the folder **Formulae** and click **Paste**.
3. In the folder **1. Martime lines / 60M lines** select all the lines, right click**,** and click **Copy**.
4. In the folder **4. Outer limit** right click the folder **Formulae** and click **Paste**.
5. In the folder **3. Sediment Data / Sediment Thickness Points** select all the lines, right click**,** and click **Copy**.
6. In the folder **4. Outer limit** right click the folder **Formulae** and click **Paste**.
7. In the folder **1. Martime lines / 100M lines** right click **atlantis's 2500 meter isobath + 100M** and click **Copy**.
8. In the folder **4. Outer limit** right click the folder **Constraints** and click **Paste**.
9. In the folder **1. Martime lines / 350M lines** right click **Baseline_Atlantis + 350M** and click **Copy**.
10. In the folder **4. Outer limit** right click the folder **Constraints** and click **Paste**.

All necessary data should now be in one of the folders in the Outer limit folder.

When digitizing the outer limit it is possible to activate different datsets to be available during the construction of

the outer limit. We can now take a closer look at the Outer Limit Menu.

Exercise

Getting familiar with the Outer Limit Menu

1. Right click the folder **4. Outer limit** and click **Generate Outer Limit**
2. Observe the different datasets and folders in the menu.
3. Check the box next to the **Formulae** folder.
4. Click the display icon at the top of the menu to display all the activated datasets.


The Outer Limit Menu

-
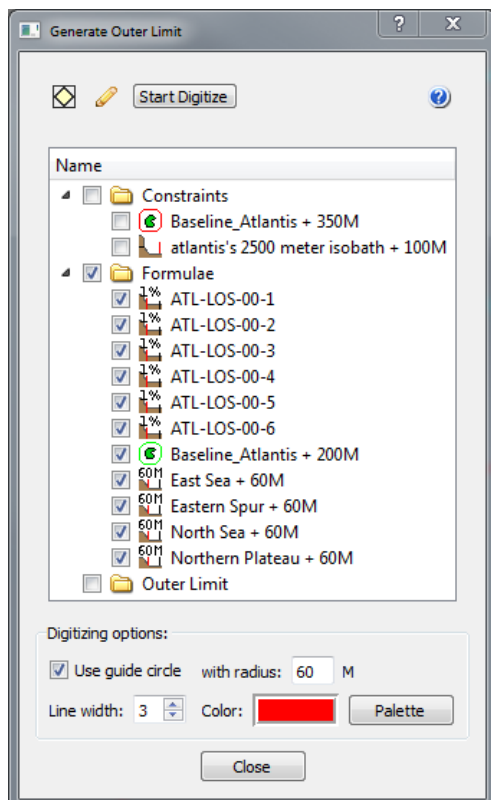
Digitize the Outer Limit

We can now start to digitize the outer limit. We will do this in three steps; first digitize the outer edge consisting of formula lines, then digitize a constraint line consisting of the distance constraints, and then finally combine these two lines.

The digitizer has a range of different keyboard shortcuts that you need to use when constructing an outer limit:

- **left click** (mouse) - will snap to the closest point on any line.
- **c** - crossing lines - will insert a new point between two lines, or a line and the 60NM circle.
- **d** - delete - will delete the last digitized point.
- **l** - connect to line - will insert a new point on the line at the cursor position.
- **s** - select line piece - will select all the points on a line until next crossing line or to end of line.
- **m** - multiple selection - will select all points from previous point to picked point (mouse cursor location).

For the first exercise we have included a screen shot of the steps in the exercise.
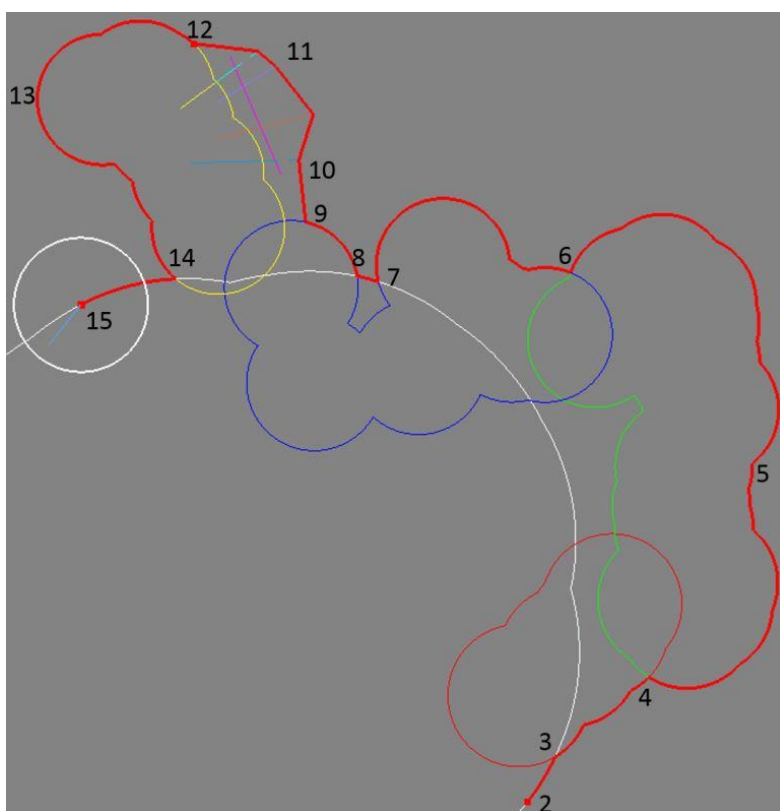
Exercise

