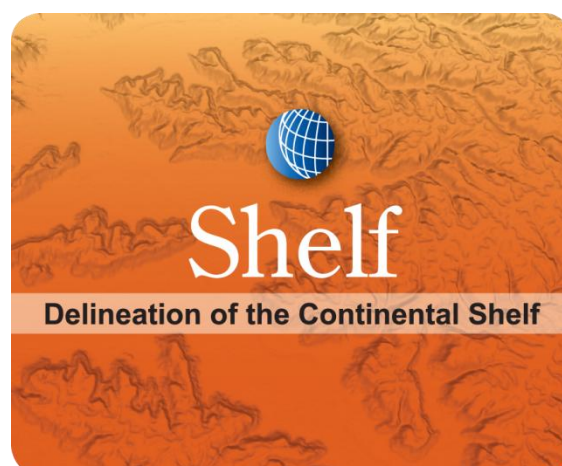


Shelf Tutorials

for Geocap 6.3

Document generated: 17.10.2012

support@geocap.no



Geocap

Shelf (UNCLOS Art. 76) Tutorial

This tutorial collection is specific to UNCLOS article 76 functionality in Geocap. The Atlantis project will be used as an example project, but other projects may be used instead when doing the exercises.

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

- Learn the basic principles of Geocap
- Create a new project
- Import data
- Create data and analysis
- Set Foot Of Slope points
- Interpret and depth convert
- Generate constraint lines and delineation lines
- Generate a final outer limit line
- Check out workflows in Geocap

	Content	Page
A.	Getting Started with Geocap Shelf	2
B.	Geocap Interface	4
C.	Create a New Shelf (UNCLOS) Project	12
D.	Import and Export	14
E.	Baseline Generation (Simplified)	17
F.	Bathymetric Grids And Generation Of Bathymetric Profiles	19
G.	Generation Of Distance Lines And Mid-Lines	22
H.	The Analysis Panel	26
I.	Foot Of Slope (FOS) Points	27
J.	Real Bathymetric Profiles	30
K.	The 2500m Isobath	31
L.	Seismic Interpretation	33
M.	Depth Conversion	36
N.	Sediment Thickness	40
O.	Final Outer Limits	42
P.	Uncertainty	45
Q.	Gridding	47
R.	Plotting	49
S.	Workflows	50
T.	Image Georeferencing	53
U.	Vertical Image Calibration	56
Z.	Additional Shelf Exercises	60

A. Getting Started with Geocap Shelf

Introduction

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

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

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

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

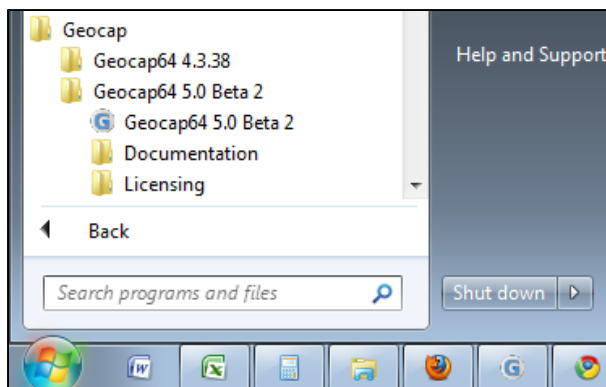
Your Geocap installation will consist of the basic Geocap platform and one or several of these plugins.

Exercises:

- Explore the different settings in the Option dialog

Start Geocap

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



Starting Geocap from the startmenu

User Documentation

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

The user documentations consists of:

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

The Options dialog

The **Options** dialog lets you define what Geocap should do on startup. This means that you can predefine a background color, working

directory, data window and automatic loading of plugins and scripts etc.

Exercise

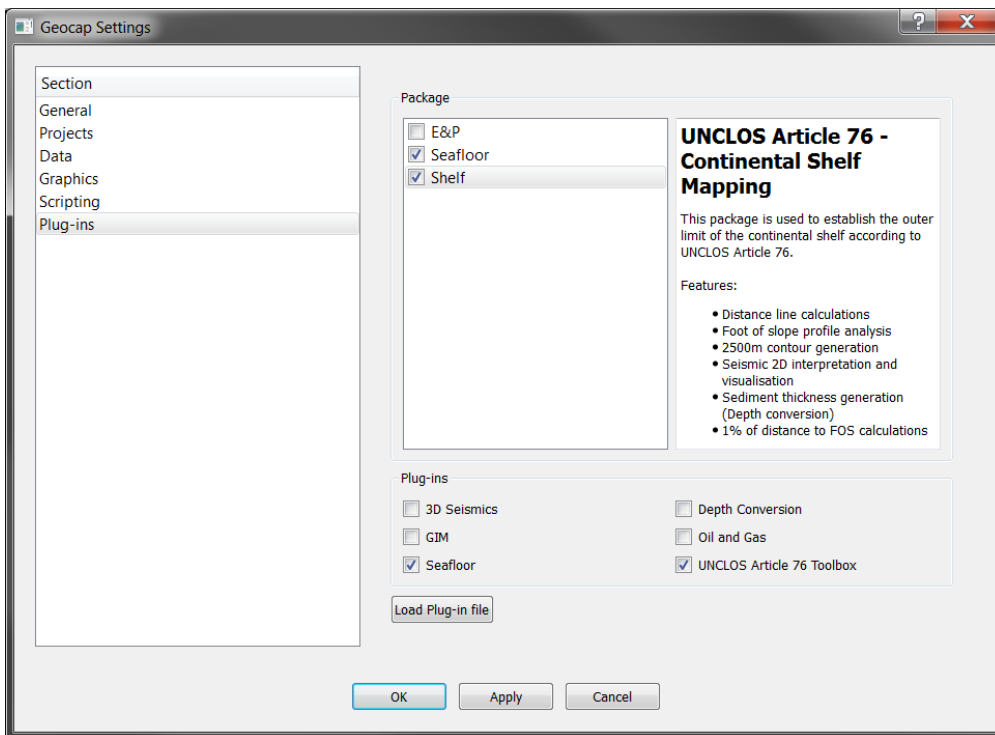
Explore the different settings in the Option dialog

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

UNCLOS functionality

We need to make sure that the UNCLOS functionality is available by checking that the Shelf Plug-in is loaded. Geocap will remember this setting, so you only need to do it the first time you start up. To enable the UNCLOS functionality:

1. Open the Geocap Settings dialog under pulldown **Tools > Options**.
2. In the plug-in section, make sure that the following plugins are selected:
 - UNCLOS Article 76 Toolbox (This is the main Unclos/Shelf functionality)
 - Seafloor
3. Click **OK**, and restart Geocap if you needed to add new plug-ins.




Geocap Plugins

B. Geocap Interface (Shelf)

Introduction

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

Exercises

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

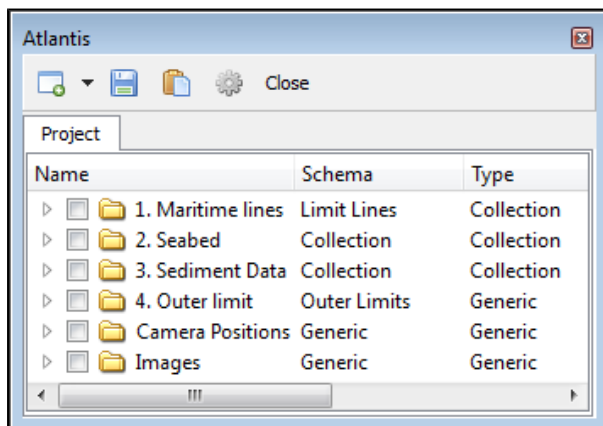
Geocap project

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

Exercise

Open the Atlantis project

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



An open project



Tip

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

Display data

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

Exercise

Explore the project folder structure

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



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


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


Exercise

Display data

Display the various datasets in the project

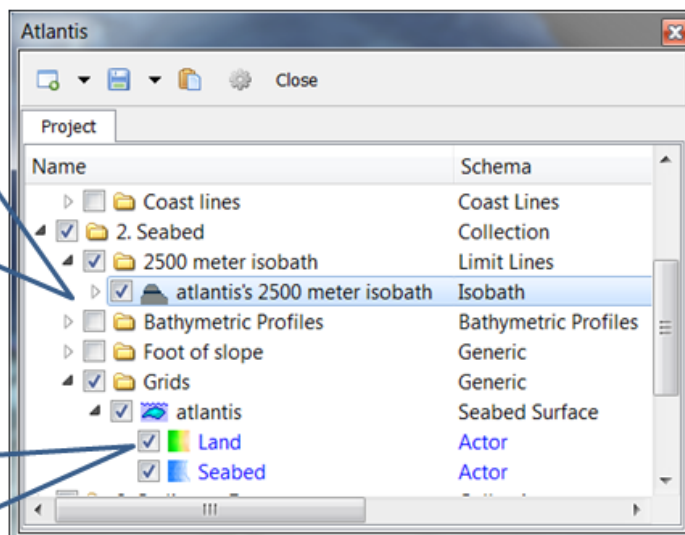
1. Locate a grid dataset (structured points or seabed) e.g. **2. Seabed/Grids/atlantis**
2. Right click the dataset and try a display command e.g.  **Display, Map Data, Map sea** or **Map land**.
3. Right click the dataset again and select **Zoom to Data**. This will make the display window center the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

 Note how the displayed items are shown and hidden from the display window.

 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.


A project item (or element) that has a visualization is checked. All folders above are also checked. You may toggle all underlying visualization on/off by the checkbox.
A dataset that is not visualized will do the "default command" when checked for the first time. See Edit > Schemas for default command

Actors (i.e. visualizations) appears in **blue text** underneath a dataset. It is possible to toggle them on/off or right-click to change appearance or delete. Note that actors are not datasets, they are visualizations.



Visualizations are called actors

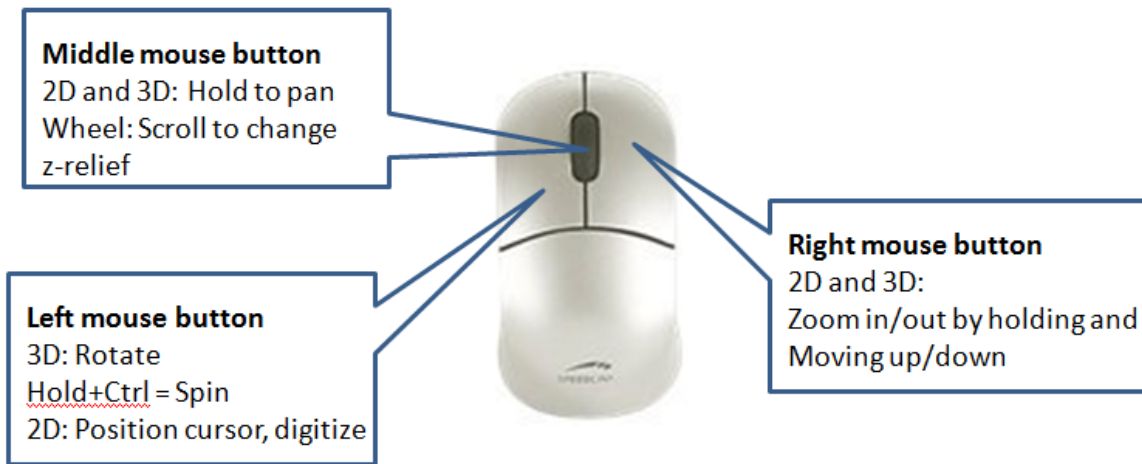
Navigating the display window

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


You may use the computer mouse to move around in the display window. You do this by pressing one of the mouse buttons while the cursor

is in the display window, and moving the mouse while keeping the button pressed.

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




The mouse buttons

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

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

Exercise

Navigate using the mouse and navigator panel

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



The navigator panel

Exercise Continued

- Play around with the buttons in the navigator.

Important Toolbar Buttons



Basic Concepts in Geocap

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

Schemas

Geocap uses schemas to classify a dataset. The Shelf Module contains several schemas. Some of the schemas used in the Shelf Module are **coast line**, **base line**, **limit line**, **seabed**, **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

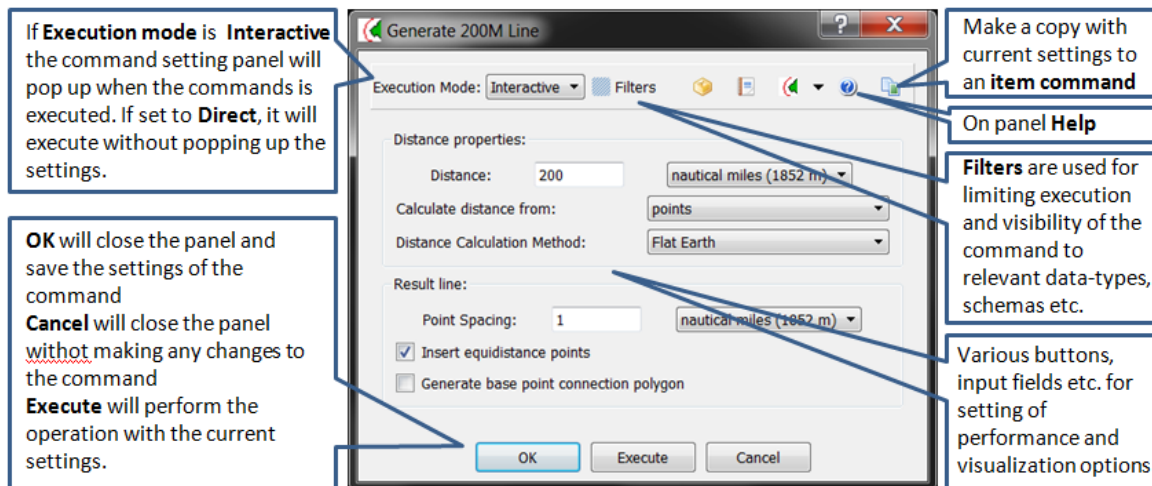
Commands are operations which can be performed on a dataset. Commands can for example be used to display a dataset in the display window, or to generate new datasets. You can even create your own scripted commands to cater to your specific needs. You execute a command by right clicking the dataset or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

Commands can be stored in three categories:

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

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

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

Item Commands

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

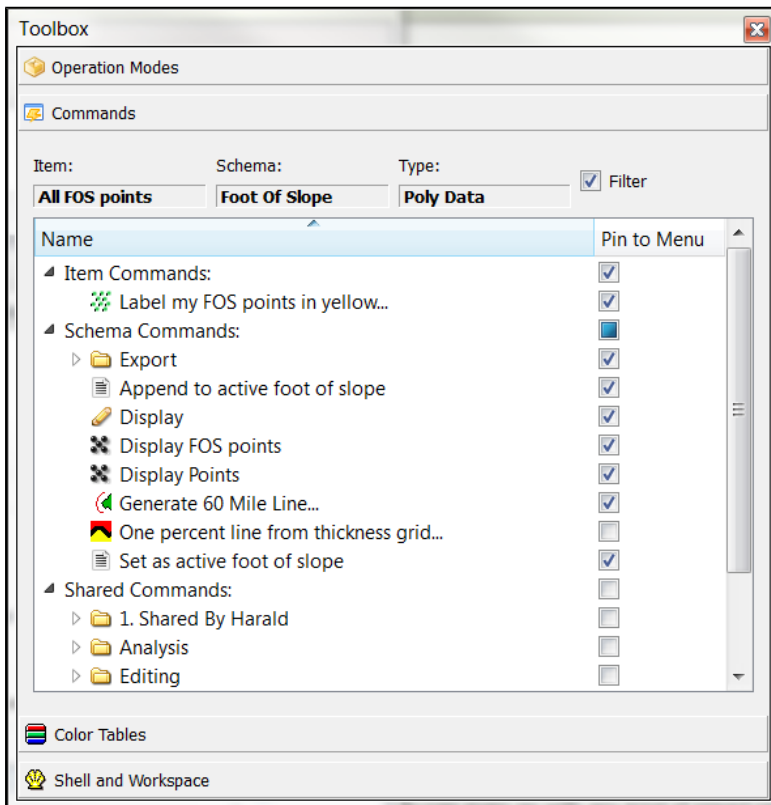
Exercise

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**.
3. Set **Line Width** to **4**.
4. Check **User defined Color** and set the color to white.
5. Click the  icon in the upper right corner of the menu and observe that the command appears in the **Toolbox** 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 white** and click **OK**.
9. Select the different lines in **1. Maritime lines / 60M lines** and observe that the new **Display in white** command is only available for the one dataset.

