Stochastic Modeling and Geostatistics for Reservoirs; Principles and Methods: Computer Workshops Release 5000.8.3

Volume 2

HALLIBURTON | Landmark Software & Services

Part Number 200083C

November 2013

Halliburton | Landmark Software & Services
2107 CityWest Blvd, Building 2, Houston, Texas 77042-3051, USA
P.O. Box 42806, Houston, Texas 77242, USA
Phone:713-839-2000
FAX: 713-839-2015
Internet: www.halliburton.com/landmark

Trademark Notice

Note

Third Party Licenses and Attributions

Halliburton acknowledges that certain third party code has been bundled with, or embedded in, its software. The licensors of this third party code, and the terms and conditions of their respective licenses, may be found in a document called Third_Party.pdf. This document is included in the installation directory of each application from Landmark® software. The path name of the file is similar to the following:

ApplicationInstallationDirectory`/docs/Third_Party.pdf`

or

ApplicationInstallationDirectory`\help\Third_Party.pdf`

Disclaimer

The programs and documentation may provide links to external web sites and access to content, products, and services from third parties. Halliburton is not responsible for the availability of, or any content provided on, third party web sites. You bear all risks associated with the use of such content. If you choose to purchase any products or services from a third party, the relationship is directly between you and the third party. Halliburton is not responsible for: (a) the quality of third party products or services; or (b) fulfilling any of the terms of the agreement with the third party, including delivery of products or services and warranty obligations related to purchased products or services. Halliburton is not responsible for any loss or damage of any sort that you may incur from dealing with any third party.

Stochastic Modeling and Geostatistics for Reservoirs; Principles and Methods: Computer Workshops

Exercise 1
Getting Started

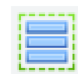
In these exercises you will learn to:

- Launch the DecisionSpace® software
- Create a new session
- Load and display data

Create a New Session

1. Click the desktop icon for the DecisionSpace® Desktop software to open the DecisionSpace® Desktop Session Manager.

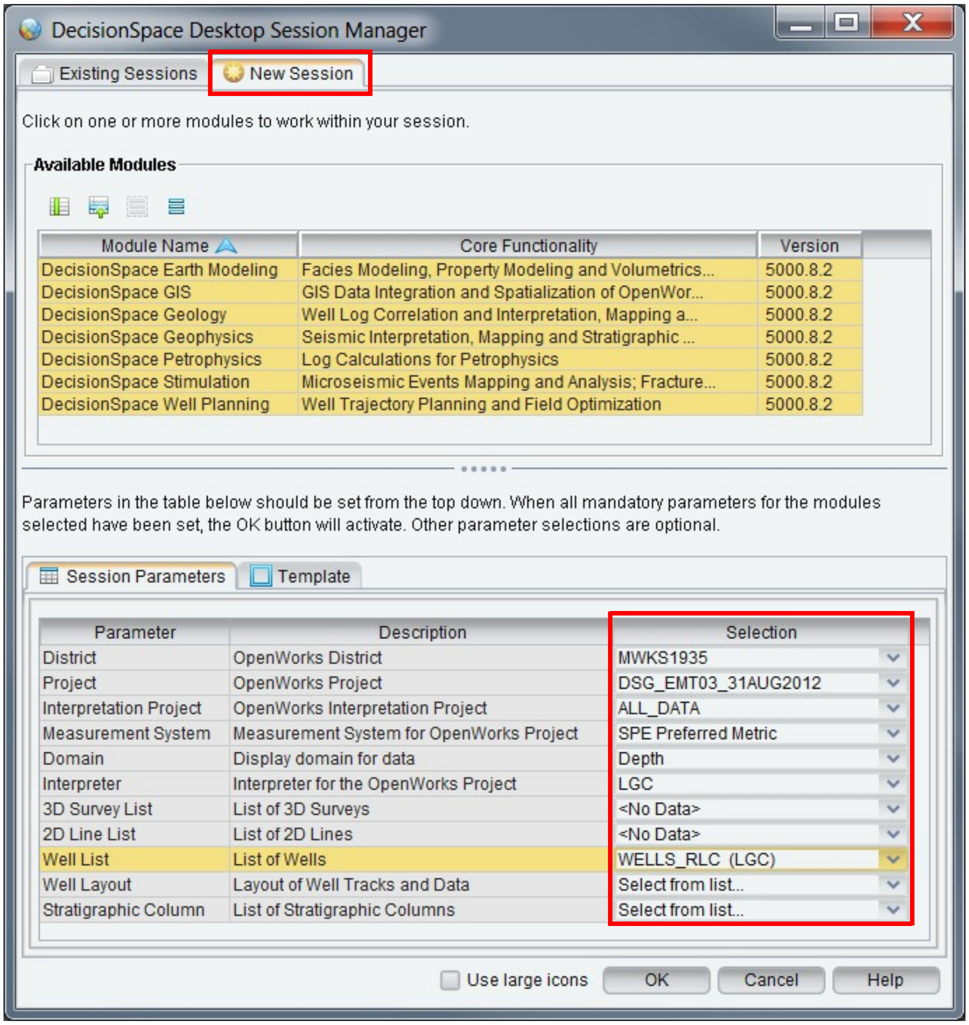
2. On the DecisionSpace® Desktop Session Manager, click the **New Session** tab to open it.

3. Click the **Select All** icon to select all modules. 

   A Session Parameters tab and Template tab appear at the bottom of the Session Manager.

4. On the Session Parameters tab, select the following parameters:

District:	**MWKS1935** [or as provided by instructor]
Project:	**DSG_EMT03_31AUG2012**
Interpretation Project:	**ALL_DATA**
Measurement System:	**SPE Preferred Metric**
Domain:	**Depth**
Interpreter:	**LGC**
Survey List:	**no data**
2D Line List:	**no data**
Well List:	**WELLS_RLC (LGC)**
Well Layout:	**Select from list…**
Stratigraphic Column:	**Select from list…**



5. Select the **Template** tab to see the available options for viewer windows.

   By default, a triple tile Map/Section/Cube is selected (outlined in blue).

6. Click **Map/Section/Cube** to deselect it.

7. Click **Cube** > **Single Window** to select it.

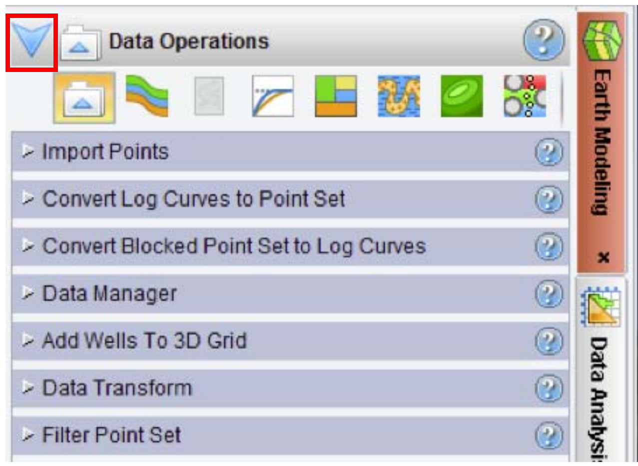
8. Click **OK** to close the Session Manager and launch the DecisionSpace® Desktop software.

   A DecisionSpace® Desktop window opens with a Cube view display tile.

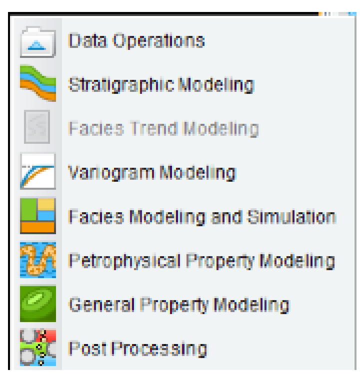
By default, the Workflow Catalog is open on the left side of the window. We will not be using the Workflow Catalog for this workflow.

9. Click the collapse icon on the right side of the orange Cube 1 bar just above the Workflow Catalog task pane.

10. Click the **Earth Modeling** tab on the right side of the window to open the DecisionSpace® Earth Modeling software. The Data Operations task pane is open by default.

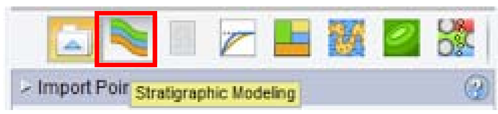


If you click the blue arrow () at the top left corner of the Data Operations task pane, you will see the components listed as shown below.



Note

Facies Trending Modeling is grayed out in the menu shown above because this is from the Cube view window and vertical proportion curves are displayed in map view only. The icons for the Earth Modeling tasks (shown in front of the names in the task menu) also appear at the top right of the Earth Modeling task pane as shown below. The corresponding task name is shown when you hover over the icon.



Clicking a task in the menu or an icon opens the corresponding task pane. A task pane may contain one or more panels with all the components used in that task.

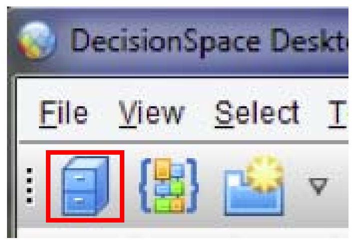
The DecisionSpace® Earth Modeling software displays all data in TVD (true vertical depth). This means that positive Z values indicate depth, not elevation, and that they increase downward. Use the Z drop-down menu on the main toolbar to change Z to TVD.

Load and Display Data

This exercise covers loading and displaying data in the DecisionSpace® software via Select Session Data and the Display Property Editor.

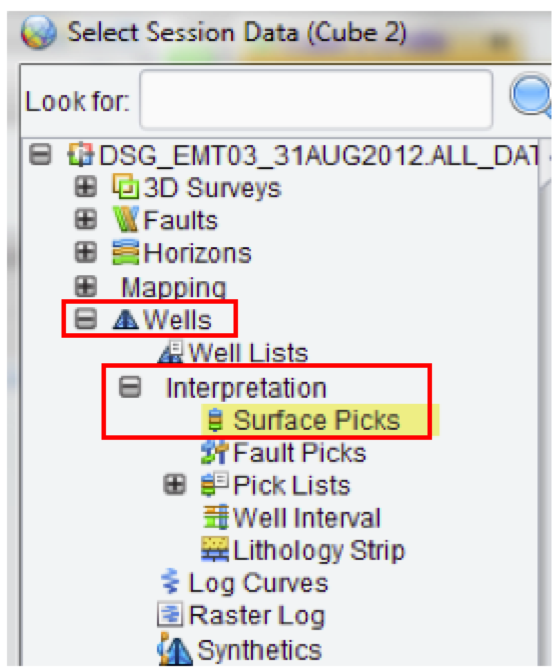
Load Surface Picks

1. Click the **Select session data** icon.



2. Select **Wells** from the data tree on the right.

3. Select **Wells > Interpretation > Surface Picks**.

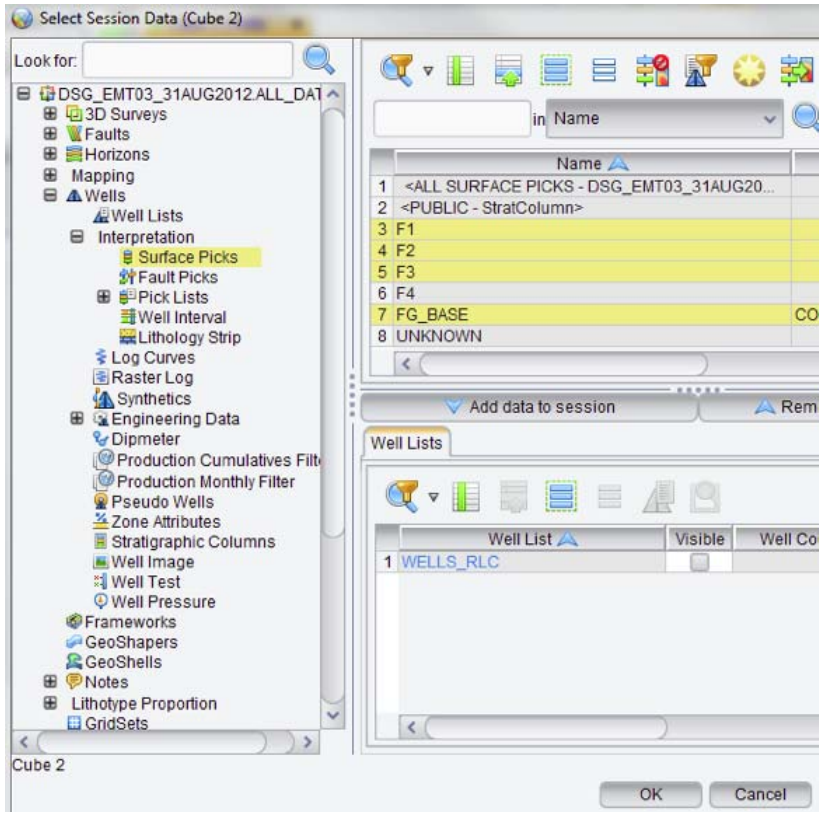


4. Use **CTRL+MB1** to select the following surface picks:

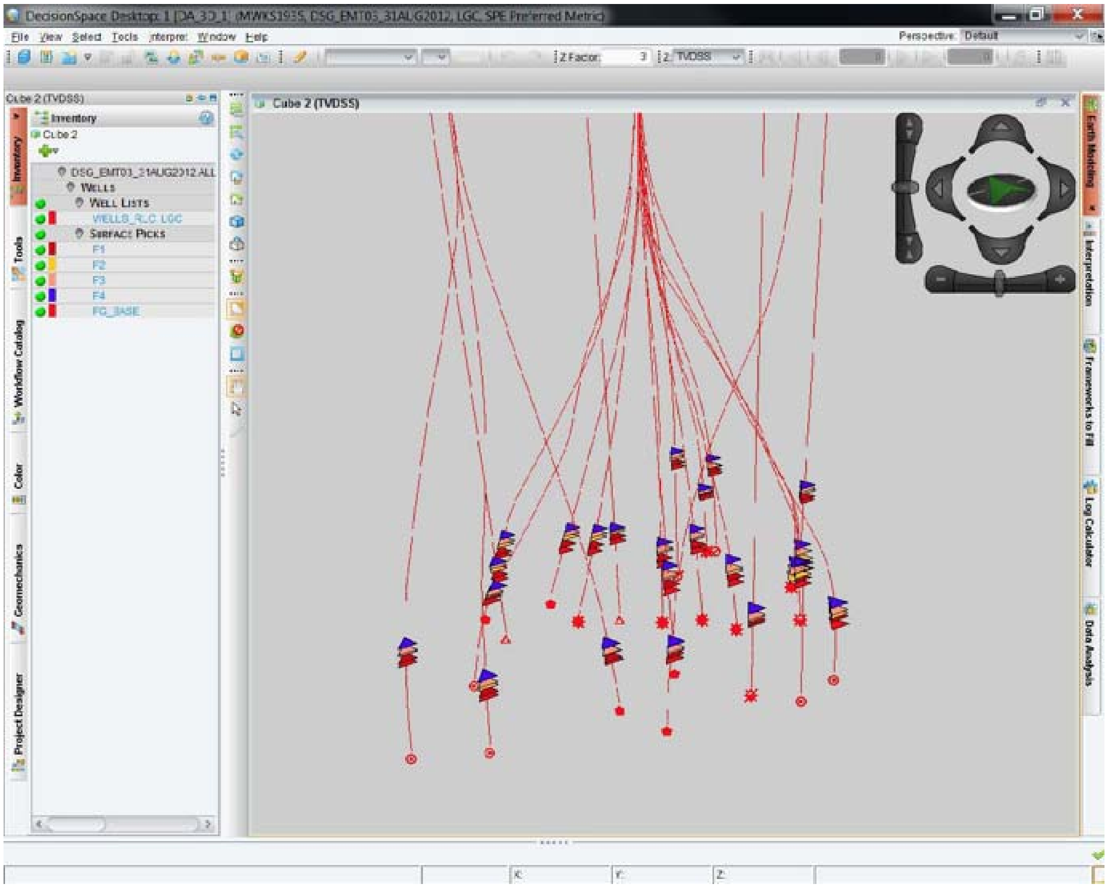
   - F1
   - F2
   - F3
   - FG_BASE

5. Click **Add data to session**.

6. Click **OK**.

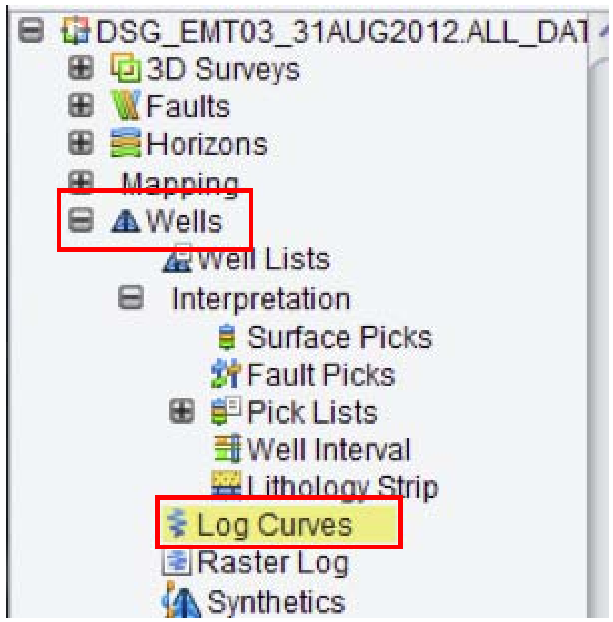


With your newly added surface picks and well list toggled on, your display should look like the following.

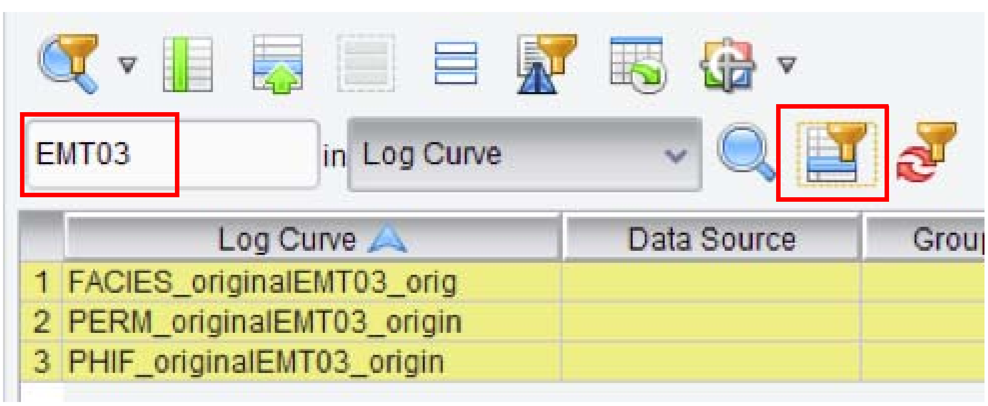


Load Log Curves

1. Click the **Select session data** icon.
2. Select **Wells**.
3. Click **Log Curves**.



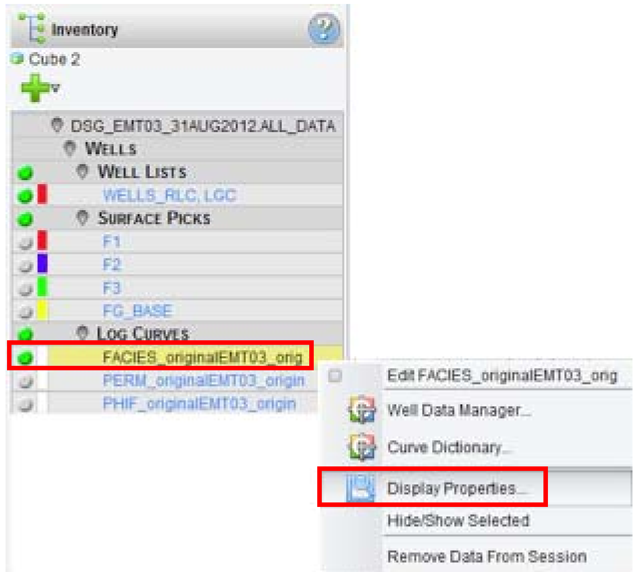
4. Use the filter option to search for **EMT03**. The three curves are selected automatically.



5. Click **Add data to session**.
6. Click **OK**.

Change Log Curve Display Properties

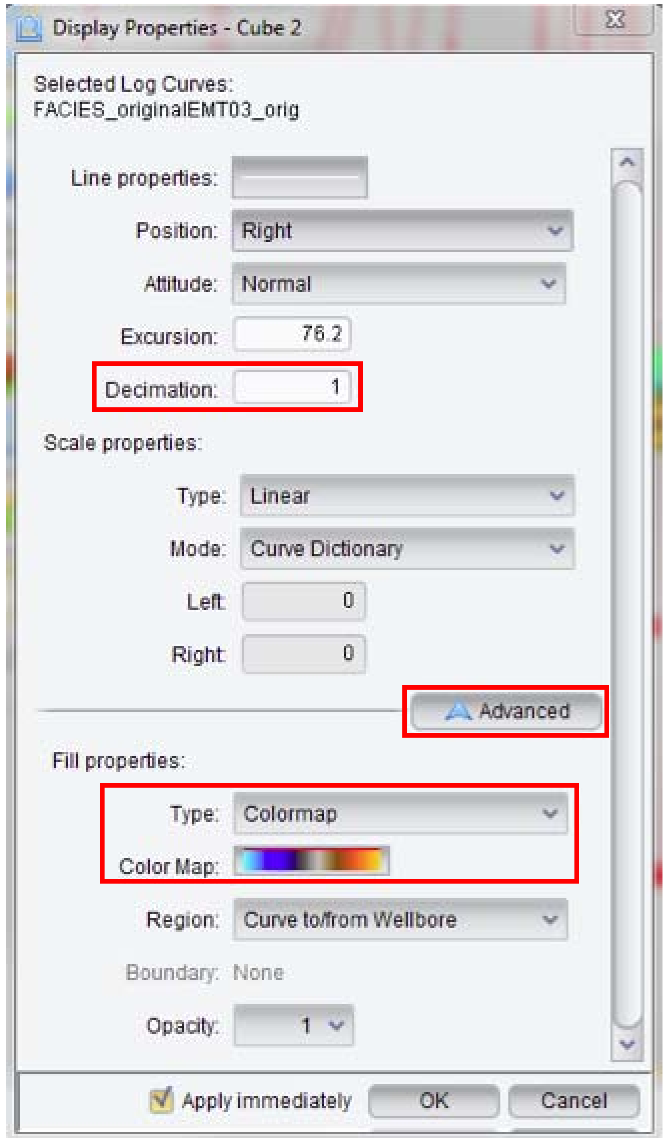
1. Click MB3 on the **FACIES** curve in inventory and select **Display Properties**.



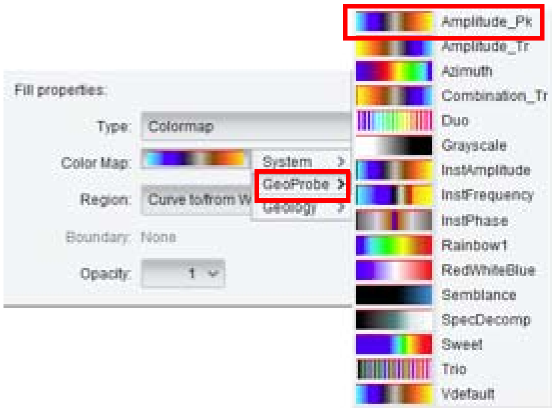
2. Change Decimation to **1**.

3. Click **Advanced**.

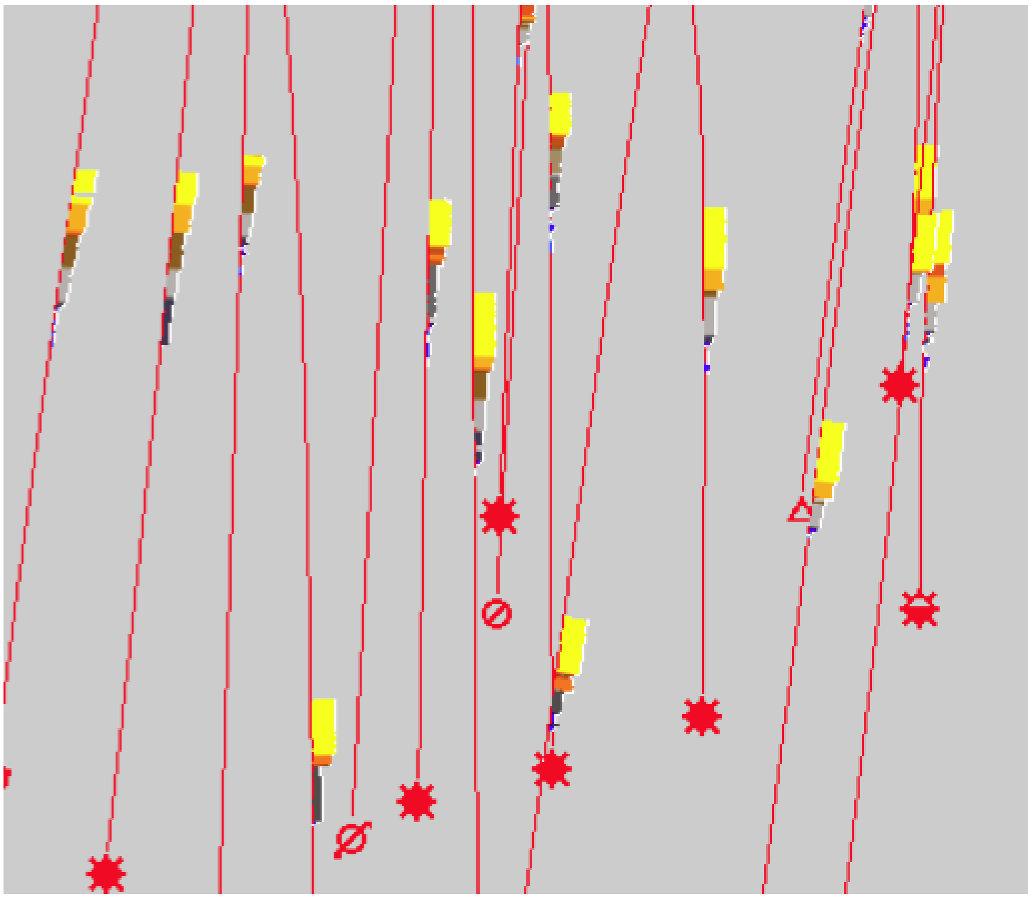
4. In the Fill properties section, select **Colormap** for Type.



5. Click the **Color Map** field and select **Geoprobe**, and then select **Amplitude_Pk**.



6. Click **OK** to finish.

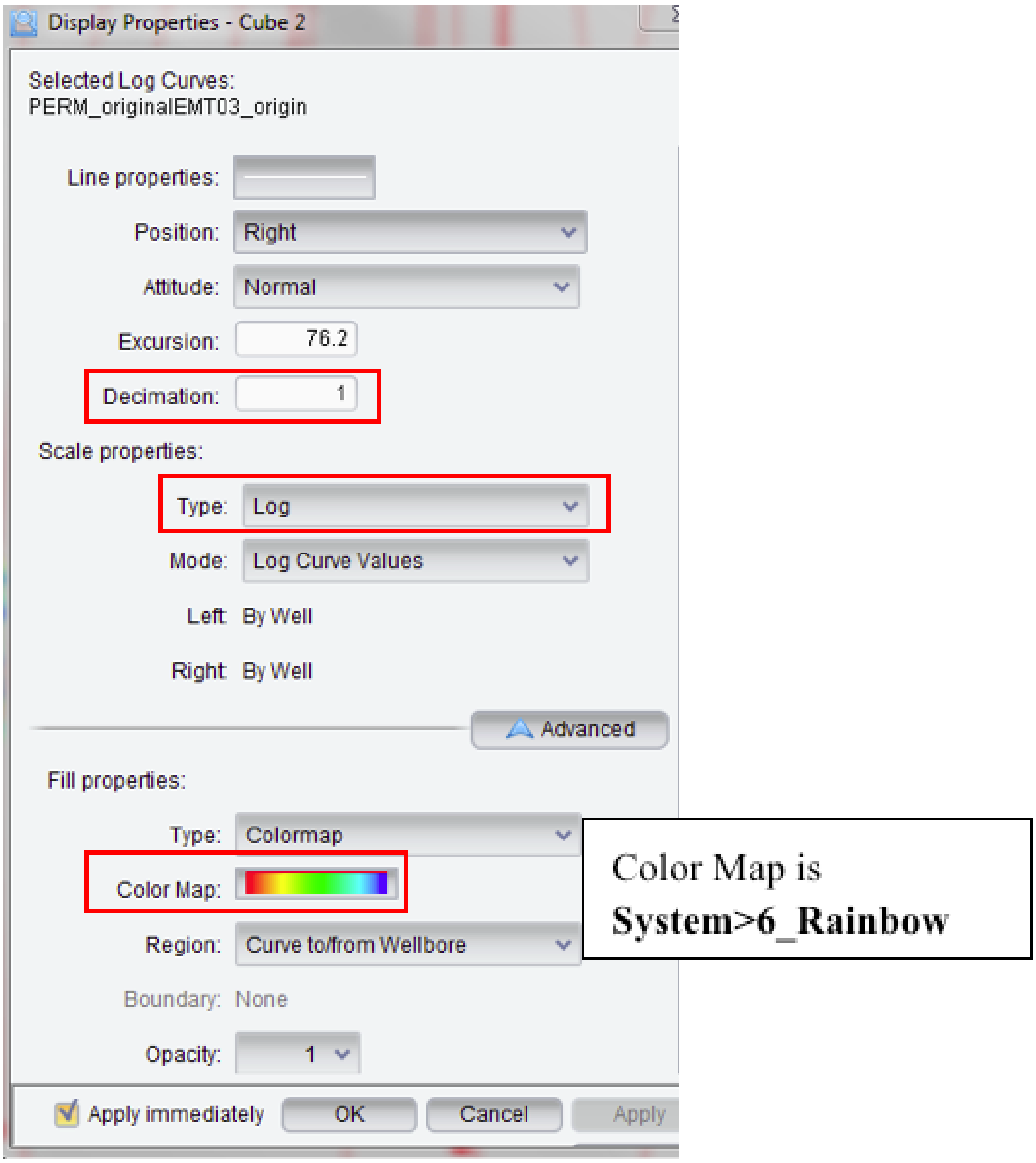


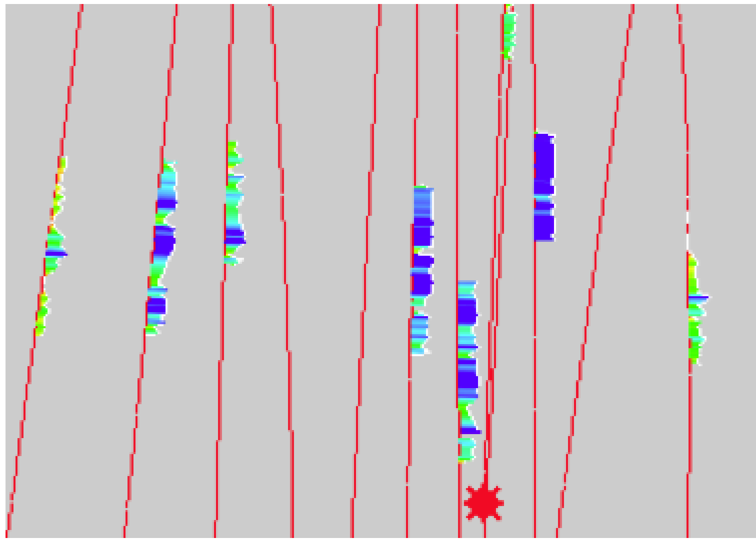
Facies Log Curve Display

PERM Curve

1. For the PERM curve click MB3 on the curve in the Inventory and select Display properties from the drop-down menu.

2. Select settings seen in the following image.



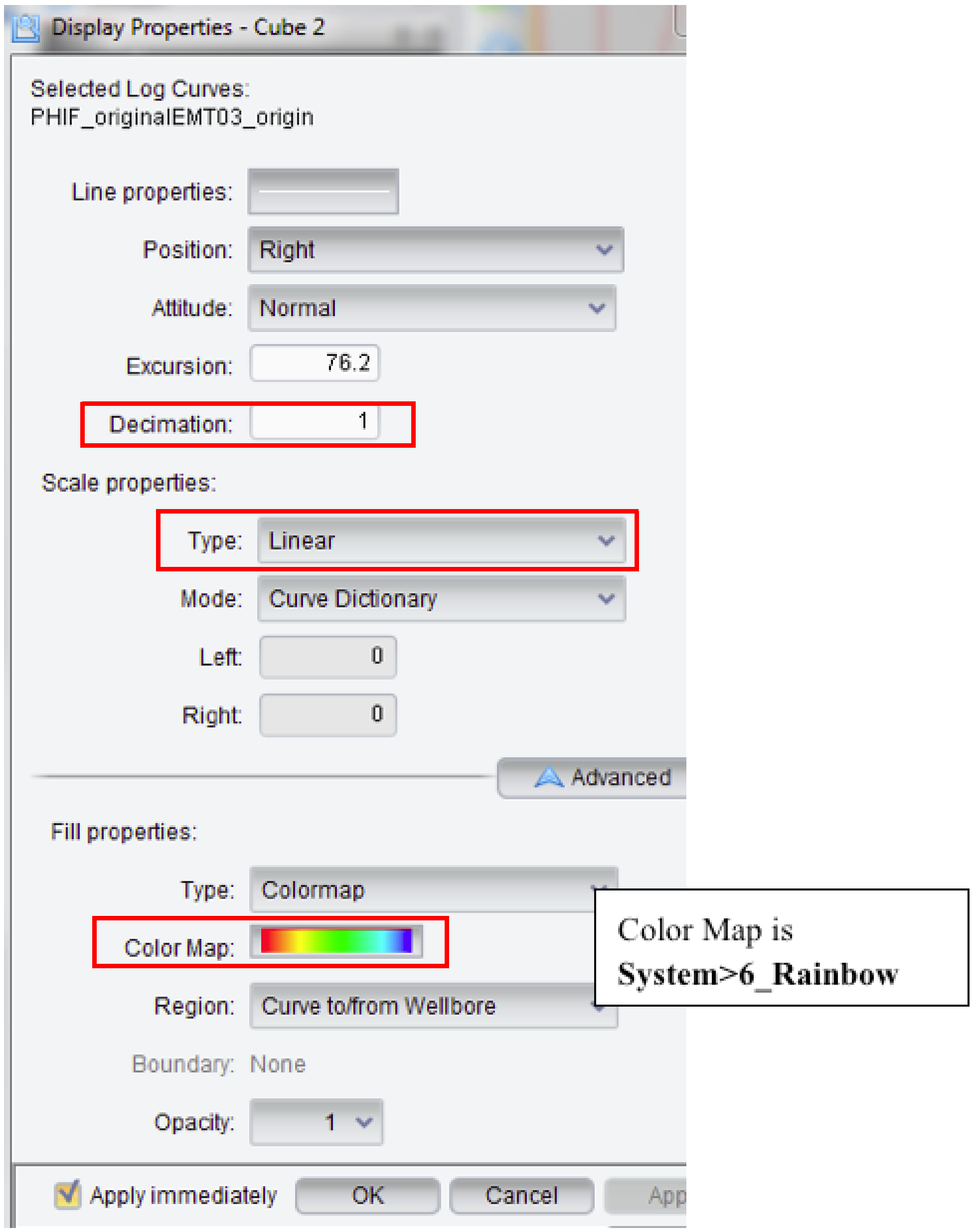


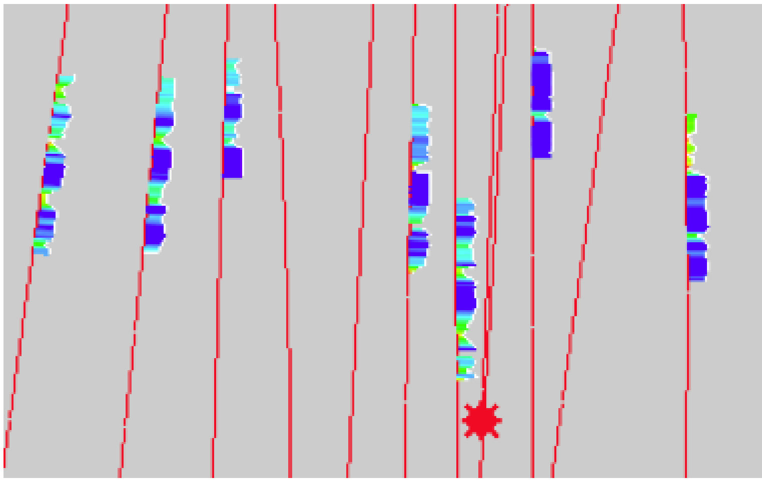
Permeability Log Curve Display

PHIF Curve

1. For the PHIF curve click MB3 on the curve in the Inventory and select Display properties from the drop-down menu.

2. Select the settings seen in the following image.

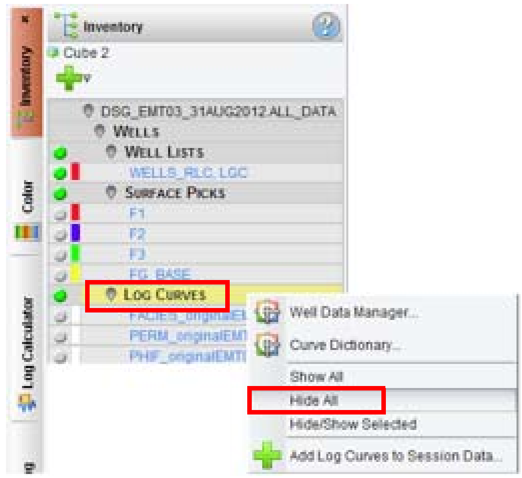




Porosity Log Curve Display

Convert Log Curves to a Point Set

1. Hide your log curves in your inventory using MB3 on **Log Curves** and select **Hide All**.



2. Select the **Earth Modeling** task pane.

3. Select the **Data Operations** icon.

4. Click to expand the **Convert Log Curves to Point Set** menu.

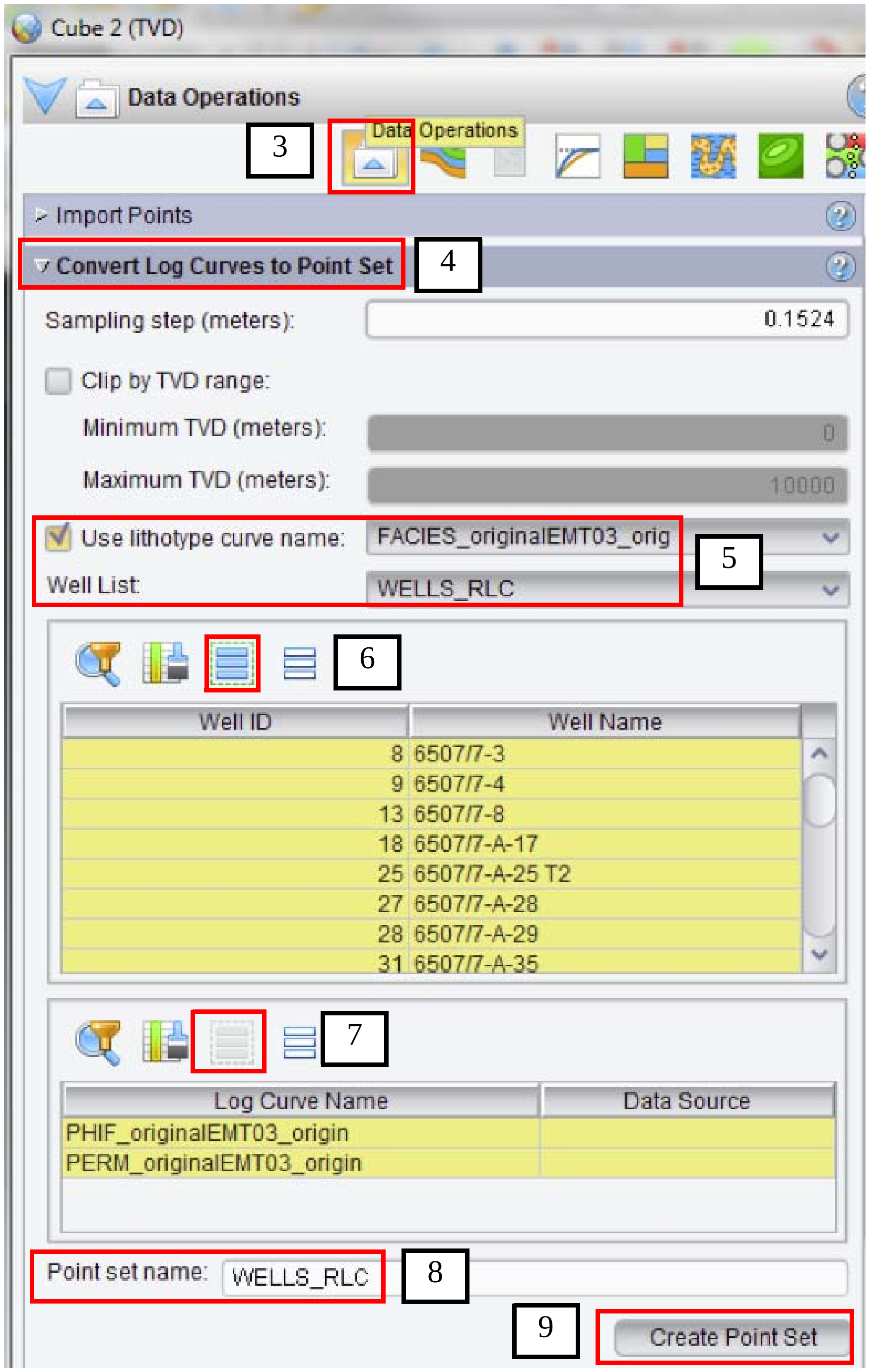
5. Check on **Use lithotype curve name** and select the **FACIES** curve and **WELLS_RLC** for the well list.

6. Click the **select all** icon to select all wells in the table.

7. Click the **select all** icon to select all available curves.

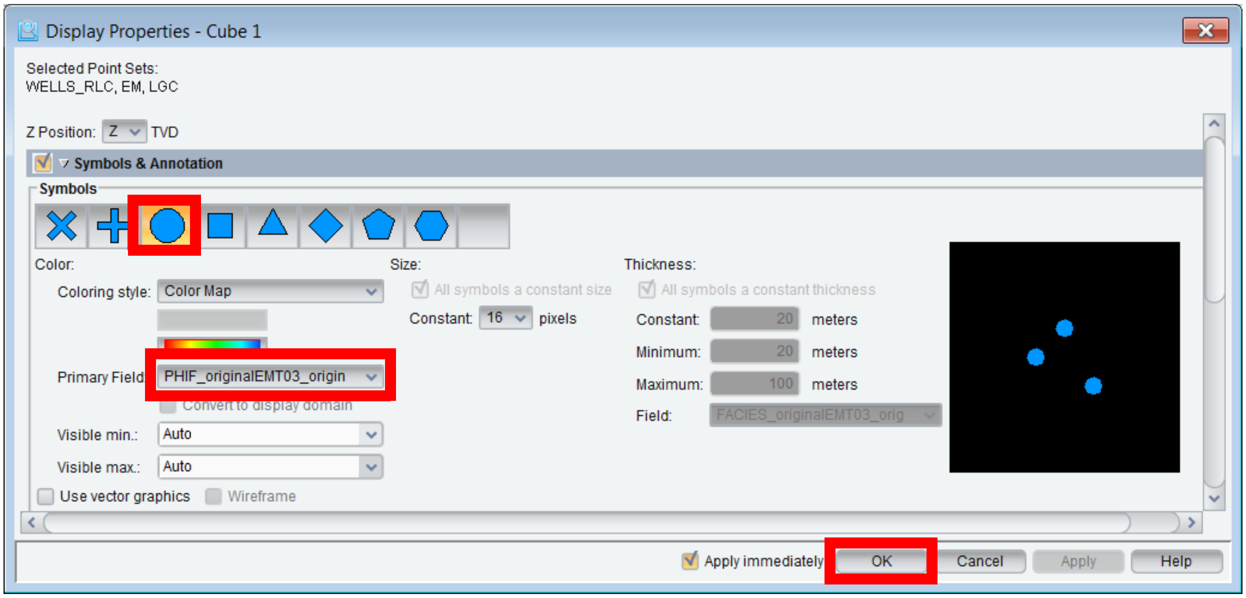
8. Name the point set WELLS_ and add your initials.

9. Click **Create Point Set**.

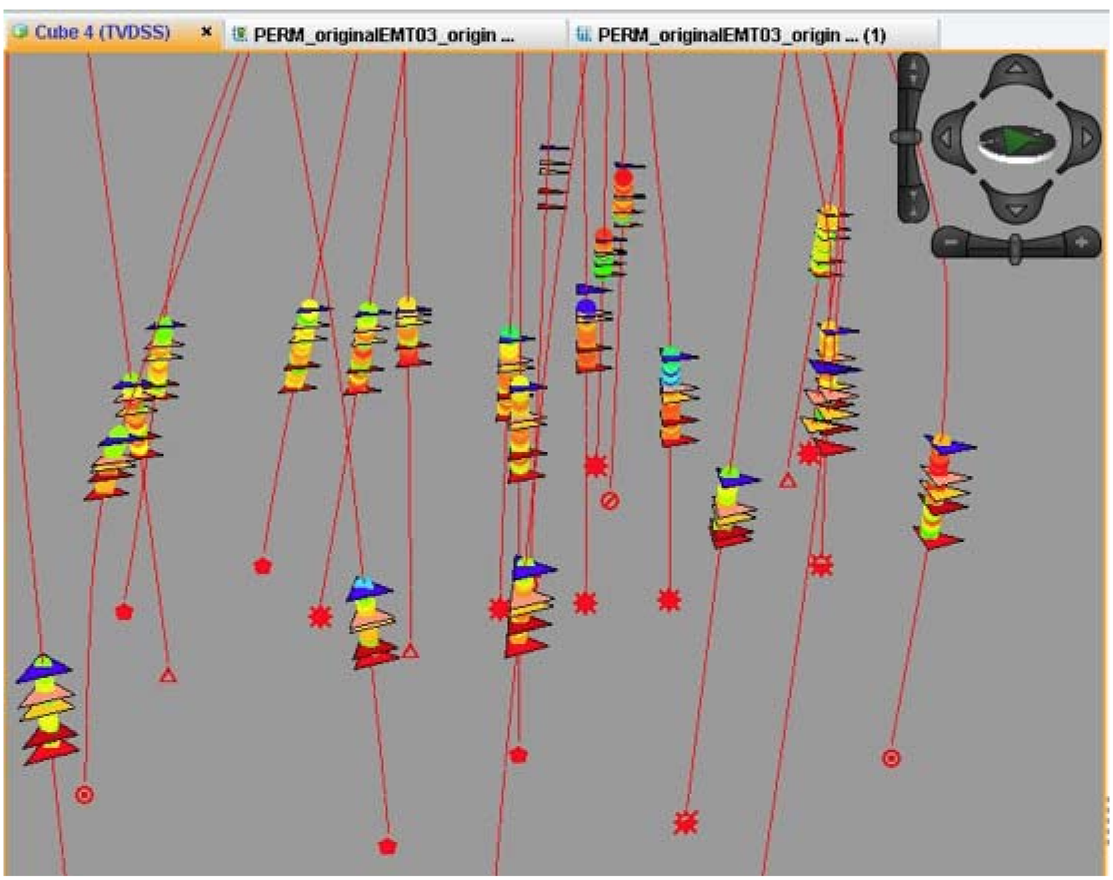


10. Click **MB3** on the point set in your inventory and select the display properties option from the drop-down menu.

11. Configure your options to look like the following image.

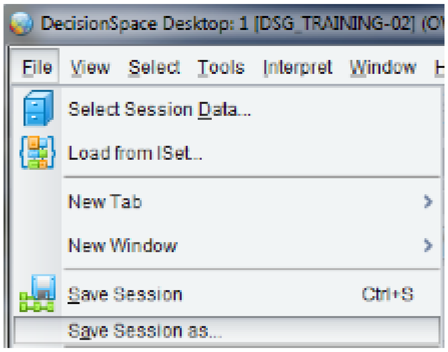


Your cube view will now look similar to the following image.

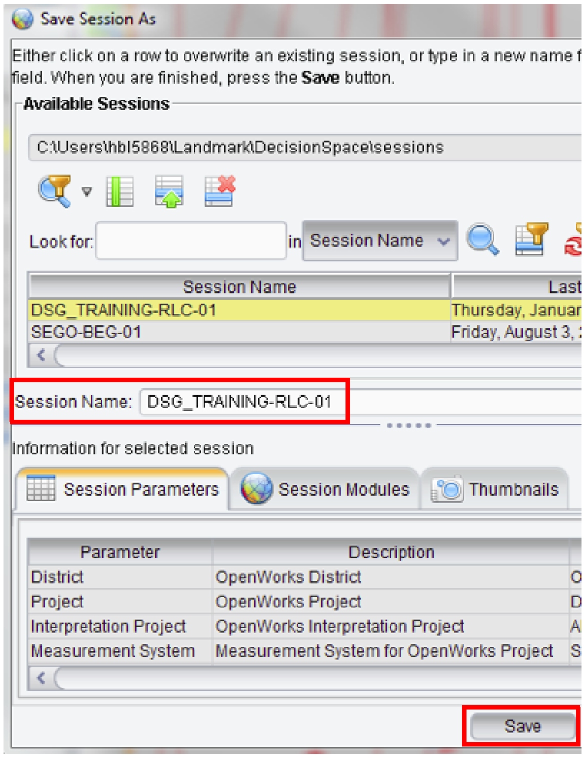


Save the Session

1. From the Menubar click **File > Save Session As**.



2. Save the session name with your initials and a number to mark the iteration using the example shown.



3. Click **Save**.

> **Hint**
>
> It is a good practice to save the session regularly using a counter even though newly created data objects are saved automatically to the OpenWorks® database.

Exercise **2**

Exploratory Data Analysis

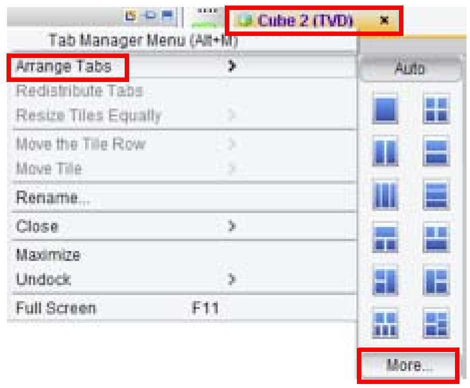
The purpose of this exercise is to introduce you to some of the basic Data Analysis tools used in exploratory data analysis, data quality control and data clean-up. You learn how to:

- Prepare the viewing environment
- Display histograms
- Create a Box Plot and use the outlier analysis tool
- Create data subsets
- Use the Filter option to create a clean version of the data
- Create a cross-plot and change the display parameters
- Use a polygon masking to remove data outliers

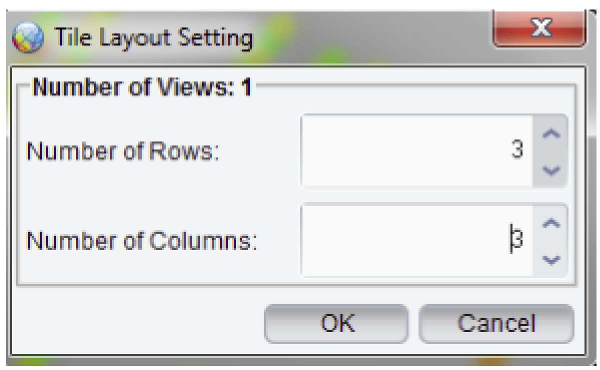
Prepare the Viewing Environment

Because data analysis creates many different displays, prepare the views by creating some empty tab views to populate as the various Data Analysis displays are created.

1. MB3 on the **Cube View**, select **Arrange Tabs**, and then click **More**.



2. Create a 3 by 3 tile layout as shown.

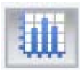


Create Histogram Displays

Histograms should be the first displays created for each property of interest as they convey important information about the behavior of the data; its distribution shape, data range, modality, and perhaps even the presence of outliers.

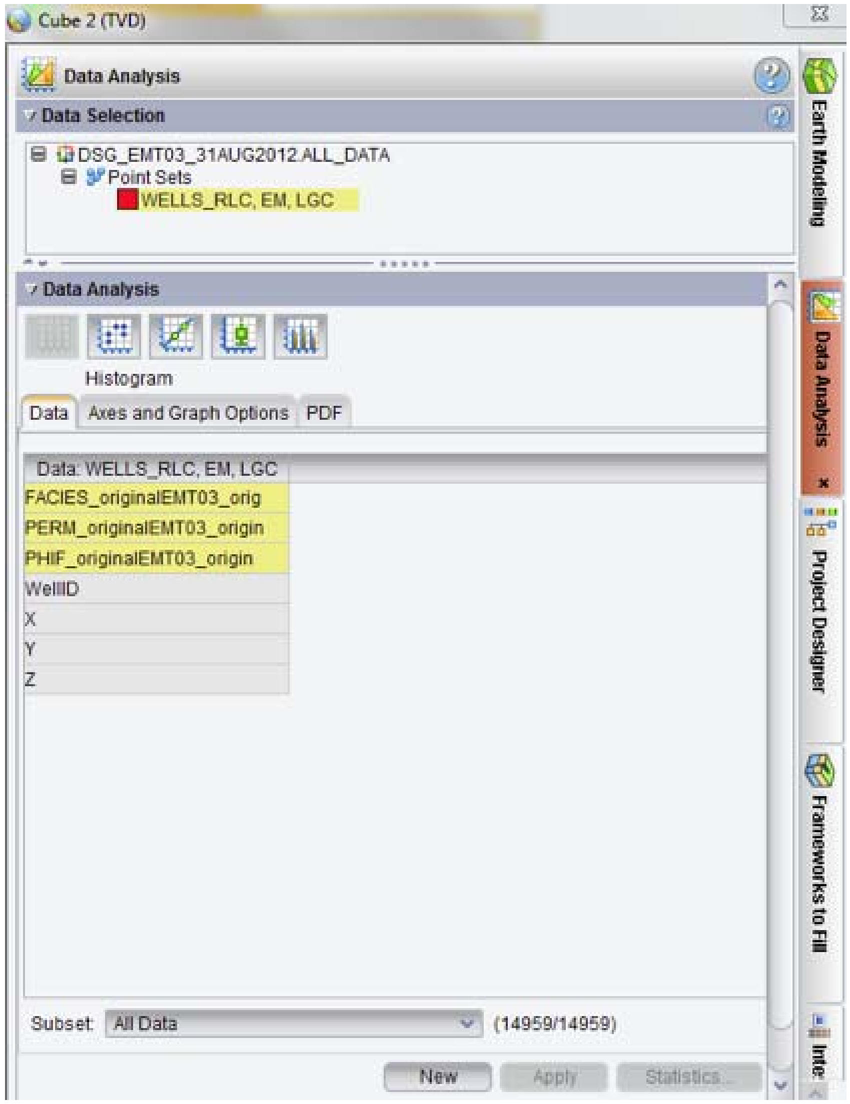
1. Click the **Data Analysis** task pane.

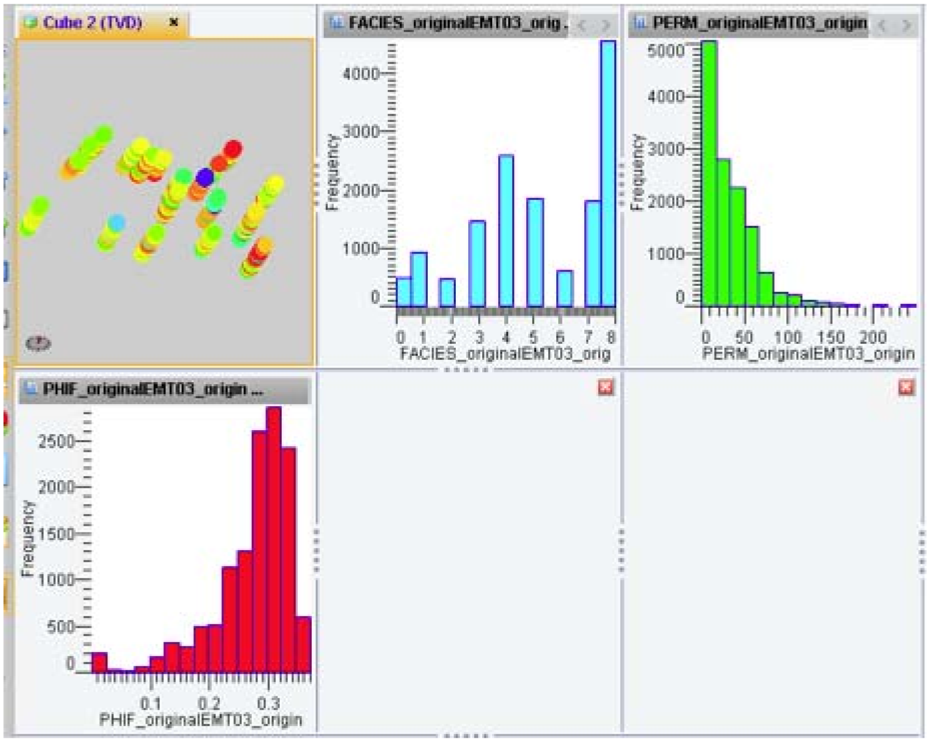
2. Use MB1 to select **Wells_your_initials** in the Data Selection panel.

3. Click the **Histogram** icon. 

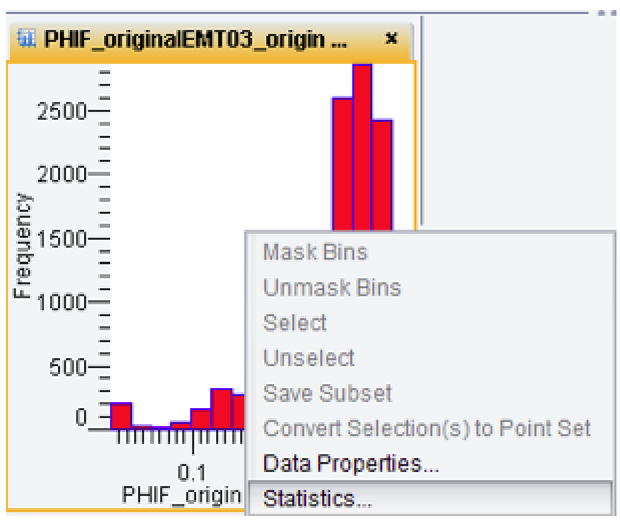
4. With MB1, drag to select **FACIES**, **PERM** and **PHIF** in the Data Analysis table.

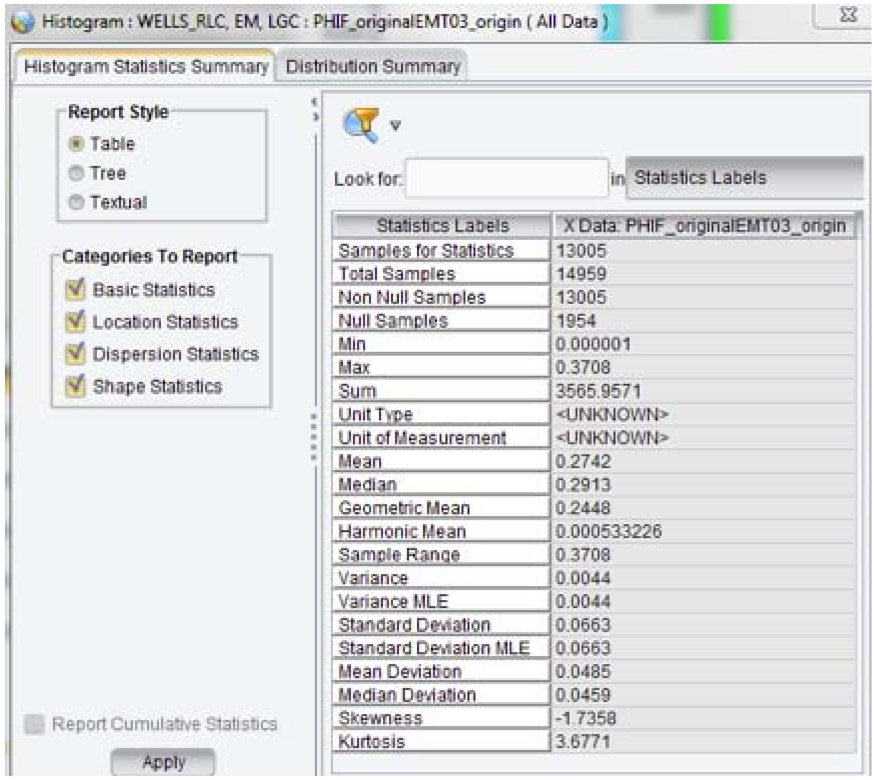
5. Click **New** to display the three histograms.





6. MB3 on the **PHIF** histogram and select **Statistics** to open the Histogram Statistics Summary Window.





7. Add the statistics for PERM to the display.

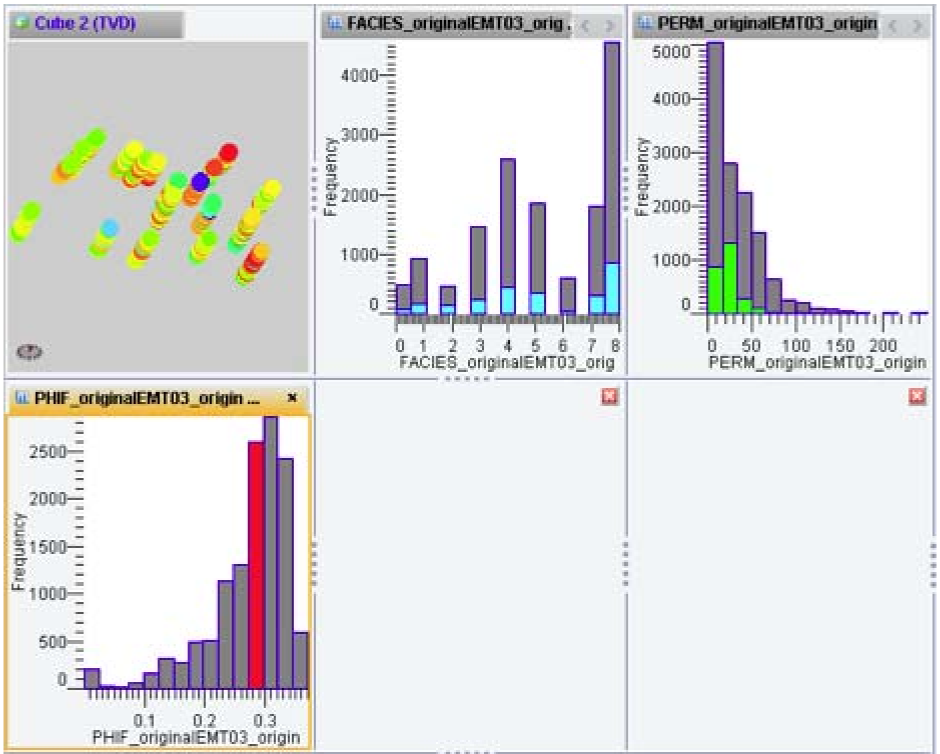
Using Dynamic Linking

Next, you will learn how to investigate the data using the dynamic linking feature between the windows. This is done using MB1 and then MB3 to create a drop-down window.

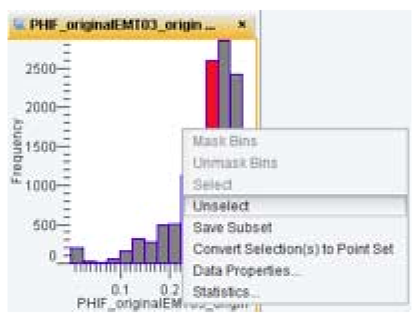
1. Select one of the bars on the PHIF histogram with MB1.

2. Use MB3 to open a drop-down menu and click **Select**.

This action has a reaction in the three displays. Investigation of the data via histograms is an effective way to understand the data.

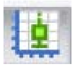


3. To reset the display, MB3 on the PHIF histogram and click **Unselect**.



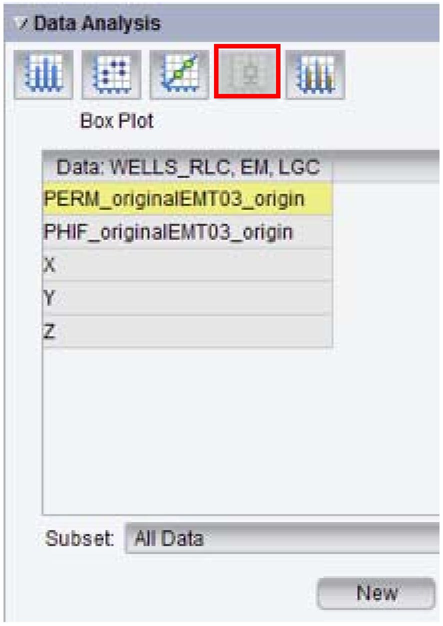
Box Plot with Outlier Analysis

Although not readily apparent, there might be a few data outliers in the PERM data, so next you will learn how to perform an outlier analysis using the Box Plot.

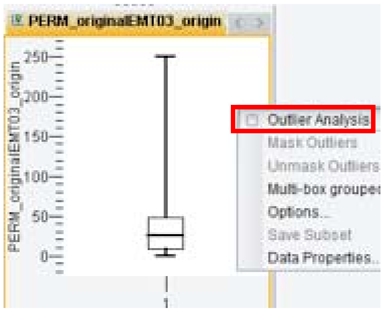
1. Click the **Box Plot** icon .

2. Select **PERM**.

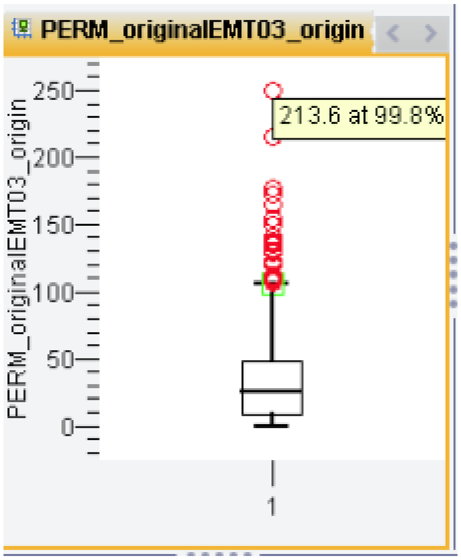
3. Click **New**.



4. MB3 on the Box Plot and select **Outlier Analysis**.



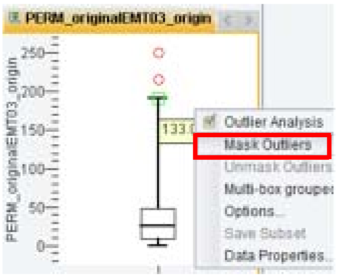
This highlights data points as red circles that might be considered as outliers. These are data that fall more than 1.5 standard deviations outside of the inter-quartile range.

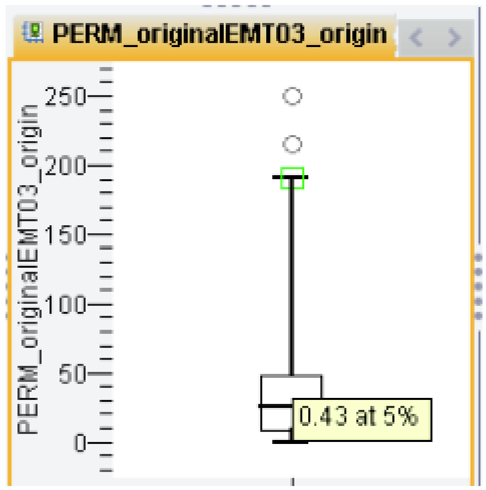


Notice that there are some values greater than 200 md, that are at the extreme tail of the histogram. However, many other points were also flagged as potential outliers. Use caution when performing outlier analysis on highly skewed data, such as permeability, as the results are based on the arithmetic mean and its standard deviation. In this example, consider only the two values greater than 200 as bad data values. The box plot offers a tool to select and mask the outliers.

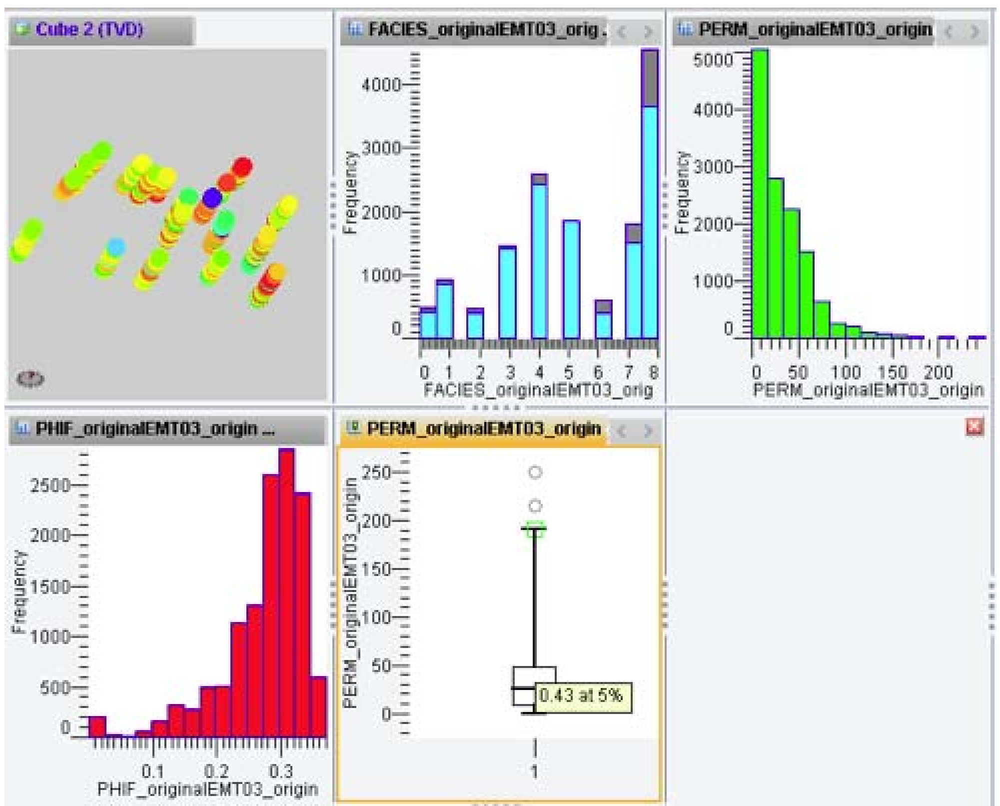
5. Click and hold MB1 on the green box inside the box plot and slide it so only the upper two red circles are in view.

6. Use MB3 to select **Mask outliers** from the drop-down menu. The red circles turn gray.

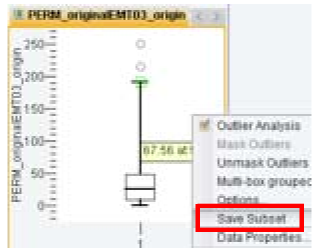




Now look at the histogram displays to see which PHIF and FACIES are related to the PERM outliers.

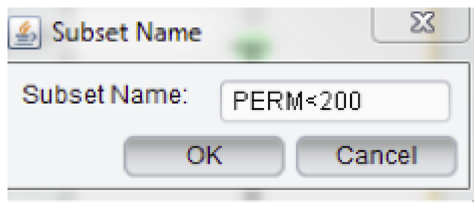


7. MB3 on the box plot and select **Save Subset** to open a new window.



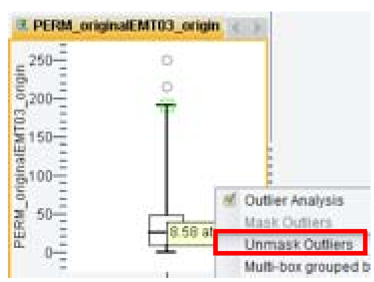
8. Name the subset PERM<200.

9. Click **OK**.



Subsets can be used in subsequent Data Analysis displays by selecting the subset name before making the display.

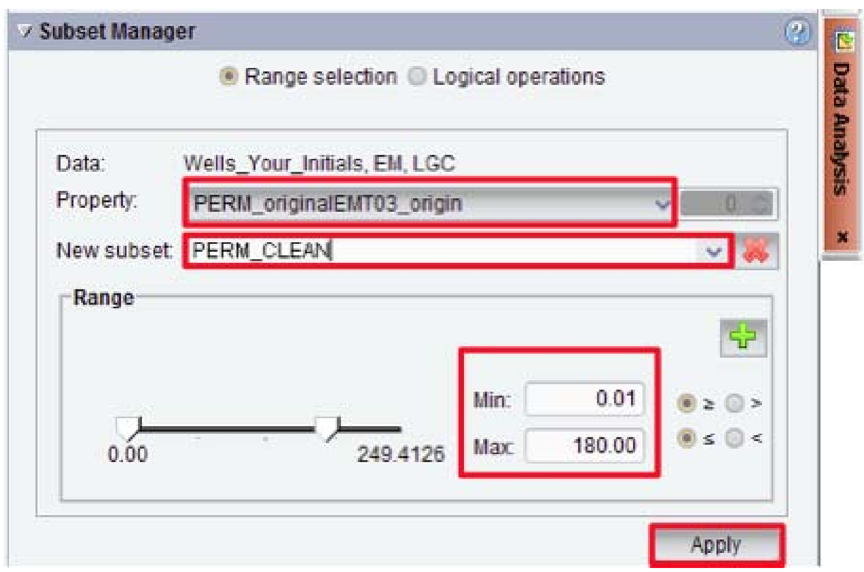
10. Before continuing click MB3 on your box plot and select **Unmask Outliers**.



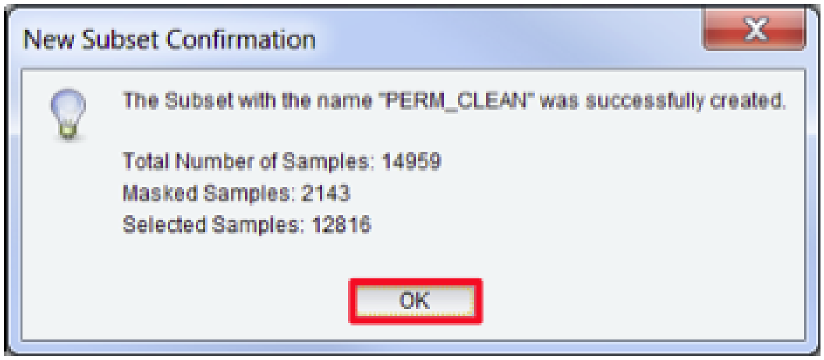
Subset Manager

You can also use the subset manager found in the Data Analysis task pane to create subsets of your data and exclude outliers or to simply range limit the data.

1. Click the **Data Analysis** task pane.
2. Select the **Subset Manager** subtask panel.
3. For PERM, enter parameters 0.01 for Min and 180.0 for Max.
4. Enter the New subset name: PERM_CLEAN.
5. Click **Apply**.



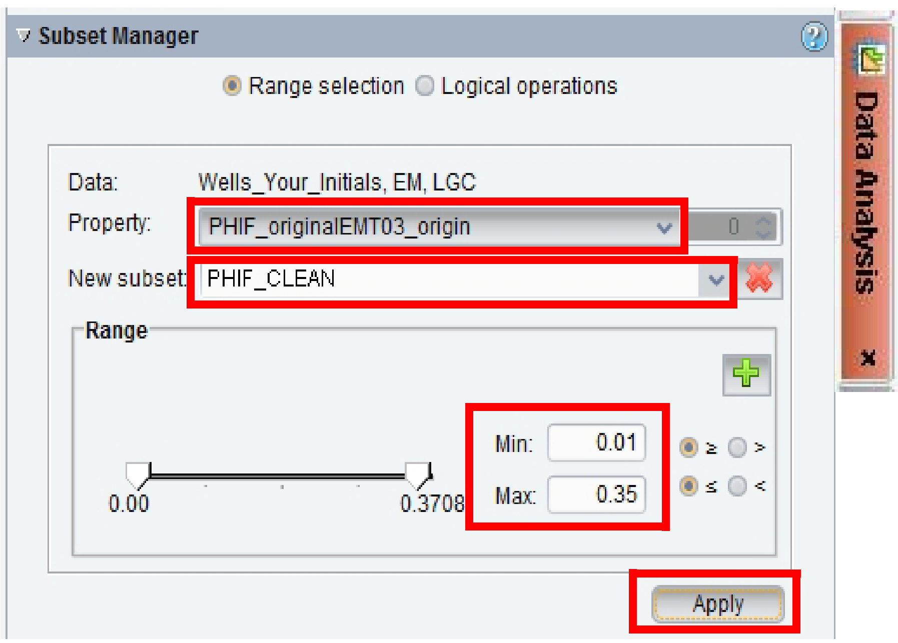
6. Click **OK** on the confirmation dialog box.



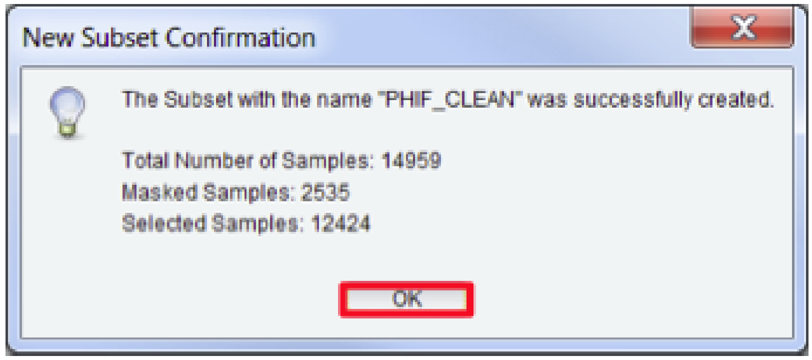
7. For PHIF, enter parameters 0.01 for Min and 0.35 for Max.

8. Enter the New subset name: PHIF_CLEAN.

9. Click **Apply**.



10. Click **OK** on the New Subset Creation dialog box.



At this step in the data analysis you created range limited versions of the PHIF and PERM. None of the values were deleted, only flagged as to what values you want to keep for subsequent analysis. Because subsets are not always easy to work with, next you learn how to create a new version of the data using the subsets.

Filter Point Set

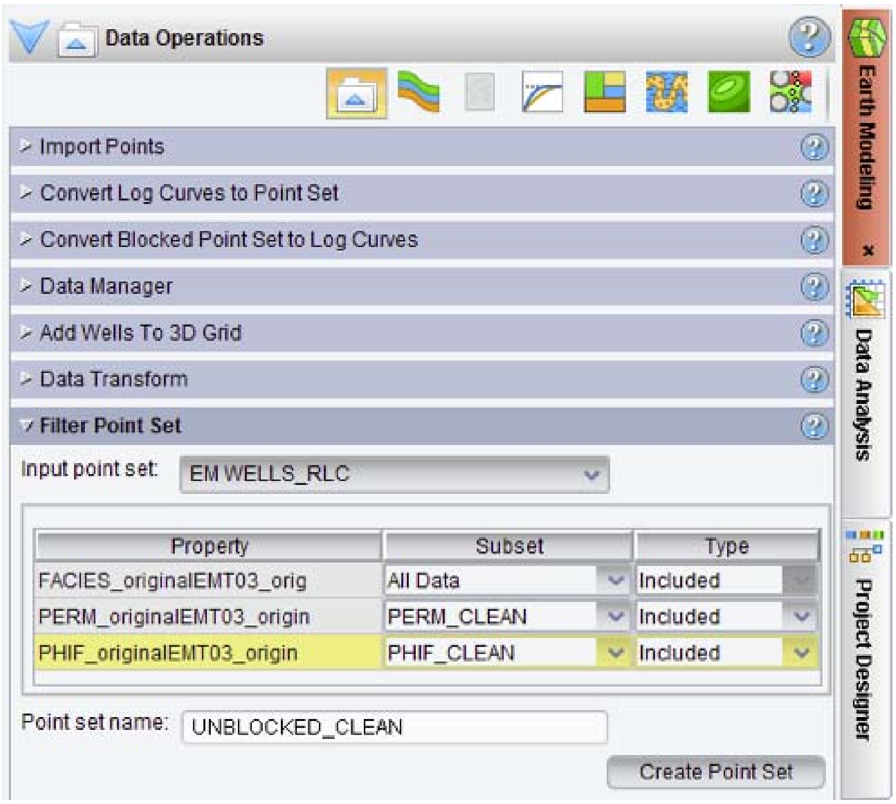
The Filter Point Set panel allows you to select a subset associated with the original point set and then either include or exclude that subset to keep or filter out invalid data values and then create a new, clean point set.

