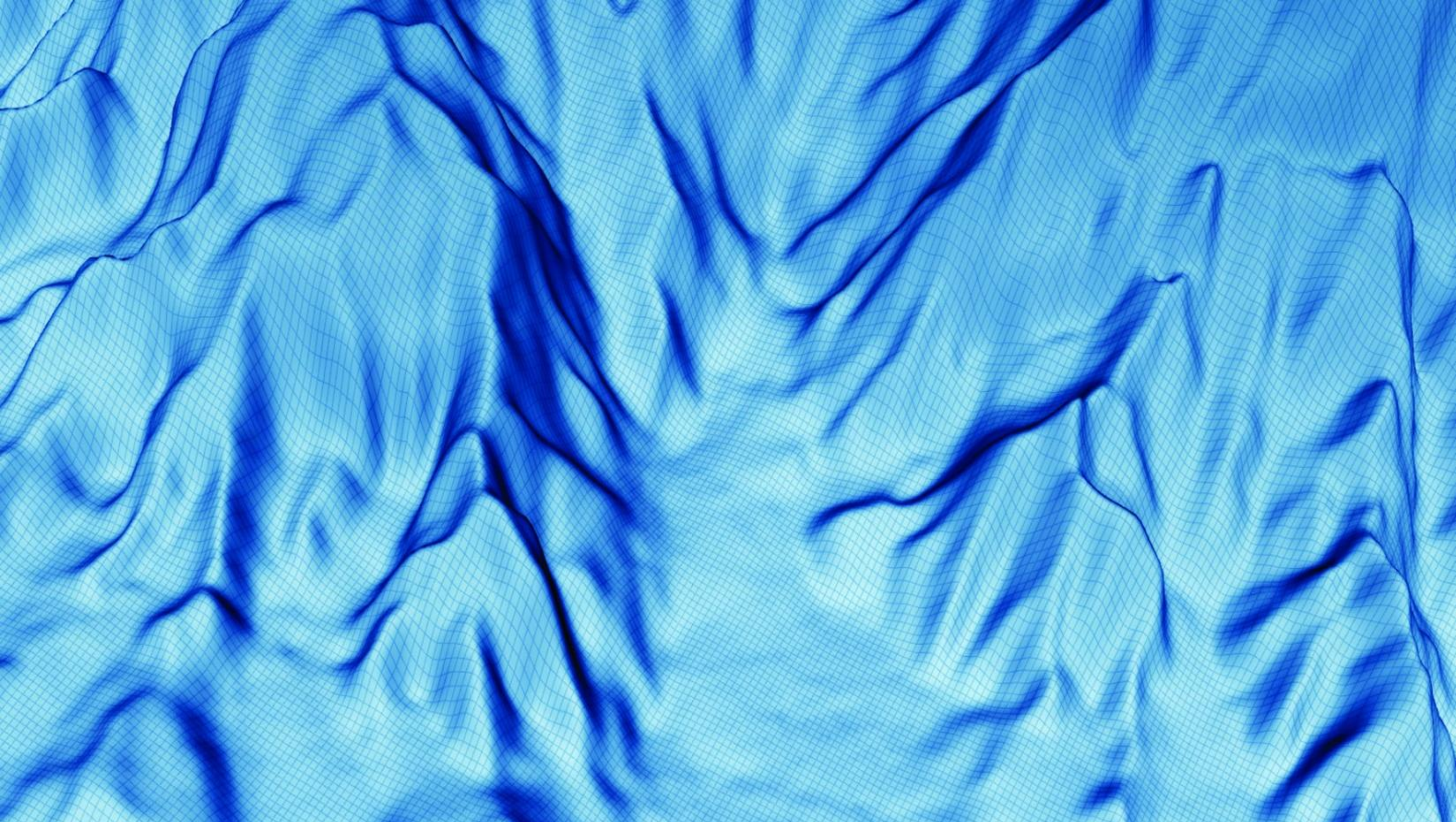


Oil & Gas Tutorials

for Geocap 6.3

Document generated: 19.09.2012

support@geocap.no

Exercises connected to training project Course B1

Introduction

These exercises requires that the specific training project **Course B1** is available as a demo project. It can be distributed in the basic training course for oil and gas or it can be downloaded from Course B1 . Be aware that the downloaded training course is a **zip** file and you have to **unzip** it.

The basic training course will use these exercises as a reference to get a good start working with Geocap in the oil and gas industry. The exercises are also suitable for self training and studying features of the Geocap software.

The primary data in the training project like seismic sections, seismic interpretations, well data, georeferenced image and etopo2 coastline and grid are loaded into the project as described in Importing Data in the user guide and the training tutorial Cultural and Bathymetric Data.

The main goal with these training exercises is then to show the correct use of data; how to visualize data and check quality; how to make the best use of Geocap algorithms and to present presentation graphics on screen and in plotting.

In this section:

The project Course B1

Course B1 is a basic primary training project for oil and gas projects. The data sets are pseudo realistic; i.e. they are from real cases, but transformed to a new place. The size and amount of data are trimmed down in order to make it downloadable. Yet they represent a typical oil and gas case that is suited to demonstrate what Geocap can do with them and how it is done.

For convenience the folders have a prefixed number so they will keep that ordering. A Geocap project has a free structure and the user can have it organized in different ways. A good rule is two have a limited number of top folders and go into the depth when necessary.

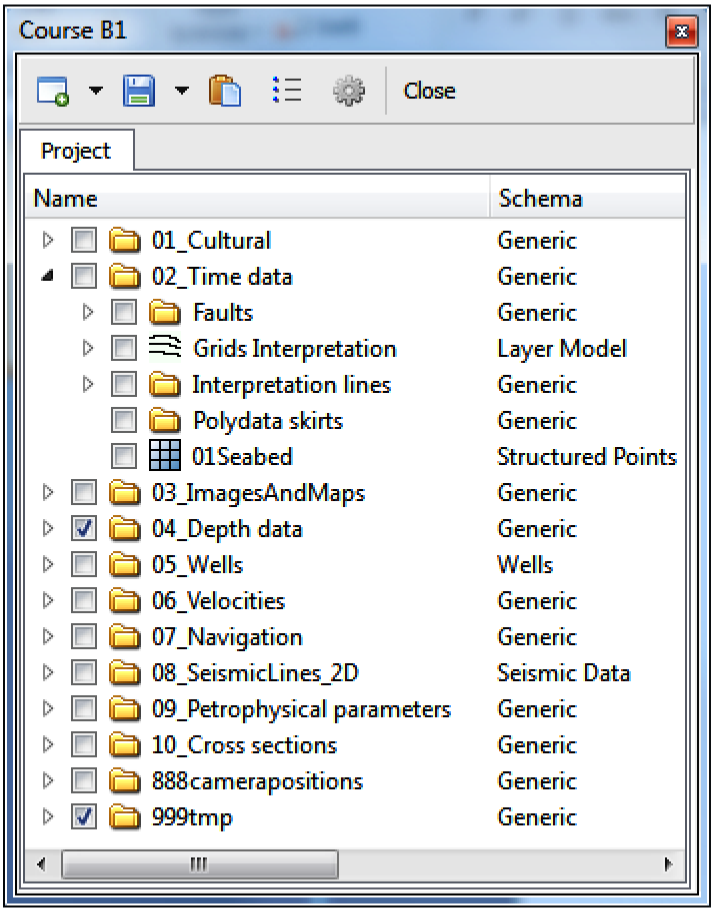
How to work with the project.

The training is organized by a set of exercises using input data from the project or creating data from scratch in some cases. The user is free to work and extend **Course B1** as the only project or creating a new project that gradually becomes more and more similar to the master project. The latter choice require that one can handle two projects simultaneously and has the advantage that the master project remains undisturbed and the exercises refer correct to it.

But the first choice to use the **Course B1** project directly is probably most convenient in a direct exercise situation. Just create parallel folders and data sets whenever needed.

Compatible Geocap release for current Course B1

- Both **Geocap** and **Course B1** are trimmed and upgraded frequently.
- The current version of the downloadable **Course B1** requires the release **Geocap 6.0.11** or later.



Course B1 project for Geocap training

Contents of the downloaded file.

The folder Course B1 contains these folders

- **db** contains the project **Course B1**. Browse in the project by clicking on this folder.
- **plot** contains a Postscript file and a pdf file showing a Geocap plot. Also the saveset for the plot and a logo.
- **workflows** contains the workflow **Seismic and reservoir models - course B1.gwf** that have 11 workflows showing project scenes.

Place the folder structure on your own computer where it suits you. All the workflows are only dependent of the project and will run fine when the project is loaded. The logo is referred to in the saveset of the plot and must be browsed over again in order to run the plot.

01 Generate project window

Project window

The **project window** is an important concept. It defines the area of interest (AOI) of the project. Usually it is equivalent to the **grid window**; i.e. the area of the grid models of the reservoir.

Exercise

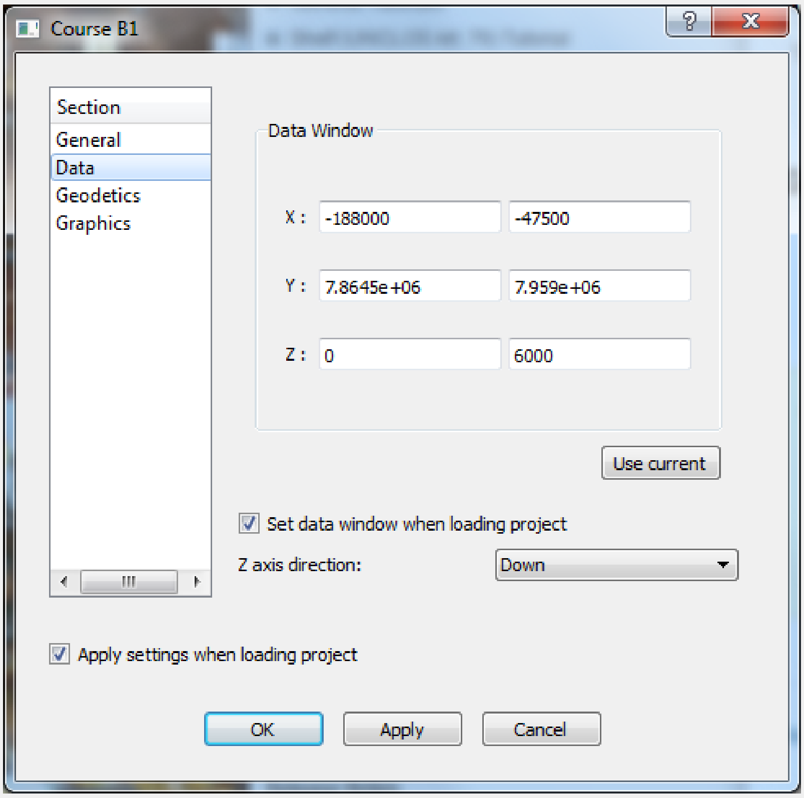
Generate project window

1. Zoom to data on the interpretation lines in the project: **_02_Time data > Interpretation lines**
2. Display the seismic interpretation lines.
3. Draw the windows frame by clicking
4. Set the screen in 2D mode.
5. Display the border line from the project: **_01Cultural > Boundary > border_line**
6. Use **Tools -> Quick Digitizer** to digitize your own border line and save it. F.inst. name it **border_line2**.

The *project window* is usually set in the Project Settings panel under Data
- Click on the tool icon on the project bar.

a. Look at the other options in the **project settings** panel.
Section - Used to change project name.
Data - Project window that will be activated when loading the project.
Geodetics - We use **UTM 51 S** as our projection and **WGS83** as our datum.
Graphics - It is an advantage to erase the screen when loading the project because the no graphics from another area will disturb the current graphics when the system calculates visualization parameters.

Features regarding the project window
- All the reservoir grids in the project are usually gridded within the *project window*.
- The gridding panels have there own window settings that should for convenience coincide with the *project window*.
- If you change the project window and use that as a new *grid window* you cannot mix grids with different grid window in a reservoir model.



Project window set in the Data section in project options

7. The file ***01Cultural > Boundary > window_aoi*** *is the window frame as a data set. Whenever you apply* ***Zoom to Data*** *on this data set the project window will be set again. The *window_aoi* is also used in the workflows for display purposes. Generate the window frame data set.

How to generate the project window data window_aoi
- Do the *shell command* **dra win sav** to display and save the window frame.
- do **mlo ^win** to retrieve the window dataset from its workspace buffer.
- Go to the project and do: **_01Cultural > Boundary > New > Workspace Data > active** to get the active data set into the project.
- Rename it to **window_aoi2** to distinguish it from the one already present.

8. Display the window frame dataset and click on the axes icon on the toolbar to see the coordinates.
9. Acivate the compass icon to see the orientation.

Notes about setting a graphical window

- The *graphical window* determines what are that will be displayed.
- When the project is loaded and the *project window_ is set it is equal to the *graphical window*.
- The *graphical window* can be set interactively by using the *z' key when the screen has focus and is in 2D mode and you draw a rubber band.
- The shell command **win *xmin xmax ymin ymax zmin zmax*** will set set window.
- The project action **Zoom to data | Zoom to folder** sets the window.
- The **project settings** panel can update its data window from the current window.