Schema Commands

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



The Geocap Toolbox. Note that the **Filter** is checked, meaning that commands not specifically relevant are invisible. Untick the filter to show all 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.

Exercise

Change the default display for seabed surfaces

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

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

Shared Commands

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

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

Exercise

Create a new command for custom display of limit lines.

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



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.


Default command

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 in a dataset's **Properties** dialog, and you can choose between schema commands and item commands.


Exercise

Change the default command for a dataset

1. Go to the **1. Maritime lines / 60M lines** folder and select the dataset that you created the item command **Display in white** for.
2. Right click the dataset and select **Properties**.
3. Click on the **Commands** tab.
4. For **Command Type** select **Item Command**
5. Select the command **Display in white** in the list.
6. Click on the  icon.
7. Click **Close**.
8. Now tick the box next to the dataset and observe that the default command has been changed.

Exercise

Commands: In panel Help

1. Open the command called "Generate 200M Line" from the toolbox.
2. Click the icon with the  question mark to open the in-panel help.
3. Read the information that pops up.



The 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 (visualized in the toolbox) under the name **sticky_surface**. If this dataset is removed from workspace, there is no sticky surface anymore.

Exercise

Display onto the Sticky Surface

1. Right-click on a Seabed surface dataset and select  **Set as sticky surface**
2. Select a line, e.g. **Atlantis > 1. Maritime lines > 200M lines > atlantis + 200M** in your project.
3. In the Toolbox under Commands > Schema commands right-click and Edit on the  **Display** command.
4. Check the **Glue to Sticky Surface** and press **Execute** to do a line display.
5. Uncheck the **Glue to Sticky Surface** and press **Execute** again and observe the difference.



Warning

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

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 (z coordinate) from graphics	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

Exercise

Keyboard shortcuts

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

C. Create a New Shelf (UNCLOS) Project

Introduction

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



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

Exercises

- Generate a new and empty UNCLOS project
- Explore the project settings
- Copy data into your new project

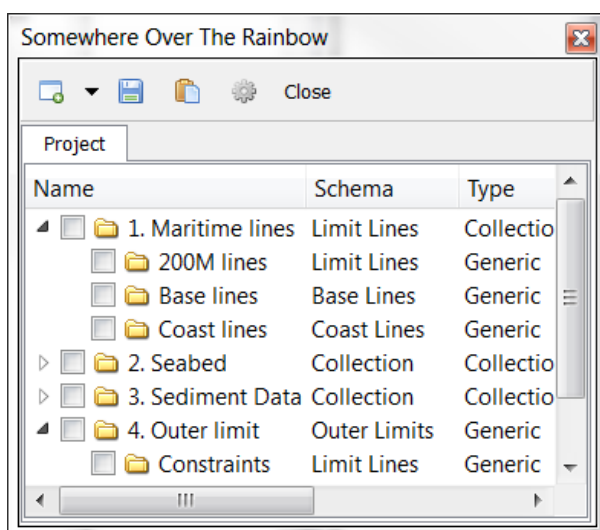
The UNCLOS Project template

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
- Type in the name of your project in the Name field. The name may consist of letters, numbers and spaces, but special characters like [, æ, ø, å, &, /, % ... should be avoided.
- 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.

Exercise

Explore the project settings

- Click the  icon on the project toolbar to look at the project settings.
- Pay particular attention to **Geodetics** section. Here you may activate geodetic settings for the project. If the geodetic settings for the project are checked, all data will be converted to these settings on import.
- Set Geocap to automatically zoom to your project area when the project is opened by first zooming to your area. Then in the **Data** section click **Use current** and tick **Set data window when loading project**
- Click **Apply** and **OK**

Exercise

Copy data into your new project

1. Open your Atlantis project (or another project)
2. Right-click and copy a Seabed surface from the Atlantis project.
3. In your new project, go to the **2. Seabed / Grids** folder, right-click and do **Paste**
4. Do the same for a coastline.

D. Import and Export

Introduction

Geocap can be used to generate accurate distance lines computed by algorithms following the earth curvature. The same operation is used when calculating distances from the base line (200M and 350M), the depth constraint (2500m depth + 100M) and the Foot of slope + 60M.

Before we can start generating a distance line, we need something to generate the distance line from. This tutorial will start by importing a coast line within your area of interest. 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.

Exercises

- Download coastline files from the Internet.
- Import and display coast line
- Convert coast line to the right projection and display
- Import Base line ASCII lat-lon format
- Convert Base line to Mercator and display
- Open the base line in table view
- Export the baseline


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/shorelines/gshhs.html>.
2. Click **Download GSHHS Data**.
3. Download and save the file **gshhs_2.1.0.zip**.
4. Uncompress the files to a folder on your hard disk.

Exercise

Import and display coast line

1. Locate the folder called **Coast Lines** in your Geocap project under **1. Maritime Lines**.
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 files you downloaded in the previous exercise.
5. Select the file called **gshhs_h.b** (this is a high resolution, but not the full resolution), 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.
Max Y = Northern boundary of import area **Min X** = Western boundary of import area
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. If it doesn't it will ask you to specify this. In that case select datum: **World Geodetic System 1984** and coordinate system: **Geodetic**.
11. Geocap will confirm that the file has been read. Click **OK**.
12. After the file has been read, click the **Cancel** button in the Generic reader dialog.
13. Observe that the imported dataset is stored in the **Coast lines** folder.
14. Right click the dataset and select **Zoom to Data**
15. Right click the dataset and select **Display**

The data is imported in Geodetic, with latitudes as Y coordinates and longitudes as X coordinates. In order to view the data in a proper projection, we convert the data to Mercator coordinates.

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 Mercator...** command in the **Operations** folder.
3. Change the result combo box to **Replace input**.
4. Click **Execute**.
5. Agree to replace the existing dataset by clicking **Yes**.


The data is now converted. Let us re-display in new coordinate system:

1. Right click the dataset and select **Zoom to Data**
2. Right click the dataset and select **Display**

If you have a data set with the base line of your country you can import it and use that as a base for the distance lines. If you do not have such a base line, skip this exercise and generate a false base line in the next exercise instead.

Exercise

Import Base line ASCII lat-lon format

1. Locate the folder called **Base Lines** in your Geocap project under **1. Maritime Lines**
2. Right click the folder and select **Import > ASCII Column...** and the ASCII Column import dialog will appear. (You can make dialog bigger by clicking in one of the corners and drag.)
3. Click the browse  button and select your base line file. There may be a suitable file somewhere under **Atlantis/Data/...**
4. Adjust the number of header lines in your file with the **Adjust header** spin box. The header lines should be displayed in the top part of the file preview, while the bottom part of the file preview should hold the coordinates.
5. If the columns are delimited by space or tab characters, you should keep the column separator at white space. If the file uses a special character, change the combo box to **MySeparator**, and change the value by typing in the separator directly in the combo box.
6. Select a coordinate format:
 - **Deg** = decimal degrees
 - **Deg min** = degrees and decimal minutes
 - **Deg min sec** = degrees, minutes and decimal seconds
 - **Hemi** = hemisphere (N/S for north or south, E/W for east or west)
7. Set the correct file columns for each data column. You see the first lines in the table. On top of each table is a spin box with the file column number. Adjust the file column by clicking the spin boxes so that the file columns are interpreted correctly.
8. All other parameters should be OK with the default settings, click **Execute**.
9. The file has now been imported. Click the **Cancel** button in order to close the ASCII Column import dialog.

Exercise Continued

Convert Base line to Mercator and display

The data is imported in Geodetic, with latitudes as Y coordinates and longitudes as X coordinates. In order to view the data with the coast line, we convert the data to Mercator coordinates.

1. Click the dataset and go to the **Shared commands** section in the **Toolbox**.
2. Double click the **Convert to Mercator...** command in the **Operations** folder.
3. Change the result combo box to **Replace input**.
4. Click **Execute**.
5. Agree to replace the existing dataset by clicking **Yes**.

The data is now converted. Let us display the data:

1. Right click the dataset and select **Zoom to Data**
2. Right click the dataset and select **Display**

 Note that the conversion is not needed if the Project Projection is activated under Project settings.

Exercise

Open the base line in table view

1. Select some points in the table view and observe that points selected in the table view are highlighted (in yellow) in the display.
2. Observe that you can copy entries from the table view and export to a text editor
3. Click Edit mode. Try to add an extra column and/or change names on the base line points.

Export

Exercise

Export the baseline

Export the baseline in Ascii column format as Degrees Minutes Second.

Tip: Make sure you are exporting a baseline in with coordinate system Geodetic.


E. Baseline Generation (Simplified)

Introduction

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 skip this exercise.

Exercises

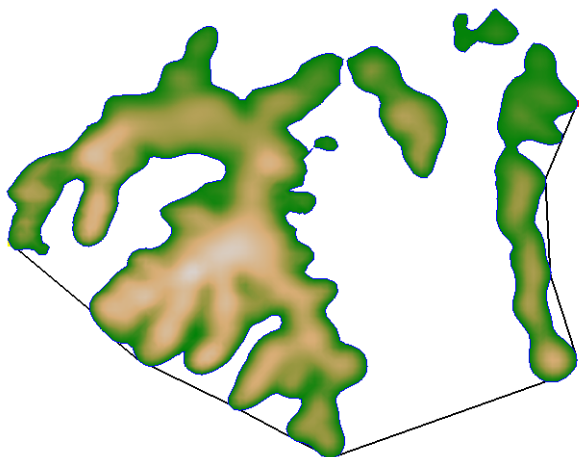
- Generate dummy baseline
- Use the  "Red snapper" to generate an alternative baseline

Exercise


Generate dummy baseline


In this exercise we will use the digitizer to snap values from the coastline, in order to make a **baseline**.



1. Locate the coast line called GSHHS, which should have been imported in **1. Maritime Lines / Coast lines**.
2. Right click the dataset and select **Make active**.
3. Open the Digitizer by selecting **Tools > Quick digitizing** from the main menu.
4. Check the option **Snap exclusively to active data points**. This will force the digitized points to stick to the points of the coast line.
5. Check the option **Save also in folder**, and click the **Browse** button.
6. The project view dialog will appear. Select the **Base lines** folder under **1. Maritime Lines**, and click **OK**
7. Click **start digitizing**
8. Digitize the points on the coast line by clicking on the coast line with the left mouse button. The picks will automatically stick 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.
9. When you are finished digitizing click the **stop digitizing** button. The digitized line should appear in the **Base lines** folder. If you are not happy with the digitized line and want to re-do the digitizing then you have to make the coast line active again before starting the digitizing.
10. Locate the digitized baseline in **1. Maritime lines / Base lines**
11. Right click the dataset and select **Rename**. Rename the data set to your country name and No_dummy_baseline.
12. Right click the dataset (not the folder), and select **Set Schema > Base line**




Digitizing a baseline

An alternative is to use the command object  **Red snapper** on the coast line and simply snap the outer points needed for a distance calculation.

Exercise**Use the  "Red snapper" to generate an alternative baseline**

1. Click the coast line called GSHHS.
2. Double click the  **Red Snapper...** command which can be found in the **Editing** folder under **Shared commands** in the **Toolbox**. The  **Red snapper** dialog will appear.
3. Click the **Display selected data with green points** button. This will display the points that the coastline is generated from.
4. Check the **Save in project** box and click the **Browse** button.
5. Select the **Base lines** folder and click **OK**.
6. Click **Start snapping**.
7. Start clicking on the outer points on the coastline with your left mouse button.
8. When you are finished digitizing click the **End snapping** button. The red snapper line should appear in the **Base lines** folder.



Observe that in the  Red Snapper command (and also in the Tools > Quick Digitizer) you may enhance functionality by enabling more keys. E.g. **m** can be used to snap all points up to next picked point. This is probably not needed here.

F. Bathymetric Grids And Generation Of Bathymetric Profiles

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 bathymetry may be imported as bathymetric profiles.
- Multibeam bathymetry 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.
- Grids may be imported directly into Geocap. Geocap supports import of several different file formats; the most common and useful kind of bathymetric data is the ETOPO1 grid. The ETOPO1 Grid is provided by NGDC (National Geophysical Data Center). This grid 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 sea floor, and in particular the approximate position of the Foot of Slope or base of slope region.


Exercises

- Download the ETOPO1 grid from the Internet
- Import the grid into a folder in your project
- Convert the grid to Mercator coordinate system
- Crop out smaller grids from your Seabed surface
- Display a detailed contour pattern on the seabed surface
- Generate a Bathymetric Profile from a grid

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: **Create Custom Grids** - in the left column on the page, below the globe. A new web page with the **GEODAS Grid Translator – Design-a – Grid** should appear.

 **GEODAS Grid Translator - Design-a-Grid** [help](#)

Windows
 Macintosh
 UNIX-LE (Linux-X86, etc.)
 UNIX-BE (Sun, etc.) [?](#)

 Your Grid Id: (Create 8-char Identifier for Grid)

[?](#) Grid Database:

[?](#) Grid Area in degrees and minutes

Upper Latitude:

Left Longitude: Right Longitude:

Lower Latitude:

[?](#) Grid Cell Size:

Number of Latitude Cells: Number of Longitude Cells:

[?](#) Grid Format:

Output Grid Format: <input type="radio"/> Binary Raster Format <input type="radio"/> ASCII Raster Format <input type="radio"/> XYZ (lon,lat,depth)	Output Grid Header: <input type="radio"/> GRD98 Header <input type="radio"/> ASCII (Arc) Header <input type="radio"/> No Header
--	---

[?](#) Greenland/Antarctica Surface Option:

Ice Sheet Surface
 Bedrock Surface

The GEODAS Grid Translator


Exercise Continued

1. Enter a grid id (for example the name of your country).
2. Select **ETOPO1 1-minute Global Relief** as your Grid Database.
3. Enter the latitudes and longitudes of your area.
4. Keep Grid cell size at **1 minute**.
5. Select Grid format: **Binary Raster Format**.
6. Select Grid header: **GRD98 Header**.
7. Click the **submit** button on the top of the page.
8. A new page will appear, Click **Compress and Retrieve your grid**.
9. A new page will appear, Click **Retrieve**. Store the data somewhere on your computer.
10. Once the data has been downloaded, decompress the files and store them on your computer.

The following exercise can be used on most binary grids which Geocap supports. You normally do not have to specify the grid file format. Geocap will automatically recognize the file format.

Exercise

Import the grid into a folder in your project

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 files you downloaded in the previous exercise. The data file is the file which ends with **.g98**, and is located in a sub folder. Select this file and click **Open**.
5. If **Specify import area** is checked, uncheck this option. This will import the entire grid.
6. Click on the **Reader Options** tab and under polydata and check the box next to **Invert** (The Grid we will import 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).
7. Click **Execute**.
8. You will now be prompted with a question of the coordinate system and datum. Select: **World Geodetic system 1984**, **Geodetic** and click **OK**.
9. Geocap will report if the grid has been imported correctly; click **OK** and close the import dialog by clicking **Cancel**.
10. Observe that the new data set has appeared in the **Grids** folder.

Exercise

Convert the grid to Mercator coordinate system


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 a Mercator coordinate system, we need to convert this grid to match the same coordinate system as the other data in the project (Mercator).

1. Click the dataset and go to the **Shared commands** section in the **Toolbox**.
2. Double click the **Convert to Mercator...** command in the **Operations** folder. (or another relevant datum / coordinate system)
3. Change the result combo box to **Replace input**.
4. Click **Execute**.
5. Agree to replace the existing dataset by clicking **Yes**.
6. Geocap will report if the grid has been converted correctly, click **OK** and close the conversion dialog by clicking **Cancel**.
7. Check that the schema of the dataset is **seabed surface**. If not then right click the dataset, choose **Set Schema** and choose **seabed surface**.

The data is now converted and ready to be displayed:

1. Right click the dataset and select **Zoom to Data**
2. Right click the dataset and select **Map Sea**
3. Right click the dataset and select **Map Land**



When you have made changes to the project it is always a very good idea to store them. Click the save button  in the project tool bar in order to save your project.