1. Click the **Earth Modeling** tab and then select the **Data Operations** icon.

2. Select the **Filter Point** Set subtask panel. The Input point set is selected automatically because it is the only current active point set in the session.

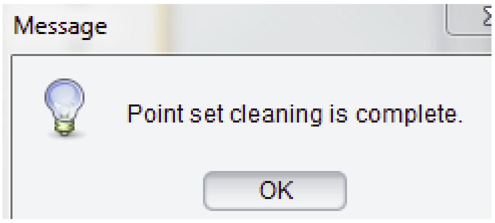
3. Choose the subsets as shown for PERM and PHIF. The type is **Included**.

4. Name the new point set: **UNBLOCKED_CLEAN**.



5. Click **Create Point Set**.

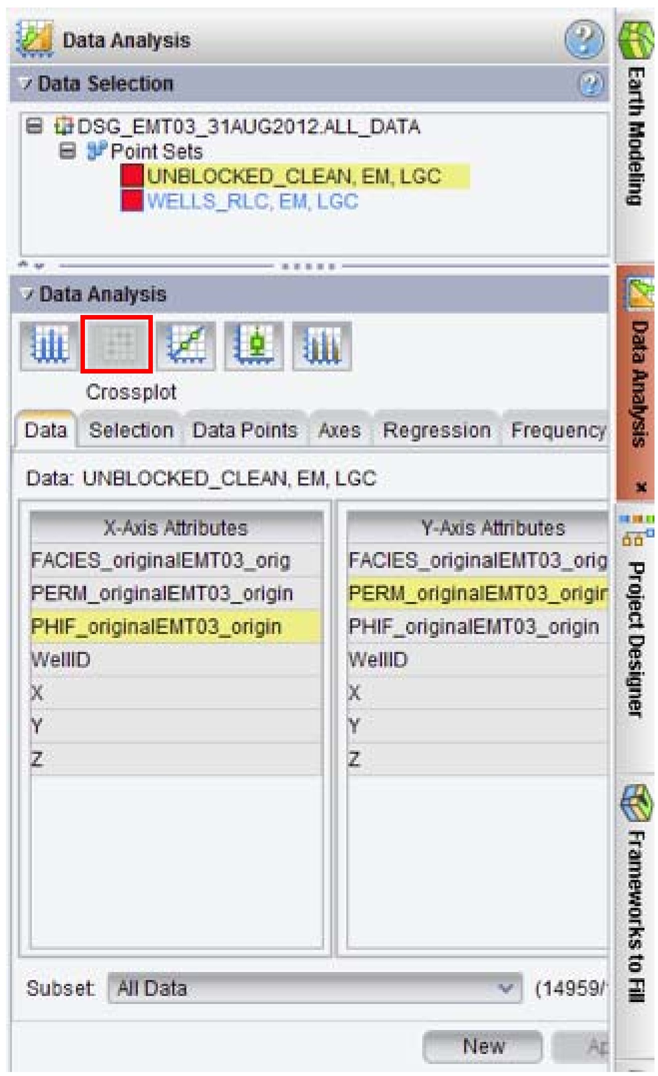
6. Click **OK** to dismiss the message.

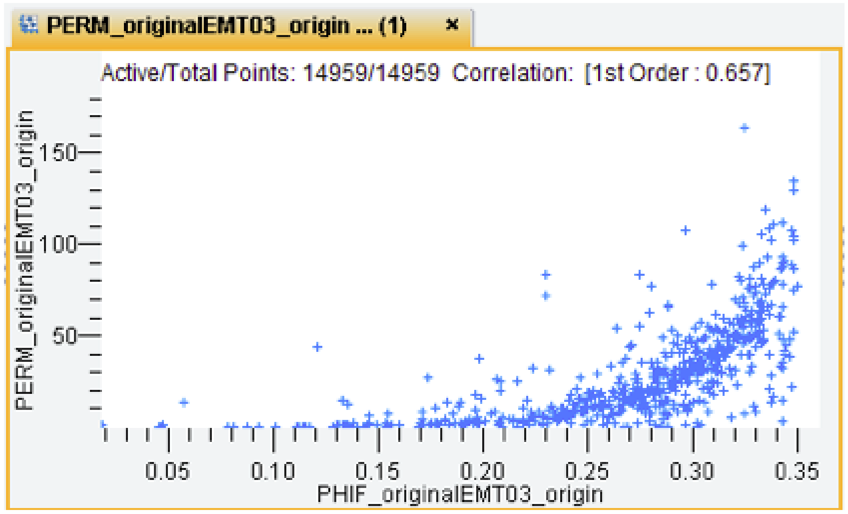


You will now find the UNBLOCKED_CLEAN point set in your Inventory and in the Data Analysis tree. Let us now look at a cross-plot of the cleaned PERM data against our cleaned PHIF data.

Create a Cross-plot

1. Click the **Data Analysis** task pane.
2. Select your **UNBLOCKED_CLEAN** point set in the data selection panel.
3. Click the **Crossplot** icon in the Data Analysis panel.
4. Select **PHIF** attribute for the X axis and **PERM** for the Y axis.
5. Click **New**. Your cross-plot will now be visible in your display.



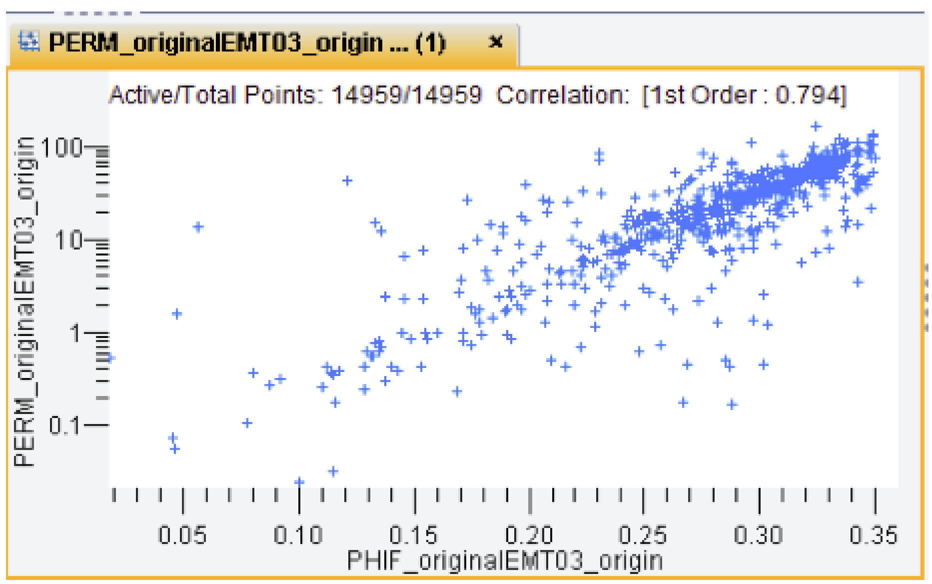


In this cross-plot we observe a reasonably good correlation between the porosity and permeability, with only a few scattered points. Next you will change to scale for PERM from linear to logarithmic and then you will use a polygon selection tool to do a final data clean-up.

6. Click the **Axes** tab. If a Warning Message appears just click **OK**.

7. Change the PERM Scale Type to **Logarithmic**.

8. Click **Apply**.

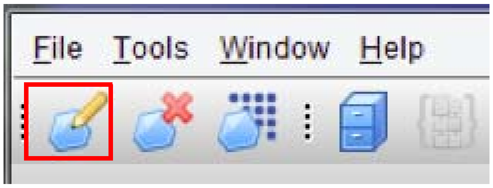


We observe an increase in the correlation once the logarithmic transform is applied, which is typically the case. There are still some possible outliers and scatter seen in the cross-plot that you might want to remove, so you will learn how to use another tool to create a subset and save a final cleaned version of the data.

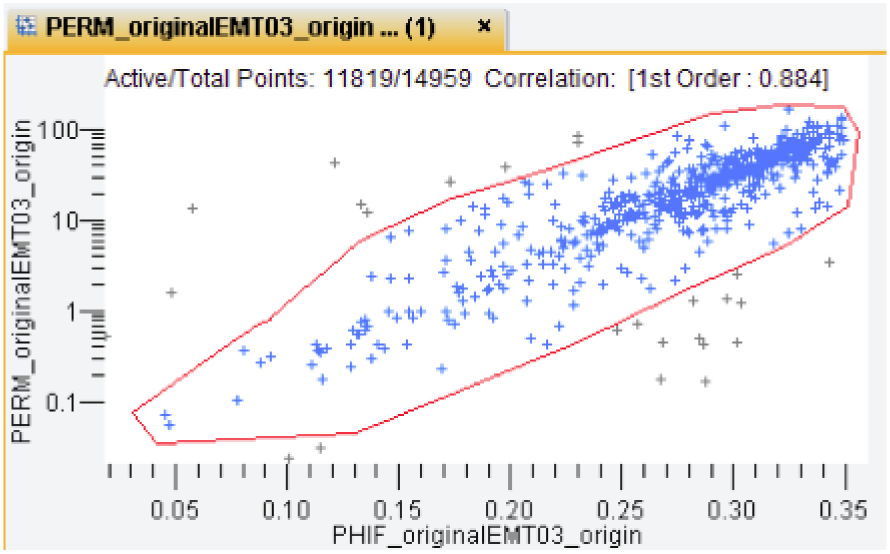
Create and Apply a Subset Using the Polygon Tool

Create a Subset

1. Select the **draw polygon** tool from the main tool bar in the upper left portion of the window.

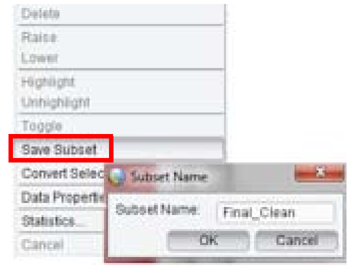


2. Use MB1 to draw a polygon similar to the one shown in the following image. Use MB2 to close the polygon.

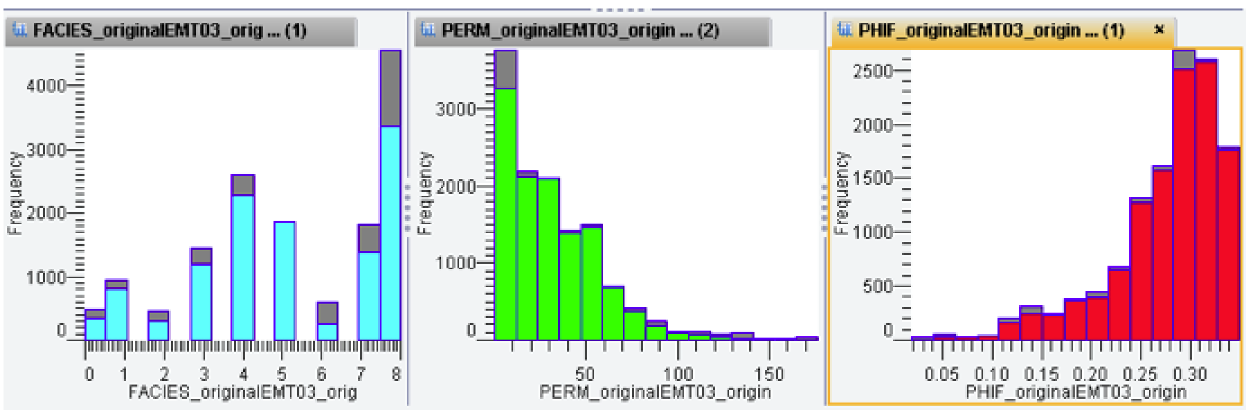


3. Click MB3 and select **Save Subset**.

4. Name the subset Final_Clean. Note that the subset is applied to both the PHIF and PERM data.



5. With your polygon selection in the cross-plot still active, display histograms of the **FACIES**, **PHIF** and **PERM** for the **UNBLOCKED_CLEAN** point set to see what data lies outside of the polygon.



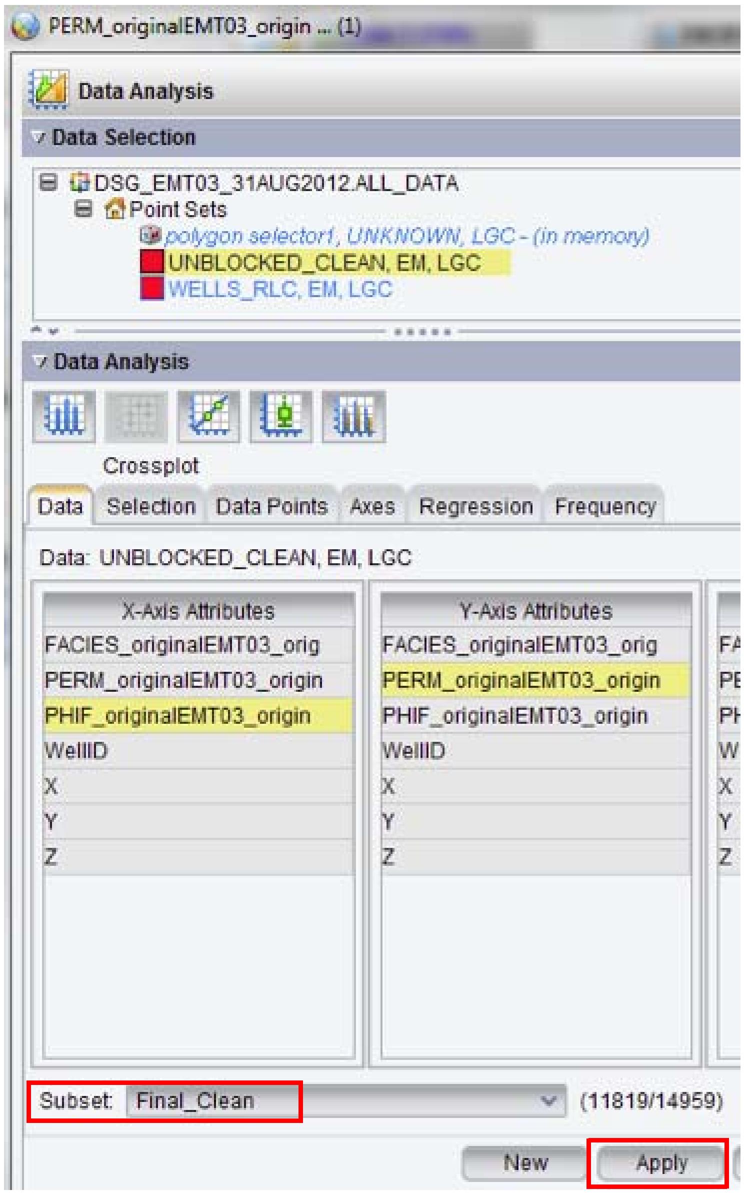
The portion of the histogram bars highlighted in gray corresponds to the data points outside of the polygon selection in the cross-plot.

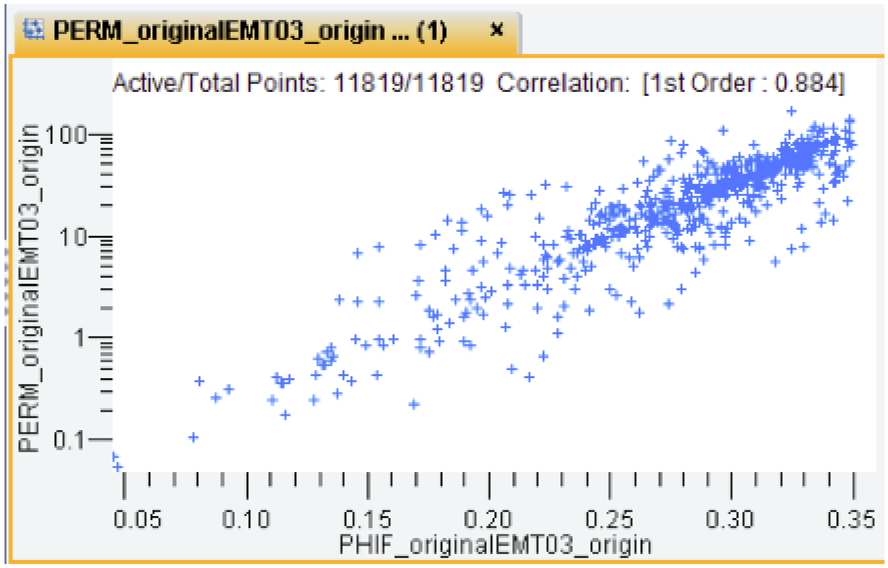
Apply a Subset

Next you can Apply the new subset for the cross-plot display. If the cross-plot with the polygon is not already the active tab, then MB1 on the tab. When a tab is active it turns to an orange color.

1. In the **Data Analysis** panel **Data** tab, click the Subset drop-down menu and select **Final_clean**.

2. Click **Apply**.





Cross-plot PHIF and PERM following data selection using the polygons tool and creation of a new subset.

Create the Final Cleaned Point Set

1. In the **Earth Modeling** task pane, select **Filter Point Set** in the Data Operations task pane.

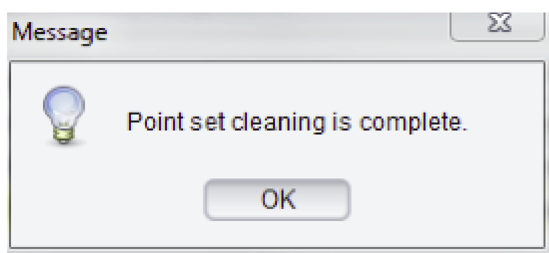
2. For the input point set, select **UNBLOCKED_CLEAN**.

3. Select the properties and subsets as shown.

4. Name the new point set UNBLOCKED_CLEAN_FINAL.

5. Click **Create Point Set**.

6. Click **OK** to dismiss the confirmation dialog box.



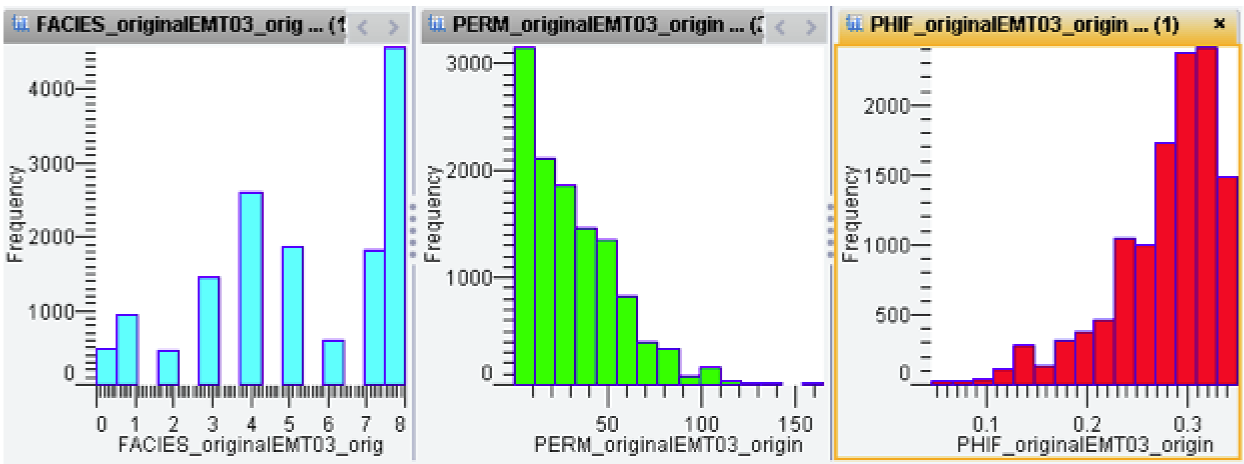
Create Final Histogram Displays

1. Click the **Data Analysis** task pane and select the **UNBLOCKED_CLEAN_FINAL** point set from the Data Selection panel.

2. Click the **Histogram** icon in the Data Analysis panel.

3. Use **CRTL+MB1** to select **FACIES**, **PERM**, and **PHIF** data.

4. Click **New**.



Histograms of the final data following data analysis and outlier removal.

Exercise 3
Variogram Modeling

In these exercise you learn the fundamentals of variogram computation and modeling using a 2D data set. In additional to data analysis, you will learn how to:

- Look for trends
- Compute and model an isotropic variogram
- Use the variogram map to look directional anisotropy
- Model an anisotropic variogram
- Create a nested variogram model

The data is a portion of a data set from a west Texas field released to the public by Amoco Production Company in 1991 for use in training, research and education. We acknowledge and greatly appreciate the use of these data.

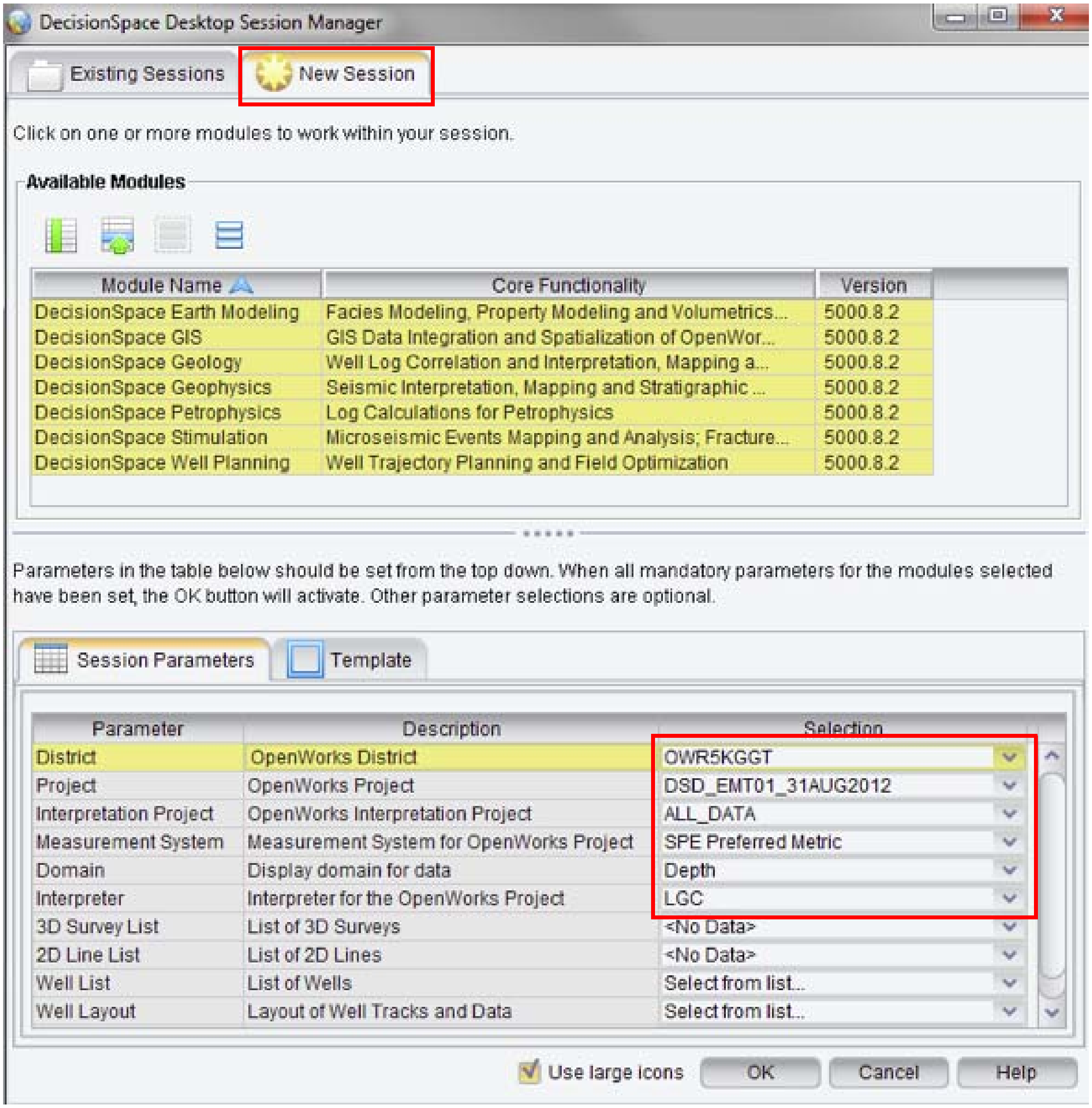
In this exercise you will use interval average porosity from 261 well locations, with wells drilled on a 40-acre 5-spot pattern, the average well spacing is 1320 feet or about 400 meters.

Start a New Session

1. Launch the software and select the **New Session** tab.
2. Select all modules.
3. Enter the following Session Parameters:

District:	**the instructor will indicate the District Name**
Project:	**DSD_EMT01_31AUG2012**
Interpretation Project:	**ALL_DATA**
Measurement System:	**SPE Preferred Metric**
Domain:	**Depth**
Interpreter:	**LGC**

4. Click **OK**.


DecisionSpace Desktop Session Manager
Existing Sessions
New Session
Click on one or more modules to work within your session.
Available Modules
Module Name
Core Functionality
Version
DecisionSpace Earth Modeling
Facies Modeling, Property Modeling and Volumetrics...
5000.8.2
DecisionSpace GIS
GIS Data Integration and Spatialization of OpenWor...
5000.8.2
DecisionSpace Geology
Well Log Correlation and Interpretation, Mapping a...
5000.8.2
DecisionSpace Geophysics
Seismic Interpretation, Mapping and Stratigraphic ...
5000.8.2
DecisionSpace Petrophysics
Log Calculations for Petrophysics
5000.8.2
DecisionSpace Stimulation
Microseismic Events Mapping and Analysis; Fracture...
5000.8.2
DecisionSpace Well Planning
Well Trajectory Planning and Field Optimization
5000.8.2
Parameters in the table below should be set from the top down. When all mandatory parameters for the modules selected have been set, the OK button will activate. Other parameter selections are optional.
Session Parameters
Template
Parameter
Description
Selection
District
OpenWorks District
OWR5KGGT
Project
OpenWorks Project
DSD_EMT01_31AUG2012
Interpretation Project
OpenWorks Interpretation Project
ALL_DATA
Measurement System
Measurement System for OpenWorks Project
SPE Preferred Metric
Domain
Display domain for data
Depth
Interpreter
Interpreter for the OpenWorks Project
LGC
3D Survey List
List of 3D Surveys
<No Data>
2D Line List
List of 2D Lines
<No Data>
Well List
List of Wells
Select from list...
Well Layout
Layout of Well Tracks and Data
Select from list...
Use large icons
OK
Cancel
Help

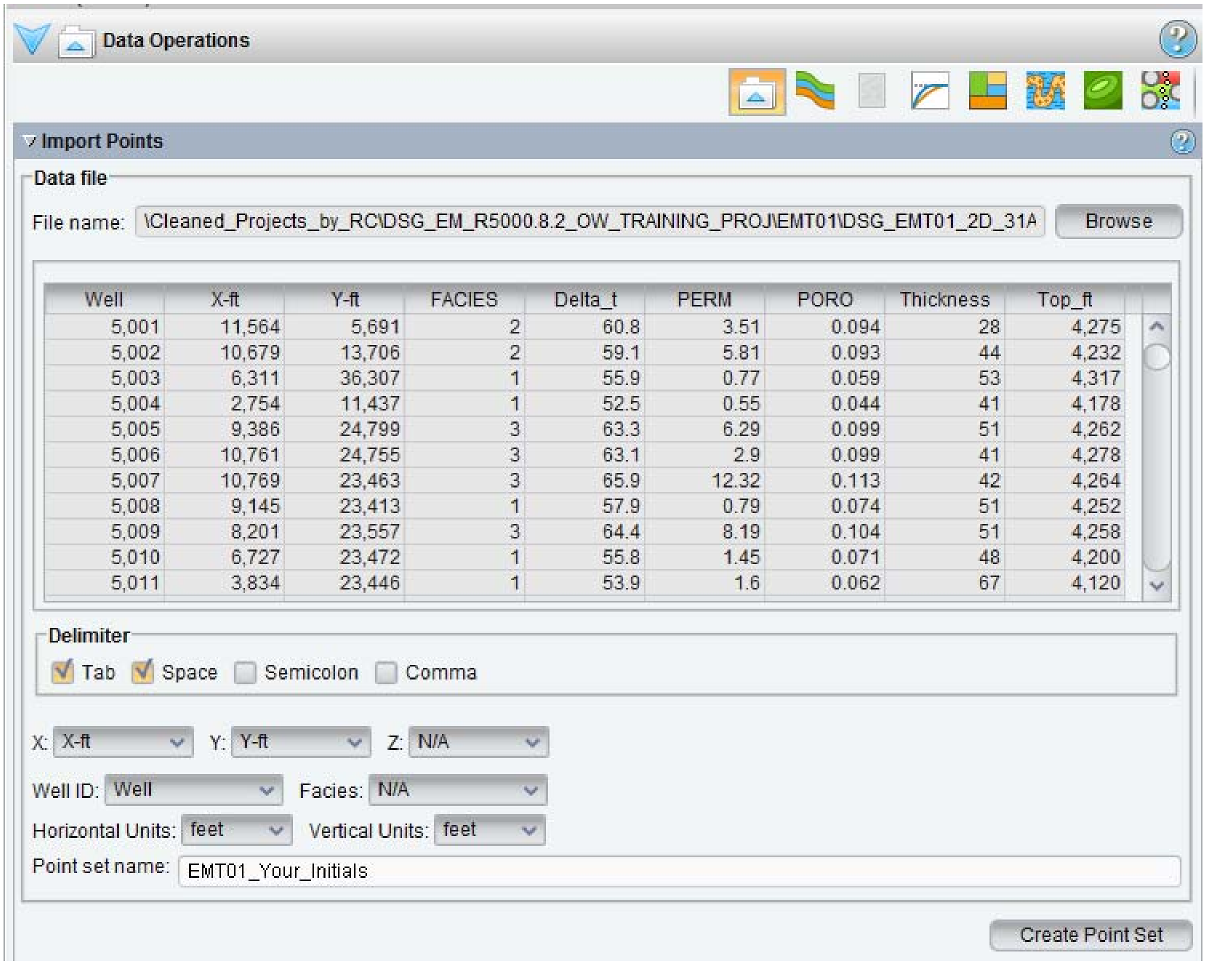
Load an ASCII File and Create a Point Set

1. The Data Operations task pane should be open.
2. Select the **Import Points** subtask pane.
3. Click **Browse** to select the **DSG_EMT01_2D_31AUG2012.txt** file. Ask the instructor if you cannot find it.
4. Select the corresponding fields.

X:	**X-ft**
Y:	**Y-ft**
Z:	**N/A**
Well ID:	**Well**
Facies:	**N/A**
Horizontal Units:	**feet**
Vertical Units:	**feet** (not really applicable as data are 2D)

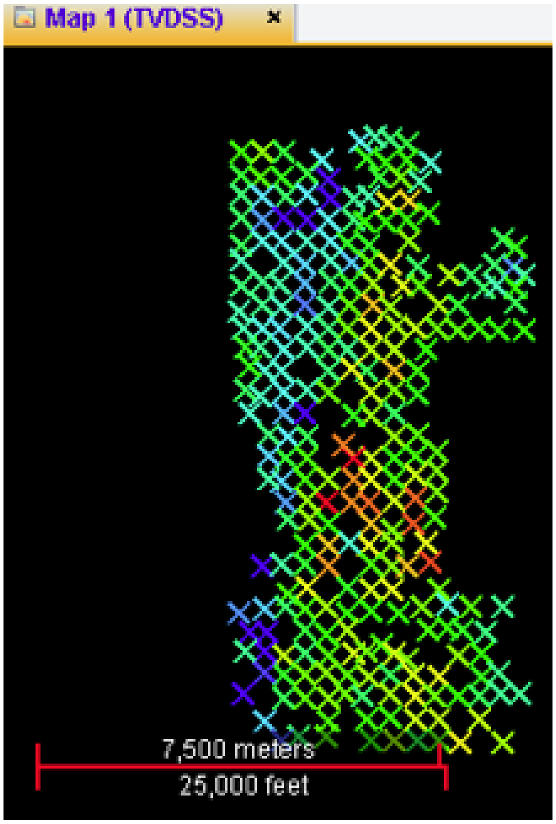
5. Name the point set EMT01 and add your initials.
6. Click **Create Point Set**.

7. Click **OK** on the Importing data complete message.



Display Porosity in a Map View

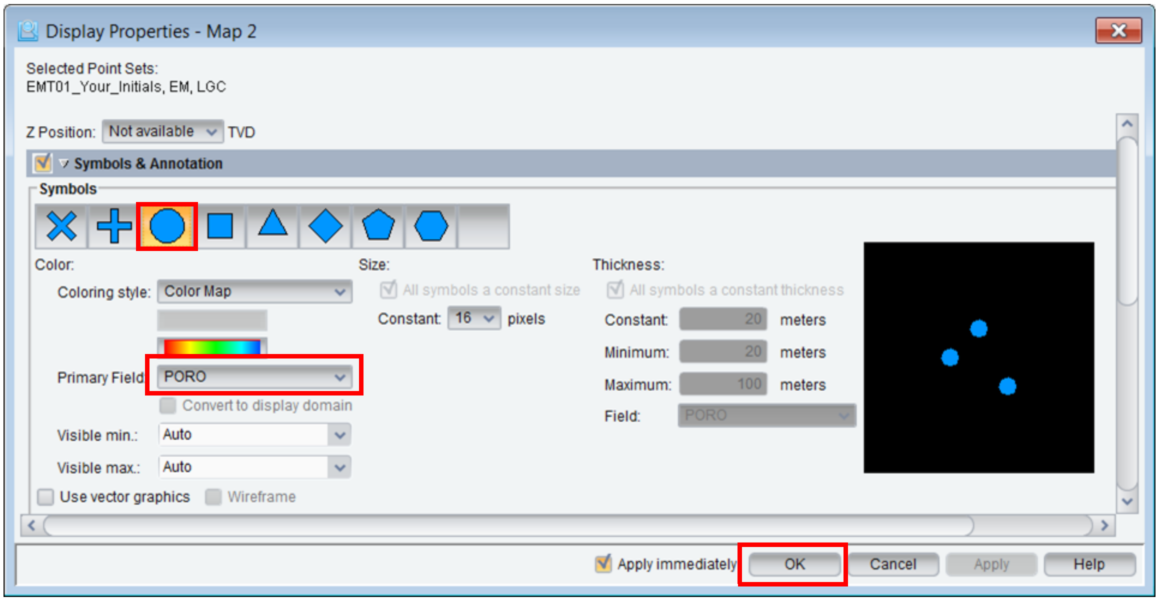
If a Map View tab is already available the new point set data should be visible and looks similar to this:



This shows the well locations, but we want a display of the porosity.

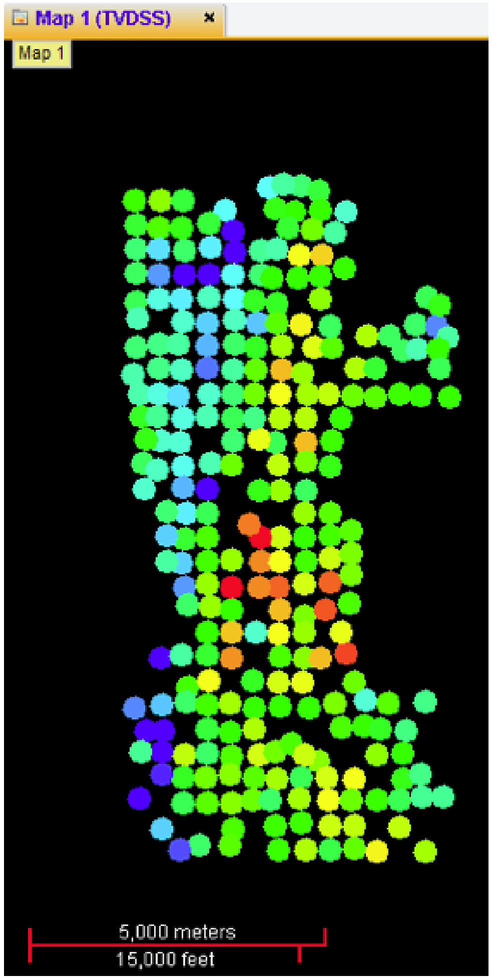
1. Click MB3 on the **EMT01 Point Set** in your inventory and select **Display Properties**.

2. Enter the parameters shown in the following image.



3. Click **OK**.

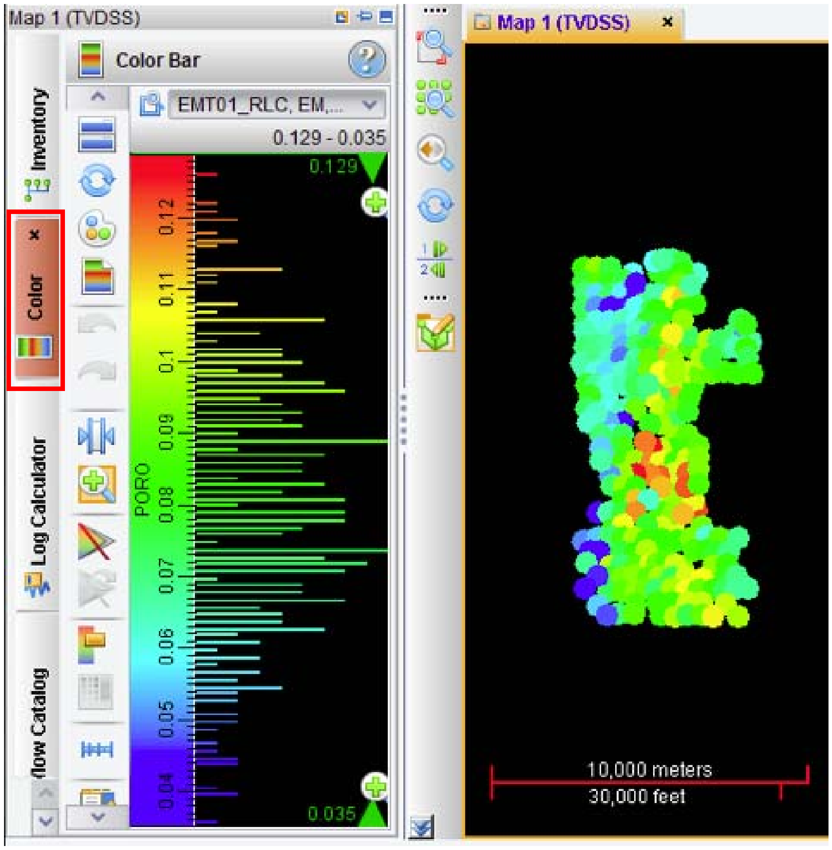
Your display should look like the following.



Observations:

- There is a North-to-South zone of low porosity (blues).
- There is a slight trend from low to high porosity from west to east.
- The highest porosity is near the center of the field (reds and oranges).

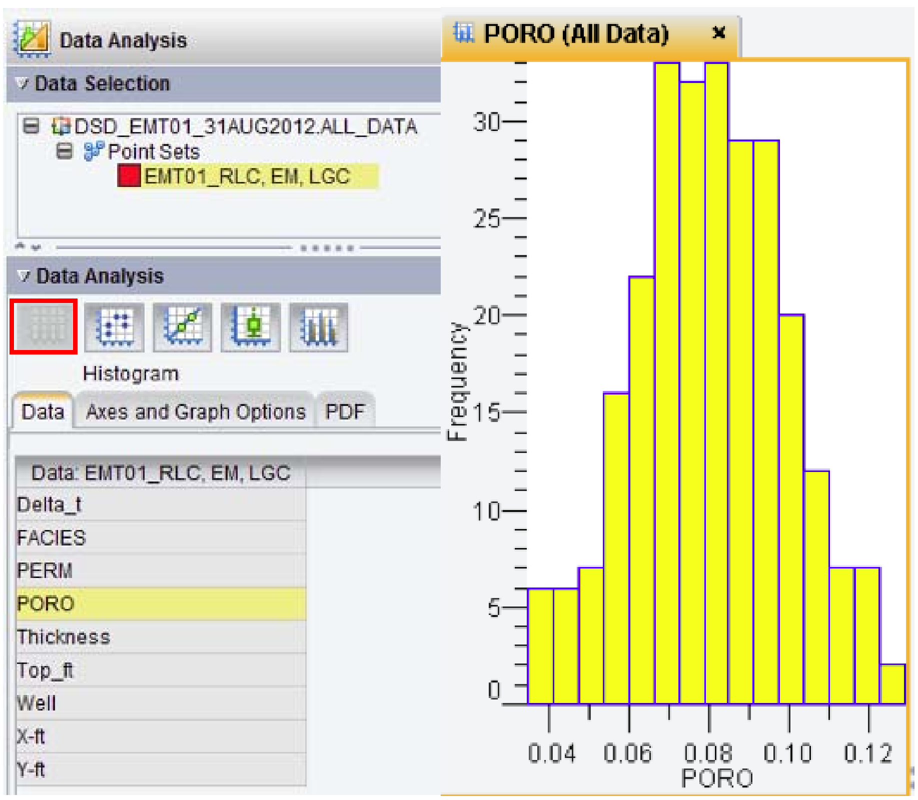
4. Click the **Color** Bar tab to display a color scale with a histogram of the porosity values.



Exploratory Data Analysis of Porosity

Histogram Display

1. Go to the **Data Analysis** task pane.
2. Click the **EMT01** point set in the data selection panel.
3. Click the **Histogram** icon.
4. Select **PORO** from the table.
5. Click **New**.

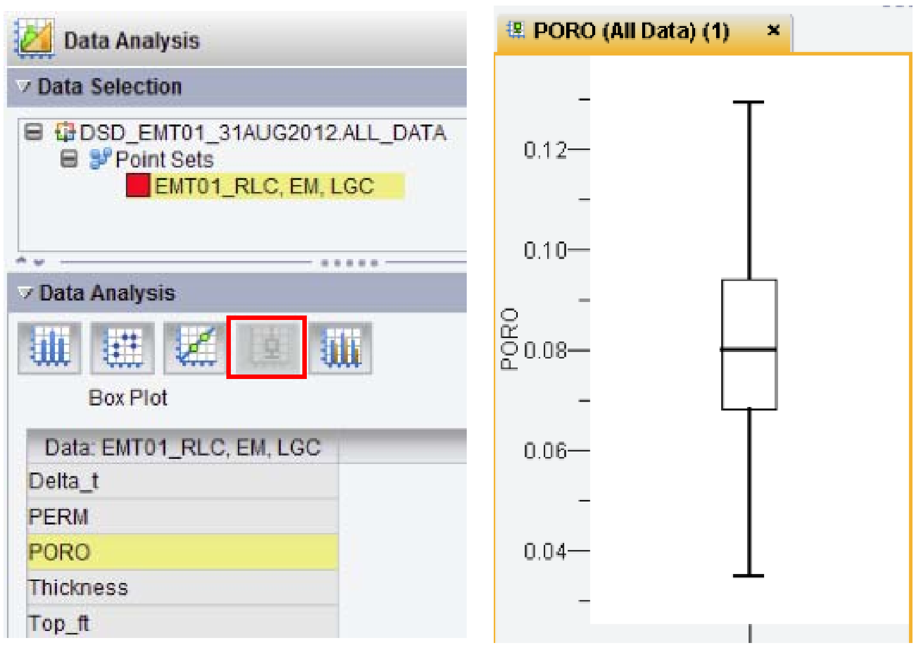


The most obvious observation is that the porosity has a very symmetrical distribution.

Box Plot

1. Select the **Box Plot** icon from the Data Analysis task pane.
2. Select **PORO**.
3. Click **New**.

The resulting box plot should look like the following image.



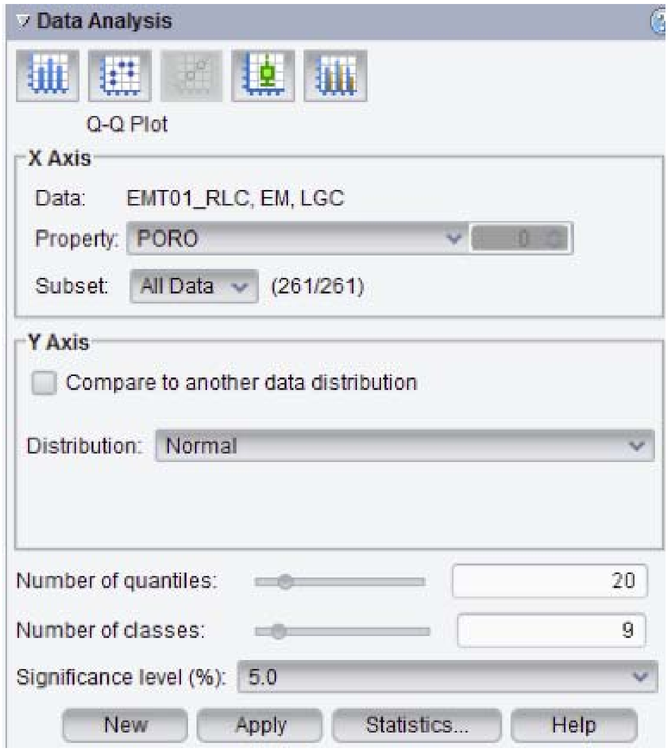
The box plot also looks very symmetrical.

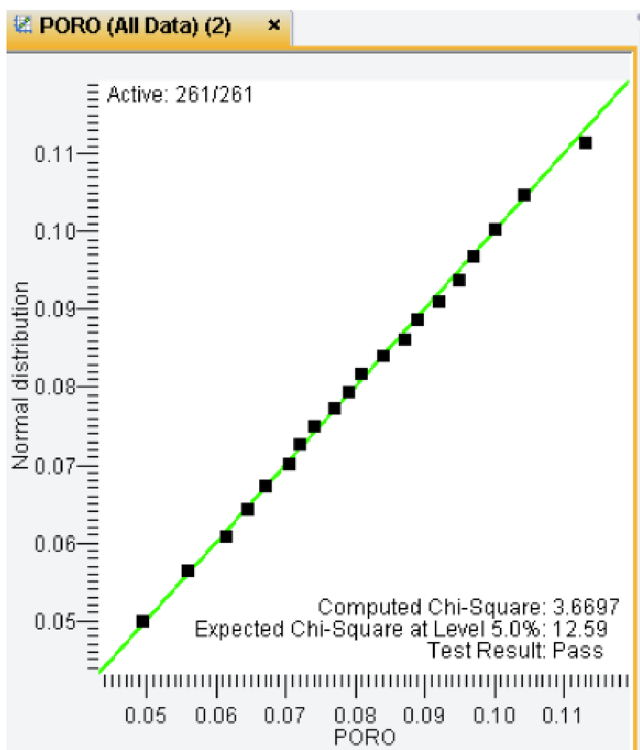
Q-Q Plot

Now we will do a Q-Q plot to test whether the porosity has a normal distribution.

1. In Data Analysis click the **Q-Q Plot** icon.
2. Select **PORO**.
3. Distribution: **Normal** (default).
4. Default other parameters.

5. Click **New**.





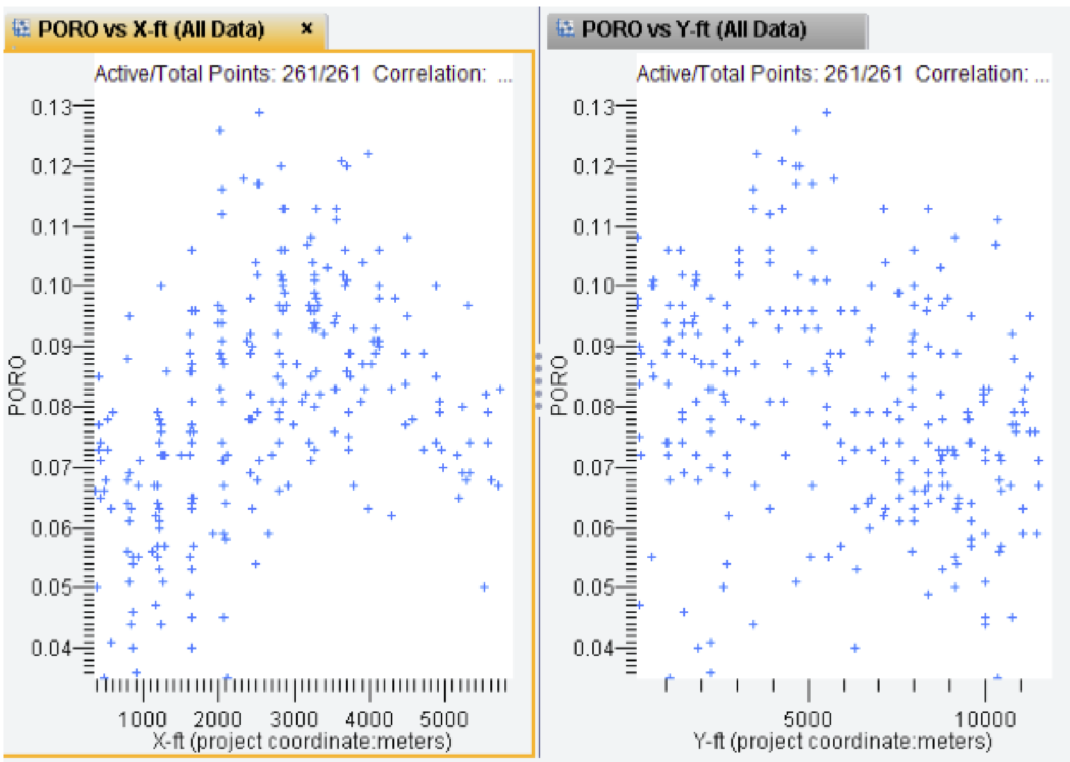
The Chi^2 test passes at the 95% confidence level, so there is no reason to reject the hypothesis that porosity does not have a normal distribution.

Look For Trends in the Porosity

Because variography requires second-order stationarity, we can use a simple cross-plot of porosity against the X and Y coordinates to look for possible trends. We already observed what appears to be a west-east trend in the Map View display.

1. Make a cross-plot of **X-ft** (X-axis) and **PORO** (Y-axis).

2. Make a similar plot with the Y-ft.



When porosity is plotted against the X-ft direction (left image), we observe porosity increasing in magnitude until about 3500 meters and then it decreases further east. This trend was also seen in the Map View. No trend is seen in the north-to-south direction when porosity is plotted against the Y-ft coordinate. For this data set the trend is mild and the slight non-stationarity should not require detrending the data before variogram modeling. We can confirm this conclusion after we compute the omni-directional variogram.

Variogram Computation and Modeling

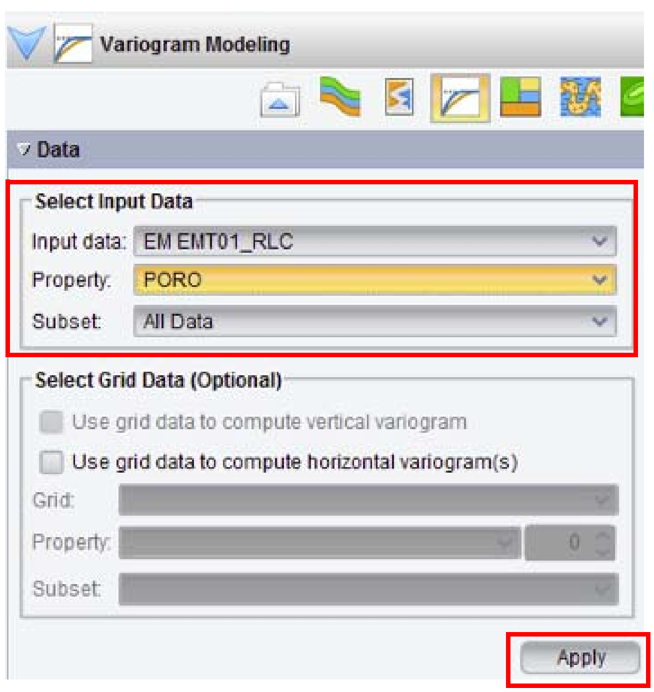
Next you learn how to define the computational parameters to compute and then to model the omni-directional variogram. This is an isotropic model which considers all azimuths to compute the average experimental variogram.

The omni-directional variogram should always be created first because it uses all the data and provides valuable information about the:

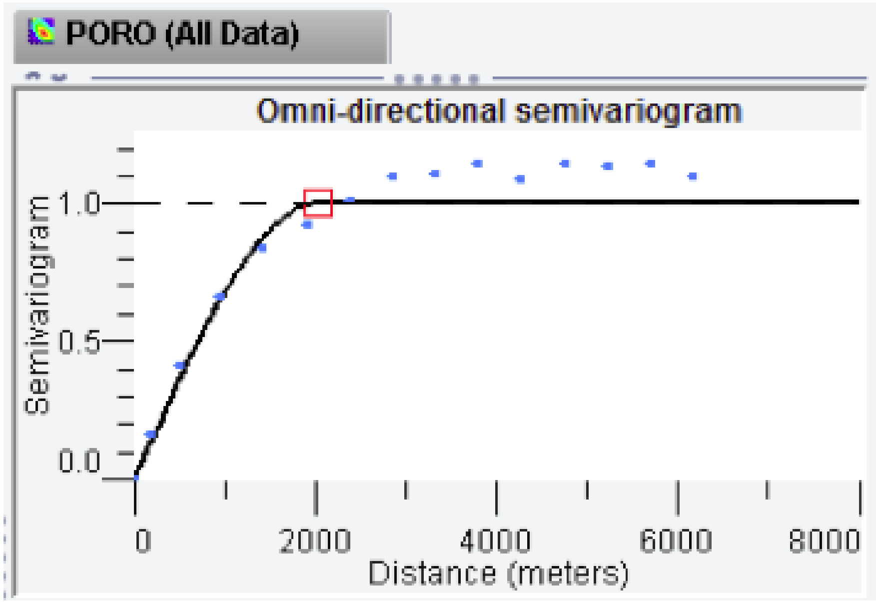
- Average spatial scale
- the variogram model type (e.g., spherical, exponential, etc.)
- and the presence of a nugget.

Isotropic Model

1. Go to the **Earth Modeling** task pane.
2. Click the **Variogram Modeling** icon. 
3. Fill-in the Select Input Data section as shown.



4. Click **Apply**.

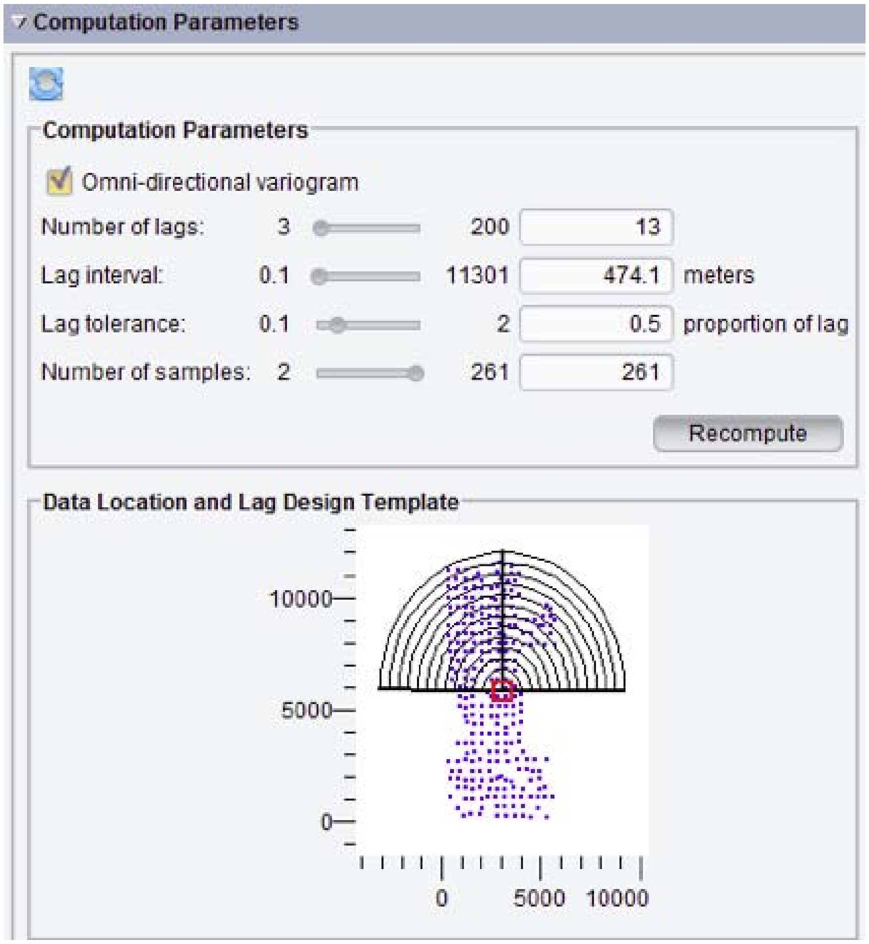


The omni-directional variogram is computed and modeled with a spherical variogram model automatically. The scale, located at the red square, is determined when the variance reaches one.

Note that the sill is always set to 1 regardless of whether it's the raw data or transformed using a Normal Score transform. This allows the variogram models to be used in the kriging or simulation algorithms.

Our conclusion that the non-stationarity is not a problem is confirmed as the experimental variogram is bounded because it does reach a sill at about 3000 meters. However, the local sill is slightly above the normalized data variance (dashed, horizontal line), but nevertheless, it does flatten. It does not display the characteristic parabolic upward shape of an experimental variogram exhibiting a trend.

5. Click the **Computation Parameters** subtask pane to view the default computational parameters. We won't make any changes yet.



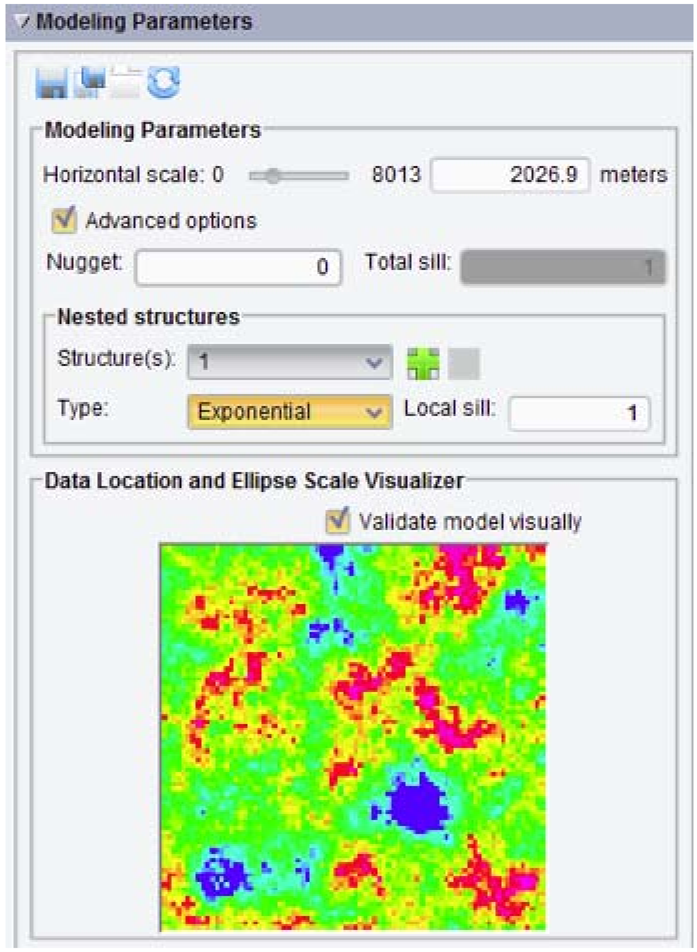
This panel controls how the experimental variogram is calculated based on defaults determined after clicking Apply. The defaults are based on the input data, so for each data set the defaults differ. How are the defaults determined?

The Data Location and Lag Design Template image shows the well locations and the search bins used in the computation. The semi-concentric circles are 474.1 meters apart; the separation based on the **Lag interval**. This is the average distance between the wells. The **Number of lags** is determined dividing 60% of the maximum well separation by the lag interval. The **Lag tolerance** is typically ± 50% of the lag interval.

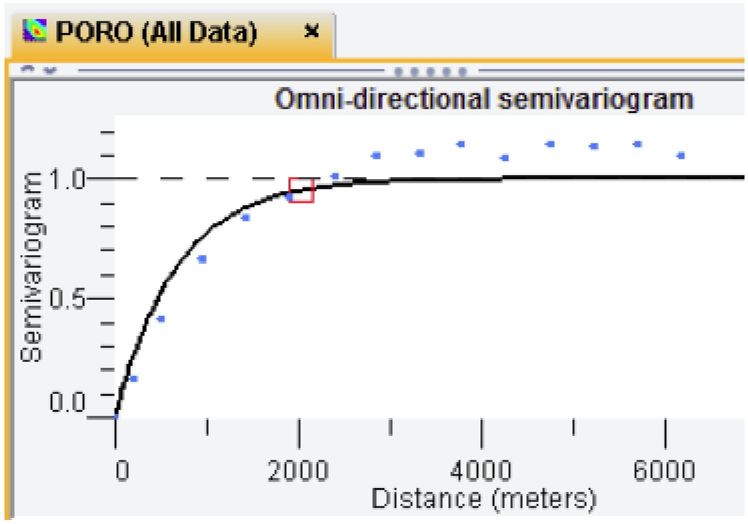
Observe that the default fitted spherical variogram model doesn't really match the shape of the experimental variogram, so select a different model type.

6. In the Modeling Parameters subtask pane, check **Advanced Options** and change the type to **Exponential** which is fit automatically to the experimental variogram.

7. Select the **Validate model visually** checkbox to display an unconditional simulation.



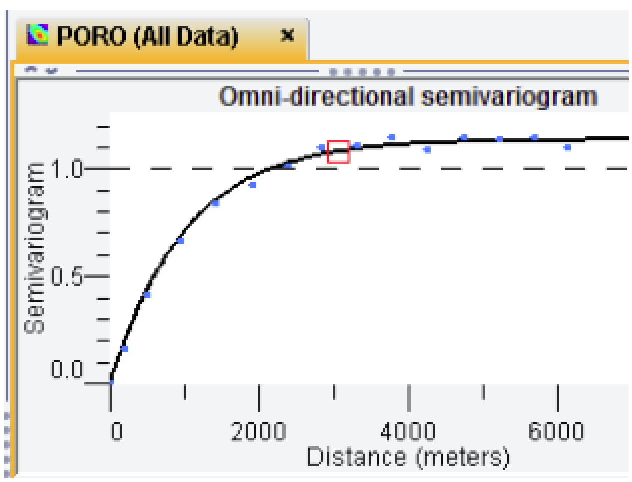
The Horizontal scale fitted automatically is about 2027 meters. The Nugget is 0. The unconditional simulation provides a preview of a simulation result based on the current variogram model parameters. You will use this image more when modeling an anisotropic variogram.

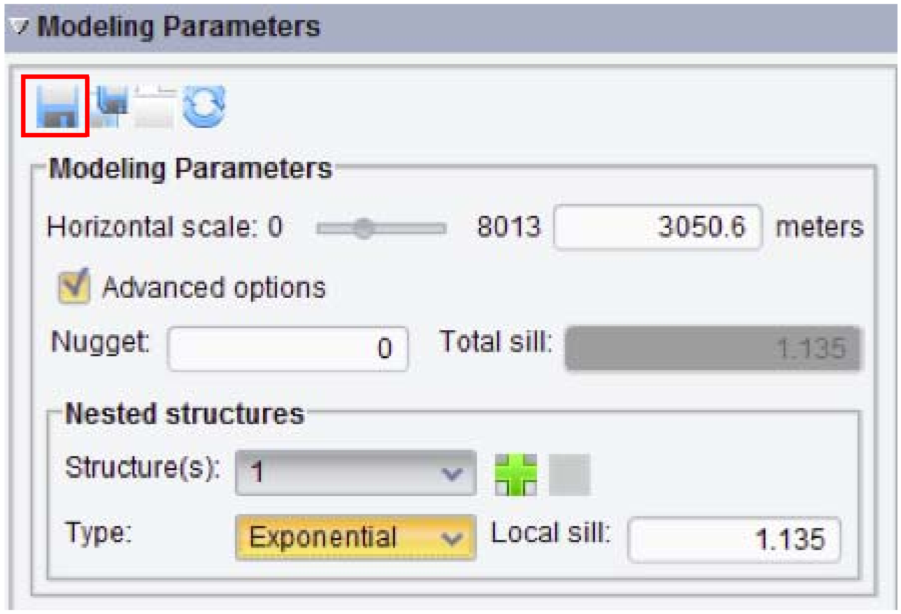


Experimental variogram fit with a scale of 2027 meters. Next you will learn how manually adjust and finalized the model.

8. With MB1, click and drag the red box to the approximate location shown in the image or enter the values for Horizontal scale and Local sill as shown in the following image.

   While dragging the red anchor point, the unconditional simulation image updates to reflect the new horizontal scale of about 3051 meters, in this example.



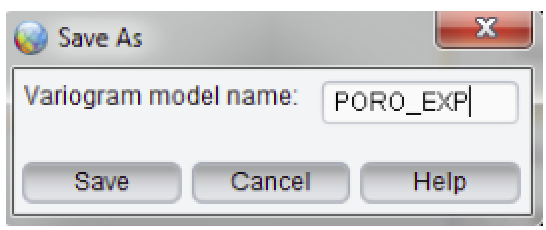


Notice that the local sill is greater than 1.0. When the model is save, the sill is automatically reset to 1.0, but the modeled range is kept. We do this to keep a valid model for use in conditional simulation algorithms. Recall that the sill would have no impact on the values interpolated when using kriging.

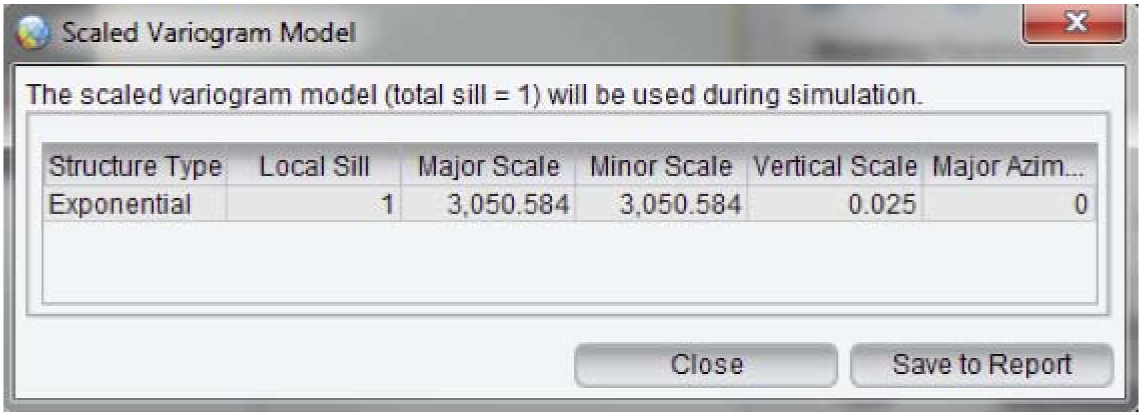
9. Click the **Save** icon (save the active variogram model).

10. Name the variogram **PORO_EXP** and add your initials.

11. Click **Save**.

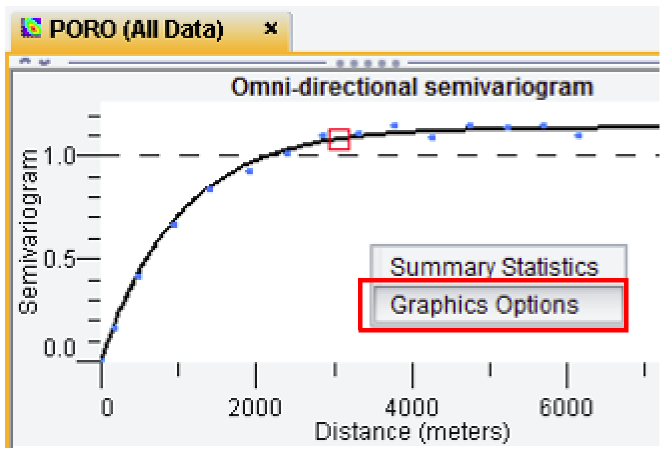


12. Click **Close** to dismiss the Scaled Variogram Model confirmation window.

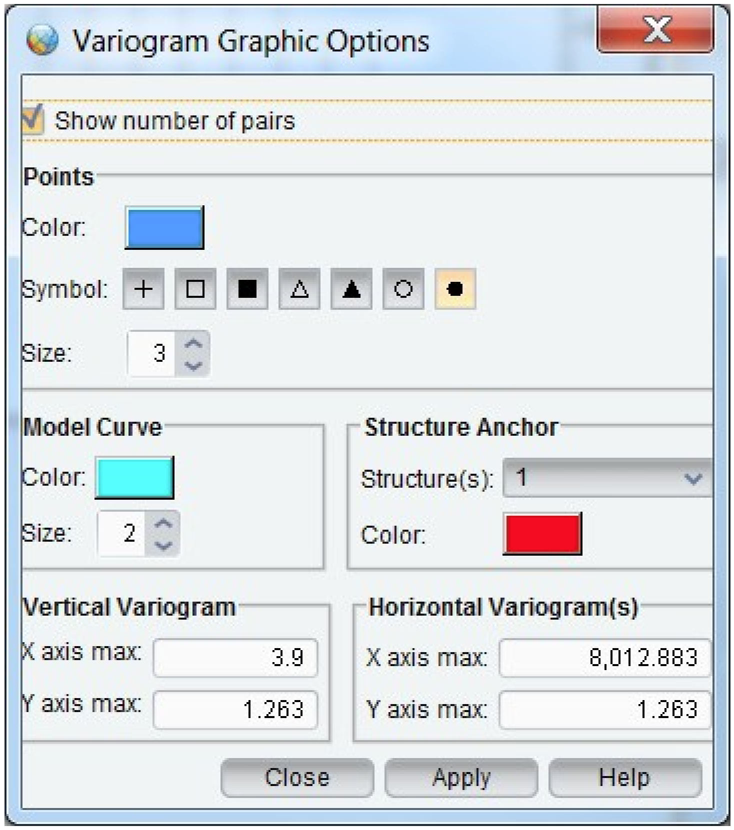


13. Save the session as DSD_VARIO_your_initials_01.

14. Click MB3 on the Variogram display and select **Graphics Options**.

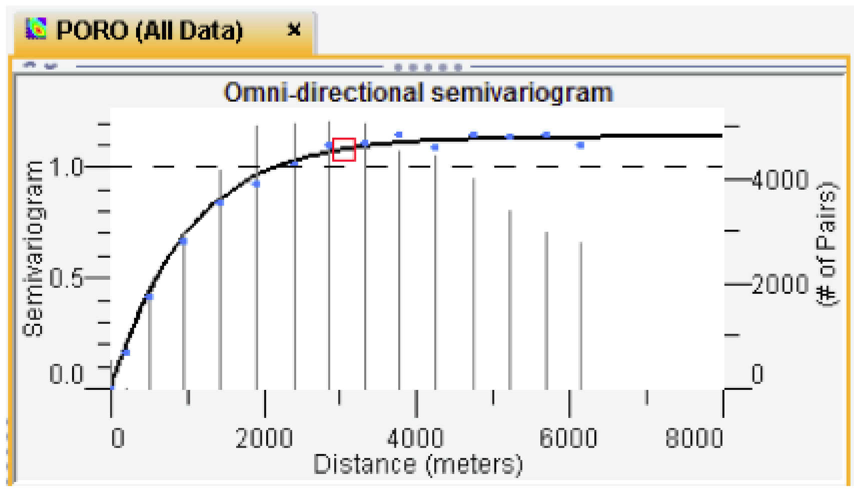


15. Experiment with the graphics options to find the display parameters you prefer.



16. **APPLIED** on **Show number of pairs** to display the number of pairs at each lag as a simple histogram.

When toggled on, gray vertical lines appear on the variogram display. The height of the gray vertical lines indicates the number of pairs in each bin. The distance between these lines may vary due to the fact that these lines are on the same point on the X axes as the variance points which are plotted at the center of gravity of the lag ± the lag tolerance.



17. Click **Close** when you are finished.

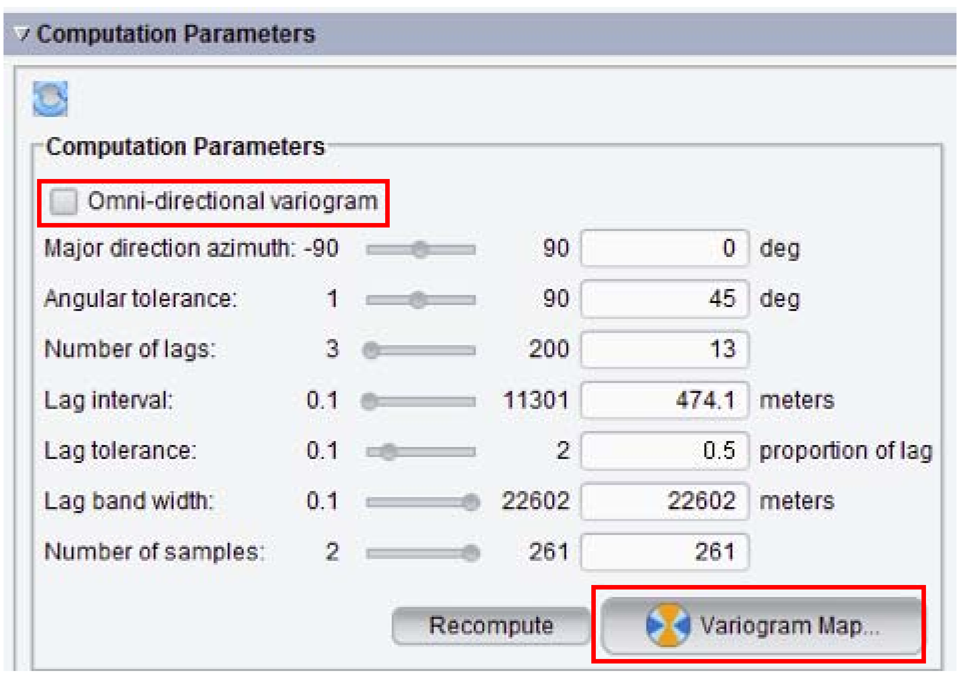
Anisotropic Modeling

The omni-directional variogram modeling provided valuable information:

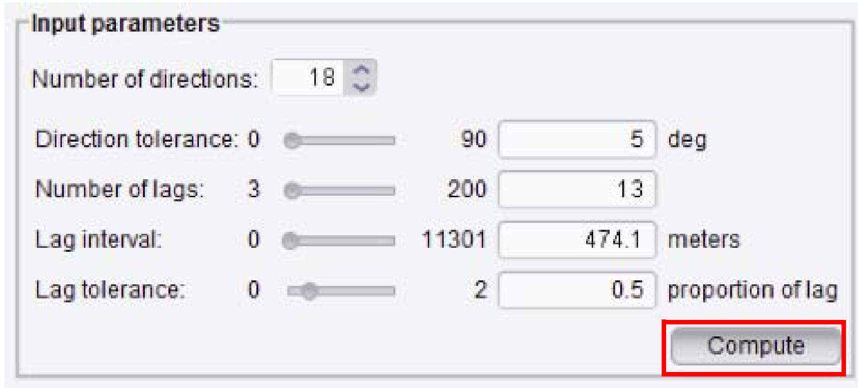
- There is no Nugget.
- The average horizontal scale is about 3050 meters.
- The exponential model fits the experimental points best.

We could use this model to create a map of porosity using the kriging algorithm or create multiple realizations using a simulation algorithm. However, most geological data have some degree of directional anisotropy. Next, you will learn the tools to determine if anisotropy is present in the porosity data and how to do the modeling.

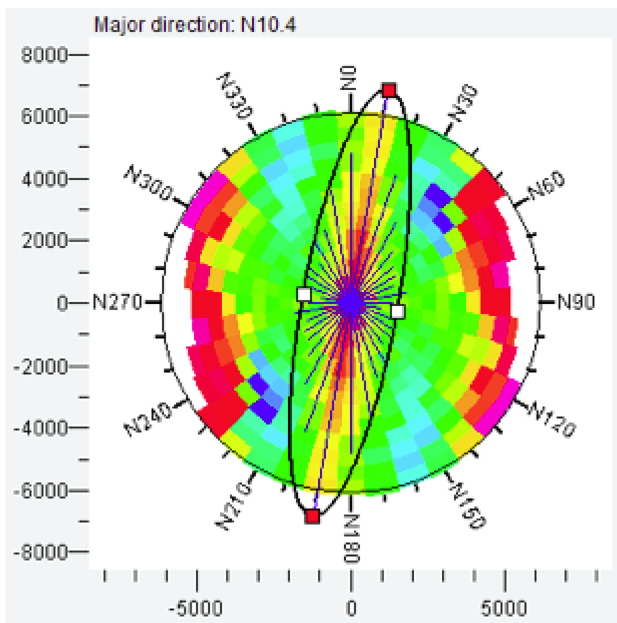
1. Click the **Computational Parameters** panel.
2. Uncheck the **Omni-directional variogram** checkbox.
3. Click **Variogram Map**.



4. Click **Compute** to create the Variogram Map (polar plot).

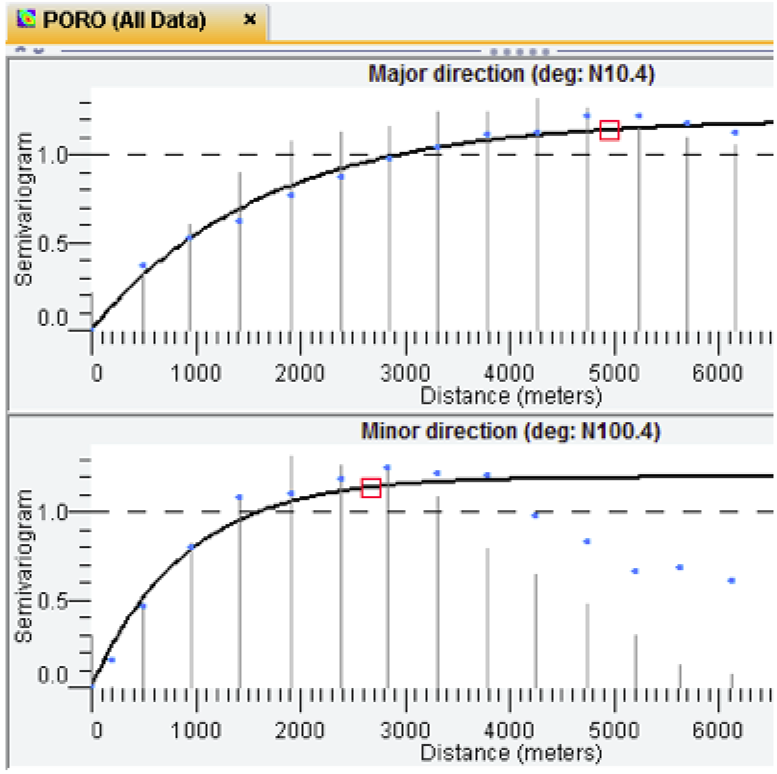


5. On the Variogram Map, click the red or white boxes with MB1 to adjust the ellipse as shown. The major direction of continuity is about N10E.



6. Once adjusted, click **OK** to transfer the parameters back to the Computation Parameters subtask pane.

7. In the Modeling Parameters section try fitting the directional variogram using a single structure and an Exponential model type.