02 Seismic sections and interpretation lines

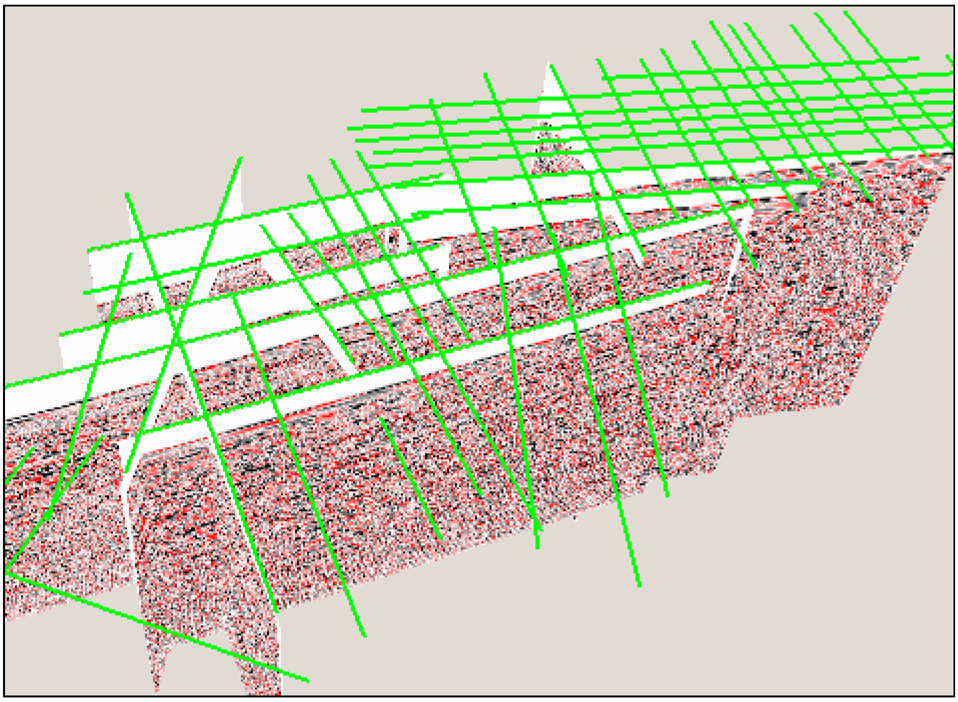
Seismic sections and interpretation lines.

The seismic 2D sections were used to interpret horizon lines and faults. Interpretation lines can in general be work out in Geocaps interpretation system. But in this case they were interpreted from an external interpretation system and imported into Geocap. This is often the case because there are many specialized interpretation systems around.

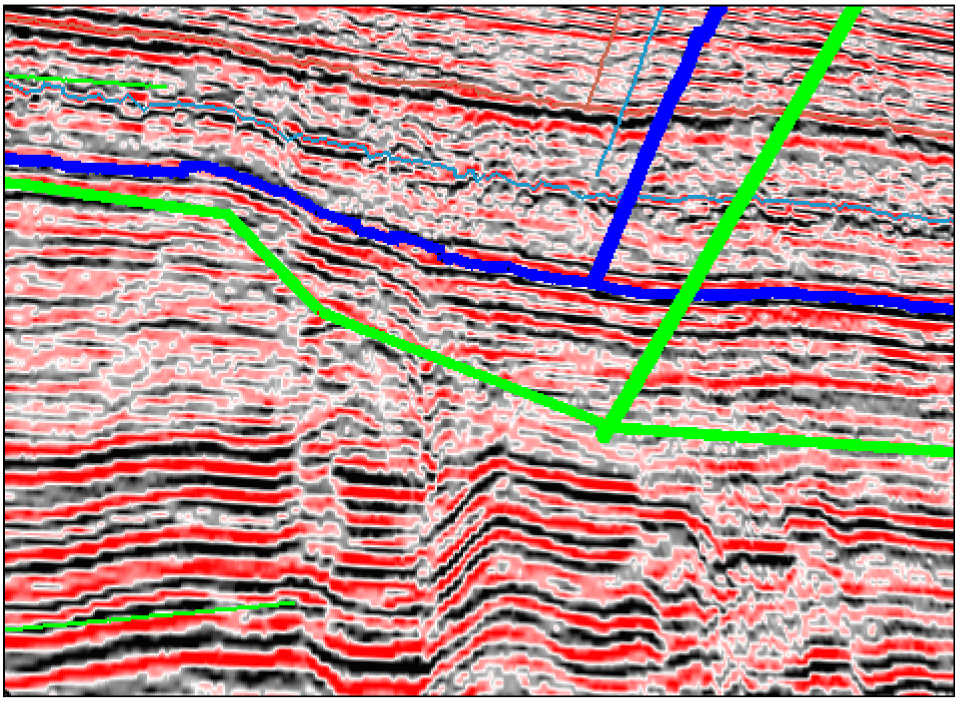
Exercise

Display and compare seismic sections and interpretation lines.

1. Display a seismic 2D section. Try LOD display and ordinary display.
2. Use the **x** key letter to set a focus and rotation point on the seismic.
3. Drop color tables on the section. Use the **Adjust color table range** icon to study color table effects.
4. Display the seismic navigation lines. Use the icon **Locate project object from graphics** to identify a seismic section.
5. Display interpretation lines in various line thickness. Try also point display.
6. Evaluate the interpretation quality. Are the lines *auto tracked* or *digitized*?



Seismic 2D sections and navigation lines



Detail of seismic and interpretation lines

If you want to correct or reinterpret some parts of the interpreted lines they have to be imported into Geocaps interpretation systems; see Importing Interpretation and Seismic Interpretation. Not all seismic 2D sections are in the course project, but they are present in the Tutorial data1 which can be downloaded.

03 Gridding

Gridding and surface modeling.

Grid models are essential in modeling reservoirs. Good grid models require good input and well tuned algorithms.

In this exercise we will use as input seismic interpretation lines and closed stick faults. There are seven interpretation sets including the seabed. The reservoir has six horizons. The bottom five have fault lines as input while the top one don't use any faults.

Exercise

Grid a time surface from seismic interpretation and no faults.

1. Display interpretation lines from **02_Time data > Interpretation lines > 02BasePliocene**.
2. Apply the command **Simple points and lines gridding**
3. Click **Update window** to get the *project window* as the *grid window*.
4. Set x and y increment to 500.

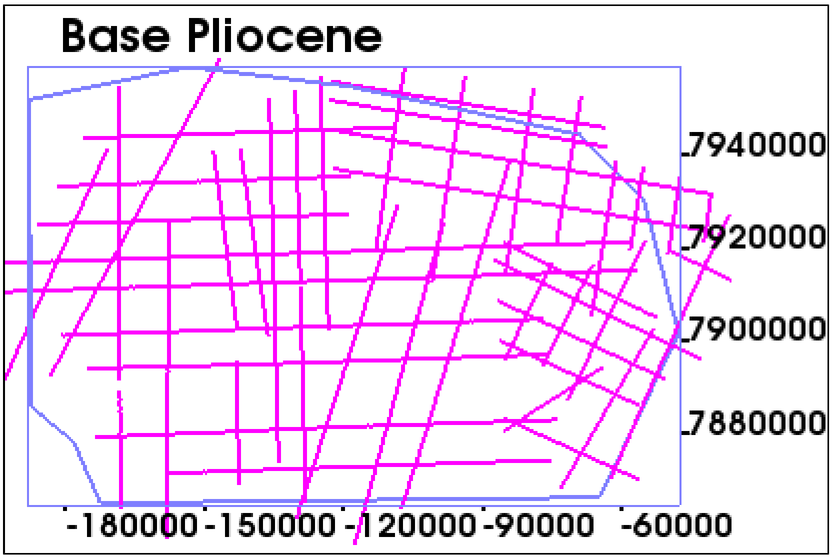
Grid window for Course B1

- xmin = -188000, xmax = -47500; ymin = 7864500, ymax = 7959000
- Using grid increment 500 gives as size of grid: **number in rows: 282 , number in columns: 190**
- It is a relatively small grid, but useful in a course and the size of the project is kept reasonable low.

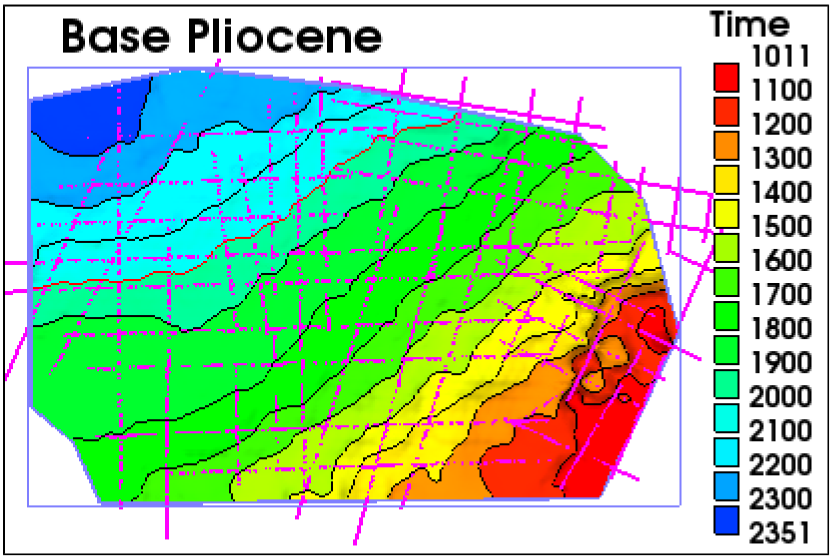
5. Check in **Apply reference grid or outline** and browse in: **01_Cultural > Boundary border_line**
6. Use gridding algorithm **Parabolic** (default).
7. Check in for **Use two steps gridding**. Grid increment will be automatic calculated if blank.
8. Set **moving average** as gridding algorithm for 1. step.
9. Click **Execute**. The result will be written to the same folder.
10. Click **OK** and save the panel with all parameters.
11. Create a new folder for interpretation grids and cut/paste the result grid into this folder.
12. Display the result as shown below.

Displaying color legend

- The color legend is often displayed with the shell command **cco**. In the example below the following procedure was used:
- Screen in 2D mode. Click a cursor point at lower right.
- Use the shell command: **cco rev cur col bla tit Time**. The **cur** argument refers to the cursor position. Look at the reference manual if necessary.



Base Plioscene interpretation lines and border line



Base Plioscene grid

Grid menus using faults

- The grid menu when faults are included is **Gridding points, lines and faults**. This menu can also be used when there is no faults, but the one we used is a simpler alternative.

Exercise

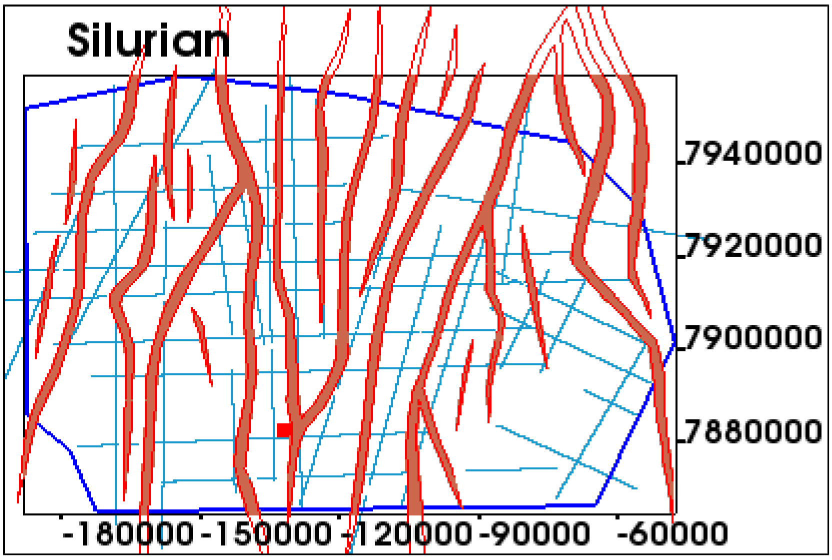
Grid a time surface from seismic interpretation and closed stick faults.

1. Display **02_Time data > Faults > 07SilurianFaultInt_Polygons**. This is how the fault polygons comes from **Kingdom** . They must be edited.
2. Use **Edit points and lines** to remove the *blobs*. See Gridding seismic interpretation with faults.
3. The correct edited fault lines are in **02_Time data > Faults > CorrectedClosedFaults**. Use this if you don't have time to complete the editing.