Digitize the outer edge

1. Click the **Start Digitize** button.
2. Snap to a point on the 200M line by placing your mouse cursor over the line in the East Sea area and clicking the **left mouse button**.
3. We are now going to follow this line north until we meet the crossing 60M line. Place your cursor close to the crossing between the 200M line and the 60M line and press **s** on the keyboard.
4. We now want to follow this 60M line until me meet the next crossing 60M line. Place your cursor close to the crossing between the 60M line we are at and the next 60M line and press **s**.
5. Again we want to follow this 60M line until me meet the next crossing 60M line, but we want to make sure it follows the outer part of the 60M line. Place your cursor somewhere on the outer part of the 60M line and press **m**.
6. Follow the line to the next 60M crossing line by placing your cursor close to the crossing between the 60M line we are at and the next 60M line and press **s**.
7. We now want to follow this 60M line until me meet the crossing 200M line. Place your cursor close to the crossing between the 60M line we are at and the 200M line and press **s**.
8. We now want to follow the 200M line until me meet the crossing 60M line again. Place your cursor close to the crossing between the 200M line and the crossing 60M line and press **s**.
9. We now want to follow this 60M line until we can reach the sediment thickness points. Place your cursor further up on the 60M line and press **m**.
10. When you can see the sediment thickness points inside the white circle you can connect to them by clicking the **left mouse button.** Tip: place the cursor outside the point furthest out to snap to the "maximum" point.
11. Continue to click the **left mouse button** until you have passed the sediment thickness area.
12. We now want to connect to the 60M line in the north. Place your cursor close to the crossing between the 60M line and the white 60M circle and press **c**.
13. We now want to follow this 60M line until me meet the crossing 200M line, but we want to make sure it follows the outer part of the 60M line. Place your cursor somewhere on the outer part of the 60M line and press **m**.
14. Follow the line to the 200M crossing line by placing your cursor close to the crossing between the 60M line and the 200M line and press **s**.
15. We now want to follow this 200M line a little bit further to the west. Place your cursor somewhere to the west on the 200M line and press **m**.
16. Stop digitizing by clicking **End Digitize.**
17. You will be asked to specify a name. Type in **Outer Edge Limit** and click **OK**.
18. The dataset is stored in **4. Outer limit / Outer Limit** click **OK**.
19. Click **Close** to close the Outer Limit Menu.

The digitized outer limit lines contains a lot of important metadata which will be used in the documentation. Each point on the outer limit line will refer back to the original FOS point or basepoint. It also contains information on point distance, shot point, sediment thickness and a lot more.

Exercise

Investigate the Outer Edge Limit line metadata

1. In the folder **4. Outer limit / Outer limit** right click **Outer Edge Limit** and click **Table View.**
2. Look at the information in the different columns and scroll down the list of points.
3. Click **Close** when you are done.

One important part of the outer limit line definition is that it should consist of: *straight lines not exceeding 60 nautical miles in length, connecting fixed points.* This means we can bridge some of the concave areas of the Outer Edge Line to gain more area

---

Exercise

Optimize the Outer Edge Limit

1. In the folder **4. Outer limit / Outer limit** right click **Outer Edge Limit** and click **Optimize outer limit**.
2. Keep the default settings and click **Execute.** The result will be stored in the same folder.
3. Click **OK** to close the menu.
4. In the folder **4. Outer limit / Outer limit** right click **Outer Edge Limit Optimized** and click **Display**.
5. Investigate the areas that have been bridged.

The **Optimize outer limit** tool is based on angle calculation and not area calculation. The angles are calculated in what ever coordinate system the data is in. Thus, It cannot be assumed this tools will automatically produce the absolute maximum achievable area.

We are now going to digitize the constraint line. It is important to remember that we want to select the constraint line which is furthest out at any point.

Exercise

Digitize the constraint line

1. Make sure the graphics window has been cleared by clicking **X**.
2. Right click the folder **4. Outer limit** and click **Generate Outer Limit**.
3. Check the box next to the **Constraints** folder.
4. Click the display icon at the top of the menu to display all the activated datasets.
5. Click the **Start Digitize** button.
6. Digitze the constraint line using a combination of your mouse and keyboard short cuts.
7. Click **End Digitze**.
8. You will be asked to specify a name. Type in **Constraint Line** and click **OK**.
9. The dataset is stored in **4. Outer limit / Outer Limit** click **OK**.
10. Click **Close** to close the Outer Limit Menu.

Finally we can produce the final outer limit. Follow the **Outer Edge Optimized** where it is inside the **Constraint Line**, else follow the **Constraint Line**. In the 1% sediment area, connect by the 60M Circle.