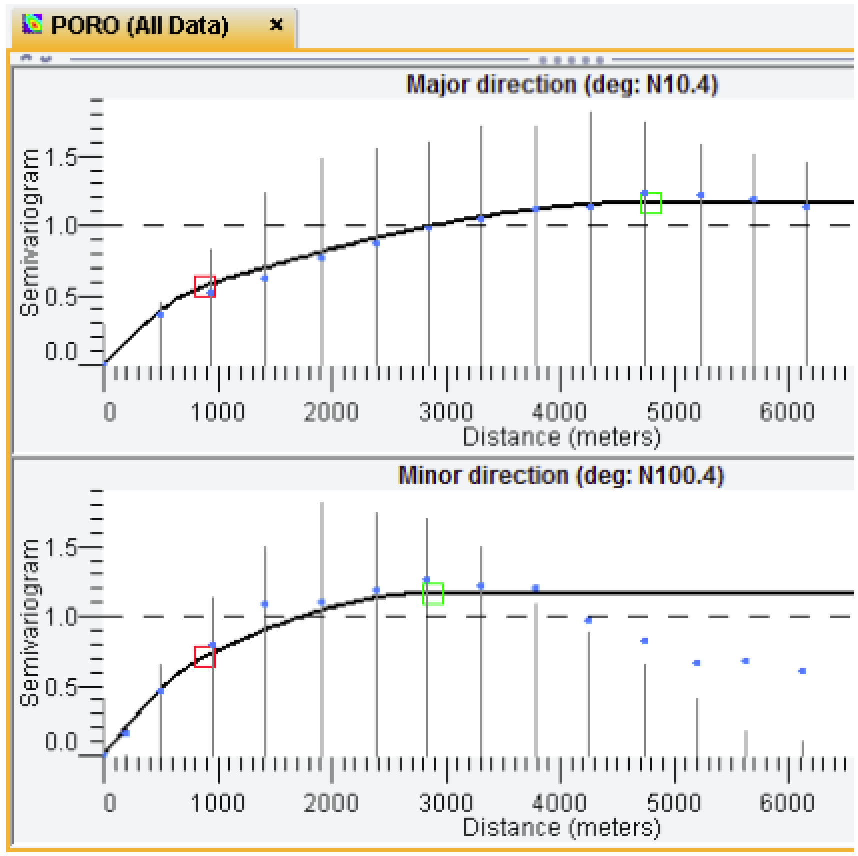
This is an excellent fit and we could save this model and use it in kriging or conditional simulation. But we will model this experimental variogram using a nested model composed of two different scale spherical models.

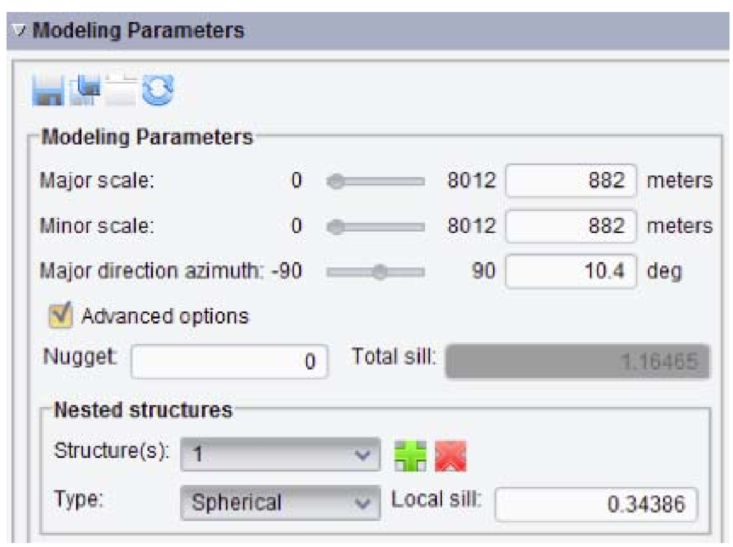
8. Change Structure Type 1 back to **Spherical**.

9. Click the **green plus icon** in the Modeling Parameters panel to add a second structure.

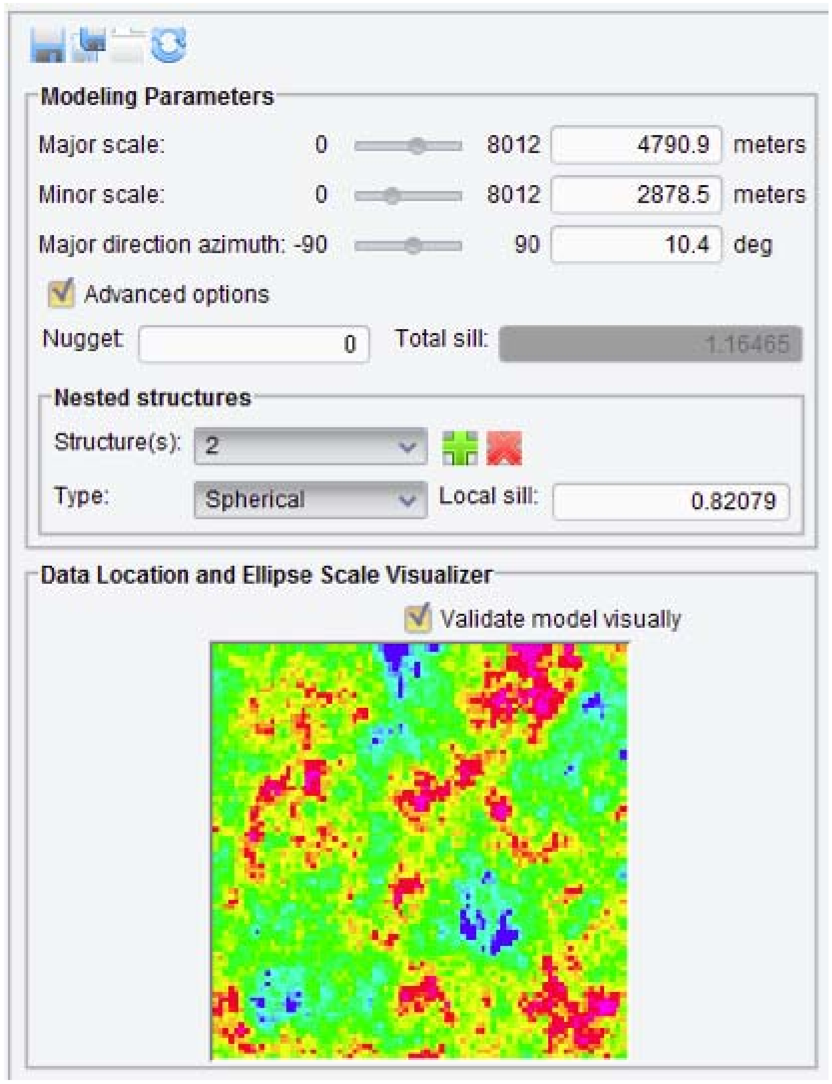
10. Change the Type for each structure to **Spherical**.

11. Now adjust each structure until you obtain a satisfactory nested modeled. You could enter the values shown below.





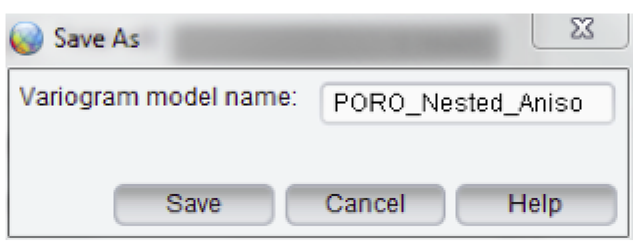
Structure 1 parameters



Structure 2 parameters and an unconditional simulation reflecting the anisotropic, nested variogram model.

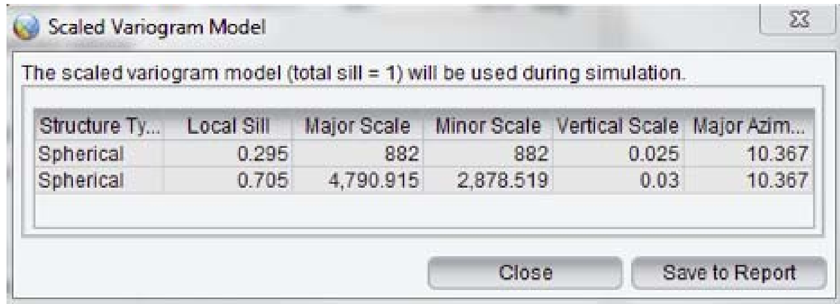
12. Click the **Save active variogram with a new name** icon. 

13. Name the Variogram model PORO_Nested_Aniso.



14. Click **Save**.

15. Click **Close** to dismiss the confirmation dialog box.



16. Save the session.

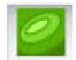
Exercise 4
2D Property Modeling

In this exercise you learn the fundamentals of 2D property mapping using geostatistical algorithms. You learn how to:

- Create an empty 2D grid.
- Populate the grid with porosity using a saved variogram model with
    - — Ordinary kriging
    - — Turning Bands simulation
- Display the results in Map View.
- Use additional Data Analysis QC displays.

Create an Empty 2D Grid

1. If the previous session on Variogram Modeling is not open, then open that session.

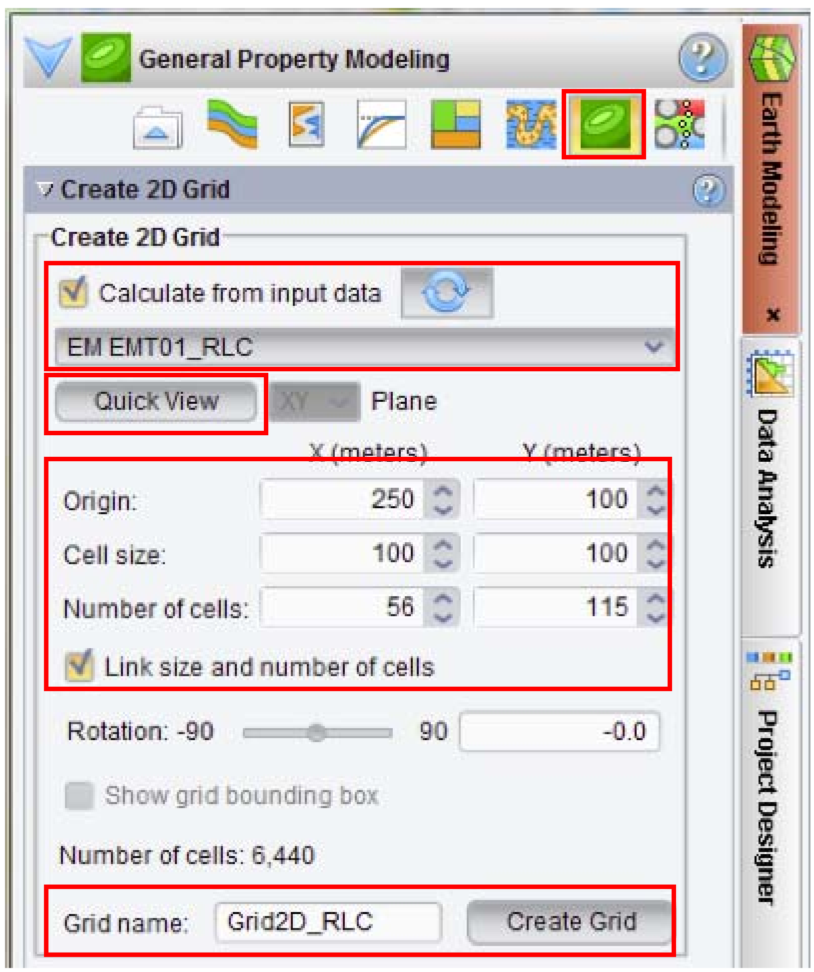
2. Click the **General Property Modeling** icon located in the Earth Modeling tab. 

3. Expand the **Create 2D Grid** subtask pane.

4. Turn on the **Calculate from input data** checkbox.

5. Select **EMT01** in the drop-down menu.

6. Enter the parameters as shown. Be sure to have the **Link size and number of cells** checkbox selected.

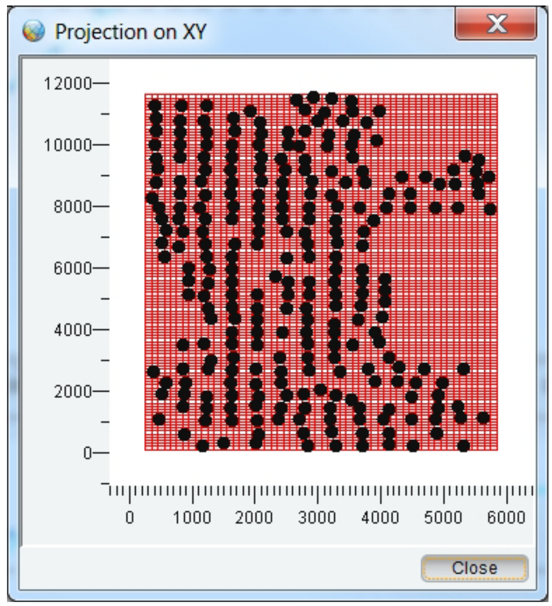


7. Click **Quick View** to see a simple display of the data and to check to see if the grid contains the data locations.

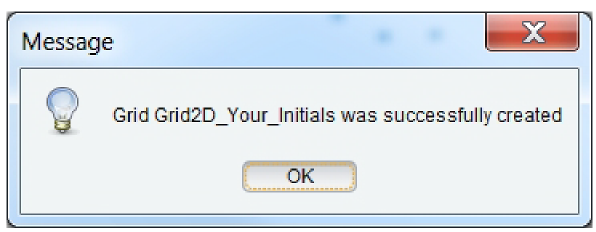
8. Name the grid Grid2D and add your initials.

9. Click **Create Grid**.

   The Quick View shows where the wells are located with respect to the 2D grid.



10. Click **OK** to dismiss the confirmation window.



Populate the Grid with Porosity

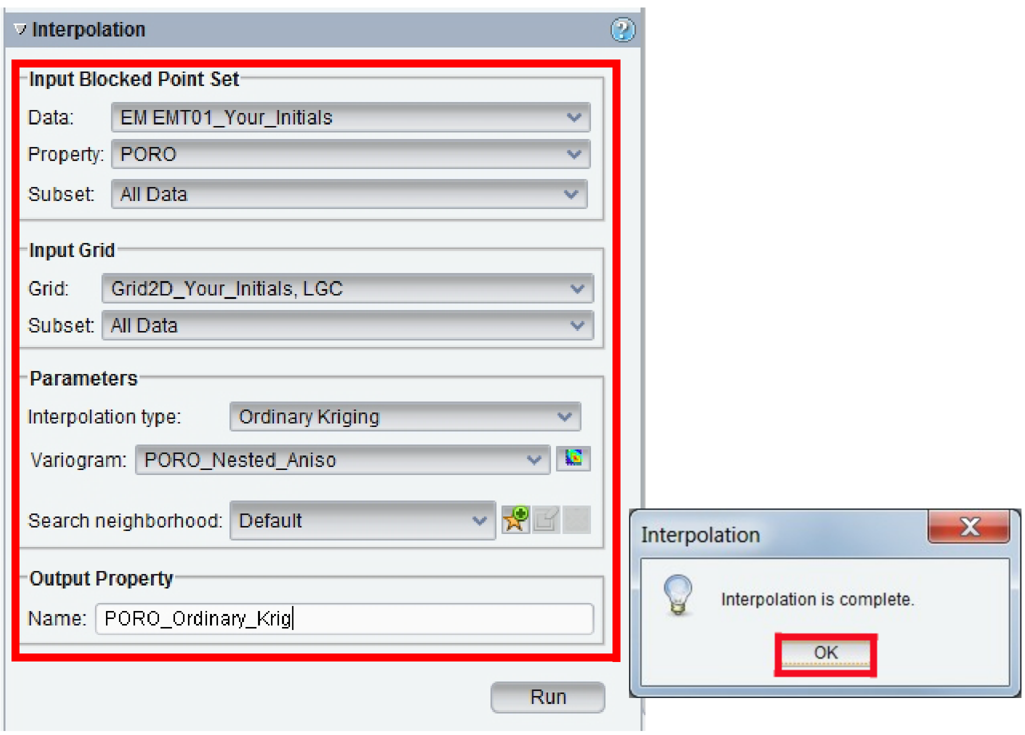
In the next set of exercises, you will learn how to use the variogram model using kriging and conditional simulation algorithms and use data analysis tools to compare and contrast the results.

Ordinary Kriging

In this exercise, you use the saved nested variogram model with the ordinary kriging option which uses the mean value based on the data in the local search neighborhood. The default neighborhood uses the 100 porosity values closest to the current target grid node.

1. Expand the **Interpolation** subtask pane.

2. Enter the input and parameters as shown.



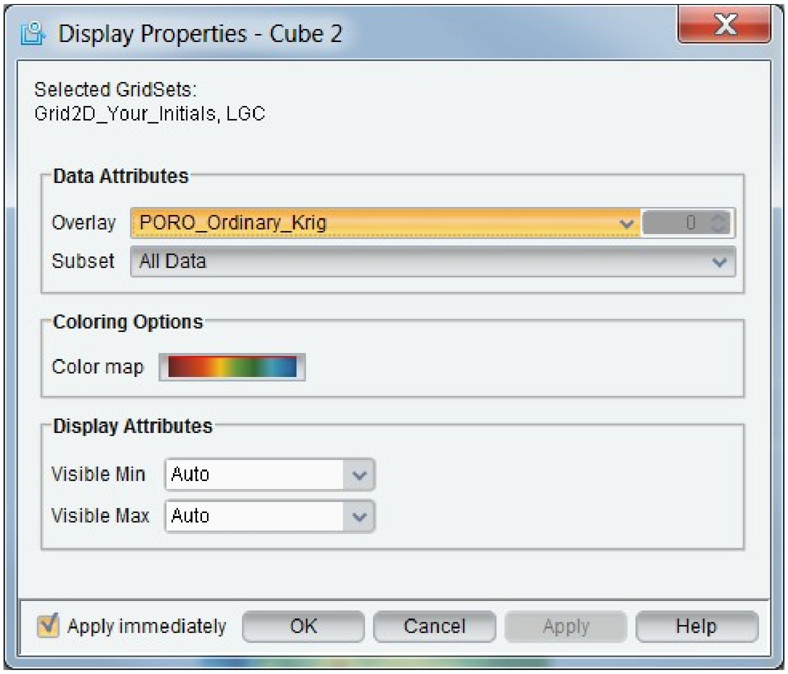
3. Name the output PORO_Ordinary_Krig.

4. Click **Run**.

5. Click **OK** to dismiss the confirmation dialog box.

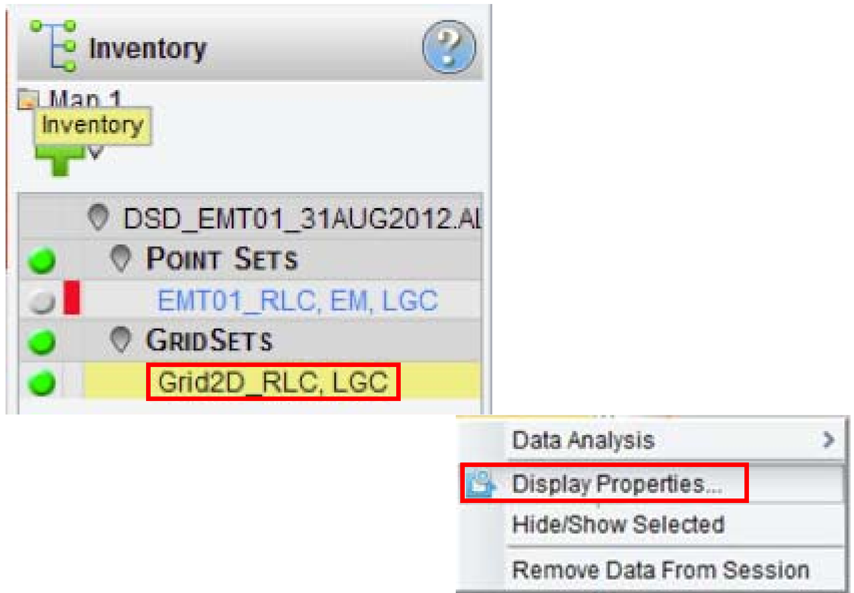
Display the Results in a Map View

1. Activate a Map View. If the point set is currently displayed in the Map View turn them off from the Inventory.



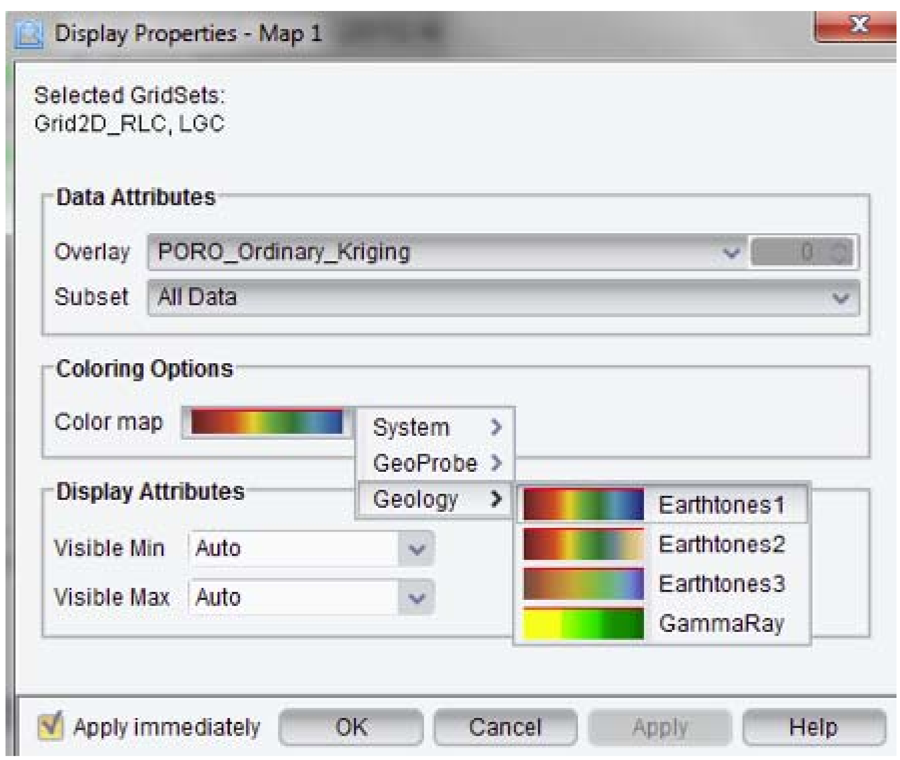
2. From the Inventory task pane turn on your newly created grid.

3. Click MB3 on the grid in the inventory and select **Display Properties** from the drop-down menu.



4. In the Display Properties editor, change Overlay to **PORO_Ordinary_Krig**.

5. For Color map select **Geology** and then **Earthtones1**.

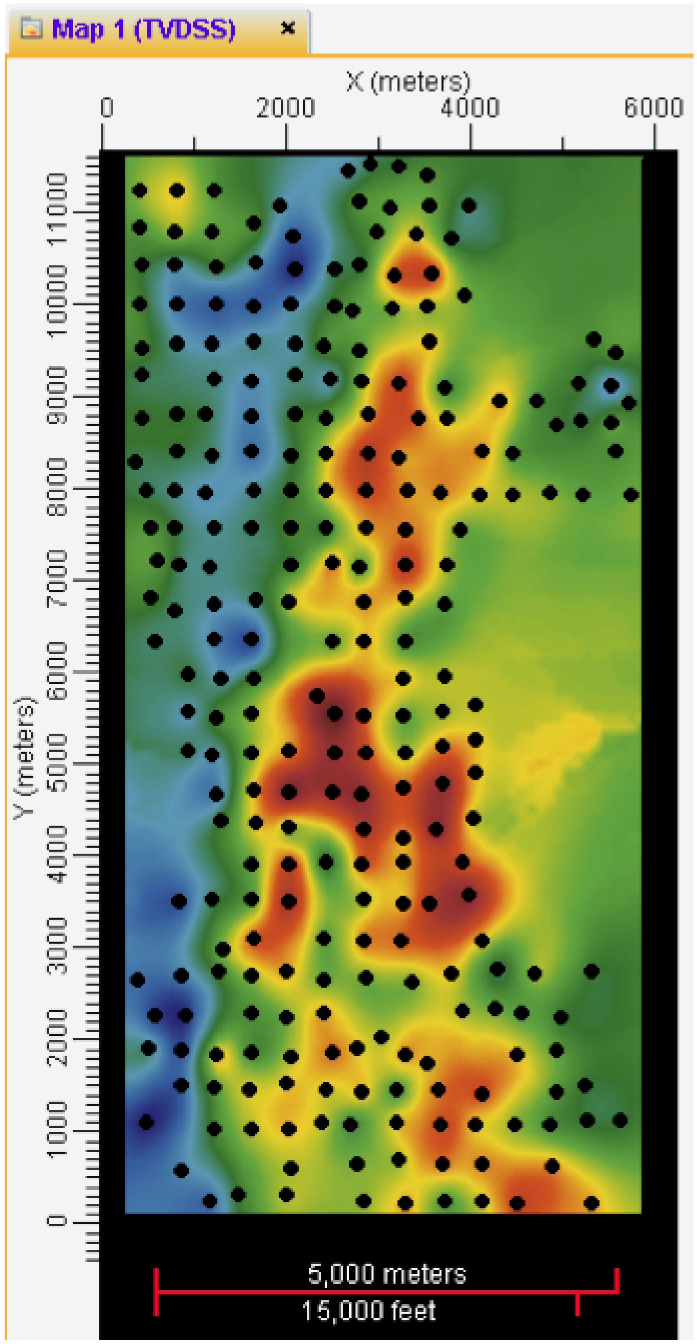


6. Click **OK**.

7. To post the well locations on the porosity map, go to the point set in the Inventory and MB3 on the point set name to select the Display Properties from the drop-down menu.

8. Change the parameters as shown.

9. Click **OK**.



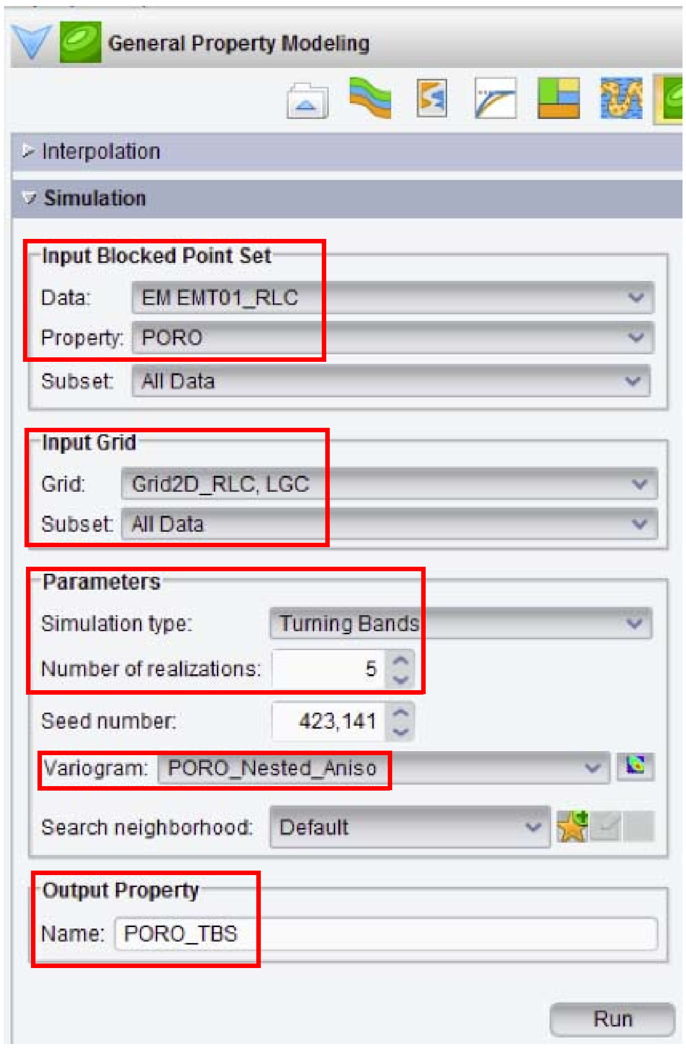
Porosity map using ordinary kriging and a nested, aniotropic variogram model.

Conditional Simulation

Next, you will use the Turning Bands conditional simulation algorithm to create five porosity realizations, create a display, and then compare a simulation result to the ordinary kriging result using a Q-Q plot and a multi-histogram.

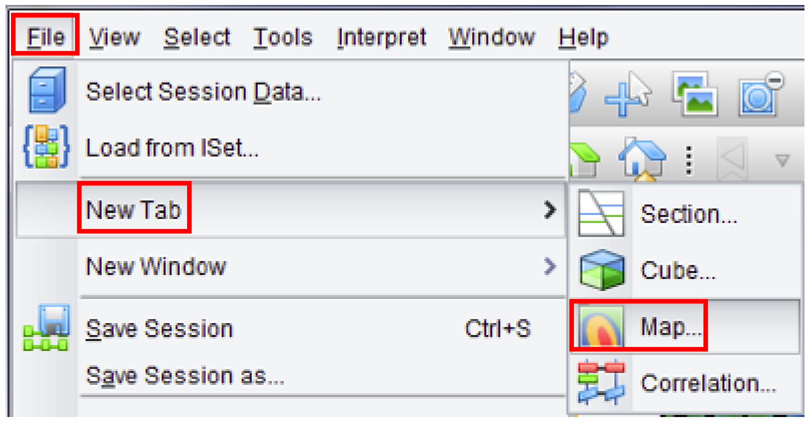
1. Return to General Property Modeling and open the **Simulation** subtask pane.

2. Fill in the subtask pane as shown.



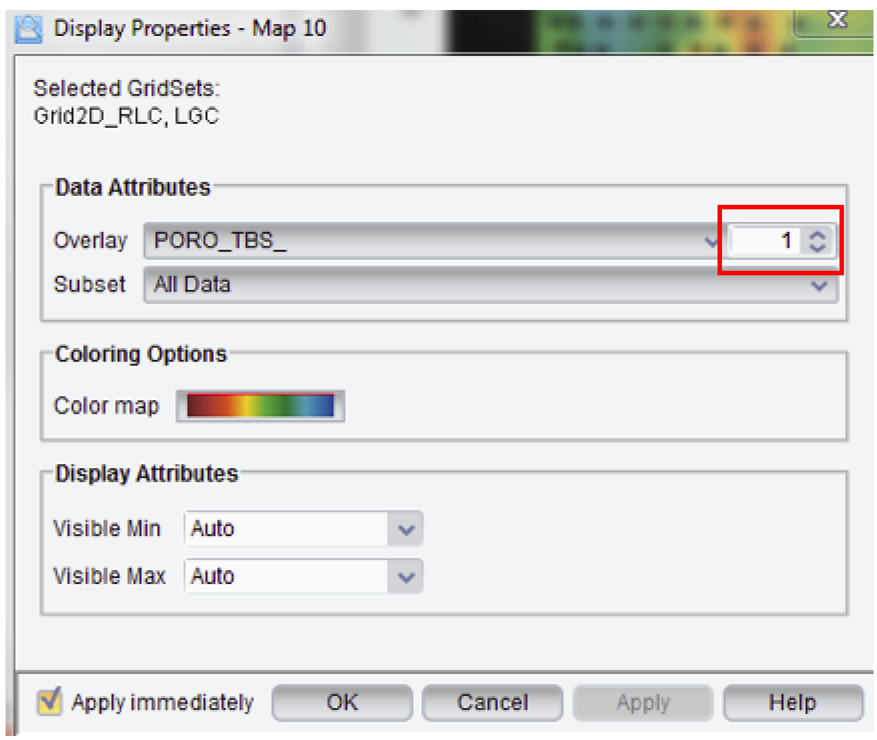
3. Click **Run**.

4. Open a new Map View from **File** > **New Tab** > **Map**. This action should display the porosity based on ordinary kriging. If this new Map View tab is not the active tab, click the tab to make it active.

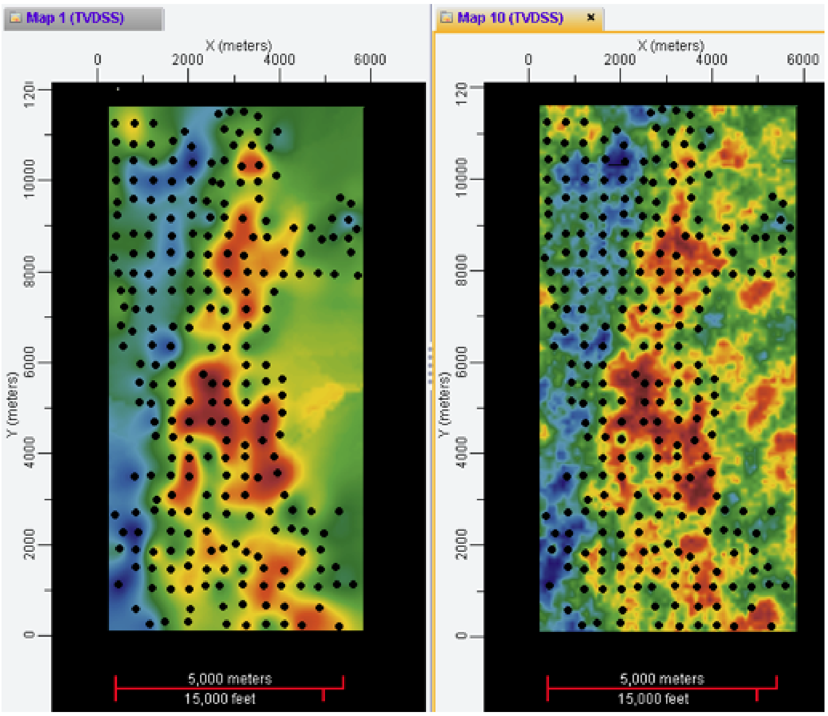


5. In the Inventory, MB3 on the Grid2D and select **Display Properties**.

6. Change the Overlay to **PORO_TBS**.



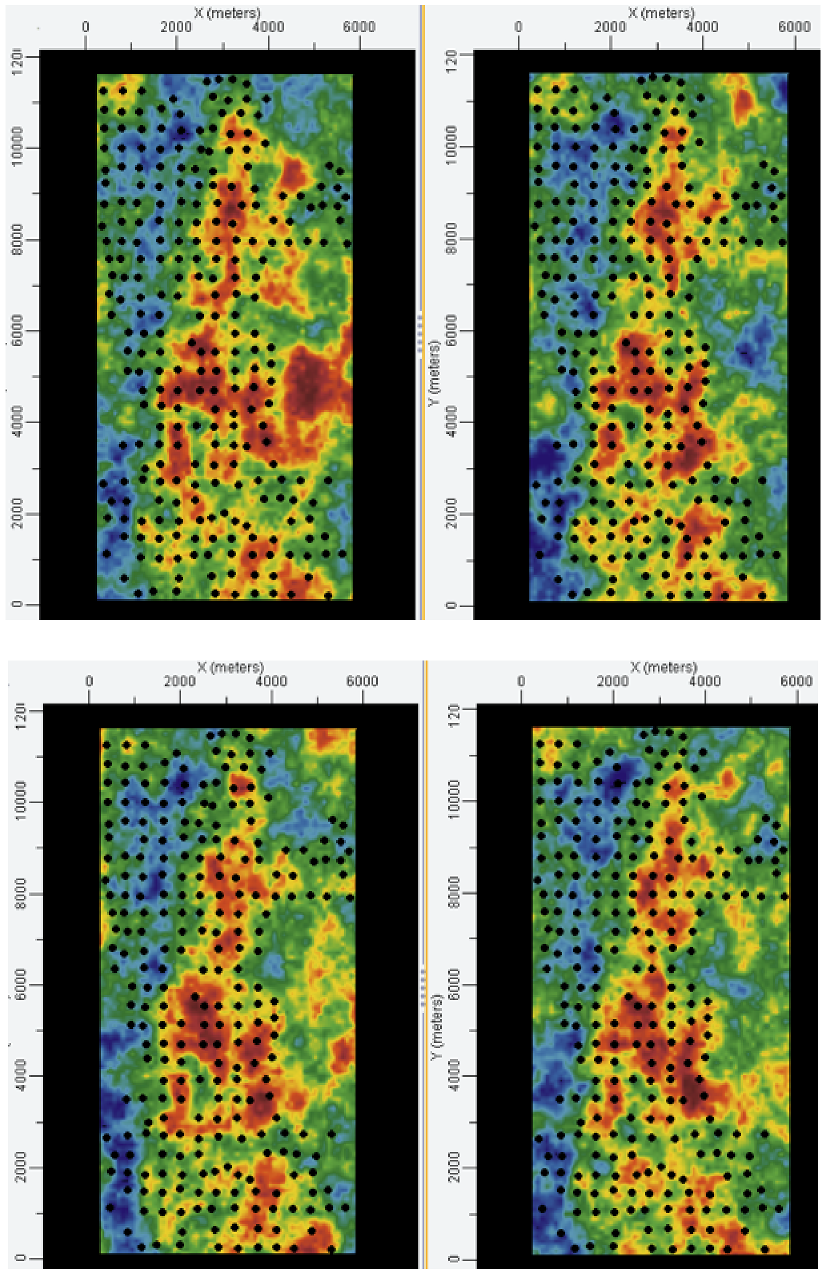
Notice that there is a spinner to the right of the Overlay. The default is 1, which is the first realization. PORO_TBS is the base name for a macro-variable containing all of the realizations created, which is five in this example. Clicking the arrows displays the next or previous realization.



Comparison of ordinary kriging (left) to Turning Bands realization 1 using the same nested variogram model and the same default neighborhood.

- How much difference do you see between the different realizations?
- Where do they differ?
- Where are the results similar?
- What might this indicate about uncertainty in the results?

Realization 2 Realization 3

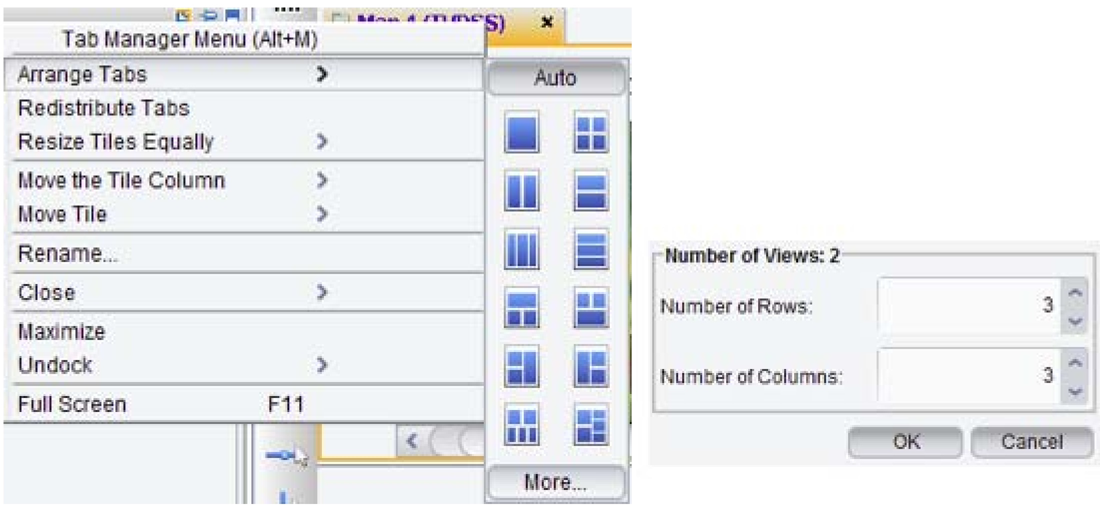


Realization 4 Realization 5

Comparative Data Analysis

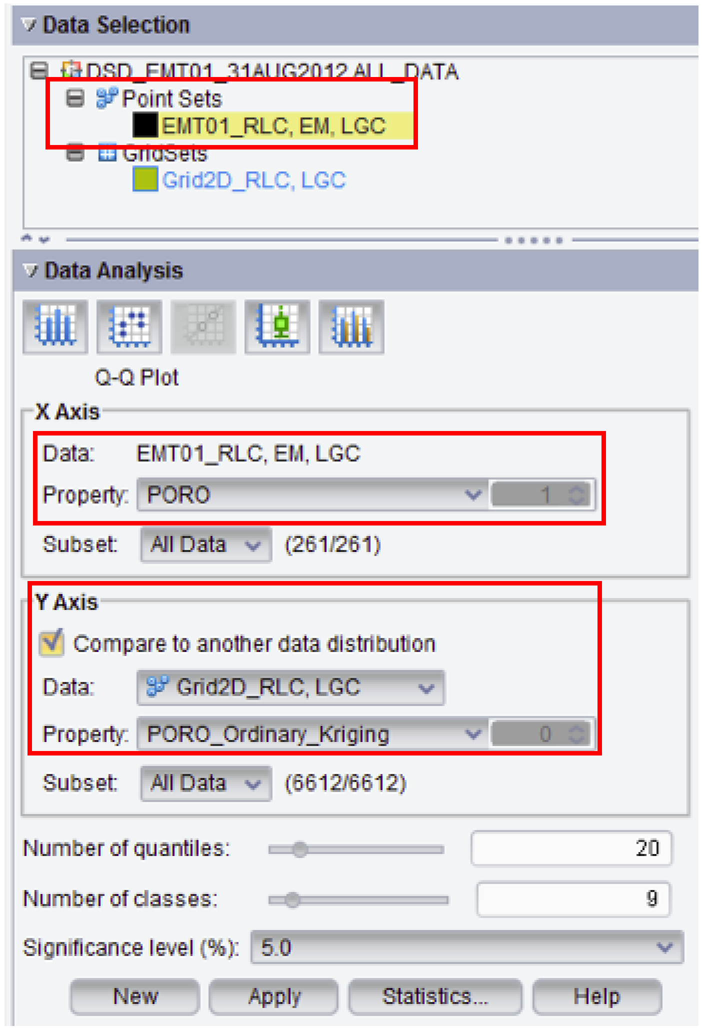
Next, you will use some Data Analysis displays to compare and contrast the results.

1. Create multiple tab views as you did in the Data Analysis exercise by using MB3 on any tab view and then select **Arrange tabs > More** and create a 3 by 3 view.

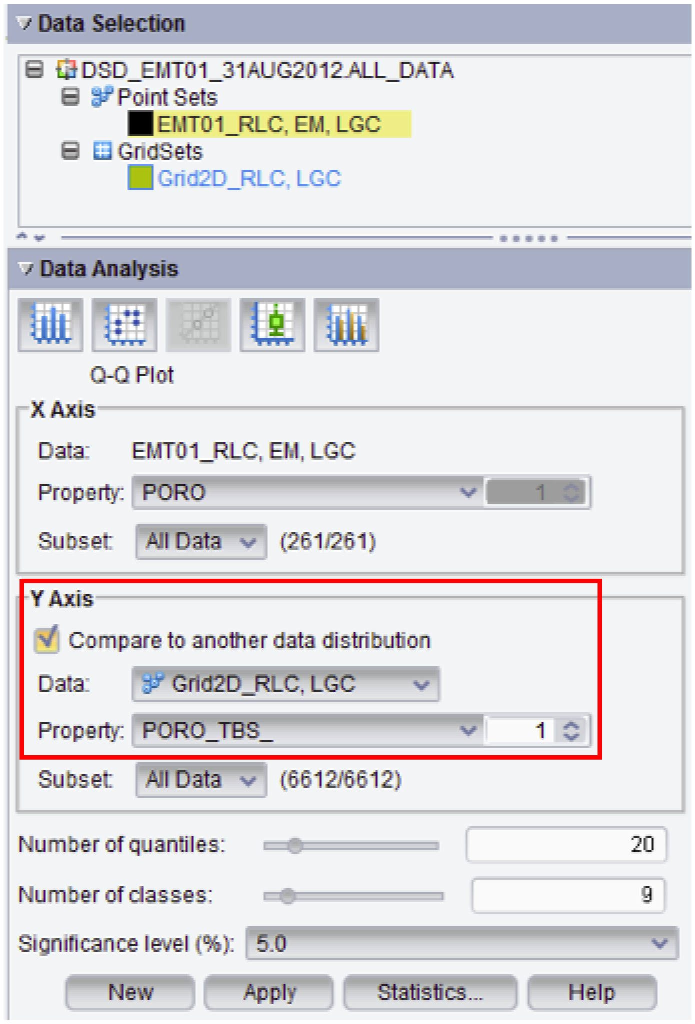


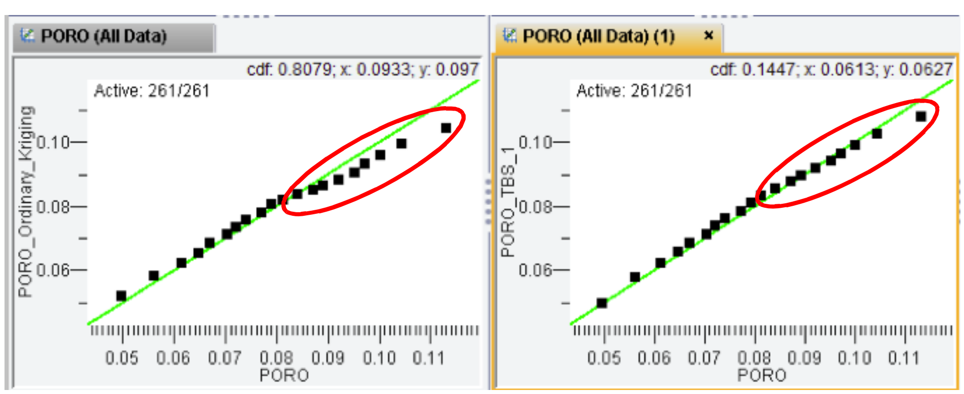
Q-Q Plot

2. Display a Q-Q plot between ordinary kriging and the PORO in the point set shown.



3. Display a Q-Q plot between TBS realization 1 and the PORO in the point set shown.





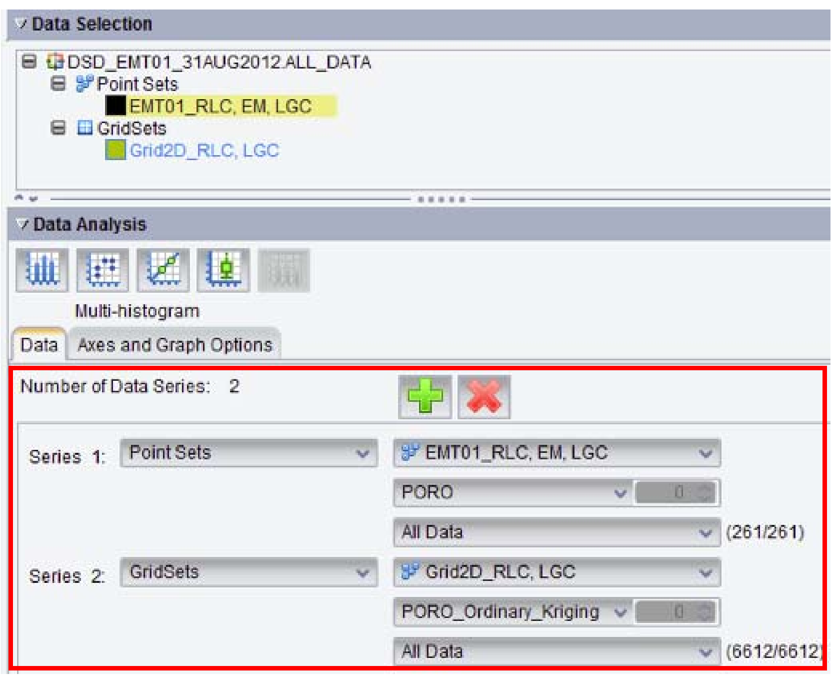
When the ordinary kriging results are compared to the input data (left), PORO, observe that the points deviate below the line at values greater than about 0.085. However, when PORO is compared to the TBS realization 1, the points do not deviate until the last point. These are expected results as the simulation method is designed to reproduce the statistical distribution of the input data, whereas kriging is designed to provide the best weighted average values. Thus kriging results tend not to reproduce the tails of the histogram as good as a simulation algorithm.

Multihistogram

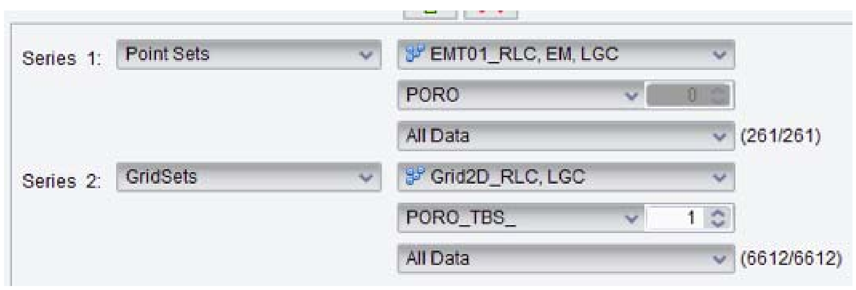
Next, you will use the multihistogram display to make the same comparisons with the input data and between ordinary kriging and TBS realization 1.

1. In Data Analysis, click the **Multihistogram** icon. 

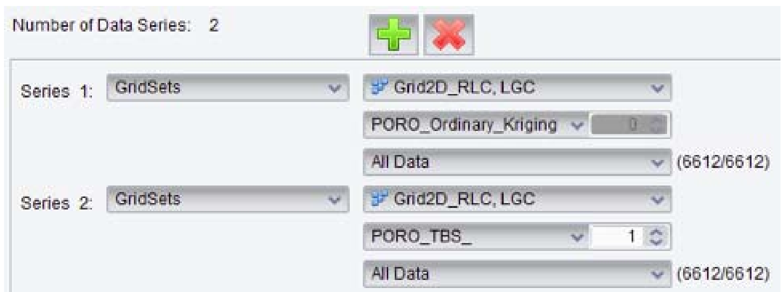
2. Create a display between ordinary kriging and PORO as shown. To add Series 2, click the green plus symbol. 

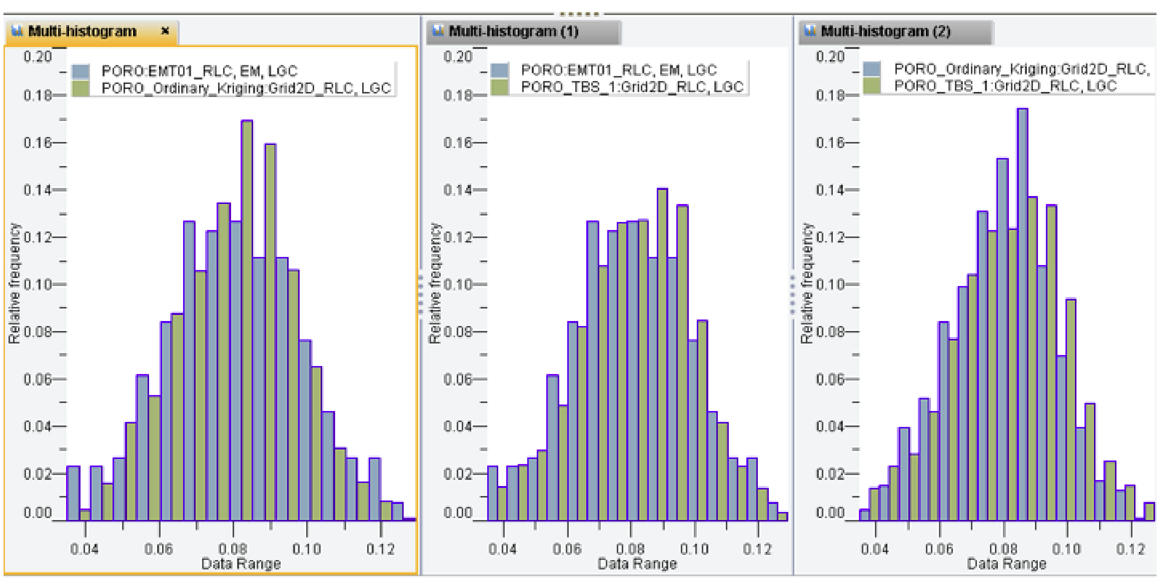


3. Create a display between TBS realization 1 and PORO as shown.



4. Create a display between ordinary kriging and TBS realization 1 as shown.





A B C

Figure A compares the PORO (blue) to the ordinary kriging. Notice that the kriging method tends to put more emphasis nearer the mean value (tallest two bars) and underestimates in the tails.

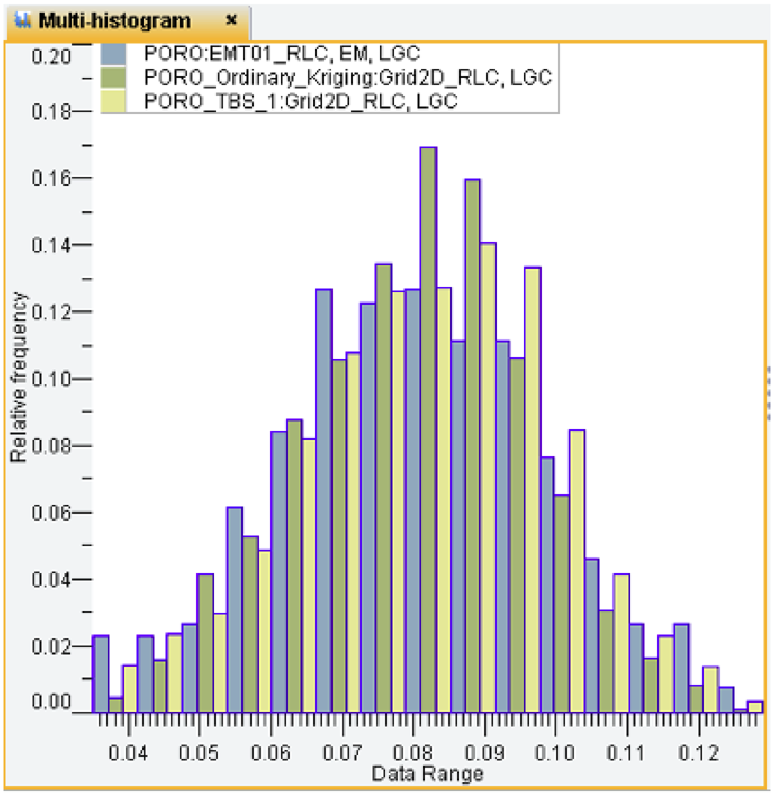
Figure B compares the PORO (blue) to the TBS realization 1. There is a much better reproduction of the input data with this method. You might want to compare the other 4 realizations to see the differences.

Figure C compares the ordinary kriging (blue) to the TBS realization 1. Again we see the over emphasis near the mean values with ordinary kriging.

Statistical Tables

You can display and compare the statistical tables.

1. Create a multihistogram with all three properties.



2. Create the statistical table.

Statistics Labels	PORO:EMT01_RLC, EM, LGC	PORO_Ordinary_Kriging:Grid2D_RLC, LGC	PORO_TBS_1:Grid2D_RLC, LGC
Samples for Statistics	261	6612	6612
Total Samples	261	6612	6612
Non Null Samples	261	6612	6612
Null Samples	0	0	0
Min	0.035	0.0357	0.035
Max	0.129	0.1266	0.1272
Sum	21.039	528.6504	537.3161
Unit Type	<UNKNOWN>	<UNKNOWN>	<UNKNOWN>
Unit of Measurement	<UNKNOWN>	<UNKNOWN>	<UNKNOWN>
Mean	0.0806	0.08	0.0813
Median	0.08	0.0816	0.0824
Geometric Mean	0.0783	0.0783	0.0792
Harmonic Mean	0.0758	0.0766	0.077
Sample Range	0.094	0.0909	0.0922
Variance	0.000350508	0.000245079	0.000301955
Variance MLE	0.000349165	0.000245042	0.000301909
Standard Deviation	0.0187	0.0157	0.0174
Standard Deviation MLE	0.0187	0.0157	0.0174
Mean Deviation	0.0151	0.0126	0.0141
Median Deviation	0.0151	0.0125	0.0141
Skewness	-0.0053	-0.1201	-0.1924
Kurtosis	-0.305	-0.2309	-0.2426

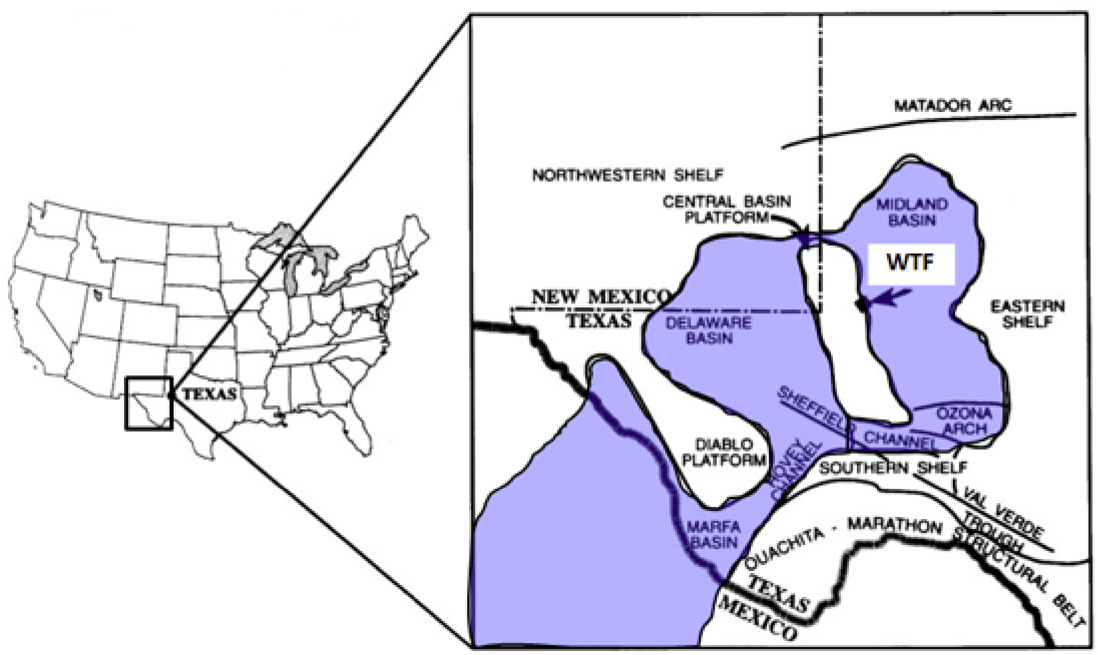
What other displays might be useful?

3. Save the session as DSD_Property_Modeling_your_intials.

Exercise 5
3D Reservoir Modeling Workshop

Introduction

For the 3D reservoir modeling workshop you will use a modified version of the Amoco data set from a field in west Texas, USA. The west Texas field (WTF) is located on the eastern edge of the Central Basin platform in the west Texas Permian basin. In the study area, production is from the Guadalupian Grayburg Formation (Permian), which is transitional between the previously more open marine conditions of the San Andres Formation and the more arid sabkha and siliciclastic eolian dune field environment of the younger Queen Formation.



Index map of the west Texas field location and the late Permian paleogeography.

Lithologically, the Grayburg is composed of alternating dolomite and siltstone for a total thickness of about 140 meters. Dolomites range from anhydritic skeletal wackestones through mudstones. Porosity is moldic or vuggy and can be extensively plugged by anhydrite. The siltstones are dominantly angular to subrounded quartz grains with angular feldspathic grains, which commonly alters to clay, often plugging pore throats. Siltstone porosity is intergranular. This formation has a characteristic shoaling-upward, prograding sedimentary motif, ranging from shallow open-marine to tidal flat/sabkha sediments. The silt is believed to be of eolian origin.

reworked by strandline processes into a series of thin, offlapping shoals. Progradation of the carbonate shelf was approximately from west to east. Structurally, the reservoir is a north-south-trending asymmetrical anticline, dipping gently eastward into the Midland basin. The Permian climatic regime was similar to the Plio-Pleistocene with major periods of glaciation. The carbonates formed during interglacial periods of relative high sea-level, whereas the eolian siltstones were most likely deposited during low sea level glacial periods with a source from the present day SE New Mexico.

References:

Chambers, R. L., M. A. Zinger and M. C. Kelly, (1994), *Constraining Geostatistical Reservoir Descriptions with 3-D Seismic Data to Reduce Uncertainty*, in J. M. Yarus and R. L. Chambers, eds., Stochastic Modeling and Geostatistics, AAPG Computer Applications in Geology, No. 3, pp. 143-158.

Yarus, J. M., R. L. Chambers, M. Maucec and G. Shi, 2012, *Facies simulation in practice: Lithotype proportion mapping and plurigaussian simulation, a powerful combination*, in the Proceedings of the Ninth International Geostatistics Congress, Oslo, Norway, June 11-15, 10 pages.

Workshop Objectives

In this workshop you learn the major workflow steps requires to make a 3D geocellular model. The general workflow is as follows:

1. Load the data into the session.

2. Create a sealed structural framework.

3. Perform stratigraphic modeling:
    a. Specify grid geometry
    b. Define lithotypes for modeling
    c. Seismic blocking
    d. Well blocking
4. Perform facies trending modeling:
    a. Seismic-to-facies calibration
    b. Create facies proportion volume from well data
5. Perform facies modeling:
    a. Sequential Indicator
    b. Plurigaussian
6. Perform petrophysical property modeling.
7. Perform static volumetric computations.
8. Assess the model uncertainty.

Getting Started

1. Launch DecisionSpace® Desktop.
2. Create a new session.
3. Load data to the active the session and view the Inventory.
    - Fault
    - Surface Grid
    - Surface Picks
    - Seismic Acoustic Impedance
4. View the data in Cube View.

Create a New Session

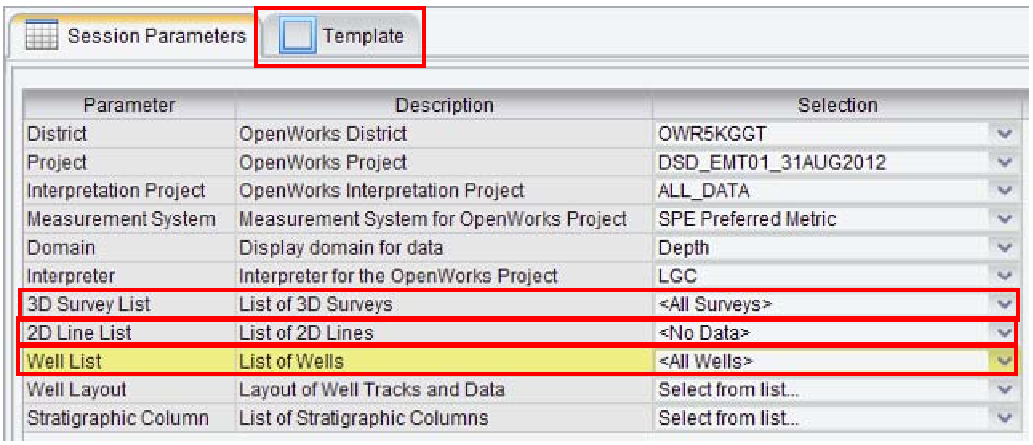
1. Start DecisionSpace® Desktop.

2. Select the **New Session** tab and then select all the Modules.

3. Select the session parameters as shown. These should be the same as the previous session except now select:

   - **3D Survey List > All Surveys**

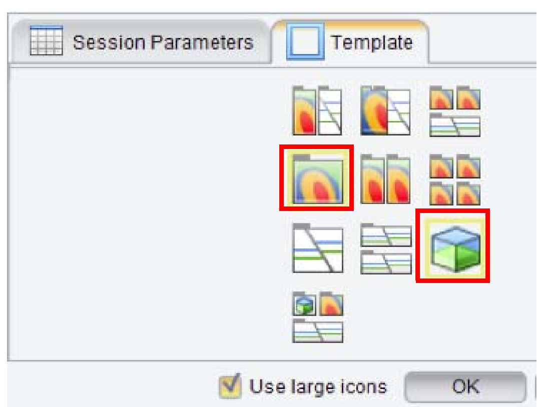
   - **Well List > All Wells**



4. Click the **Template** tab.

5. From the available templates, select the **Map** and **Cube Views**.

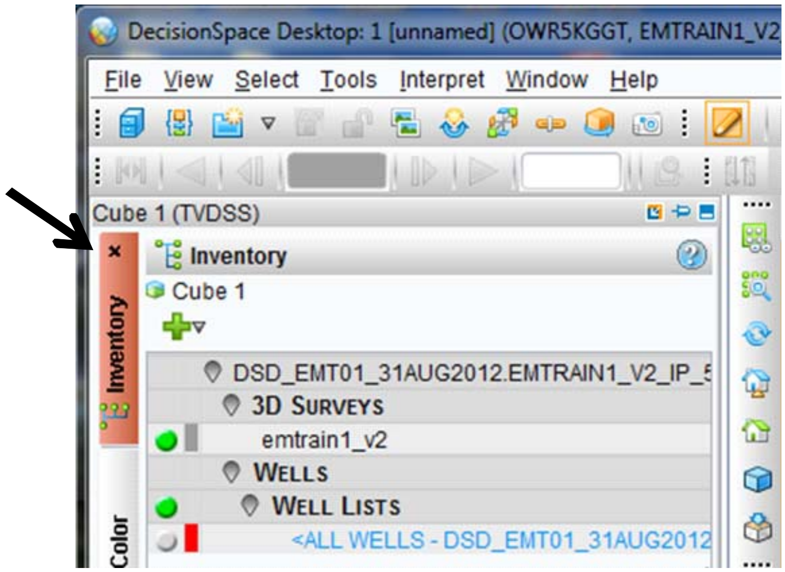
   Selecting Use large icons is optional.



6. Click **OK**.

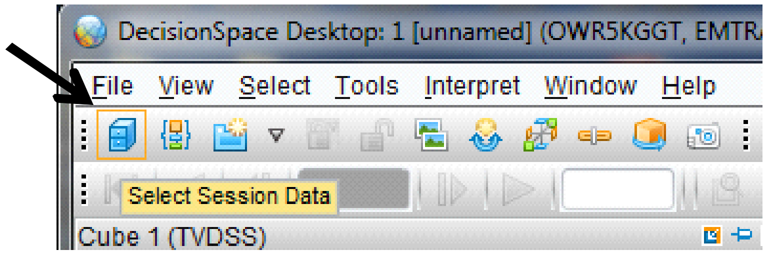
Load Data into the Session

1. Click the **Inventory** tab to see what information is already loaded into the active session.

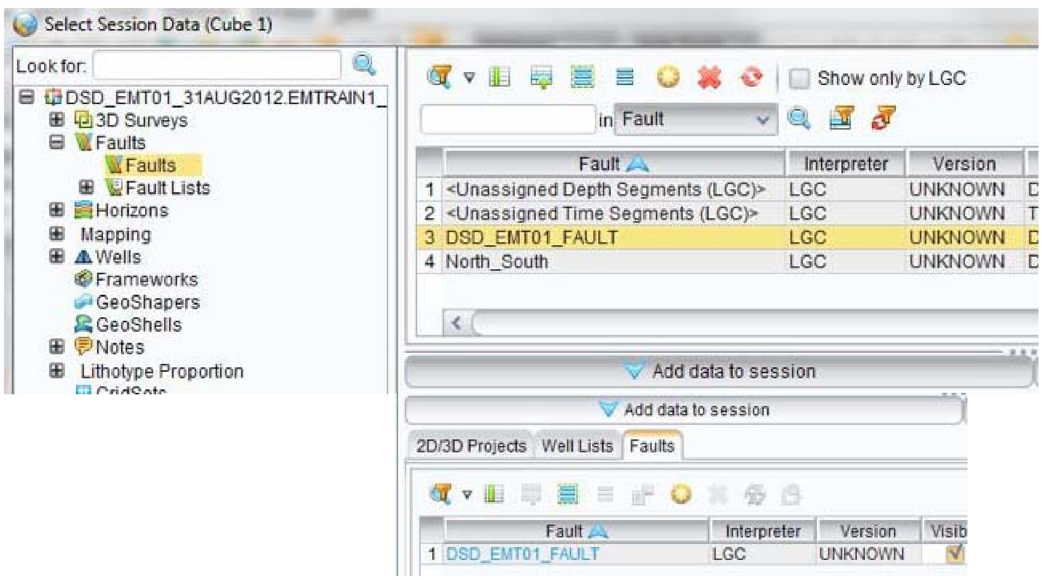


Add a Fault

1. Click the **Select Session Data** icon to open the Select Session Data (SSD) window (the SSD Window).

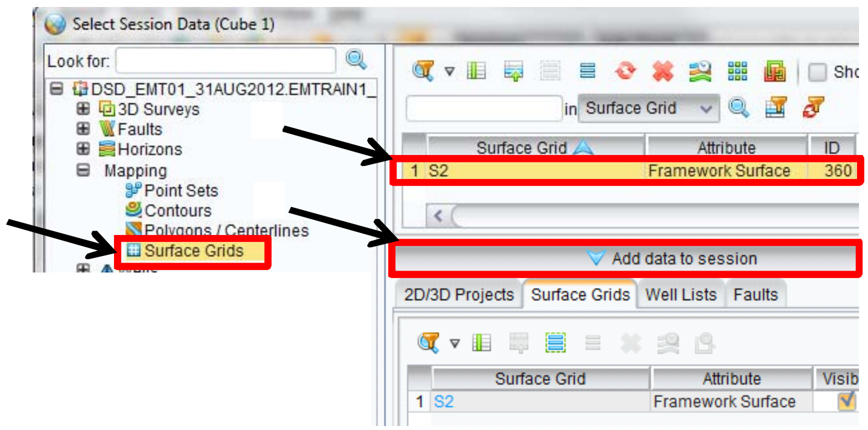


2. Select **Faults** and add the **DSD_EMT01_FAULT** fault to your session.



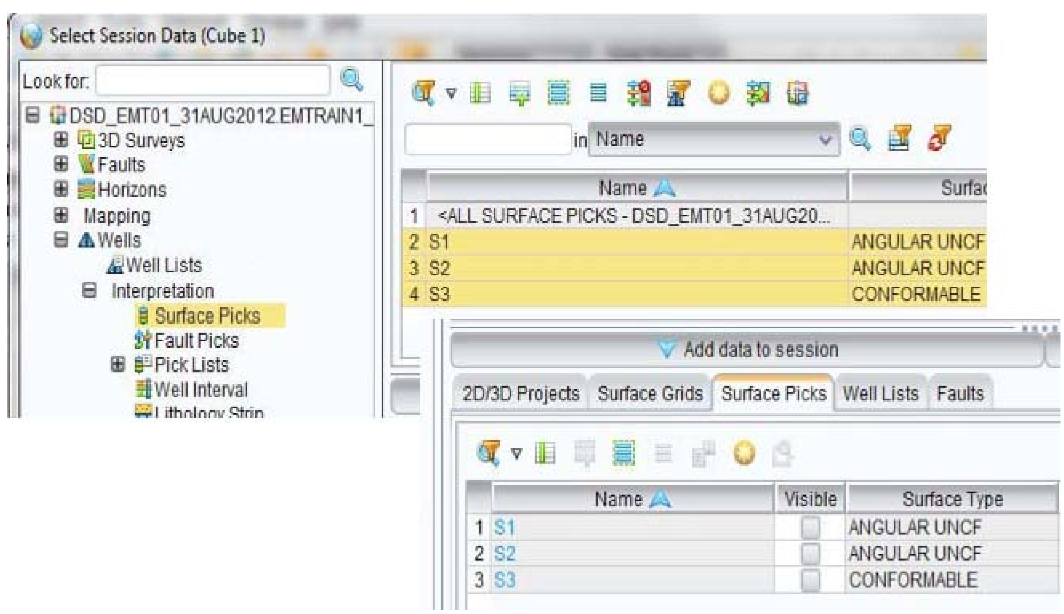
Add Surface Grid

3. Select **Mapping** > **Surface Grids** and add Surface Grid **S2** to your session.



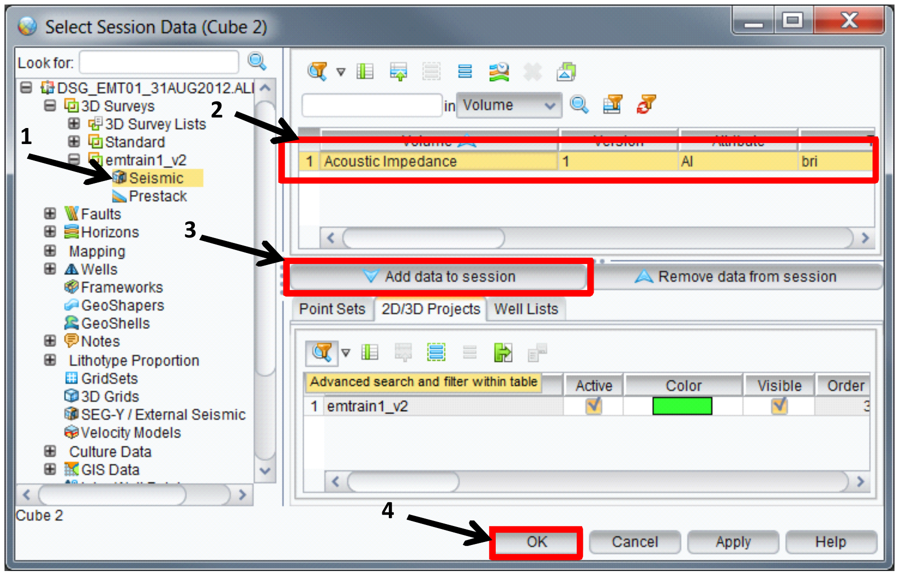
Add Surface Picks

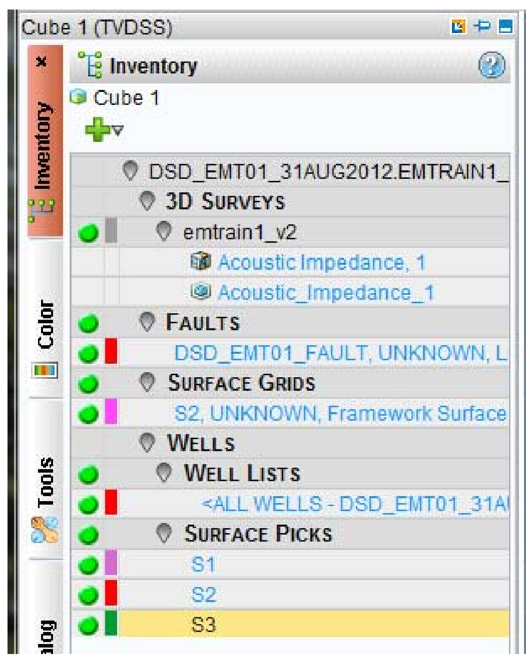
4. Select **Wells** > **Interpretation** > **Surface Picks** and add Surface Picks **S1**, **S2**, and **S3** to your session.



Add Seismic Acoustic Impedance

5. Select **3D Surveys > emtrain1_v2 > Seismic** and add **Acoustic Impedance** to the session.





Data in the active session.

6. If not selected already, turn on the Green Button beside each data object so it will display in the Cube View.

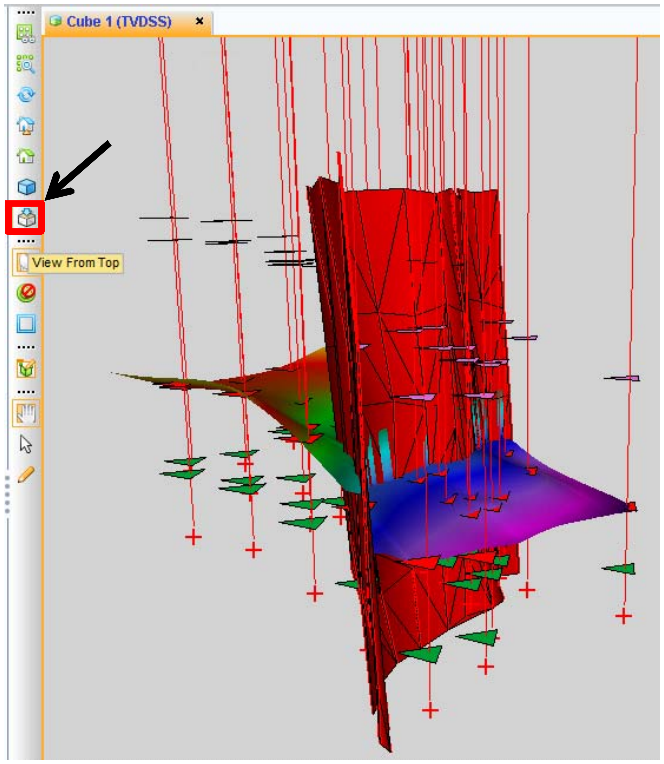
View Data in a Cube View

If the data are not visible in the Cube View use:

- MB1 to click the View From Top icon  located on the tool bar on the left side of the Cube View
- MB1 to rotate
- MB2 to move laterally or vertically
- MB1+MB2 (zoom: or use track wheel) to adjust the view.

This is the basic data that is used to create a sealed framework with three bounding surfaces and a single fault.

The middle surface is used as a guide in the construction of the other two surfaces which are known from only well tops picked by the geologist.



Save the session as EMT01_3D- your initials-01.

Create a Structural Framework

Using the single fault, surface picks and the S2 surface grid you will learn how to create a sealed structural framework. There are two types of frameworks:

- Dynamic – this type of framework is always the first one created and can be done while interpreting the data. If an interpretation changes then the entire framework updates automatically or you can have it update on demand.
- Static – once you have a framework completed to your satisfaction, you can save a Static version such that new updates won't change it. We recommend using the static version when creating the geocellular grid which creates reproducible grids, provided that the gridding parameters don't change.

Workflow

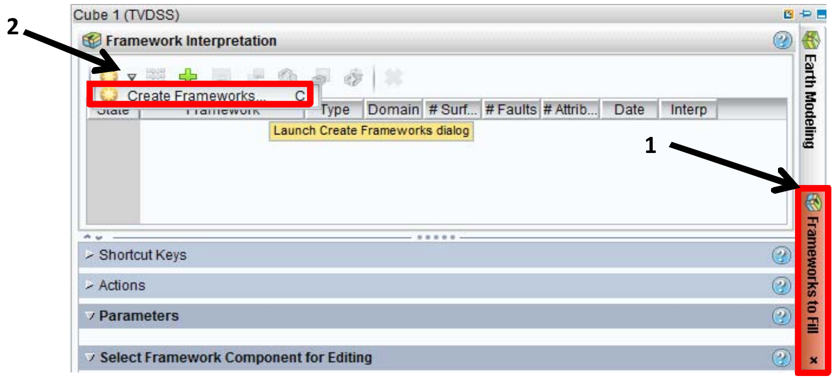
Open Frameworks to Fill to create framework:

- Create new dynamic framework
- Add data to framework builder
- Select surface grid as master surface and set gridding parameters
- Define conformance rules
- Define cleaning parameters
- Save dynamic framework to OpenWorks

Create a New Framework

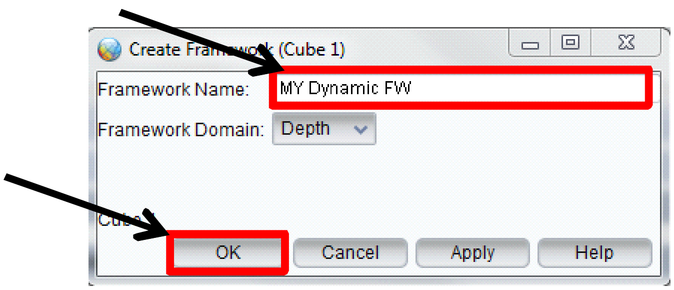
1. Select the **Frameworks to Fill** tab to open its task pane.