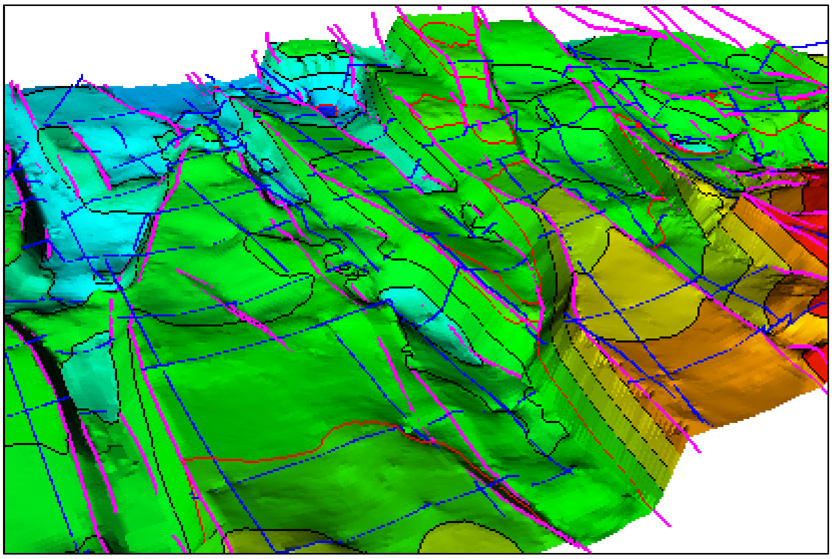
Tips when editing fault lines with *blobs*

- To remove the *blobs* just delete the two extra points in the *blobs*.

4. Apply the command **Gridding > Gridding points, lines and faults** and activate it on the proper interpretation lines.
5. Don't alter type of input data (under development). Use default: **Single points**.
6. Click **Use active window** to get the *project window* into the *grid window*.
7. Set the x and y *increment* to 500. Click **Adjust to Increment** to secure that the grid window is adjusted to the increments. If the grid window changes update the *project window* in **Project Settings**. It is an recommended that the *project window* and *grid window* are equal and adjusted to the grid increments.
8. Click **Calculate** to get number of elements in rows and columns: **282 x 190**.
9. Go to the **Fault Input** tab.
10. Check in **Closed stick faults** and browse in: **02_Time data > Faults > CorrectedClosedFaults**
11. Clik **Test** at bottom to do the blanking test. The fault polygon should be filled inside. No lines outside.
12. Go to the **Grid options** tab.
13. Check in **Use reference grid or outline** and browse in: **01_Cultural > Boundary border_line**.
14. Check in **Use two gridding steps**.
15. Go to to **Result Grid** tab.
16. Set the saving option to **Save in project**. The result grid will be saved in the same folder as input unless another folder is browsed in.
17. Click **Execute**
18. A default display of the result grid appears. Delete this and display using **Display Contours**.
19. The faults lines have been assigned z values and are located in workspace as **closedGenFaults**. Transfer them to the project using: ***folder* > New > Workspace Data > closeGenFaults**.
20. Click **OK** to save the grid panel.
21. If time, repeat the procedure for all faulted horizons. If not use the data in the course project.



Silurian interpretation lines and faults


Silurian grid with faults and input lines

04 Layer models and cross sections

Layer models and cross section.

A layer model is a collection into a layer folder of grids which have the same grid position and resolution. The layer folder is of type **Generic** and has the Schema **Layer model**.

Exercise

Create a layer model.

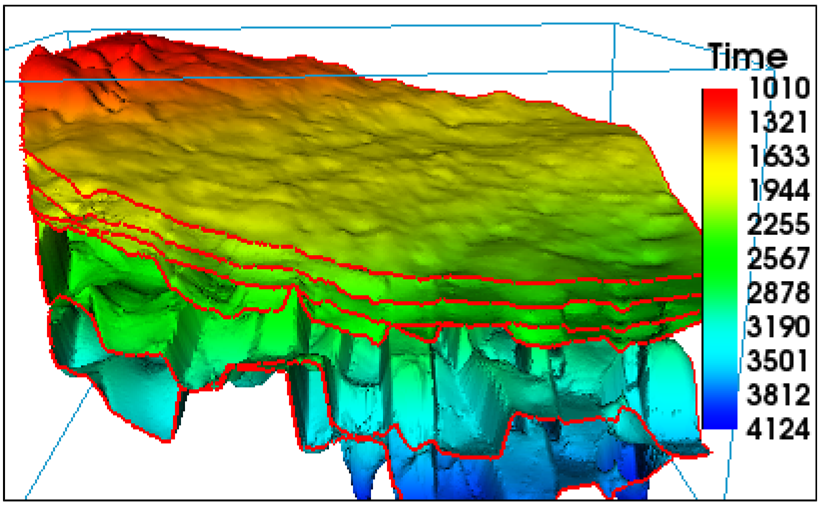
1. Create a *generic* folder and set the schema to **Layer Model**.
2. Copy all time grids into this folder and secure the correct ordering from top to bottom by placing a prefix number 01 02 03 ... etc.
3. Perform **Zoom to Folder** on the time layer model and draw the window frame and axes.
4. Display all the layers in random colors.
5. Create a map command that display the color using color range from 1000 to 4200.

A layer model consisting of interpretation grids may have grids that go deeper that the next grid in sequence. We then have to check the layer model and if necessary do **truncation**.

Exercise

Check and truncate the layer model.

1. Visualize the layers one by one to see if there is any overlap.
2. Activate upon the layer model the command **Check and truncate layers from top and downwards**.
3. We run on defaults; therefore click on **Execute** to perform truncation from top and downwards.



Layers mapped with the same color range

Having a layer model it is convenient to make cross sections.

Exercise

Create cross sections.

1. Highlight layer model folder and read the command **Information about the layer model**.
2. Activate the command **Initialize the layer model into Workspace**. The layer model grids are read into workspace for fast retrieval.

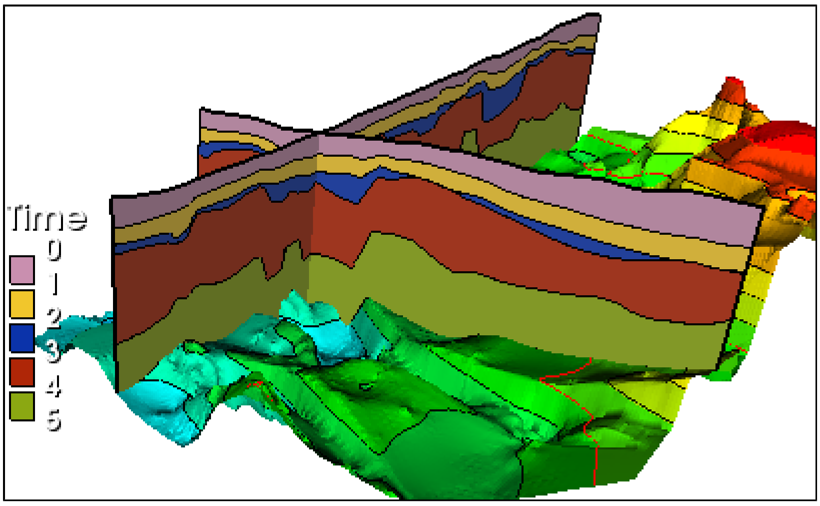
Layer model grids

- When the layer model grids are in workspace it is quick to perform cross sections and other calculation.
- If a another layer model shall be studied just activate **Initialize the layer model into Workspace** on the new layer model.
- Cross sections will be performed on the layer models grids in workspace.

3. Create a new **Generic** folder and give it the name *Cross sections* and set the schema to **Cross sections**. Cross section commands will now be visible on this folder.
4. Do on the cross section folder **Display fence type cross section**. It will display and the fence data is saved in the cross section folder.
5. Erase the screen and open the cross section folder perform on the fence data **Vertical redisplay from cross section data**.
6. Activate on the cross section folder **Legend for cross section**.
7. Try the other cross section commands.

Setting cursor for cross section action

- When doing cross sections it is smart to have the bottom layer displayed.
- Place a cursor position on the bottom layer using key letter **y** when in 3D mode, or just click when in 2D mode.
- Cross section between two cursor points: Set two cursor marks. Geocap remember them even if only the latest is shown.



Cross section study of the time layer model

Another option for fast cross sections is to generate a solid **layer cube**.

Exercise

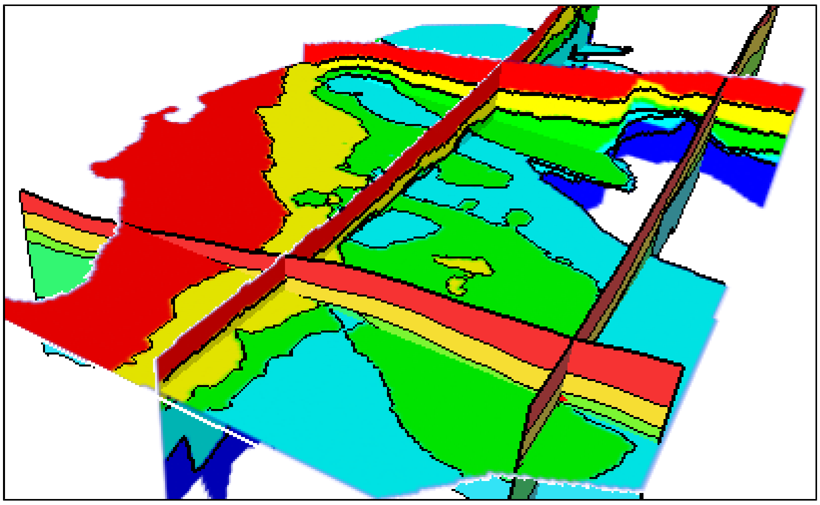
Create a layer cube.

1. Use **Zoom to Folder** on the layer model to get the overall layer window.
2. Highlight layer model folder and activate the command **Cube generation of all layers**.
3. Click **Use Active Window** to set the cube window.
4. Set the cube grid increments to **500 500 20** to get **282 x 190 x 20** grid nodes.
5. Click **Execute**. The cube is saved in workspace **cubegrid**.
6. Get the cube gid into the project under the Time folder and rename the cube to **time_cubegrid**.
7. Apply the **Manipulator** to display the layers in the cube in a fast way in all directions.

Setting cursor for cross section action

- The manipulator is for a cube is activated by ***cube*** **> Utilities > Manipulator cube display**.

8. Run the manipulator in all directions on the cube.
9. Display a layer grid to verify that the cross sections from the cube have correct extensions.



Cross sections displayed from a layer cube

05 Stacking velocities

Stacking velocities.

The **stacking velocities** are collected along the seismic lines and processed to show *time* versus *velocity* of the seismic reflectors. The data is used to generate a **time velocity cube**.

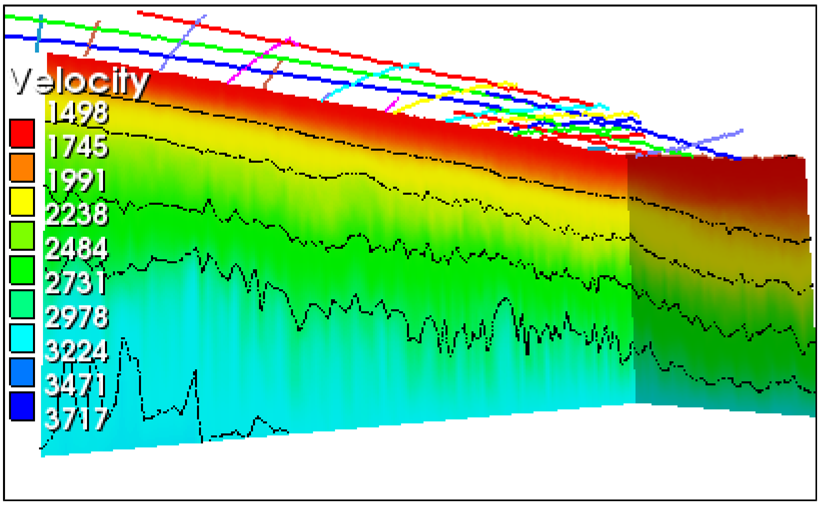
Before using then uncritical it is common to display and check the quality and extent to see how deep the cube should go.

The stacking velocities are located as in **06_Velocities** as *Survey03* and *Survey04*. There are more surveys in the dataset Tutorial data1 , but in this project we only need 3 and 4.

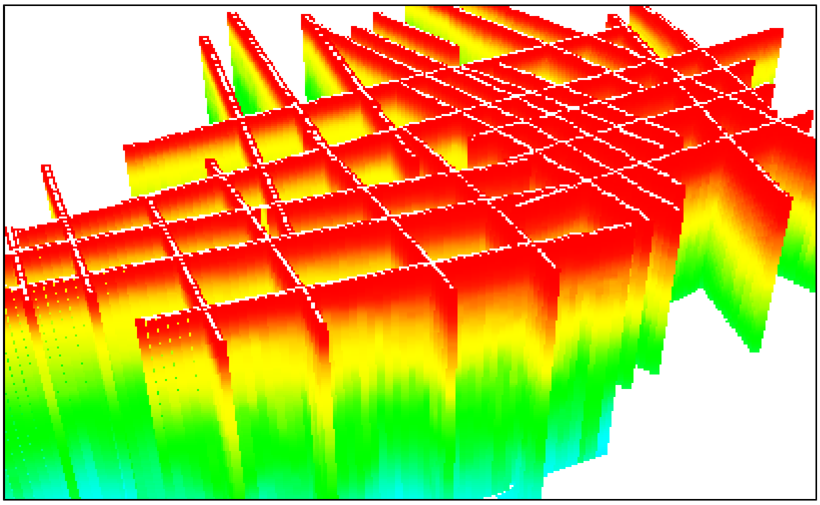
Exercise

Display stacking velocity.