Exercise

Produce Final Outer Limit

1. Make sure the graphics window has been cleared by clicking **X**.
2. Right click the folder **4. Outer limit** and click **Generate Outer Limit**.
3. Check the box next to the two datasets **Outer Edge Limit Optimized** and **Constraint Line**.
4. Click the display icon at the top of the menu to display all the activated datasets.
5. Click the **Start Digitize** button.
6. Digitze the final outer limit line using a combination of your mouse and keyboard short cuts.
7. Click **End Digitze**.
8. Keep the default name and click **OK**.
9. The dataset is stored in **4. Outer limit / Outer Limit** click **OK**.
10. Click **Close** to close the Outer Limit Menu.

K. 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

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

Actors

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

Visualizations are called actors

Schemas

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

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

The Shelf Module contains several schemas. Some of the schemas used in the Shelf Module are **coast line**, **ba**

se line, **limit line**, **seabed surface**, **sediment thickness**. You can define the schema of a dataset in the project by right clicking it, and selecting **set schema** in the pop-up menu. The choice of a dataset´s schema controls which commands you see in the pop-up menu when you right click the dataset. You may create your own schemas as well as edit existing schemas by selecting **schemas** under edit in the main menu. You can also edit the commands associated with the schemas.

Commands

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

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

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

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

An example of a command front end panel

Commands can be stored in three categories:

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

You will find the commands sorted into the different categories in the **Toolbox** or on the right-click menu of a dataset or a folder. Commands can also be put together in sequence in **Workflows** to perform visualizations or data operations.

Item commands

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

Exercise

Create an item command

1. Go to **1. Maritime lines / 60M lines** and select one of the datasets.
2. In the **Toolbox** right click **Display** and select **Edit.**
3. Set **Line Width** to **4**.
4. Check **User defined Color** and set the color to **Red**.
5. Click the

   icon in the upper right corner of the menu and observe that the command appears in the **Tool box** under **Item Commands**.
6. Click **Cancel** (the settings will not be saved for the schema command).
7. Right click the **Display** command under **Item Commands** and select **Rename**.
8. Rename it to **Display in red** and click **OK**.
9. Select the different lines in **1. Maritime lines / 60M lines** and observe that the new **Display in red** command is only available for the one dataset.

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 **2.Seabed / 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. **1. Maritime lines > 200M lines > Baseline_Atlantis + 200M** 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.

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

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.

L. Build your own Shelf (UNCLOS) Project

Introduction

We have now been through all the main steps in building a Shelf project, and it is time to look at a real example. In this section we are going to build a project from scratch in an area you want to look closer at. We are going to use public data available online or provided by us, analyze them and generating the datasets we needed for a submission. The first part of this section will mainly be about importing data and converting it to the correct projection. In the last part we will produce the distance lines, analyze the bathymetry, generate Hedberg points and Gardiner points and hopefully produce an outer limit.

Exercises

- Generate a new and empty UNCLOS project
- Download coastline files from the Internet
- Import and display the GSHHS coastline files
- Convert coast line to the right projection and display
- Download the ETOPO1 grid from the Internet
- Import the seabed grid
- Convert the grid to the right projection and display
- Import the sediment thickness grid
- Convert the grid to the right projection and display
- Generate dummy baseline
- Import baseline
- Generate the 200M and 350M lines
- Clean up the 200M and 350M lines
- Generate the 2500m isobath
- Edit the generated 2500m isobath
- Generate the 2500m isobath + 100M distance constraint
- Generate Bathymetric Profiles
- Download Bathymetric Profiles from GEODAS
- Import Bathymetric Profiles from GEODAS
- Analyze your profiles
- Group FOS points
- Generate 60M Hedberg points
- Calculate 1% of the distance to the closest FOS point
- Copy your data to the Outer limit folder
- Generate the outer limit

The UNCLOS Project Template

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

Geocap can provide an empty folder structure for UNCLOS projects. This will give you a starting point to get organized with your own UNCLOS 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 example a sub-folder for each survey or for each region. Other subfolders commonly created are folders for FOS Collections, Images etc.

Exercise

Generate a new and empty UNCLOS project

- In the main menu, click **File > New > Project**.
- Select UNCLOS project template.
- In the **Name** field type in **Geocap_Training_<Year>** . Project names may consist of letters, numbers and spaces, but special characters like [, æ,ø, å, &, /, % ... should be avoided.
- Click the **Browse** button, and select where you want to store your project on your hard disk.
- Click **Finish**.
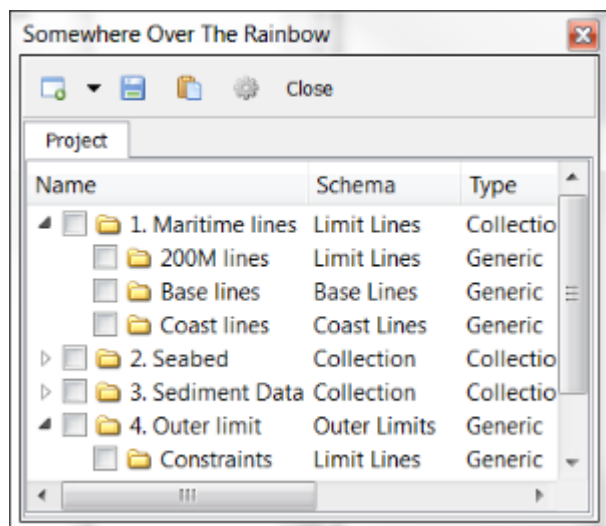
An empty Shelf project

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.