2. Click the arrow  next to the Create Frameworks icon and select **Create Frameworks**.

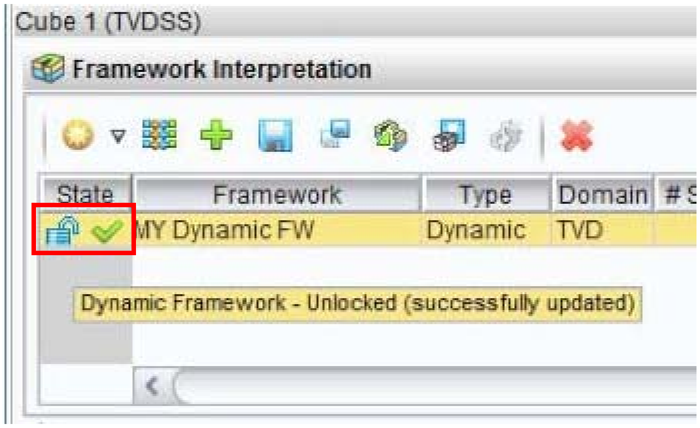


3. Enter a name for the framework.

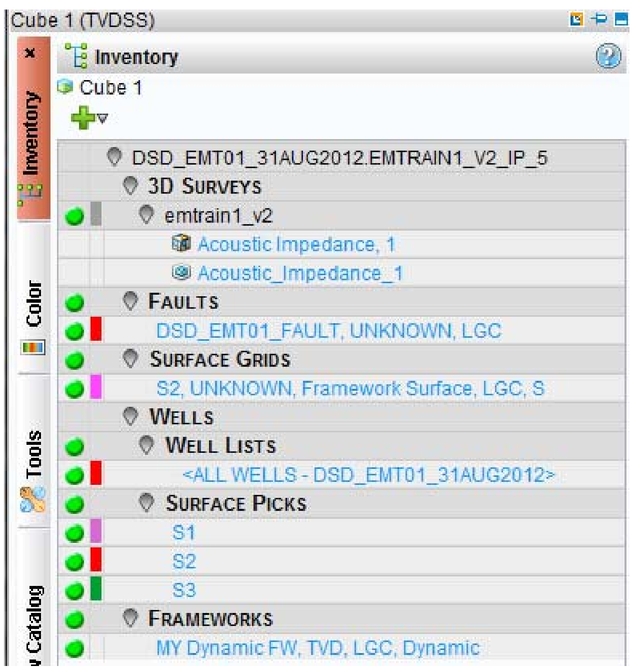
4. Click **OK**.



5. Make sure that the Lock is **unlocked** to allow framework updating.



Your empty framework can now be listed in the inventory.

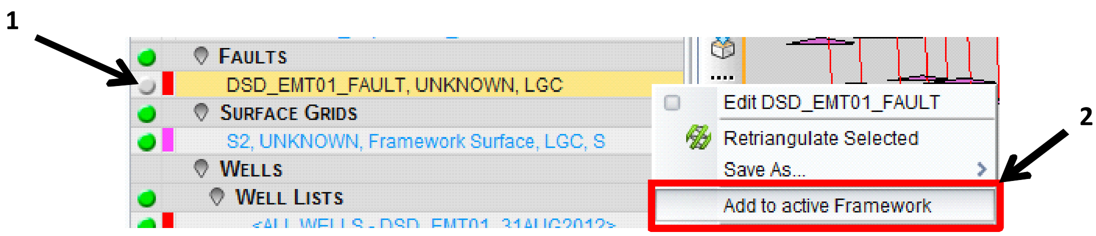


Add Data to the Framework

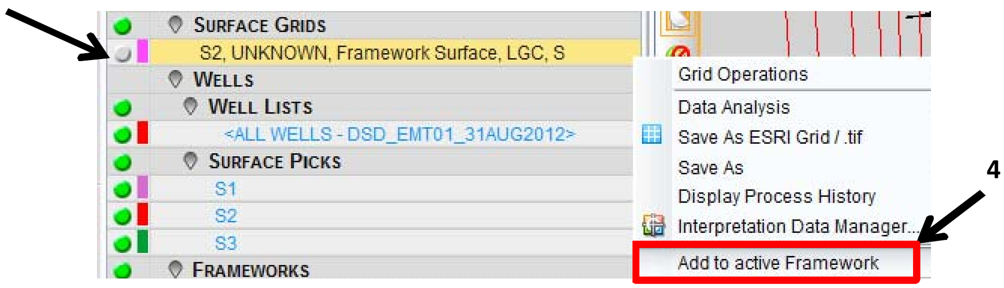
- Fault
- Surface grid
- S1 & S3 surface picks

1. Click the green button for the fault in the inventory to hide it, then click MB3 on the fault name and select **Add to Active Framework**.

   By hiding it first you will see only the version of this fault that is generated by the Framework. It is not required to hide it first, but visually it is easier to manage.

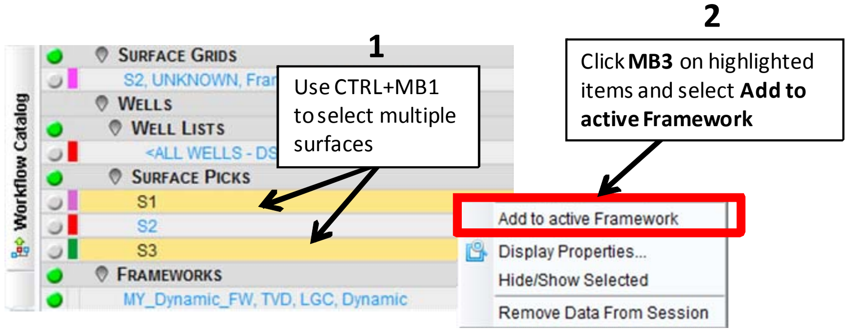


2. Uncheck the surface grid and add it to the framework.

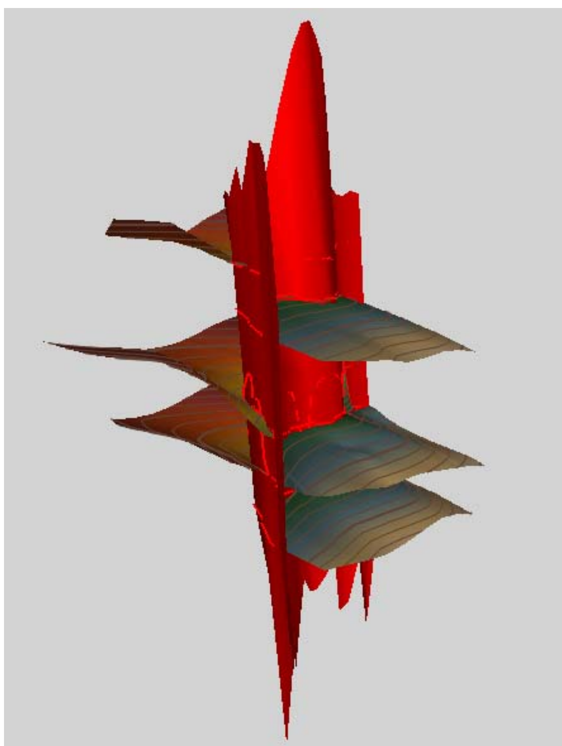


3. Uncheck all of the surface picks and then add the S1 and S3 picks to the active framework.

Notice that active surfaces are created based only on the wells picks.



Current Inventory of data in the Active Framework.



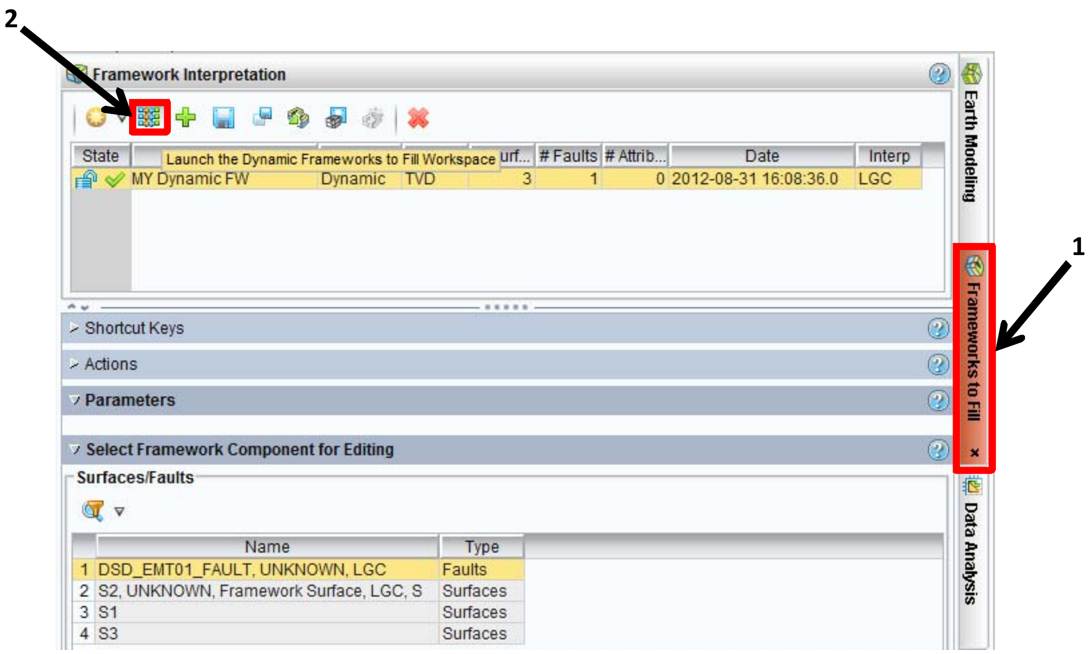
Cube View display of data.

Choose Master Surface

The Frameworks to Fill Task pane contains the elements necessary to create the framework.

1. Select the **Frameworks to Fill** tab.

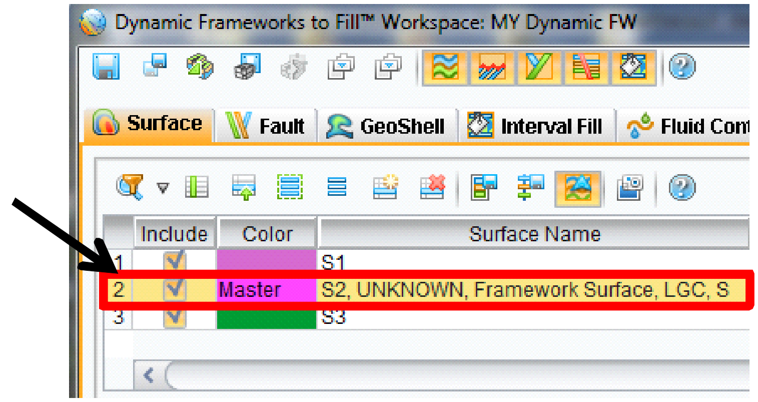
2. Click the  icon to launch the workspace.

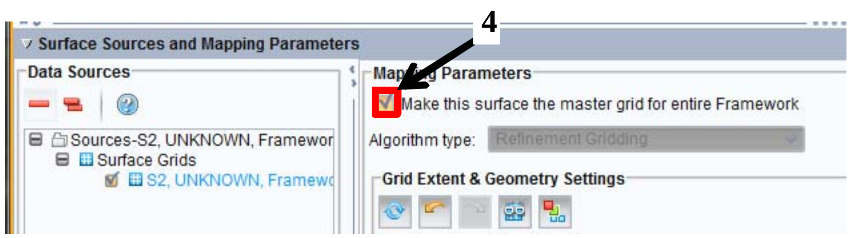


3. Select the **S2 surface** in the Surface Sources and Mapping Parameters.

4. Select the **Make this surface the master grid for the entire Framework** check box. The other two surfaces will be made conformable to this surface.

3





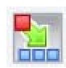
Grid Extent & Geometry Settings

1. Change Grid extents to **User Specified**.

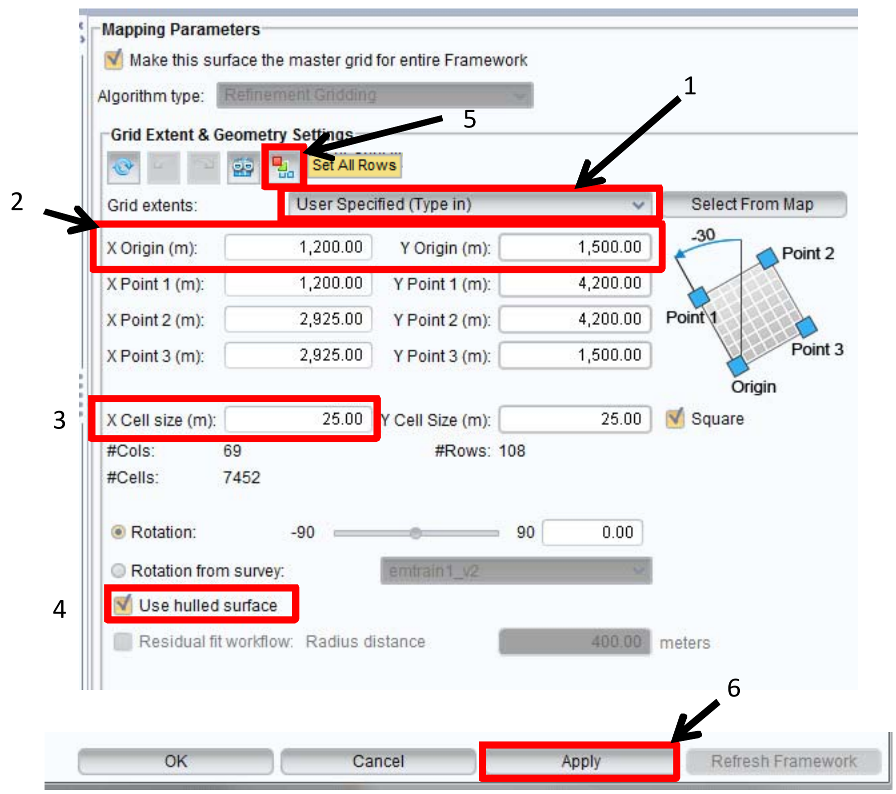
2. Set X & Y origins as shown.

3. Set X cell size as shown and select the **Square** checkbox.

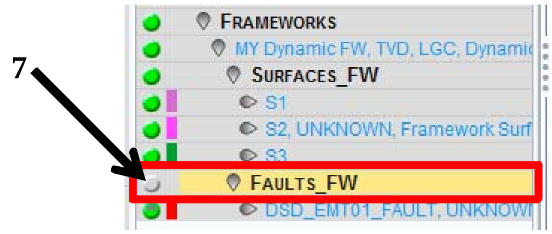
4. Leave Use hulled surface checked.

5. Click the **Set All Rows** icon  to have an equal number of columns (69) and rows (108) for each surface.

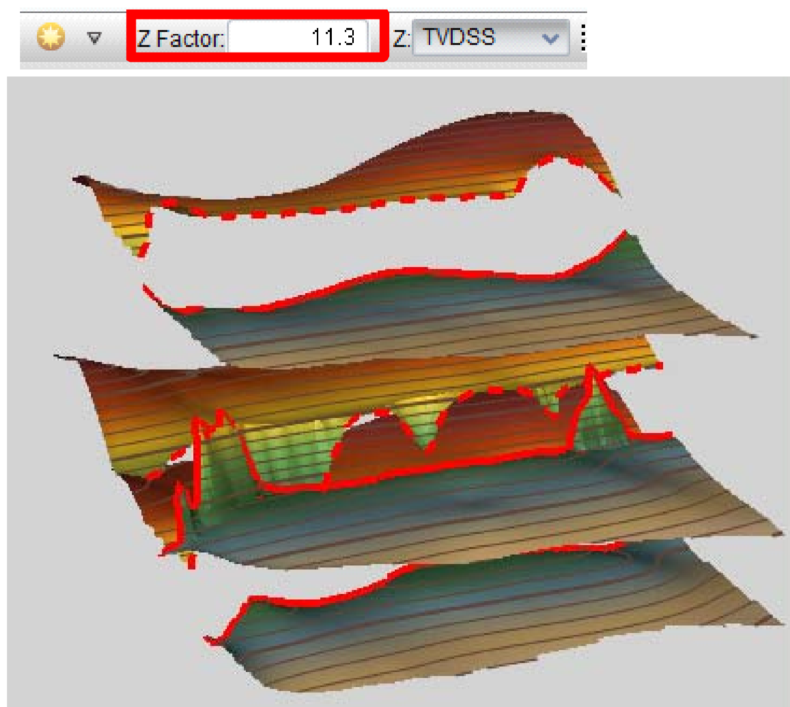
6. Click **Apply**.



7. Turn off the fault and look at the surfaces in the Cube View.



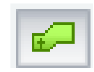
8. You can adjust the Z Factor (vertical exaggeration) to make your display look like the following image.

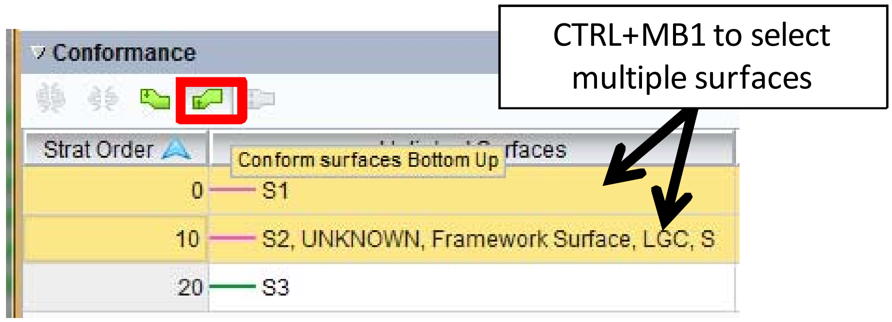


Notice that the middle surface (S2 surface grid) is not yet trimmed by the fault. Also the S1 and S3 surfaces are not conformable to the S2. The next steps set the conformity rules and trim and seal the surfaces to the fault.

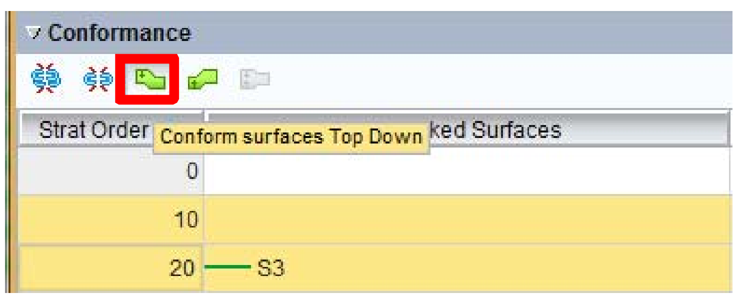
Set the Conformance Rules

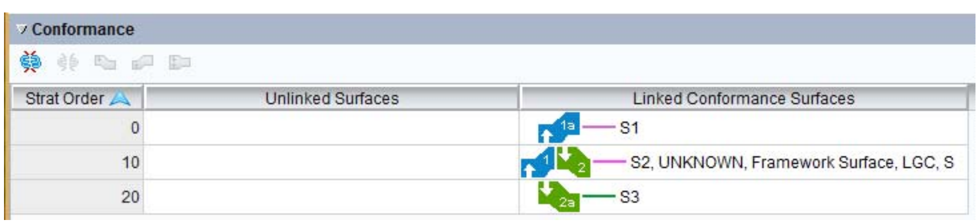
1. Open the Conformance Parameters. If not in Strat order click .

2. Use CTRL+MB1 to select the **S1** and **S2** surface and click  to make the S1 conform to the S2 (conformable upwards).

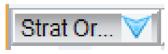


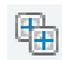
3. Use CTRL+MB1 to select the S2 & S3 surface and click  to make the S3 conform to the S2 (conformable downwards).

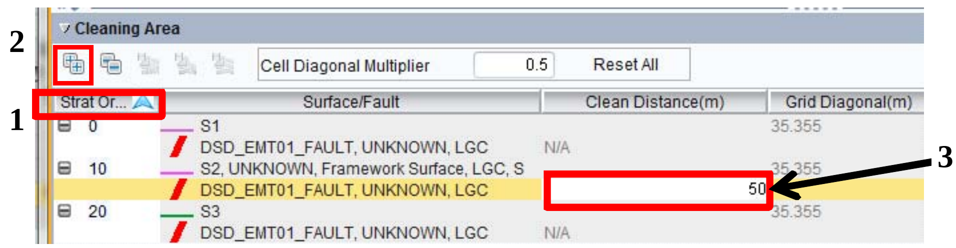




Set the Cleaning Parameters

1. Click  to reorder the surfaces.

2. Click the icon  to expand the view.



3. For the S2 surface change the cleaning distance to **50** and press the Enter key.

4. Click **OK**.

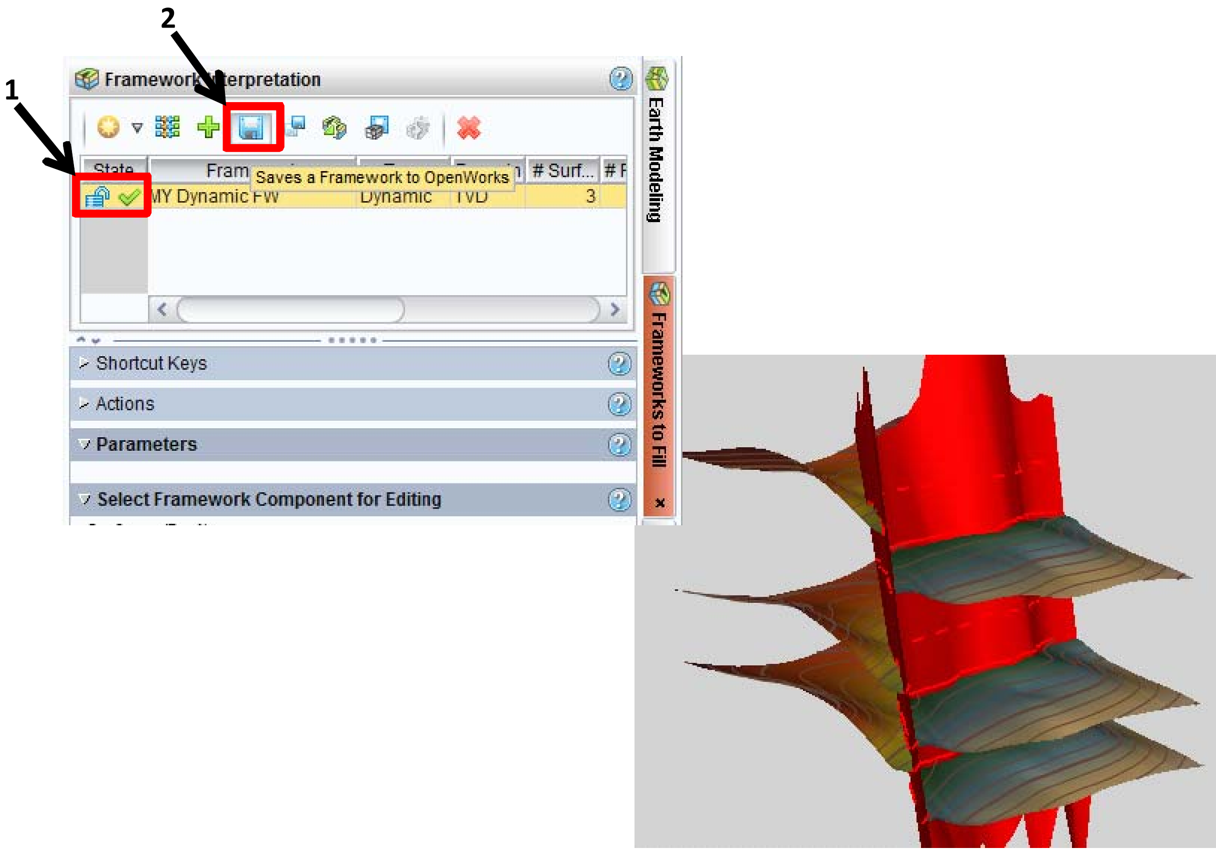
   You now have a sealed framework consisting of three conformable horizons and a single fault.

5. Save the session.

Save the Dynamic Framework

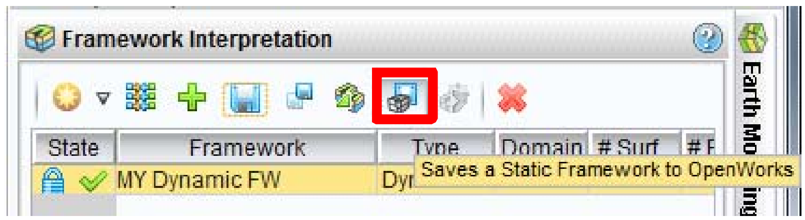
1. Click the State icon to lock the framework to override dynamic updating.

2. Click the icon to save the framework to OpenWorks.



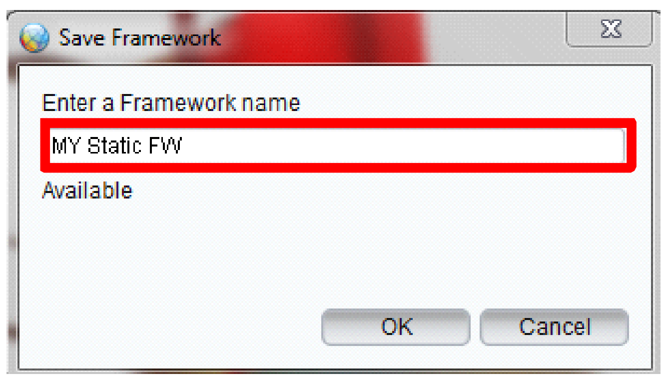
Create a Static Framework

Now that we have a sealed framework it is best to create a static version of it for use in the Earth Modeling workflow. Even though the framework is locked to prevent automatic updating while interpretation, it is not a static version, which is required for repeatability in Earth Modeling for use in the creation of the 3D geocellular grid.



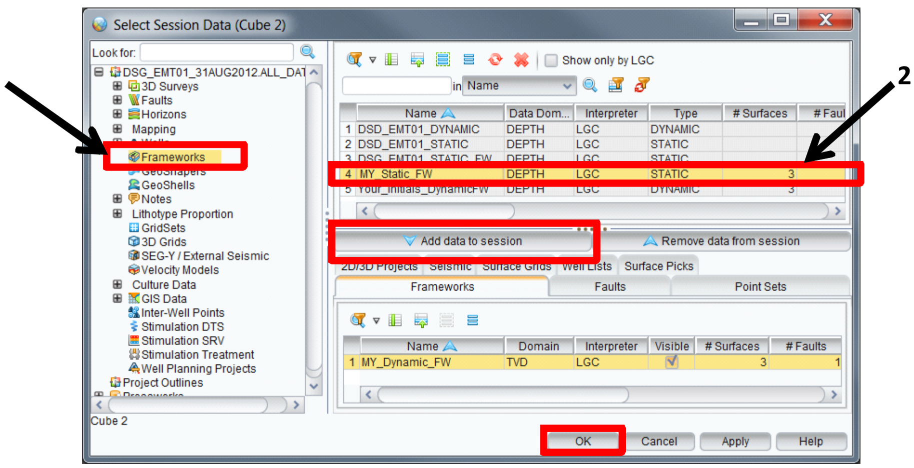
1. Click the  icon to name and save the static framework.

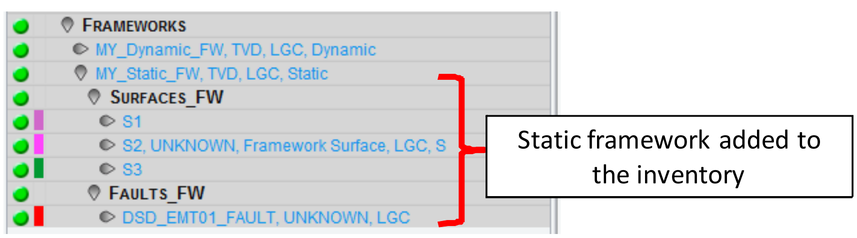
2. Enter a name and click **OK**.



Load the Static Framework

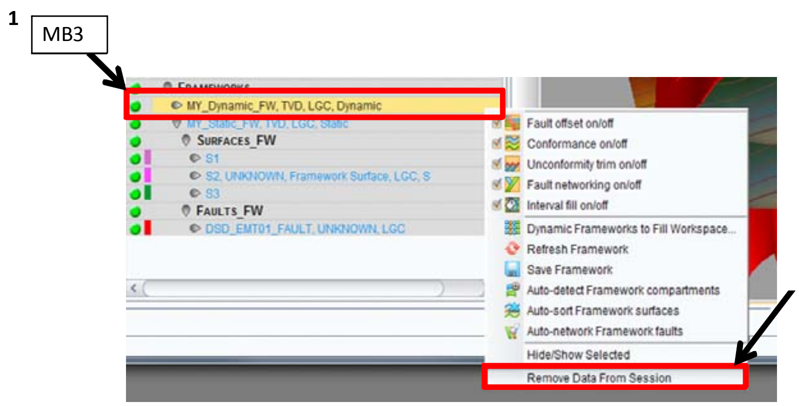
1. Click the **Selection Session Data** icon on the main tool bar.
2. Select and add the static framework to the session.
3. Click **OK**.





Remove the Dynamic Framework

1. Click with MB3 on the **Dynamic** framework and then select **Remove Data From Session** to remove it from the working session. This only removes the dynamic framework from the current session as it is still stored in OpenWorks.



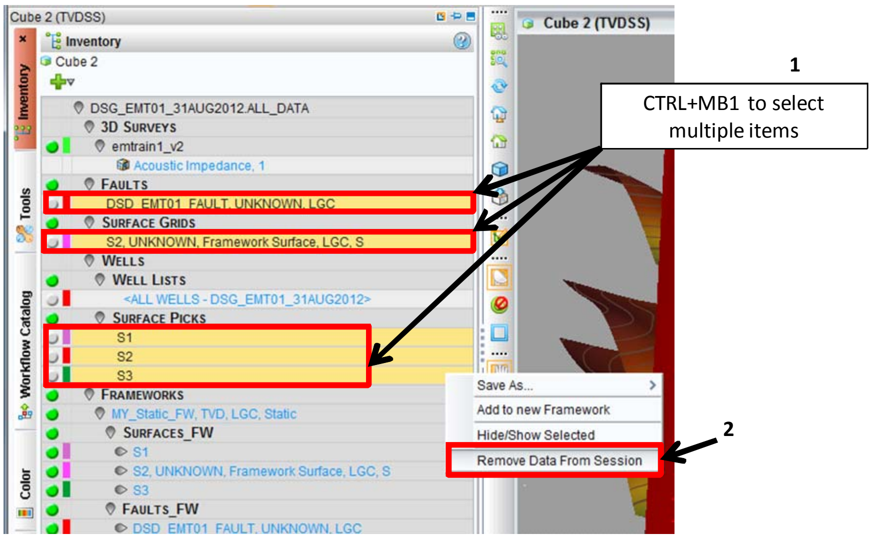
Remove Framework Data

We no longer need the fault nor the surface grid and surface picks used to create the framework so remove these items from the working session.

1. Press Ctrl + MB1 to select the items shown in the image.

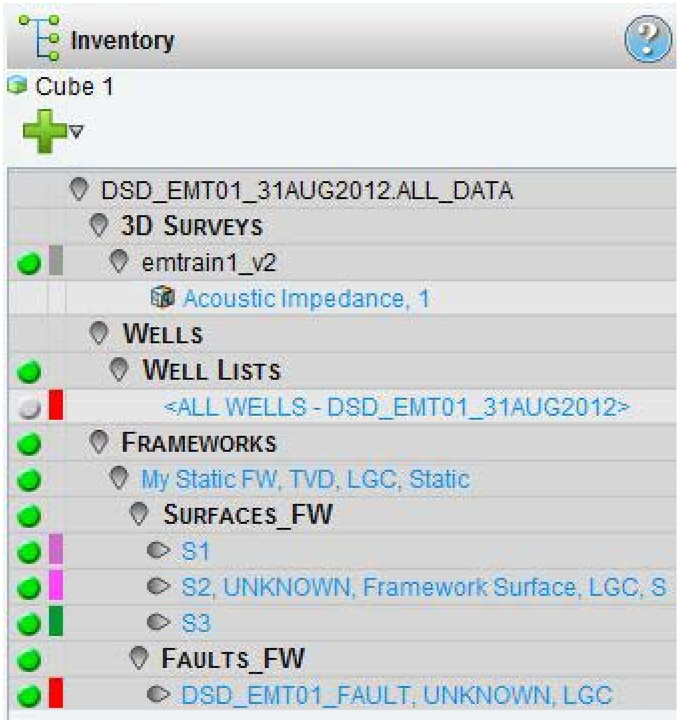
2. Click MB3 and select **Remove Data From Session**.

3. Save the session.



Data

Current Inventory of Data



Data Analysis of Unblocked Data

Workflow

1. Import a point set from an ASCII.txt file.
2. Exploratory Data Analysis.
    - Histograms
    - cross-plots
    - Create subsets
    - Box Plots
    - Outlier Analysis
3. Exploratory Spatial Analysis.
    - Compute lithology indicator variograms
    - Design vertical cell thickness for the 3D geocellular grid based on the vertical variogram scales.

Import External ASCII File

Now that you created a static framework, next you will add the well data to the session. Although the data exists as log curves in OpenWorks and can be added to the session via Select Session Data, you will import an ASCII.txt file with new well data just created by the project geologist. There is a need to load the data first into OpenWorks.

The ASCII file contains a simple one line header that describes the data as illustrated below. Each line has data for each depth in the well and each set of well data are concatenated to the end of the previous well. There are 16 wells in the data set.

DSD_EMT01_WELLS.txt

Well_Name	X_UTM	Y-UTM	TVDss-m	Lith_Code	PORO	PERM
1001	2060.00	3100.00	1262.65	4.000	0.131	179.468
1001	2060.00	3100.00	1263.35	4.000	0.117	171.927
1001	2060.00	3100.00	1263.86	2.000	0.146	303.753
1001	2060.00	3100.00	1264.16	2.000	0.170	102.796
1001	2060.00	3100.00	1264.46	2.000	0.179	113.037
1001	2060.00	3100.00	1264.77	2.000	0.184	124.426

Well 1001 data continues until data for well 1002 begins, etc.

1001	2060.00	3100.00	1349.20	2.000	0.172	105.118
1001	2060.00	3100.00	1349.50	2.000	0.177	117.036
1001	2060.00	3100.00	1349.81	2.000	0.179	123.564
1001	2060.00	3100.00	1350.11	2.000	0.182	135.604
1002	1760.00	1660.00	1243.91	2.000	0.182	130.284
1002	1760.00	1660.00	1244.10	2.000	0.189	147.037
1002	1760.00	1660.00	1244.35	2.000	0.195	161.808
1002	1760.00	1660.00	1244.65	2.000	0.173	114.707

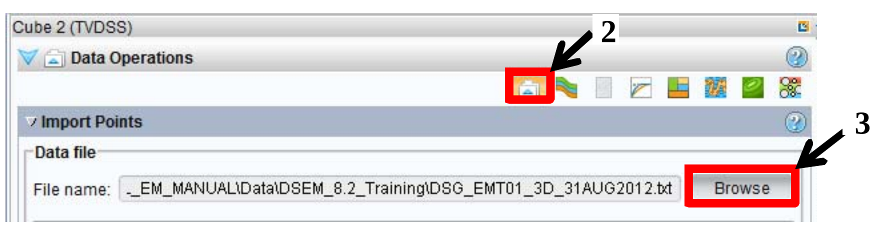
1. Click 

to open Earth Modeling.

2. Click the  icon to access Data Operations.

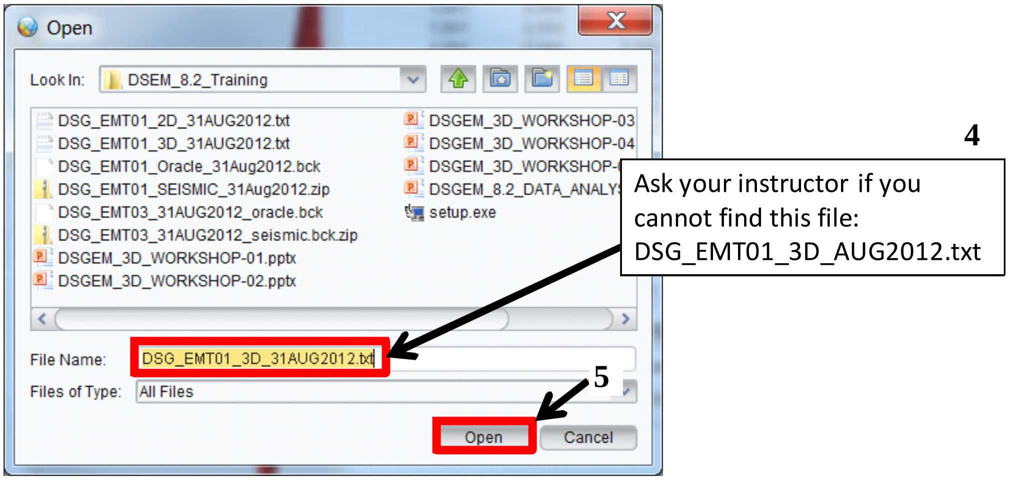
3. Open the Browser 

.



4. Select the **ASCII.txt** file.

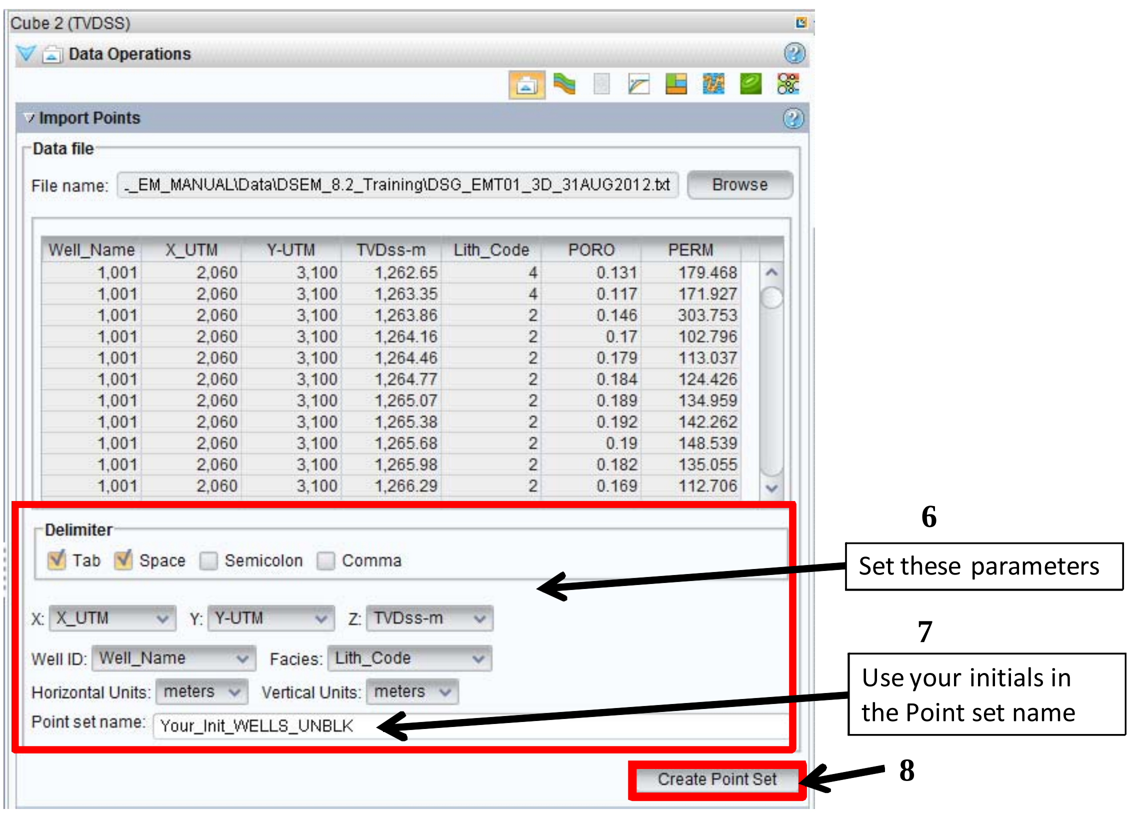
5. Click **Open**.



6. Set the parameters as shown.

7. Enter a point set name.

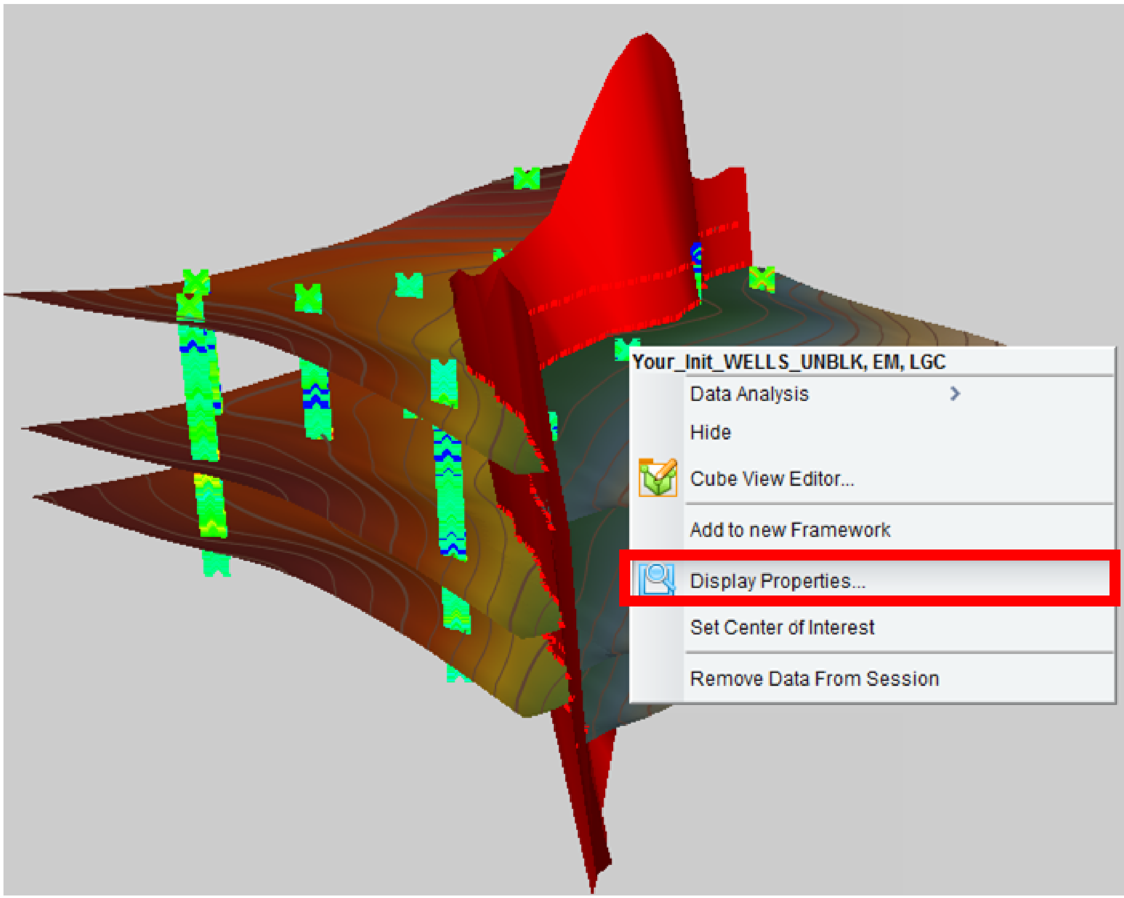
8. Click **Create Point Set** once all parameters are entered. This action automatically displays the well data as a point set in the Cube View.



Change Point Set Display Properties

1. Use the Display Property Editor to change how the information is displayed. Place the cursor over one of the wells and with MB3 open the drop-down menu and select 

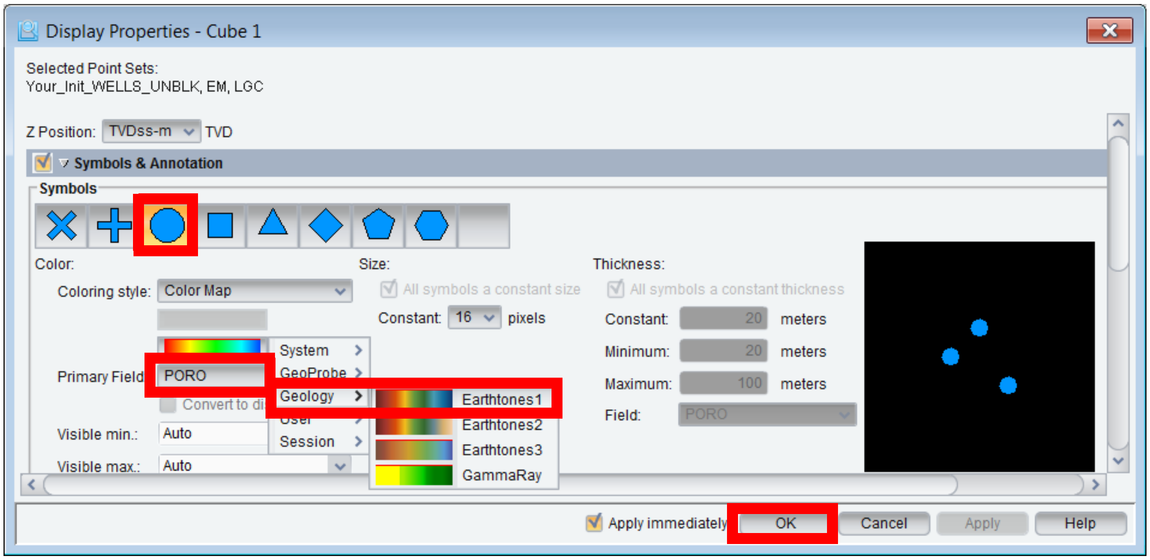
.



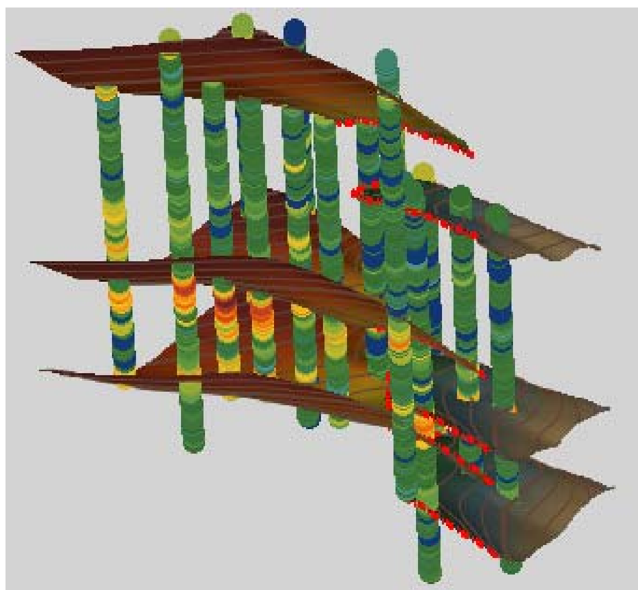
2. Change Color map to **Geology** > **Earthtones 1**.

3. Change the Overlay property to **PORO**.

4. Change **Symbol to Circle**.



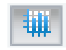
There is a zone of high porosity below the middle surface in interval 2.

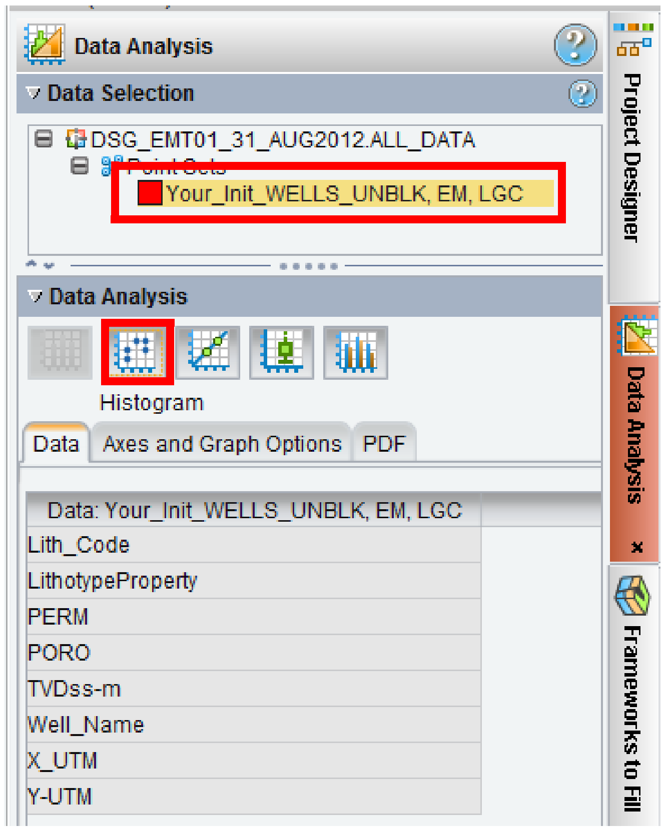


Histogram Displays

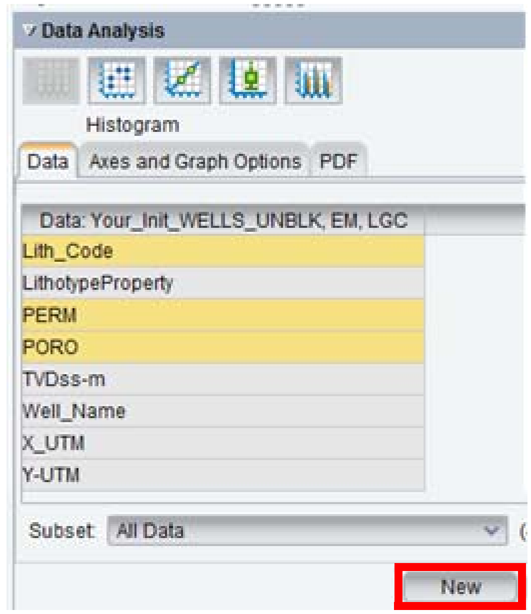
1. Click the **Data Analysis** tab.

2. Select the unblocked point set in the Data Selection section.

3. Click the **Histogram** icon. 



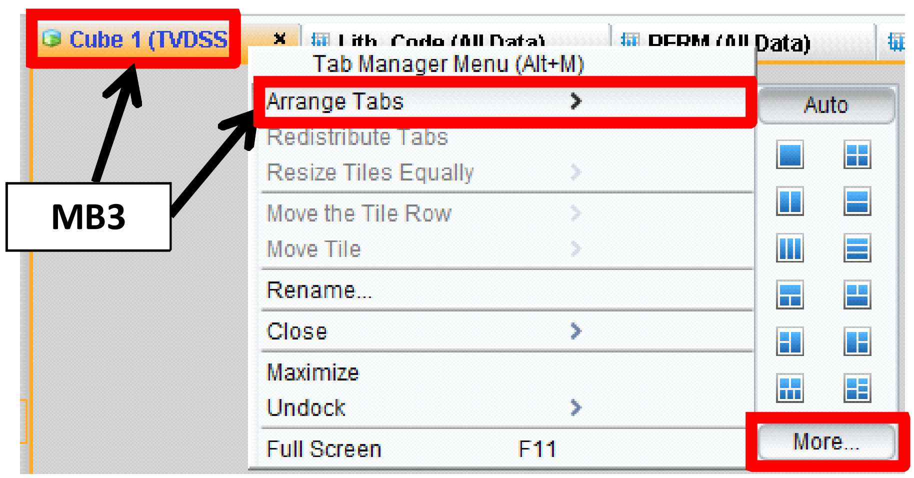
4. Select the following parameters using MB1+Ctrl.



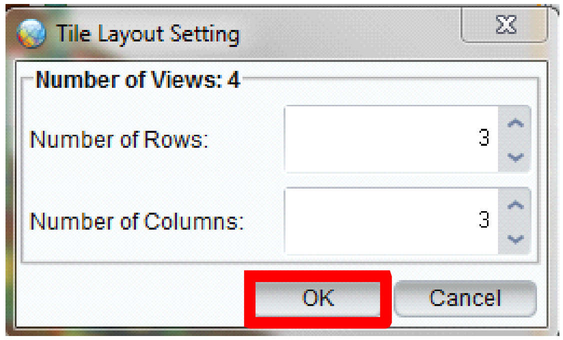
5. Click **New** to display the three histograms.

6. Click the Cube View with MB1 and then with MB3.

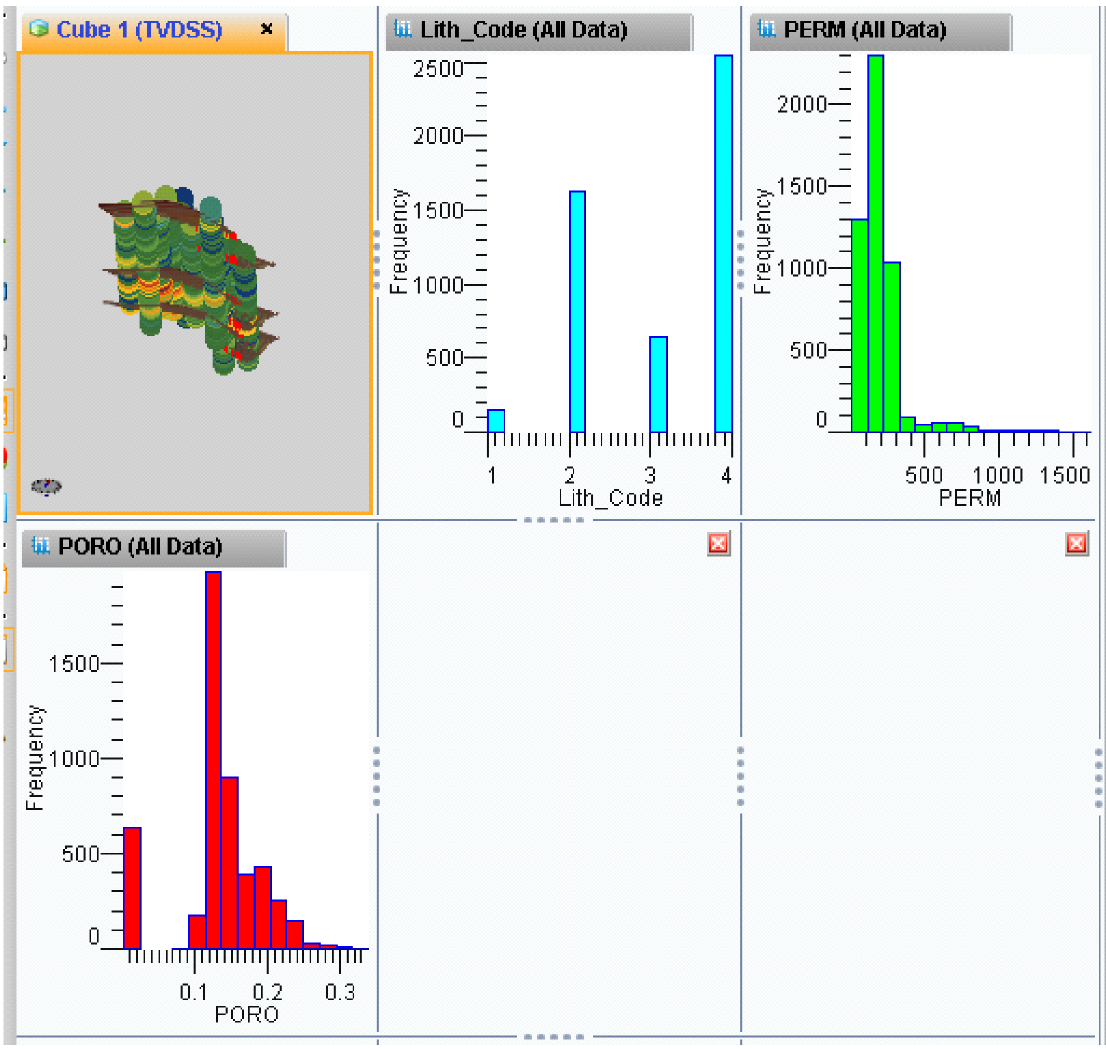
7. Select **Arrange Tabs** and then click **More**.



8. Change the Tile Layout to 3 by 3 to see the Cube View and three histograms. As more displays are created the windows are automatically populated.

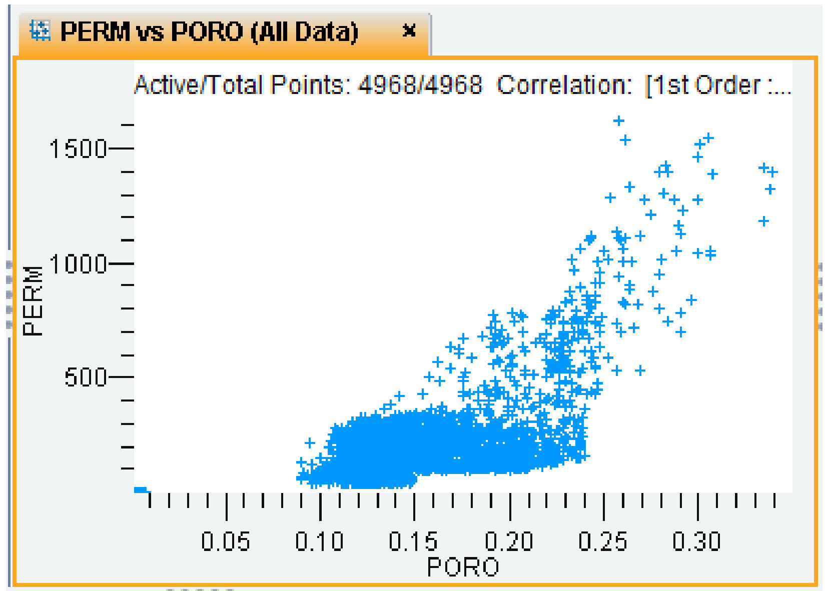


At this point your display should look like the following image.



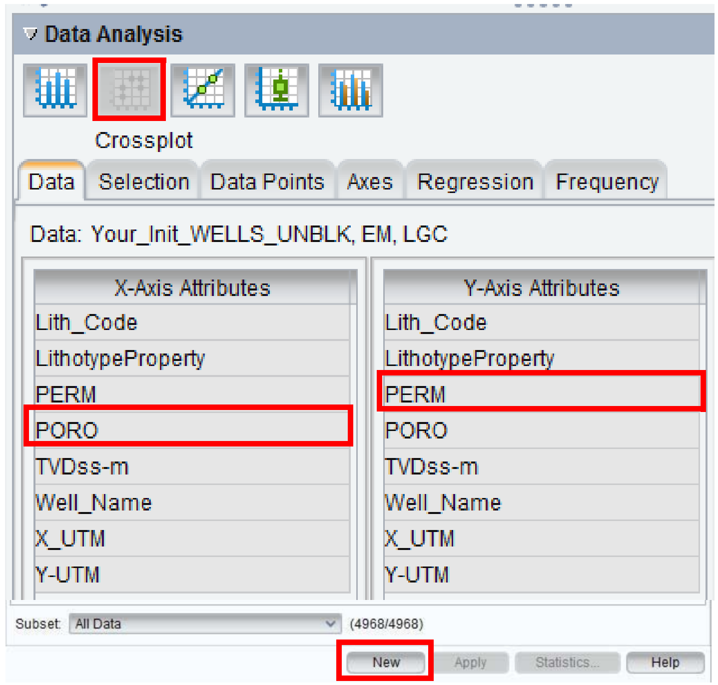
Cross-plot Display

Create a cross-plot of PORO (X-Axis) and PERM (Y-axis).



1. Click the **Cross-plot** icon.
2. Select **PORO** for the X-Axis.
3. Select **PERM** for the Y-Axis.

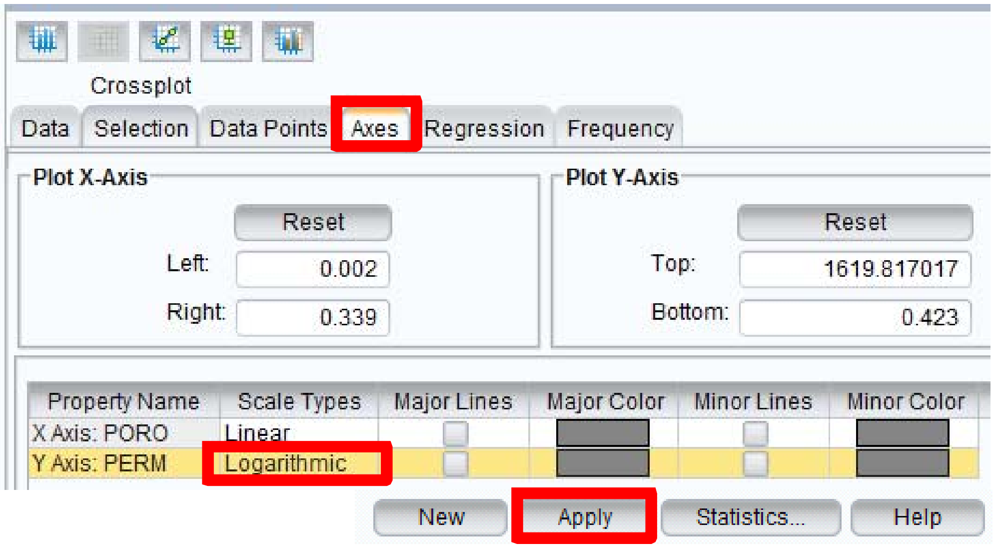
4. Click **New**.

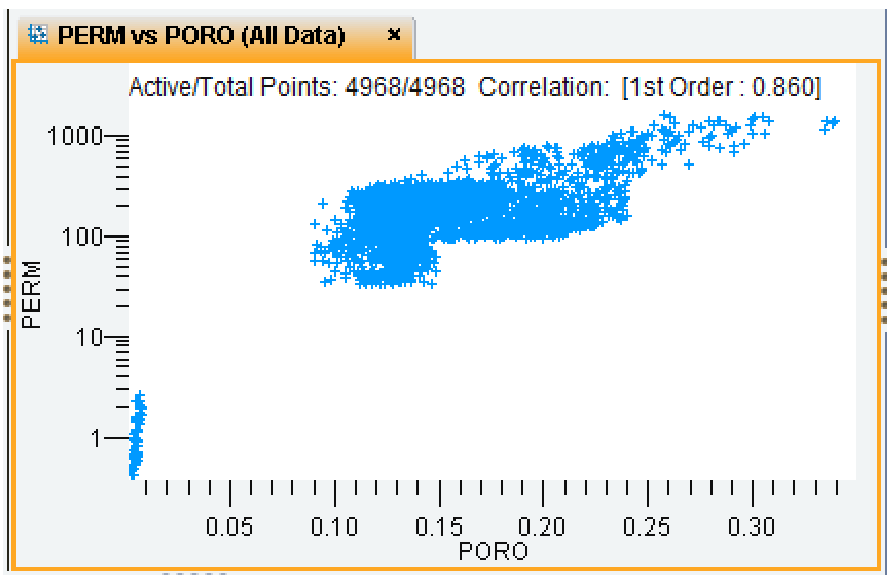


5. Click the **Axes** tab.

6. Change Y-Axis Scale Type to **Logarithmic**.

7. Click **Apply**.



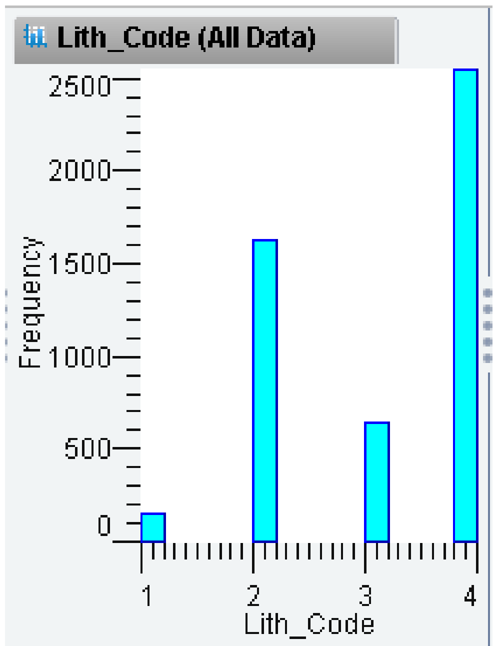


Display PERM on a Logarithmic scale.

Create Subsets

From the Histogram of the Lith_Code, we observe four codes which are:

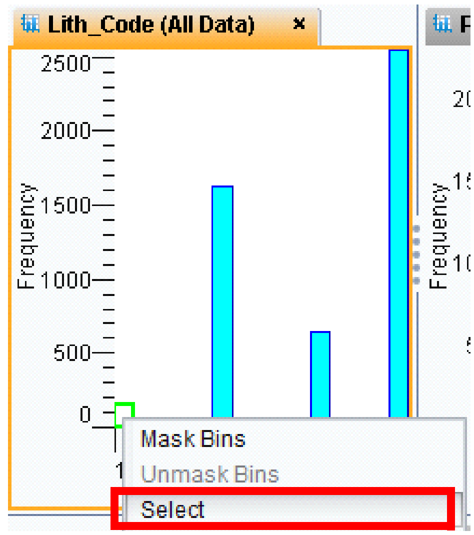
1. High Poro and Perm Eolian Siltstone
2. Eolian Siltstone
3. Carbonate muds (non-reservoir)
4. Carbonate Platform (Dolomite)



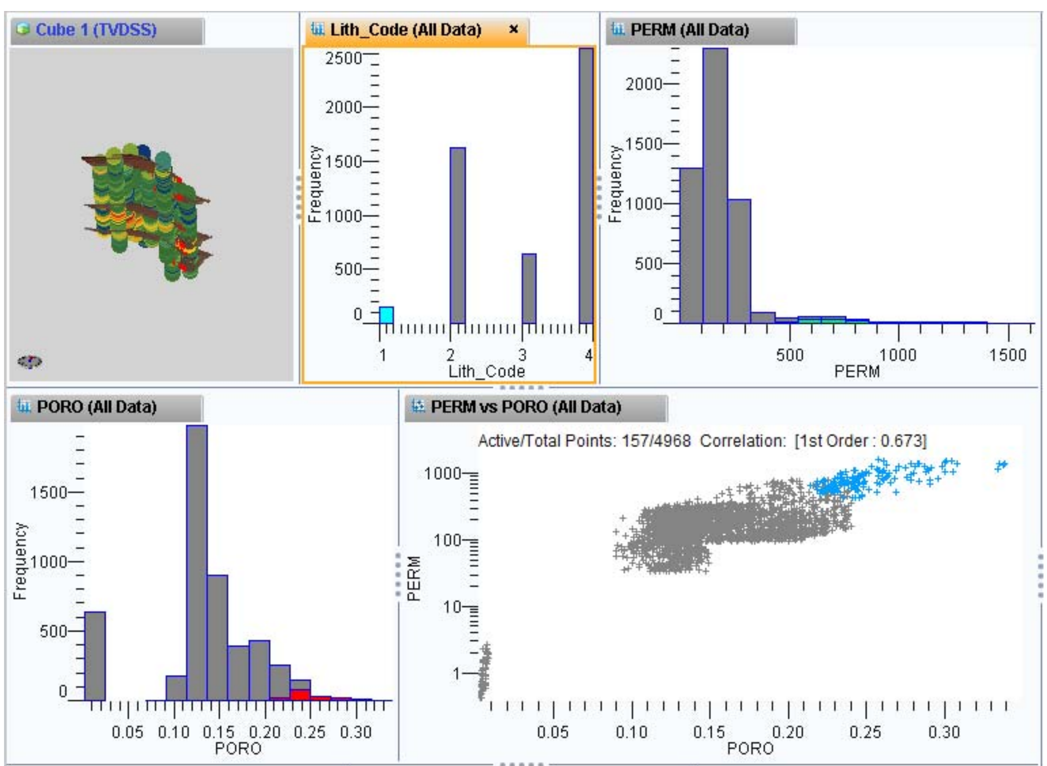
The next step is to create a subset for each Lith_Code to perform a more detailed Data Analysis.

1. With MB1 click the **Lith_Code 1** histogram bar.

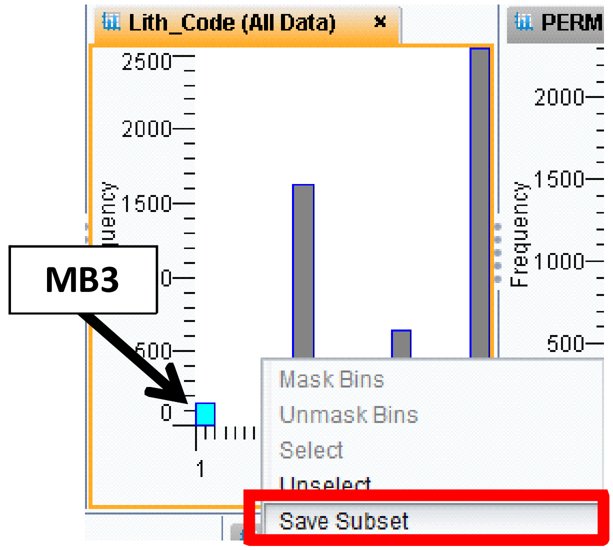
2. With MB3 click **Select**.



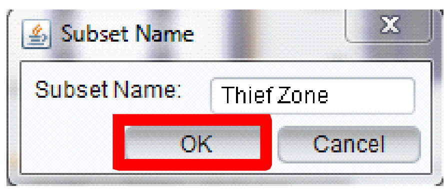
Notice that the other bars turn gray and only a small portion of the PORO and PERM histograms are not gray. Also the cross-plot shows points that are not grayed out. These are data related to Lith_Code=1.



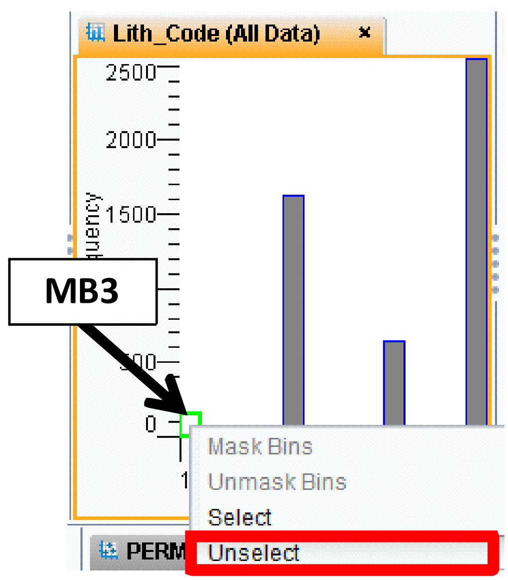
3. MB3 on the **Lith_Code** Histogram and select **Save Subset**.



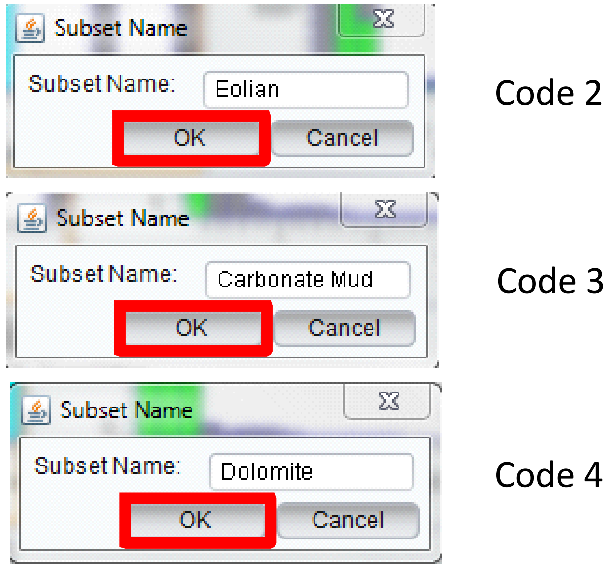
4. Name the subset Thief Zone and click **OK**.



5. Now MB3 on the same bar and click **Unselect**.

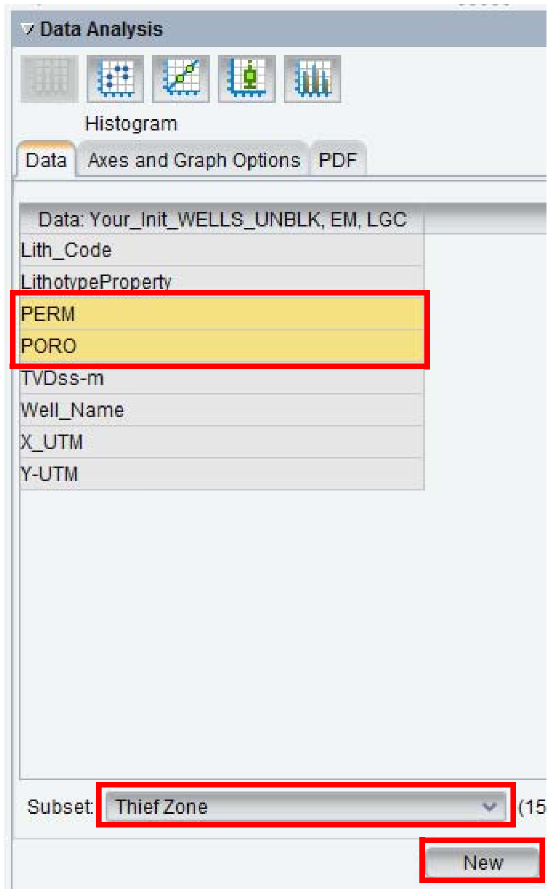


6. Create subsets for codes 2, 3, and 4.



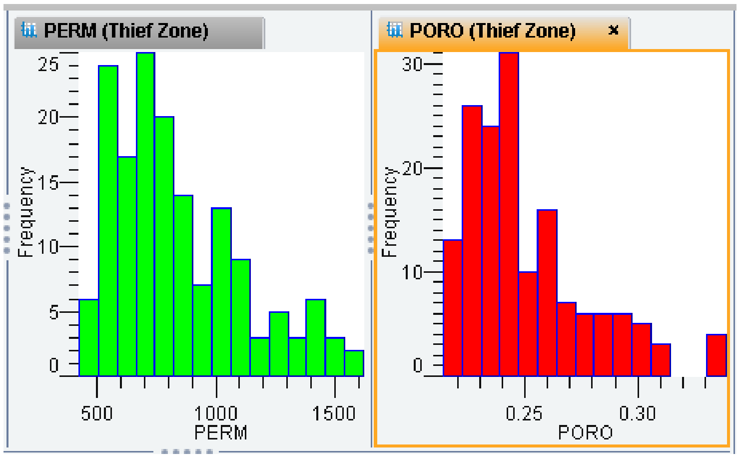
Data Analysis of the Thief Zone Subset

1. Create a histogram for PERM and PORO using the Thief Zone subset.

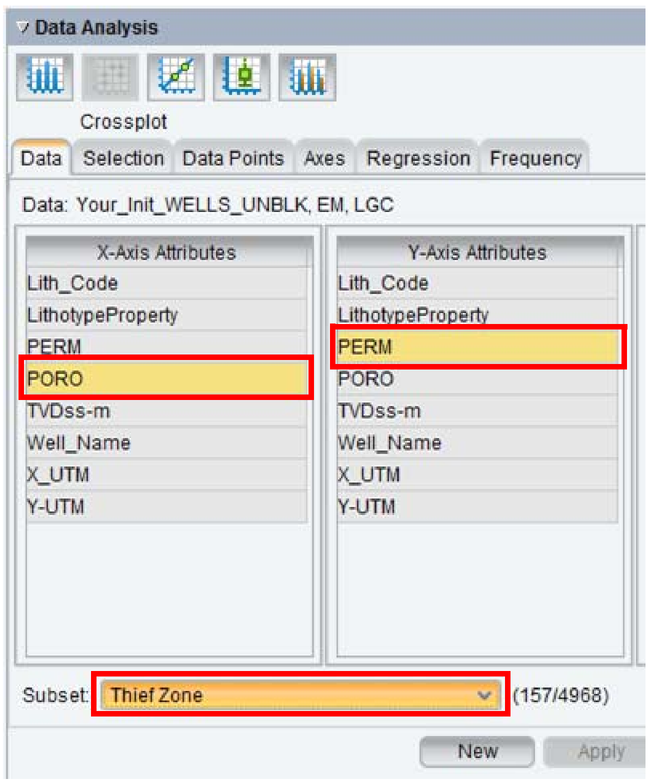


It looks like there could be some porosity outliers just by the look of its histogram; although it could be just a sampling artifact.

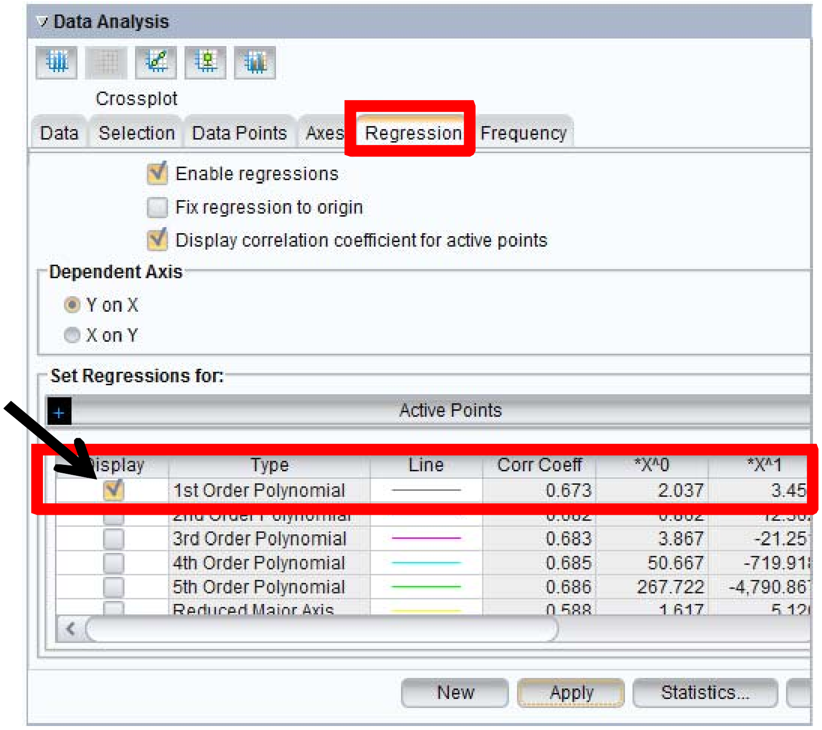
Histograms of Thief Zone subset



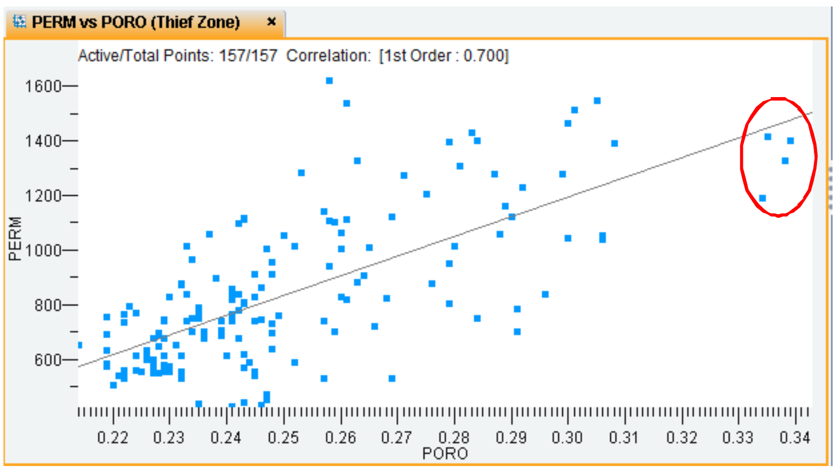
2. Create a cross-plot of the PORO and PERM using the Thief Zone subset.