Exercise


Crop out smaller grids from your Seabed surface

- Use the command **Crop** to chop out 2-4 smaller grids from your main seabed surface.

Use keyboard shortcut **z** in 2d mode to set a window frame

Exercise

Display a detailed contour pattern on the seabed surface

Use the command  **General Display** to display contours. Try to display line contours at depths relevant to base of slope and see if you can observe the area of the deep ocean floor and areas of base of slope from the contour pattern.

Explore the different display options of this command, and observe that this command is different from most other commands. Observe that this command can produce several **actors** (visualizations).

Exercise

Generate a Bathymetric Profile from a grid

1. Locate the dataset you imported in the previous exercise, which should be located in **2. Seabed / Grids**. The data set should have the schema Seabed Surface. If it does not have this schema, right click it and select **Set Schema > Seabed Surface**.
2. Display the seabed by right clicking it and selecting **Map Sea**.
3. Right click the dataset and select the command **Display contours...**
4. A menu will appear. We will just use the default settings. Click **Execute**, and **Cancel**.
5. Right click the dataset and select the command **Generate Bathymetric Profile**. A menu will appear.
6. 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.
7. 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.
8. After you have digitized both the start and stop point, you will be prompted with a dialog asking you to provide a name for the profile. Keep the default name and click **OK**.
9. Geocap will notify you where the resulting Bathymetric profile has been stored in the project. Click **OK**.
10. Locate the new profile in **2. Seabed / Bathymetric Profiles**; this is the default position of bathymetric profiles. In order to structure our project we want to move the profile into a sub folder and rename the profile.
11. Right click the folder **2. Seabed / Bathymetric Profiles** and select **New > Folder**.
12. Change the folder name to **Etopo1** and click **OK**.
13. Right click the profile and select **Cut**.
14. Right click the new folder and select **Paste**. The profile is now moved into the new folder.
15. Right click the data set and select **Rename**. Give the profile a suitable name, for example the name of the area you are working in.


G. Generation Of Distance Lines And Mid-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.

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

Exercises

- Generate the 200M line
- Read about distance lines in the  in panel help
- Generate new 200M line with 1M point spacing
- Generate the 200M baseline critical points
- Generate 350M line
- Try the Distance Calculation tool
- Clean up the distance lines
- Generate mid-lines
- Mid-lines using weighted base points


The 200M constraint line is measured from the baseline.

Exercise

Generate the 200M line


1. Display the baseline you imported or generated in a previous exercise.
2. Make sure the baseline has the **Base Lines** schema in the project window (If it uses a different schema, right click the dataset and select **Set Schema > Base Line**).
3. Right-click the baseline and select the command **generate 200M Line**.
4. The new line should now be available under **1. Maritime Lines/200M Lines**. Click **OK**.
5. Display the 200M line.



When you have made changes to the project it is always a very good idea to store them. Click the save button  in the project tool bar in order to save your project.

Exercise

Read about distance lines in the in panel help

1. Open the command called "Generate 200M Line" from the toolbox.
2. Click the icon with the  question mark to open the in-panel help.
3. Read the information that pops up. If you want to use the method of **tracés parallèles** you may consider resampling (geodetically correct) your baselines and use distance from points.

Exercise

Generate new 200M line with 1M point spacing

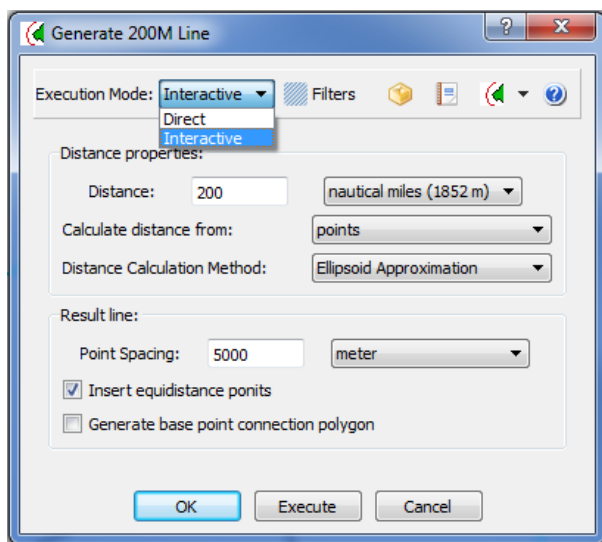
1. Right click the baseline dataset and select **Edit Schema Commands**. The Base line schema command list will appear.
2. Double click the **Generate 200M Line** command object in the command list and the command editor will appear.
3. Type in the new point spacing 1 and change the unit to Nautical miles.
4. Click **OK** in the command editor and the schema command list.

You have now set the **Generate 200M Line** to use 1M point spacing on the distance line. Repeat the first exercise in this tutorial in order to generate the line. Note that if you reduce the point spacing the algorithm needs more time to do the calculation.

Exercise

Generate the 200M baseline critical points

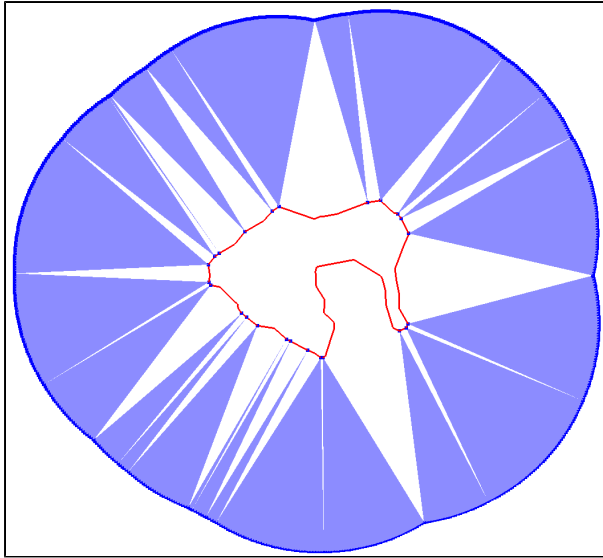
1. Click the baseline dataset and go to the **schema commands** section in the **Toolbox**.
2. Right click the **Generate 200M Line > Edit**, and the command editor will appear.
3. If necessary, change the Execution Mode to **Interactive** and click **OK**. You have now set the **Generate 200M Line** command to be interactive and the panel will pop up every time you choose to use this command.



Setting the Schema Command to **Interactive**

Exercise Continued

1. Right click the baseline dataset again and choose **Generate 200M Line...** (the three dots indicate this Schema Command is set to interactive mode).
2. Check the box next to **Generate base point connection polygon**.
3. Click **Execute** and Display the new dataset found in the 200M lines folder. Also display the points by right clicking and choosing **Display Points**.
4. Re-name the dataset to something more meaningful like, **200M critical points**.



200M line with critical points on baseline in blue

Exercise Continued

Notice not all points on the baseline are included as **critical points** used to generate the **200M** line. This is helpful in determining and visualizing which points on the baseline contribute to the limit lines, whether the user generates **200M**, **24M**, **12M** or any other distance calculation with this option.

1. Additionally, open the table view for this dataset and notice the column for **Base Line Index**. This shows which point on the baseline contributes to each point on the 200M line.
2. Toggle the display between polygon mode and wire mode with the **W** and **S** keys.

Exercise

Generate 350M line

The same procedure as above.

Exercise

Try the Distance Calculation tool

Use the **Distance Calculations** tool to verify that the distance between the baseline and the generated 200M line is correct.

1. Open the tool from **Tools > Distance Calculations**.
2. Select an appropriate datum and coordinate system.
3. For distance calculation method, select **ellipse approximation** or **Vincenty calculation**. **Flat earth** calculates in the metric values of the projections.
4. Make sure that the graphic window is in 2D.
5. Click in the graphics window to measure distance to previous position. The distance is displayed in the lower right corner of the panel and coordinates (lat/long) of the cursor position are also shown in the bottom of the panel.

Exercise

Clean up the distance lines

Remove the parts of the generated distance lines that are outside our area of interest.

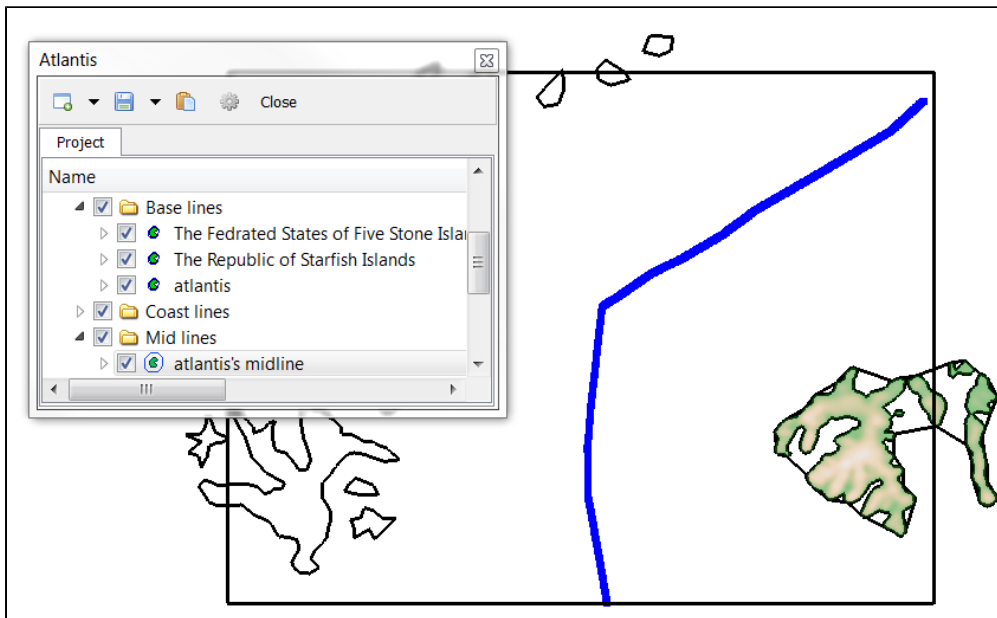
1. Right-click on the dataset and select **Edit points and lines...**
2. Click on the **Delete** tab, then on the **By closed line** tab.
3. You may want to redisplay your data using the **Display** button in the lower part of the panel.
4. Click the **Start** button and start digitizing, encircling the parts of the lines that should be removed.
5. When encircling is almost complete, click the **Connect to start and End** button. Then press **Delete points INSIDE** to remove the encircled points. (Alternatively use OUTSIDE).
6. The edited dataset is not written back into the project until **Execute** is clicked. Observe the options for **Save as an extra copy**, **Overwrite input dataset** etc.

Exercise

Generate mid-lines

Generate mid-lines between The Kingdom of Atlantis and the Republic of Starfish Islands.



1. The coastline of Starfish Isl. is found in the **Coast lines** folder.
2. You may either use the coastline as a baseline - or create a new baseline.




The mid-line calculated from Atlantis towards it's neighboring states. The coastlines of Starfish Islands and Five stone islands has been copied into the baseline folder and their schema is changed to **Baseline**. The blue line is the resulting baseline. Note that is is generated within the window frame (shown in black).

Exercise Continued

The mid-line will be generated in the **Active window**. This window is set when you use **Zoom to Data** or **Zoom to Folder**.

1. Right click the folder containing the base lines and select **Zoom to Folder**.
2. Click the  **Draw frame** icon to draw a rectangle around your window. The mid-line will be generated inside this rectangle only. If you do not like the extent of the window, try the keyboard shortcut **z** to draw a rectangle to set a new window. Display this window frame over again by  **Draw frame**
3. On a dataset of schema "Baseline" you can right-click and select **Generate mid-line**
4. In the dialog that appears, leave the default options and click **Execute**.
5. In a popup menu you are then asked about what to generate mid-lines against. Select the other baselines and click **OK**.
6. The result mid-line will be generated in a new folder called **1. Maritime Lines / Mid lines**.



Instead of using an entire coastline as baseline, you may use the  **Red Snapper** command to snap contributing / important points and use that set as a baseline.

Exercise

Mid-lines using weighted base points

Try to add weights on base points and see what happens to your mid-line. Read the In-Panel Help in the **Generate midline** command to understand how to use weight on base points.

H. The Analysis Panel

Introduction

The Analysis Panel is the tool used in Geocap to analyze bathymetric profile data, which can then be used to determine the Foot of the Continental Slope in accordance with UNCLOS Article 76. This section provides an overview of what the analysis panel looks like, how to create new bathymetric profiles from existing profiles when this panel is open and how to change the look of the various displays in the panel.

The usage and elements of the analysis panel is discussed in greater details in the next chapter, I. Foot Of Slope (FOS) Points.

Exercises

- Generating a bathymetric profile from another bathymetric profile


Exercise

Generating a bathymetric profile from another bathymetric profile

Moving your profile to another position on the seabed surface to create a new profile.

1. Right click the profile you want to use as a starting point and select **Analyze Profile**. The Bathymetric Profile analysis dialog will appear.
2. Locate the profile in the 3D window.
3. Click with the left mouse button on the gray transparent part of the frame in the 3D window, and some white spheres will become visible in the corners of the frame.
4. Click with the left mouse button the gray frame again and keep the button pressed while you move your mouse. This will translate the profile to the side.
5. Click and drag on the spheres This will rotate the profile around the spheres on the opposite side of the profile.
6. Use these controls to move the gray frame to a new position, where you would like to store a profile. For example, next to the original profile.
7. Click the **Save Profile As** button, specify a name for the new profile and click **OK**.
8. Observe that a new profile has been stored in the same folder as the original profile.
9. Select the Base of Slope with the **Select Area** button as described in the previous exercise.
10. Click **New Analysis**. This will store the analysis and store the Foot of Slope points.
11. You may now continue creating new profiles by moving the gray frame in the 3D window.

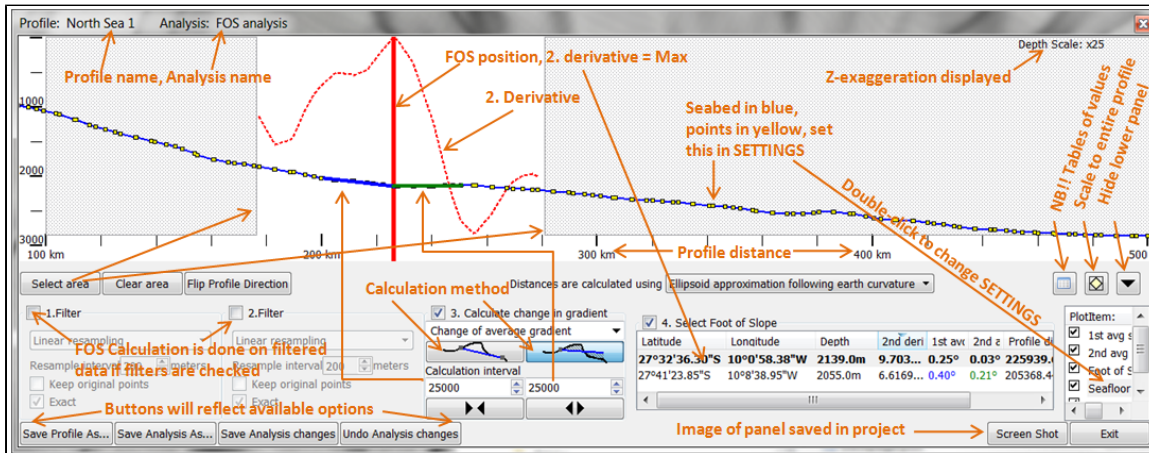


When you have made changes to the project it is always a very good idea to store them. Click the save button  in the project tool bar in order to save your project.

I. Foot Of Slope (FOS) Points

Introduction

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. FOS analysis is crucial because both formula lines (Gardiner and Hedberg) are calculated using FOS positions.



The elements of the FOS analysis panel

Some of the key variables that can be adjusted in calculating the change in gradient are:

- Change of average gradient.
- Change of gradient.
- Gradient.
- Average gradient.

Some of the other key features are:

- Select area (base of slope zone)
- Clear area
- Flip Profile Direction
- Save Profile As...
- Save Analysis As...
- Calculation interval
- Depth scale
- Various Filters for smoothing noisy data or complicated morphology

Exercises

- Calculating the Foot of Slope (FOS)
- Change the settings of the Analysis panel
- Joining FOS points into one FOS dataset
- Adding 60M to the FOS points

Exercise

Calculating the Foot of Slope (FOS)

Use the analysis panel to set foot of slope points.