Import Data

We will now explain the basics of importing datasets and working with coordinate systems in Geocap. Most datasets in Geocap are imported using the **ASCII import** or the **Generic import.** The **Generic import** is used to import a number of different formats, such as **images**, **grids**, **shapefiles** etc. When browsing in a file, the **Gener ic import** will recognize the file format and let you define an **Area of Interest** to import. The **ASCII import** reads data from an ASCII file into the Geocap project. **ASCII import** can be used for many types text data files, e.g. whether the data columns have fixed or variable width, and also whether the coordinates are in X Y or Lat Lon. When data is read into the project, it can be organized in a number of different ways. It is for example possible to tell Geocap to split the data into different dataset whenever a value in a specific file column changes. There are also options for creating project folders whenever a column value changes.

We have already used the ASCII import in a previous section so in this section we are going to import different datasets using the **Generic import** method and convert the datasets to the projection we want to use in our project.

Coastline

We are now going to import a coastline file. National Geophysical Data Center (NGDC) provides coast lines covering the entire world in a data set they call GSHHS (Global Self-consistent, Hierarchical, High-resolution Shoreline Database). Geocap can read both the raw data format (the files gshhs_*.b files), and the shapefiles directly. In this exercise we are going to import the *.b files.

If you have limited internet connection you can skip this exercise and continue with the next one since we have provided the dataset already.

Exercise

Download coastline files from the Internet

1. Open a web browser and go to this NGDC web page: http://www.ngdc.noaa.gov/mgg/shorelin es/gshhs.html.
2. Click **Download GSHHS Data version 2.3.4** (January 1, 2015).
3. Download and save the file **gshhs-bin-2.3.4.zip** *(09-Mar-2015 11:14 113M)*.
4. Uncompress the files to a folder on your hard disk.

Exercise

Import and display the GSHHS coastline files

1. Locate the folder **1. Maritime Lines/Coast Lines** in your Geocap project.
2. Right click the folder and select **Import > Generic…**.
3. The format should be set to **Automatic**.
4. Click the browse button and locate the folder with the GSHHS files.
5. Select the file called **gshhs_f.b** and click **Open**.
6. Click the **Area of Interest** tab.
7. Check the **Import Area** check box.
8. Type in the minimum and maximum latitude and longitude in decimal degrees.
   a. **Max Y** = Northern boundary of import area **Min X** = Western boundary of import area.
   b. **Max X** = Eastern boundary of import area **Min Y** = Southern boundary of import area.
9. Click **Execute**.
10. Geocap will recognize the datum and coordinate system of the file and confirm that the file has been read. Click **OK**.
11. After the file has been read, click the **OK** button in the Generic reader dialog.
12. Observe that the imported dataset is stored in the **Coast lines** folder.
13. Right click the dataset and select **Zoom to Data**.
14. Right click the dataset and select **Display**.

The data is imported in geographical coordinates with latitude as Y and longitude as X. In order to view the data in a proper projection we convert the data to to the projection we want to use.

Exercise

Convert coast line to the right projection and display

1. Click the dataset and go to the **Shared commands** section in the **Toolbox**.
2. Double click the **Convert to...** command you want to use, in the **Operations** folder.
3. Define the appropriate settings.
4. Change the result combo box to **Replace input**.
5. Click **Execute**.
6. Agree to replace the existing dataset by clicking **Yes**.
7. The data is now converted. Click **OK.**
8. Click **OK** to close the conversion menu.
9. Observe that the display is automatically updated to the new projection.

-

Global Seabed Grid

ETOPO1 is a 1 arc-minute global relief model of Earth's surface that integrates land topography and ocean bathymetry. In the next exercises we will download and import this grid into Geocap.

Exercise

Download the ETOPO1 grid from the Internet

1. Open a web browser, and go to the following address: http://www.ngdc.noaa.gov/mgg/global/global.html.
2. Click the link: **Extract Custom Grids** - in the left column on the page, below the globe. A new web page with a interactive map should appear.
3. Zoom into the area where you want to extract the grid.
4. Click the icon with and **i** in the upper left corner of the map window.
5. Drag a rectangle over the area where you want to extract the grid. You should see a red rectangle on the map.
6. Keep Layer as **ETOPO1 (ice)**.
7. Select Output Format **XYZ**.
8. Click "**click here to download**".

Exercise

Import the seabed grid

1. Locate the folder called **Grids** in your Geocap project under **2. Seabed**.
2. Right click the folder and select **Import > Generic…**
3. The format should be set to **Automatic.**
4. Click the browse button and locate the ETOPO file.
5. Click the **Area of Interest** tab.
6. Check the **Import Area** check box.
7. Type in the minimum and maximum latitude and longitude in decimal degrees.
   a. **Max Y** = Northern boundary of import area **Min X** = Western boundary of import area
   b. **Max X** = Eastern boundary of import area **Min Y** = Southern boundary of import area
8. Click on the **Reader Options** tab and under polydata and check the box next to **Invert** (The grid has negative values for sea depths, and positive values for heights above sea level. Usually bathymetry data has positive values for depths. We therefore have to multiply the depth and height values by -1 with this option to "flip" the data).
9. Click **Execute**.
10. You will now be prompted with a question of the coordinate system and datum. Select: **World Geodetic system 1984**, **Geodetic** and click **OK**.
11. Geocap will report if the grid has been imported correctly; click **OK** and close the import dialog by clicking **Cancel**.
12. Observe that the new data set has appeared in the **Grids** folder.