1. Do **Zoom to Folder** on *Survey03*.
2. Open up folder *Survey03*; click on a stacking velocity dataset.
3. Perform **Map Data**. Zoom in and study the display. The lines are mapped.
4. Erase and perform **Make bands of velocity an map**. Zoom in and study the display. It is solid.
5. Try different display commands on the various velocity data sets.
6. Go to the **Command** panel. Use **Display survey lines**. Transfer this command to the **Toolbar**.
7. Highlight folder *Survey03* and apply **Display survey lines** from the **Toolbar**.



Survey lines and stacking velocities


Stacking velocities collected and mapped

Gridding stacking velocities requires that all lines are collected into one file.

Exercise

Appending stacking velocity into one file.

1. Highlight folder *Survey03* and apply **Append all lines in a folder** from the **Command** section under **Schema Commands**.
2. The result is *appendedData* in workspace. Create an temporary folder and put it there. Rename it to *appendedData1*.
3. Perform **Append all lines in a folder** on *Survey04*.
4. Put *appendedData* into the temporary folder.
5. Perform **Append all lines in a folder** on the temporary folder. The final result is now *appendedData* in workspace.
6. Transfer *appendedData* in workspace to the project under *06_Velocities* and rename it *all_lines*.
7. Delete the temporary folder.

Note on appending data

- We did not use **Append all lines in a folder** on the *06_Velocity* folder because there are data in it that should not be appended.
- Appending could have been done quicker with the *shell command* **mhi/mlo *data* app**. The menu uses these commands.
- We could have avoided the temporary folder if we appended once; then edited the **Append all lines in a folder** on the next survey and clicked on **Prestep** and removed the command **dhi appendedData** which means delete workspace data *appendedData*. Then we can continue to append without deleting in the *prestep*. The final result is *appendedData* which we can transfer to the project.

06 Velocity cubes

Velocity cubes.

Velocity cubes are generated from **stacking velocities** and **check shot** wells. The dimension of the cube is in the time domain, while the contents: i.e the cube nodes are velocity values. The cube is thus also called a time velocity cube. The purpose of the velocity cube is to have a fast and reliable method for **depth conversion**.

There are several methods of generating the velocity cube:

Methods for cube generation

1. An overall gridding method that transfers the stacking velocities into the cube by one master command. This method is used in Geocaps cube gridding panel.
2. Gridding individual cube layers and insert them into the receiving cube. This method can be used by users that will script up sophisticated logic for detailed control and quality.

The stacking velocities were collected into one dataset called *all_lines* in the *06_Velocity* folder.

Exercise

Digitize a border line for the cube.

Border line for the cube

- We need a border line for the interior of the cube. This line should go a little outside _border_line_used for the grids.
- The cube size should extent the time grids a little bit in order to get no edge effects when depth converting.

1. Put the screen in 2D mode.
2. Digitize a closed line just outside *border_line*.
3. Transfer it to the **Boundary** folder and name it *cube_border*.
4. Set the screen back into 3D mode.
5. Perform **Zoom to Data** on *window_aoi* (or use one of the grids) to secure the grid window is back.

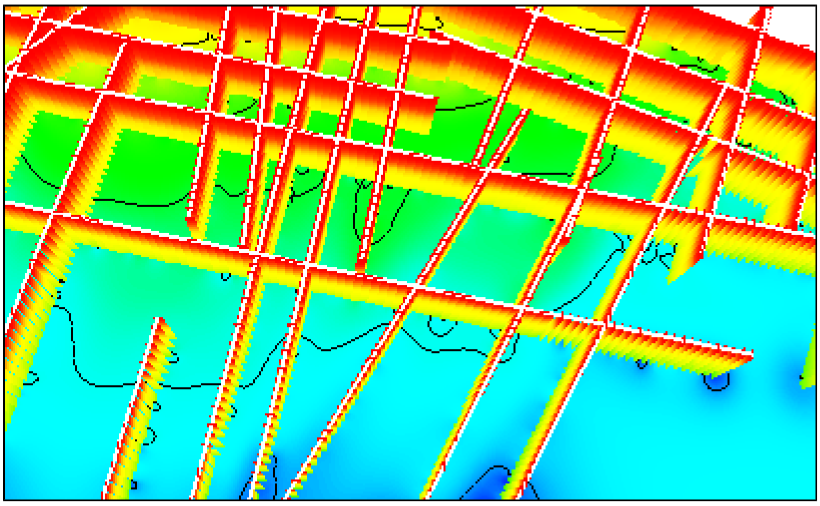
Exercise

Grid stacking velocity into a cube.

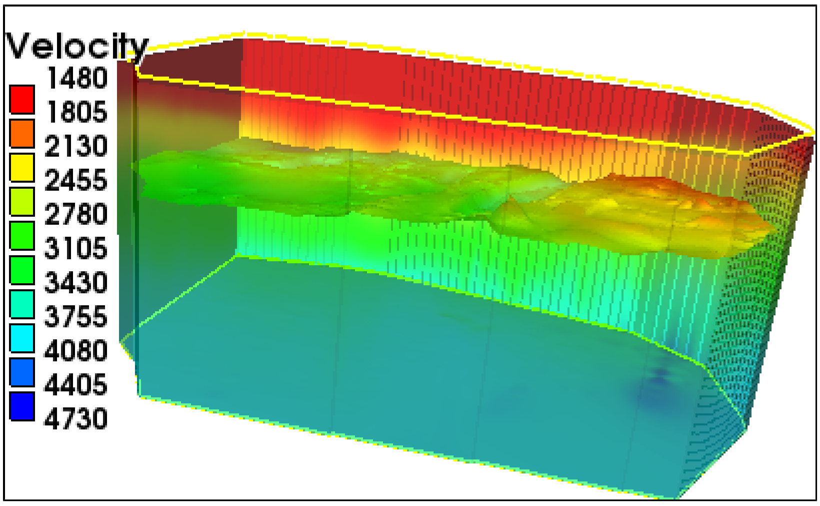
1. Highlight the stacking velocities *all_lines* in the **Velocity** folder. Activate **Cube gridding from stacking velocities**.
2. Select correct attribute by clicking **Set**. The combobox should shift to **Scalar**. Sometimes the velocity is the field data **velocity**, but in this case it is in the scalar part.
3. Click **Use Active Window** to set the cube window.
4. Set **zmin to 0 and zmax to 6000** which will be the size of the cube in the z direction.
5. Set the cube increments to **500 500 50** which is an acceptable resolution in our case.
6. Click **Expand** to expand the cube window a little bit to avoid edge effects when depth converting.
7. In our case the size of cube is now **286 194 121**.
8. Go to the **Grid Options** tab.
9. Click **Use reference grid or outline** and browse in *cube_border*.
10. Go to the **Cube Options** tab.
11. Click **Apply base grid** and browse in *01Seabed_time from the _06_Velocity* folder.

> **Note on the base grid *01Seabed_time***
> - A base grid tells where the values from the stacking velocities start. Above the base grid it is only water and the values are the water velocity.
> - The ***seabed in time*** is the natural base grid and *01Seabed_time* was gridded from time interpretation lines.
> - The grid window and x y grid increments for *01Seabed_time* must be exactly equal to the cube.
> - You can do that gridding as an exercise later on. For the moment use the one in the course project.

12. Go to the **Result Cube** tab. Observe that the result cube by default is written back to the same folder as the input.
13. Go to the **Testing** tab.
14. Drag the slider bar and set the z value to around 2500.
15. Click **Make velocity grid** to perform a test for that level.
16. If everything is OK click **Execute** to generate the velocity cube. It will be saved as *all_lines time_velocity*.



Testing the cube gridding



Velocity cube displayed in transparent mode wit one time layer inside

Check shot data
The velocity cube will usually be checked against **check shot** wells; i.e. wells that have correct time and depth values downwards.
This project does not have appropriate check shot wells for the area at the moment so here is only the technical procedure.

- Use the command **Deviation velocity cube and wells** to see the difference between the cube layers and the crossing with the wells.

Exercise

Using check shot data to adjust the velocity cube.

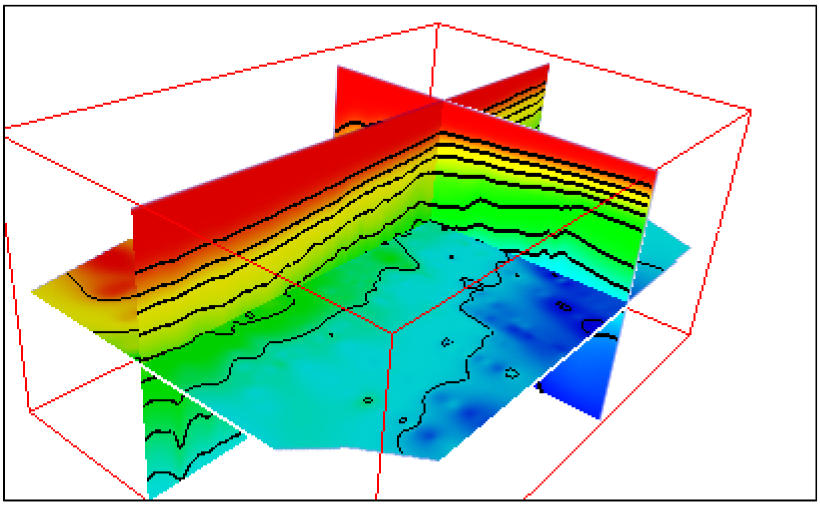
1. Highlight *all_lines time_velocity* and activate **Utilities > Adjust velocity cube to checkshots**.
2. Browse in the check shot data.
3. Specify the correct format of check shot data.
4. Click **Execute** to perform the adjustment.

Exercise

Displaying the velocity cube with the manipulator.

There are many ways to display velocity cube data. They are all visible on cubes. Try them out and be familiar with them. The primary one to learn is to use the **manipulator**.

1. Highlight *all_lines time_velocity* and activate **Utilities > Manipulator cube display**.
2. Draw the manipulator in all directions by checking X Y or Z.



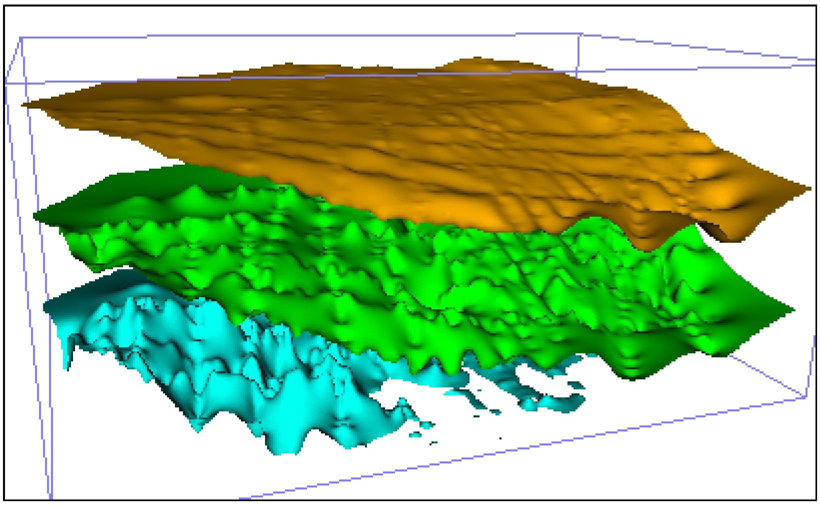
Using the manipulator to see cube planes

Exercise

Displaying different iso contours from the cube.

Iso contours is a direct way to see the interior of the cube. If too noisy one should consider smoothing the cube.

1. Highlight *all_lines time_velocity* and activate **Cube contours**.
2. Draw the contours for the values **2000 3000 3800**
3. Evaluate if it is noisy. In that case smooth the cube.
4. Activate **Smoothing Cube** and perform smoothing and redisplay the iso contours.



Checking the cube by displaying cube contours

07 Depth conversion

Depth conversion.

Depth conversion will transfer the time grids to depth grids. In that process we use the velocity cube in the algorithm.
The time grids are *probed* into the velocity cube and the transition to depth grids occurs by sampling velocity values from the cube onto the tme grids.
The standard conversion formula is then used: **depth = time x velocity**.