1. Right click the profile you generated in the previous exercise.
2. Select the command object called **Analyze Profile**.
The profile analysis tool will appear at the bottom of your screen. You may need to un-dock the panel or expand it by dragging the edge of the window up, allowing more space to view it properly.
The black line with gray dots is the seafloor, the red dotted line is the change of gradient and the thicker red vertical line marks the point of the maximum change of gradient on your profile. 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.
3. If you want to flip the direction of the profile in the 2D window, you may do so with the **flip profile direction** button.
4. Click the select area button.
5. Click with the left mouse button in the profile window, and keep the left button pressed. Drag the mouse to the side, and release the button. This is how you select the area where the software will calculate the change of gradient.
If you do not use **evidence to the contrary**, the foot of the continental slope should be placed at maximum change of gradient at the base of the slope. So we want to restrict the calculation of the change of gradient to the **base of the slope**. Determining the **Base of the Slope** may be quite complicated, and is subject to individual interpretation, therefore we will not go into details about determining the base of slope zone in this tutorial. 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
6. Click the select area button again and select the part of the profile which you want to use as the base of the slope. You may change the parameters of the algorithm several times but for this exercise we will keep the default settings.
7. Click the **New Analysis** button. This will store the analysis, and the foot of slope point.
8. Click the plus sign (or triangle) next to the profile in the project manager and you will see that the FOS analysis is stored as a child of the profile in the tree.
9. Click the plus sign (or triangle) next to the **FOS analysis** dataset, and you will see the **Foot of Slope** dataset. This is the data set holding the foot of slope point we just made.
10. Click the **Exit** button in the bathymetric profile Dialog in order to close it.



Info

Whether you see a plus sign or a triangle in the project manager is determined by your Windows operating system or settings. The default for Windows XP is a plus sign and the default for Windows 7 is a triangle.

Exercise

Change the settings of the Analysis panel


Change the colors and other settings of the lines in the panel

1. Double-click the **Seafloor** entry in the right part of the panel.
2. Change line width and color.
3. Observe that the settings are remembered by Geocap.
4. Change the settings in the way that you want to use for all your FOS analysis documentation in the future.
5. Press the **Screen Shot** button to write an image of the analysis to your project for documentation. Alternatively, you may use the standard **Print Screen** windows option to produce a picture to the Windows clip-board.

Exercise

Joining FOS points into one FOS dataset

It is often useful to work with a group of FOS points instead of working with them individually. This exercise will show you how to collect many FOS points into one foot of slope dataset. A foot of slope point is stored as a dataset under an analysis which again is stored under a profile.


1. Click the small plus signs (or triangles) to the left of the profiles and the analysis datasets made in the previous exercise to show the foot of slope datasets. 
2. Right click the first foot of slope point and select the command, **set as active foot of slope**.
3. Right click the next foot of slope point and select the command, **Append to active foot of slope**.
4. Repeat the last step as many times as necessary to append all the foot of slope points you want in one collection.
5. Right click the folder **2. Seabed** and select **New > Folder**. The New Folder dialog will appear.
6. Select the Generic type (which is default choice).
7. Select the name to be **Foot Of Slope**.
8. Click **OK**.
9. Right click the new folder (**Foot of slope**) and select **New > Workspace data > fos**.
10. The Foot of Slope collection is now added to the folder.
11. Right click the new dataset and select **Set Schema > Foot of Slope**.
12. Right click the same dataset again and select **Rename**.
Give the foot of slope a proper name, for example the area you are working in.

Exercise

Adding 60M to the FOS points

1. Display the FOS points which you created in the previous exercise.
2. Right-click the dataset and select the command object **Generate 60M Line**.
3. The new line should now be available under **1. Maritime Lines/60M Lines**.
4. Display the 60M line.



When you have made changes to the project it is always a very good idea to store them. Click the save button  in the project tool bar in order to save your project.


J. Real Bathymetric Profiles

Introduction

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).



Tip

To locate a specific line amongst hundreds of lines in a folder, display them all, then click at the icon  **Locate project object from graphics**, then point at the relevant line. This action will select your line in the project.

Exercises

- Analyze on a bathymetric line from Geodas
- Save an adjusted bathymetric profile as a new bathymetric profile
- Combine portions of two or more bathymetric profiles into a new bathymetric profile

Exercise

Analyze on a bathymetric line from Geodas

1. Grab the handles of the analysis in the 3D view to narrow the analysis range. The handles appear as white spheres at the ends of the profile in the 3D window and they can be adjusted by left clicking on the sphere and dragging it in one direction or another.



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.


Exercise

Save an adjusted bathymetric profile as a new bathymetric profile

1. Right click a real bathymetric profile (not one created from a seabed surface) and choose **Analyze**.
2. 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. Specify a new name for the adjusted bathymetric profile and click **OK > OK**. The new profile will be saved in the same folder as the original.
5. Right click the new bathymetric profile and choose **Analyze Profile**. Notice the new bathymetric profile is now a sub-set of the original profile.

Exercise

Combine portions of two or more bathymetric profiles into a new bathymetric profile

1. Combine two or more datasets into one dataset. (The command **Append all lines in folder** may be used.)
2. Use the command  **Red Snapper** with the **m** option to sample points to your new bathymetric profile.

Note that it may be even easier to flip the folder schema temporarily to "Outer limit" and use the outer limit digitizer to combine bits and pieces of bathymetric profiles.

K. The 2500m Isobath

Introduction

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.

Exercises

- Generate the 2500m isobath from a Seabed Surface
- Edit the generated 2500m isobath
- Use the Red Snapper to prepare for generating the 2500m + 100M
- Generate 2500m isobath + 100M constraint line


Exercise

Generate the 2500m isobath from a Seabed Surface

1. Locate the etopo1 dataset which is located in **2. Seabed / Grids**. The dataset should have the schema **Seabed Surface**. If it does not have this schema, right click it and select **Set schema > Seabed Surface**.
2. Right click the dataset and select **Display 2500m isobath**. Black lines will appear where the 2500m isobaths are located. You may notice that the isobaths cover a larger area than you are interested in so we will restrict the area where the isobath is calculated.
3. Set the display window to 2D mode by clicking the **2D View** button, **2D** or typing **2** on the keyboard.
4. Zoom the display window so that it only covers the area you are interested in.
5. Right click the seabed surface and use the command, **Generate 2500m Isobath**. This will use the full resolution of the grid and calculate the 2500 isobath only in the area we are focused on. If the display window is in 3D mode it will create the isobaths for the entire dataset. If it is in 2D mode it will only create the isobaths inside the display window.
6. Locate the result dataset which should be in **2. Seabed / 2500 Isobath**. The Isobath dataset you have created may still have some line pieces which you want to remove before generating the 100M line, for instance the pieces that are seaward of the FOS points.

Exercise

Edit the generated 2500m isobath


1. Clear the display window by clicking the  button in the main toolbar.
2. Right click the dataset and select the command **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**.
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, and end the digitizing.
8. Click **Delete points INSIDE**.

Exercise continued


The part inside the closed polygon is now deleted. You may repeat the three last points in order to delete some more. The changes you make are not performed on the dataset directly, but as a local copy. In order to store the changes in the **Edit points and lines** menu, click the **Execute** button at the bottom. A new copy of the data set is stored in the folder.

1. Click the **Cancel** button in the **Edit points and Lines** menu.
2. Clear the display window again.
3. Display the edited isobaths and confirm that the unwanted parts have been removed.





When you have made changes to the project it is always a very good idea to store them. Click the save button  in the project tool bar in order to save your project.

Ordinarily, a 2500m isobath generated on a surface (as a contour) contains very many points, often thousands of points. It is recommended to snap a subset of these data to create a much smaller dataset containing fifty to a few hundred more or less critical points to use as input

for the 2500m + 100M generation. For this we use the command "Red Snapper" .


Exercise

Use the Red Snapper to prepare for generating the 2500m + 100M

1. Blank the screen
2. Select the 2500m isobath dataset in the project.
3. Double click the  **Red Snapper...** command which can be found in the **Editing** folder under **Shared commands** in the **Toolbox**. The  **Red snapper** dialog will appear.
4. Click the **Display selected data with green points** button. This will display the points that the coastline is generated from.
5. Click **Start snapping**.
6. Keep your cursor at a distance outside the 2500m isobath, and observe how nearest point is snapped when clicking the left mouse button.
7. Continue snapping a representative "critical points" dataset from your original 2500m isobath.
8. When you are finished digitizing click the **End snapping** button. The red snapper line should appear in workspace or folder according to your settings.



Tip


Observe that in the  Red Snapper command (and also in the Tools > Quick Digitizer) you may enhance functionality by enabling more keys. E.g. **m** can be used to snap all points up to next picked point. This is probably not needed here.

Exercise

Generate 2500m isobath + 100M constraint line

1. Display the 2500m isobath which you created in the previous exercise.
2. Right-click the isobath and select the command **Generate 100M Line**.
3. The new line should now be available under **1. Maritime Lines/100M Lines**.
4. Display the 100M line.



When you have made changes to the project it is always a very good idea to store them. Click the save button  in the project tool bar in order to save your project.

L. Seismic Interpretation

Introduction

In order to use the sediment formulae criterion in UNCLOS Article 76, we need a dataset containing the sediment thickness. This dataset can be based on seismic data. This tutorial has some exercises which cover the basics for working with seismic data.

Exercises

- Import SEG-Y files
- Interpret a seismic line

Exercise

Import SEG-Y files

This exercise will guide you through import of a seismic line. 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. In the example we will use the SEG-Y file which is stored with the Atlantis project.

A SEG-Y file can only be imported into a **seismic data** folder. If you are using Geocap's standard UNCLOS folder structure then such a folder is located in **3. Sediment Data/Seismic Lines**. In this exercise we will also generate a sub-folder in order to organize our data.

1. Locate the folder **3. Sediment Data/Seismic Lines** in your project
2. Right click the folder and select **New > Folder**
3. Right click the folder and select **Rename**. Give the folder the name of the survey. In this case **ATL-LOS-00** (or give it another name if you are importing into the Atlantis project and the folder already exists)
4. Right click the folder and select **Import > SEG-Y 2D**. The SEG-Y import menu will appear.
5. Click the **Defaults** button in order to reset the menu to the default options.
6. Click the file browse button and select the **ATL-LOS-00-1.segy**-file located in the Atlantis folder structure on disk (Atlantis/data/Atl-los-1-segy).
7. Click Open. The file follows the SEG-Y standard and we do not have to change the default settings.
8. It might be a good idea to check that the **Storage:** in the lower right corner is set to **8 bit**.
9. Click **Execute** and the file is imported.
10. Click close in order to close the SEG-Y import command.

In order to calculate the sediment thickness based on a seismic line we have to interpret the seabed surface and the base of sediments. This is done by using the Seismic Interpretation (Xi) functionality.

Exercise

Interpret a seismic line

In this exercise we are going to interpret the seabed and basement of the seismic line. First we need to display the seismic line we want to interpret.

1. Right click the seismic line and select **Seismic display**.
2. Right click the seismic line and select **Zoom to Data**


Next we open the Seismic Interpretation (Xi) dialog:

- Locate **Seismic Interpretation** under **Tools** in the main menu.

First we need to select where our interpretations should be stored:

1. Click **Browse** button to select the interpretation folder
2. Select the seismic interpretation folder **3.Sediment Data / Seismic Interpretations**

Next we need to specify the horizons we want to interpret:


1. Click the  **New** button in order to create a new IHorizon. The IHorizon dialog will appear
2. Type in **Seafloor** as the name of the new horizon, and click **OK**
3. Click the same new button again, and call the next horizon **Sediment Base**

Exercise Continued

Before we can start interpreting we need to specify which horizon it is that we want to interpret.

- Click the **Browse** button in the IHorizon row. Select Seabed and click **OK**

Next we need to select which seismic line we want to interpret:



- Click the  **Select** button and click on the seismic line in the display window (with your left mouse button)

Finally, since we're starting off with manual interpreting, we turn off the snap.



- Set **Snap mode** to  **Off**

Exercise Continued


We have now set up everything we need and are ready to start interpreting. Interpreting manually:

1. Click the  button.
2. Pick points on the seismic line by pointing the mouse cursor on the seismic line and clicking the **p** key on the keyboard. If you want to undo a pick, you can click the **d** key.
3. When you are finished interpreting click the  button.

Editing an interpretation:

1. Click the  button.
2. Re-pick points in an area where you have picked points before. You pick points the same way as you did the first time.
3. When you are finished interpreting click the  button.
4. Observe that the old interpretation is updated with the new interpretation.

Deleting parts of an interpretation:

1. Click the  **Erase** button.
2. Pick two points: the start- and the end point of the part of the interpretation that you want to delete.
3. The part of the interpretation between the two points is automatically deleted.



Exercise Continued

Interpreting with auto-tracking:

1. To select **Snap mode** look at the seismic display and identify the reflector representing the seabed. Compare the color of the seabed reflector with the seismic color table. The color on the left side of the color table represents minima; the color on the right represents maxima. The seismic color table used is one of three:
 - current active seismic color table – look in the lower right corner of Geocap.
 - the color table specified in **Seismic display**, if any.
 - the color table dragged and dropped on top of the seismic line, if any.
2. Select **Snap mode** accordingly.
3. Select **Track mode** to be **Auto**.
4. Set **Delta Z**. **Delta Z** is the maximum allowable vertical distance between two auto-tracked points. Z is usually time in ms. A good rule of thumb is that Delta Z should be the double of the sample interval in the seismic. In our case that is **8**.
5. Pick the horizon to autotrack by clicking the **p** key on the keyboard
6. The horizon is automatically interpreted

Exercise Continued

Interpreting with insertion (semi-auto):

1. Select **Snap mode** to be **Max**
2. Select **Track mode** to be **Insert**
3. Set **Interval** to **5**. This will interpret a point at every fifth trace.
4. Click the  button.
5. Pick a point on a strong reflection (for example the seabed).
6. Pick a new point a bit to the side, on the same reflection. Observe that auto tracker automatically interprets the line.
7. When you are finished interpreting click the  button.
8. Observe that the old interpretation is updated with the new interpretation.

Exercise Continued

The auto tracker can be very useful on strong reflectors. However when interpreting the base of sediments, the manual approach is probably better.

1. Use a combination of the editing methods above to do an interpretation of the seabed
2. Before starting on the base of sediments, we need to tell the interpreter that we are starting on a new horizon.
3. Click **Browse** on the IHorizon row, and select **Sediment Base**
4. Interpret the sediment base using the same methods as for the seafloor.
5. When you are finished with both horizons, click the **Close** button.

M. Depth Conversion

Introduction

In order to use the sediment formulae criterion in UNCLOS Article 76, we need a data set containing the sediment thickness. This data set can be based on seismic data. This tutorial has some exercises which cover the basics for working with seismic data.

Exercises

- Generate Sediment Profile using a velocity look-up table
- Import a velocity data set
- Convert Stacking velocities or RMS velocities to Interval velocities
- Generate sediment thickness profile from seismic interpretation using interval velocity dataset

Exercise

Generate Sediment Profile using a velocity look-up table

In this exercise we are going to use seismic interpretation of the seabed and the sediment base combined with interval velocity table to generate a sediment profile.

The seismic line interpreted in the previous tutorial 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 order to generate the sediment thickness, we need at least an interpretation of the sediment base, and the seabed surface. Geocap can also use more interpreted horizons if you provide them. You will then need to add velocities to these horizons as well. The interval velocity used at a horizon should be the interval velocity in the interval between the horizon and the horizon above.

1. Locate the folder **3. Sediment Data / Velocities / Velocity Profiles**
2. Right click the folder and select the command **Generate Velocity Profile**
3. Browse in the Seismic line, and the interpretation folder in the Input data group box.
4. Select **Use interval velocities from: table**
5. Type in horizon name **Seabed Surface** and Interval velocity **1500**, and click add
6. Type in horizon name **Sediment Base** and the interval velocity **2100**, and click add

It is important that the horizon names match exactly the names you used when interpreting the seismic lines. The velocities are the interval velocity to use in the interval above the specified horizon. In this case we use 1500 meter/second for the velocity in water, and 2100 meter/second for the interval velocity in the sediments. It is also possible to use a velocity function. This is however not covered in this exercise.