The data is imported in Geodetic, with latitudes as Y coordinates and longitudes as X coordinates. In order to view the grid with the data we have already imported and converted to another projection, we need to convert this grid to match the same projection as the other data in the project .

<table>
<tr><td colspan="2">Exercise</td></tr>
</table>

Exercise

Convert the grid to the right projection and display

1. Click the seabed grid and go to the **Shared commands** section in the **Toolbox**.
2. Double click the **Convert to...** command you want to use, in the **Operations** folder.
3. Define the appropriate settings
4. Change the result combo box to **Replace input**.
5. Click **Execute**.
6. Agree to replace the existing dataset by clicking **Yes**.
7. Geocap will report if the grid has been converted correctly, click **OK** and close the conversion dialog by clicking **Cancel**.
8. Check that the schema of the dataset is **seabed surface**. If not then right click the dataset, choose **Set Schema** and choose **seabed surface**.
9. Display the dataset together with the coastline.

Global sediment thickness

The Global Sediment thickness can be downloaded as a grid here: http://www.ngdc.noaa.gov/mgg/sedthick/index.html

Exercise

Import the sediment thickness grid

1. Locate the folder called **Grids** in your Geocap project under **3. Sediment Data** (you might have to create it)
2. Right click the folder and select **Import > Generic…**
3. The format should be set to **Automatic.**
4. Click the browse button and locate the sediment thickness file.
5. Click the **Area of Interest** tab.
6. Check the **Import Area** check box.
7. Type in the minimum and maximum latitude and longitude in decimal degrees.
   a. **Max Y** = Northern boundary of import area **Min X** = Western boundary of import area.
   b. **Max X** = Eastern boundary of import area **Min Y** = Southern boundary of import area.
8. Click on the **Reader Options** tab and make sure **Invert** is unchecked.
9. Click **Execute**.
10. Geocap will report if the grid has been imported correctly; click **OK** and close the import dialog by clicking **Cancel**.
11. Observe that the new data set has appeared in the **Grids** folder.

Exercise

Convert the grid to the right projection and display

1. Click the seabed grid and go to the **Shared commands** section in the **Toolbox**.
2. Double click the **Convert to...** command you want to use, in the **Operations** folder.
3. Define the appropriate settings.
4. Change the result combo box to **Replace input**.
5. Click **Execute**.
6. Agree to replace the existing dataset by clicking **Yes**.
7. Geocap will report if the grid has been converted correctly, click **OK** and close the conversion dialog by clicking **Cancel**.
8. Right click the dataset, choose **Set Schema** and choose **Sediment Thickness**.
9. Display the dataset together with the coastline.

-

Generate or import a Baseline

The baseline is needed in order to generate distance lines. The coast may also be used as a baseline (by changing the schema accordingly), but then the computing time may be very long. If you do not have a baseline for your area, you may generate a dummy to use as an approximation based on the coast line you imported in a previous exercise. If you already have imported an official baseline you may jump to the import baseline exercise.

Exercise
Generate dummy baseline

1. Display the coast line called GSHHS, which should have been imported in **1. Maritime Lines / Coast lines**.
2. Open the Digitizer by selecting **Tools > Quick digitizing** from the main menu.
3. Check the option **Save also in folder**, and click the **Browse** button.
4. Select the **Base lines** folder under **1. Maritime Lines**, and click **OK.**
5. Click **Start digitizing.**
6. Digitize the points on the coast line by pressing **j** on the keyboard. The picks will automatically snap to the nearest point on the coast line data set. Digitize a rough version of the coast line by picking the outermost points on the coast line.
7. When you are finished digitizing click the **Stop digitizing** button. The digitized line should appear in the **Base lines** folder.
8. Click the **X** to close the digitizing menu.
9. Locate the digitized baseline in **1. Maritime lines / Base lines.**
10. Right click the baseline dataset and select **Rename**. Rename the data set to your country name and _baseline_. Click **OK**.
11. Right click the baseline dataset, and select **Set Schema > Base line**.
12. Right click the baseline dataset, and select **Properties > Geodetics**.
13. Click **Set geodetic meta data** and specify your settings.
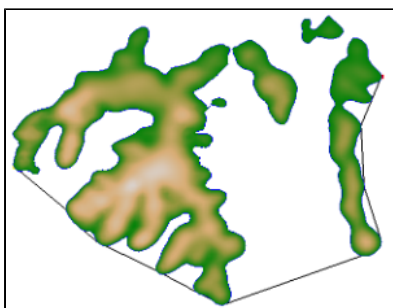
Digitizing a baseline

If you already have a baseline you want to import you can import it using the ASCII import menu, or Generic import if you have a Shapefile. The exercise below is for a baseline in ASCII format. If you created a dummy baseline in the previous exercise you can skip this exercise.

Exercise
Import baseline

1. Right click the **Base lines** folder under **1. Maritime Lines** and select **Import > ASCII Column**
2. Click the browse button and browse for your baseline file.
3. Make sure the **Coordinate format, File column, Cell type and Schema** is set correctly.
4. You may add a column for point name if the file contains point names.
5. Click **Execute** to import the file.
6. If the coordinates in your file was in Latitude and Longitude you need to convert it to the right projection.