Exercise

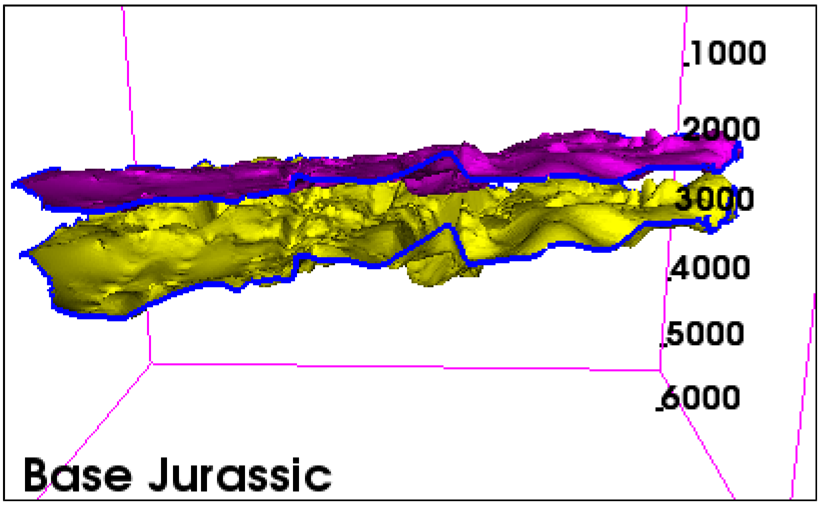
Transfer time grids into depth grids.

1. Copy the time grid folder and paste it into the *Depth data* folder.
2. Activate on the pasted grid folder **Utilities > Depth conversion of time grids in folder using velocity cube**.
3. Read the info and browse in the velocity cube.
4. Click **Execute** to perform the depth conversion.

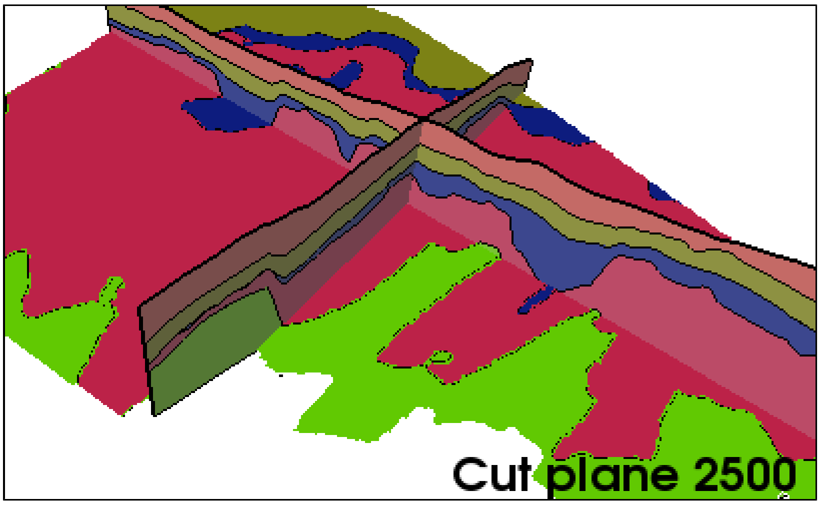
Exercise

Display depth grids.

1. Display the same time grid and depth grid to see how it is shifted.
2. Activate **Initialize layer model into Workspace** on the depth grid folder.
3. Make a new cross section folder and perform cross sections in all directions.



Base Jurassic is deeper when transferred to depth



Cross section through layer model of depth grids

08 Reservoir and volume calculations

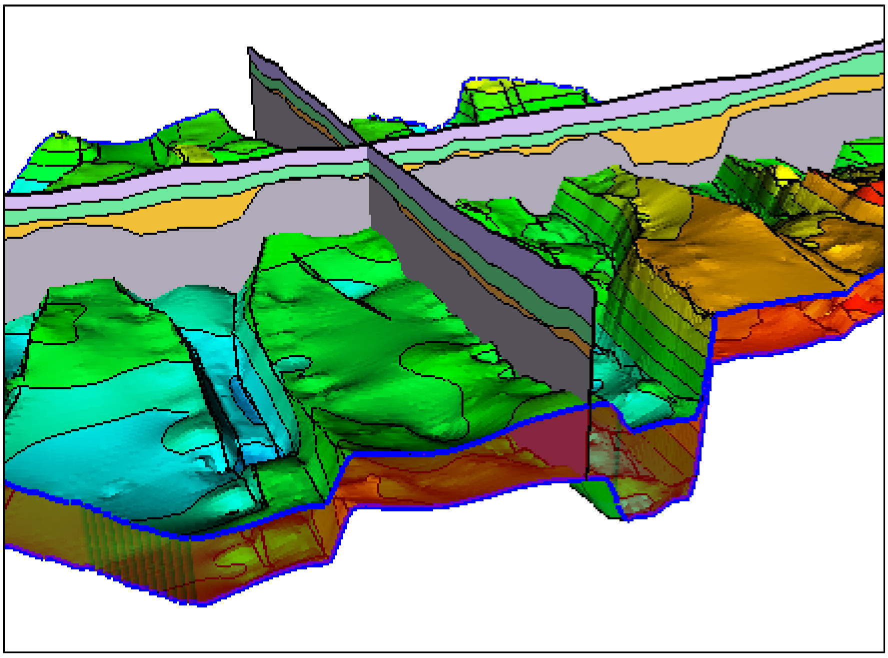
Reservoir models

In order to calculate **HCPV** (i.e. hydrocarbon pore volume) for the reservoir there has to be a **reservoir model**. This is a folder with reservoir horizons in **layer model** and folders with petrophysical parameter grids.

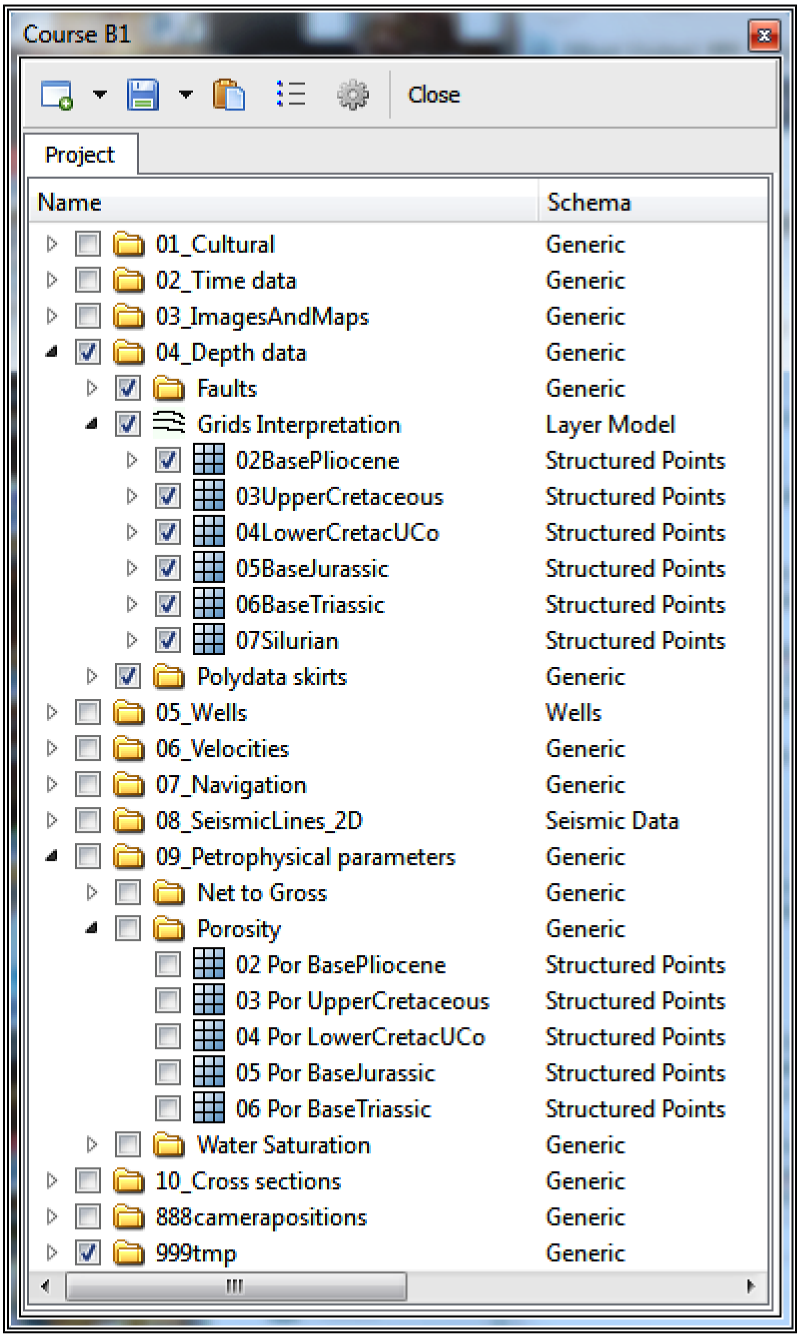
The parameter grids are connected to the zone between two horizons and the zones in the **layer model** start with the first horizon. There are seven horizons and six zones; therefore the last horizon will have no petrophysical parameter grids associated.

Petrophysical parameter grids
The petrophysical parameter grids in the volumetric model are random generated between 0.1 and 0.6 approximately. That corresponds to realistic value domains. In real cases the petrophysical parameter grids are gridded from well data, f.inst. using the gridding panel **Simple points and lines gridding**.

The parameter grids should be organized as shown in the folder *09_Petrophysical parameters*.



Layer model and cross sections



Folders showing the layer model and petrophysical folder with parameter grids

Exercise

Calculate hydrocarbon pore volume within a license.

1. Right click on ***Depth data > Grids interpretation*** and activate **Volume layer model calculation**.
2. Set **Type of volume to be calculated** to ***Between plane1 and plane2***.
3. Check in **1: contact** and write in the level value **2700** (suitable for this project). Also called **top** contact.
4. Check in **2: contact** and write in the level value **3800** (suitable for this project). Also called **bottom** contact.

> ⚠
> - If there is only one contact plane you just check in **1: contact**.
> - There are two types of calculation:
>
> a. From top reservoir to **Top contact**.
> b. Between **Top contact** and **Bottom contact**.

5. Check in **Apply folder or single area grid or outline for licenses or sub areas**.
6. Browse in ***01_Cultural Licenses > License 07-18***.
7. Go to page **Parameters**.
8. Check in boxes for petrophysical parameters and browse in the corresponding folders.
9. Check in **Expansion oil** and set the value to **1.2**.
10. Check in **Recovery factor oil** and set the value to **0.4**.
11. Click **Execute** to start calculation.
12. Go to **Result** page.
13. Click on **Volume report**.

> ⚠
> - Goes default to your home area on disk and is placed in folder. The names are made of parameters set and a time stamp.
> - The folder has a typical name: ***Volumetric reports_Grids Interpretation_Wed Nov 16 2011***.
> - The report has a typical name: ***VR_Grids Interpretation_2700_3800_Licens 07-18_b_p_n_w_Wed Nov 16 08_55_46 2011***.

14. Take the volume report into a editor and read it.

Volumetric report

```
Project: Course B1
Volumetric report for horizons in folder: Grids Interpretation.
Calculated and saved at: Wed Nov 16 08_55_46 2011

Parameters present and used:
Type of volume to be calculated: Between plane1 and plane2
First horizon used: 02BasePliocene
Last horizon used: 07Silurian

Contact plane 1: 2700
Contact plane 2: 3800

License or sub area: License 07-18

Porosity folder: Porosity
Net to Gross folder: Net to Gross
Water saturation folder: Water Saturation

Expansion value oil: 1.2
Recovery factor oil: 0.4

Units used in report table: Cubic km (10^9)

 Zone                 Bulk          Pore          HCPV          InPlace       Recoverable
 02BasePliocene       57.859400     33.751980     10.205310     12.246370     4.898550
 03UpperCretaceous    446.479170    174.687160    41.260900     49.513080     19.805230
 04LowerCretacUCo     794.343640    384.093490    79.636180     95.563410     38.225360
 05BaseJurassic       4603.565830   1667.313830   563.336510    676.003810    270.401530
 06BaseTriassic       1299.398520   542.952390    85.428700     102.514440    41.005780
 Total                7201.646570   2802.798850   779.867600    935.841120    374.336450
```

As one can see the names are generated by a simple system created by parameters used and a time stamp. For the folder name, the time stamp follows the day, while for the report itself, the time stamp is the actual day time down to seconds so the name will be unique. The ***b_p_n_w*** *stands for* ***_bulk porosity nettogross watersaturation*** to show that these parameters were present in the calculation.