- Click **Execute**

If you select folders containing the seismic lines as input instead of single data sets, Geocap will try to generate more velocity profiles in one go. A dialog with a list of the matching seismic lines will appear.

1. Click **OK**, and the velocity profiles will be generated in the **Velocity Profiles** folder
2. Click **Close** in order to close the **Generate velocity profile** dialog

Exercise Continued

1. Right click the new velocity profile and select **Map Data** in order to see the profile

The velocity profile holds all the data necessary to do generate a sediment thickness. It holds both the interpretations in time, and the interval velocities to use in the interval between the interpretations.

- In order to generate a sediment thickness profile, right click the velocity profile, and select **Calculate Sediment thickness**

A new data set will be created in **3. Sediment Data/ Sediment profiles**

- Right click the new data set, and select **Table view**. The **Table view** dialog of the data set will appear.

The x,y and z is the position and the sea depth at that point. The **scalar** value is the sediment thickness. The Shot point column is the points corresponding shot point from the seismic line.

1. Click **Close** in order to close the dialog.
2. Right click the data set again, and select **Display profile as cross section**

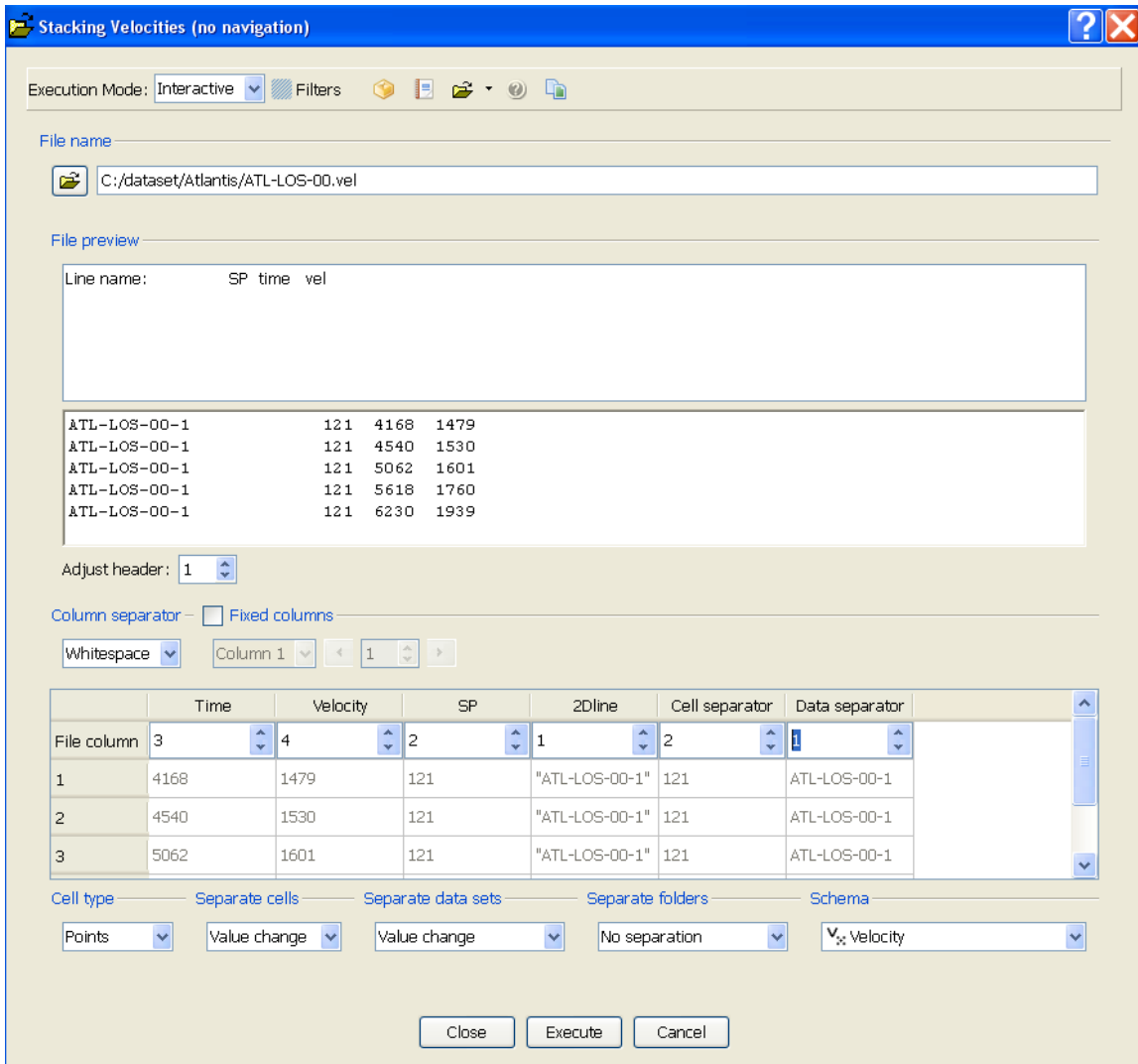
A sediment profile should now be displayed, you may need to un-toggle the Velocity display and the seismic display, if you are still displaying them in order to get a good view of it.

Exercise

Import a velocity data set

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.

1. Locate the folder **3. Sediment Data / Velocities / Stacking Velocities**
2. Right click the folder and select the command object **Import > Stacking Velocities (no navigation)**
3. Browse in the data set called **ATL-LOS-00.vel**
4. Adjust the columns according to the following screen shot, and click **Execute**



Stacking Velocities import menu

Exercise Continued

The data set has now been imported into your project. However it does 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. Geocap will match the shot points in the navigation file with the shot points in the velocity file.

1. Right click one of the velocity data sets, and select the command object **Assign Navigation**
2. Browse to **3. Sediment Data /Seismic Lines** and highlight the corresponding seismic line
3. Click **OK**
4. Right click the velocity data set again, and click **Map Data**

A color map of the velocity data should appear. If you display the seismic line at the same time, you should be able to see that the velocities match the seismic line.

The velocity data sets should have the exact same name as the seismic line. If this is the case, then you can use the assign navigation command object on the folder containing the velocity data. You should then browse in the folder containing the navigation. Geocap will match the data sets by their name, and assign navigation accordingly. This way you do not need to assign navigation to one and one data set, but can do them all in one go.

Exercise

Convert Stacking velocities or RMS velocities to Interval velocities

If you have imported stacking velocities or RMS velocities, we need to convert them to interval velocities before we can use them as input to the **Generate Velocity Profile** command.

1. Locate the folder **3. Sediment Data / Velocities / Stacking Velocities**
2. Right click the folder and select the command object **Compute Interval Velocities (Dix Formula)**

The result will be stored in the folder **3. Sediment Data / Velocities / Interval Velocities**.

Dix formula will produce interval velocities if it is run on RMS velocities. You may also use this formula on stacking velocities, and thereby use the stacking velocities as an approximation for RMS velocities.

Exercise

Generate sediment thickness profile from seismic interpretation using interval velocity dataset

This exercise is quite similar to the first exercise in this tutorial. The only difference is that we will base the velocities on a data set in the project instead of a look-up table

In order to generate the sediment thickness, we need at least an interpretation of the sediment base, and the seabed surface. Geocap can also use more interpreted horizons if you provide them.

1. Locate the folder **3. Sediment Data / Velocities / Velocity Profiles**
2. Right click the folder and select the command object **Generate Velocity Profile**
3. Browse in the Seismic line, and the interpretation folder in the Input data group box.
4. Select **Use interval velocities from: Data set in Project**
5. Browse in the Interval velocity input data group box.

It is important that the Interval velocity data set has the same name as the corresponding seismic line. It is also important that the shot point numbering in the velocity file correspond to the shot point numbering of the seismic line.

- Click **Execute**

Exercise Continued

If you select folders containing the seismic lines and the velocity data as input instead of the data sets, Geocap will match the names, and try to generate more velocity profiles in one go. A dialog with a list of the matching seismic lines will appear.

1. Click **OK**, and the velocity profiles will be generated in the **Velocity Profiles** folder
2. Click close in order to close the **Generate velocity profile** dialog
3. Right click the new velocity profile and select **Map Data** in order to see the profile

The rest of this exercise is similar to the end of the first exercise in this Tutorial. The velocity profile holds all the data necessary to do generate a sediment thickness. It holds both the interpretations in time, and the interval velocities to use in the interval between the interpretations.

1. In order to generate a sediment thickness profile, right click the velocity profile, and select **Calculate Sediment thickness**. A new data set will be created in **3. Sediment Data/ Sediment profiles**
2. Right click the new data set, and select **Table View**. The table view dialog of the data set will appear.

The **X,Y** and **Z** is the position and the sea depth at that point. The **scalar** value is the sediment thickness. The Shot point column is the points corresponding shot point from the seismic line.

1. Click **Close** in order to close the dialog.
2. Right click the data set again, and select **Display profile as cross section**

A sediment profile should now be displayed, you may need to un-toggle the Velocity display and the seismic display, if you are still displaying them in order to get a good view of it.

N. Sediment Thickness

Introduction

Aim of this tutorial:

1. Generate Sediment Profile from seismic interpretation
2. Import Sediment Thickness points from ASCII file
3. Import Sediment Thickness Grid
4. Compute Sediment points > 1% of distance to FOS

Exercises

- Import Sediment Thickness points from ASCII file
- Import Sediment Thickness Grid
- Compute Sediment points > 1% of distance to FOS

Exercise

Import Sediment Thickness points from ASCII file

This exercise will assume that you have an ASCII Column file containing coordinates with sediment thicknesses. In the exercise we will use a file which is included in the Atlantis Project the file includes more than one profile.

1. Locate the folder **3. Sediment Data / Sediment profiles**
2. Right click the folder and select **add folder**
3. Add a sub folder with the name of the survey for example **ATL-LOS-00**
4. Right click the folder and select **Import > ASCII Column...** and the ASCII Column import dialog will appear.
5. Click the browse button, and select the sediment thickness file located in the Atlantis folder: **Atlantis \ Data \ ATL_Sediments.txt**
6. Adjust the number of header lines in your file with the **Adjust header** Spin box to one. The header lines should be displayed in the top part of the file preview, while the bottom part of the file preview should hold the coordinates.
7. Set the column separator to **whitespace**
8. Select coordinate format **X Y**
9. Check the **Z** and the **Scalar** Columns checkboxes
10. Check the **Additional attributes**
11. Type in **Shot point** under Additional attributes, and select number, then click **Add**. A new column will appear for the Shot points
12. Set the cell type to **Line**
13. Set **Separate Data Sets** to **Value change**
14. Set Schema to **Sediment Thickness**
15. Correct file columns for each data column by adjust the spin boxes . x: 3 y: 4 z: 5 Scalar: 6 Shot point: 2 Data separator: 1
16. click **Execute**
17. Click the **Close** button in order to close the ASCII Column import dialog. The data has now been imported. Let us display the data:
18. Right click one of the datasets and select **Scale window to data**
19. Right click the data set again, and select **Display profile as cross section**

Exercise

Import Sediment Thickness Grid

The NGDC has made the **World sediment thickness grid** which is an approximation of the sediment thicknesses covering the entire world. It is available from the internet with the link: <http://www.ngdc.noaa.gov/mgg/sedthick/sedthick.html>
Import that grid

1. Open the link in your internet browser. Geocap should be able to import all of the formats which are presented. For this exercise you may download the **GMT NetCDF** file.
2. Click the **Download GMT NetCDF grd file**. Store the file, and Unzip the grid to your hard drive.
3. Locate the folder **3. Sediment Data / Grids** If you do not have this folder you may create it. Use the generic folder type which is the default option.
4. Right click the folder and select **Import > Generic...** and the Generic import dialog will appear. Click the browse button, and select the sediment thickness file.
5. Click the **Area of Interest** tab. Here you will see extent of the file. You may specify the import area at the bottom.
6. Check the import area button, type in the coordinates of your area in decimal degrees.
7. Click **Execute**.
8. You will now be prompted with a question of the coordinate system and datum. Select: World Geodetic system 1984 Geodetic. Click **OK**
9. Click **close**.

The data is imported in Geodetic, with latitudes as Y coordinates and longitudes as X coordinates. In order to view the data with the other data we have imported need to convert the grid to Mercator coordinates:

1. Right click the dataset and select **shared commands > convert to Mercator**
2. Change the result combo box to **Replace input**
3. Click **Execute**. The data is now converted.

Let us display the data:

1. Right click the dataset and select **Scale window to data**
2. Right click the dataset and select **Map**

You should now see the sediment grid in your area.

Exercise

Compute Sediment points > 1% of distance to FOS

In the previous exercises we imported and calculated sediment thicknesses. In order to apply the sediment thickness formula in UNCLOS Article 76, we need to compare the sediment thickness with the distance to the foot of slope. In this exercise we will cover how you can do this calculation. We assume that you already have a foot of slope dataset.

1. Right click the sediment thickness data set.
2. Select the command **Generate >1% of distance to FOS** A dialog asking for the Foot of slope data set will appear
3. Browse and select the Foot of slope data-set you want to use for the calculation, click **OK**.

If the data-set contained any points thicker than 1% of the distance to the nearest FOS point, it will store those points in the project, and inform you of the folder used. It will also display these points.

O. Final Outer Limits

Introduction

The construction of outer limits is (obviously) a complex operation. Sometimes it is best to do this area by area and in multiple steps, then combining points into one or more final outer limits. Sometimes the outer limit line may be constructed from a combined formula line and a combined constraint line.

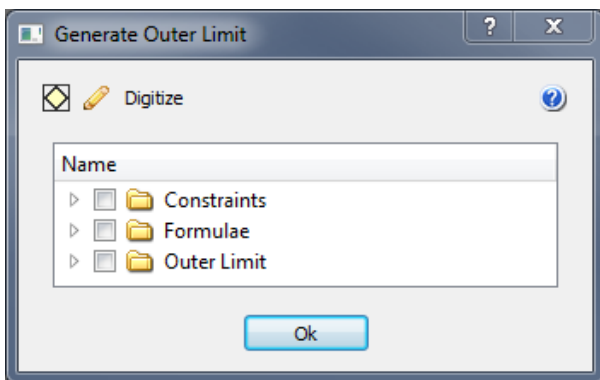
The ultimate construction may still be a complex logical and strategical operation. The Geocap Final Outer Limit tool is designed to digitize to combine line segments and keep distances between points less than 60M.

Exercises

- Try the Outer Limit tool
- Generate the Outer Limit

About the Outer Limit Menu

The "Outer Limit Menu" is used in order to select the final outer limit points from the formula and constraint lines. You may open the outer limit menu by right-clicking on the project folder **4. Outer limit** and selecting **Generate Outer Limit**.



The Outer Limit Menu

The blue question mark in the top right corner of this menu contains instructions on how to use this menu.

The folders in the Outer Limit menu correspond to the sub folders in **4. Outer limit**. You will also be able to see the datasets in this list. Before opening this menu, you can copy the files you want to use in your outer limit to the correct folders.

You can display the different data sets by checking the data sets you want to display, and clicking the pencil. Multiple select is allowed in this list. You can scale the display window to the data sets the same way by checking the datasets, and clicking the scale button.

The digitize button is used to digitize points from the checked data-sets. You can use this to pick points on your final outer limit. Each time you pick a point, a white circle of 60M will be displayed around the last picked point. The circle is geodetic correct. This will be a guide to you when digitizing the final outer limit. The distance between the final outer limit points should be less than 60M, so the next point you pick should be inside the circle.

Exercise

Try the Outer Limit tool

Before doing the exercise below, get a hand on feeling with the functionality of the digitizer, just by playing around with two datasets.