Generate Distance Lines

We can now proceed to generate the distance lines we need for our project. The 200M line and the 350M line.

Exercise
Generate the 200M and 350M lines   1. In the folder **1. Maritime Lines / Base lines** right click your baseline and click **Generate 200M line.**   2. Use the settings you want and click **Execute.**   3. Generate the 350M line also.

Depending on your project area parts of your 200M and 350M lines might be on land or in none relevant areas. We can clean this up and remove this part of the line using the **Edit points and lines** command.

Exercise
Clean up the 200M and 350M lines   1. Select the 200M line in your project.   2. Find the **Edit points and lines** command in the **Toolbox.** You might have to add it by right clicking a folder in the **Toolbox** and selecting **New command**.   3. Use the command to delete the part of the line you don`t need.   4. Do the same with the 350M line if needed.

-

Distance Constraint

We are now going to create the last distance constraint, the 2500m isobath + 100M.

<table>
<tr><td colspan="1">Exercise</td></tr>
</table>

Generate the 2500m isobath

1. Right click the seabed grid in the **2. Seabed / Grids** folder and click **Generate 2500m Isobath.**
2. The command will generate a new dataset. Click **OK**.

<table>
<tr><td colspan="1">Exercise</td></tr>
</table>

Edit the generated 2500m isobath

1. Clear the display window by clicking the **X** button in the main toolbar.
2. In the folder **2. Seabed / 2500 meter Isobath** right click the 2500m isobath and select **Zoom to Data** and then display by selecting **Light Blue (2500m Isobath)**.
3. In the folder **2. Seabed / 2500 meter Isobath** right click the 2500m isobath and select **Edit points and lines..**.
4. Use the command do delete the parts of the line that we are not going to use as input to generate the 100M line.
5. Click **Execute** when you are finished.

<table>
<tr><td colspan="1">Exercise</td></tr>
</table>

Generate the 2500m isobath + 100M distance constraint

1. In the folder **2. Seabed / 2500 meter Isobath** right click the 2500m isobath and select **Generate 100M Line**.
2. Keep the default settings and click **Execute**.
3. The command will generate a new dataset. Click **OK.**

-

Generate or Import Bathymetric Profiles

Before we start analyzing the foot of slope we need bathymetric profiles. Initially you might want to create bathymetric profiles from the seabed grid, but you also need real bathymetric profiles. In the next two exercises we are going to create a few bathymetric profiles from the seabed grid and then import real bathymetric profiles from GEODAS.

Exercise

Generate Bathymetric Profiles

1. Display your seabed grid in the folder **2. Seabed / Grids**.
2. In the folder **2. Seabed / Grids** right click your seabed grid and click **Generate Bathymetric Profile**.
3. Find and area where you want a bathymetric profile and digitize it.
4. Continue to generate profiles until you think you have enough.
5. Rename your profiles so they have a useful name.

In the next exercise we will download a "Corrected Depth Bathymetry" from the U.S. National Oceanic and Atmospheric Administration's (NOAA) Geophysical Data System (GEODAS) and then import this profile using the ASCII Column importer in Geocap.

Exercise

Download Bathymetric Profiles from GEODAS

1. Go to this link: http://www.ngdc.noaa.gov/mgg/geodas/geodas.html
2. Click **Trackline Geophysical Data Viewer**.
3. Zoom in to the area you are working in.
4. Click on a line which you thing would be useful to analyze.
5. Click on **Get Marine Data for these Surveys** in the popup.
6. On the site that comes up, change the format to **XYZ** with **Z value** as **Bathymetric corrected depth**.
7. Follow the instructions on the site to download the data.

Exercise

Import Bathymetric Profiles from GEODAS

1. Right click the **Bathymetric Profiles** folder under **2. Seabed** and select **Import > ASCII Column**
2. Click the browse button and browse for your bathymetric profile.
3. Make sure the **Coordinate format, File column, Z, Cell type and Schema** is set correctly.
4. Click **Execute** to import the profile.
5. If the coordinates in the profile was in Latitude and Longitude you need to convert it to the right projection.

-

Foot of Slope Analysis and Hedberg points

You should now have several bathymetric profiles in your project, and we can proceede to analyze these to find FOS points.

Exercise
Analyze your profiles   1. Right click on of the bathymetric profiles under **2. Seabed / Bathymetric Profiles** and click **Analyze.**   2. Try out different settings in the analysis window in order to find a FOS point you think the Commission would accept.   3. Save the analysis and type in a useful name for your FOS point.   4. Analyze FOS points for the rest of your bathymetric profiles.

After you have analyzed all your profiles and created FOS points we will merge the FOS points in order to create one set of Hedberg points.

Exercise
Group FOS points   1. Create a new folder under **2. Seabed**, called **FOS points.**   2. Right click one of the FOS points you have created and click **Merge Foot of Slope points**.   3. Add the rest of your FOS points to the list and set the **FOS points** folder as result location.   4. Click **Execute**.   5. Type in a useful name for your FOS point group.

We are now going to generate the 60M line from the merged FOS points.