3. Click the **Regression** tab in the Data Analysis panel and enable **1st Order Polynomial** regression line.



4. Click **Apply**.

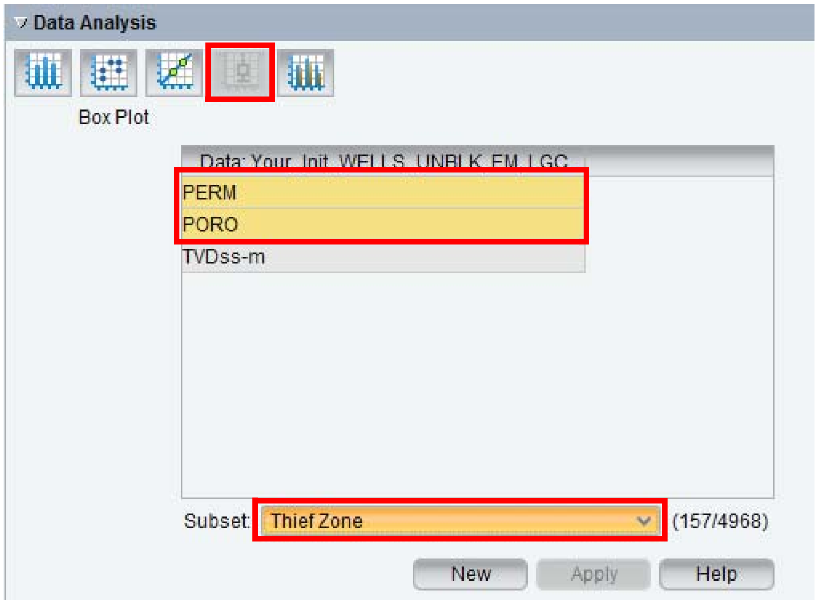


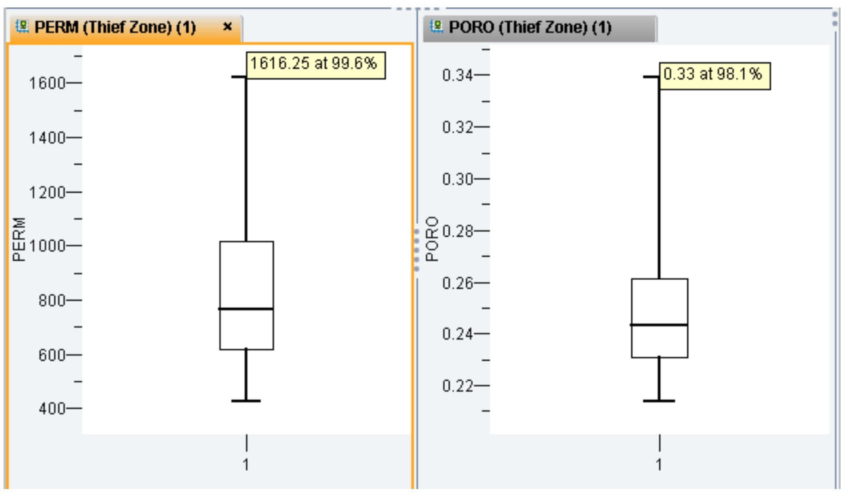
Cross-plot of Thief Zone Subset.

Notice the high correlation between PORO and PERM (0.700) with both plotted on linear scales. The apparent outliers are on trend with other data even though they all fall below the regression line.

We'll use PORO as a collocated attribute during the property modeling step.

5. Create box plots for PERM and PORO using the Thief Zone subset.

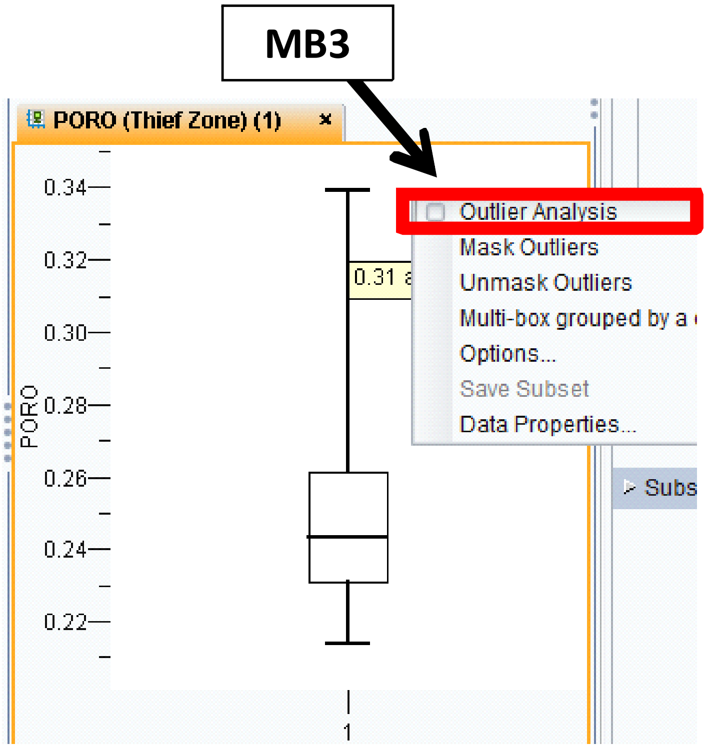




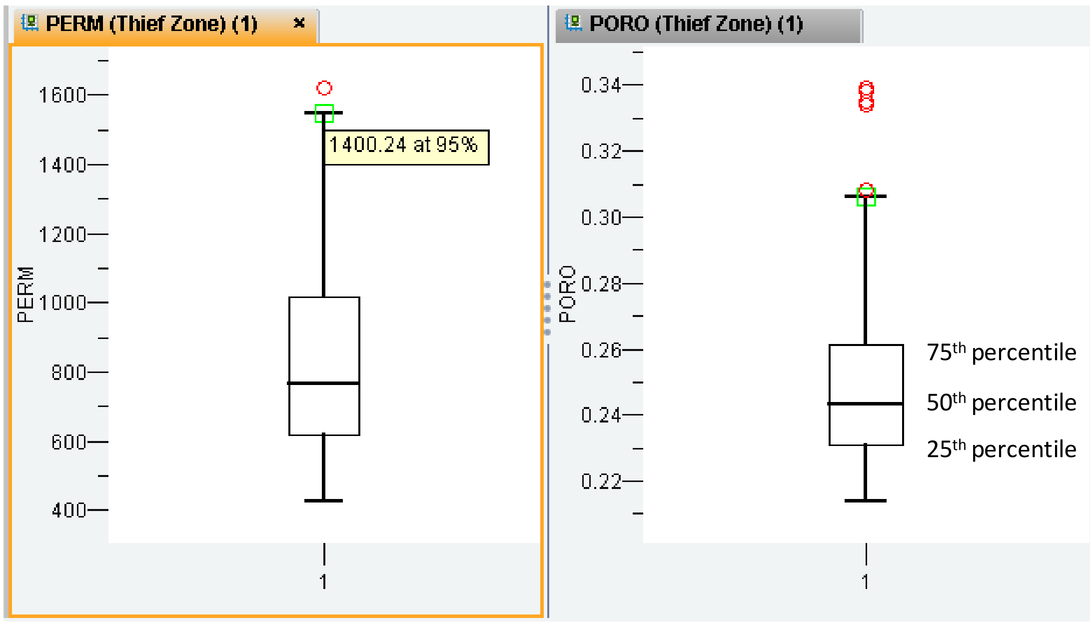
Box Plots of Thief Zone subset.

The Box Plot is a simplified histogram illustrating the skewed nature of the data (towards the high values; a long tail). Box Plots can be used for outlier analysis to further clean the data.

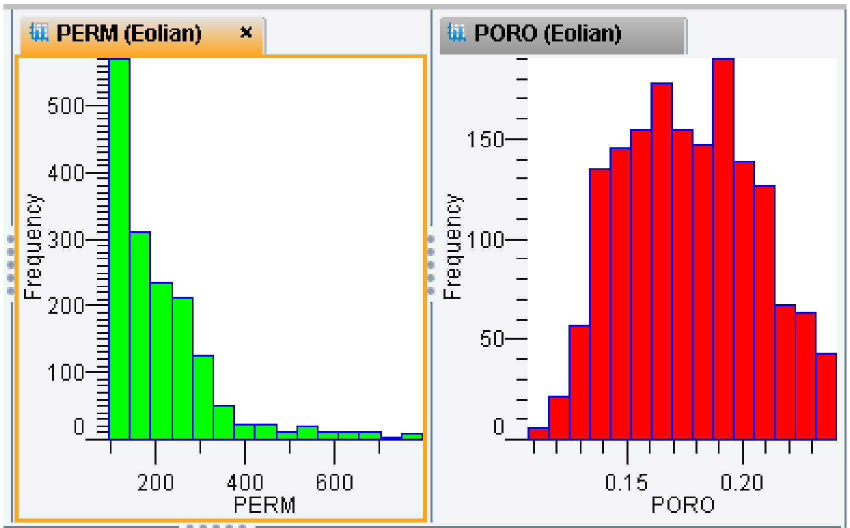
Activating the Outlier Analysis in each Box Plot highlights some potential outliers; those data values that differ significantly from the 75th percentile.



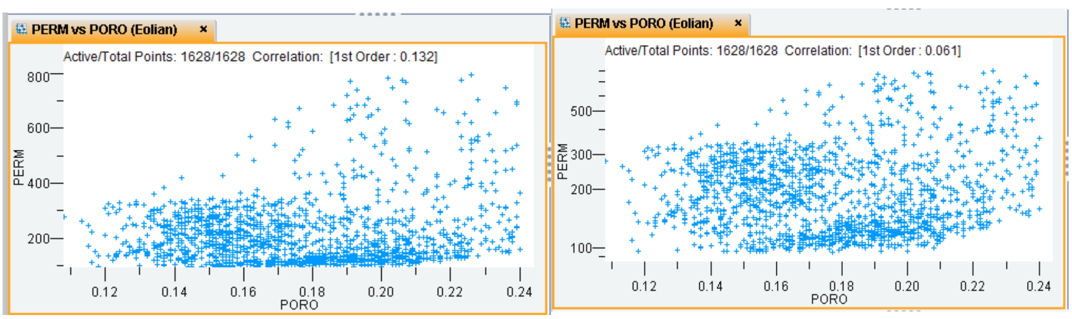
For PERM, the differences are small and about 1.5 PUs for PORO. In this case we won't consider these as bad data, but just high values that are on trend with the other Thief Zone data.



Data Analysis of Eolian Subset



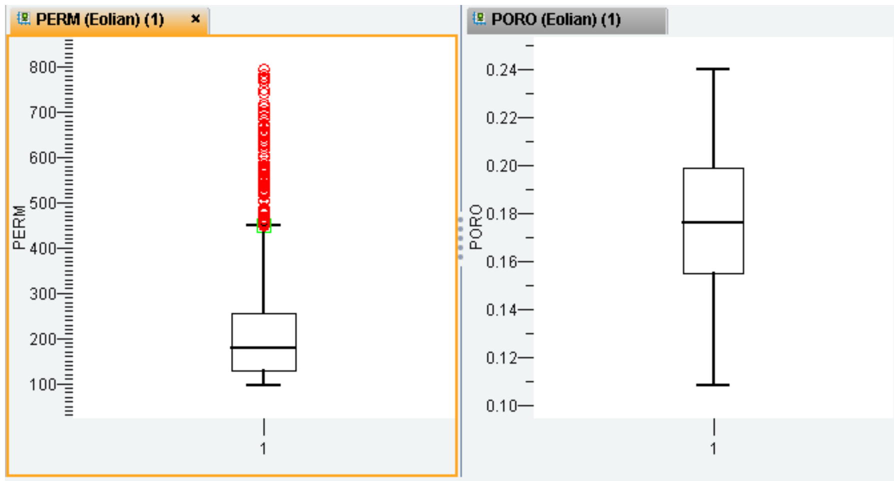
Histograms of PERM and PORO using the Eolian subset.



Cross-plots of PORO vs PERM on a linear (L) and logarithmic (R) scale. There is no correlation between these properties.

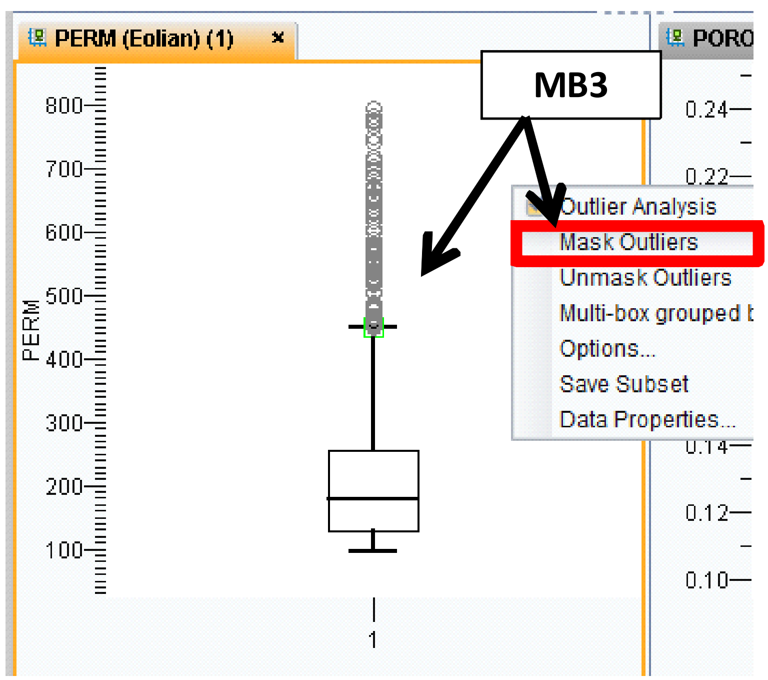
1. Create Box Plots for PERM and PORO using the Eolian Subset.

2. Use MB3 to activate Outlier Analysis.



Box Plots of Eolian subset.

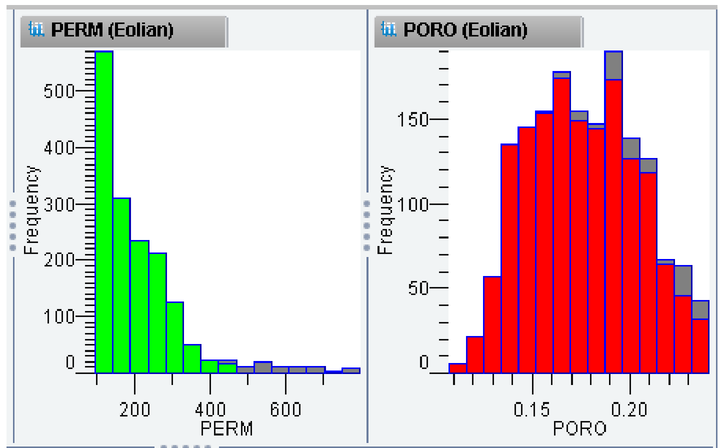
Outlier Analysis indicates many possible high PERM outliers in the Eolian subset, but none for PORO.

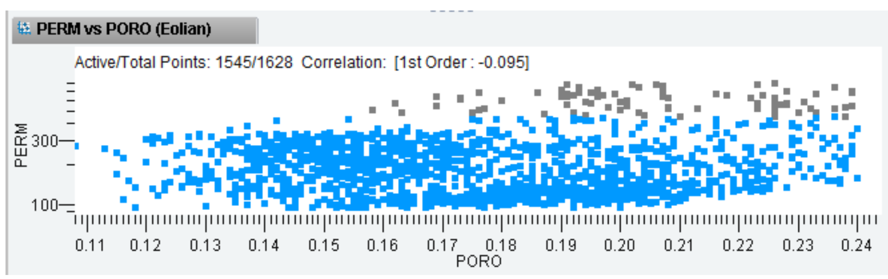


Box Plot of PERM Eolian subset.

Masking the outliers turns them gray in all the Data Analysis plots and indicates which porosity values are paired with the PERM outliers.

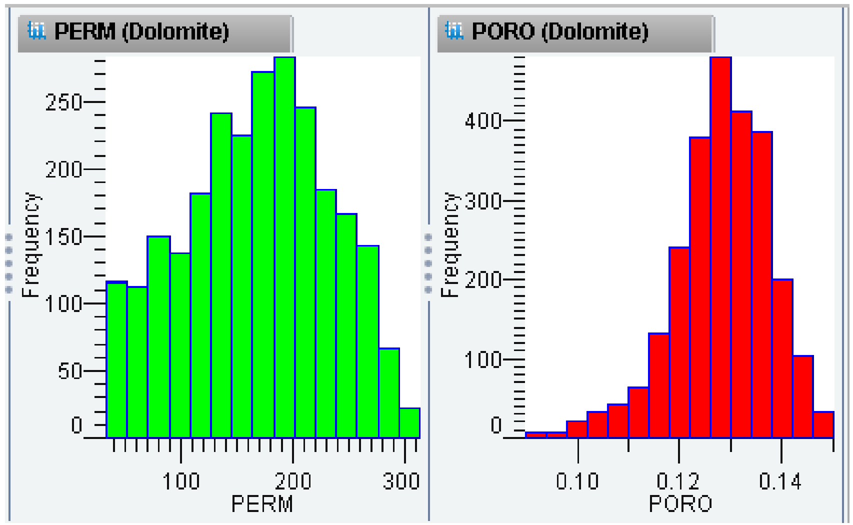
The histograms don't really look odd and outlier analysis on a property like permeability isn't always reliable due its most often highly skewed distribution. So we'll again consider this as good data.



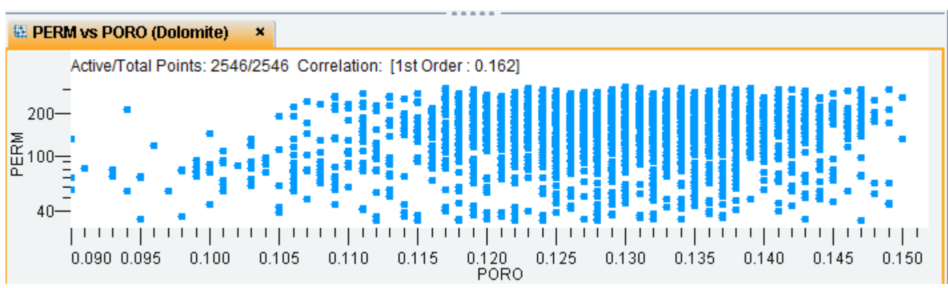


Unmask the outliers.

Data Analysis of the Dolomite Subset



PERM and PORO histograms of the Dolomite subset.

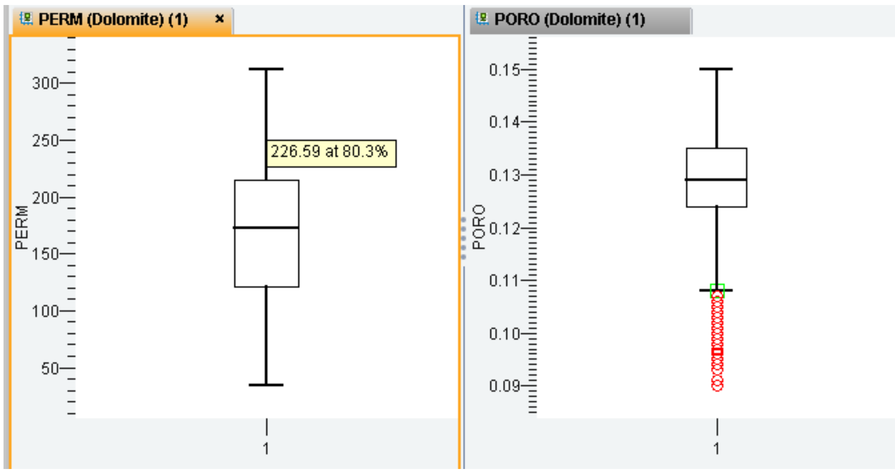


Y Axis is set to Logarithmic.

There is no correlation between these properties.

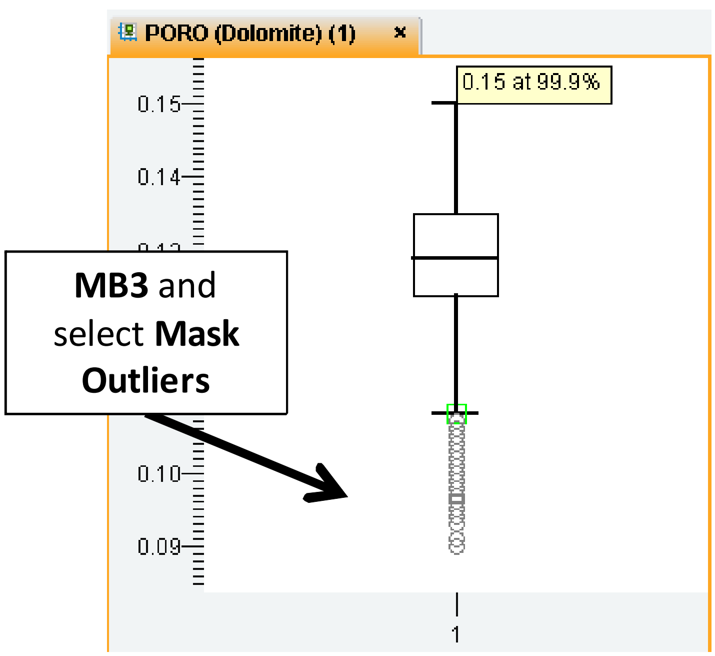
1. Create Box Plots for PERM and PORO using the Dolomite subset.

2. Use MB3 to activate **Outlier Analysis**.



Box Plots of Dolomite subset.

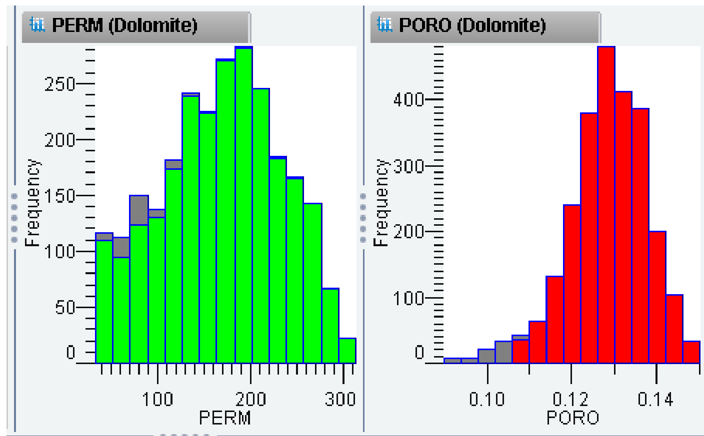
Outlier Analysis indicates many possible low PORO outliers in the Dolomite subset, but none for PERM.



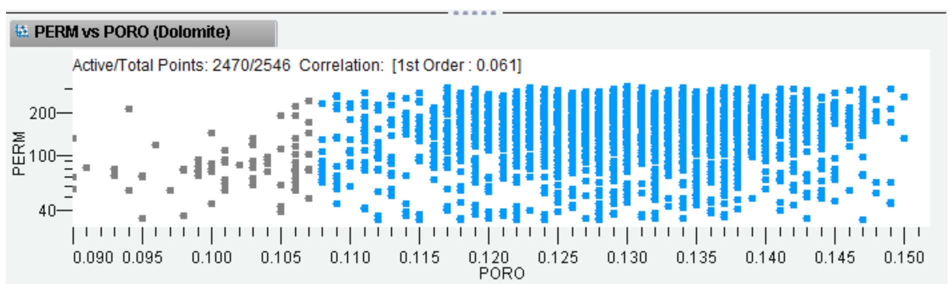
Box Plot of PORO Dolomite subset.

Masking the outliers turns them gray in all the Data Analysis plots and indicates which porosity values are paired with the PORO outliers.

The histograms don't really look odd and outlier analysis on a property like permeability isn't always reliable due its most often highly skewed distribution. So we'll again consider this as good data.



Y Axis is set to Logarithmic.



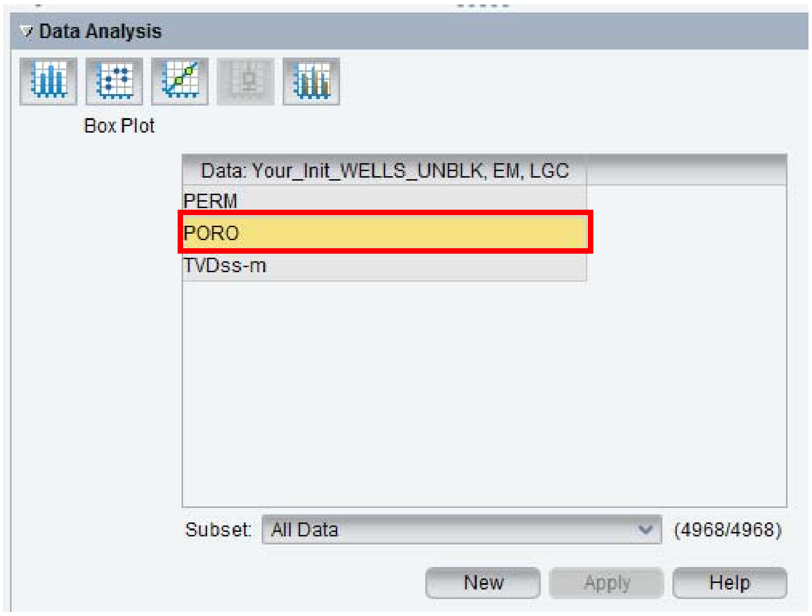
Unmask outliers.

Multi-Box Plot Displays

The Box Plot has two other display options for additional analysis:

1. Multi-box grouped by a discrete property (eg., by lithology or by well)

2. Options (provides additional display properties on the Box Plot)

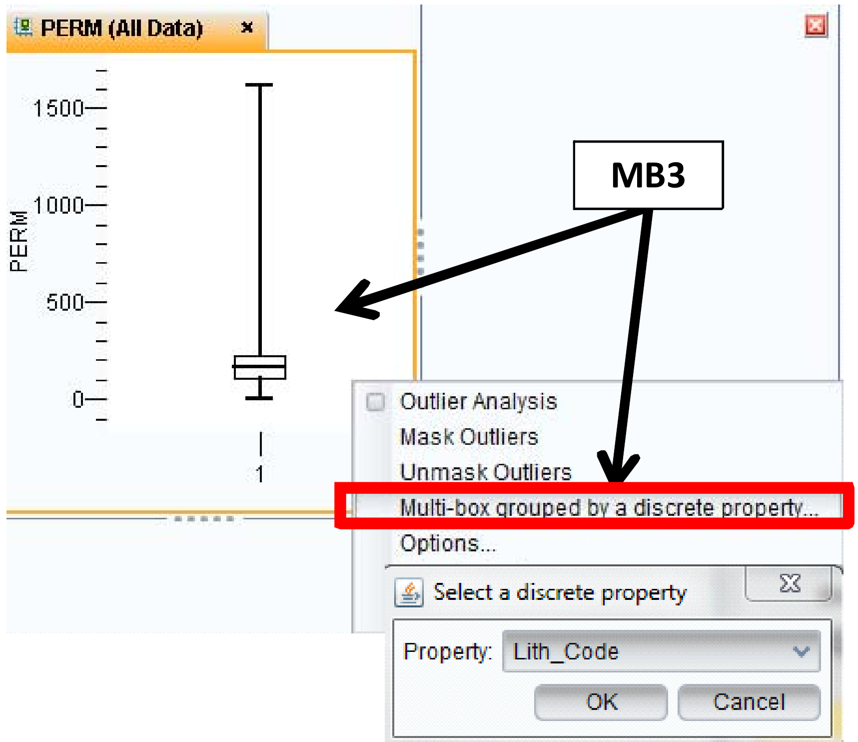


To make a Multi-box plot display,

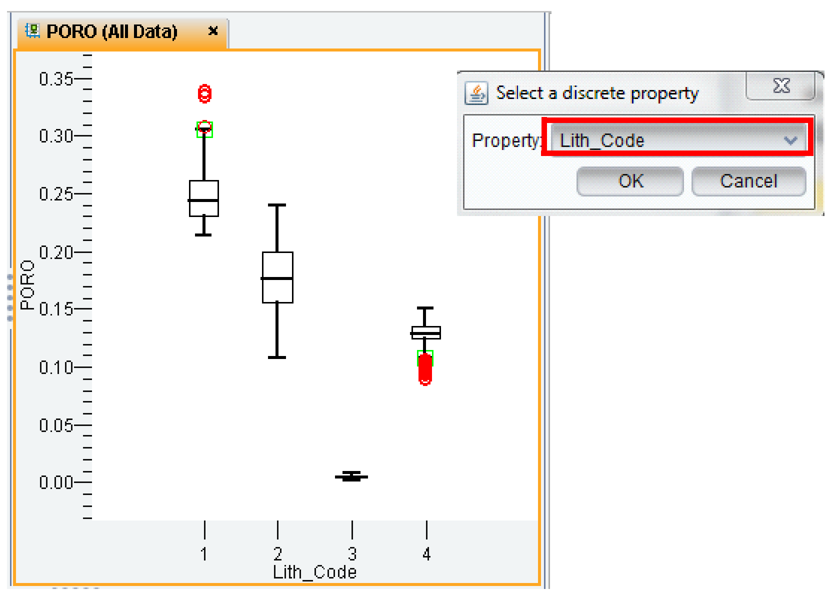
1. Select **PORO**.

2. Subset = All Data > click **New**.

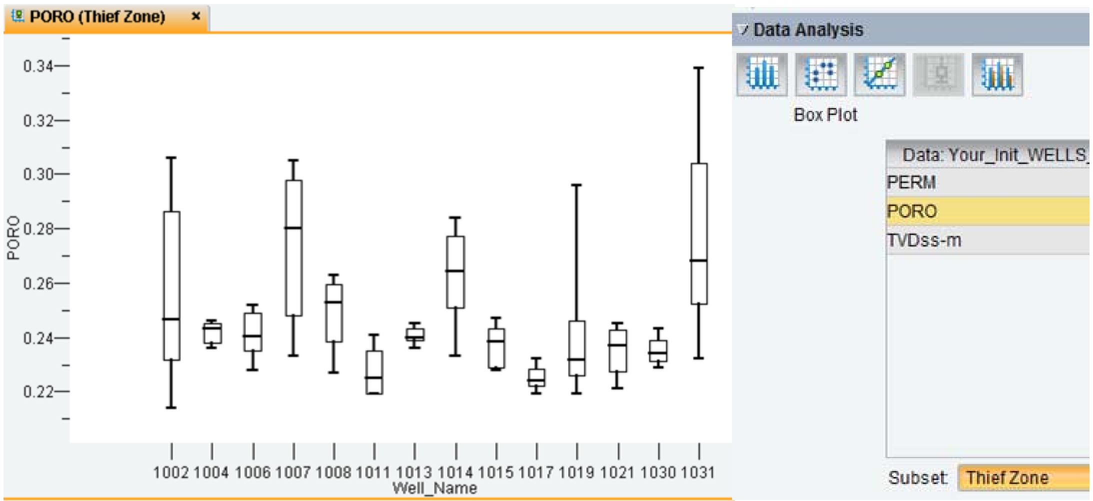
3. Use MB3 on the Box Plot and select **Multi-box grouped by a discrete property**.



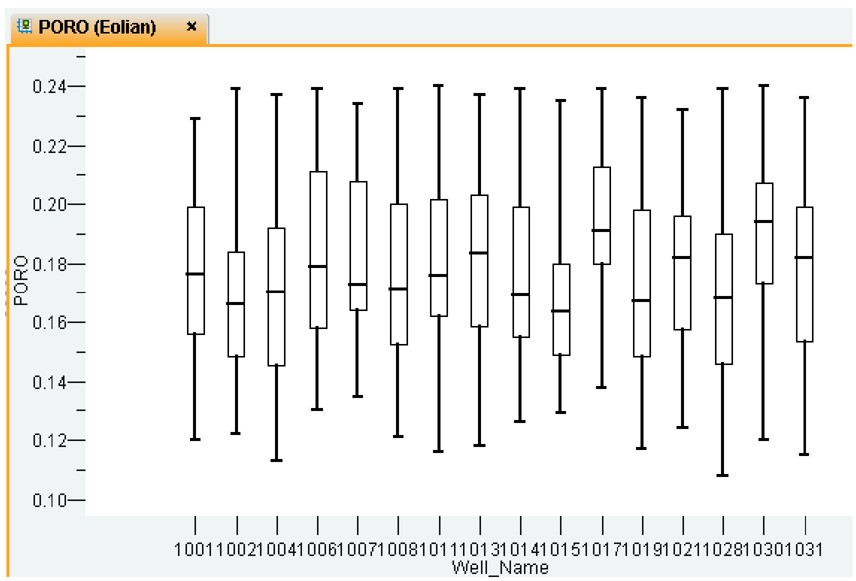
4. Select the property shown for grouping.



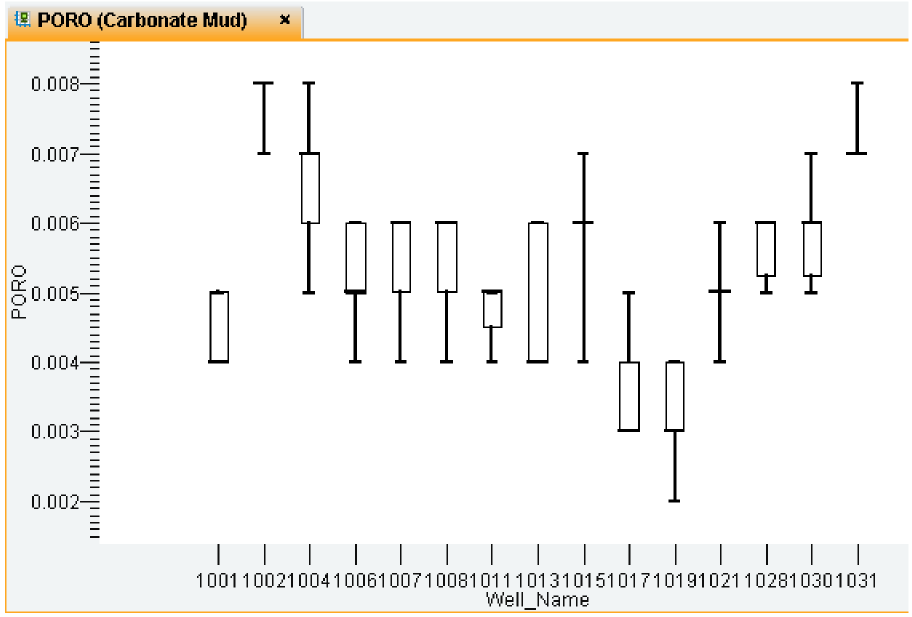
This display provides a graphical overview of the differences or similarities between the data distributions for each lithology code. Outlier analysis is activated.



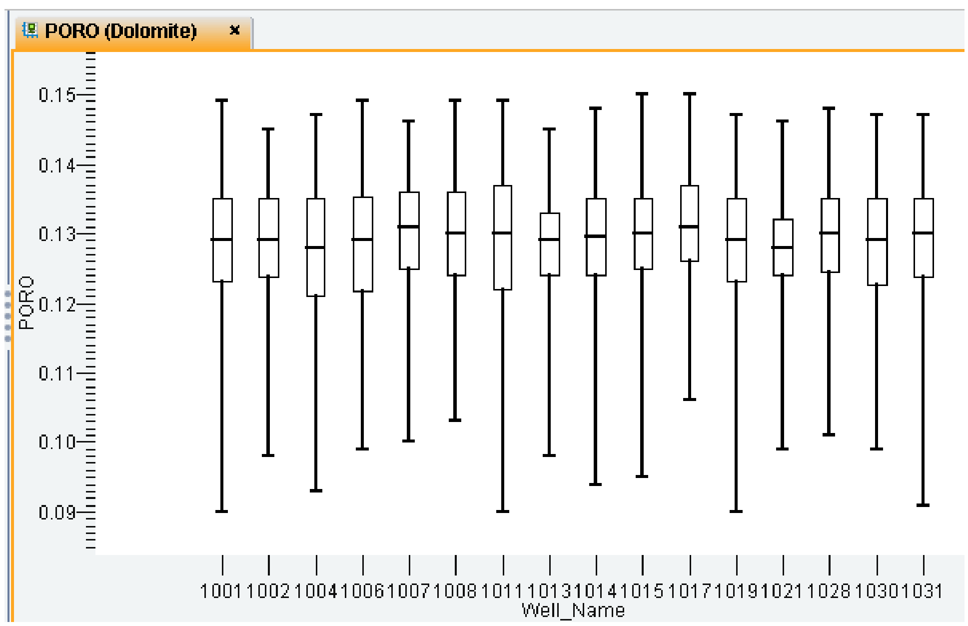
Box Plot display by well for the Thief Zone Subset.



Box Plot display by well for the Eolian subset.



Box Plot Display by well for the Carbonate Mud subset.



Box Plot display for the Dolomite subset.

Variogram Analysis Workflow: Unblocked Data

Spatial Data Analysis or Variography is an extension to the traditional data analysis workflow. In the previous exercises we explored the nature of the three primary properties (Lith_Code, PORO and PERM) in the unblocked well data set.

During Variography Analysis you will compute and model the indicator variograms for each of the traditional variograms for PORO and PERM by lithology code. Because there are only 16 wells, the horizontal variogram will look very unstructured or appear to be a pure Nugget effect. However, the vertical variograms have a short scale structure (spatial correlation) which can be used to determine the vertical cell size when you create the 3D geocellular grid in the next workshop. For this exercise you will use the unblocked data.

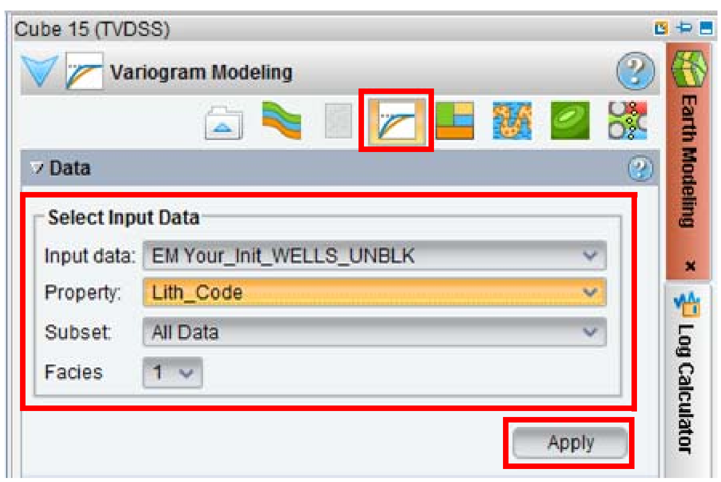
The Variogram Modeling task pane is located under the Earth Modeling tab.

1. Click the **Earth Modeling** tab.

2. Click the **Variogram Modeling** icon.

3. Fill in the Select Input Data section as shown. Note that Facies 1 is the Thief Zone facies.

4. Click **Apply**.

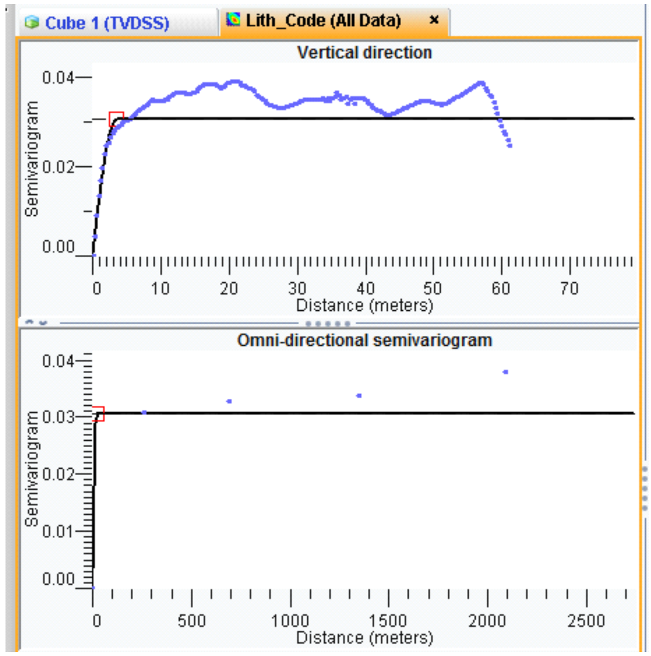


Clicking Apply does the following:

- Determines smart computation parameters based on the data
- Computes the vertical and Omni-directional (horizontal) variograms
- Applies a best-fit variogram model using a default Spherical variogram model type

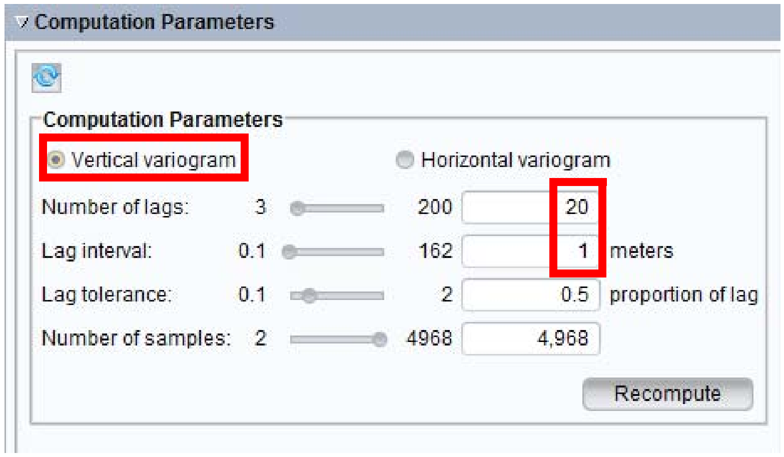
Thief Zone Variogram (Lith_Code = 1)

Thief Zone Variograms (Lith_Code = 1)



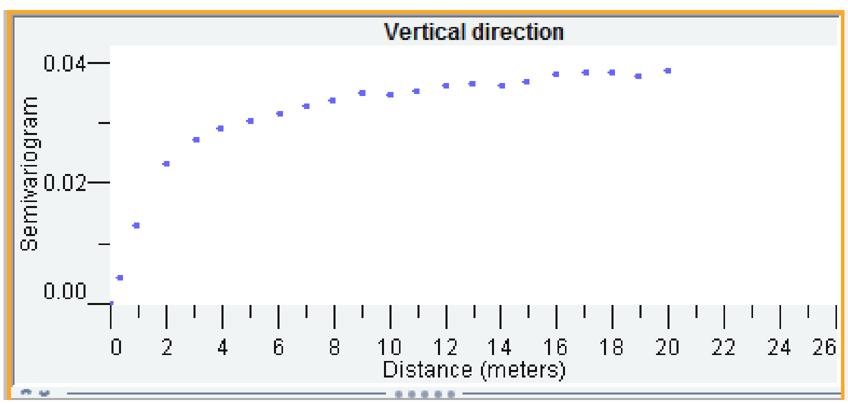
Next you will learn how to adjust the computational and modeling parameters.

Thief Zone Computational Parameters (Vertical Variogram)



Expand the Computation Parameters subtask pane:

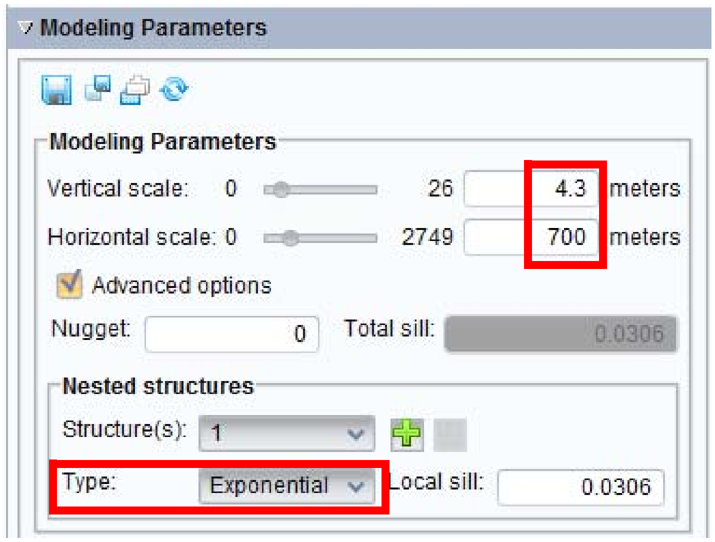
- Select **Vertical Variogram**
- Set Number of Lags = **20**
- Set Lag interval = **1**
- Click **Recompute**



New Vertical Variogram after defining new computational parameters.

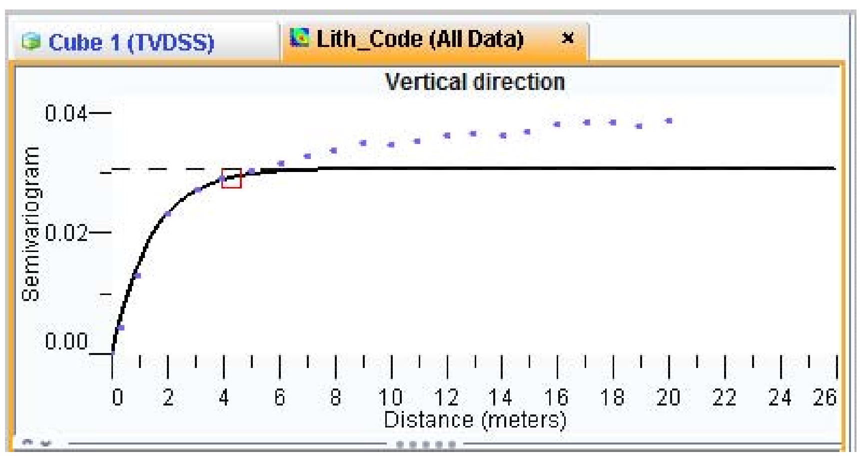
Because there are only 16 wells we won't change the computation parameters for the horizontal variogram.

Thief Zone Modeling Parameters



Expand the Modeling Parameters subtask pane:

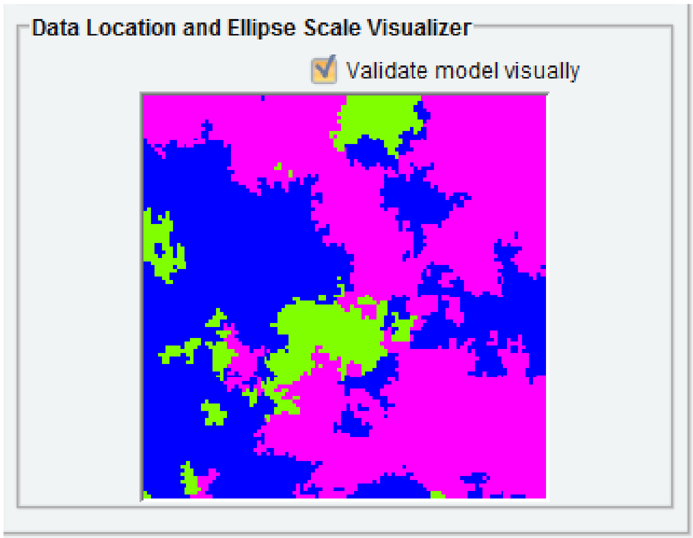
- Set Vertical scale = **4.3**
- Set Horizontal Scale = **700**
- Change Structure Type = **Exponential**



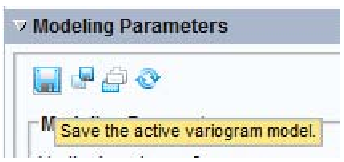
Vertical variogram after defining new computational parameters.

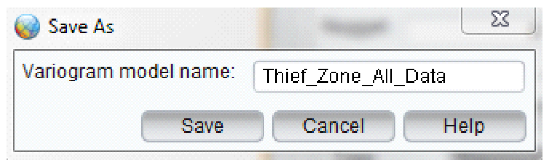
Because there are only 16 wells we won't change the computation parameters for the horizontal variogram.

The Model Visual Validator shows and unconditional indicator simulation using the new parameters. This display is a preview of what a full simulation of the lithologies once computed.



Save the model of the Thief Zone.



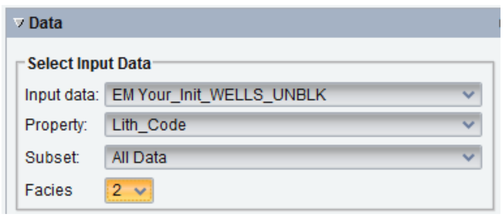


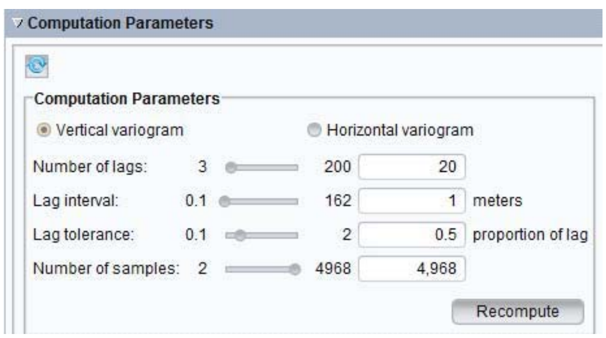
Repeat the previous steps for Lith_Codes 2, 3 and 4.

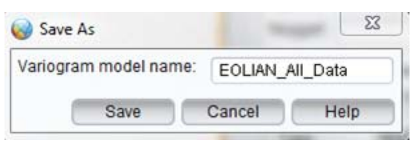
Eolian Variogram (Lith Code = 2)

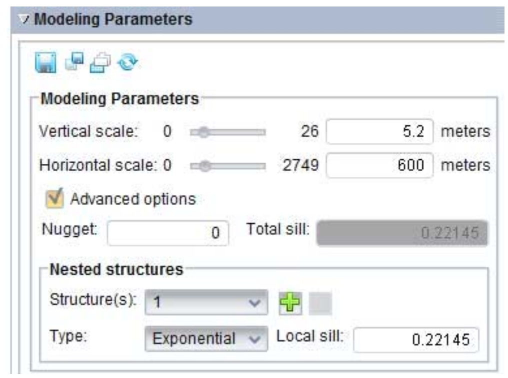
1. Enter the computational parameters and modeling parameters as shown in the following images.

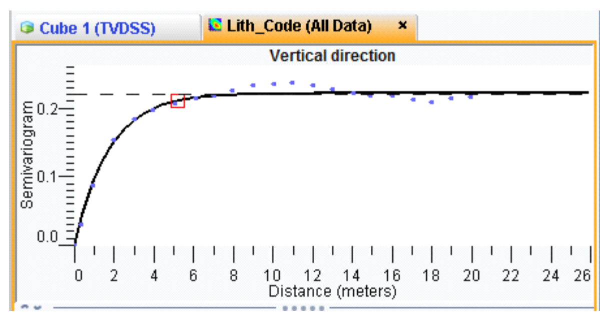
2. Name the variogram and click **Save the active variogram model with a new name**.







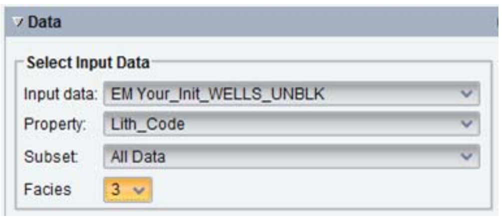


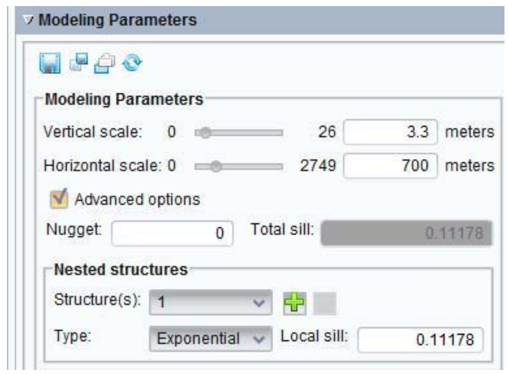


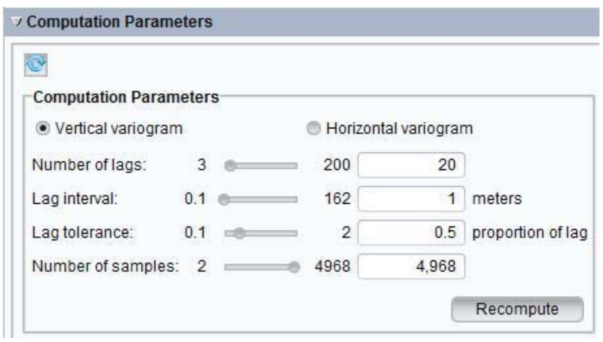
Carbonate Mud Variogram (Lith Code = 3)

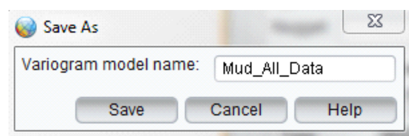
1. Enter the computational parameters and modeling parameters as shown in the following images.

2. Name the variogram and click **Save the active variogram model with a new name**.





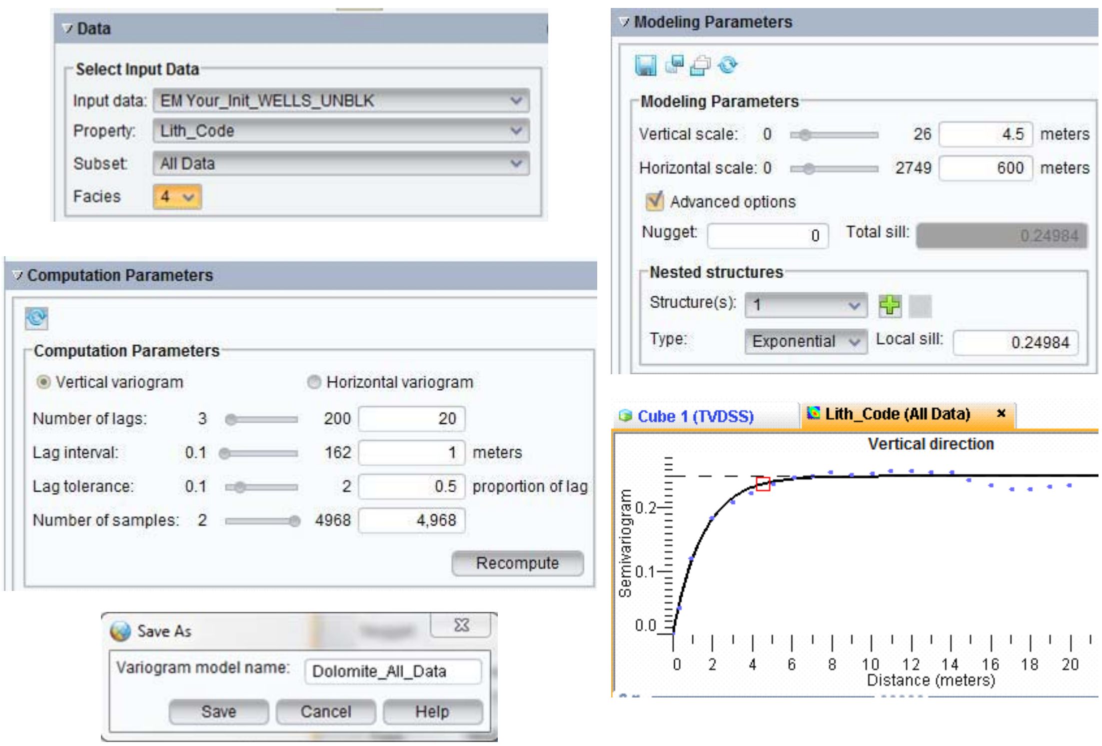




Dolomite Variogram (Lith Code = 4)

1. Enter the computational parameters and modeling parameters as shown in the following images.

2. Name the variogram and click **Save the active variogram model with a new name**.



Vertical Variogram Scale by Environment

Depositional Environment	Vertical Variogram Scales (meters)
Eolian Thief Zone	4.3
Eolian	5.2
Carbonate Mud	3.3
Carbonate Platform (Dolomite)	4.5

Based on the vertical scales, we see that the Carbonate Mud has the shortest scale. Because this is a non-reservoir lithology and presents barriers to flow, it is important to capture this lithology with sufficient resolution in the 3D geocellular grid. Therefore, the maximum vertical cell thickness should be no greater than 1-meter. For this workshop we'll set the vertical resolution to 0.5 meters as the model will be less-than 275,000 cells.

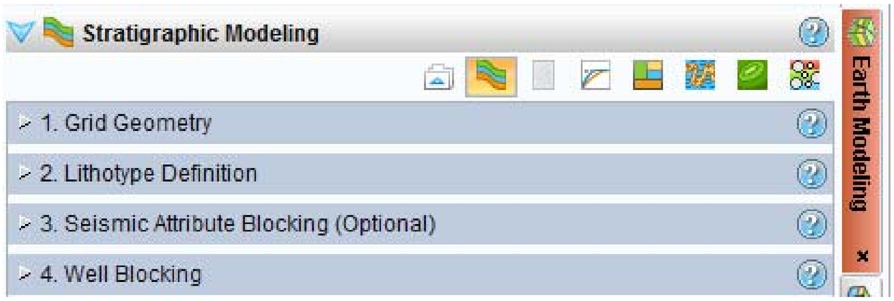
Rule of Thumb: To resolve the thinnest of the lithologies it is best to have a sampling rate (cell thickness) of about 1/3 to 1/6 of the average lithology thickness.

Save the session as (e.g., give it a new incremental value).

Stratigraphic Modeling Workflow

1. Define 3D grid geometry.
2. Define grid subsets for quick modeling and QC.
3. Define lithotypes to model.
4. Block seismic acoustic impedance onto the 3D grid.
5. Block well data.

The Stratigraphic Modeling task pane is accessed by clicking on the  icon. The task pane has four subtask panes:



1. **Grid Geometry**: Where parameters are defined to specify the geometry of the 3D geocellular grid.
2. **Lithotype Definition**: Define what lithologies to model, regroup and create a color map.
3. **Seismic Attribute Blocking (optional)**: If you want to use a seismic attribute during the modeling it is necessary to block the information into the grid created in Step 1.
4. **Well Blocking**: Resamples (blocks) the well data based on the vertical cell thickness of the 3D geocellular grid.

Grid Geometry

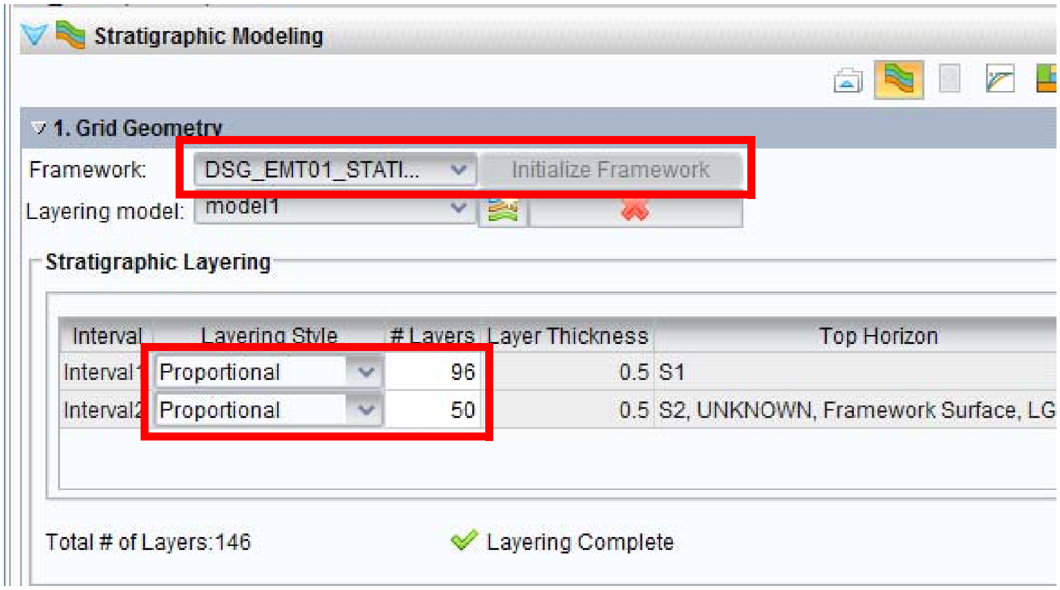
Within this subtask pane you will:

1. Select the static framework and intialize it.
2. Define the Layering style.
    a. Parallel to Top of Base (requires layer thickness)
    b. Proportional (required number of layers)
3. Define the gridding parameters.
    a. X & Y origin
    b. X & Y cell size
    c. Grid rotation angle
    d. Digitize (AOI (requires a Map View)
    e. Delimit reservoir base by constant z (depth)
    f. User specified Grid name
4. Select the static framework and initialize it.

5. Define the Layer style.

   a. Interval 1: Proportional (96 layers)

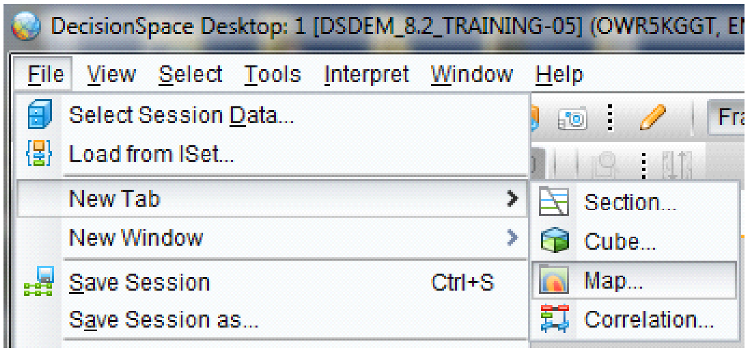
   b. Interval 2: Proportional (50 layers)



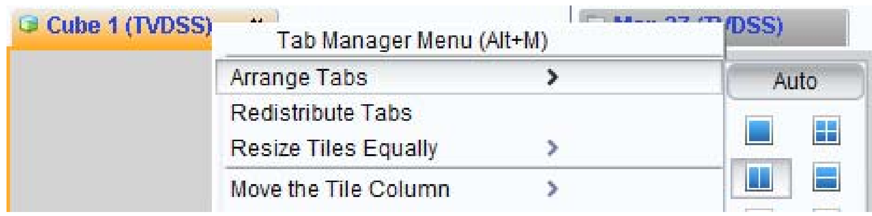
Proportional layering was chosen because the isochore thickness is based on differential compaction and not erosional removal of sediment.

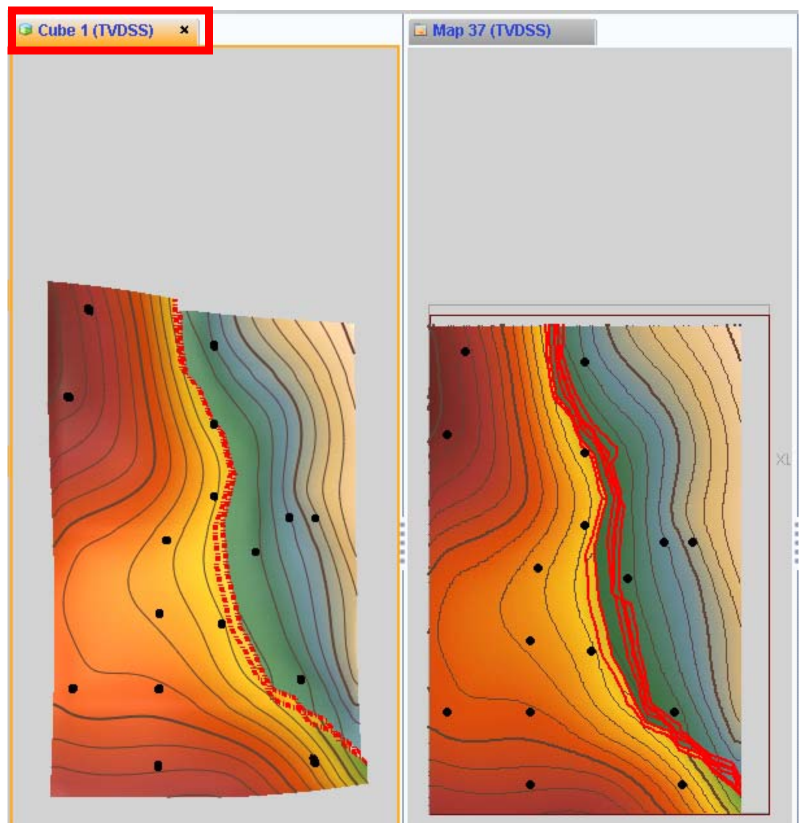
Digitize an AOI

Because you will learn how to digitize an AOI it is necessary to have a Map View tab open, so from the main Menu select **File > New Tab > Map View**.



MB1 on the **Cube View** tab and then with MB3 **Arrange Tabs** and select the  icon.



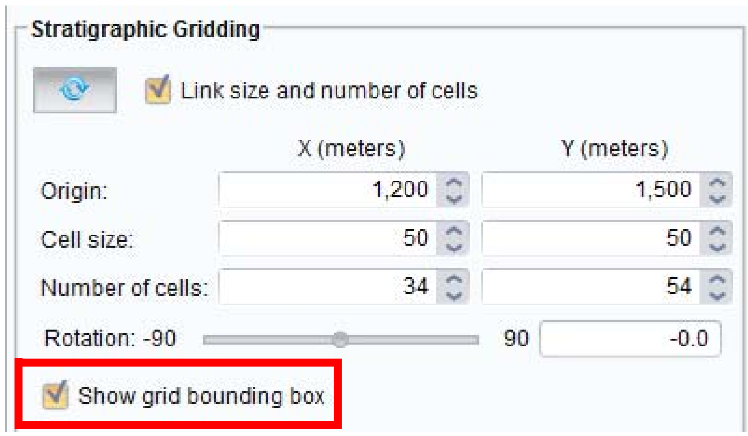


Framework displayed in Cube and Map Views.

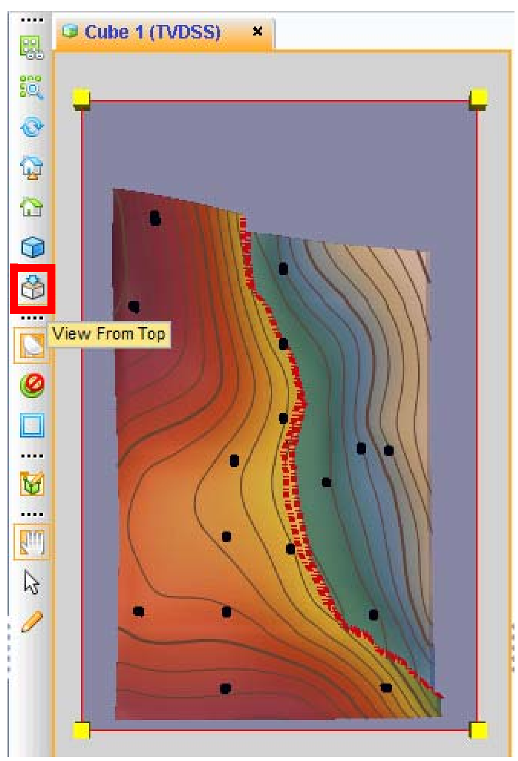
Digitizing the Area-of-Interest (AOI) requires the interaction between Cube and Map View. Perform these steps.

1. MB1 on the **Cube** tab to activate it. You should still have the framework displayed, if not, display it.

2. Check on the **Show grid bounding box** checkbox under Stratigraphic Gridding.



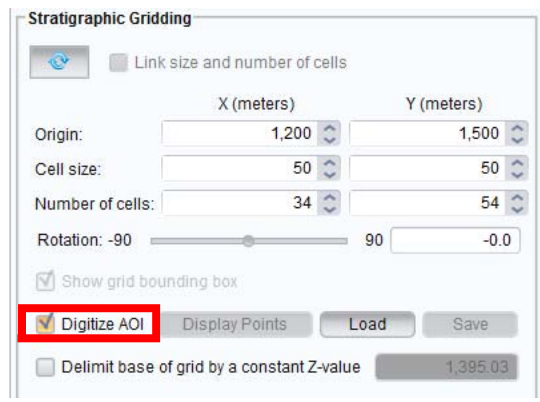
3. In the Cube View, click the **View from Top** icon. 



Create an AOI polygon

1. MB1 on the **Map View** tab to make that the active view.

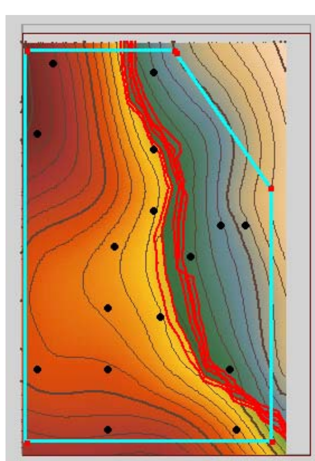
2. Check on **Digitize AOI** in Stratigraphic Gridding which allows you to digitize an AOI polygon within the Map View.

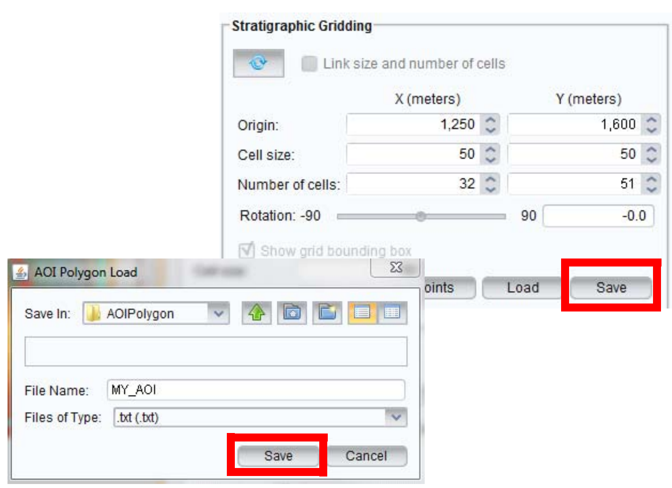


The polygon you digitize probably won't match exactly what's illustrated, which is fine. The exercise illustrates how to create an AOI if necessary.

3. Use MB1 to digitize the polygon and MB2 to close it.

4. Click **Save** to save the polygons as an ASCII.txt file, which can be used again if necessary and give it a name.





In Stratigraphic Gridding

1. Provide a Grid name.

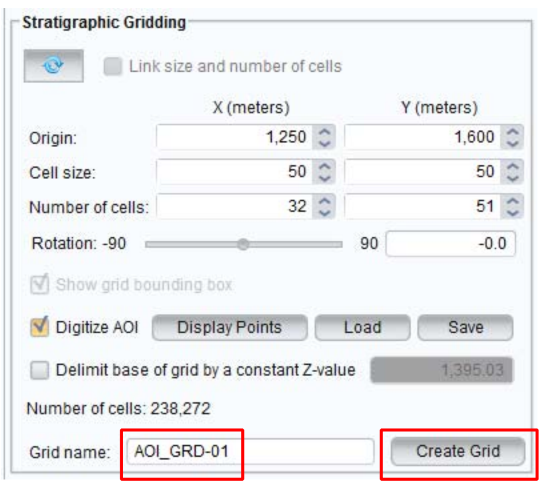
2. Click **Create Grid** which creates the empty 3D geocellular grid with proportional layers in both intervals.

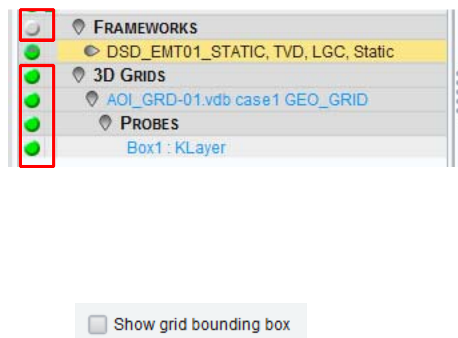
3. Toggle **off** the Show Grid Bounding Box checkbox in the Stratigraphic Gridding task pane (make Cube View active).

4. In the Inventory with the Cube View as the active view.

    a. Turn off the framework.

    b. Turn on the Box1 Probe, if not already activated.





In the Cube View

1. Use MB1 to rotate the grid.

2. With the cursor on the grid, use MB3 and select **Display Properties**.

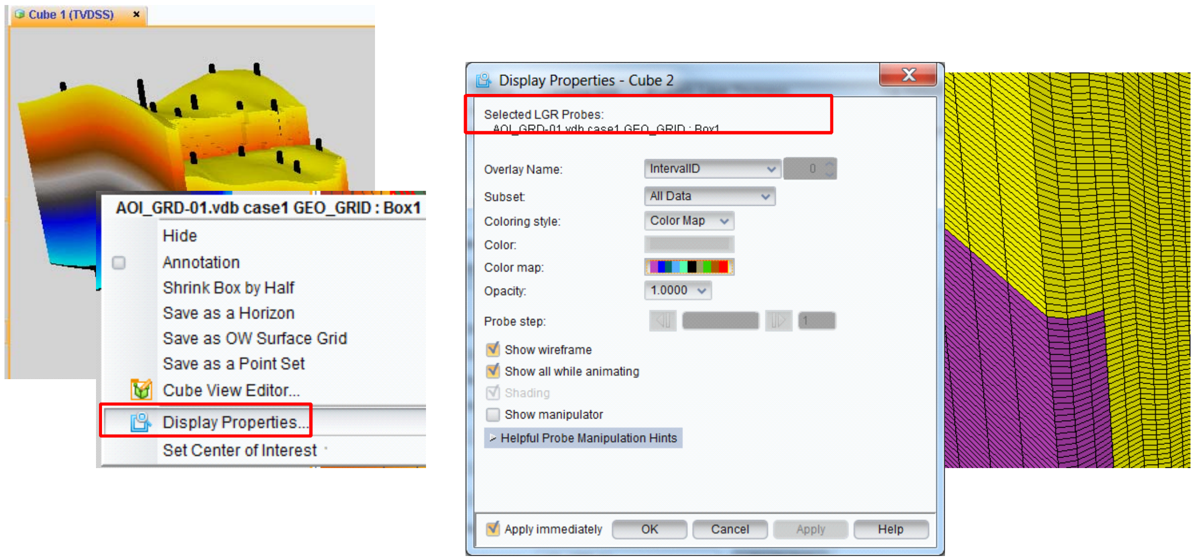
3. In the Display Property Editor, do the following:

    a. Change Overlay Name to IntervalID.

    b. Change colormap to **SYSTEM** > **BLOCK**.

    c. Check on **Show wireframe** and then click **OK**.

4. Now go back to the Cube View and zoom in to see the interval and layering.



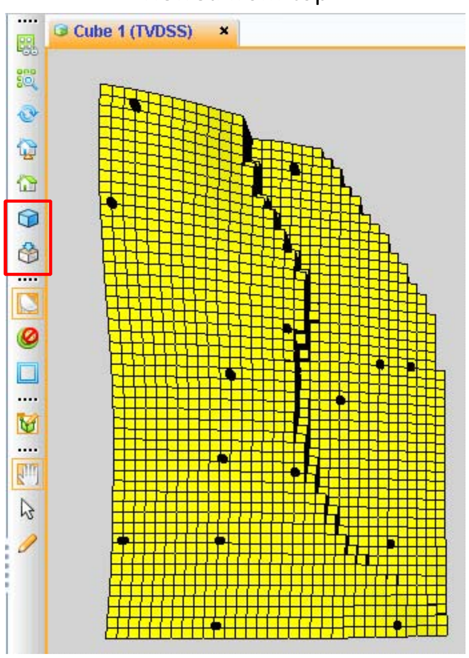
Create Grid Subsets

Now that you created a grid using the AOI option, you also have the option to create subsets of the grid, such as a critical cross-section, or perhaps your reservoir engineer wants you to create a sector model around a few key wells. Note that grid sub-setting can be done at any time and then those subsets can be modeled independently of the entire grid.

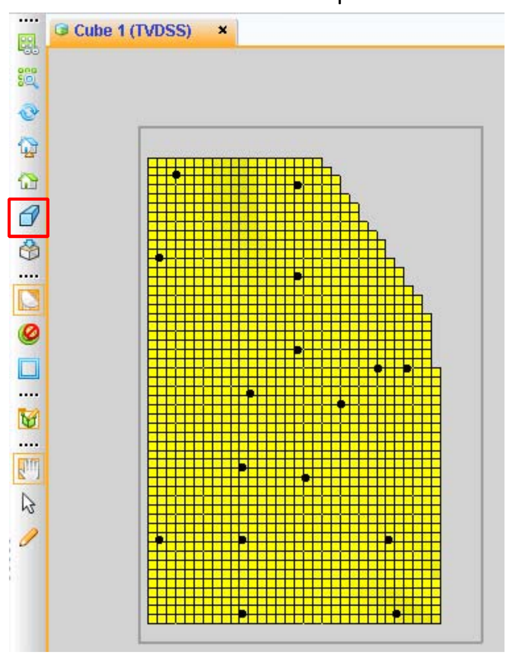
Using grid subsets allows you to test modeling parameters quicker in a See it Now mode which executes much faster than running a whole field model.

To create grid subsets make the Cube View the active mode and then view the grid from the top (with the wireframe on) and viewed with an Orthographic view.

Perspective View and viewed from top



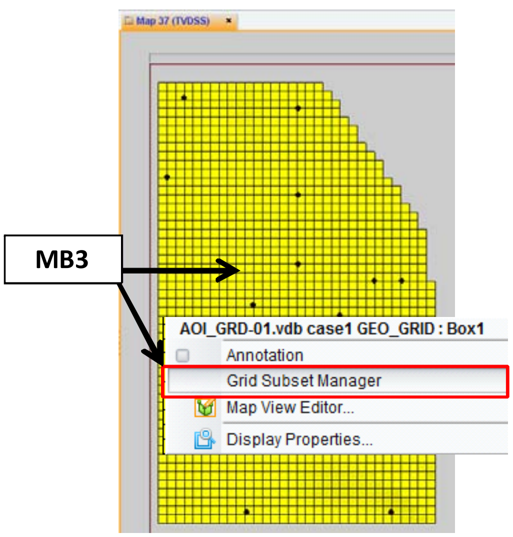
Orthographic View and viewed from top

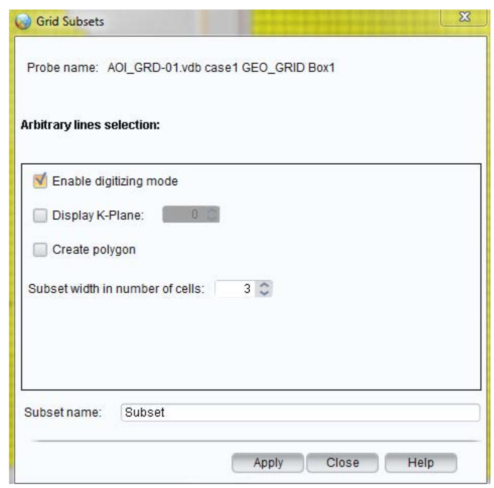


Creating cross-sectional subsets is similar to digitizing the AOI, except that you don’t close a polygon.

1. Make the Map View the active view.
2. In the inventory:
   a. Turn off the framework.
   b. Turn on the Box1 probe which is located under the grid name.
3. Display the Unblocked point set in both views. In the Display Property Editor use a solid black color and a solid circle of size 8.

4. In Map View, use MB3 and select **Grid Subset Manager**.





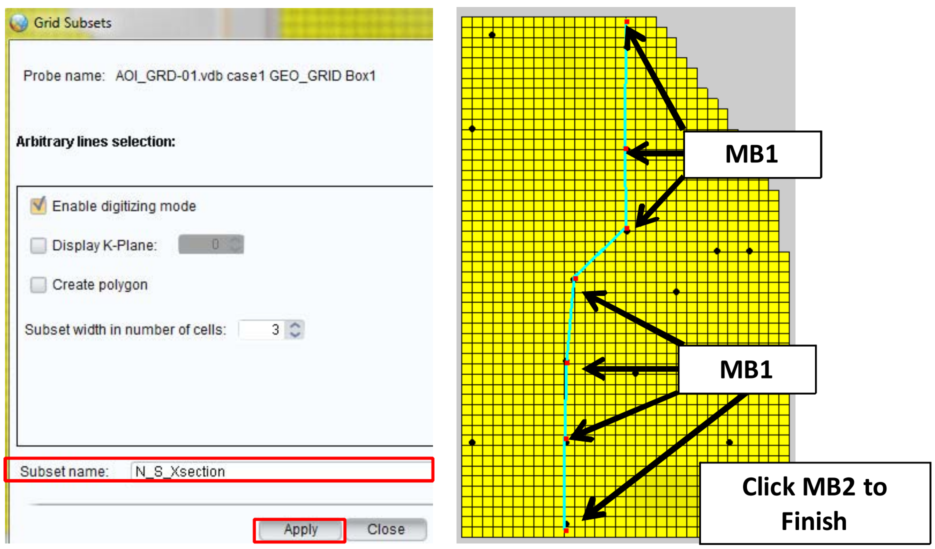
Digitize a N-S Cross-Section Through Some Key Wells

1. Use default options as shown in the dialog box.

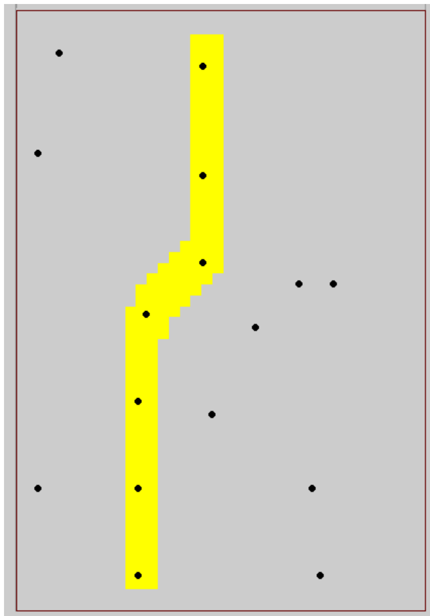
2. Provide a Subset name.