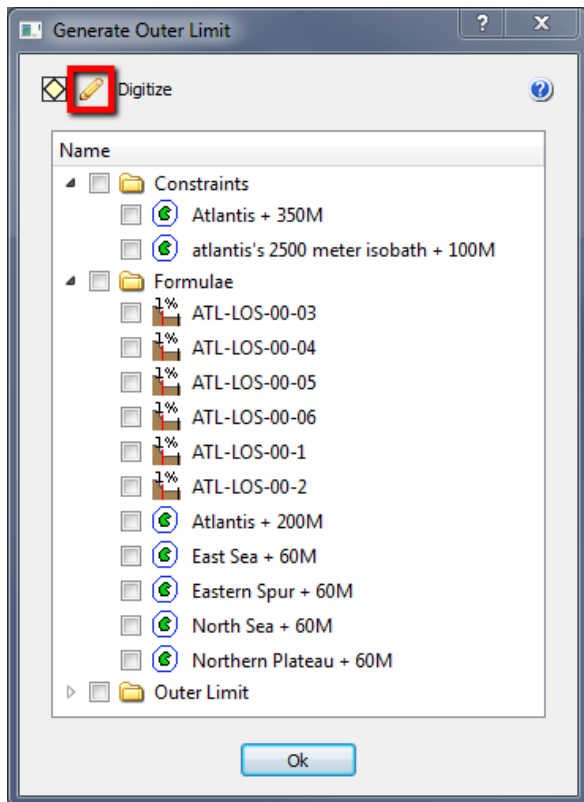
- Copy the 200M line into **4. Outer limit / Formulae**
- Copy your 2500m + 100M into **4. Outer limit / Constraints**
- Display the data from the Final Outer Limits panel. Note that the data may also be displayed from the project, e.g. Display Points.
- Try the various digitizing operations.

Exercise

Generate the Outer Limit

In this exercise we are going to digitize the outer limit from your formula, and constraint lines using the Outer Limit menu.

1. Copy all your formula lines to the **4. Outer limit / Formulae** folder (200M, FOS+60, Sed>1% points)
2. Copy all your constraint lines to the **4. Outer limit / Constraints** folder (350M, 2500m + 100M)
3. Open the Outer limit menu, and check some of the datasets and use the **Display** and **scale** buttons

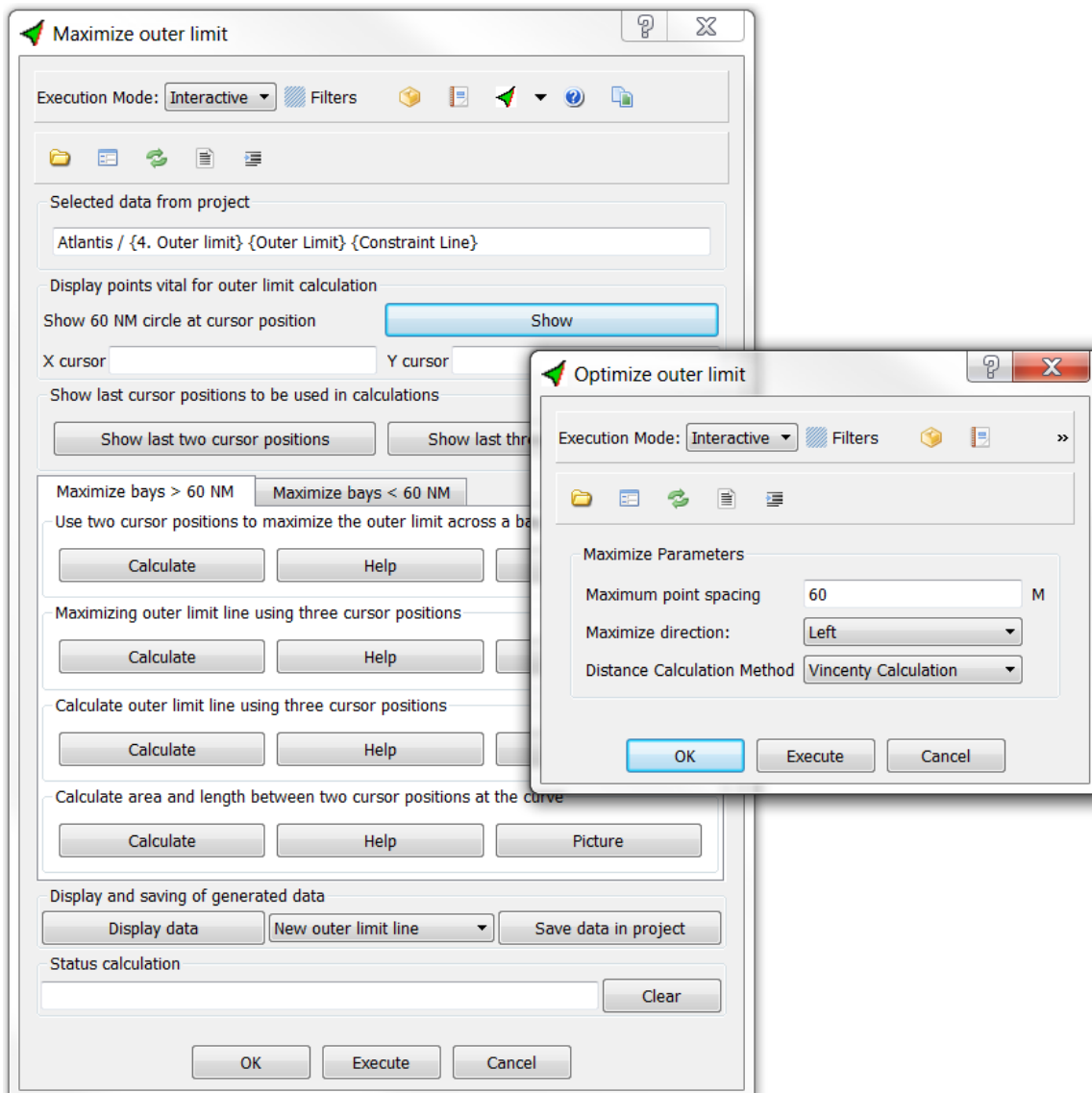


The Outer Limit menu with Datasets Copied

Exercise Continued

1. Digitize a combined formula line, by clicking the digitize button and following the instructions
2. When you have finished digitizing and clicked the end digitize button, give the new dataset a name, such as **combined formula line**.
3. Right click the new line in the project, and select **Table view**
4. Observe the new columns in the table

There are also two tools which have been developed in order to **Optimize** the outer limit and to **Maximize** the outer limit. These tools should be used with great care in order for them to be utilized correctly. It cannot be assumed these tools will automatically produce the absolute maximum achievable area. They are interactive tools that should be experimented with, using different settings until the desired result is obtained.



The Maximize outer limit panel together with the Optimize outer limit panel

P. Uncertainty

Introduction

Exercises

- Vertical to horizontal uncertainty
- Vary sediment thickness

Exercise

Vertical to horizontal uncertainty

In this exercise we will investigate the implications on the 1%-thickness points/area of varying the sediment thickness. Create alternative sediment thicknesses, either by:

1. Multiplying the thickness by 0.9 to produce a minimum thickness and multiply by 1.1 to produce a maximum thickness.
2. Change the depth conversion to produce an alternative thickness.

How to vary sediment thickness (uncertainty in thickness)

The thickness along a profile can be adjusted up and down in order to measure the horizontal effect on the 1% point (1%point is used in the following for the point used in UNCLoS art. 76 where the thickness of sediments are equal to 1% of the distance to the FOS). FOS = Foot of slope point. If you have a seabed (in depth) and an interpretation of the base of sediments, you may use formulas or methods in Geocap to depth convert the base of sediments. If you have used Geocap methods with stacking velocities and Dix interval velocities described in chapter 17 in the user manual, you will end up with **Sediment profiles** and **Sediment thickness points** as present in the Atlantis project.

Exercise

Vary sediment thickness

- Right-click **ATL-LOS-00-2** in **3. Sediment Data/Sediment profiles** and click **Table view**

This table contains x, y and z for the seabed bathymetry. The Scaler (S) contains the sediment thickness. The dataset also contains shotpoint numbers.

	X	Y	Z	scalars	distance to fos	nearest fos index	percent of distance to fos	shot point
2588	-1 205 404.7...	-2 750 783.250	3 102.073	1 702.056	167 170.7	3.000000	1.018154	6 430.000
2589	-1 205 383.7...	-2 750 768.000	3 102.083	1 693.096	167 194.3	3.000000	1.012652	6 431.000
2590	-1 205 363.2...	-2 750 753.000	3 102.094	1 684.157	167 217.4	3.000000	1.007166	6 432.000
2591	-1 205 342.5...	-2 750 737.750	3 102.260	1 674.997	167 240.9	3.000000	1.001548	6 433.000
2592	-1 194 745.2...	-2 742 934.000	3 118.961	1 794.460	178 881.6	2.000000	1.003155	6 944.000

Display marked points

Close Edit Mode

Table view (Note the checkbox allowing a visualization of the table entries).

Exercise Continued

We want to create a thinner (minimum) profile and a thicker (maximum) profile.

1. Right-click **ATL-LOS-00-2** and click **Copy**
2. Right-click on the **Sediment profiles** folder and click **Paste** twice.
3. Name the new datasets **ATL-LOS-00-2- max** and **ATL-LOS-00-2- min**
4. Right-click on **ATL-LOS-00-2- max** and click **Make active**
5. Go to the **shell** in the **toolbox** and type: **mul sca 1.1**

Now, the scalar (i.e. the thickness) is increased by 10%.

1. Right-click on **ATL-LOS-00-2- max > Assign from workspace > active**
2. Right-click on **ATL-LOS-00-2- max > Generate 1% dist to FOS area**. (You may be asked for projection which is Mercator, central meridian 0.)
3. You will be asked for foot of slope point(s), and you can select the **FOS's Northern Plateau**.
4. Repeat this for **ATL-LOS-00-2- min** but multiply by 0.91.

The two new datasets that are created have all points on the lines that falls within the 1% thickness criterion.

Q. Gridding

Introduction

Read about gridding in the User Manual. The gridding commands collection can be reached on the right-click menu on any dataset of schema **Poly Data**. Below follows a step-by-step procedure for gridding, using the Seafloor functionality in Geocap.

Exercises

- Creating a Seafloor project.
- Importing data
- Displaying raw point data (soundings)
- Creating a chart
- Extract soundings
- Gridding chart
- Display chart data
- Display accepted and rejected points

Gridding multibeam XYZ-data in Seafloor

This demo will take you through the steps from importing Multibeam XYZ data to displaying the final grid. The multibeam XYZ data can be found in the **/Data/Seafloor** folder in the Atlantis project. Note that the Atlantis project is in Mercator, central meridian 0.

Exercise

Creating a Seafloor project.

1. Click **File > New > Project**
2. Select the **Seafloor** project template and a name and path for your project. Click **Finish**.
3. Click **Seafloor** at the menu bar and open the **Seafloor Module Main Panel**.
4. Click **Settings** and **Geodetic Settings**. Select an appropriate datum and coordinate system and click **OK**.

Exercise

Importing data

1. Click **Import to XYZ data folder...**
2. Import all the XYZ-files with an appropriate import method.
3. You should now be able to see your data in the **XYZ data** folder under the **1. Survey Data** folder.

Exercise

Displaying raw point data (soundings)

1. Right-click the **XYZ data** folder and select **Display Soundings**
2. Select **Show soundings**, **Use colour range** and **Use same range for all**
3. Select all files (click one and then ctrl + a)
4. Select **Scale to selected files** and click **Execute**

Exercise

Creating a chart

1. Click **Create and Modify Chart**
2. Select the **Create Single Chart** tab
3. Click **Scale to soundings**
4. Check the box next to the **XYZ data** folder and click ok
5. Click **Execute**
6. You can now find your chart frame in the **Chart** folder under the **2. Charts** folder.

Exercise

Extract soundings

1. Click **Extract Soundings...**
2. Check the box next to the **XYZ data** folder.
3. Select the **Chart** folder so it is marked.
4. Click **Execute**
5. You will now see that the **Chart** folder has been updated with a **Soundings** dataset.

Exercise

Gridding chart

1. Click **Gridding Chart...**
2. Select **Parabolic** gridding algorithm with an appropriate x-increment and YInc=XInc. Use the Tools > Distance calculator to find an appropriate increment.
3. Algorithm Parameters may have search radius: 5. **Use Boundary Polygon** and **Automatically Create Boundary Polygon** should be checked.
4. Use **filter input points** and **save filtered points** with standard deviation, shallow factor: 2 and deep factor: 2
5. Use **Apply smoothing to result grid** with convolution, filter width: 1 (3x3) and filter weight: 0.5
6. Select the **Chart** and click **Execute**
7. When the gridding has finished you are presented with a dialog box providing you with details and statistics about the gridding process. At this point, this is not saved, but you can copy and paste this into a text document.

Exercise

Display chart data

1. Click **Display Chart Data...**
2. Have the following options checked: **Frame**, **Seafloor**, **Use same range for all** and **Use LOD**
3. Select the **Chart**
4. Have the **Scale to selected charts** box checked and click **Execute**

Exercise

Display accepted and rejected points

1. Right-click the **Soundings_Accepted** dataset in the **Chart** folder and select **Display**
2. Select the **Soundings Rejected** dataset in the **Chart** folder and right-click the **Display** command in the **Toolbox**
3. Select **Edit**
4. Check the **User Defined Color** box and select another color using the **Palette**
5. Click **Execute**

R. PostScript Plotting

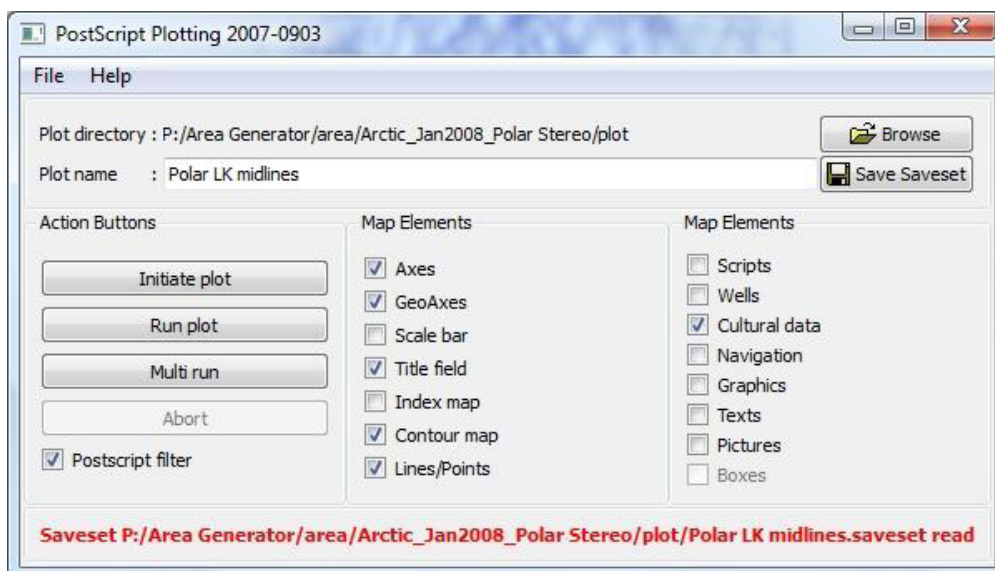
Introduction

PostScript Plotting

Exercises

- Generate a plot using your seabed surface as the main dataset.

Start the PostScript plotting panel from **Tools > PostScript Plotting**.



The panel for PostScript plotting

Read about plotting in **Help > Quick Guide to Plotting with Geocap**.

Exercise

Generate a plot using your seabed surface as the main dataset.

- Follow the steps in the **Help > Quick Guide to Plotting with Geocap** to produce a plot.

S. Workflows

Introduction

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

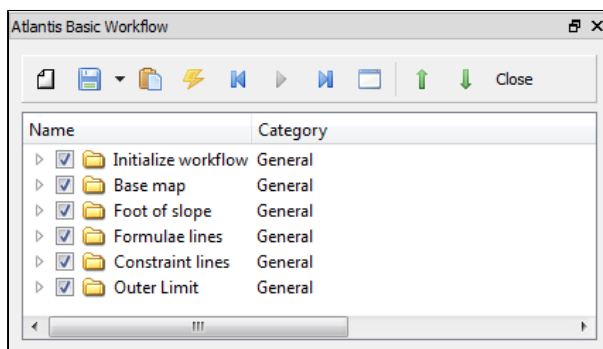
Exercises

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

Exercise

Examine the workflows provided in the Atlantis project.