Display features

Exercise

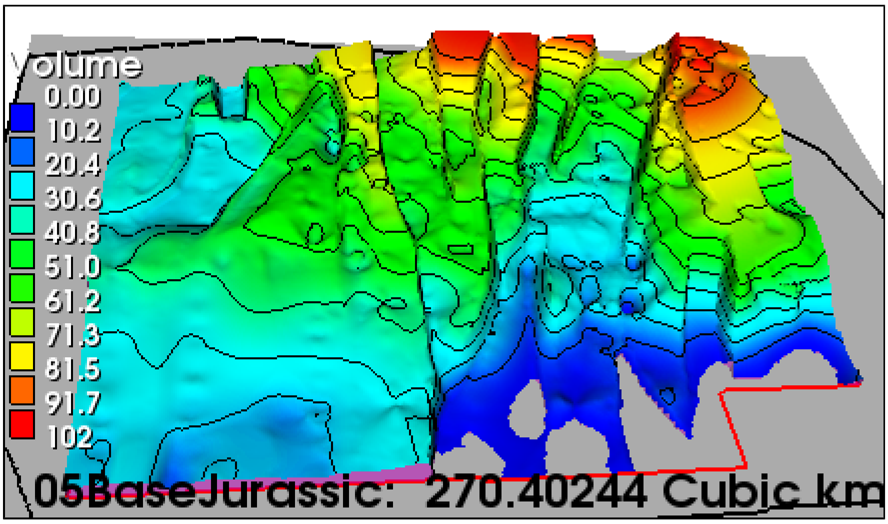
Display recoverable hydrocarbon volume within a license.

1. Click on **Display** to display volumetric results.
2. Select zone 4.
3. Set option to ***As single volume***.
4. Click **Display volume or contact area**.
5. Study the display and explain the map.

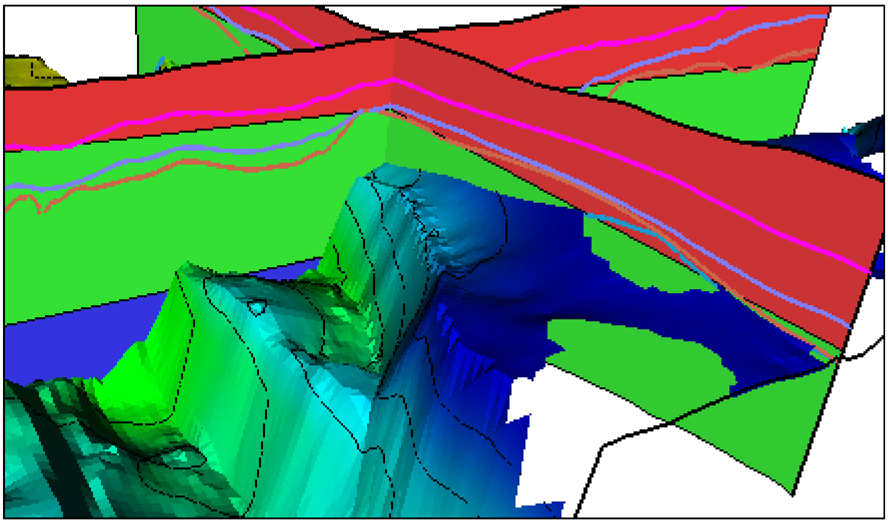
Exercise

Display hydrocarbon volume and cross sections.

1. Click on **Display** to display volumetric results.
2. Select zone 3.
3. Set option to ***On bottom of horizon***.
4. Click **Display volume or contact area**.
5. Make a cursor position with the **y** key.
6. Click **Get cursor values**.
7. Click **X direction**.
8. Click **Perform cross section from position values**.
9. Study the display and explain the map.



Base Jurassic recoverable volume as single volume grid



Base Jurassic draped on Base Triassic and cross sections

Exercise

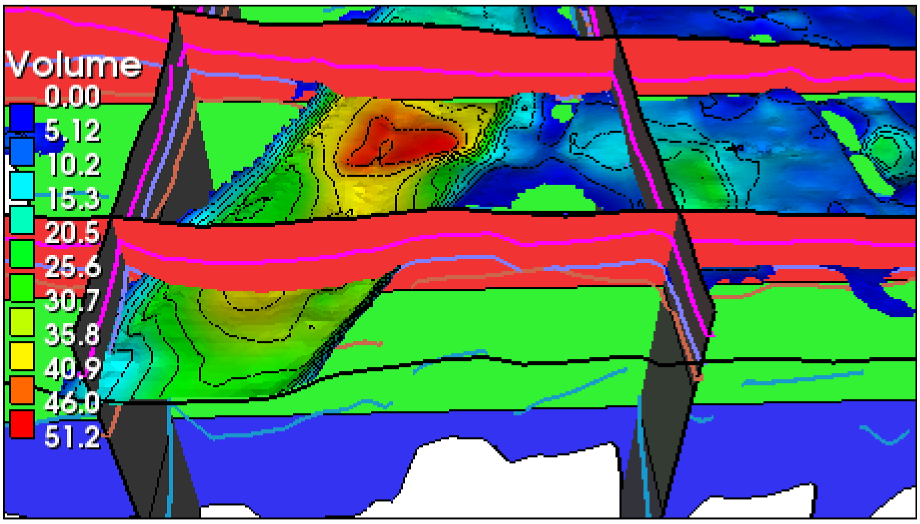
Display hydrocarbon volume and fence diagram.

1. Click on **Display** to display volumetric results.
2. Select zone 4.
3. Set option to ***On top of horizon***.
4. Click **Display volume or contact area**.
5. Click **Fence diagram cross sections**.
6. Study the display to and explain the map.

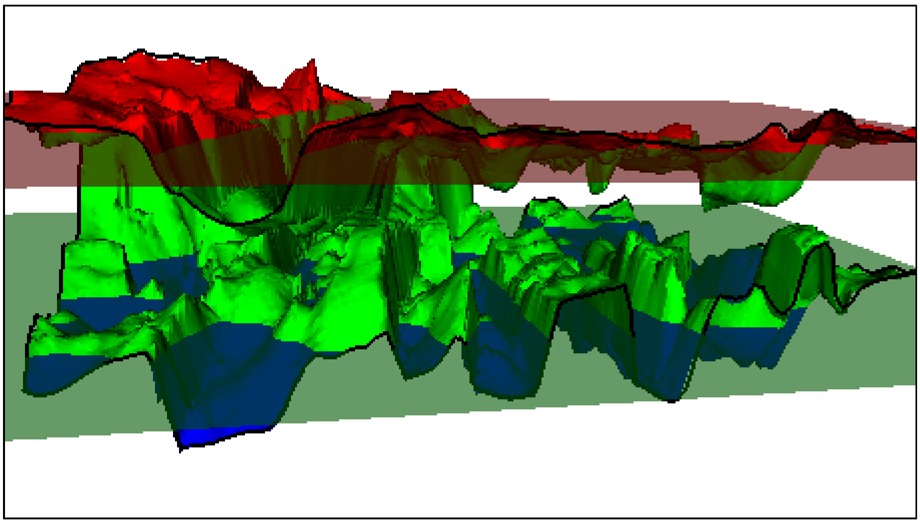
Exercise

Display recoverable hydrocarbon volume within a license.

1. Click on **Display** to display volumetric results.
2. Select zone 4.
3. Set option to ***On top of horizon***.
4. Check in **Display contact area (on zone top only)**.
5. Click **Display volume or contact area**.
6. Check off **Erase screen**.
7. Select zone 5.
8. Click **Display volume or contact area**.
9. Go to the **Volume** page and display the contact planes.
10. Study the display to and explain the map.



Volume draped on Base Jurassic and fence diagram



Base Triassic and Silurian and contact planes

Advanced display of volumetric results

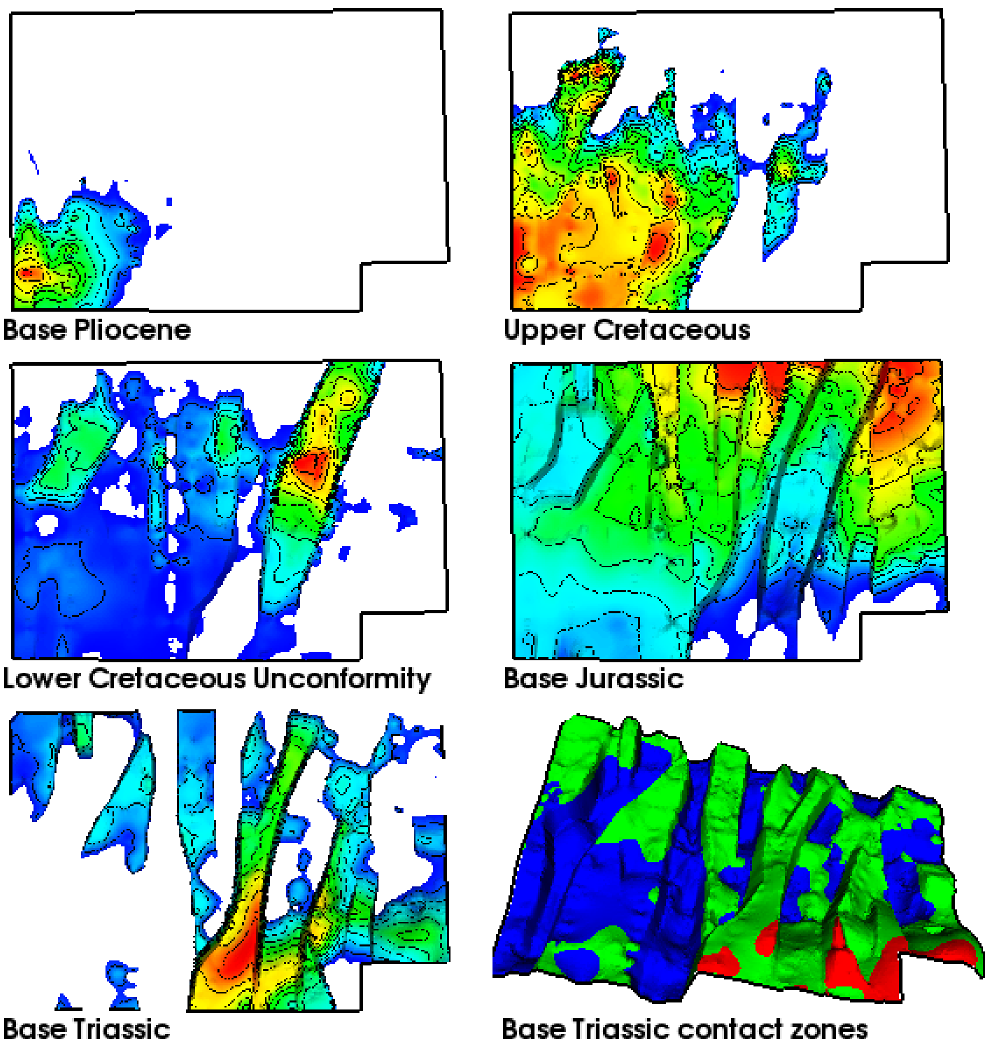
Here is a viewport presentation of all volume grids draped on the top horizon for the zones. Try to reproduce it by following the steps.

Exercise

Display the volume grids for all zones in the layer model.

1. Click on the viewport icon on the ***toolbar*** and open the viewport menu.
2. Check in **Connect all viewports**.
3. Create 2 x 3 viewports. You now have 6 connected viewports.
4. Click on viewport at upper left to get that viewport frame in red.
5. Set all parameters correct in the volume menu and **Execute** it.
6. Go to the **Display** page.
7. Select zone 1. Select **On top of horizon**. (All selections will do, and one must be consequent).
8. Click **Display volume or contact area**.
9. Go to the shell and type the text: **tx2 .7 lle col bla txt "Base Pliocene"**. (2D text at lower left; see the ref. manual).
10. Click on the next viewport.
11. Select zone 2 and repeat the previous steps for zone 2.
12. Repeat the steps for zone 3, 4 and 5.
13. Put the viewports in 2D mode to see it from above.
14. To display the lower right viewport (it is number 2) in 3D mode do as follows:
15. Go to the command shell and type: **vie 2 con off**.
16. Check in **Display contact area (on zone top horizon only)**.
17. Select zone 5.
18. Click **Display volume or contact area**.
19. Make a proper size of the screen and apply ***generic folder*** **>New > Screenshot** to save the picture.

Viewport display of volume maps draped on the top horizon for each zone.



*Volume display for all zones in the training project **Course B1***

The procedure for making the viewport display is described as many interactive steps. It is however possible to set up a workflow to do it all in one action.

The documentation for volumetric calculation is located at Volumetric Calculations. Near the end of the chapter is a larger version of the viewport display.

09 Volume calculation for grids

Volume calculation for one or two grids.

Volume calculation for one grid and an isopach or between two grids and optionally two contact planes does not require a *layer model* and can be done quite easy with a dedicated menu called **Volume calculation** that is directly visible on a grid. The volume calculation operates with singles values for the petrophysical parameters and singles values or grids for contact planes.

Below is a sample task to illustrate volume calculation.

Exercise

Calculate HCPV volume for a zone.

We want to calculate HCPV volume between *Upper Cretaceous* and *Base Jurassic* in the layer folder for depth grids.
The volume will also be between the contact planes 2500 and 2900 and within the license area.