3. Digitize the cross-section with MB1 as shown, and MB2 to end the action.

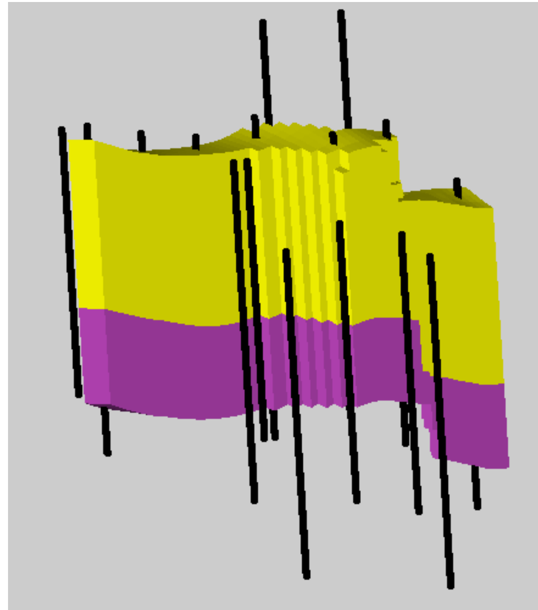
4. Click **Apply** and then **Close**.



Cross-section in Map View



Cross-section in Cube View



Digitize a Sector Polygon Around Some Key Wells

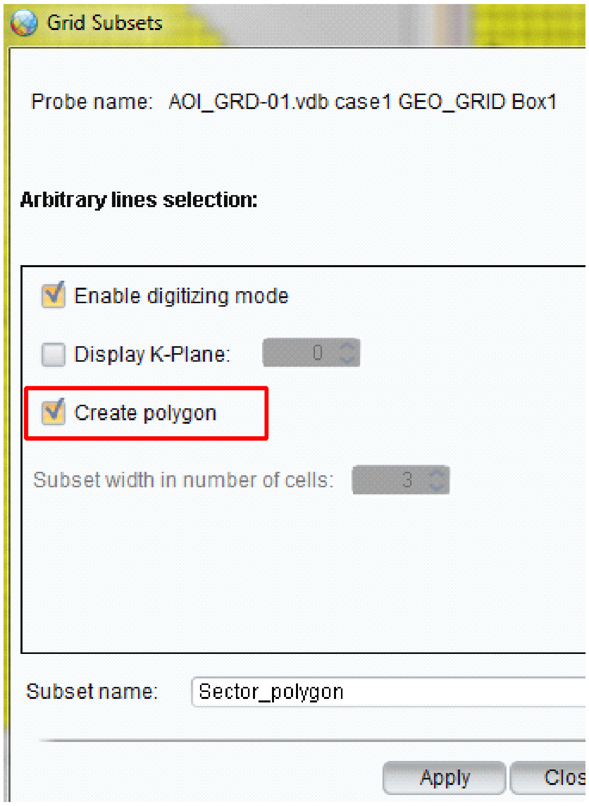
1. In the map view, MB3 on cross section of grid select **Display Properties** and choose all data for the Subset and then click **OK**.

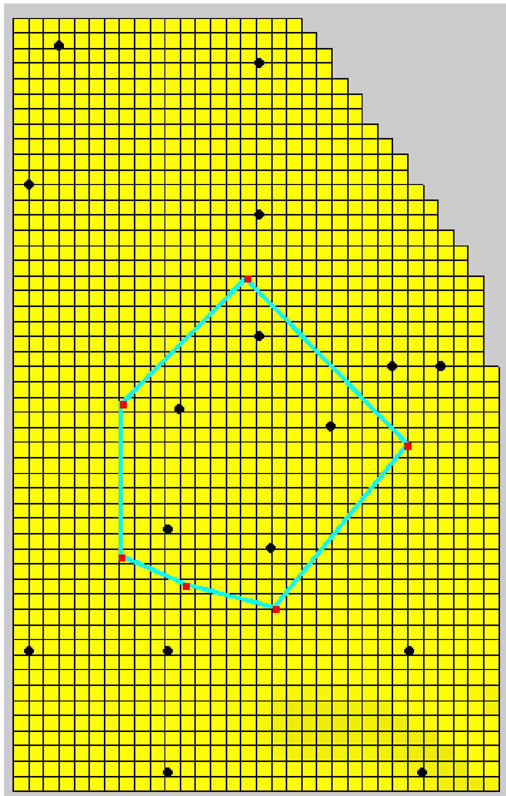
2. Use MB3 and select **Grid Subset Manager** and then **enable Create polygon** as shown in the dialog box.

3. Provide a Subset name.

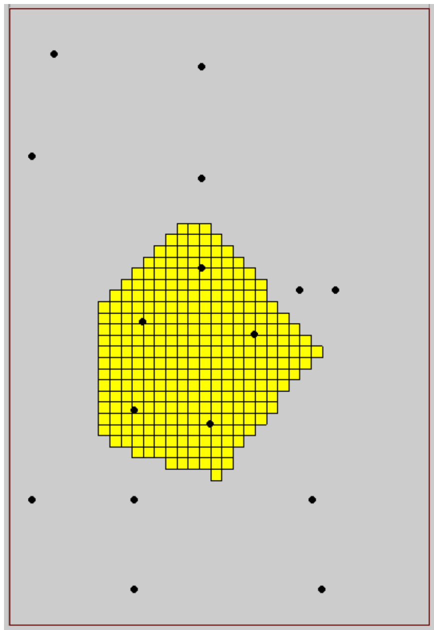
4. Digitize the polygon with MB1 and use MB2 to close polygon.

5. Click **Apply** then **Close**.

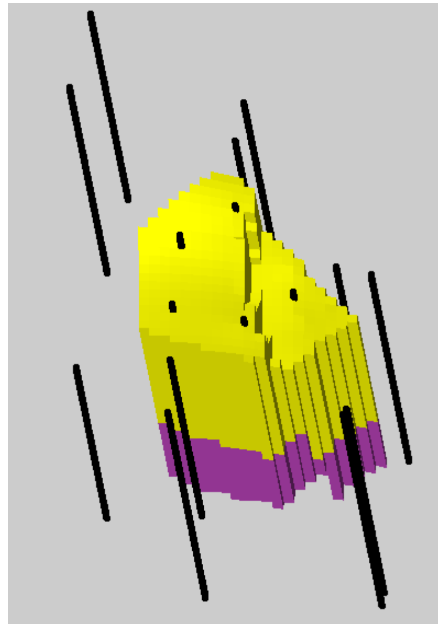




Polygon in Map View



Cross-section in Cube View



6. Save the session.

Process Check

So far you have created

1. Sealed framework with 3 horizons and 1 fault
2. A 3D geocellular grid with a user defined AOI
3. Subsets on the 3D grid
    a. N-S cross-section
    b. Sector polygon

Remaining Stratigraphic Workflow:

1. Define lithotypes to model
2. Block seismic acoustic impedance onto the 3D grid
3. Block well data

Lithotype Definition

In DecisionSpace® Earth Modeling it is possible to model up to 14 lithotypes per reservoir interval.

Definition: For reservoir modeling, a *lithotype* consists of two or more lithologies that share a common geometry and petrophysical properties.

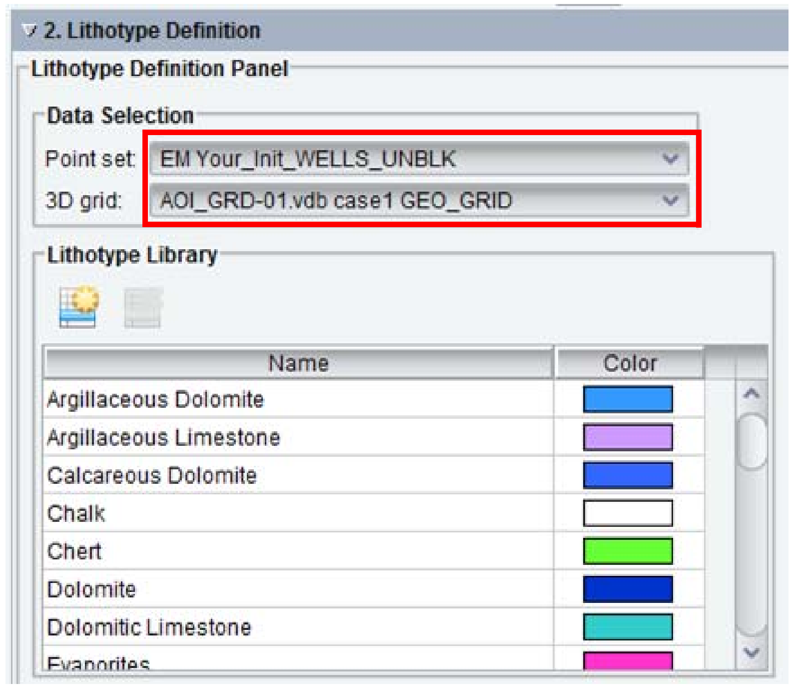
- For example, there might be two or three defined shales but they can be grouped into a single lithotype to simplify the modeling.
- However, distributary channel sand should not be grouped with distributary mouth bar sand as the geometries are different.

For this data set there are four lithology codes which you will model and no regrouping is necessary.

Facies Code	Depositional Facies	Lithotype Code	Lithotype
1	Eolian Thief Zone	1	Eolian Thief Zone
2	Eolian Siltstone	2	Eolian
3	Lagoon	3	Carbonate Mud
4	Carbonate Platform (Dolomite)	4	Platform

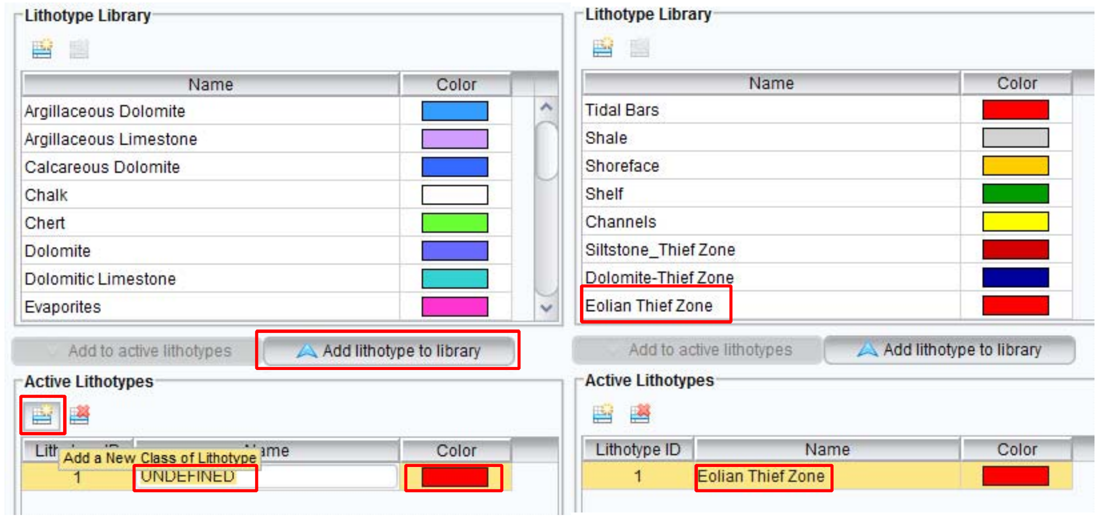
1. Select the **UNBLOCKED_WELLS** in the Lithotype Definition subtask pane.
2. Select the grid you created in the previous step (a grid not loaded to session will not be available).

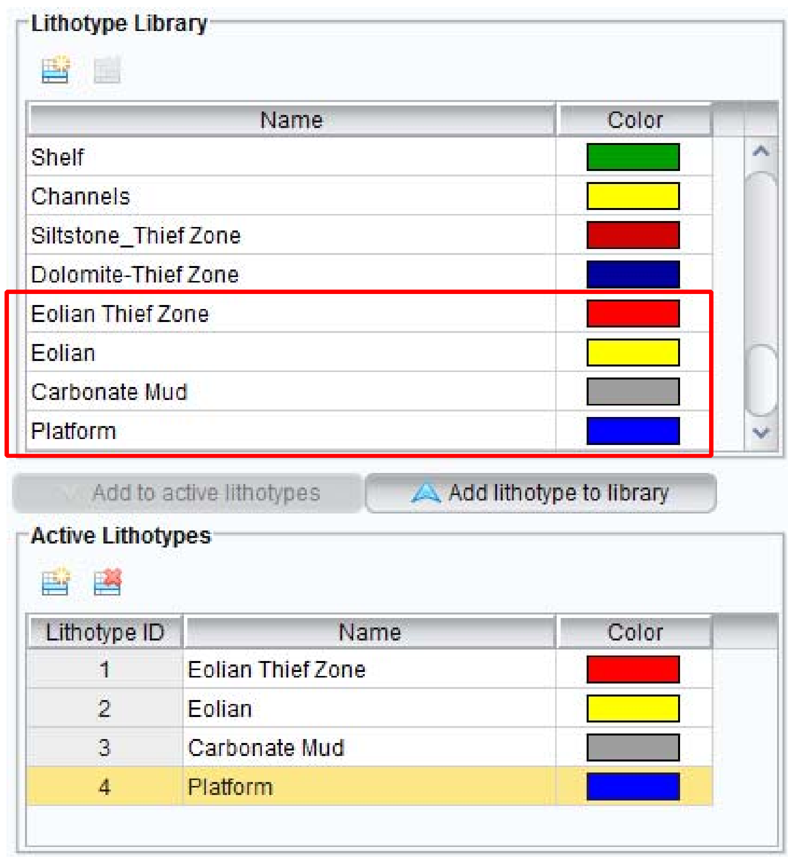
Because the four lithotypes are not in the existing library, the next steps illustrate how to create and assign them.



3. Click the **Add a New Class of Lithotype**  icon.
4. Click in the **UNDEFINED** cell and type the lithotype name.
5. Change the color.
6. Click **Add lithotype to library** which saves the information to the flat file area for use in other projects.

7. Repeat the steps for the other three lithotypes.

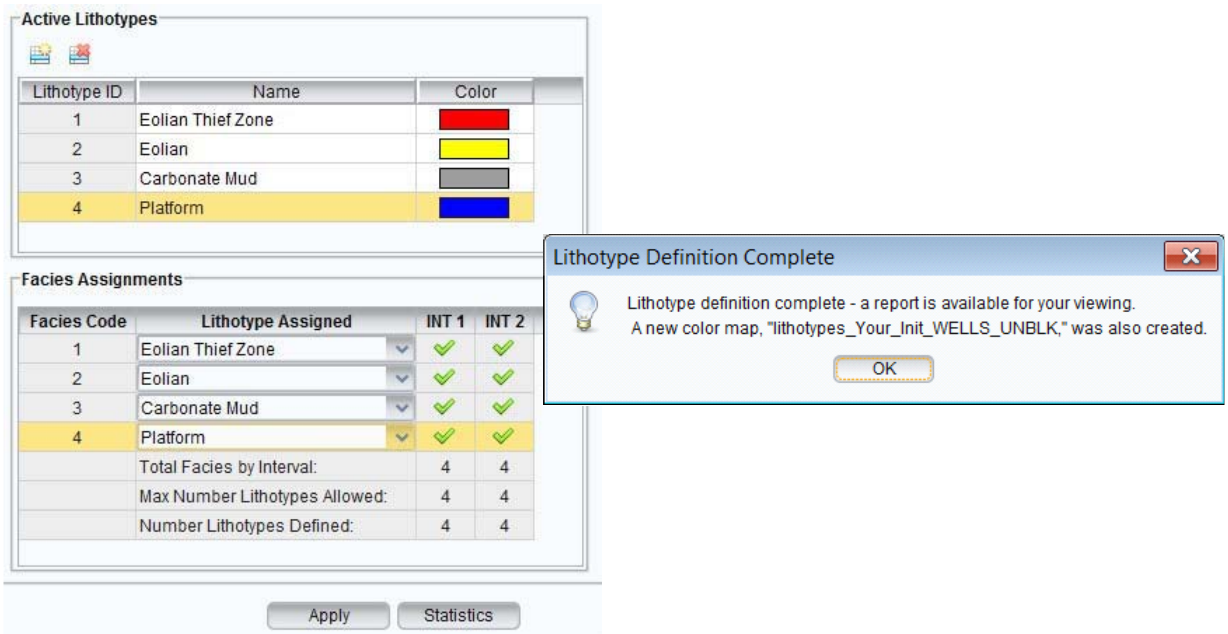




4 lithotypes created and added to the library.

Assign Lithotype to Facies Code

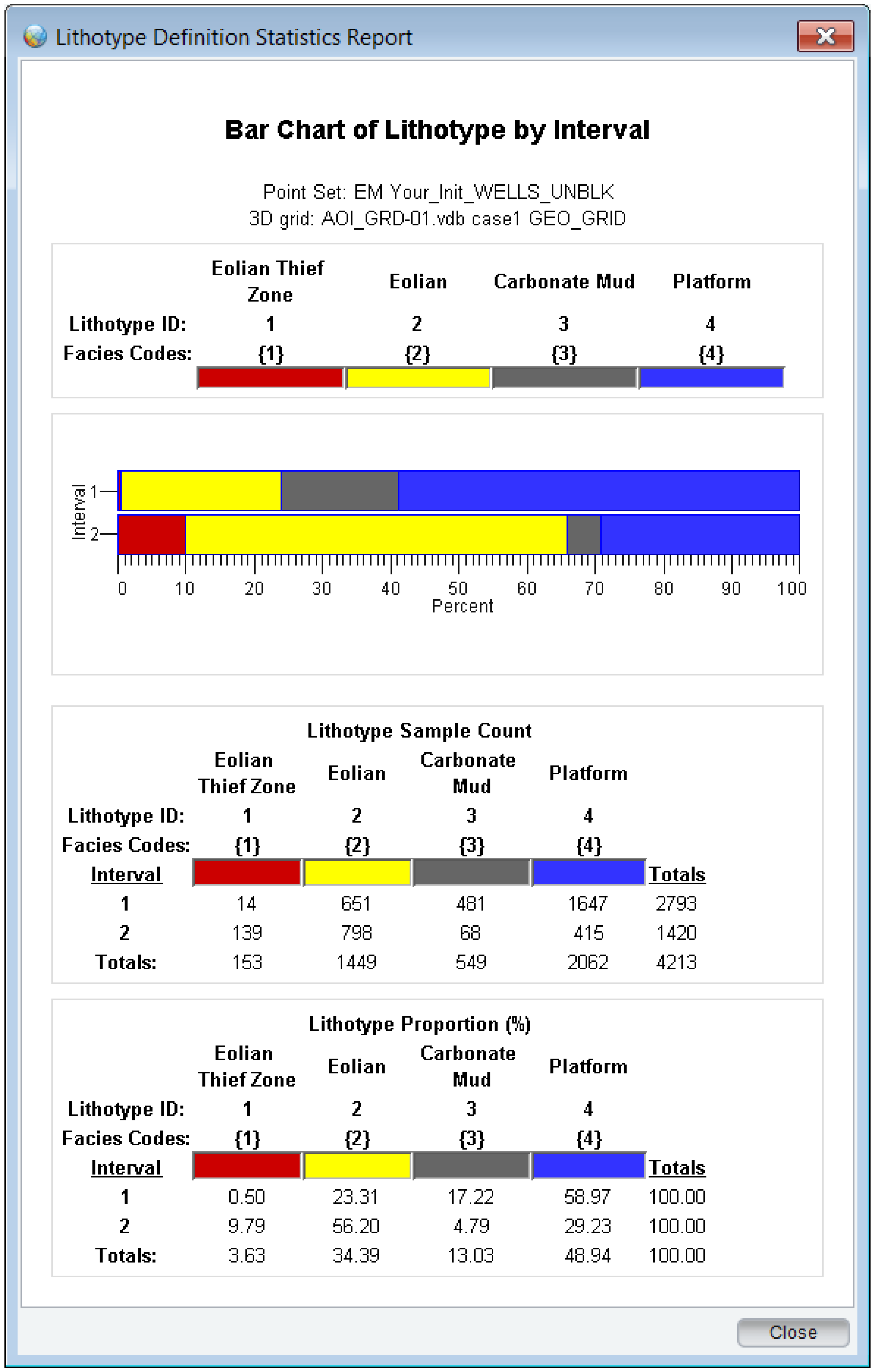
1. Assign the lithotype code to the same facies code.
2. Click **Apply** to assign the code and to create the color map.
3. Click **Statistics** to view the report.



The Statistics Report provides information about the proportions for each lithotype by interval based on the unblocked wells. A new report is created once the well data are blocked to the 3D grid.

Interval 1 is dominantly the carbonate platform lithology with very little of the Eolian Thief Zone.

Interval 2 is dominated by the Eolian Siltstone and with almost 10% of the Thief Zone.



Lithotype Sample Count

Interval	Eolian Thief Zone (1) {1}	Eolian (2) {2}	Carbonate Mud (3) {3}	Platform (4) {4}	Totals
1	14	651	481	1647	2793
2	139	798	68	415	1420
Totals:	153	1449	549	2062	4213

Lithotype Proportion (%)

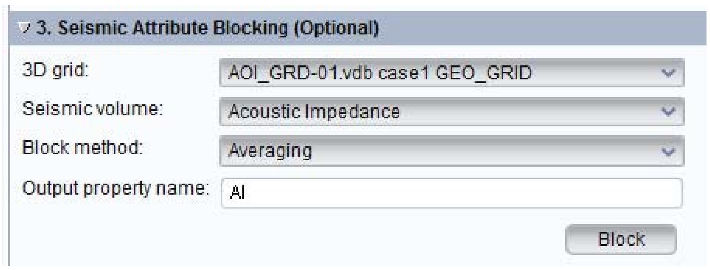
Interval	Eolian Thief Zone (1) {1}	Eolian (2) {2}	Carbonate Mud (3) {3}	Platform (4) {4}	Totals
1	0.50	23.31	17.22	58.97	100.00
2	9.79	56.20	4.79	29.23	100.00
Totals:	3.63	34.39	13.03	48.94	100.00

Seismic Blocking (Optional)

From 5-93 to 5-95, the exercise is optional.

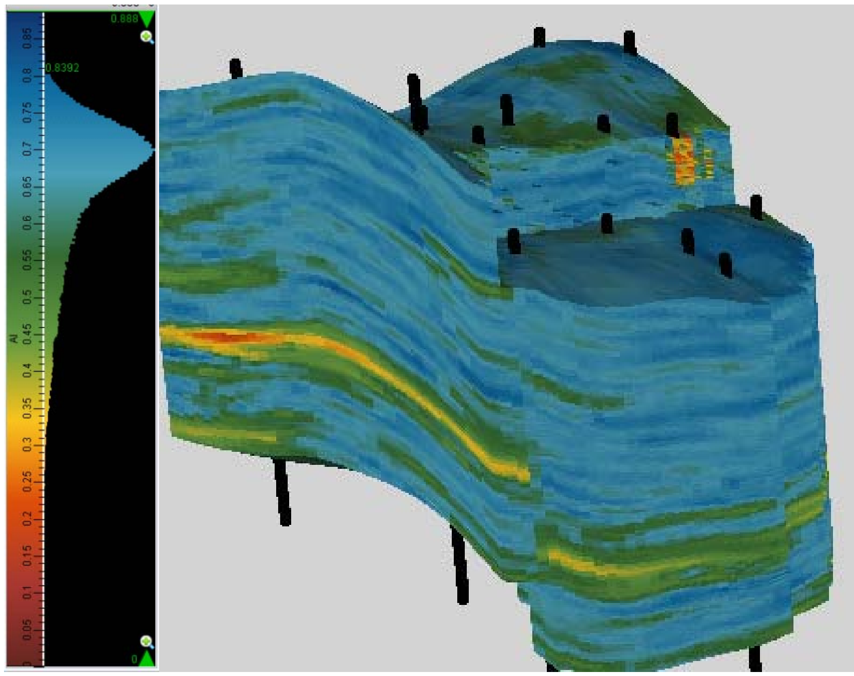
If seismic attributes are available in OpenWorks, the 3D volume can be blocked at the scale of the stratigraphic grid using two methods.

- Averaging (using the closest four data points)
- Nearest Point (uses the closest data point to the cell gravity center)



1. Block the seismic as shown in the image above.

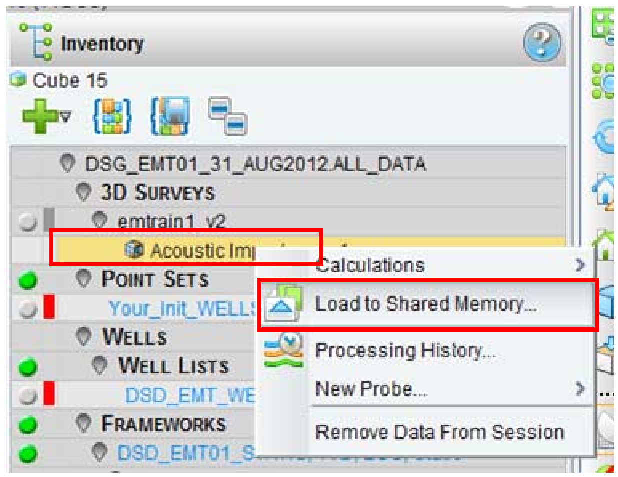
2. Display the blocked seismic data in Box Probe 1 using the Earthtones 1 color map. You may need to flip the color scale.



The low impedance values (warm colors) are confined dominantly to the upper part of interval 2.

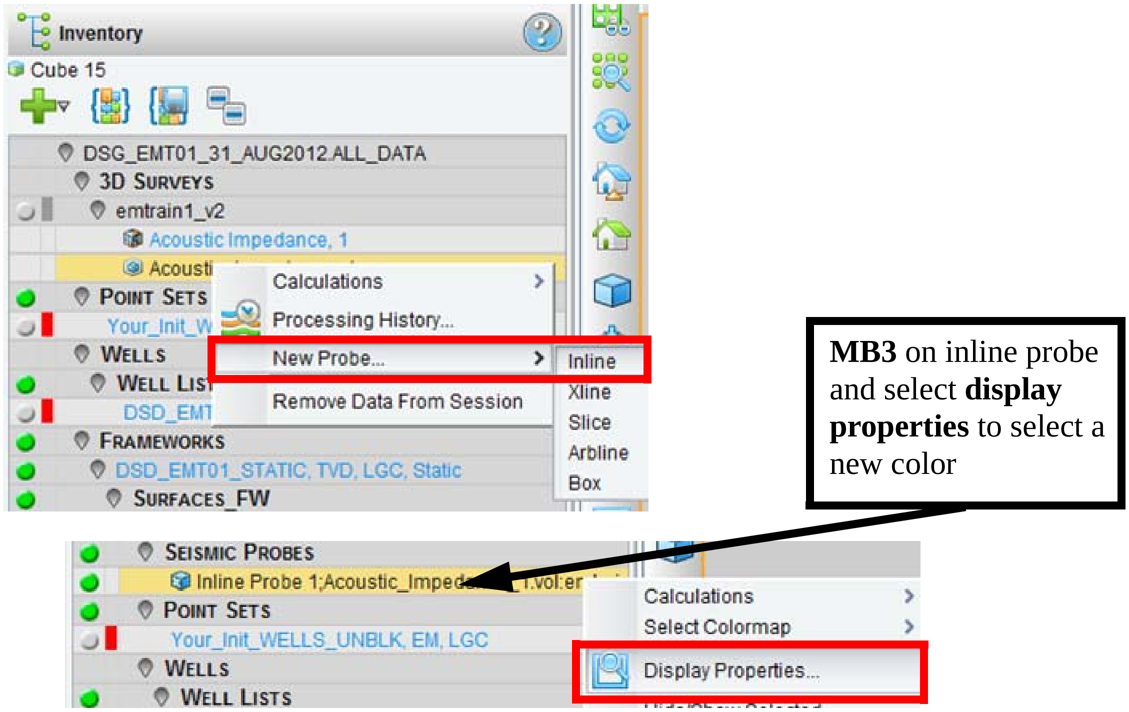
Load Seismic Volume to Memory

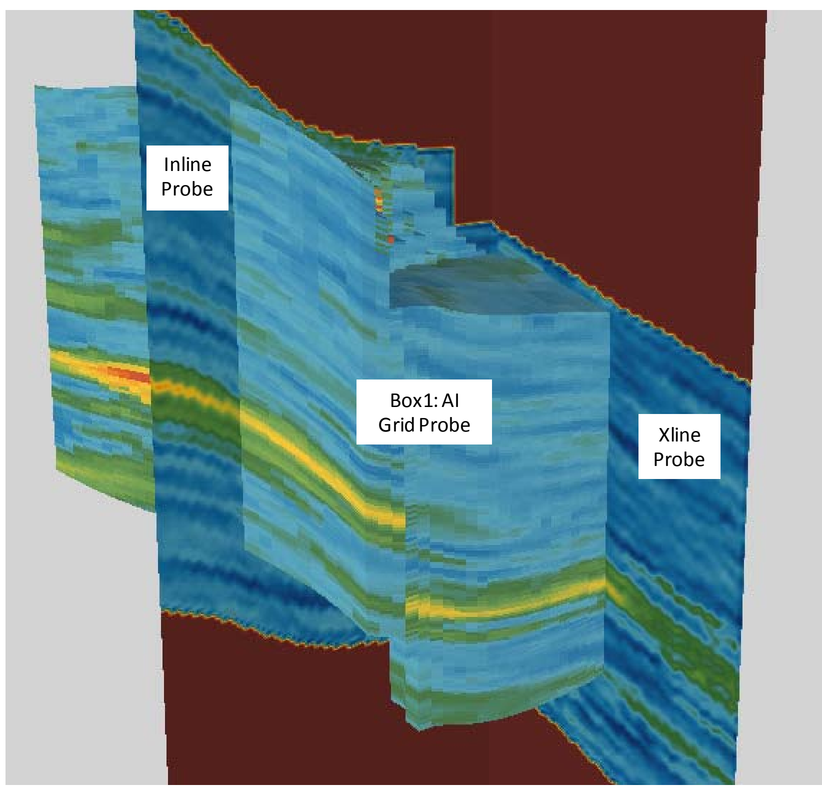
1. Load the seismic volume to memory by doing and MB3 on the **Acoustic Impedance** volume and selecting **Load to Shared Memory** (if not already in memory).



2. From the inventory and with cube view active, create an Inline Probe from the In Memory 3D seismic volume and compare it to the blocked seismic AI on the grid.

3. Change the probe color to match that on the grid.



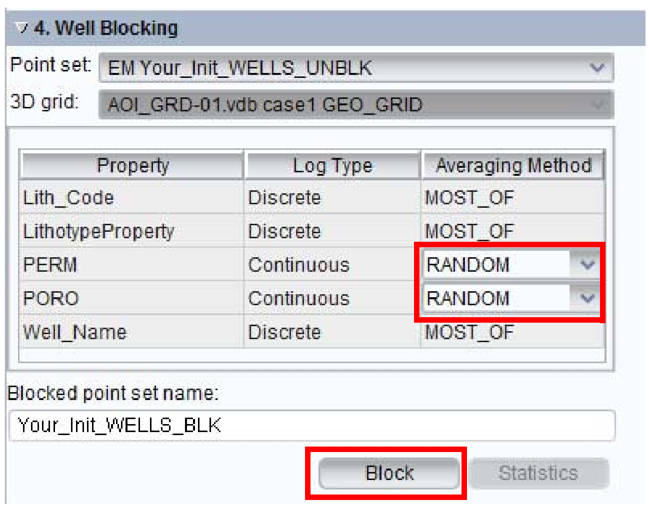


Well Blocking

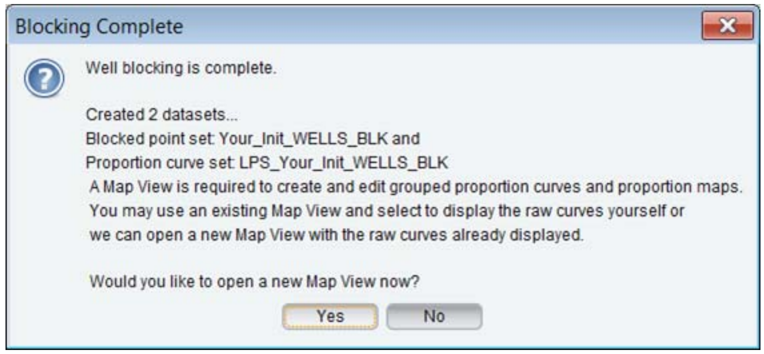
You can block and unblocked point set properties onto the 3D geocellular grid using the following methods:

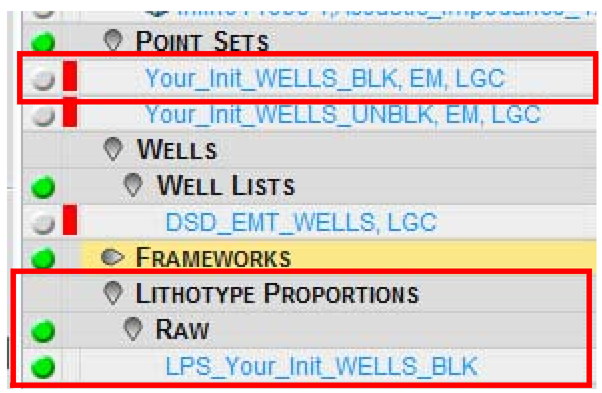
- Most of (for discrete properties this is the only method used)
- Arithmetic average
- Minimum value
- Maximum value
- Mid-point (the value that lies closest to the cell gravity center)
- Random (selects at random a value that falls within a cell for the selected lithotype) reproduced unblocked statistical properties best
- Geometric average (often used for permeability)
- Harmonic average (often used for permeability)

1. Select the unblocked point set.
2. Choose **Random** for PERM and PORO.
3. Click **Block**.

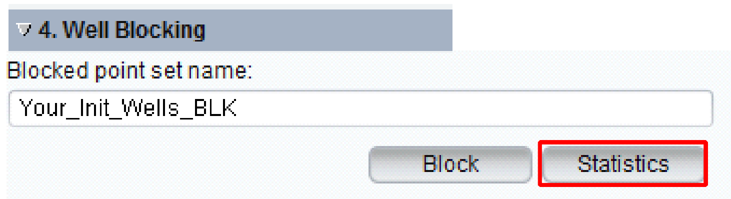


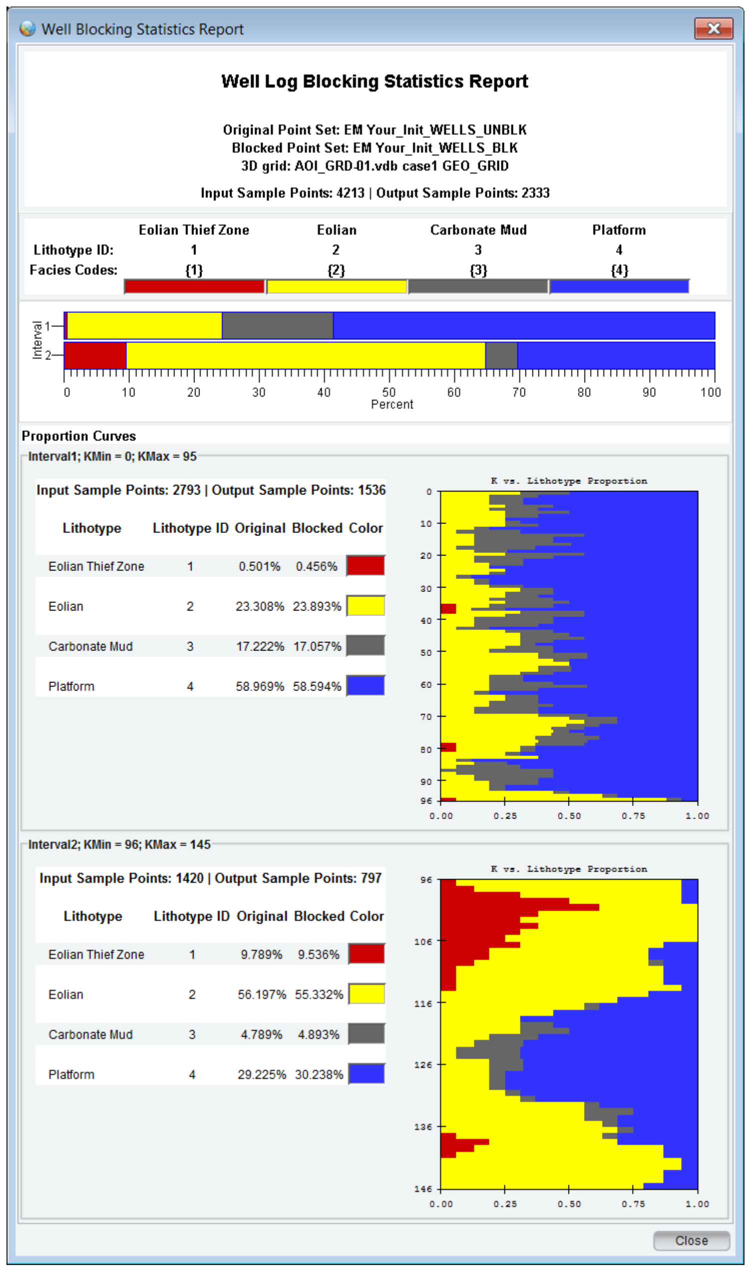
After the well blocking completes you will receive a message to open a Map View (if not already open) to display the information required to create the Lithotype Proportion Map. Two new items show up in the inventory: 1) Blocked Curves and 2) the Lithotype Proportions computed from the Blocked Lithotypes.





Display the Well Blocking Statistics





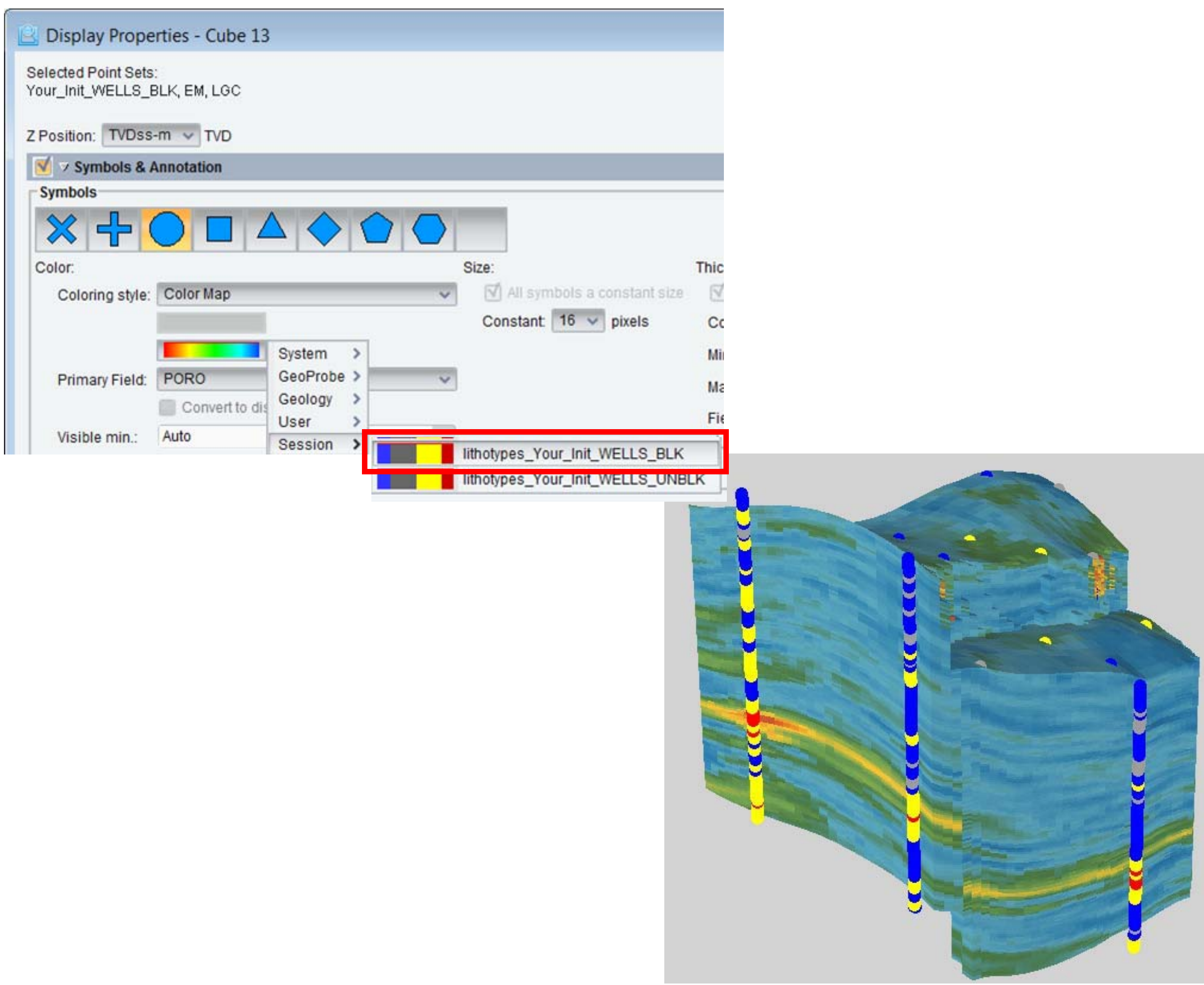
The report shows the Global proportion curves for each interval and the lithography proportion statistics for the blocked well data. Review these curves because they have information to assist you during the facies modeling and rules definition.

Save the session.

We observe that the Eolian Thief Zone is wholly within the Eolian Siltstone and it is never in contact with the other lithologies. But all of the other lithologies can be in contact with each other.

1. Display the blocked point sets and then overlay with the new Lithotype Property created in Blocked Well point set.

2. Change to the new Blocked Color Map.



Display of the LithotypeProperty in the blocked well data and the blocked seismic AI in the 3D grid

Data Analysis of Blocked Well Data

Once the well blocking is completed it is a good idea to check the results against the unblocked data (that resides inside the grid) to check for any blocking bias.

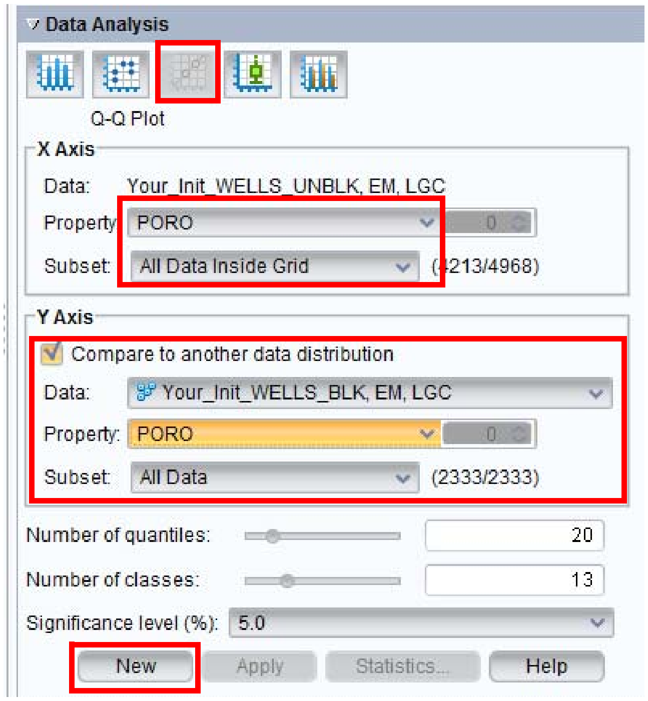
The two most useful plots are the (plots can be made for any subset created during blocking):

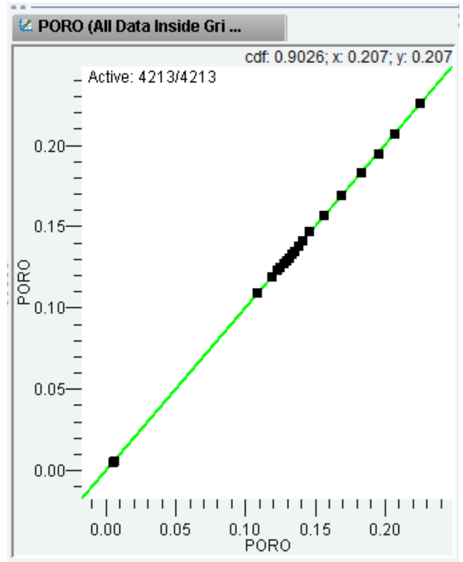
1. Q-Q Plot (plot one distribution against the other)

2. Multi-histogram for the two data sets

Q-Q Plot of PORO

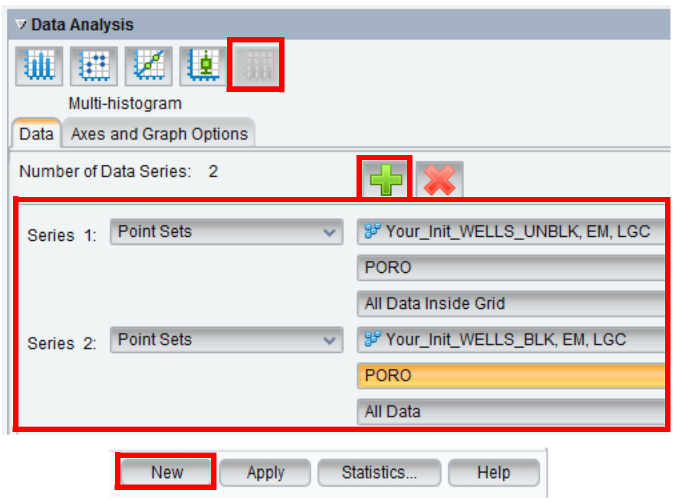
Create a Q-Q Plot of PORO as shown below.

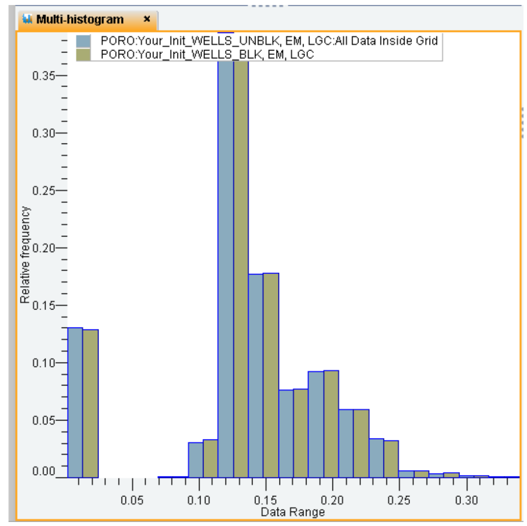




Multi-histogram Plot of PORO

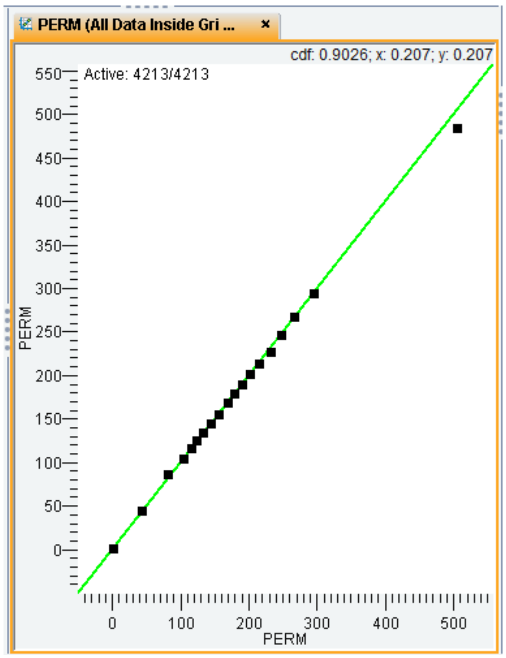
Create a multi-histogram of PORO as shown below.

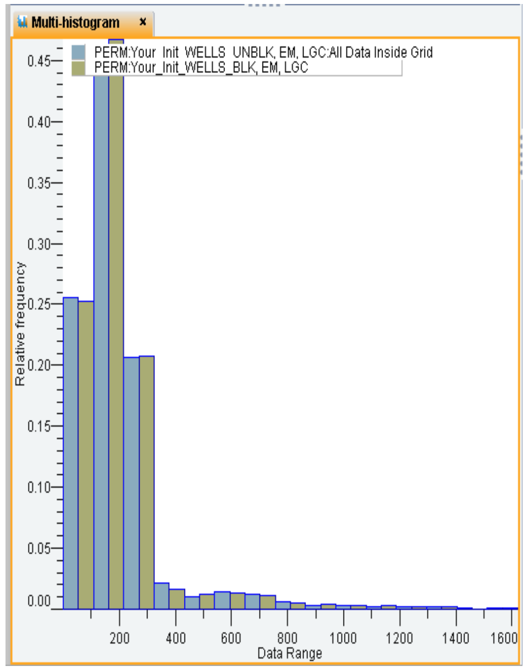




Based on the Q-Q Plot and the multi-histogram, no sampling bias was introduced during the blocking of PORO.

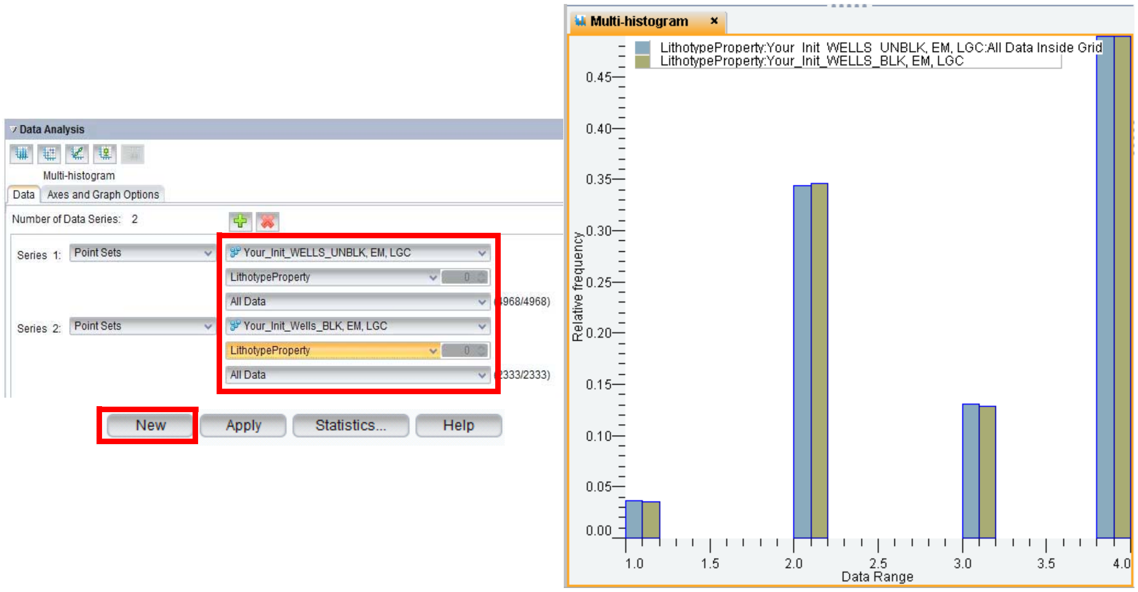
Q-Q Plot and Multi-histogram Plot of PERM





Based on the Q-Q Plot and the multi-histogram, no sampling bias was introduced during the blocking of PERM.

Multi-histogram of Lithotype Property



Based on the multi-histogram, no sampling bias was introduced during the blocking of Lithotype Property.

It is not possible to create a Q-Q Plot of a discrete property.

Facies Trend Modeling Workflow: Option 1

There are two options to create Lithotype proportion cubes:

1. Calibration with a seismic attribute for use only in Sequential Indicator Simulation (facies modeling)

2. Proportions (default) can be used with all facies modeling methods (Sequential Indicator Simulation, Truncated Gaussian and Plurigaussian)

There are two mapping options for the creation of Lithotype proportion cubes:

1. Inverse Distance (default)

2. Linear Model Kriging (LMK) which uses a linear variogram model

Seismic Attribute-to-Facies Calibration Workflow

1. Perform a lithology to seismic attribute calibration to create a lithology proportion volume for use in Sequential Indicator Simulation facies modeling.

2. Display seismic lithology proportions in Cube View.

3. Create indicator variograms to perform an Sequential Indicator Simulation facies simulation.

4. Create 10 Sequential Indicator Simulation facies models using the lithology calibrated seismic AI for:

    - Cross-section grid subset—using seismic calibration and display in Section View

    - Sector Polygon grid subset—using seismic calibration

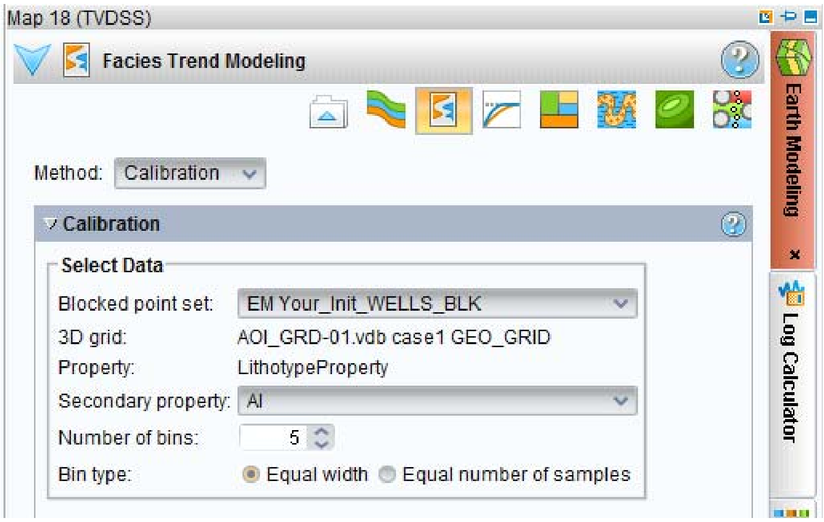
    - Entire grid—without seismic calibration

5. Create a Multi-histogram to compare blocked lithology proportions with simulated lithology proportions.

Calibration Method

1. In the Facies Trend Modeling task pane select the **Calibration** Method. Once the Blocked point set is selected, the grid is selected automatically. In this example Acoustic Impedance was blocked into the grid and stored as AI.

   Map View must be active.

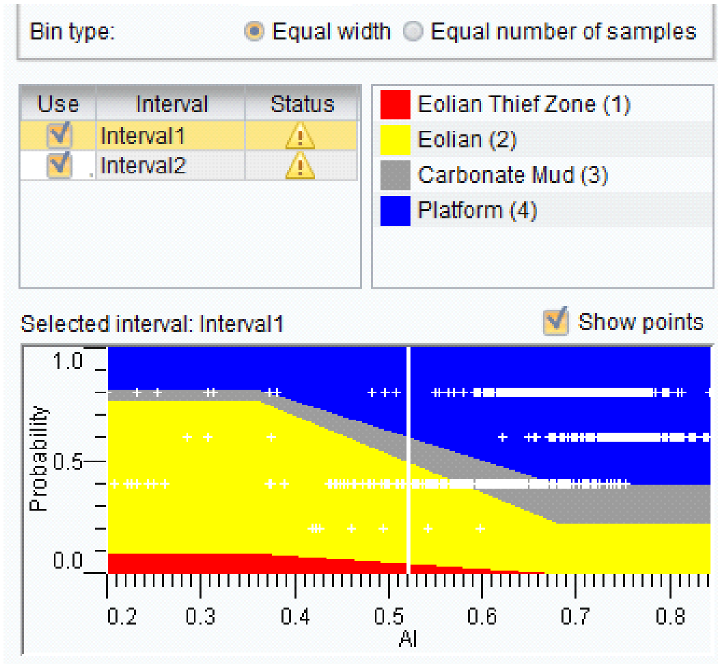
   There are two Bin type options used to create the calibration cross-plot.

   Calibrating seismic properties, such as acoustic impedance (AI), to lithologies is not a clear-cut process. Ideally each lithology would correspond to a different range of AI values; unfortunately this rarely occurs, so there is more often an overlap between the lithologies and the range in AI (or whatever attribute is used). The calibration process is therefore subjective.

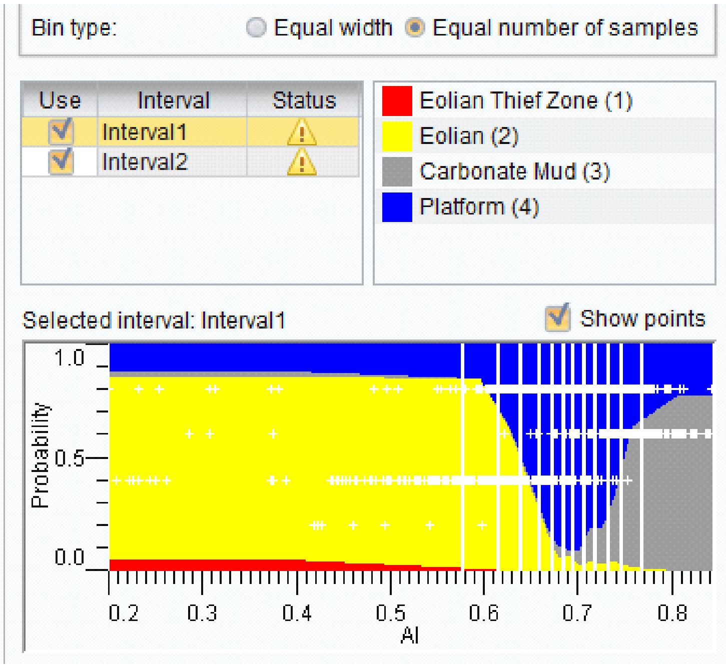
   The calibration involves the use of a cross-plot, where the seismic attribute values lie on the X-axis with the Y-axis as probability. The next slide details the information in the cross-plot.

   In this example there are four lithologies. It is recommended that the calibration method is used with a maximum of 3-4 lithologies; however, in reality, it is often difficult for a seismic attribute to differentiate more than 2-3 lithologies at best. There is a choice to use equal width or equal number of samples per bin.

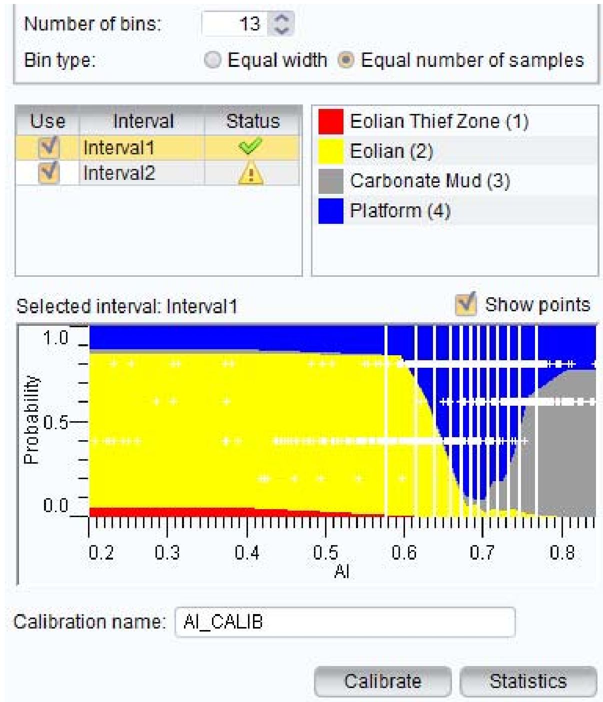
When Equal width is selected the X-axis is divided into bins of an equal range of *AI*. The white points show the number of samples for each lithology falling into each bin. The number of bins is chosen to maximize the difference in the proportions falling in the bin. Here we see that the yellow (siltstone) is associated predominately with bin 1 when 2 bins of equal width are chosen.



If Equal number of samples is chosen the bins are of unequal size. This option is good when the data are sparse and equal width could result in a severe bias in the calibration. In this example either calibration method would produce similar results using 5 bins.



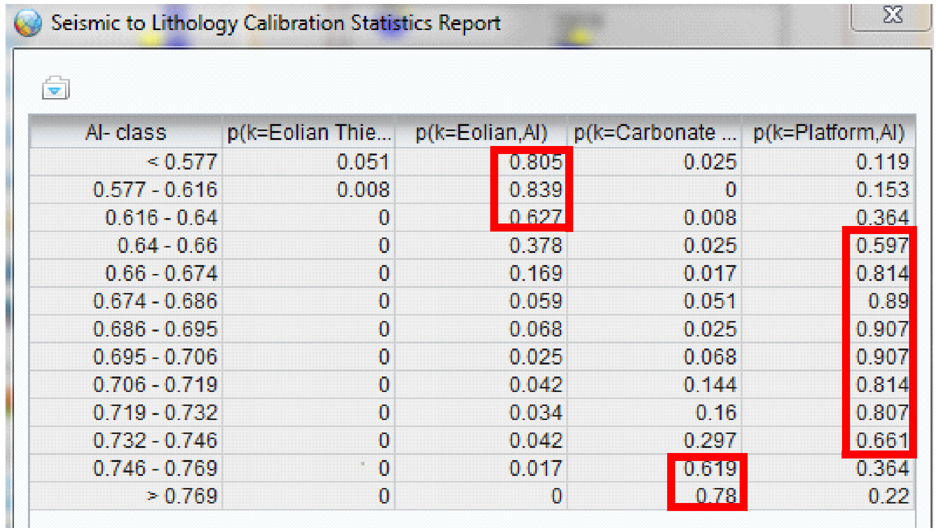
2. For the final calibration of Interval 1, using 13 bins with equal number of samples seems to give the best calibration.



3. Create a Calibration name which is used for all intervals you want to calibrate. It is not necessary to calibrate an interval if no relationship is found.

4. Click **Calibrate** and the Status is indicated completed with a green check.

The statistical is a table of the results (your values could differ slightly).

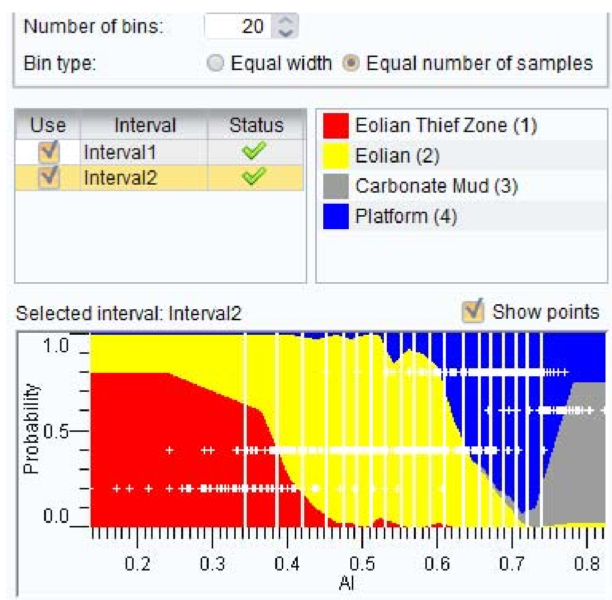


AI- class	p(k=Eolian Thie...	p(k=Eolian,AI)	p(k=Carbonate ...	p(k=Platform,AI)
< 0.577	0.051	0.805	0.025	0.119
0.577 - 0.616	0.008	0.839	0	0.153
0.616 - 0.64	0	0.627	0.008	0.364
0.64 - 0.66	0	0.378	0.025	0.597
0.66 - 0.674	0	0.169	0.017	0.814
0.674 - 0.686	0	0.059	0.051	0.89
0.686 - 0.695	0	0.068	0.025	0.907
0.695 - 0.706	0	0.025	0.068	0.907
0.706 - 0.719	0	0.042	0.144	0.814
0.719 - 0.732	0	0.034	0.16	0.807
0.732 - 0.746	0	0.042	0.297	0.661
0.746 - 0.769	0	0.017	0.619	0.364
> 0.769	0	0	0.78	0.22

This is a reasonably good calibration even though the Thief Zone is not well represented, because there is very little of this lithology to calibrate in Interval 1.

5. For the final calibration of Interval 2, 20 bins with equal number of samples seem to give the best calibration.

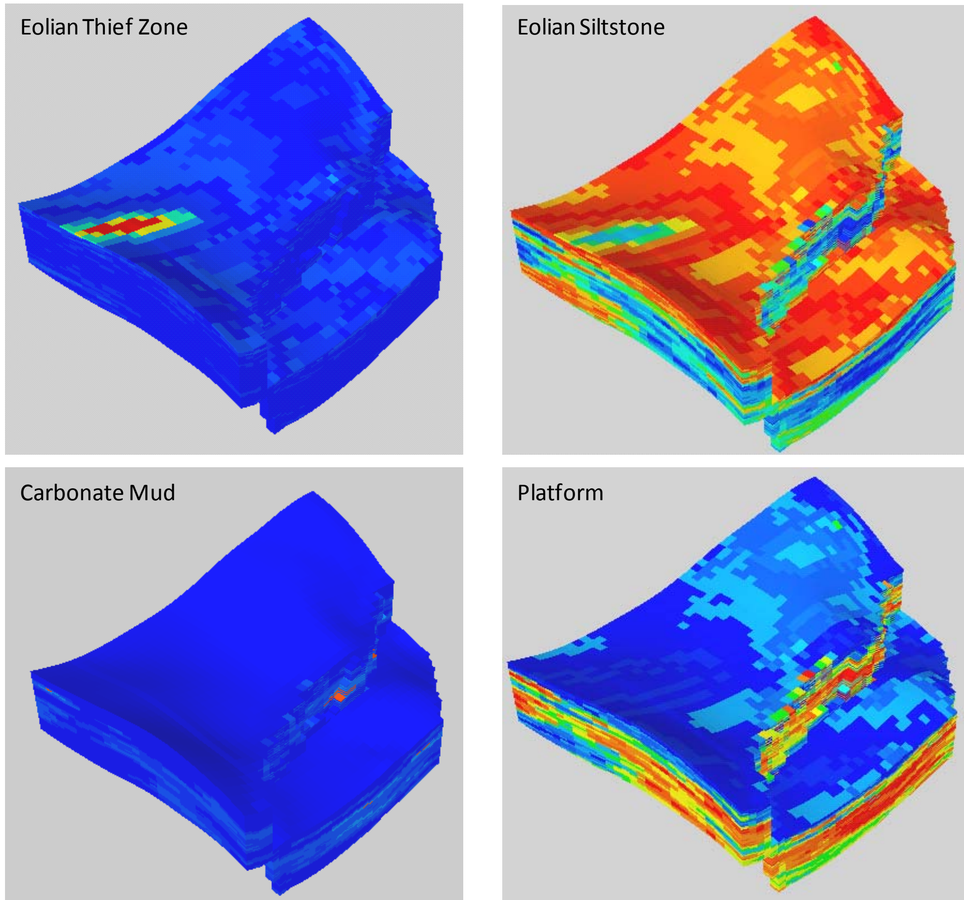
   Do not forget to click **Calibrate**, but use the same calibration name.



This is a reasonably good calibration. The interval 2 Thief Zone falls predominantly within the first two bins.

AI- class	p(k=Eolian Thie...	p(k=Eolian,AI)	p(k=Carbonate ...	p(k=Platform,AI)
< 0.344	0.8	0.2	0	0
0.344 - 0.387	0.6	0.4	0	0
0.387 - 0.421	0.25	0.75	0	0
0.421 - 0.454	0.103	0.872	0	0.026
0.454 - 0.476	0.025	0.975	0	0
0.476 - 0.493	0.025	0.95	0	0.025
0.493 - 0.514	0	1	0	0
0.514 - 0.531	0.05	0.95	0	0
0.531 - 0.553	0.025	0.825	0	0.15
0.553 - 0.57	0	0.923	0	0.077
0.57 - 0.592	0	0.9	0	0.1
0.592 - 0.611	0.025	0.8	0	0.175
0.611 - 0.636	0	0.55	0	0.45
0.636 - 0.657	0	0.35	0	0.65
0.657 - 0.672	0	0.275	0.025	0.7
0.672 - 0.688	0	0.175	0.025	0.8
0.688 - 0.706	0	0.103	0.051	0.846
0.706 - 0.721	0	0.025	0.05	0.925
0.721 - 0.74	0	0	0.1	0.9
> 0.74	0	0.025	0.725	0.25

Box probe display of the AI_CALIB for the 4 lithologies near the top of Interval 2.



Sequential Indicator Simulation

Sequential Indicator Simulation is a facies modeling method that can use the proportion volume just created in the seismic-to-facies calibration step.

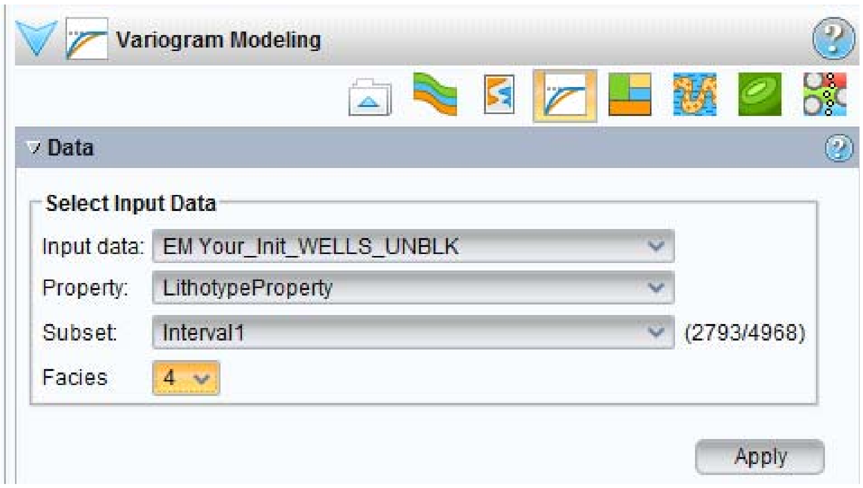
Because Sequential Indicator Simulation requires an Indicator Variogram for each modeled interval, you learn how to:

- Compute and model indicator variograms
- Create facies models using Sequential Indicator Simulation and the calibration proportion volume
- Display the results in Cube View

Compute and Model Indicator Variograms

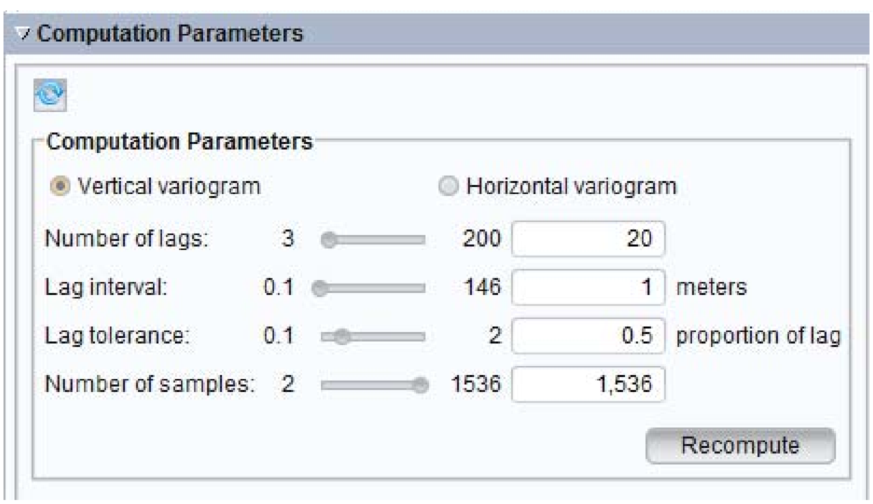
1. In the Earth Modeling task pane, click the **Variogram Modeling** icon.

2. Fill-in the Select Input Parameters section as shown.

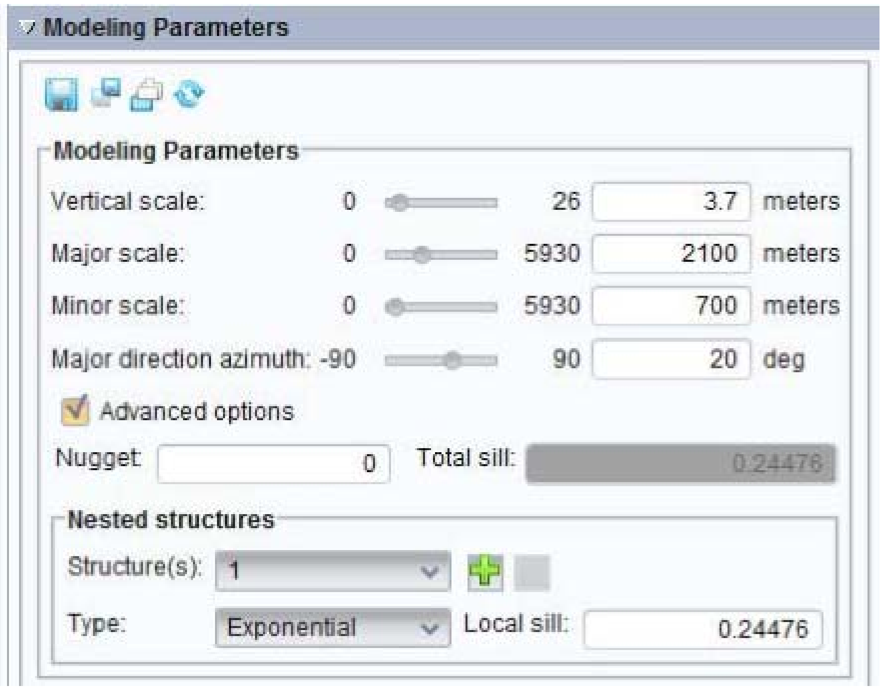


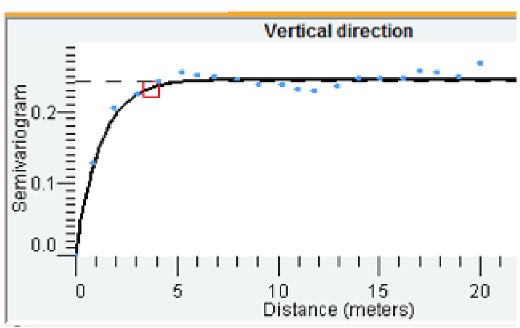
Facies 4 (Dolomite Platform) is selected because it is the most abundant lithology in Interval 1. Sequential Indicator Simulation requires only 1 indicator variogram to be computed and modeled. The others are computed automatically at run-time.

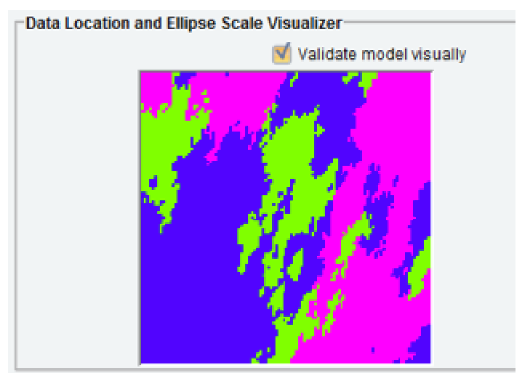
3. Change the Computational Parameters as shown. There is no need to change the horizontal parameters as there is not enough well data. Uncheck the omni-directional variogram for the horizontal variogram first.



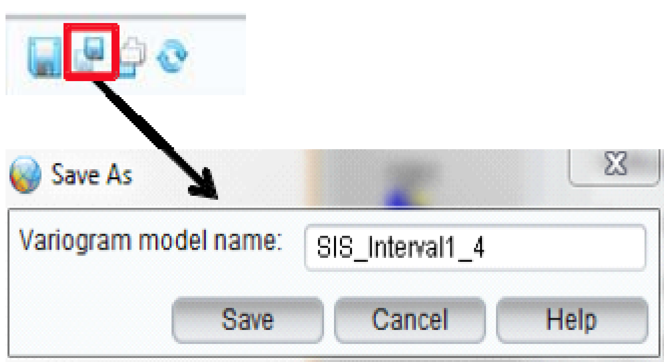
4. Change the Modeling Parameters as shown.



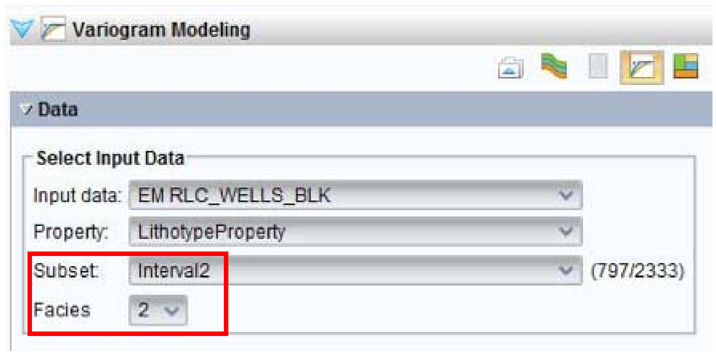


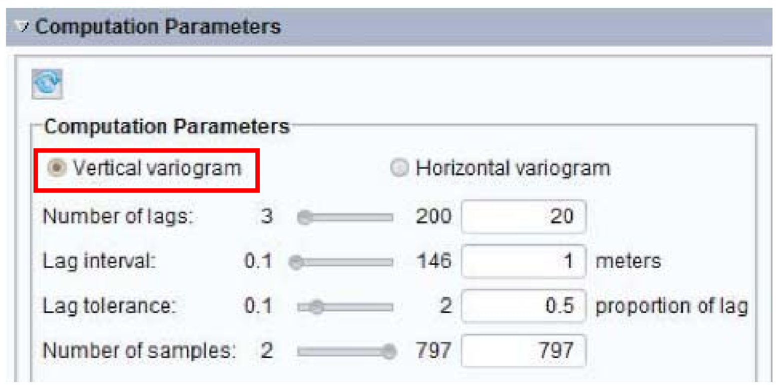


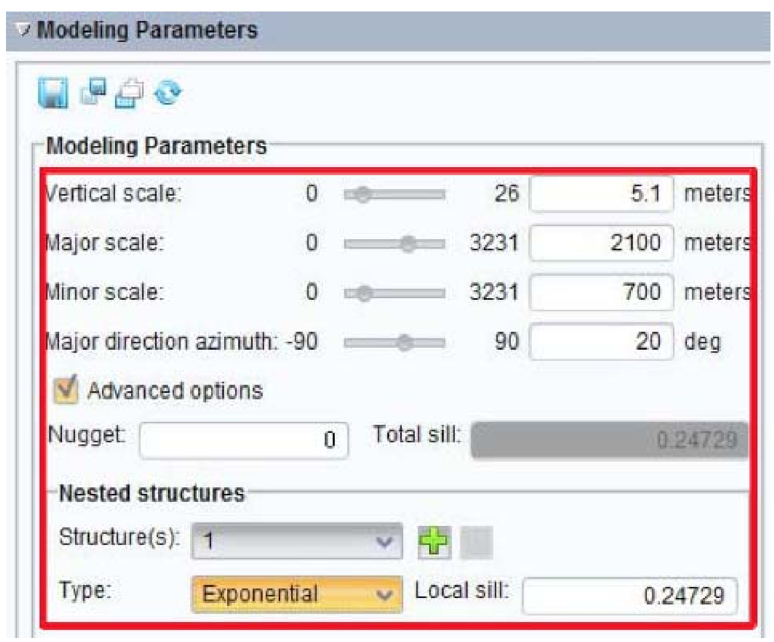
5. Save the model.