Exercise
Generate 60M Hedberg points   1. In the folder **2. Seabed / FOS points** right click your group of FOS points and click **Generate 60M line.**   2. Use the settings you want and click **Execute.**

-

Generate Gardiner points

We will now investigate the sediment data in the area and try to generate Gardiner points using the global sediment thickness grid and the FOS points we just created. Note that there might not be enough sediments to produce Gardiner points in the area you are working in.

Exercise

Calculate 1% of the distance to the closest FOS point

1. In the folder **3. Sediment Data / Grids** right click **Sediment Thickness** and click **Generate >1% dist to fos Area**
2. In the menu that pops up, click the browse icon and browse for the group of FOS points**.**
3. Make sure that **Store in project** is **checked.**
4. Click **Execute**.
5. Investigate the Gardiner points.

Create Outer Limit

Now that we have all the relevant data in our project we can proceed to copy all our formula lines and constraint lines into the Outer Limit folder and generate the outer limit.

Exercise

Copy your data to the Outer limit folder

1. **Copy** the **200M line**, **350M line**, **Hedberg points**, **Gardiner points**.
2. **Paste** them into the appropriate folder in the **4. Outer limit** folder.

Exercise

Generate the outer limit

1. Right click the folder **4. Outer limit** and click **Generate Outer Limit**.
2. Use the tool to generate your outer limit.

M. Workflows

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. In this section we will have a look at one of the workflows provided with the Atlantis project and then we will create a workflow our self. It is up to you if you want to create a workflow from the Atlantis project or from the new project you built.

Exercises

- Examine the workflows provided in the Atlantis project.
- Create a new workflow
- Build the workflow
- Execute the workflow
- Create a flight tour

Exercise

Examine the workflows provided in the Atlantis project.

1. Make sure the Atlantis project is open.
2. In Geocap, click **File > Open > Workflow**, and browse to the Atlantis project folder, then open the the sub-folder called **workflows**.
3. Select **Basic_Workflow.gwf** and click **Open**. This will open the workflow below the project.
4. Examine the contents of the workflow items by clicking on the triangles next to the root folders.
5. Click on the **Go to first executable element**

   ◀◀

   icon and then click on the **Execute next element**

   ▷

   icon. (This will initialize the basic display settings and nothing will appear in the display window yet)
6. Click on the **Execute next element**

   ▷

   icon again and notice everything in the **Base map** folder will be displayed.
7. Continue clicking on the **Execute next element**

   ▷

   icon until you reach the end of the workflow and observe how different elements of the Atlantis project are displayed.
8. Close the workflow by clicking **Close**

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

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. Go to **File > New > Workflow**.
2. Browse for **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. Select a dataset in your project that you want to display.
2. In the **Toolbox** right click the display command you want to use in your workflow and select

   Copy
3. Click the

   button in the workflow. The command will now be opened in edit mode.
4. Check the settings and click **OK**
5. Continue to add elements to your workflow by selecting a dataset and copying a command into the workflow.

Exercise

Execute the workflow

1. Click the

   icon.
2. Click the

   icon to execute next element or the

   icon to execute all elements in 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
    ▶
    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 .**

N. DataLink for ArcGIS

Introduction

The Datalink for ArcGIS plugin provides interoperability between Geocap and the ArcGIS suite of products from Esri. The DataLink allows you to open existing ArcMap Documents (.mxd) in Geocap and lets you display the data directly in the graphics window of Geocap.

It is also possible to open Esri File Geodatabases inside a Geocap project allowing you to copy data between the Geocap project and the geodatabase.

> The Datalink for ArcGIS is only available in the 32bit version of Geocap. You will also need an existing ArcGIS 10.x desktop license to be available on your machine.

Exercises

- Load an ArcGIS map document the Geocap project
- Visualize map data in the Geocap project
- Create a new file geodatabase in the project
- Copy data from Geocap to the geodatabase

ArcMap documents in Geocap

Exercise
Load an ArcGIS map document the Geocap project   1. Open the Atlantis project   2. In the project toolbar click the **Add Data Source** toolbutton   3. Click **Add Data Source**   4. On the connection dialog type inn a name that describes the map file. This is optional.   5. In the **Datasource** field choose **ArcMap Document**   6. Click the Browse button to select the **Atlantis.mxd** map file   7. Click **Connect**   8. A map node will be created in the project. This is a link to the existing mxd file. No data is copied.

Exercise
Visualize map data in the Geocap project   1. Select a layer that is not a group layer.   2. Right click on the layer node   3. If the layer is a feature layer use **Display Features**   4. If the layer is a raster layer use **LOD Grid Display.** Open the command in edit mode an check **Flip Z values.**

-

File Geodatabases

Exercise

Create a new file geodatabase in the project

1. Create a new project or open an existing one
2. In the project toolbar click the **Add Data Source** toolbutton
3. Click **Create ArcGIS File Geodatabase**
4. The database creation dialog pops up. There are two alternatives for the location of the new geodatabase:
   a. If you create the database **inside the project** directory the geodatabase and relative path names will continue to work even if you change the location of the Geocap project.
   b. If you create the database **outside the project** it can more easily be shared by several projects or other applications, but the database can no longer be accessed by the project if the project is moved..
5. Choose the database location and name and click **OK**
6. The geodatabase is create on disk and a new database object is created in the project.