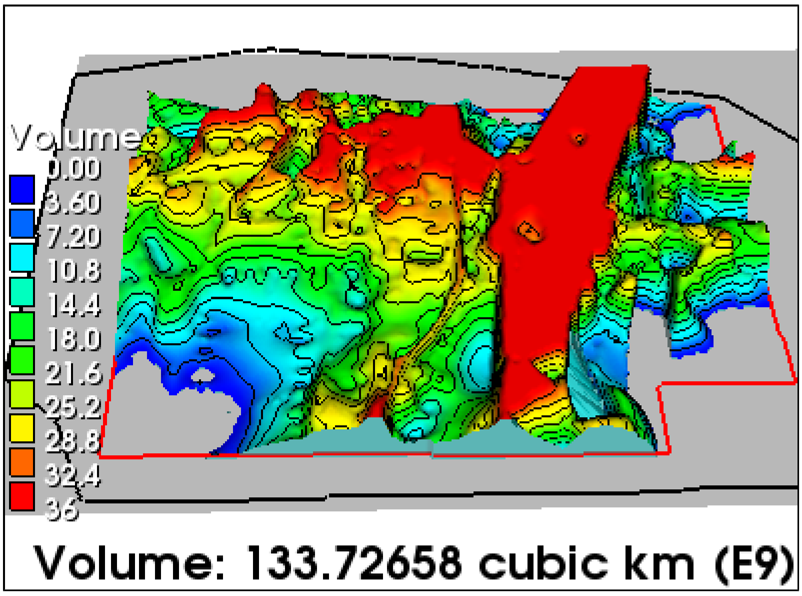
1. Highlight *{04_Depth data} {Grids Interpretation} 03Upper Cretaceous* and activate **Grid Operations > Volume calculation**.
2. Check in **Apply a bottom grid in calculations**.
3. Browse in *{04_Depth data} {Grids Interpretation} 05BaseJurassic* as the bottom horizon.
4. Set **Type of volume to be calculated** to *Between plane1 and plane2*.
5. Check in **Specify plane 1 level** and set the value to 2500.
6. Check in **Specify plane 2 level** and set the value to 2800.
7. Check in **Use area grid or outline** and browse in *01_Cultural Licenses {Licens 07-18}*
8. Write in values for *porosity* = 0.4, *water saturation* = 0.5 and *net to gross* = 0.6.
9. Click **Execute** to perform HCPV volume calculation.
   The result is: Volume calculated from 2500 to 2800: 133.72658 cubic km (E9)

The *display options* are placed under the **Display** tab. Open that tab.

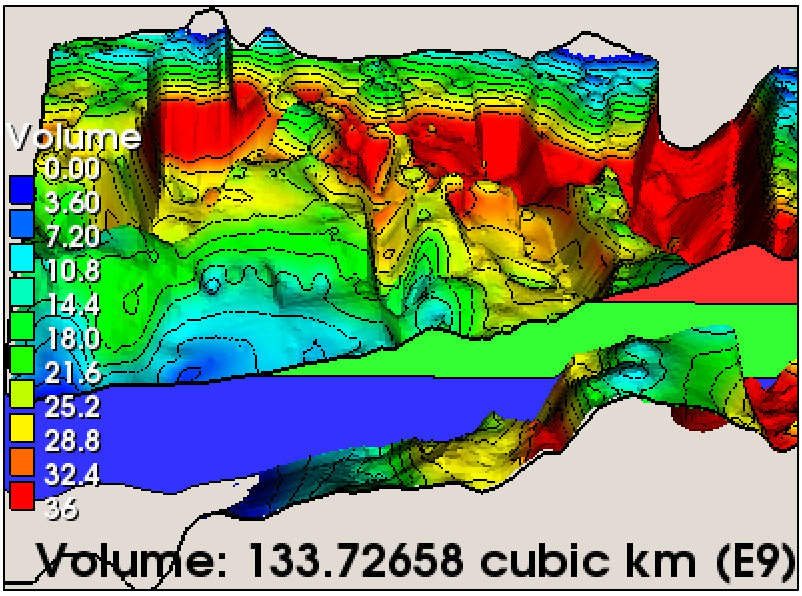
Exercise

Display volume grids.

1. Click on **Display calculated volume** when the combo box is **As single volume**.
   The result is the lower left display.
2. Click on **Display calculated volume** when the combo box is **On bottom of grid**.
3. Set a cursor marker with **y** and click **Get cursor values**.
4. Click **Perform cross section from position value**.
   The result display is shown below to the right.



Volume grid displayed as single volume



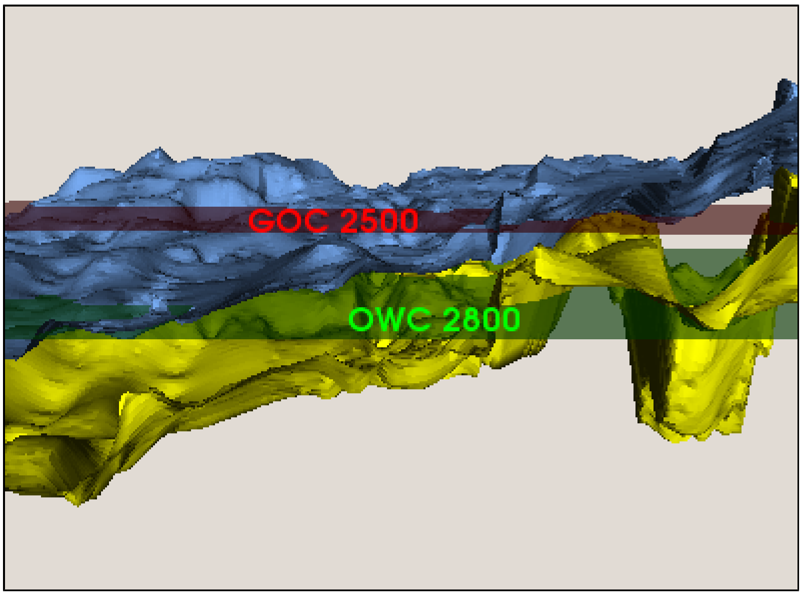
Volume grid and cross section displayed on bottom grid; see comments

Comments on *Volume grid displayed on bottom grid*
The picture shows the volume grid draped onto *Base Jurassic*. The grid is not present outside the area of positive volumes. In addition a cross section is displayed and shows clearly where there are no volume because the top grid is below the bottom contact plane. The cross section study illustrates that clearly.

Exercise

Display top and bottom grids and contact planes.

1. Display the top and bottom grid from the project.
2. Display the two contact planes.
3. Mark with **y** a cursor position at the top plane and type the shell command: **tx2 col red cur fol txt "GOC 2500"**
4. Mark with **y** a cursor position at the bottom plane and type the shell command: **tx2 col gre cur fol txt "OWC 2800"**

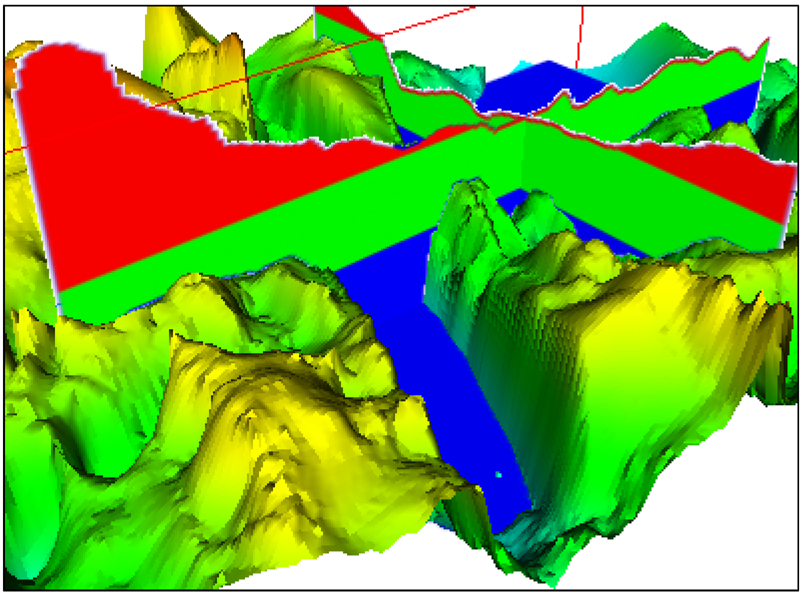


Top and bottom grids and contact planes

Exercise

Create a cube with top and bottom grids and contact planes.

1. Click **Create cross section cube**.
2. Click **Activate frame** and move the manipulator to see the cross section planes in **red green and blue** colors.
3. Display the bottom horizon.

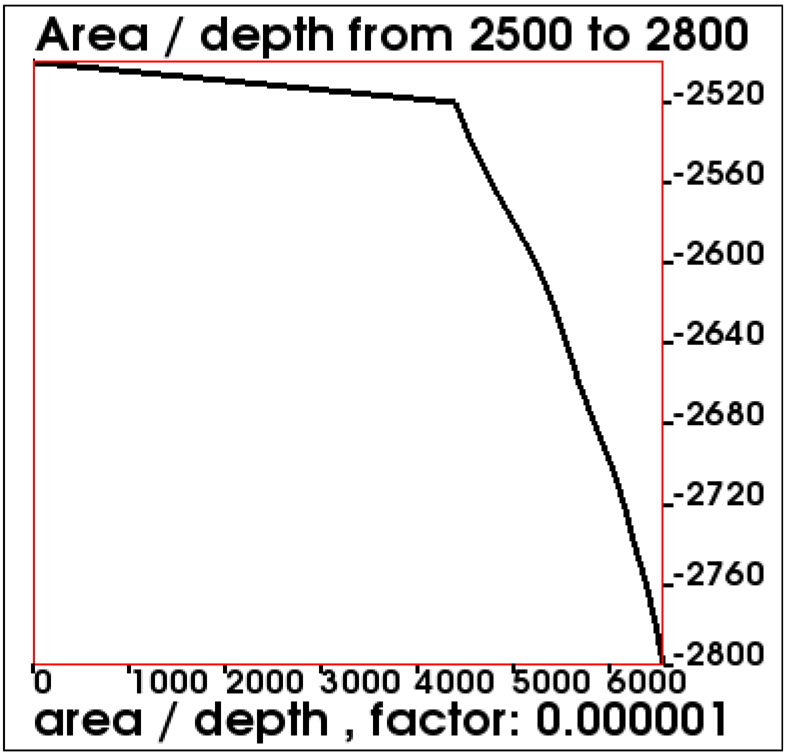


Cross section cube for gas, oil and water

Exercise

Calculate curve for area versus depth.

1. Use exactly the same settings as in Exercise **Calculate HCPV volume for a zone**.
2. Check in **Calculate area or volume against depth**.
3. To calculate area / depth the combo setting is **Calculation is done using:** ***area***.
4. Push **Execute**
5. The **area / depth** curve is saved in the project as *{Course B1} / 01_Cultural Curves_area_depth 03UpperCretaceous_area_depth_curve*.
6. Display the **area / depth** curve by pushing **Show curve**. The screen is blanked and the **area / depth** curve is displayed. Instead of using the existing screen window a new window or a new **Tab** could have been established.
7. Write the text **Area / depth from 2500 to 2800** at top of the window. This is done most easily by clicking a cursor point (in 2D mode) at top left. Then go to the command shell and write: **tx2 cur col bla txt "Area / depth from 2500 to 2800"**
8. The **area / depth** factor is set low to have the area numbers small. The real area numbers are multiplied by the factor.



Area versus depth between 2500 and 2800

10 Plotting

Plotting

Geocap can produce Postscripts plot of very high quality. That is: If the user puts some effort into the details and run some test plots until the required plot quality is achieved.

However, it is easy to get a nice plot and afterwards one can produce lots of plot with the same setting and quality,

Another advantage with Geocaps plotting system is that the output is very small. It has been filtered down to absolute minimum and is very suitable for plotters of all kind as well as converting to PDF files.

The downloaded file **Course B1** contains a folder called **plot** which contains a Postscript file and a PDF file of a Geocap plot. Display the PDF file.
The folder contains also the saveset for the plot and a logo. The location of the logo in the saveset must be changed; see below.