1. In Geocap, click **File > Open > File**, and browse to the Atlantis folder, then open the the sub-folder called **workflow**.
2. Double click on the file called **Atlantis Basic Workflow.gwf** to open it, or single click to highlight it and then click, **Open**. This will open the workflow below the project.



Atlantis Basic Workflow

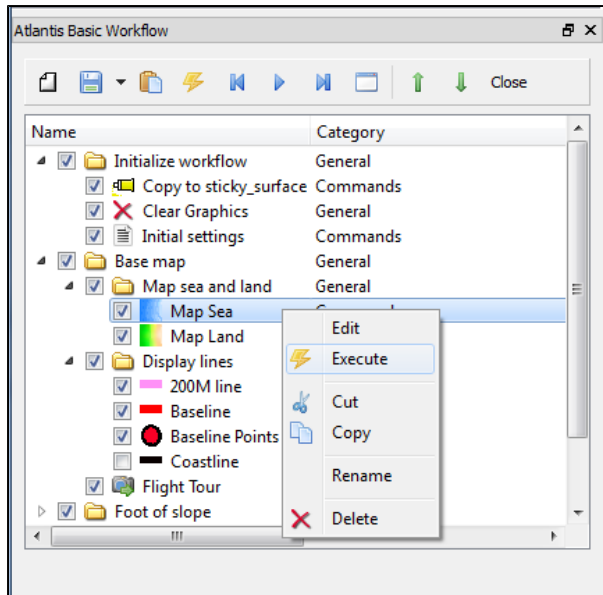
Exercise Continued

1. Examine the contents of the workflow items by clicking on the triangles next to the root folders.
2. 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)
3. Click on the **Execute next element** icon again and notice everything in the **Base map** folder will be displayed along with a short **flight tour** (animation).
4. 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.

Following the steps above, open and examine the other workflows in the **workflows** folder.



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.



Executing individual commands one by one



Exercise

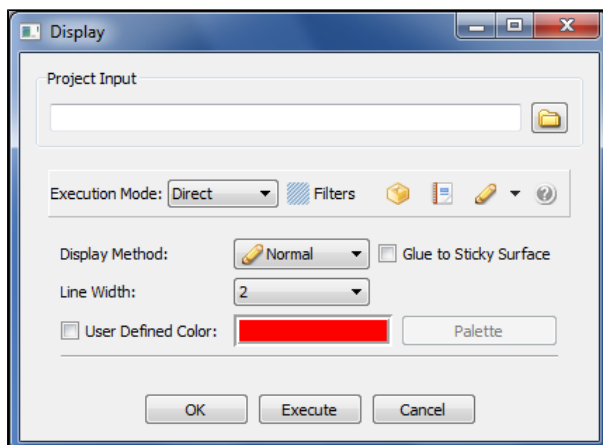
Create a new workflow

In this exercise we will create a workflow that displays seabed and land and some important lines from the project.

- Click **File > New > Workflow**. Then a browser pops up.

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

1. Press the **New Element** icon to create the first entry. That may also be achieved by a right-click **New Element**.
2. Select the **Display**  command found in the **Commands** folder and click **OK**.
3. Right-click the **Display**  command in the workflow and select **Edit**. Then this panel pops up:






The Display edit panel

Exercise Continued

In a workflow, you will need to browse in an object from the project. The command usually requires a dataset to do an action on (such as a type of display).

1. Browse to the coastline folder under **1. Maritime lines**, highlight **The Kingdom of Atlantis** and click **OK**.
2. Choose the display settings you like (color, glue to surface or not, etc.) and click **OK** again.
3. Right click the element in the workflow and choose **Execute**. Alternatively, click the **Execute** icon.
4. Examine how your display looks and edit the command if necessary to achieve the results you want by right clicking on the command and choosing **edit**.
5. In order to keep your workflow organized it is recommended to change the name of the newly added commands from **Display** to something more descriptive like, **Display Coastline Yellow** which is done by right clicking the command and choosing **Rename**.

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

1. Right click the **Map sea** command in the **Toolbox** and select **Copy** . (It should be located under **Schema Commands** if you have selected a **Seabed Surface**.)
2. Click the **Paste**  button in the workflow.
3. Again you must edit it and browse your seabed surface into the panel.
4. Remember to save the workflow by clicking .

T. 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. In this tutorial we will use a map of the structural elements of the Norwegian continental shelf, which can be downloaded from Norwegian Petroleum Directorate website. This map has a graticule showing latitude and longitude which works great as tie points.

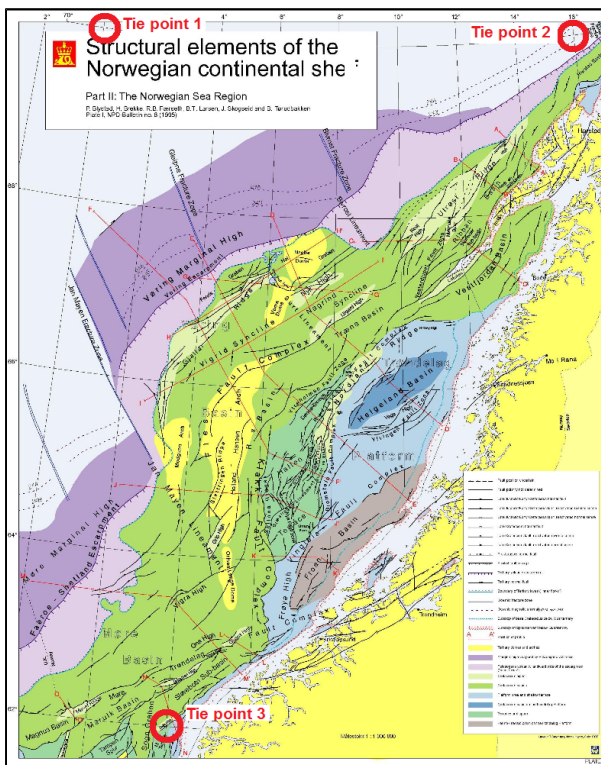
Exercises

- 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**.



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

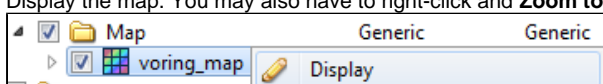
Exercise

Georeference an image

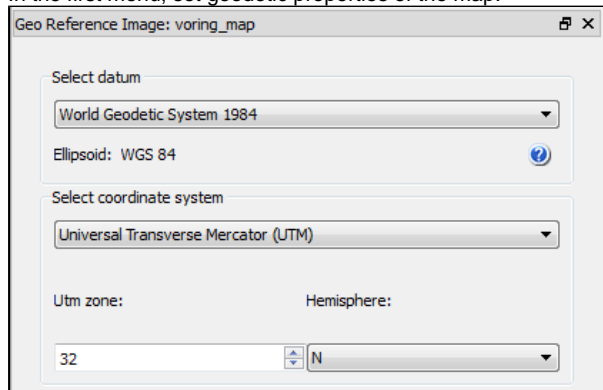
Use the Georeferencing wizard to georeference an image. Follow the step-by-step procedure below.

Georeferencing step by step

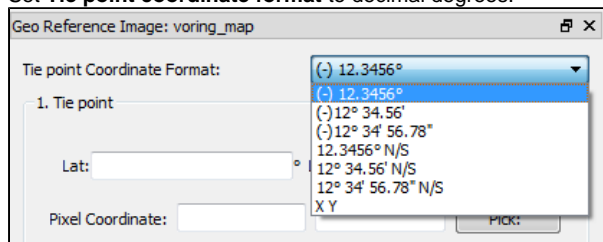
- Display the map. You may also have to right-click and **Zoom to data**



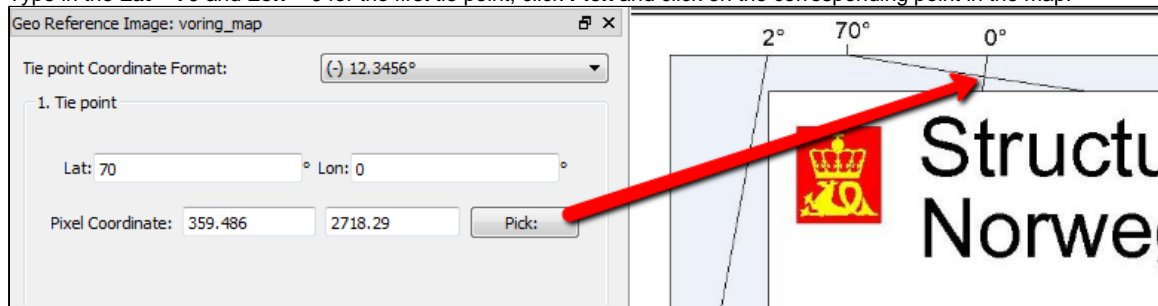
- Make sure the image is selected in the project and double-click **Geo Reference** command in the **Toolbox**. Notice that the graphic window was changed to 2D mode.
- In the first menu, set geodetic properties of the map:



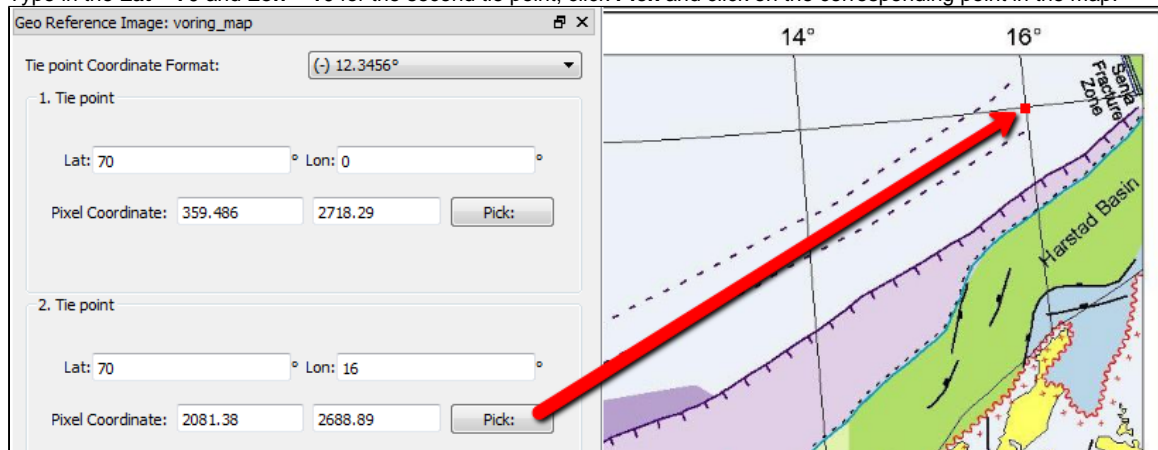
- Set **Tie point coordinate format** to decimal degrees.



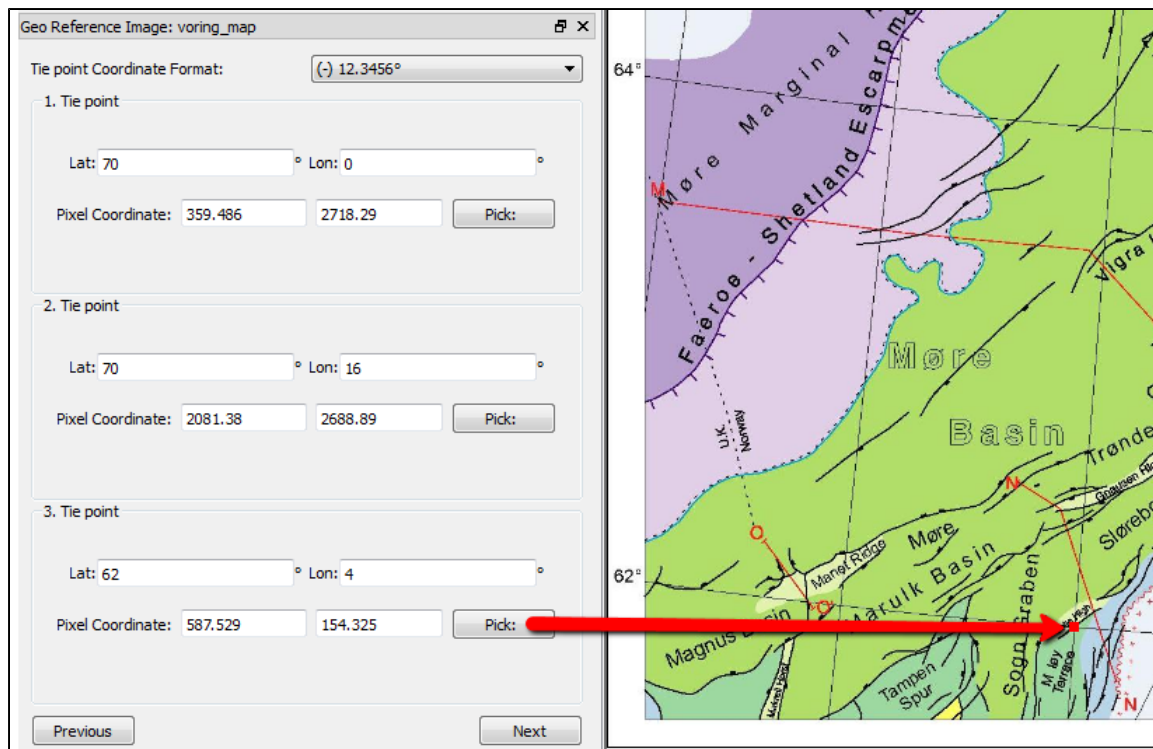
- Type in the **Lat = 70** and **Lon = 0** for the first tie point, click **Pick** and click on the corresponding point in the map.



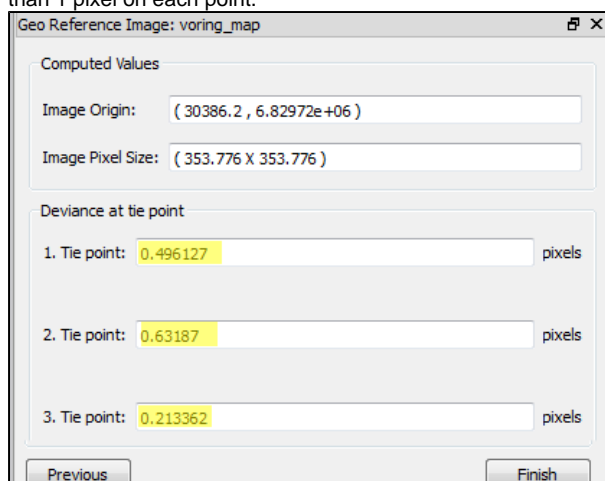
- Type in the **Lat = 70** and **Lon = 16** for the second tie point, click **Pick** and click on the corresponding point in the map.



- Type in the **Lat = 62** and **Lon = 4** for the third tie point, click **Pick** and click on the corresponding point in the map.



- Click **Next**
- Evaluate the result. The **Deviance at tie point** represents the accuracy of the georeferencing. The value should not exceed more than 1 pixel on each point.