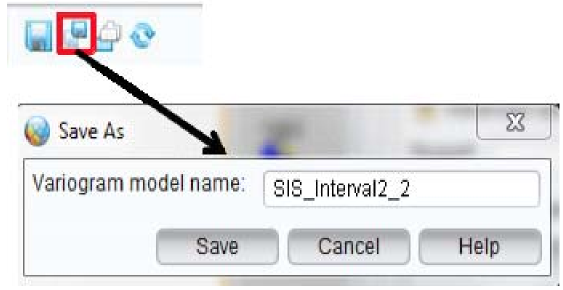


Compute & Model Indicator Variograms for Interval 2 Using Eolian Siltstone Lithotype (2)



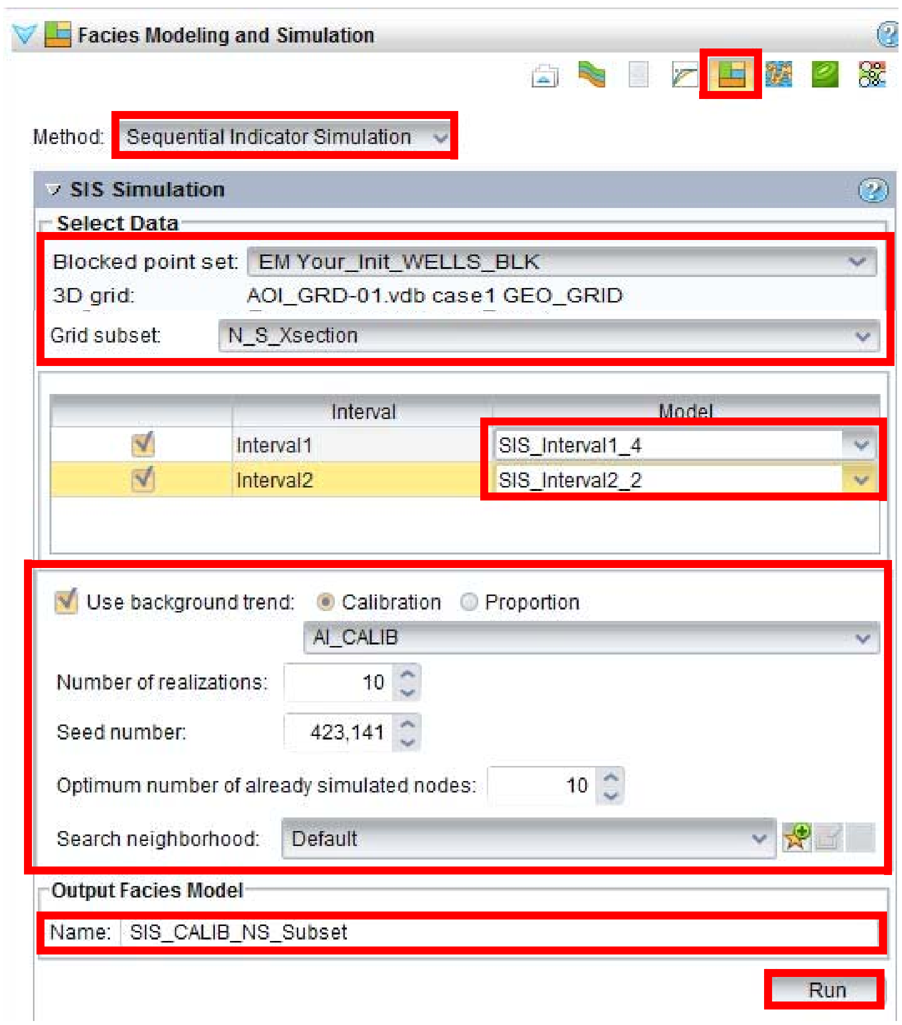




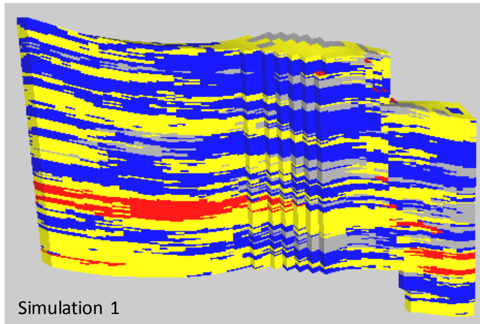

Save As
Variogram model name:
SIS_Interval2_2
Save
Cancel
Help

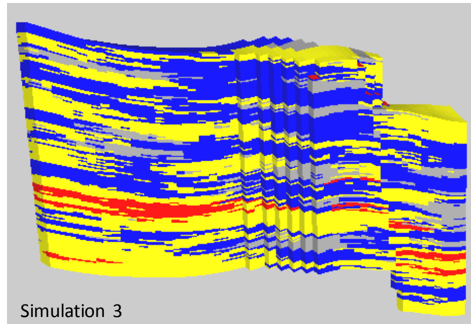
Sequential Indicator Simulation Facies Modeling with N-S Subset

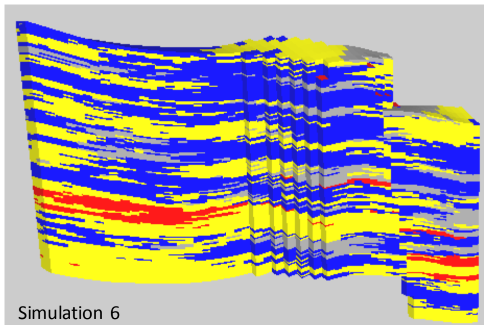
1. In the Earth Modeling task pane click the **Facies Modeling** icon.
2. Select the Method: **Sequential Indicator Simulation**.
3. Fill-in the task pane as shown.
4. Click **Run** to compute 10 realizations.
5. Save the session.
6. Display realizations in the Cube View.

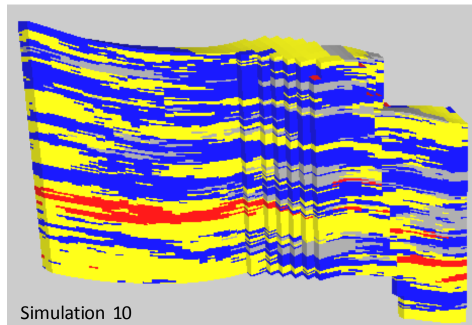


Displays of the Sequential Indicator Simulation Facies Modeling with N-S Subset in Cube View. Your simulations will only look identical if you have exactly the same grid cells as in this example.





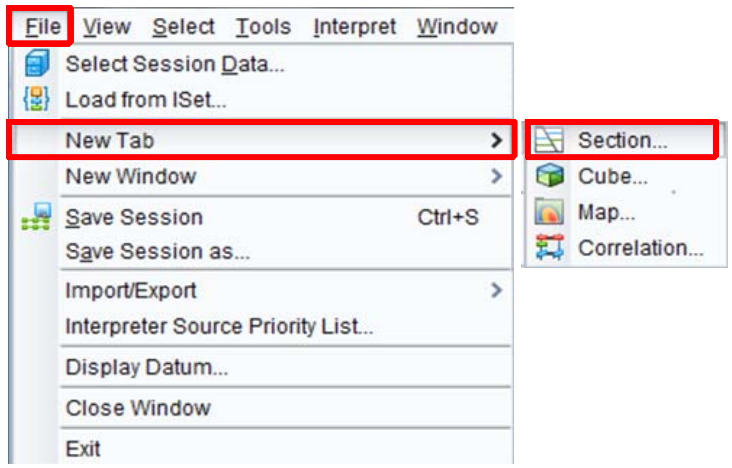




Notice that the Thief Zone (red facies) is sometimes in contact with the carbonate mud and dolomite; an observation not seen when you created the proportion first during well blocking. One of the limitations of Sequential Indicator Simulation is its ability to honor facies boundary relationships.

Display a Section View of the Sequential Indicator Simulation Facies Model N-S Subset

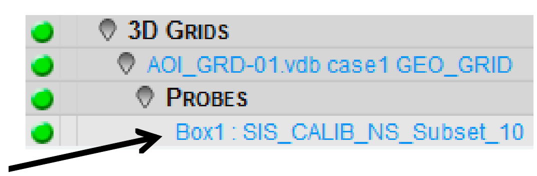
1. Open a Section View from the Main Window.



2. From the Cube View, use MB3 to Arrange Tabs.

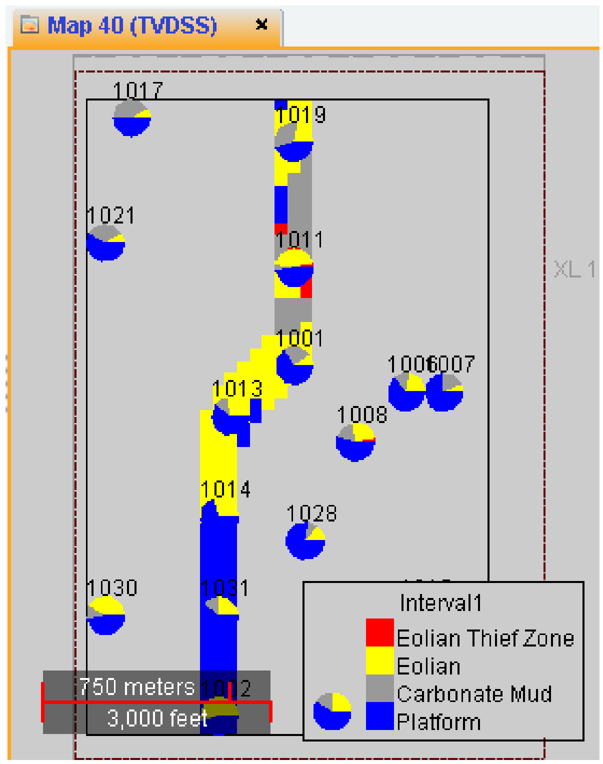
3. Make the Map View the active view.

4. Display the Box1 Probe of the Sequential Indicator Simulation N-S subset in Map View.

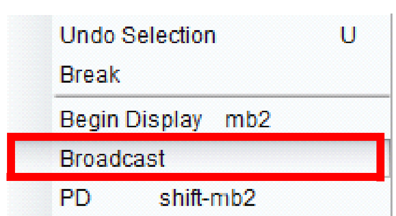


5. With Map View Active, click the icon  on the toolbar to activate Listening.

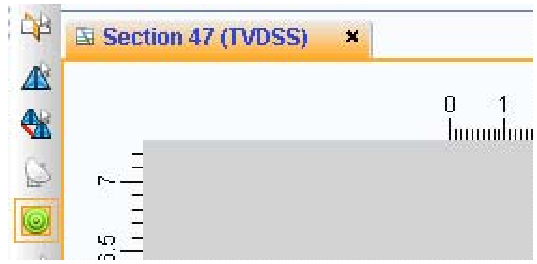
6. Click the  icon to digitize (with MB1) a point-to-point line in the Map View.



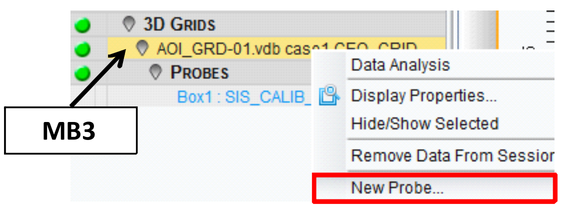
7. After digitizing the line, use MB3 to Broadcast it to the Section View.

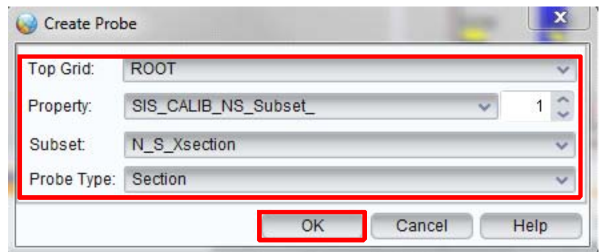


8. The Section View must be the active view.

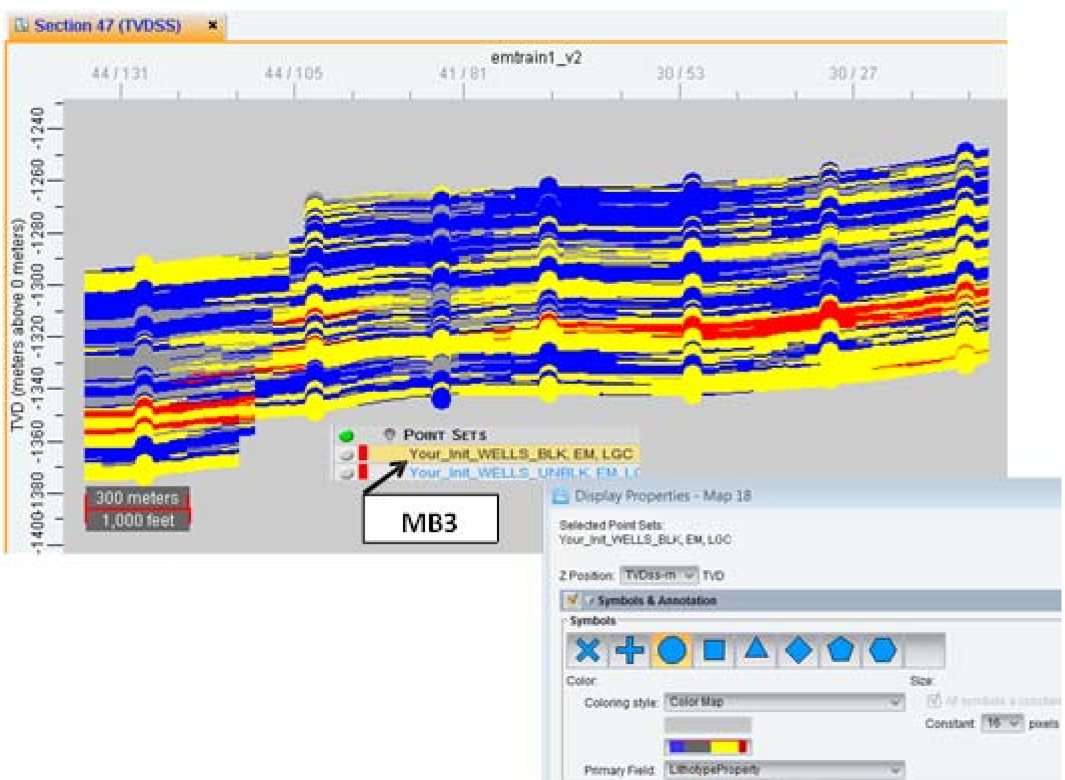


9. In the Inventory create a Section Probe to display the cross-section.

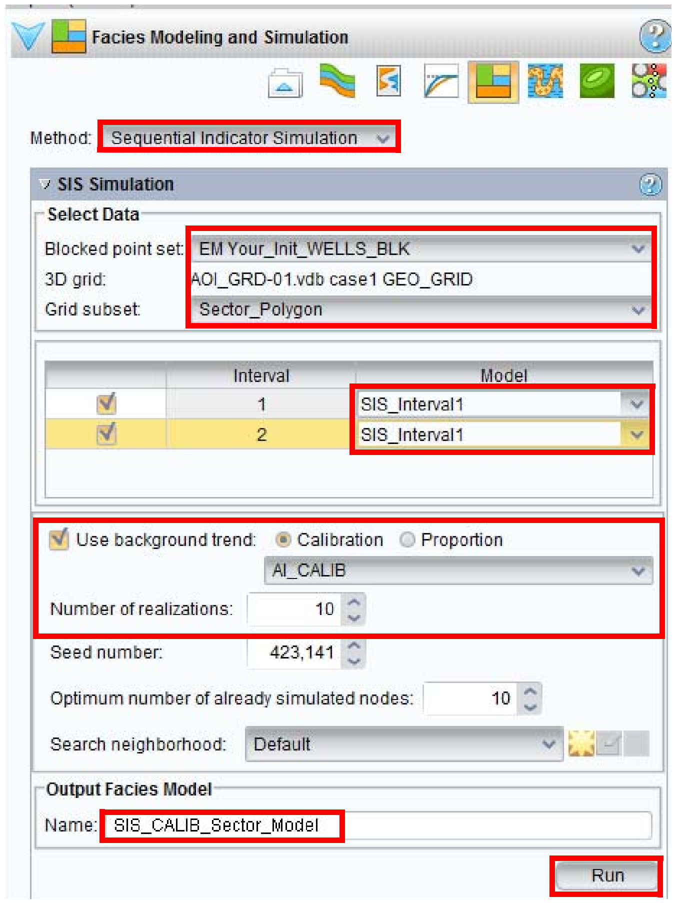


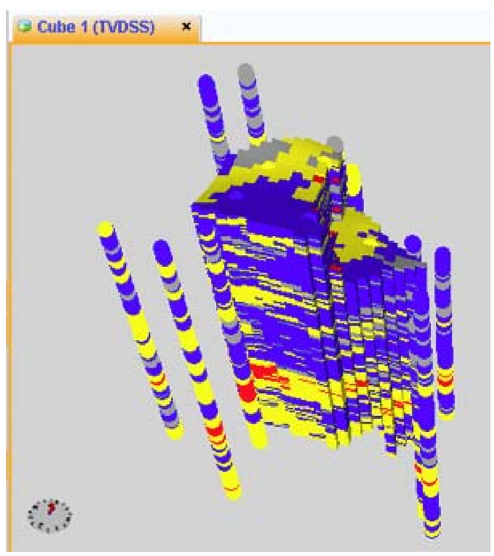


Display the **LithotypeProperty** in the Blocked Point Set. To see that the well data are honored in the simulation. Be sure to use the same color map while comparing the results.

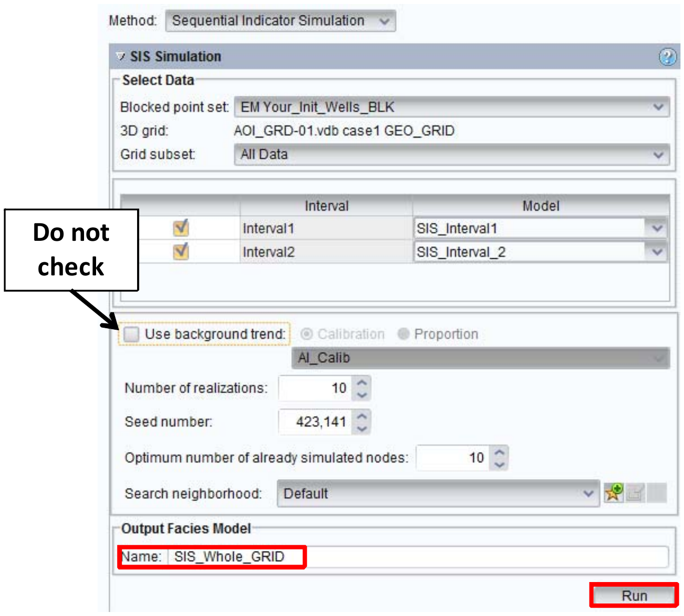


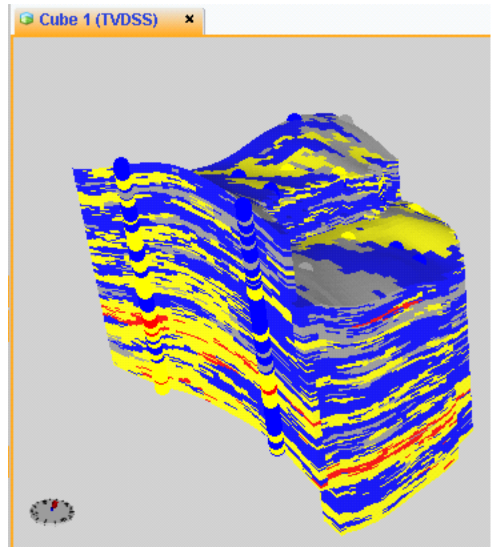
Rerun the Seismic Calibration Sequential Indicator Simulation Model Using the Sector Polygon Subset





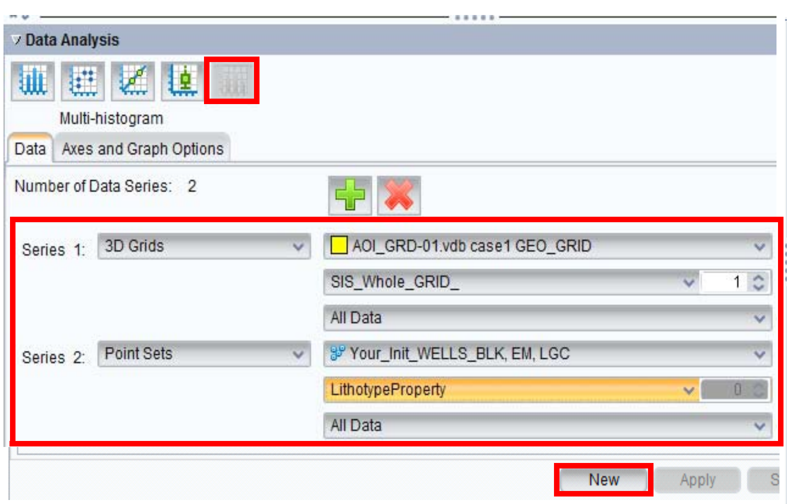
Create an Sequential Indicator Simulation model on the entire grid without the seismic calibration as a background trend. You will compare these results to Truncated Gaussian and Plurigaussian in another workshop.

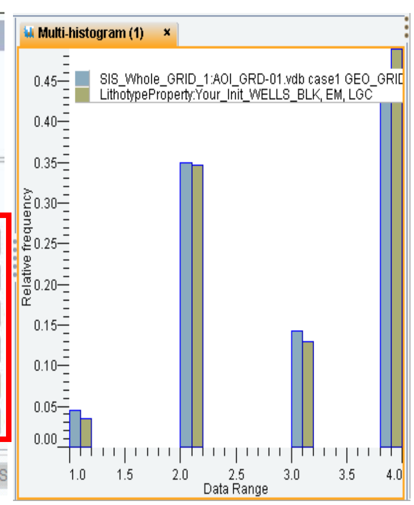




Simulation 1 of the SIS using the entire grid without seismic calibration as background trend.

Create a Multi-histogram to compare blocked lithology proportions with simulated lithology proportions.





Save the session.

Facies Trend Modeling: Option 2

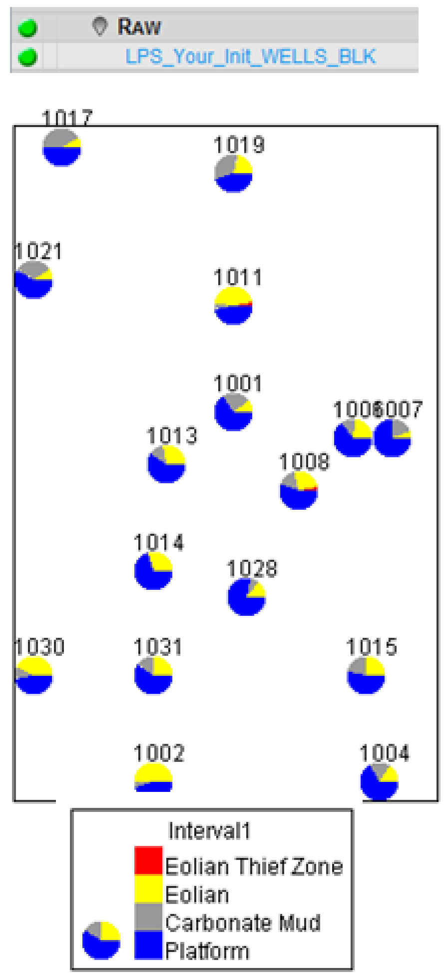
In the previous workshop you learned how to calibrate a seismic attribute to lithology codes at the wells to create a background trend for use in Sequential Indicator Simulation. In this workshop you learn how to create a background trend using the well data for use in Truncated Gaussian and Plurigaussian Facies Simulation. The lithology proportion matrix created with this option can be used with Sequential Indicator Simulation.

Workflow:

1. Create a Lithology Proportion Matrix using well data only.
2. Learn how to copy and edit lithology curves.
3. Learn how to use Linear Model Kriging to control the construction of the Lithology Proportion Matrix.
4. Perform Sequential Indicator Simulation with the Lithology Proportion Matrix.
5. Perform Truncated Gaussian.
6. Perform Plurigaussian.
7. Create a Multi-histogram to compare the results with the blocked well data.

Create a Lithology Proportion Matrix

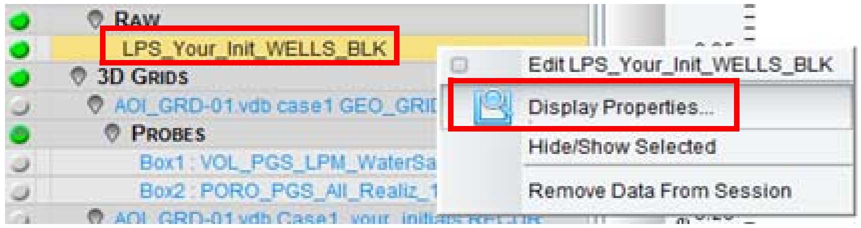
1. Make sure the a Map View is the active view and display the Raw Lithotype Proportion Curves available in the Inventory and display the Pie Charts for Interval 1.



- The pie charts show the proportions for each of the 4 lithotypes at the 16 well locations. Although they give information about the proportions, you don't know the vertical lithotype ordering which is an important piece of information required for the modeling.

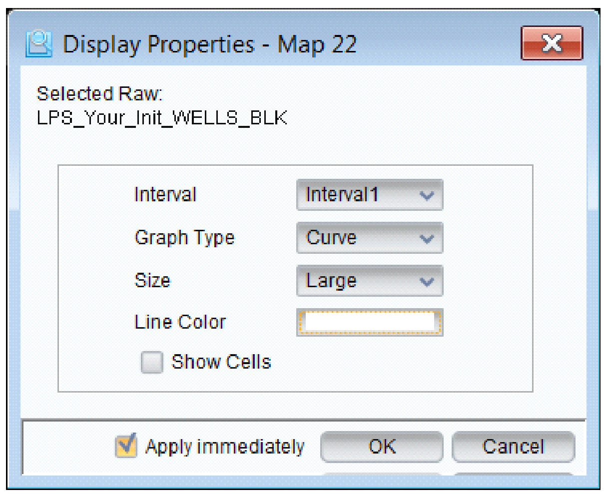
- We observe that the Platform dolomite lithotype dominates followed by the Eolian Siltstone, Carbonate Mud, and then the Thief Zone.

- To display the curves, open the Display Property Editor using the MB3 on the Raw LPS name.

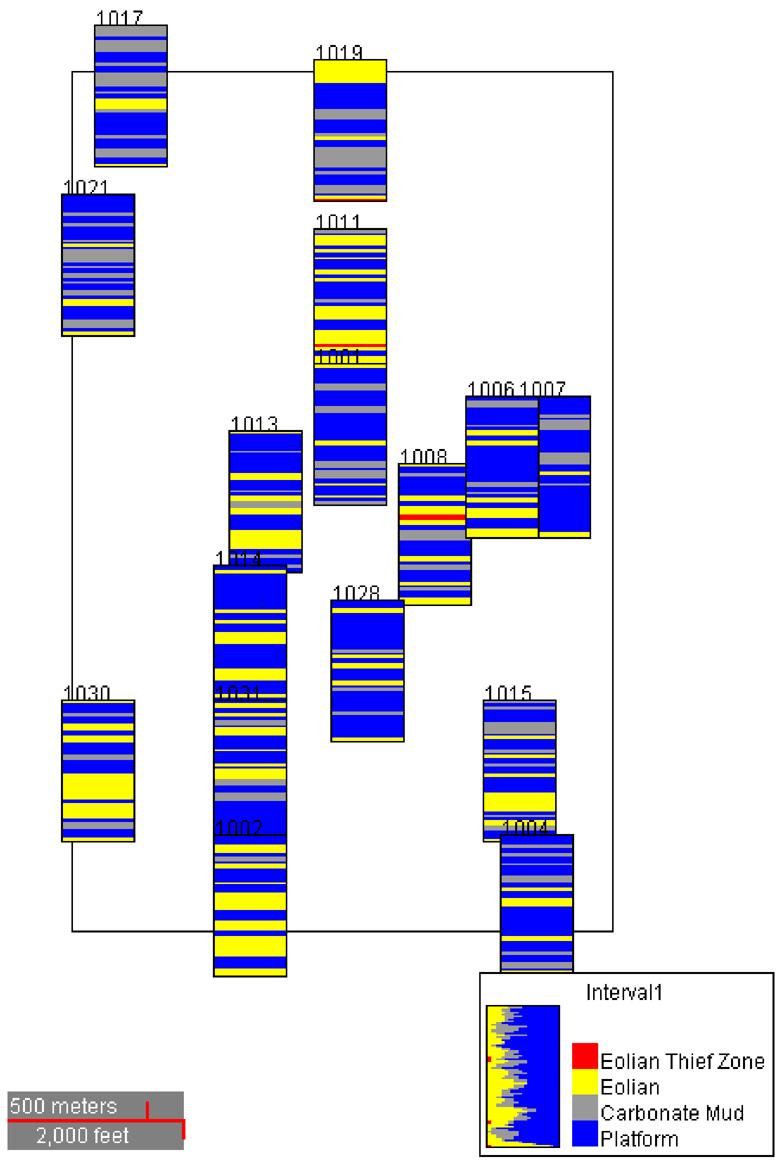


Change the Display Property Editor

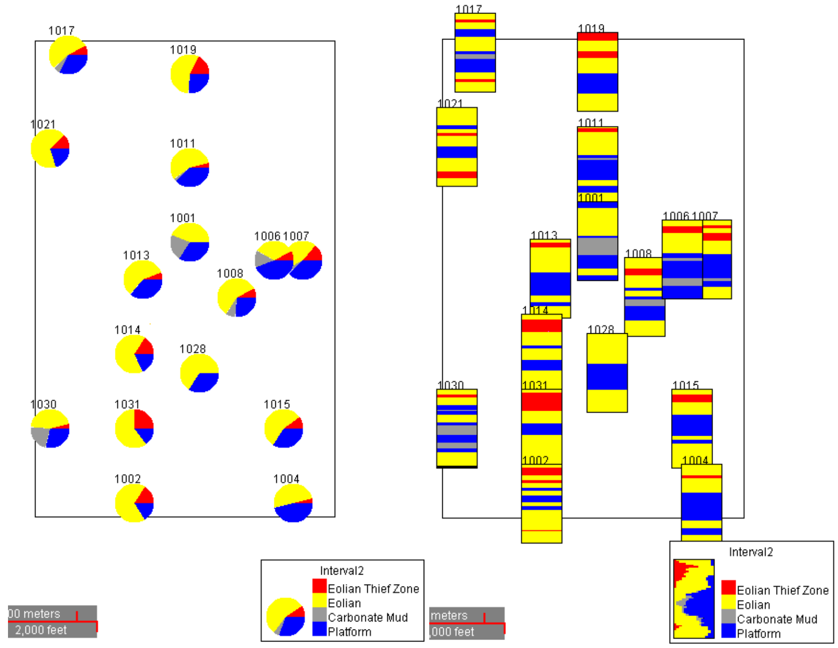
- Graph Type: **Curve**
- Size: **Large**
- Line Color: **Black**
- Show Cells: **unchecked**



Observe that the Eolian Thief Zone occurs in wells 1008 and 1011 only and is encased within the Eolian Siltstone and not in contact with the other lithotypes. The summary Vertical Proportion Curve (VPC) is shown in the Interval Legend Box. We see that there *several* sub-cycles of eolian and carbonate deposition within this interval.



Display the Interval 2 Pie Charts and VPCs.

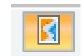


Observe that the Eolian Thief Zone is present in all wells, except 1001 and 1028, and is predominately in the upper part of interval 2. It is always encased within the Eolian Siltstone.

The Global VPC illustrates two episodes of eolian deposition interrupted by a rise in sea-level during which deposition of the carbonate occurred. Locally both the Platform Dolomite and Carbonate Mud are in contact with the Eolian Siltstone.

Create Grouped VPCs

1. Make the Map View the active view.

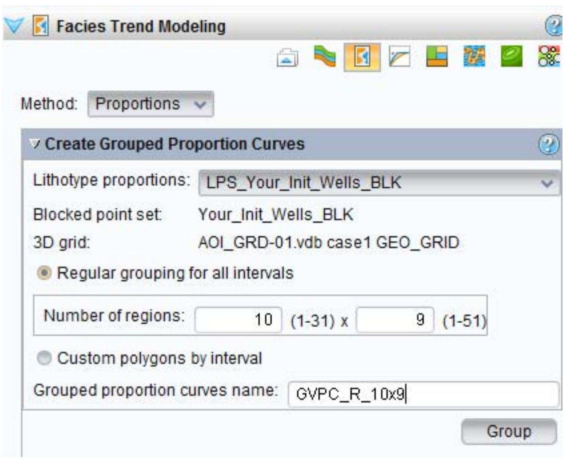
2. Click the **Earth Modeling** tab and then click the  icon to open the Facies Trend Modeling task pane.

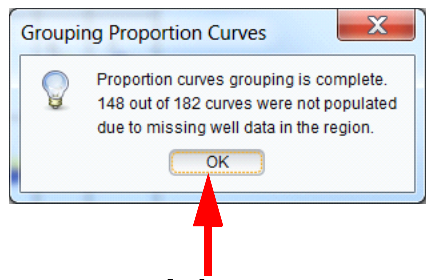
3. Method should be set to **Proportions**.

4. Use **10 by 9 Regular grouping** for all intervals.

5. Name the Grouped proportion curves.

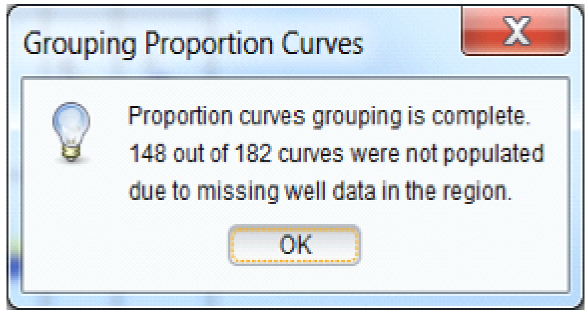
6. Click **Group**.

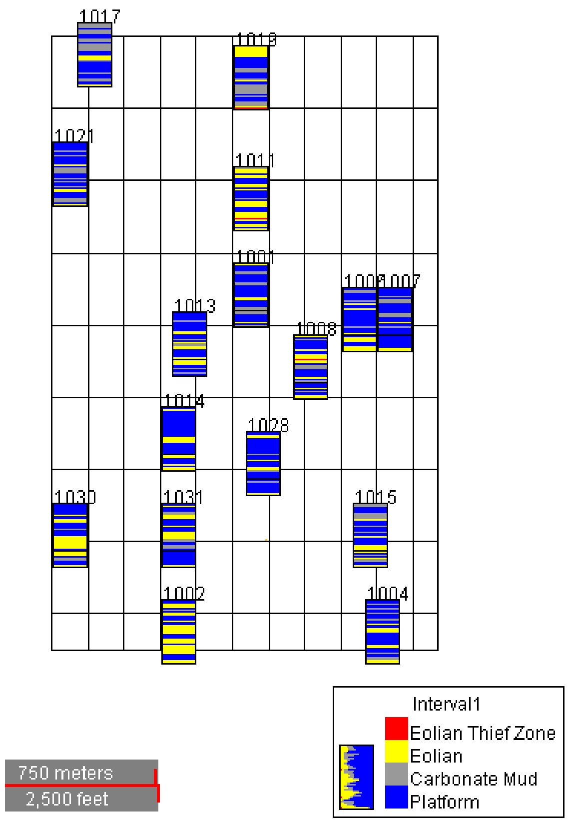




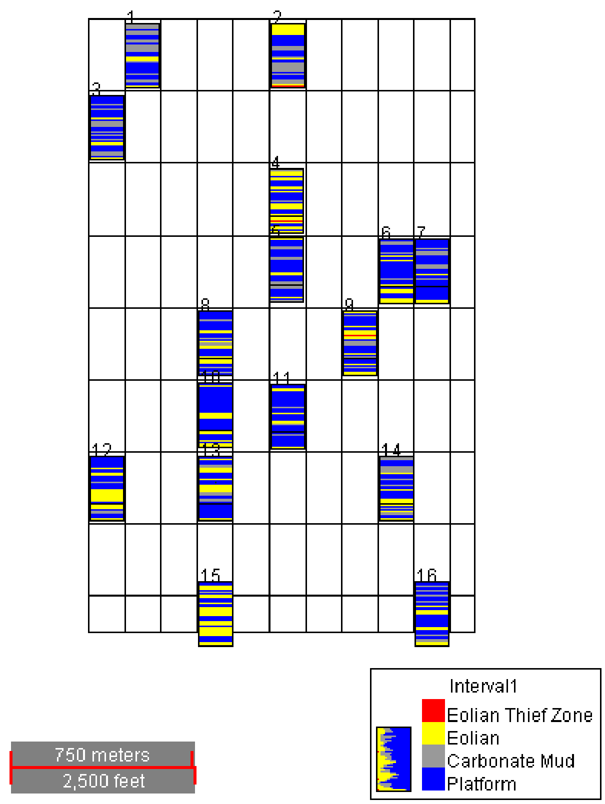
Click **OK**

7. Click **OK** to dismiss the message. What this means is that wells are not present in most of the coarse grid blocks.



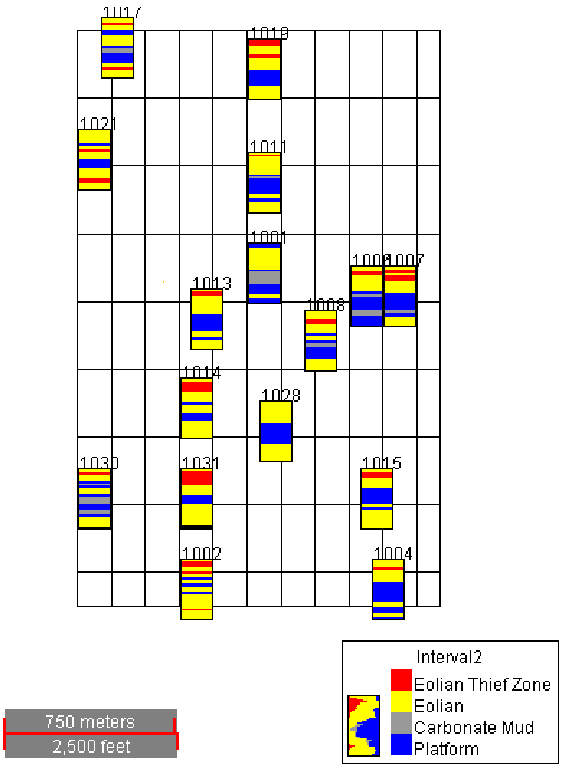


Interval 1 before grouping

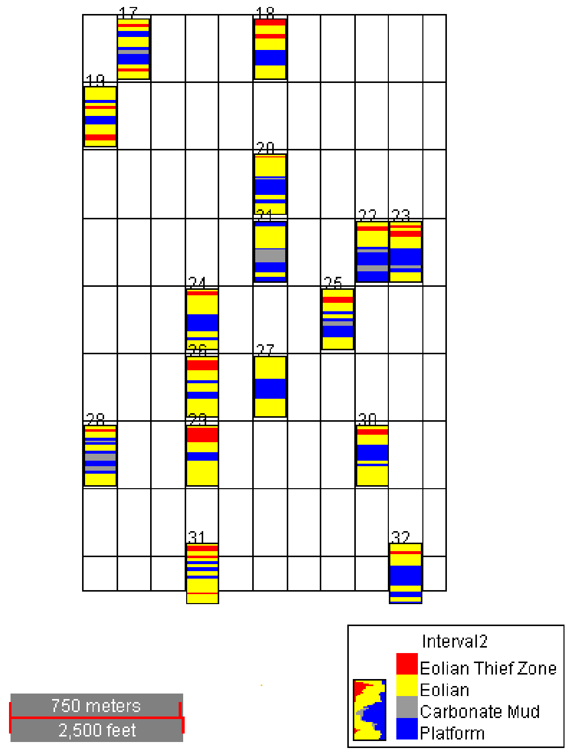


Interval 1 after grouping

This illustrates the 10 by 9 regular grid overlay before grouping (left) and after grouping (right). The intent is to have each well within one of the grids following the grouping. The wells are renumbered 1-16. The reason that the wells have a new number is because if the Regular grid was designed so that two or more wells fall within a grid they are averaged into a single VPC for that grid.



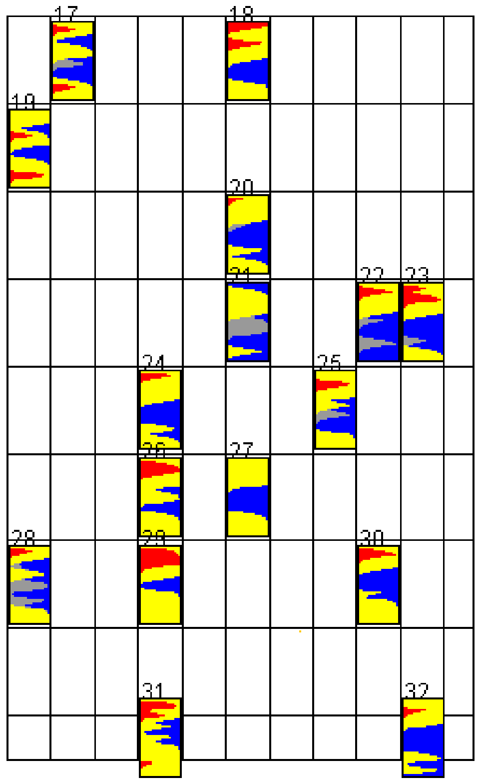
Interval 2 before grouping



Interval 2after grouping

Smooth the Grouped VPCs

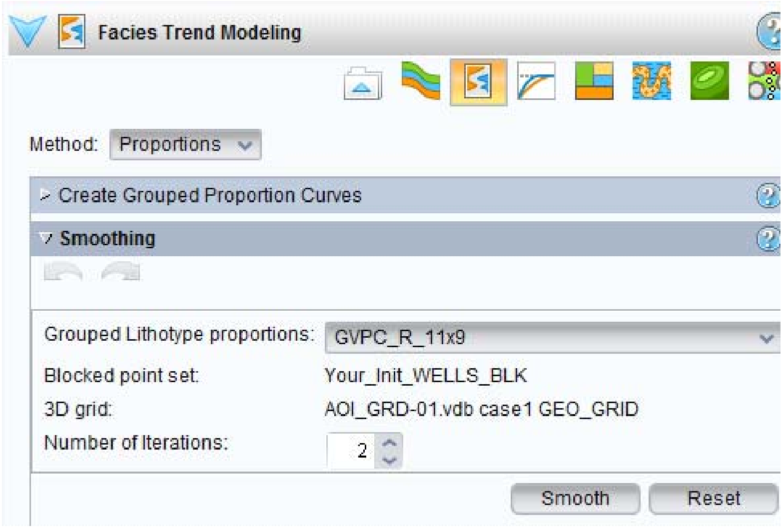
Smoothing the VPCs provides for a vertical gradation in the lithologies to provide for natural transitions between the lithologies.



Smoothed VPCs for Interval 2

For this example change the

- Number of iterations = **2**
- Click smooth once.
- The arrow is a undo previous action.



For the next set of exercises you will learn how to use the Copy and Edit functionality to have greater control on how the background trend is used during the facies modeling by imposing your interpretation on how the trend is created.

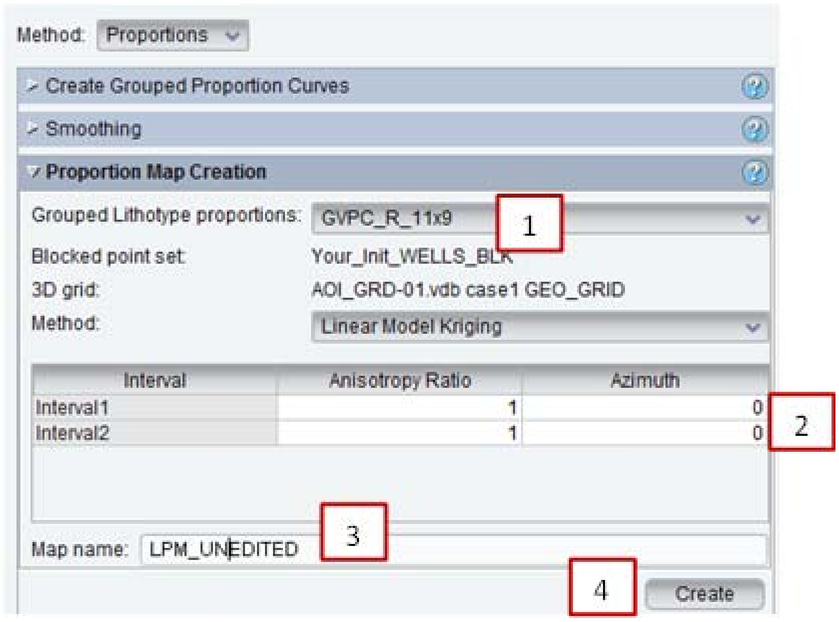
Before modifying the VPCS first create a Proportion Map (Lithology Proportion Matrix); a 3D volume of facies probabilities at every I, J, K location in the grid. Because you will use Linear Model Kriging to create the editing Lithology Proportion Matrix, create the unedited Lithology Proportion Matrix using the same method.

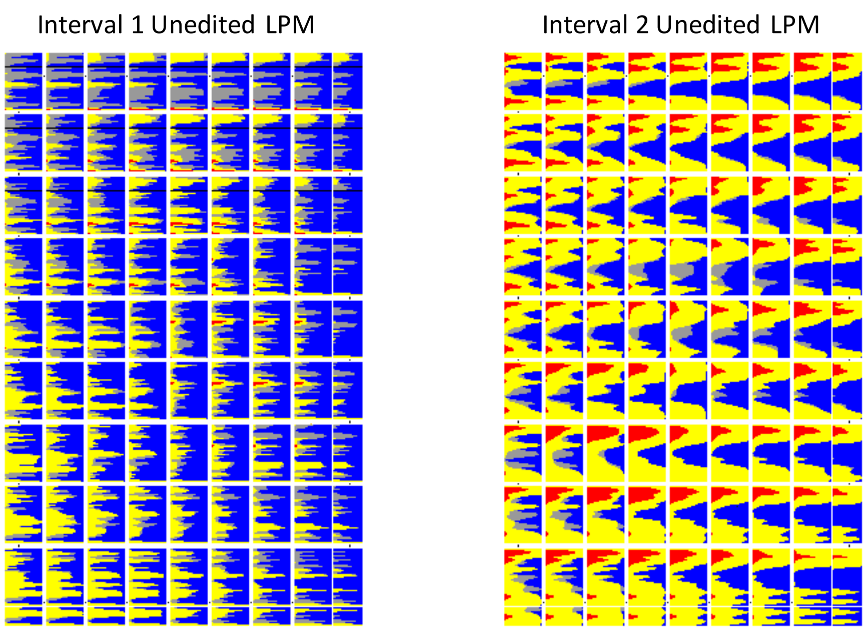
1. In Proportion Map Creation section change the Method to **Linear Model Kriging**.

2. Use the Default parameters.

3. Name the Map: LPM_UNEDITED.

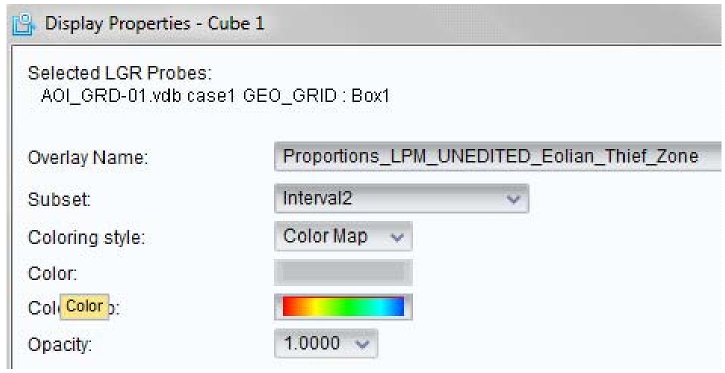
4. Click **Create**.





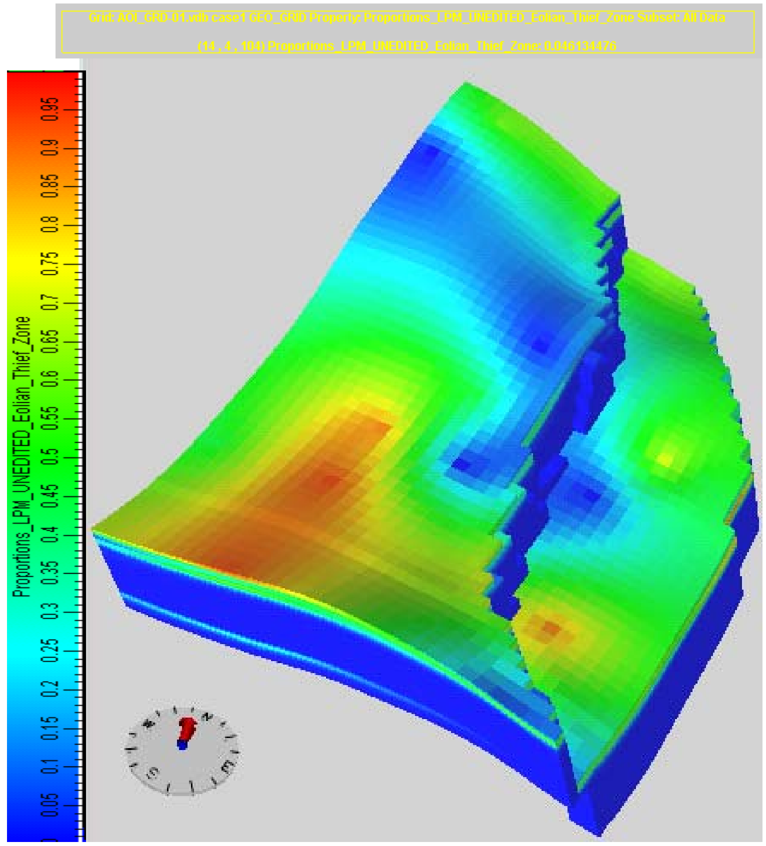
Results of Proportion Map Creation

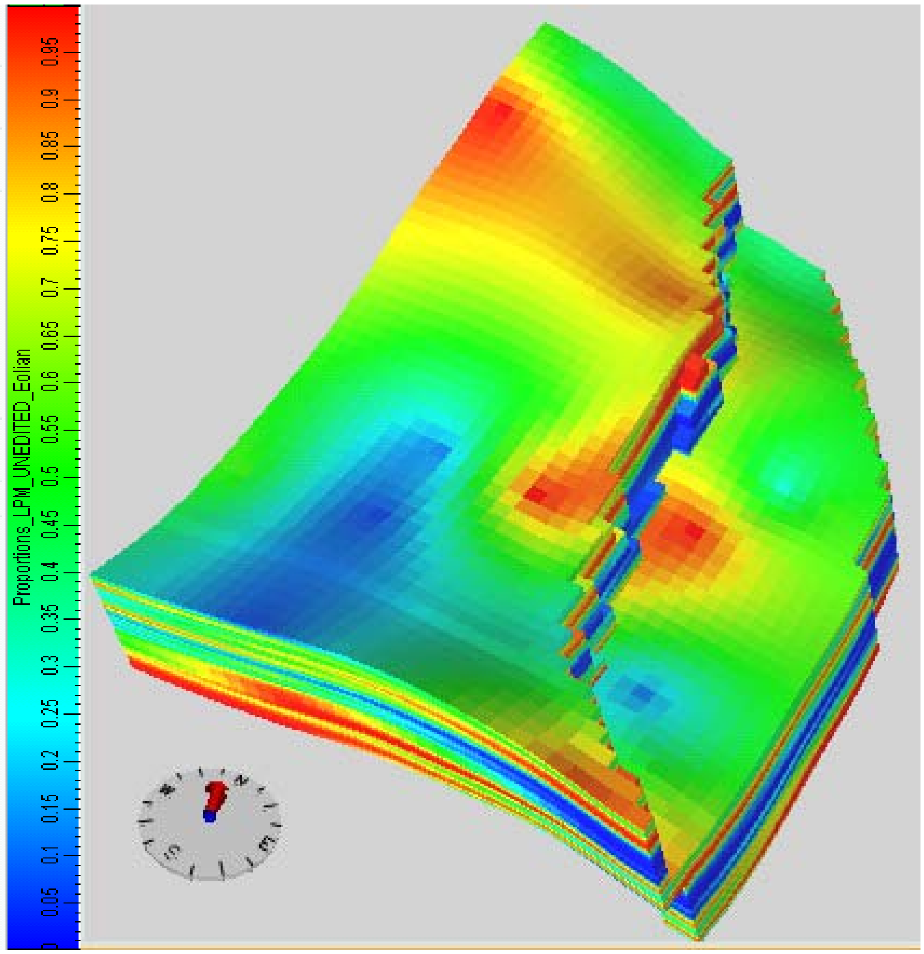
Because Interval 2 has the better quality reservoir with a higher proportion of the Thief Zone, you will modify the trend in this interval only. Creating the Lithology Proportion Matrix also creates the probability of each lithotype which can be displayed in the Cube View.



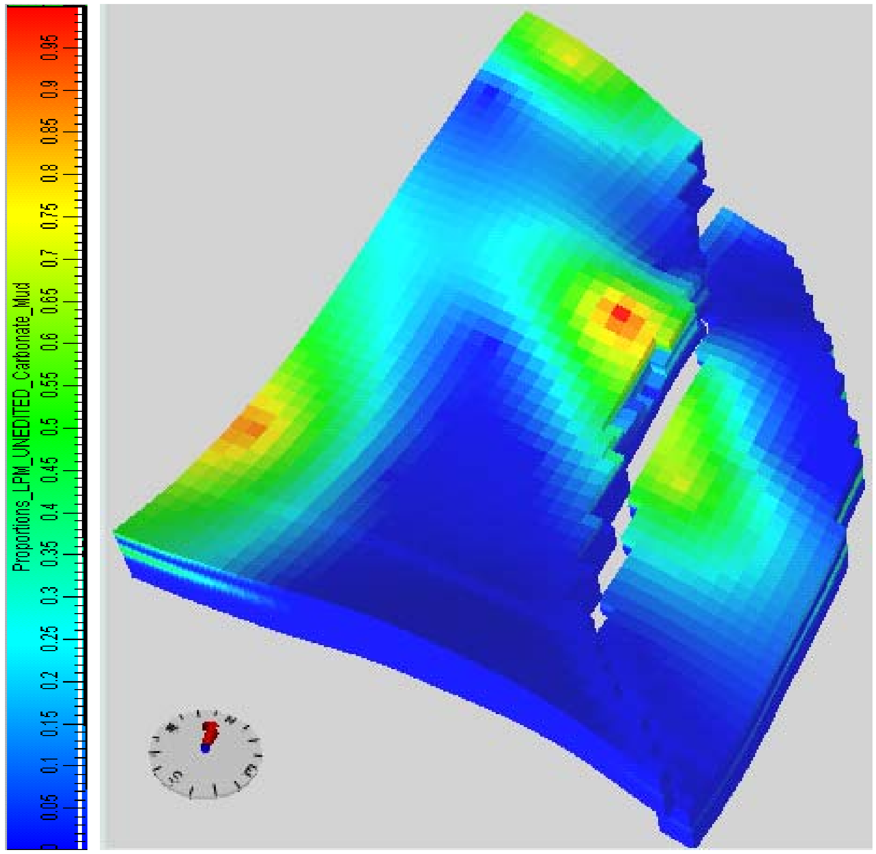
Display of the proportional of the Eolian Thief Zone for Interval 2 with K-plane 101 as the top surface: MB3 and turn on the annotation.

Then use SHIFT+MB1+DRAG to scroll through K-layers until layer #101 is displayed.

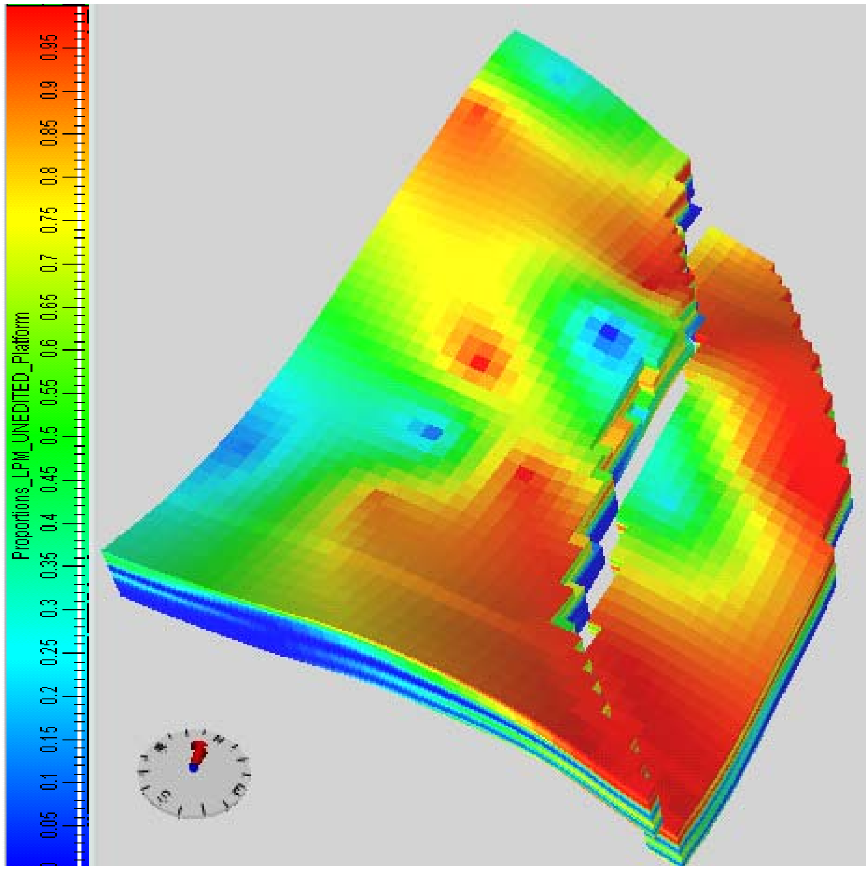




Display of the proportion of the Eolian Siltstone for Interval 2 with K-plane 101 as the top surface.



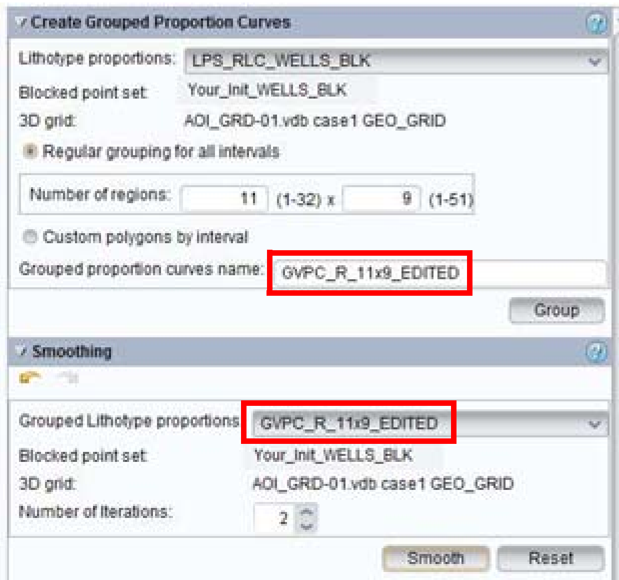
Display of the proportion of the Carbonate Mud for Interval 2 with K-plane 124 as the top surface.



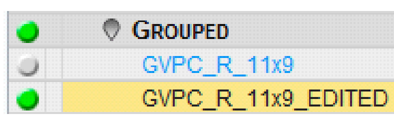
Display of the proportion of the Platform Dolomite for Interval 2 with K-plane 124 as the top surface.

Copy and Edit Interval 2 VPCs

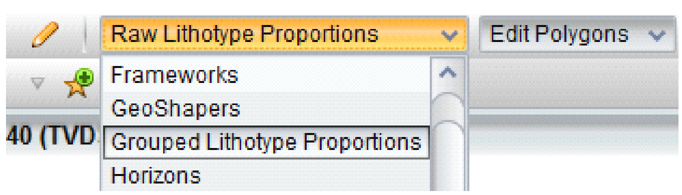
1. Make the Map View the active View.
2. Make another version of the Grouped VPCs following the previous steps.
3. Smooth the new grouped curves.



4. Display the **edited Interval 2** VPCs in Map View as a curve type.

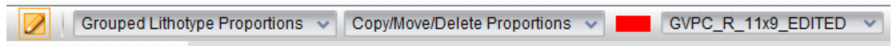


5. In the Main Tool Bar (next to the Pencil) change the Raw Lithotype Proportions to **Grouped Lithotype Proportions**.

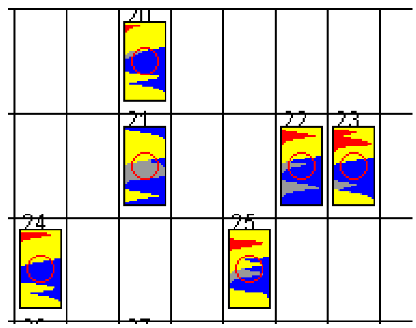


6. This activates the **Copy/Move/Delete Proportions** for the edited grouped VPCs.

7. Using MB1, click the **Pencil** icon. In the map view you should see a red circle on each VPC.



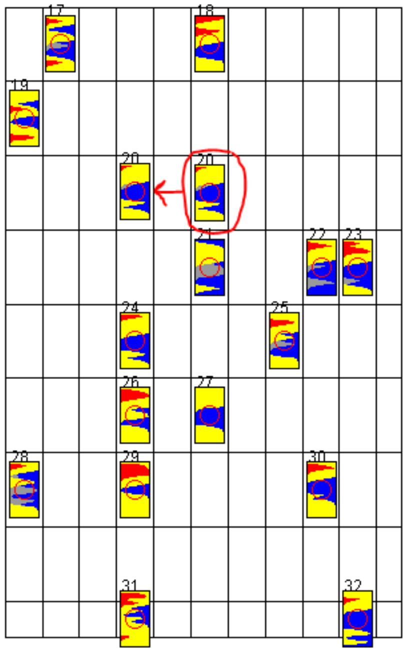
When a red circle is displayed on a VPC you can Copy (MB1), Move (MB1) or Delete (MB2) any VPC.

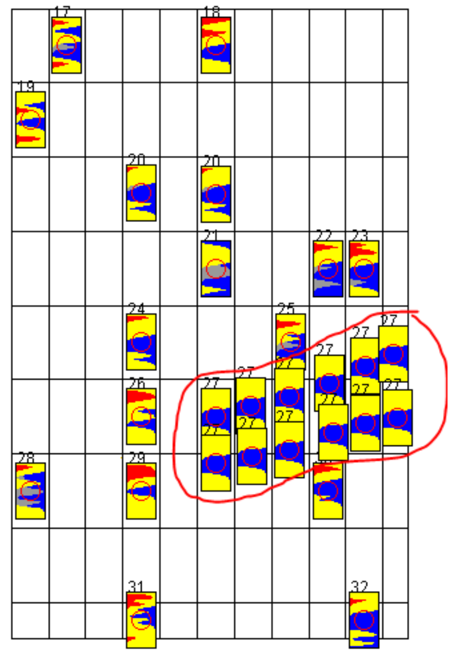


Next you will copy a VPC and edit it, then make several copies of the edited VPC.

8. Copy **VPC 20** (Interval 2) by clicking on the VPC and holding down MB1 and move it two cells to the left as shown in the illustration. Later you will edit this curve and make more copies.

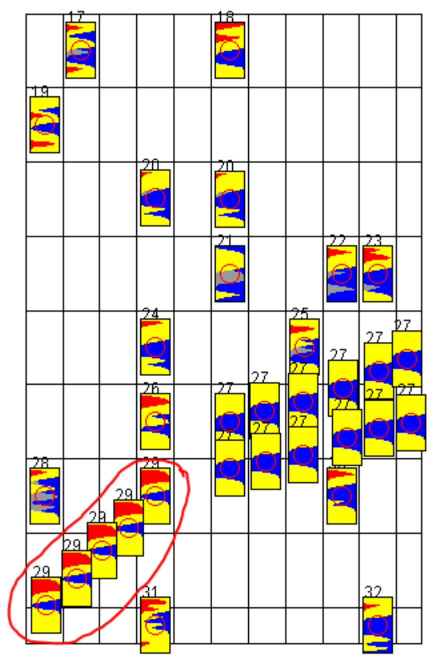
9. Copy **VPC 27** several times as to make a barrier for the Thief Zone.

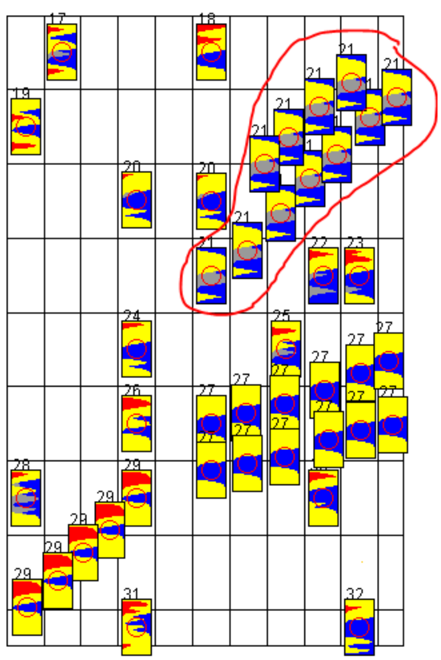




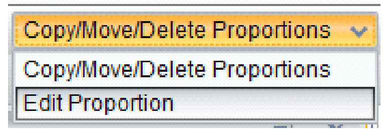
10. Copy **VPC 29** several times enhance the Thief Zone tend to the SW.

11. Copy **VPC 21** to impose a barrier to the NE.





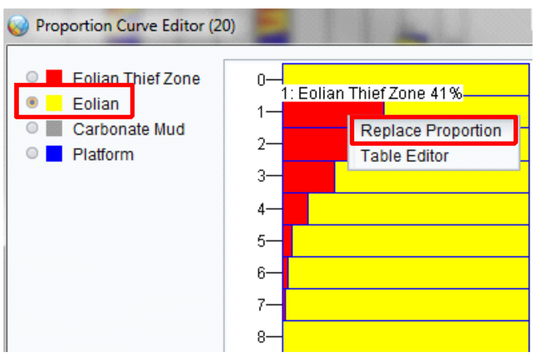
12. To edit the copied version of VPC 20 to remove the Thief Zone, go to the Main Tool Bar and Change Copy/Move/Delete Proportions to **Edit Proportion**.



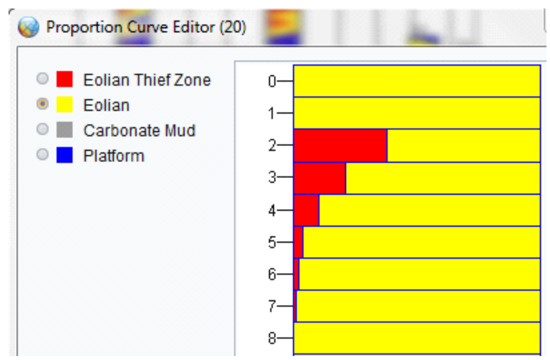
13. Using MB1 inside the red circle, click the **copied VPC20** to open the editing window. The Red lithotype (Eolian Thief Zone is shown to be in layers 1-7 (your example might differ slightly).

14. If not already selected, activate the **Eolian lithotype** (Yellow) radio button to modify the proportions layer-by-layer.

15. With MB 3 on layer 1, put the cursor over the lithology to be replaced, in this example, the red lithology which is the Eolian Thief Zone, then select **Replace Proportion**.

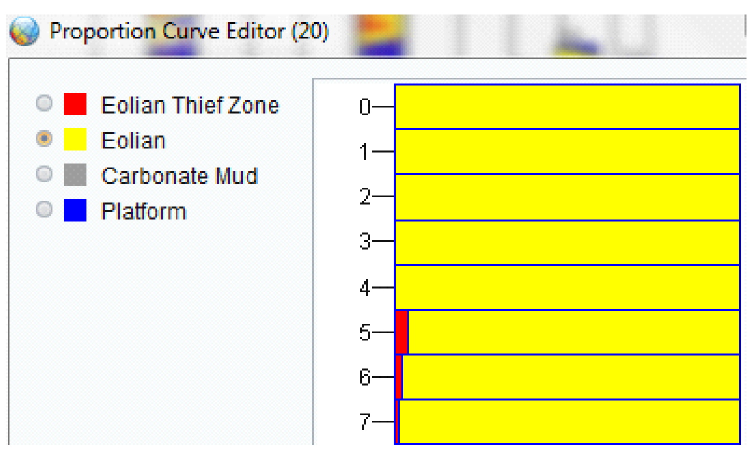


Before replacement

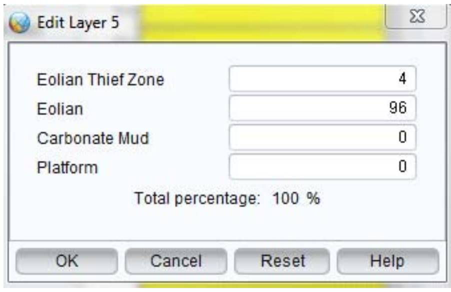


After replacement

16. Use the Replacement option for layers 2-4.

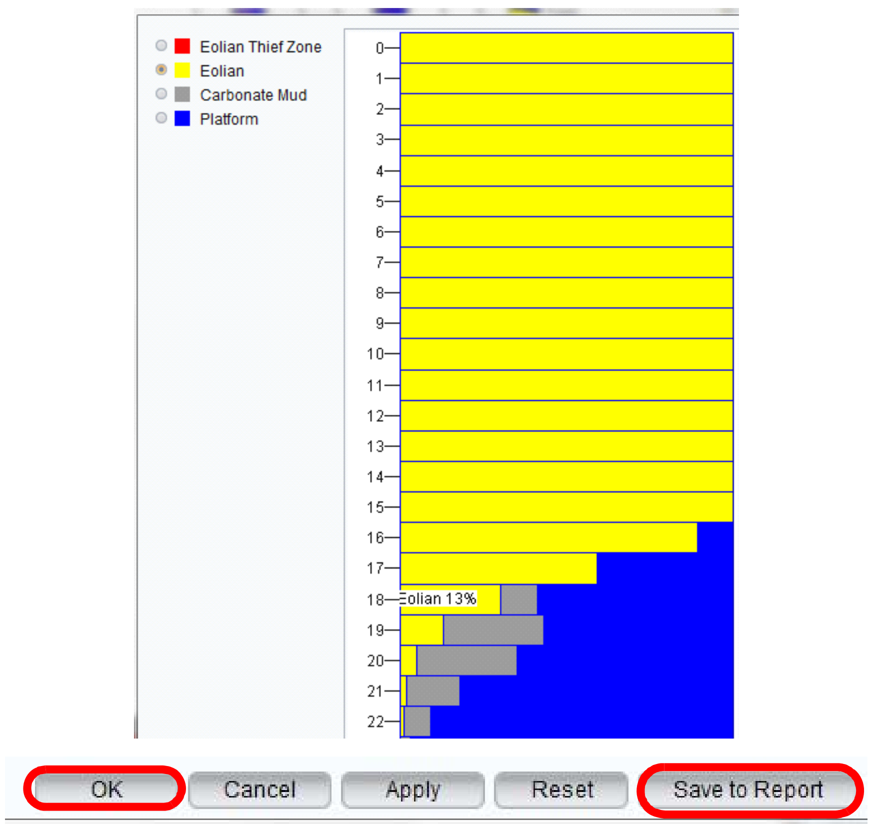


17. Use the Table Editor for the remaining layers.



Before editing layer 5

After editing layer 5

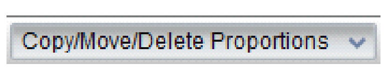


Final edited copied version of VPC 20

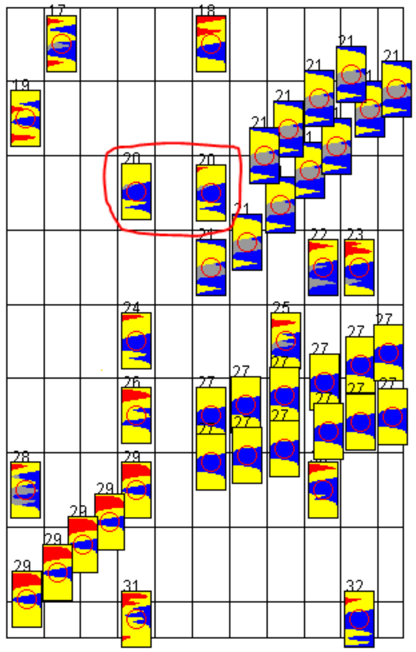
18. When editing is finished Click Save to Report.

19. Click OK to close the editing window.

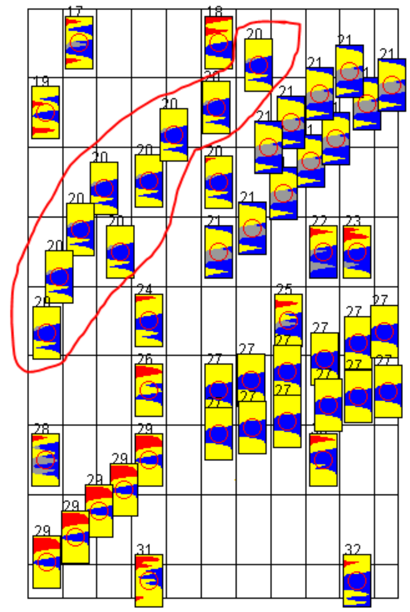
20. Return to the Copy/Move/Delete Mode.



21. Copy edited VPC 20 as shown.

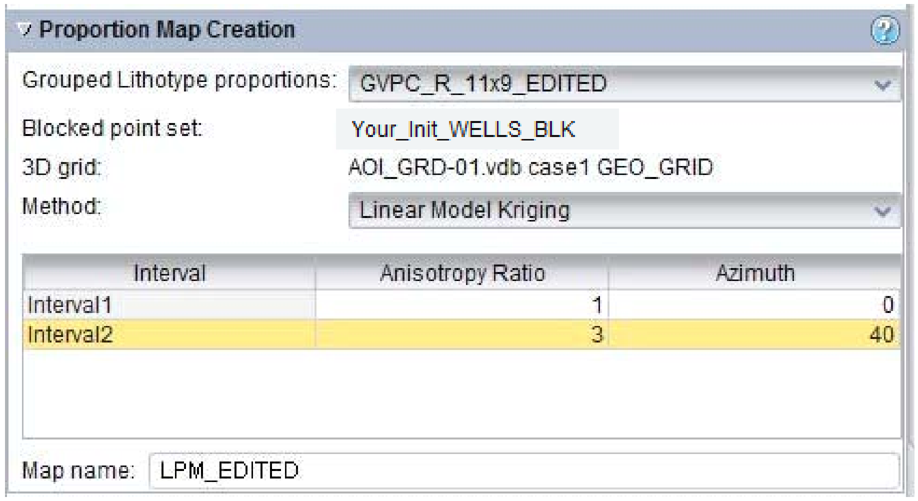


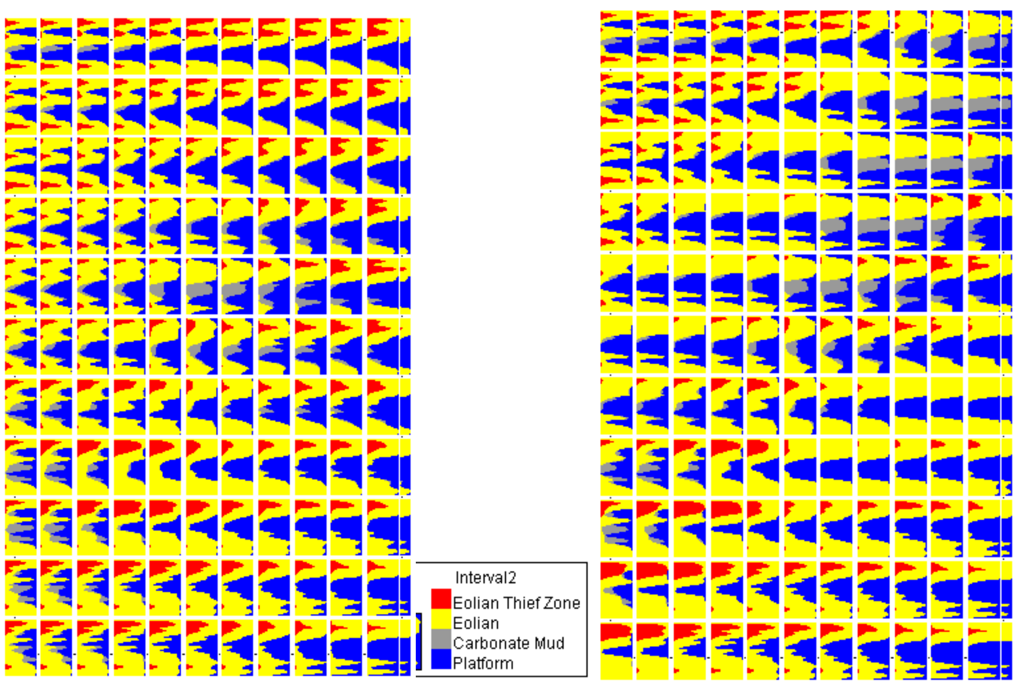
Edited and unedited version of VPC 20



22. Create a new Lithology Proportion Matrix for the Edited grouped VPCs using the setting as shown below.

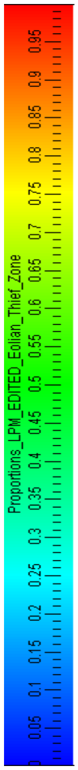
23. Provide a new Lithology Proportion Matrix name and then Create.

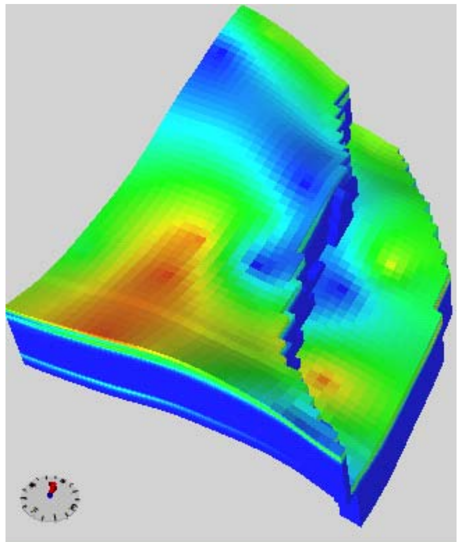




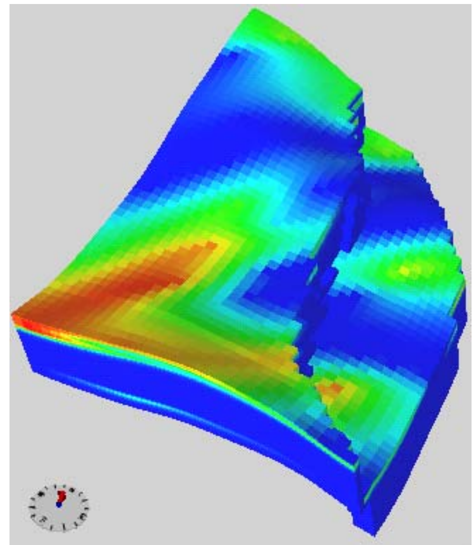
Unedited LPM Edited LPM

This illustrates how you can impose an interpretation to control the background trend used in the facies modeling. Your edited Lithology Proportion Matrix may differ as it depends upon how you copied and placed the additional VPCs.