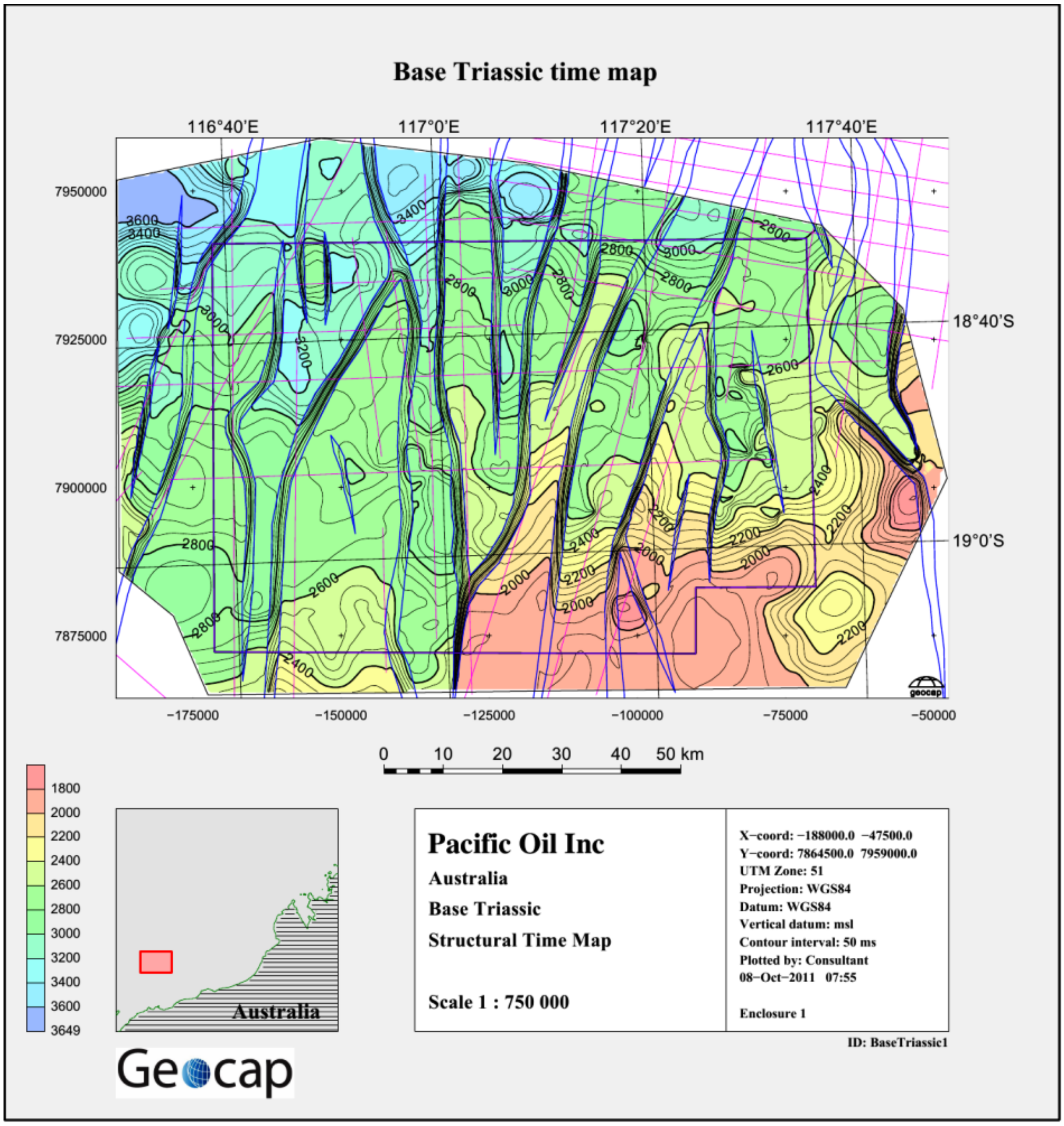
Exercise

Start the plotting system and read the saveset for Course B1.

1. Activate **Tools > PostScrip Plotting**. The plotting menu appears.
2. Do **File > Read saveset** and browse in **BaseTriassic1.saveset** in the **plot** folder.

   Notice that the plot name is **BaseTriassic2** so the original **BaseTriassic1** will be kept.

3. **NB! The disk location of the logo is in 14. Pictures. Click twice on the checkbox and browse in the new location**.
4. Click **Generate Plot** and wait until completion.
5. Go to the **plot** folder and you will find **BaseTriassic2**. It is a Postscript file.
6. Display **BaseTriassic2** using *GsView* or any system that can read and display Postscript. It should look like the display below.
7. Transfer **BaseTriassic2** Postscript file to PDF. There are many freeware systems on internet that can do that: try f.inst. **PDFill PDF Tools**. The PDF file was generated with that system. Also the image below which was converted from the PDF file.
8. Use the **Help** entry on the toolbar for more information about the plotting system and how to use it.
9. Optional: Replace the Geocap logo with your own logo.



Postscript plot of Base Triassic and survey lines and license area (70 % reduced size)

11 Workflows

Workflows

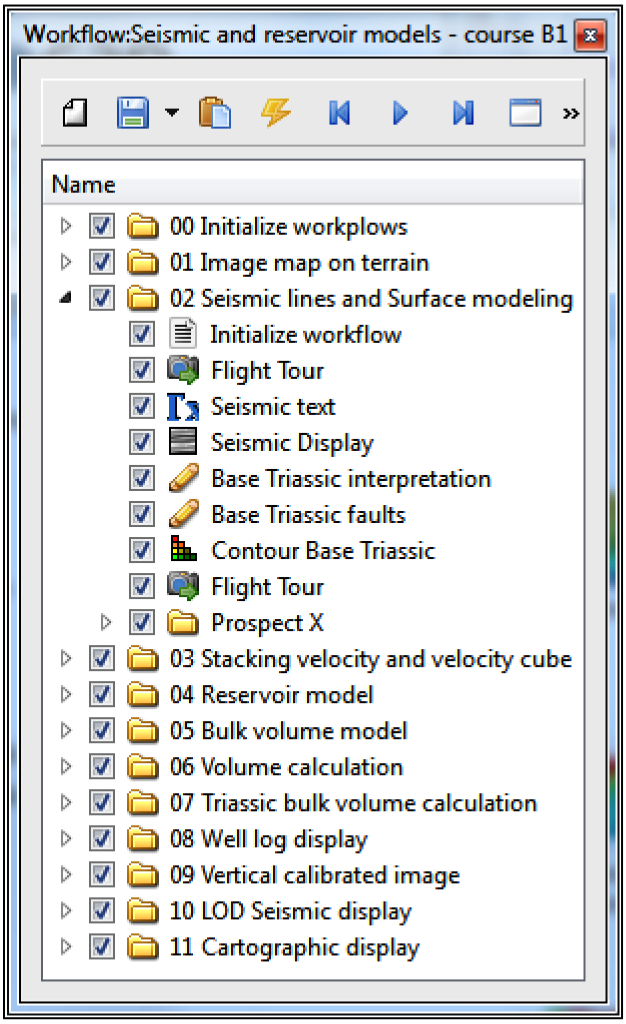
The **Course B1** project has an associated workflow called **Seismic and reservoir models - course B1**. It is located in the **workflow** folder of the downloaded file.

The workflow is a set of *commands* that are chained together to show data from the project and is very suited in presentations of all kinds. It relieves the user from remembering where the data are located and also in which sequence they shall be presented.

A workflow is created in a tree structure much like a project. The actual workflow here has many topics that are separate workflows. They have prefix numbers to have a natural ordering.

The **Flight Tour** command makes the workflow shift from one scene to another in a smooth way much like flying. They are used frequently to get a proper start scene and to fly to new camera positions to see other details.

A workflow is unprecedented in a presentation and is impressive when elaborated.



The workflow created to show data from project Course B1

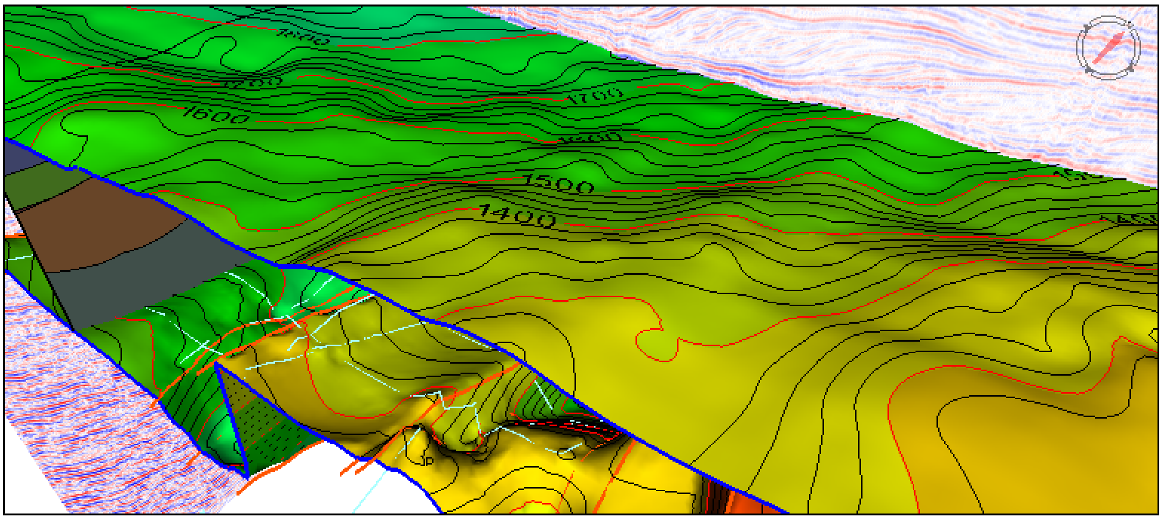
Exercise

Read the workflow file and execute the workflow.

1. Click **File > Open > Work Flow**.
2. Browse in **Seismic and reservoir models - course B1** from the worflow folder.
3. Make sure that the project **Course B1** is loaded.
4. Maximize the Geocap screen.
5. Run the different workflows by right click and **Execute**.
6. Try also the **Execute** button on the toolbar of the workflow.

Study the workflows and see how they are created

- Do **Edit** on the various workflow elements.
- Create your own workflow, f.inst. just displaying a grid and some associated data.



A scene from the workflow

12 Presentation graphics

Presentation graphics

Nice plots are always a wish to achieve. Good plots can be done by our plotting system. But sometimes a screen image must do.

Below is a cartographic plot that is produced by the workflow **11 Cartographic display** in **Seismic and reservoir models - course B1**

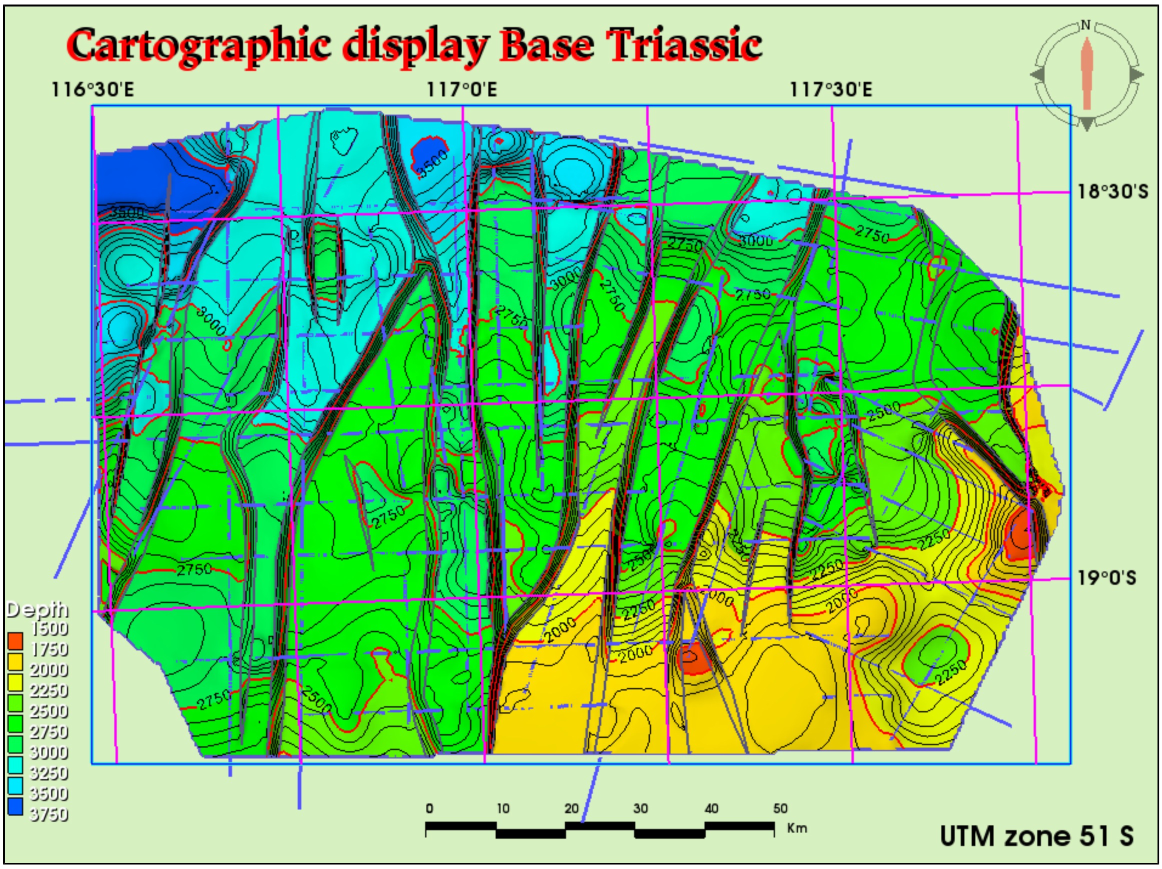
Exercise

Study the workflow 11 Cartographic display.

1. Run the workflow.
2. Open the workflow folder and see which elements are present.
3. Do **any image folder > New > Screenshot** to save the image.
4. Change some parameters to see the effect.
5. Add some text by introducing a new text command to get familiar with the method.

Notes on presentation graphics

- To achieve high cartographic standard it is an advantage to know Geocap and all details well.
- Personal preference is important so presentation graphics is a relative issue.



Presentation of Base Triassic with cartographic standard