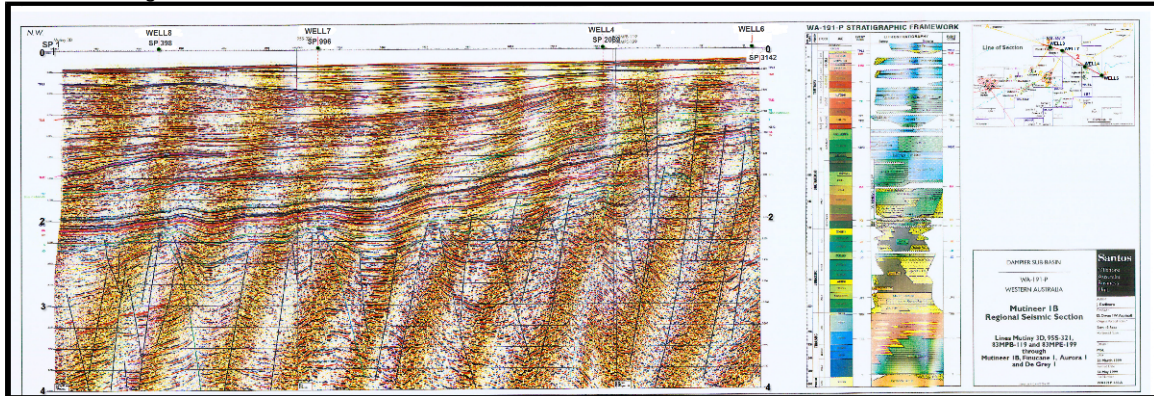


U. Vertical Image Calibration

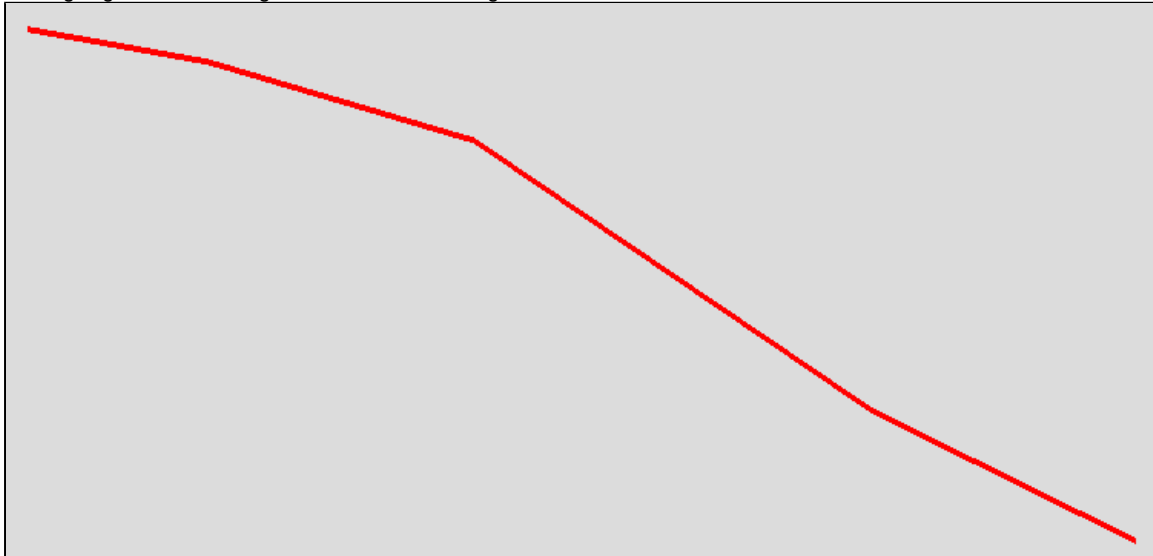
Introduction

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

The actual image to calibrate:



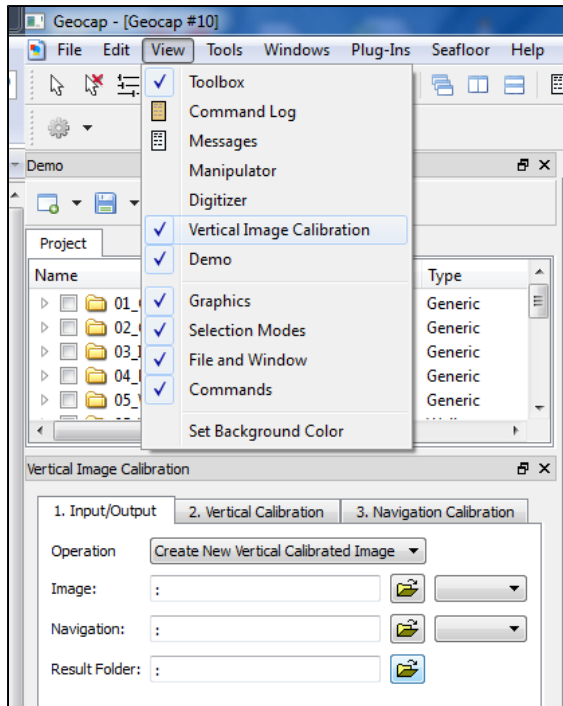
A navigation file or a digitized line to calibrate against:



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

Open VCI menu

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



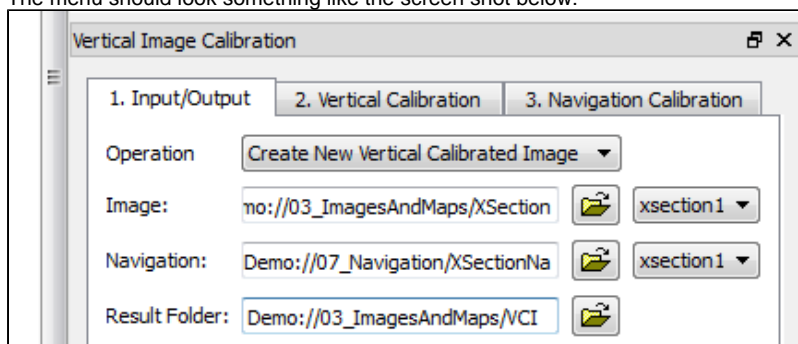
Opening the VCI menu

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

Input/Output

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

The menu should look something like the screen shot below:

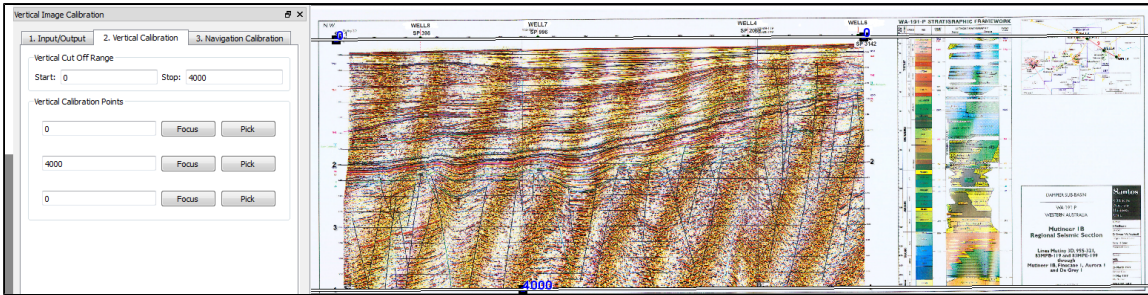


Vertical Calibration

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

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

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

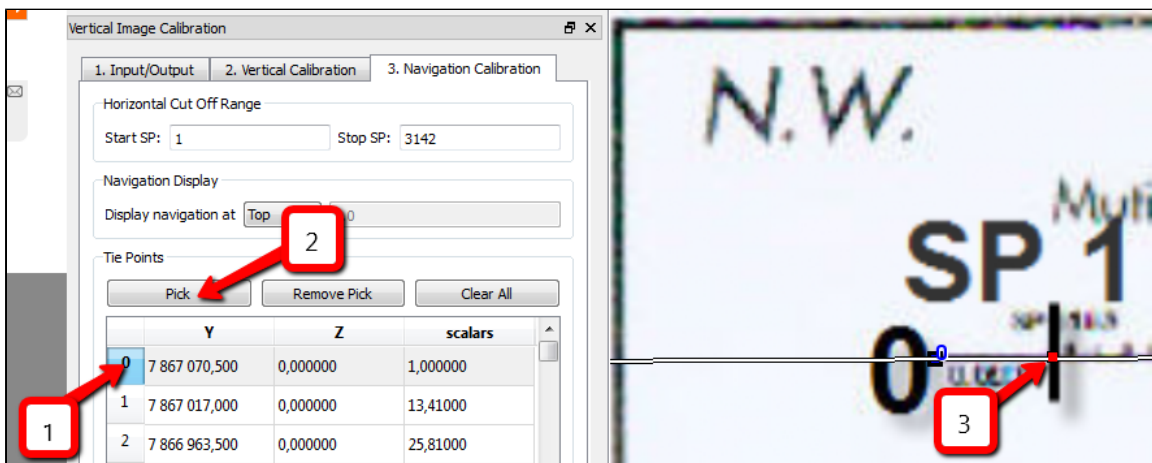


Vertical calibrated

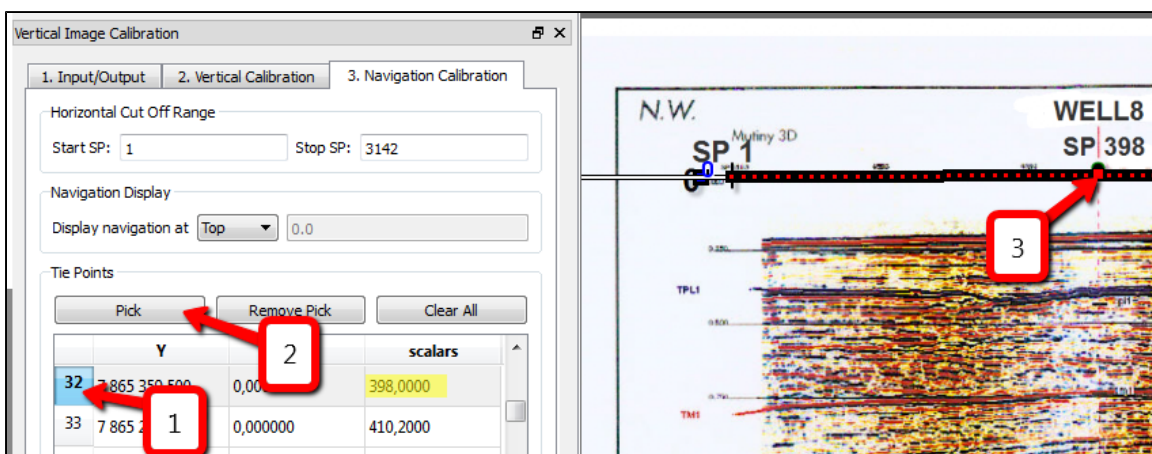
✔ If you want to keep parts of the image above 0 or below the max depth of the seismic image you can do it by adjusting the cut off range after you have set the calibration points. In the example above we could set the cut off start to -100. This would preserve the well-information at the top of the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

Navigation Calibration

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

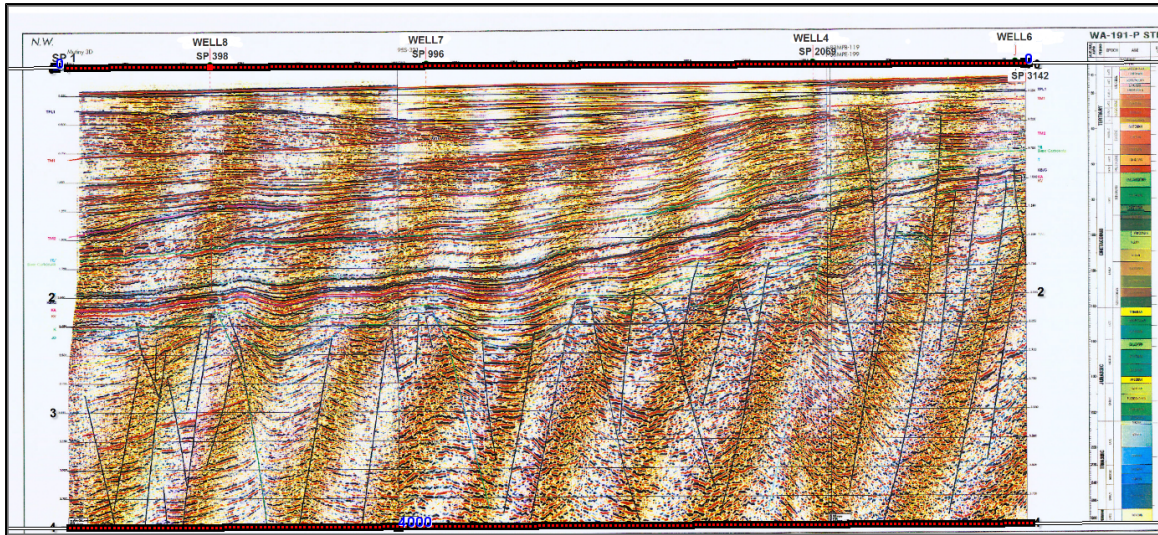


- Locate the next shot point in your image and find the corresponding shot point in the **Tie Points** list in the menu.
- Click on the **Tie Point** in the list, click the **Pick** button and click on the shot point in the image.



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


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

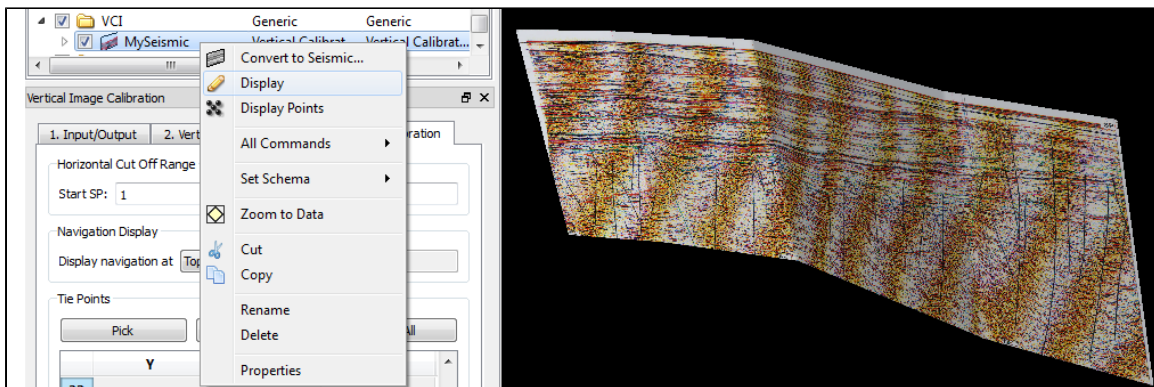



Everything to the left and right of the dotted line and everything above and below the white lines will be cut off on the completed VCI.


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

Display

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



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

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

Z. Additional Shelf Exercises

Introduction

The following exercises are just suggested for further training topics, and may not be explained in detail.

Exercises

- Make a movie
- Check my "Shared commands"
- Hello World!
- Install the script system "Area Generator Mini"
- The function command
- Composed grid
- Combine parts of two bathymetric lines into one single bathymetric profile

Exercise

Make a movie

Use a workflow or just use a the command **Flight Tour** to create a movie in avi format.

Exercise

Check my "Shared commands"

Import and activate my shared commands and see if there are commands you do like. My shared commands are found here: <http://andeby.no/geocap> Just follow the instructions on the page.

Exercise

Hello World!

Type the following line into the shell and press enter:

```
qinformation Information "Hello World"
```

Observe the popup. This is a Geocap GUI (Graphical User Interface). Study the examples in the **Reference > QTcl** section of the documentation and observe that it is fairly easy to build your own front ends to Geocap. Combined with the internal scripting language and Tcl, this is a very powerful tool.

Exercise

Install the script system "Area Generator Mini"

Read about the Area generator, download the scripts and follow the step-by-step procedure here: <http://andeby.no/geocap>

Chop out a portion of the world and build the project.

Exercise

The function command

Try the function command to time-convert your seabed surface.

```
z = 2000 * z / 1490
```

Exercise

Composed grid

Use two or more grids of different resolution (e.g. Etopo1 and a multibeam grid) and create a composed grid. Do an FOS-analysis across this composed grid. Read more about composed grids in the Article section of the [main documentation](#).

Exercise

Combine parts of two bathymetric lines into one single bathymetric profile

Use the Outer Limit Tool to do this.

- Copy two or more bathymetric lines into a sub folder of **4. Outer limit**. You may prefer to create a new folder.
- Open the Generate Outer Limit Dialog from the **4. Outer limit** folder.
- Carefully read the In-panel help under the question mark.
- Use the tool and especially option **m** to digitize a new bathymetric line.
- Use this new line for an analysis.

Exercise

Create a bathymetric profile in a grid using a line

In this exercise we are going to create a bathymetric profile from a digitized or already existing line to sample the `sticky_surface` to create a bathymetric profile. The exercise assumes that a seabed surface is activated as the sticky surface:

- Right-click on the surface in the project and select **Copy to sticky_surface**.

A line of any kind can be used, e.g. a digitized line. To digitize a line, use **Tools > Quick Digitizing** or **View > Digitizer**

1. Create a new shared command of type **Script** and rename the command to **Generate Bathymetric profile on sticky surface**
2. Enter this content:

```
mak lin grid sticky_surface
mhi bathymetric_profile
qinformation "Mission accomplished" "New profile is found in workspace
bathymetric_profile"
```

3. Execute on the line to produce bathymetric profiles.
4. Set the filter on the command to run on **Generic** folders and **Polydata** only.

Notes

Notes

Notes