Exercise

Copy data from Geocap to the geodatabase

1. Right-click the dataset that you wish to move to the file geodatabase and select **Copy**
2. Right click the geodatabase object and click **Paste.**
3. A dialog will pop up showing you various options for tranforming the VTK data to a corresponding ESRI data format. Which conversion dialog is shown depends on whether the VTK data format is a raster dataset or polydata. If the source dataset is VTK Polydata the dataset will be split into two feature classes, one for points and for the the cell geometry. You may choose one or both. In most cases you will also choose to flip the Z values, which is default.
4. Geocap will try to determine the ESRI spatial reference that corresponds to the one in the source dataset. If you think the chosen suggested spatial reference is wrong you may browse and select one of the standard ESRI spatial references.
5. Click **OK** to perform the conversion and close the dialog
6. The result data will be found in the project under the geodatabase connection.

O. Image Georeferencing

Introduction

This tutorial will guide you through the process of georeferencing a map. To georeference a map or image you need 3 tie points. These points should cover as much of the map as possible. The map in this exercise has a graticule showing latitude and longitude which works great as tie points.

Exercises

- Import an image into your project
- Georeference an image

Exercise

Import an image into your project

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

Exercise

Georeference an image

1. Right click the image you want to geo reference and select **Geo Reference**
2. A new window will show the image and the georeference menu will pop up in the lower left corner.
3. In the first menu specify the original spatial reference of the map. In this case **WGS84** and **Mercator**
4. Click **next**
5. In the **Tie point Coordinate Format** drop down select **12°34.56' N/S**
6. Type in the values **Lat: 32 00 S** and **Lon: 13 00 W**
7. Zoom into this crossing point on the map
8. Click on **Image pick** and click on the crossing graticules in the map.
9. Select **Tie point 2** in the drop down list
10. Type in the values **Lat: 32 00 S** and **Lon: 1 00 W**
11. Zoom into this crossing point on the map
12. Click on **Image pick** and click on the crossing graticules in the map
13. Select **Tie point 3** in the drop down list
14. Type in the values **Lat: 20 00 S** and **Lon: 1 00 W**
15. Zoom into this crossing point on the map
16. Click on **Image pick** and click on the crossing graticules in the map
17. Once you have georeferenced three tie points the table will provide you with an pixel error. The value should not exceed more than 1 pixel on each point.
18. Click **Finish** to complete the georeferencing
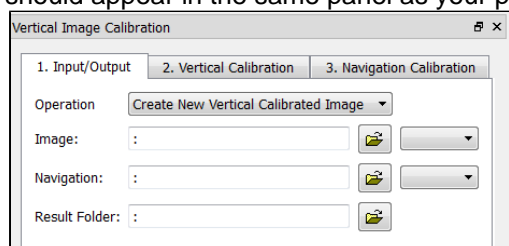
P. Vertical Image Calibration

Introduction

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

- Image
- Navigation

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

Open VCI menu

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

This menu consists of 3 steps (tabs) that you need to complete in order to complete the vertical calibration.

Input/Output

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

Vertical Calibration

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

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

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

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

Navigation Calibration

1. Click the **3. Navigation Calibration** tab to start the navigation calibration
2. Locate the first shot point in your image and find the corresponding shot point in the **Tie Points** list in the menu.
3. Select the **Tie Point** in the list, then click the **Pick** button and click on the shot point in the image.
4. Locate the last shot point in your image and find the corresponding shot point in the **Tie Points** list in the menu.
5. Select the **Tie Point** in the list, then click the **Pick** button and click on the shot point in the image. You should now see a line of points along the shot point line.
6. Continue this process with the points between the start and end points if needed.

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

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

Display

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

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

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

Q. Other Topics

In this section:

Introduction

Shelf and Geocap in general contains a lot more functionality than we have been through in these tutorials. This section highlights some of these topics.

Maritime Zones plugin

Maritime Zones plugin extends the Shelf package with additional capabilities for calculating maritime zones. It works directly with ArcGIS feature classes and can calculate any maritime zone from both normal and straight baselines. The calculation also extracts critical points for each zone and store them in a feature class. More information here: https://geocap.atlassian.net/wiki/display/ug/Maritime+Zones+plugin

Equidistance lines and weighted midlines

It is possible to create equidistance lines and weighted midlines between nations. The command is available on the base line schema.

Composed grids

It is possible to combine grids of different resolution into a composed grid and then produce bathymetric profiles from the composed grid.
More information here: https://geocap.atlassian.net/wiki/display/art/Composed+grids

Area Generator

The **Area Generator (AG)** is an automatic project generator that is used to chop out portions of the world from a database of global or regional data. Select an area by latitude and longitude, select data to include, select datum and coordinate system and click one button to automatically produce a Geocap project. More information here: www.geocap.no/doc -> **Geocap Extensions**

Graphic Export

The graphics window in Geocap can be exported to a range of different formats. Select **File > Save graphics as.**
It is also possible to copy the graphics window to clip board using **Ctrl + C,** and then pasting into powerpoint etc.

Scripting

It is possible to create your own commands and scripts in Geocap.
More information here: https://geocap.atlassian.net/wiki/display/ug/Scripting+commands+with+a+menu

The Function command

The Function command enables you to manipulate datasets in many different ways. You can find this command in the **Operations** folder.

Note sheets