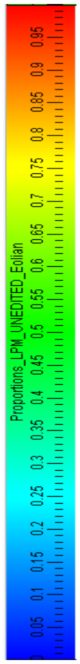


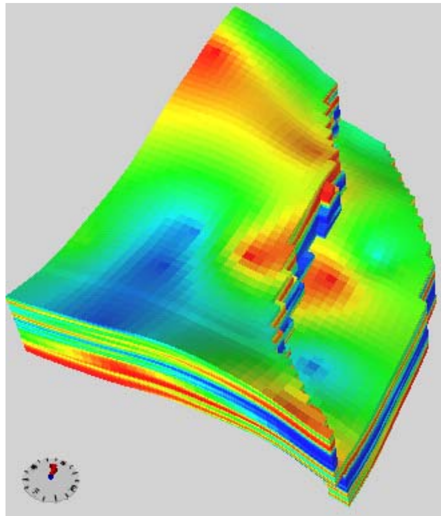
Unedited Eolian Thief Zone with k-layer 101 as top surface



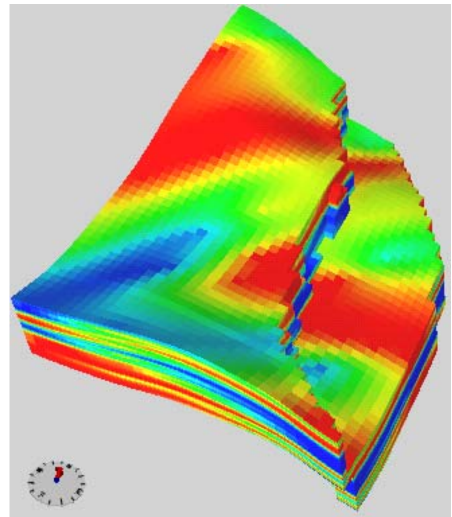
Edited Eolian Thief Zone with k-layer 101 as top surface

Unedited and Edited Proportion Volumes for Interval 2



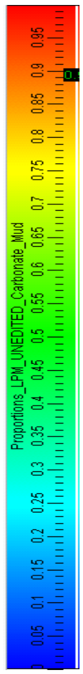


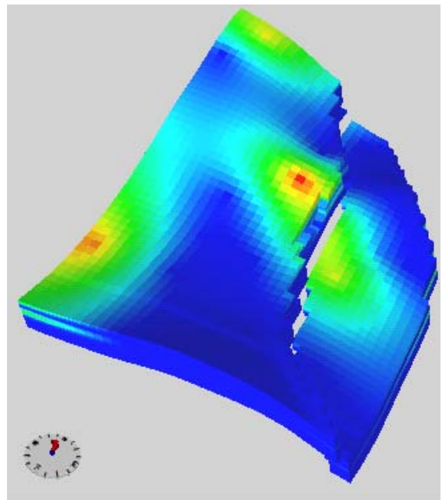
Unedited Eolian Siltstone with k-layer 101 as top surface



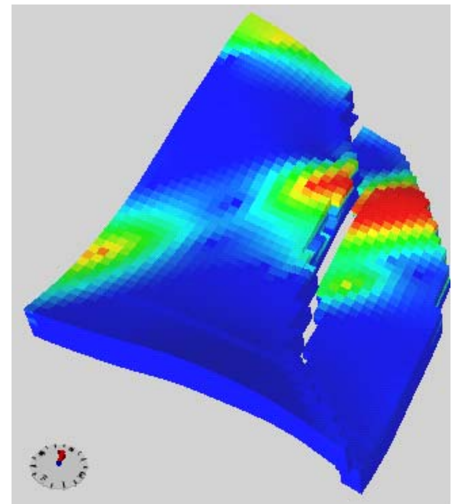
Edited Eolian Siltstone with k-layer 101 as top surface

Unedited and Edited Proportion Volumes for Interval 2



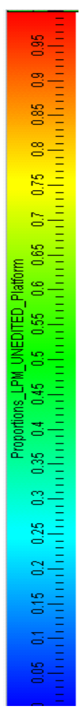


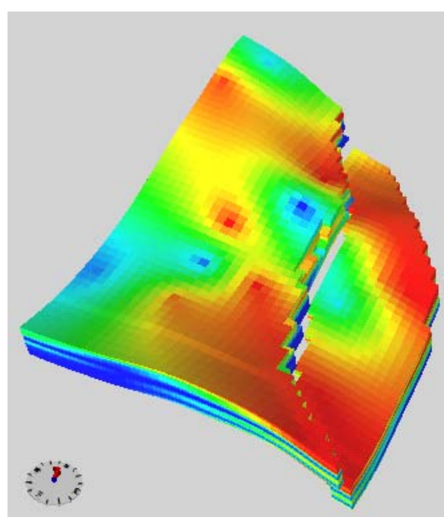
Unedited Carbonate Mud with k-layer 124 as top surface



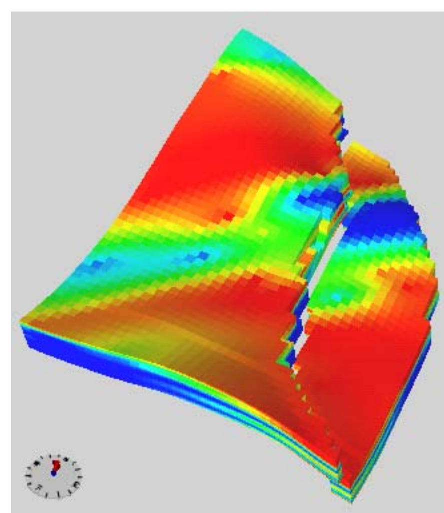
Edited Carbonate Mud with k-layer 124 as top surface

Unedited and Edited Proportion Volumes for Interval 2





Unedited Platform Dolomite with k-layer 124 as top surface



Edited Platform Dolomite with k-layer 124 as top surface

Save the session.

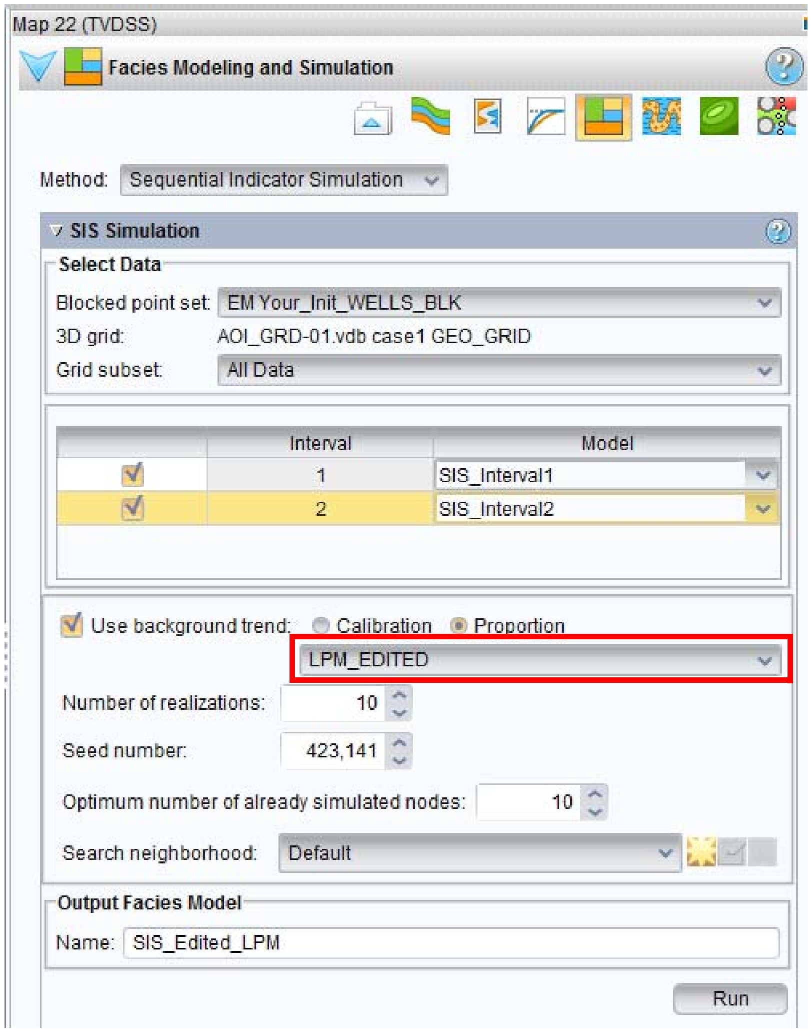
Facies Modeling and Simulation

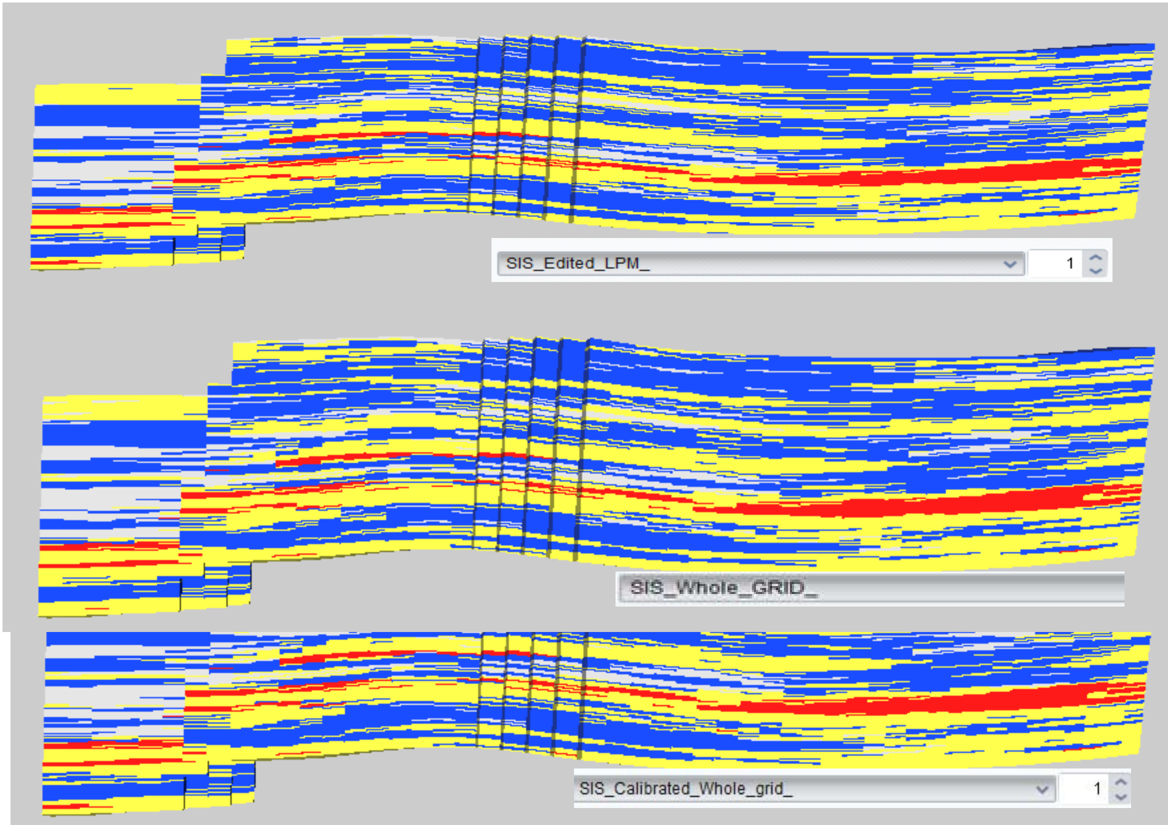
Sequential Indicator Simulation (SIS)

For the SIS you will use the edited Lithotype Proportion Matrix.

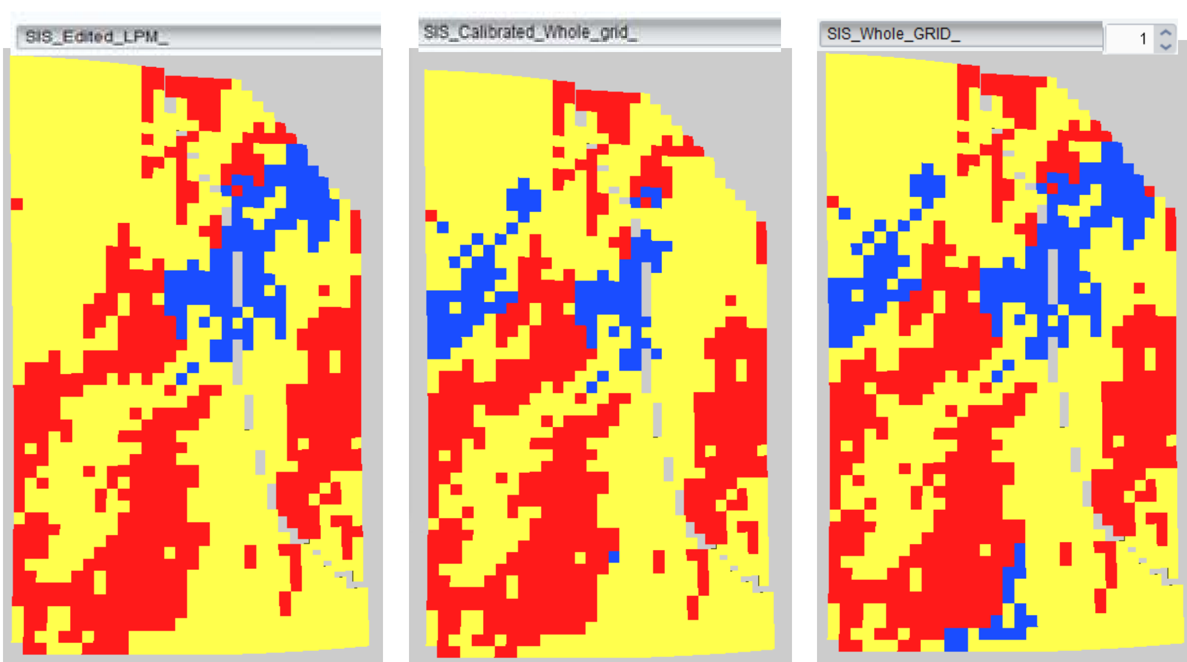
1. Open the Facies Modeling and Simulation task pane.
2. Method: **Sequential Indicator Simulation**.
3. Enter Select Data as shown.
4. Use **Proportion** as the background trend.
5. Number of simulation: **10**
6. Name: SIS_Edited_LPM
7. Run.

8. Display the realizations.





Section View of N-S section Subset



K-plane 101 probes for different realizations

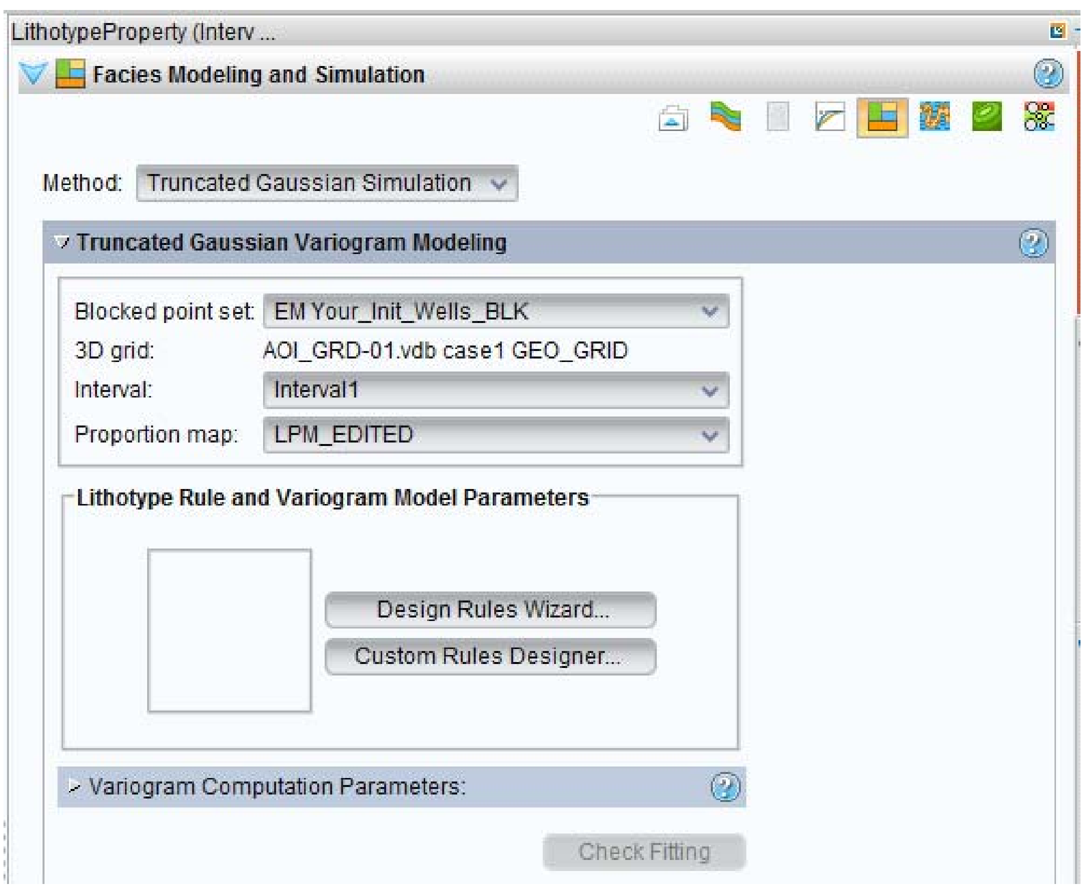
Truncated Gaussian Simulation

Truncated Gaussian Simulation is a simple facies transition rule based method. You will learn how to define rules using two different methods:

- Using the Design Rules Wizard which allows you to select from templates, or using the
- Custom Rules Designer

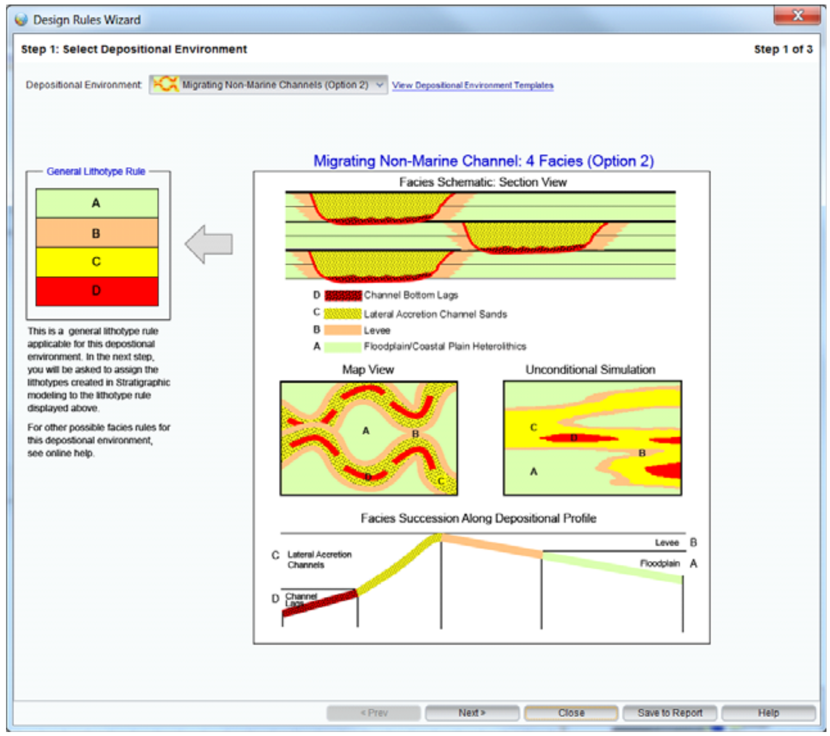
From the Facies Modeling and Simulation task pane:

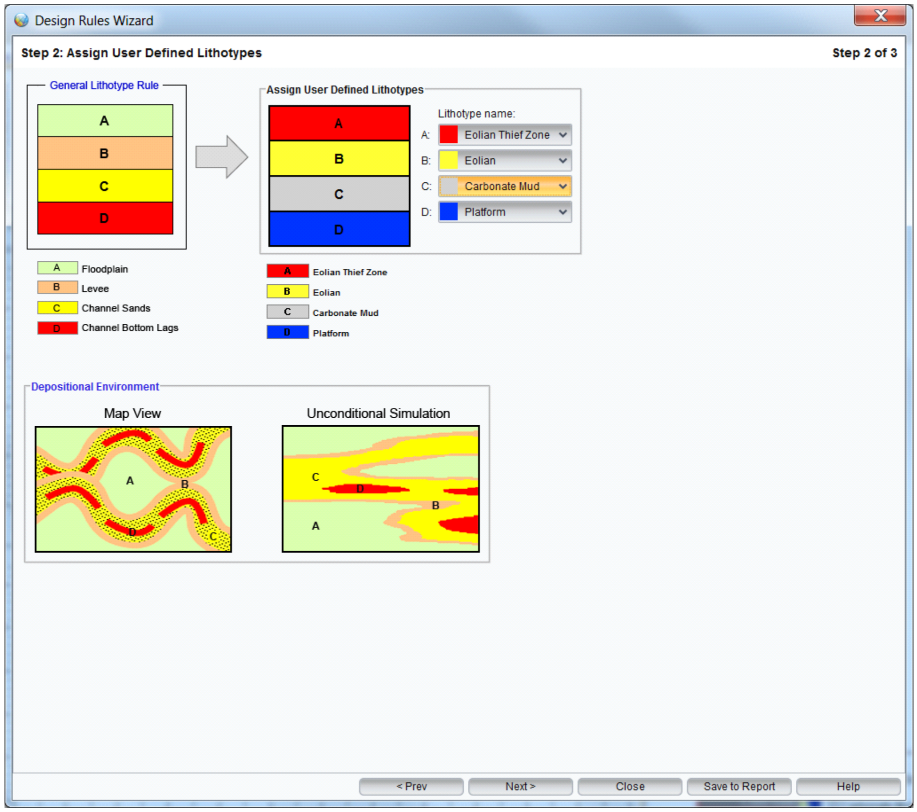
1. Select **Truncated Gaussian Simulation Method**.
2. Select **Interval 1**.
3. Select **LPM_EDITED** as Proportion map.
4. Click **Design Rules Wizard**.



Simple Transitions

A simple transition facies template isn't available for the west Texas depositional environment, but this template will work. Simply follow the steps in the Wizard and fill in the Assignments as shown.





Design Rules Wizard
Step 2: Assign User Defined Lithotypes
Step 2 of 3
General Lithotype Rule
A
B
C
D
Floodplain
Levee
Channel Sands
Channel Bottom Lags
Assign User Defined Lithotypes
Lithotype name:
A: Eolian Thief Zone
B: Eolian
C: Carbonate Mud
D: Platform
Eolian Thief Zone
Eolian
Carbonate Mud
Platform
Depositional Environment
Map View
Unconditional Simulation
< Prev
Next >
Close
Save to Report
Help

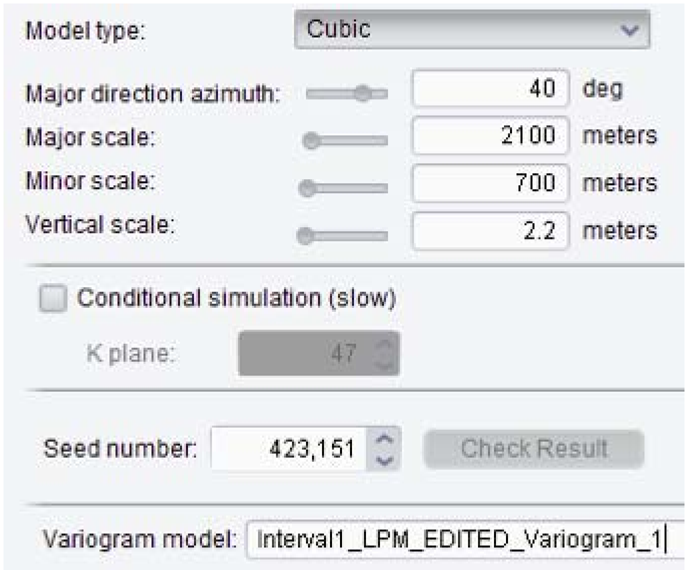
Adjust Lithotype Rules

This window allows you to change the variogram parameters to control the simulation. Because there are not enough wells to actually compute and model the variogram, you will need to enter parameters that are reasonable for the depositional setting.

1. You can load an existing variogram or in this case, type the parameters for the new one.

2. You can check the transition Statistics to see how often each lithotype is in contact with the others.

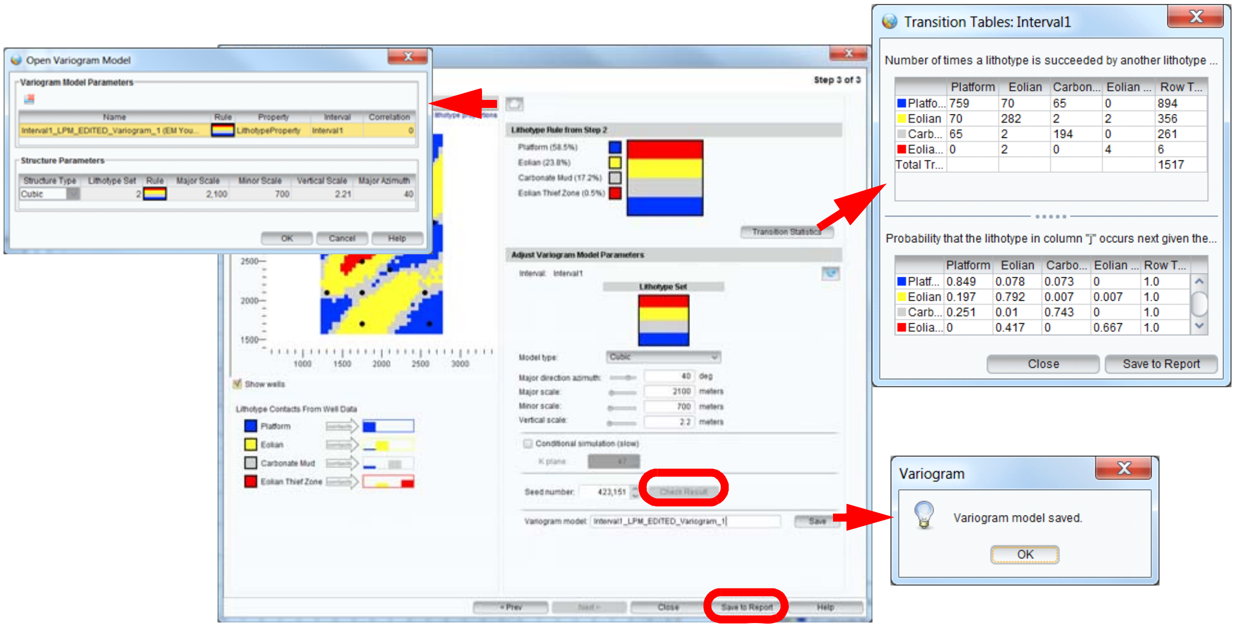
3. Use the following parameters.



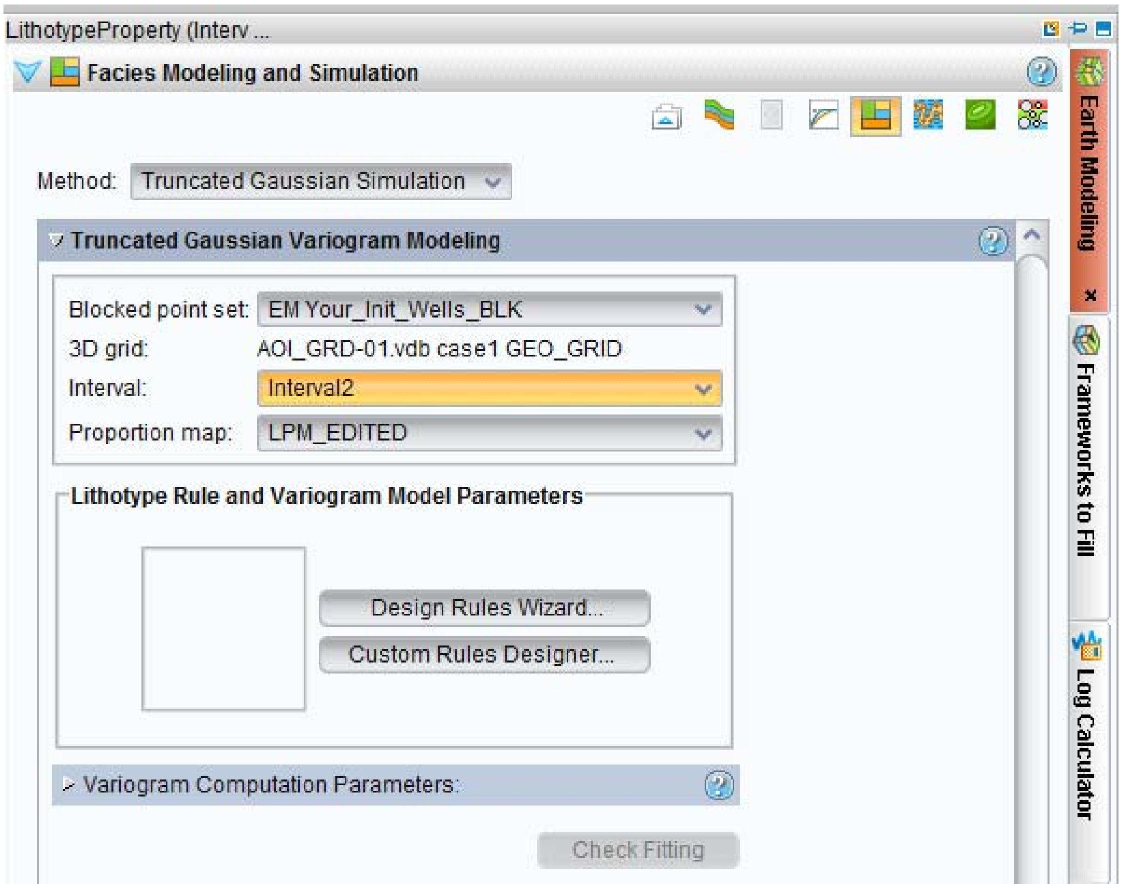
4. Check results.

5. Save variogram.

6. Close.


Open Variogram Model
Transition Tables: Interval1
Number of times a lithotype is succeeded by another lithotype ...
Probability that the lithotype in column "j" occurs next given the...
Close
Save to Report
Variogram
Variogram model saved.
OK

- Select **Truncated Gaussian Simulation** Method.
  - Select **Interval 2**.
  - Select **LPM_EDITED** as Proportion map.
  - Click **Custom Rules Designer**.

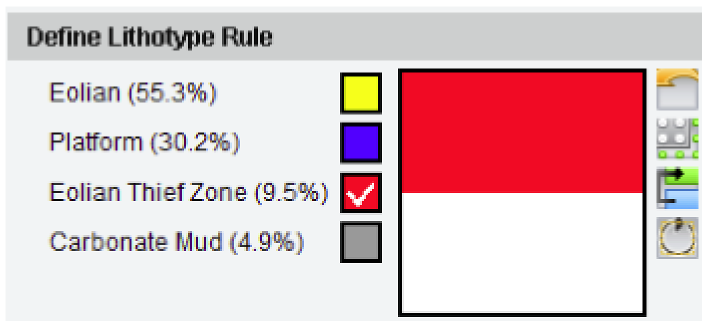


Customer Rules Designer

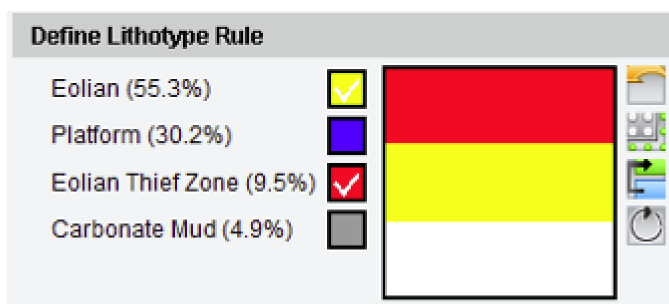
When you first enter the Custom Rules Designer Dialog the Define Lithotype Rule box is empty. Each Lithotype for the active interval is display with its proportions. Your task is to fill-in the Rule Box with the correct transition rules. Remember that in Truncated Gaussian only simple A>B>C>etc. transitions can be defined.

1. With MB1, select the red box for Eolian Thief Zone to active it. It has a green border when active.

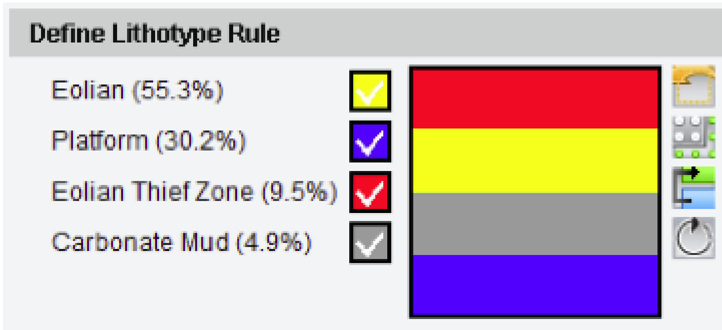
2. With MB1, click inside the empty box near the top border to place the Thief Zone lithotype. The red box is checked when assigned.



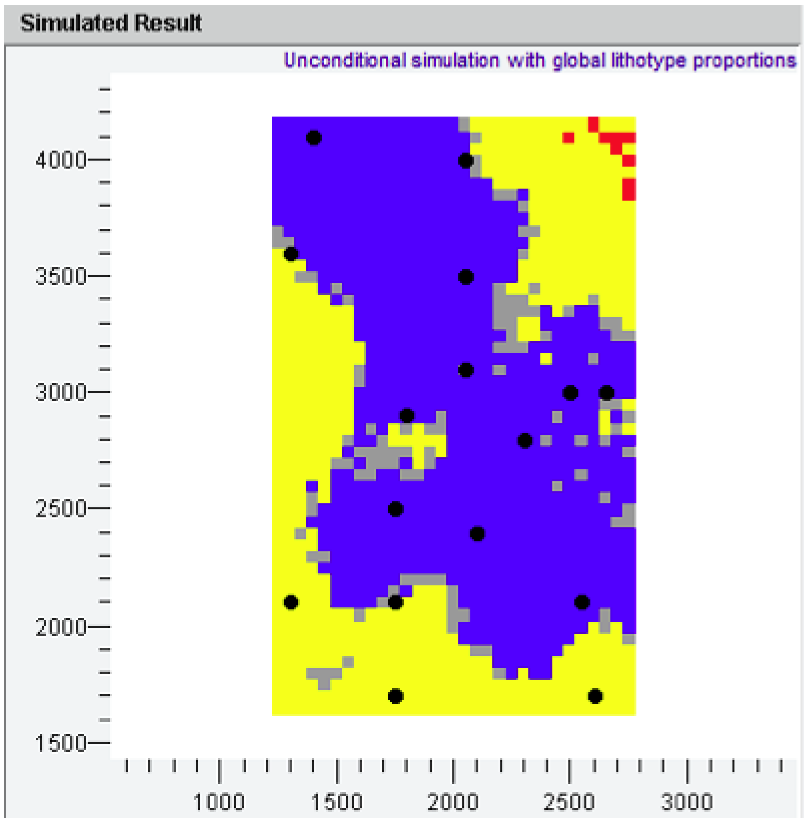
3. Select **Eolian** and click below the red bar to add it.



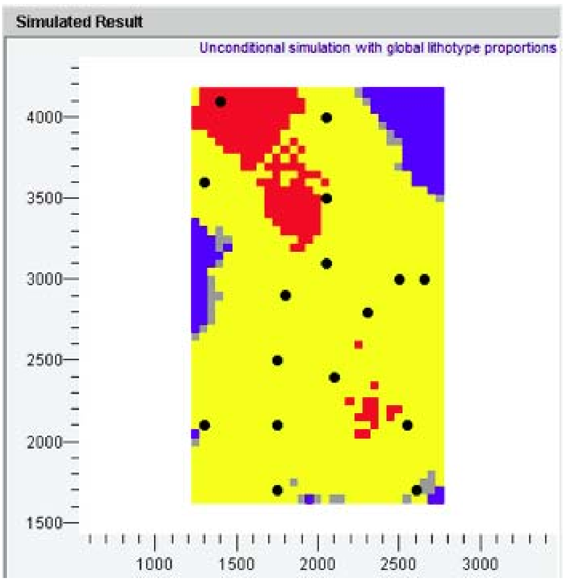
4. Add the Carbonate mud, which also adds the Platform below it.

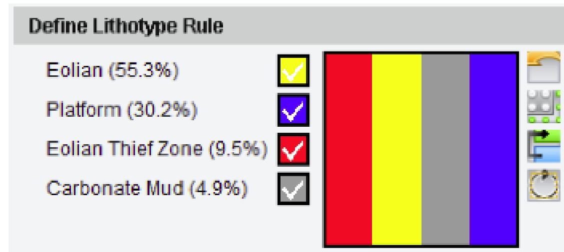


Once the rules are defined, an unconditional simulation is displayed using default variogram parameters.

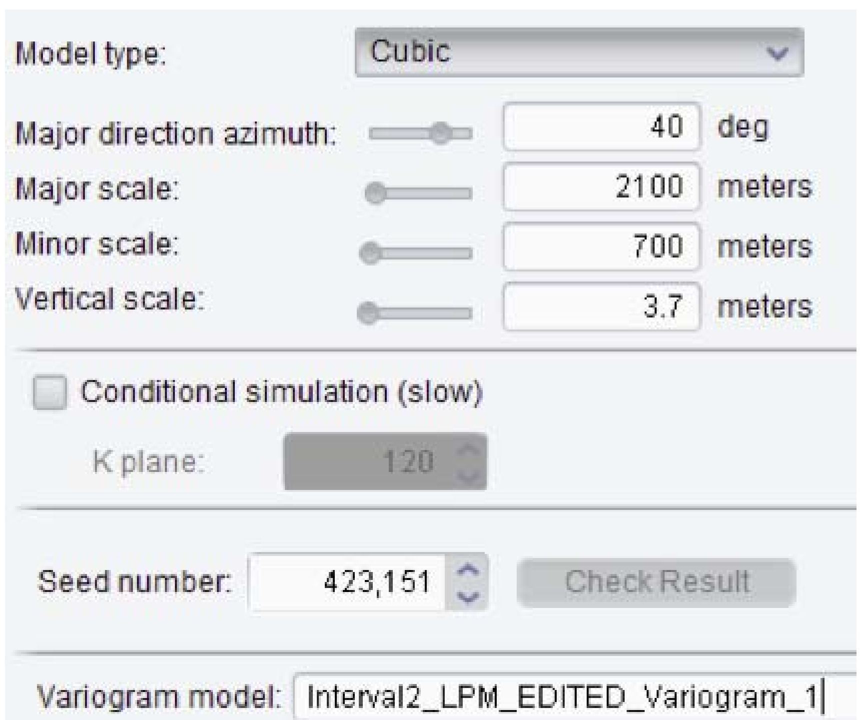


5. Click the rotate icon to change the rule order. Notice that the Thief Zone is obviously inside the Eolian when the truncation order is switched. Although either rule is correct.





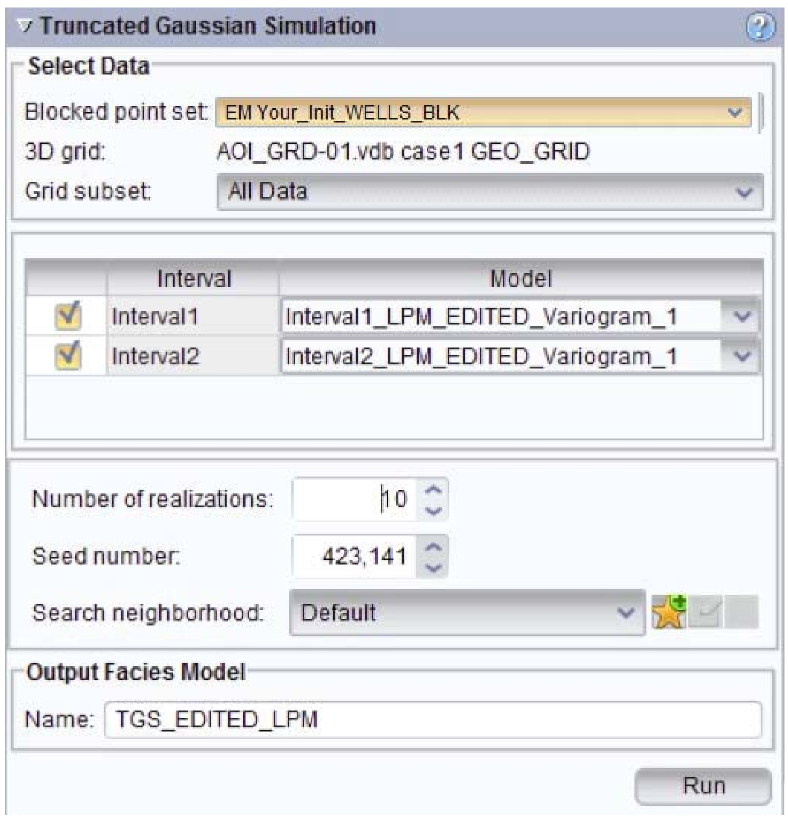
6. Use these variogram parameters.



7. Click **Check Results** to update the unconditional simulation.
8. **Save** the variogram.
9. **Close**.
10. **Save the Session**.

Run the Truncated Gaussian Simulation

1. Select the point set, which selects automatically
   - The 3D grid
   - And the Grid subset (make sure to use All Data)
   - Last saved variogram models
2. Number of realizations: **10**
3. Default **Seed number** and **Search neighborhood**.
4. Output name: **TGS_EDITED_LPM**
5. **Run**. This may take longer than Sequential Indicator Simulation as it is a more sophisticated algorithm.



Before making any displays you will create 10 Plurigaussian realizations and then do some comparisons of the methods.

Plurigaussian Simulation

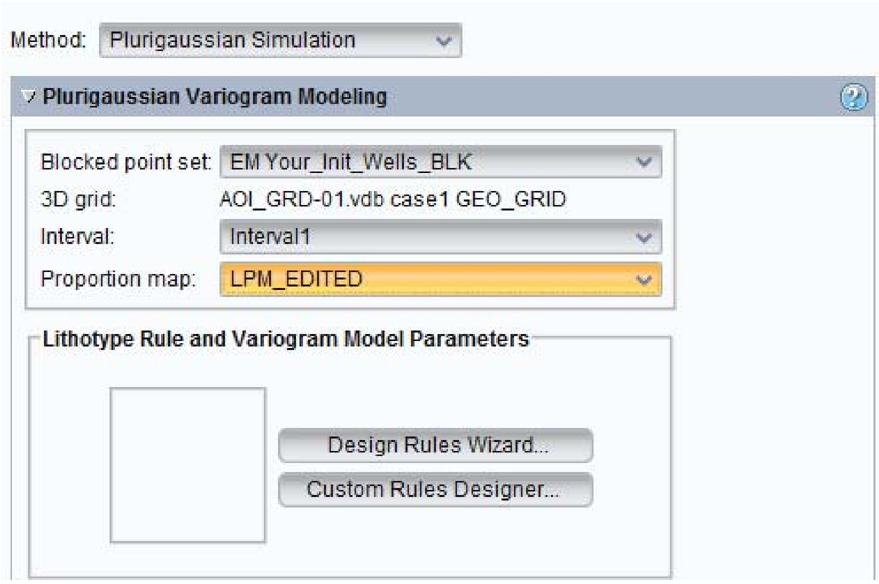
Plurigaussian Simulation is an extension to Truncated Gaussian allowing for more complex facies relationships. In this exercise you learn how to design more complex rules, perform k-layer conditional simulation and use the variogram correlation feature. For this exercise the rules designed are not necessarily the rule that fits this environment, rather the exercise is designed to illustrate the flexibility when designing the rules.

1. In Facies Modeling and Simulation select the **Plurigaussian Simulation** Method.

2. Select **Interval 1**.

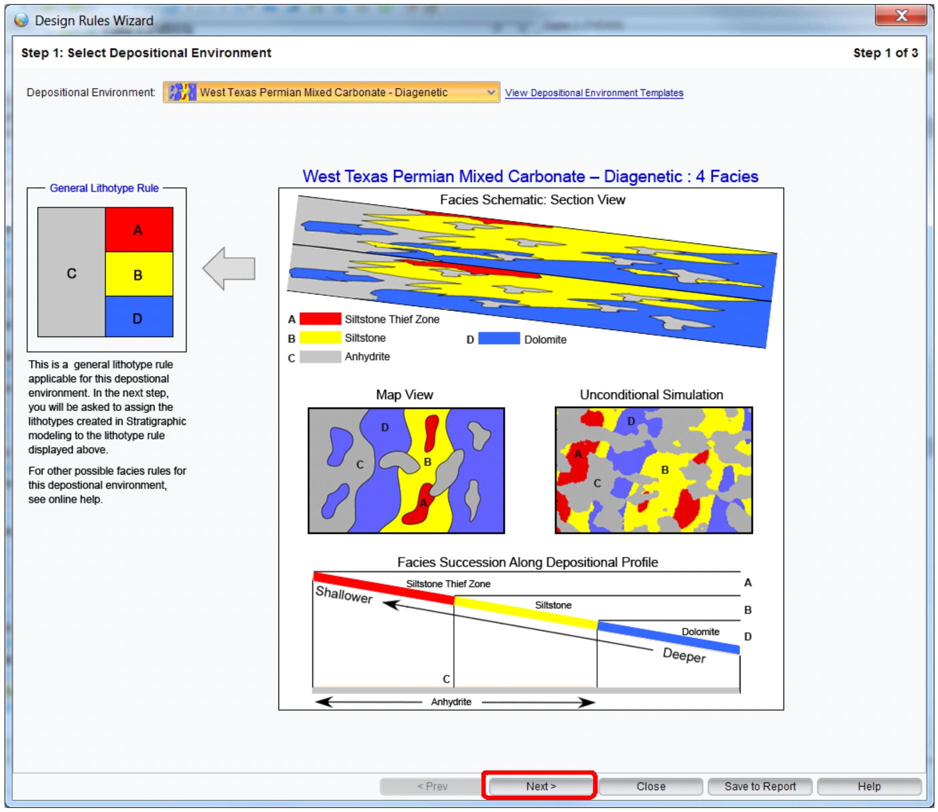
3. Select **LPM_EDITED** as Proportion map.

4. Click **Design Rules Wizard**.

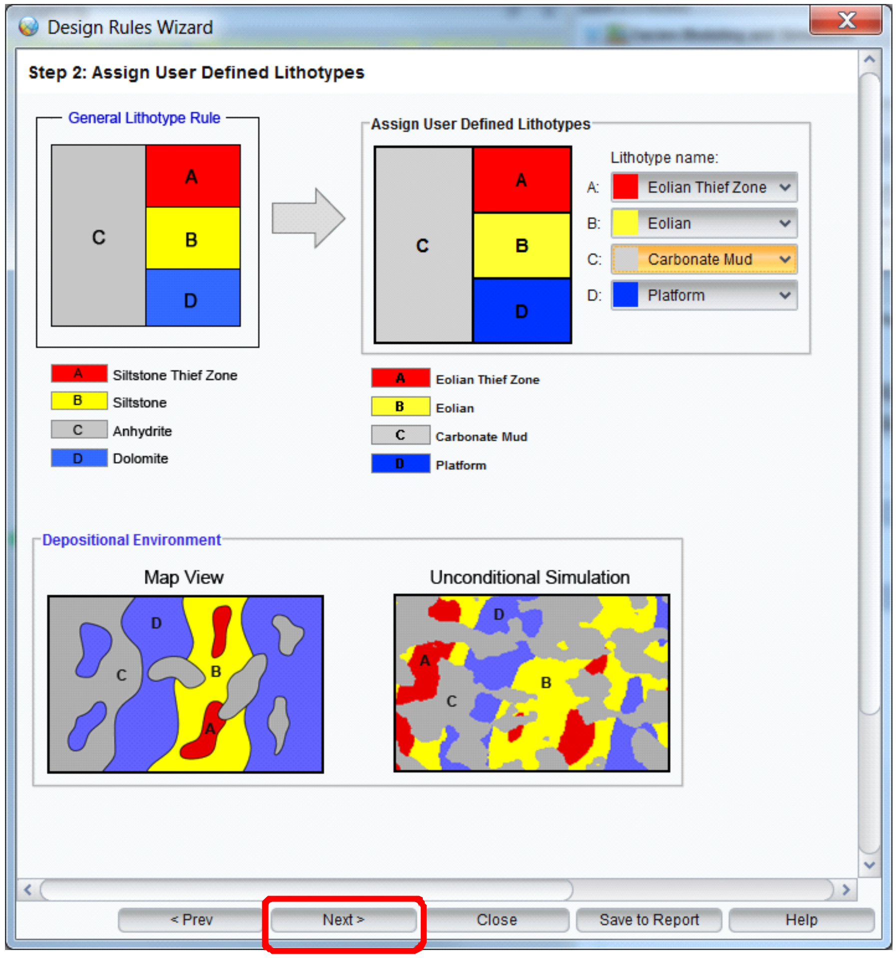


5. Select the **West Texas Permian Mixed Carbonate – Diagenetic** template. For this set of rules treat the carbonate mud as a diagenetic overprint.

6. Click **Next**.


Design Rules Wizard
Step 1: Select Depositional Environment
Step 1 of 3
Depositional Environment:
West Texas Permian Mixed Carbonate - Diagenetic
View Depositional Environment Templates
General Lithotype Rule
A
B
C
D
This is a general lithotype rule applicable for this depostional environment. In the next step, you will be asked to assign the lithotypes created in Stratigraphic modeling to the lithotype rule displayed above.
For other possible facies rules for this depostional environment, see online help.
West Texas Permian Mixed Carbonate – Diagenetic : 4 Facies
Facies Schematic: Section View
A Siltstone Thief Zone
B Siltstone
C Anhydrite
D Dolomite
Map View
Unconditional Simulation
Facies Succession Along Depositional Profile
Siltstone Thief Zone
Shallower
Siltstone
Dolomite
Deeper
Anhydrite
< Prev
Next >
Close
Save to Report
Help

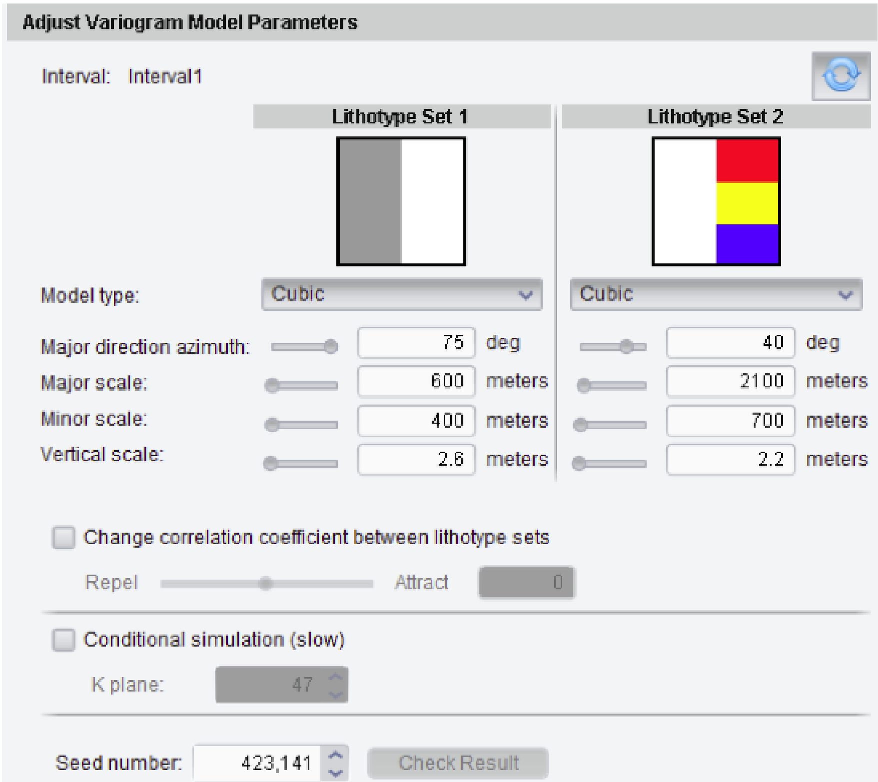
7. Assign the lithotypes as shown.



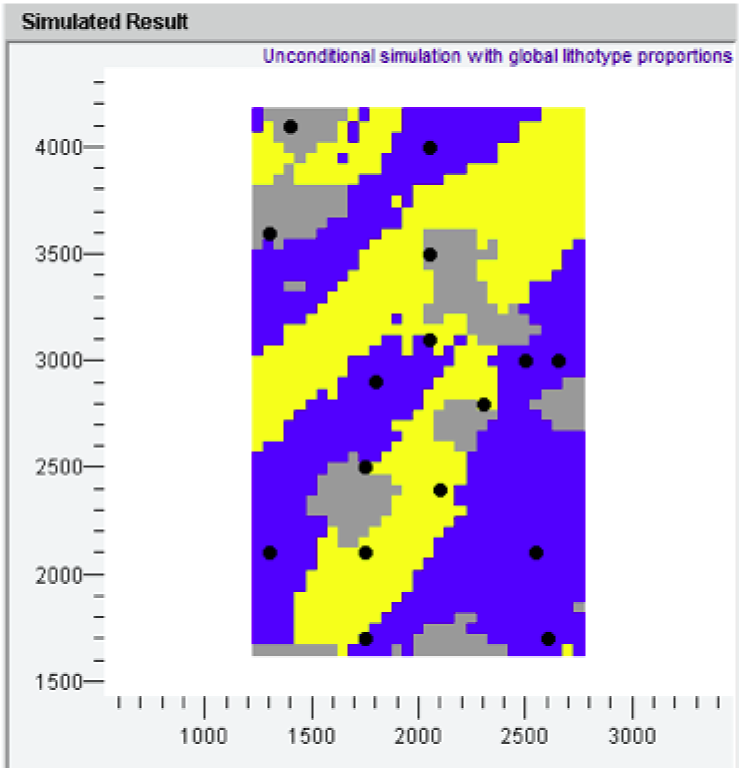
8. Click **Next**.

9. Adjust the variogram parameters as shown.

10. Click **Check Result**.

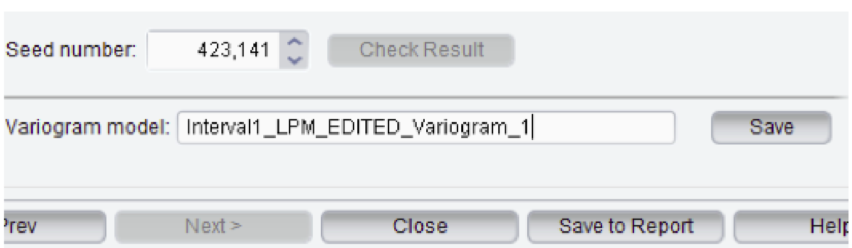


Look at the image of the unconditional simulation and observe that the carbonate mud appears as patches that cross-cut the other lithologies. The rule shows that the carbonate mud can be in contact with the other 3 lithologies and that it will appear as an overprint with a scale that on average is 600 by 400 meters with its long scale oriented N75E.



11. Save the variogram model.

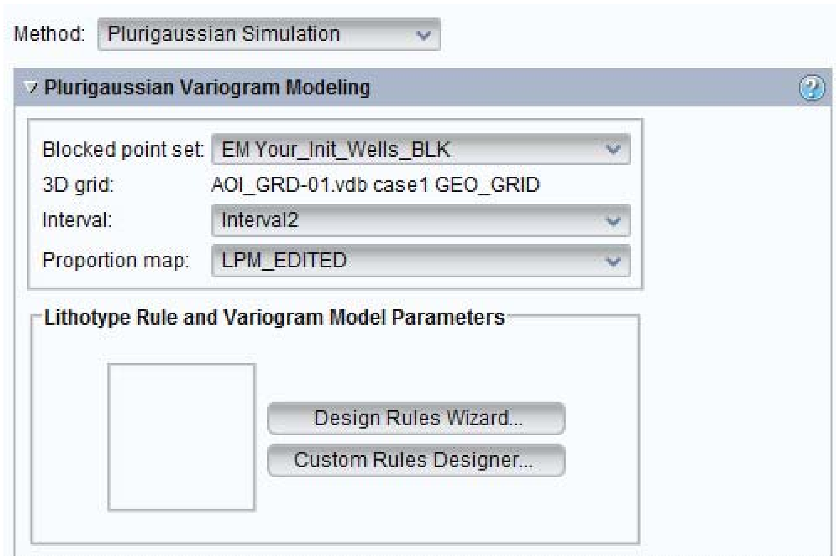
12. Close the Window.



For interval 2 you will learn how to manually design the transition rules and create a k-plane conditional simulation and use the correlation feature.

Plurigaussian Interval 2: Custom Rules

1. Select **Plurigaussian Simulation Method**.
2. Select **Interval 2**.
3. Select **LPM_EDITED** as Proportion map.
4. Click **Custom Rules Designer**.



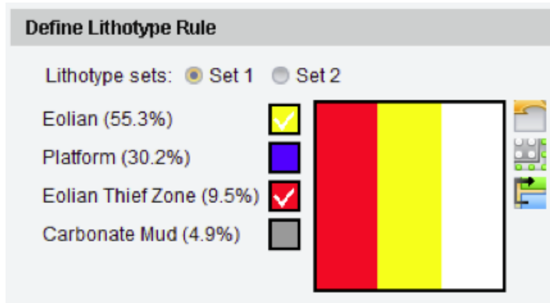
Next you need to understand how to use the Plurigaussian rule designer.

Plurigaussian uses two variogram models, one for each lithoset, which define lithologies sharing common geometries. In the rule designer you define Lithotype Set 1 and Set 2.

If a lithotype for Set 1 is selected it shows up as a vertical bar in the rule box.

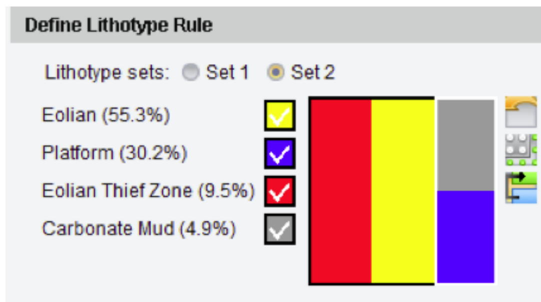
Set 2 is defined as a horizontal bar.

5. Use **Set 1** and add the **Thief Zone**.



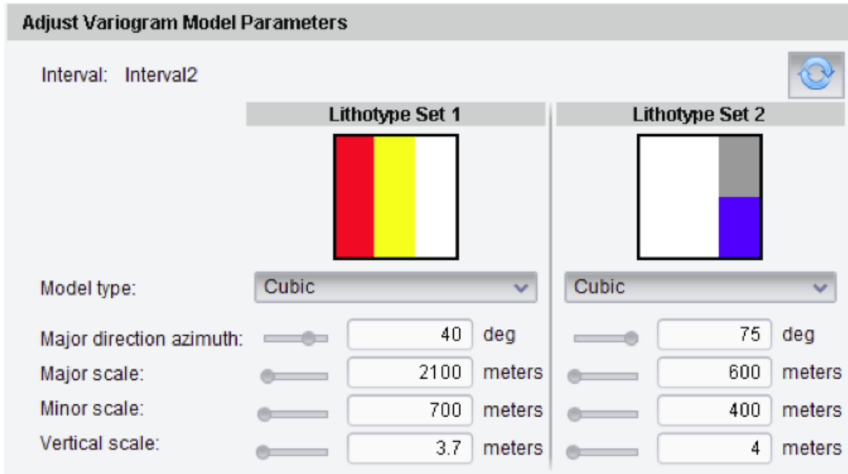
6. Add the **Eolian** as shown.

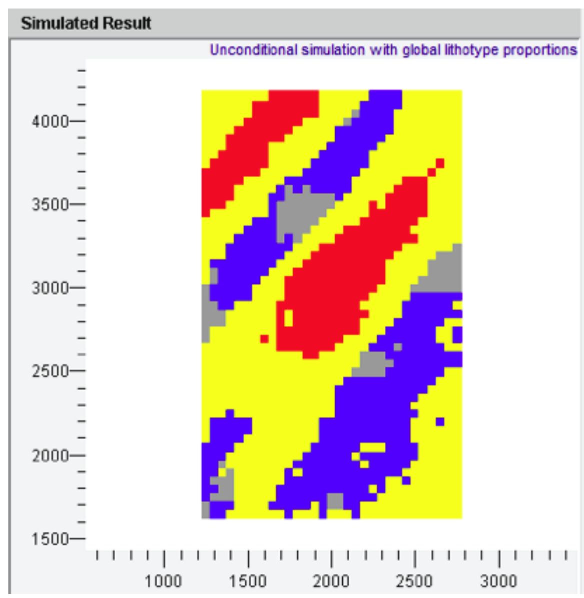
7. Change to **Set 2** and select the **Platform** and add it to the bottom, the Carbonate Mud is then added automatically.



8. Adjust the variogram model parameters as shown.

9. Check Results.

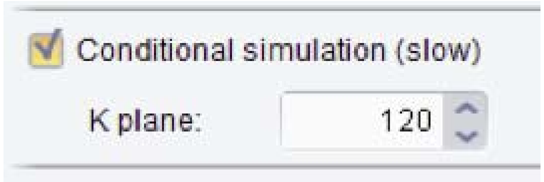


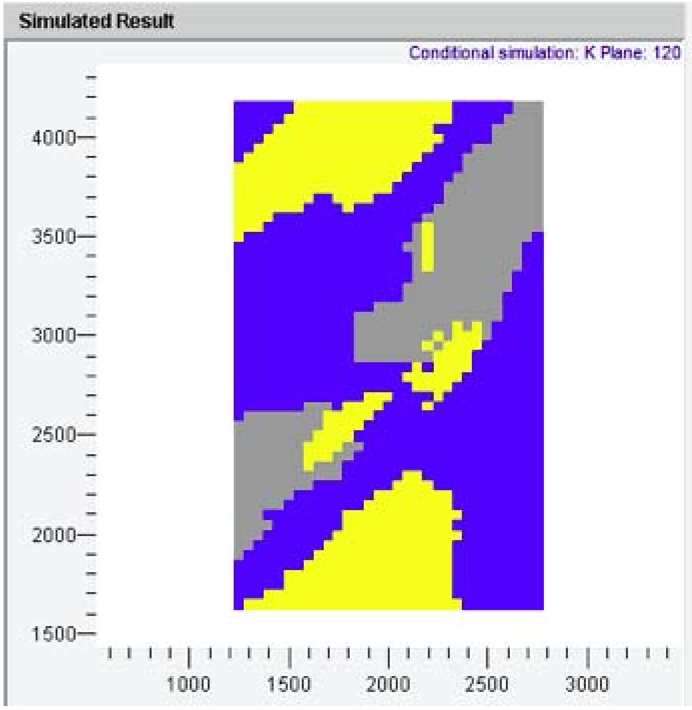


We observe in the unconditional simulation that the Thief Zone is inside the Eolian and not in contact with the other lithotypes. Eolian, Mud and Platform can be in contact. Because the fraction of mud is small (4.9%), it expresses the variogram scales, although the platform is controlled by the same variogram.

K-plane Condition Simulation

1. Select the **Conditional simulation (slow)** checkbox.
2. Check Result.

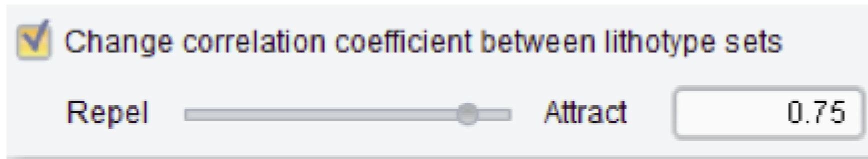


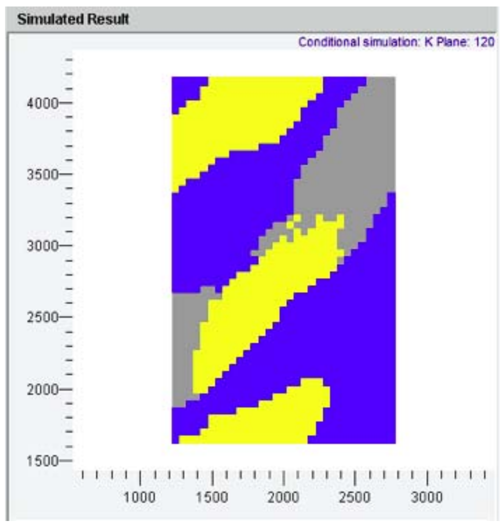


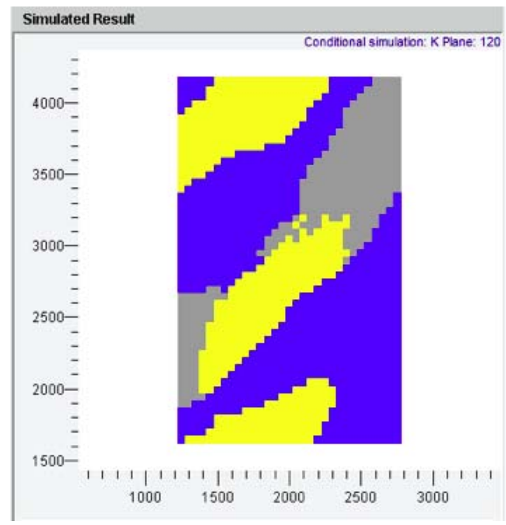
Conditional simulation of K-pane 120 with default correlation

3. Select the **Change Correlation** checkbox.

4. Move the slider to the right.







Correlation = 0.75 Correlation = -0.75

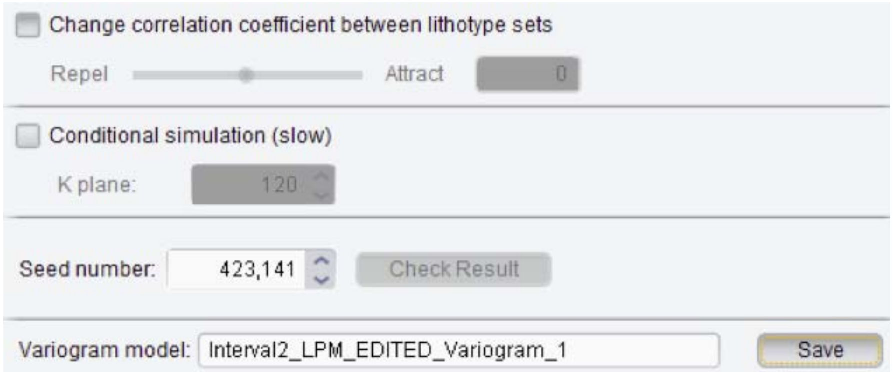
Changing the correlation between the variogram (independent when correlation = 0) changes the relationships between the facies transitions and provides more flexibility in the rule design.

5. Change the correlation = **0**.

6. Uncheck **conditional simulation**.

7. Check Result.

8. Save the variogram.

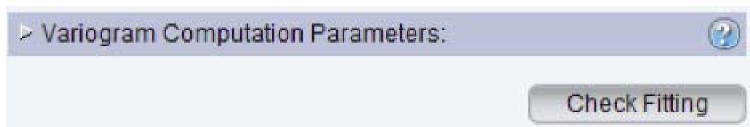


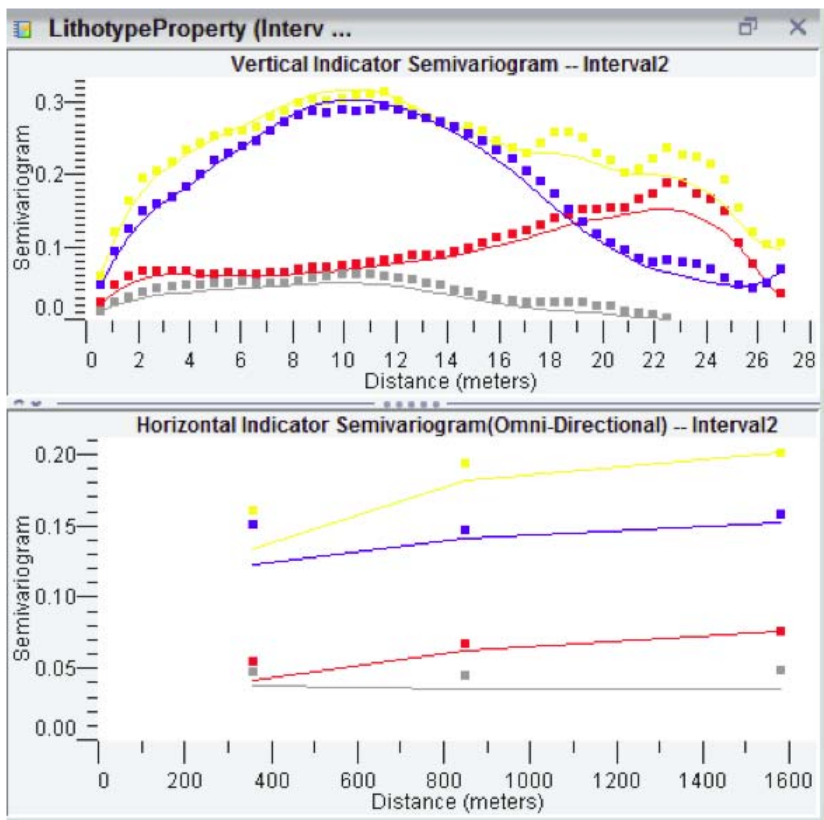
9. Close the dialog box.

Check Fitting

It is possible to check the variogram fitting to the experimental points based on the variogram parameters provided when designing the rules.

1. Click **Check Fitting** on the main Facies Modeling and Simulation task pane.

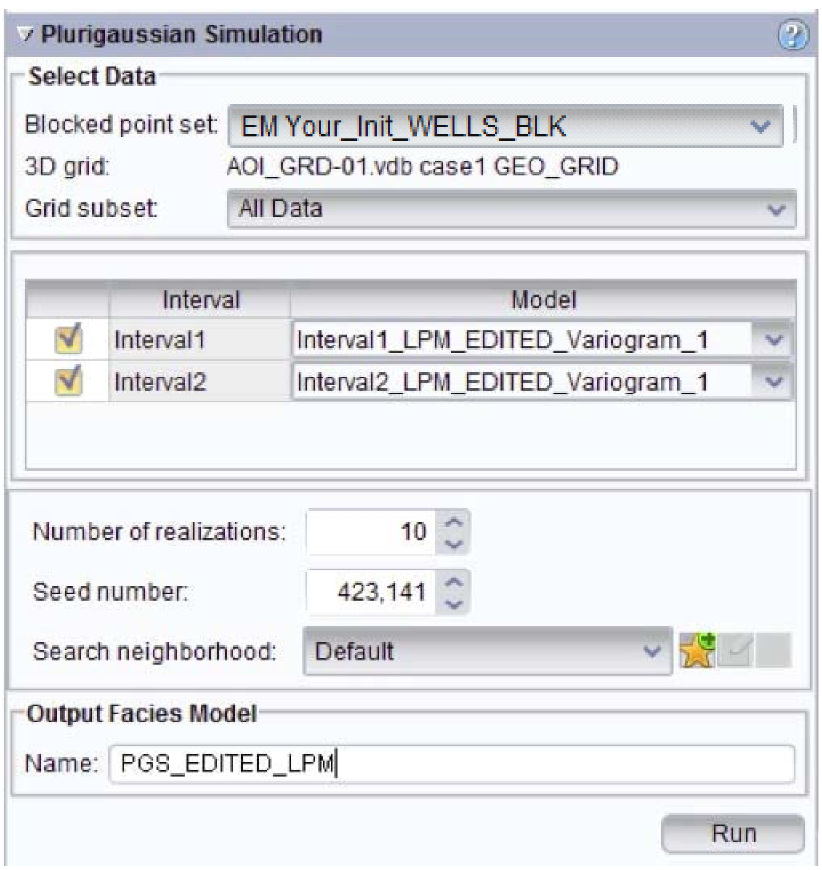


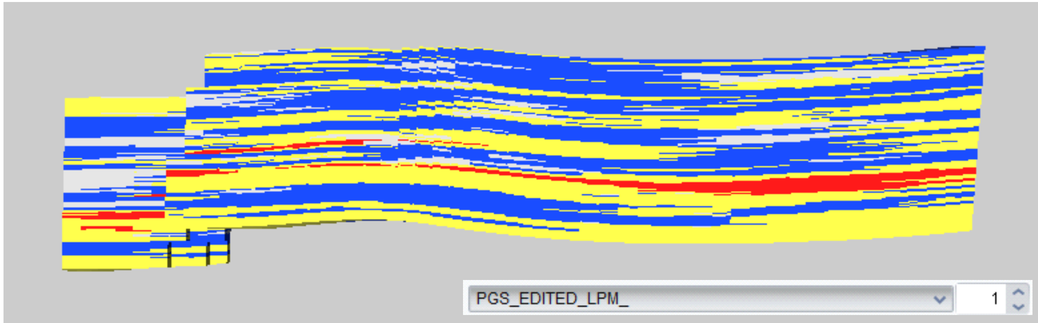


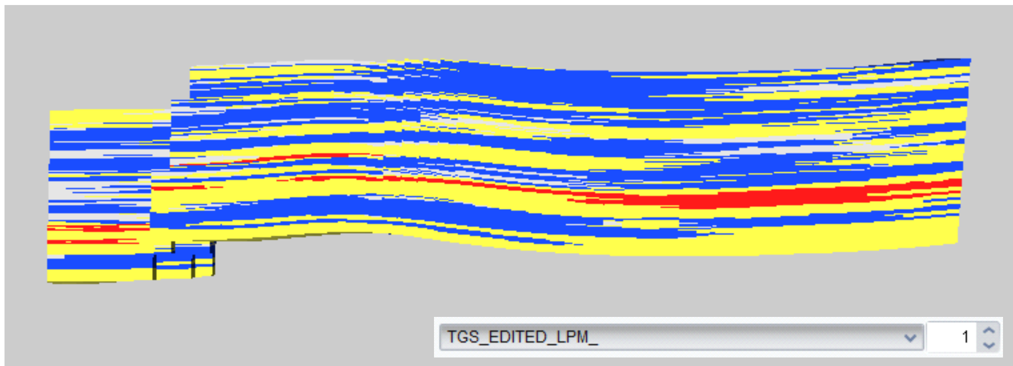
The parameters provided for the vertical (top) fit the experimental points very well, but the Omni-directional variogram is not a good fit because there are too few data points to compute the horizontal variogram. This will often be the situation.

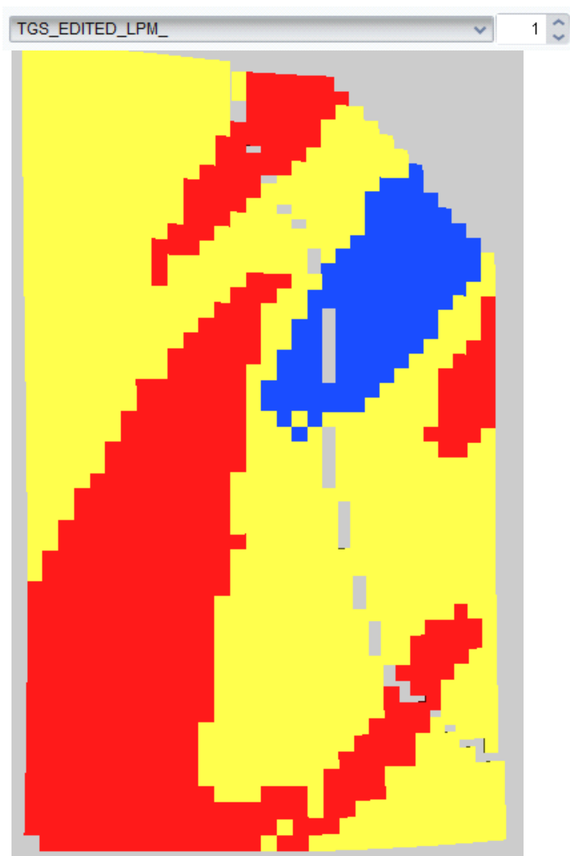
Run the Simulation

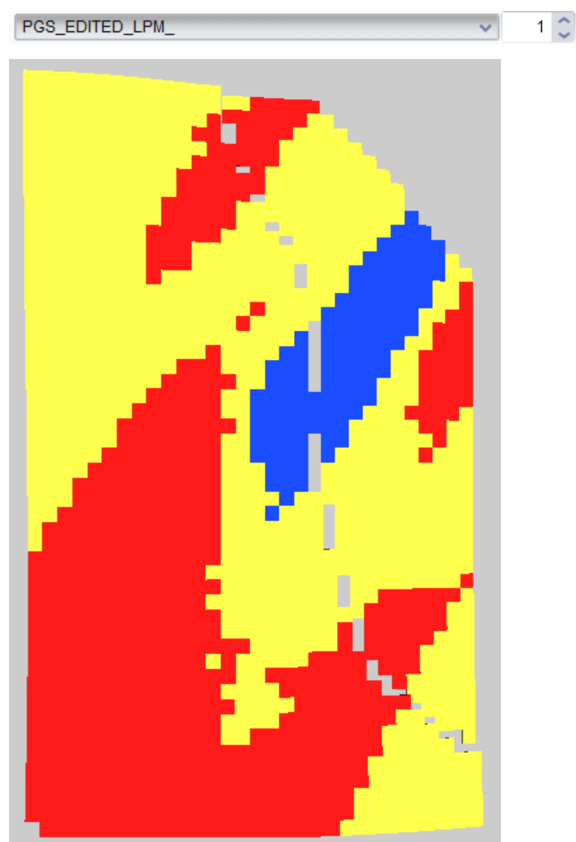
1. Expand the **Plurigaussian Simulation** subtask pane.
2. Select the point set, which selects the grid and variogram models last saved.
3. Number of realizations: **10**
4. Default Seed number and search neighborhood
5. Output model name: **PGS_EDITED_LPM**
6. **Run**.
7. Save session.
8. Make some displays as shown.

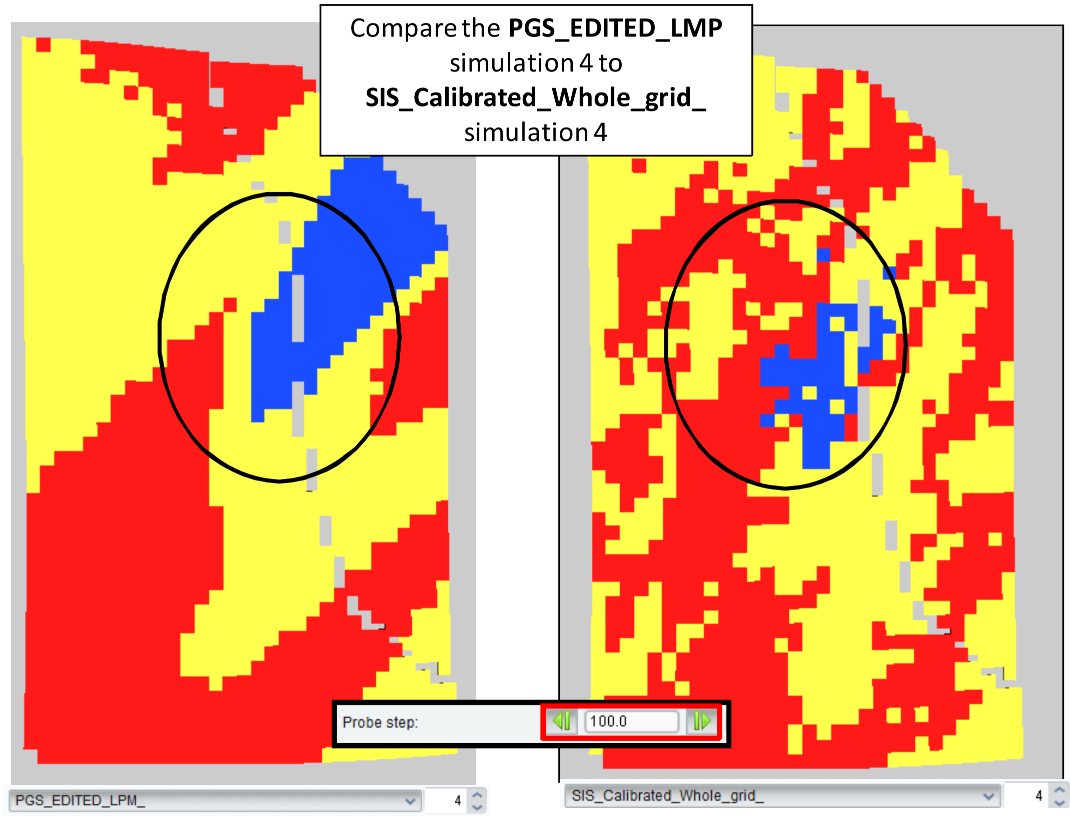


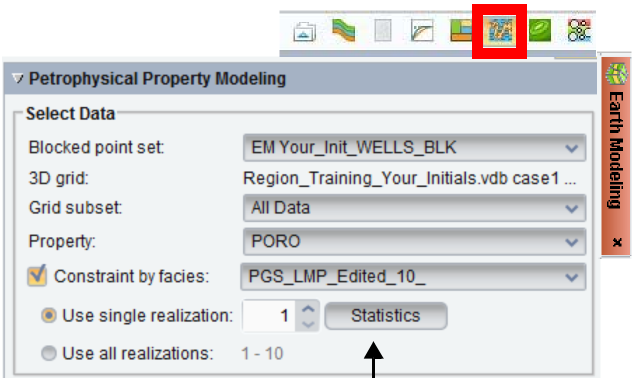

PGS_EDITED_LPM_
1


TGS_EDITED_LPM_
1

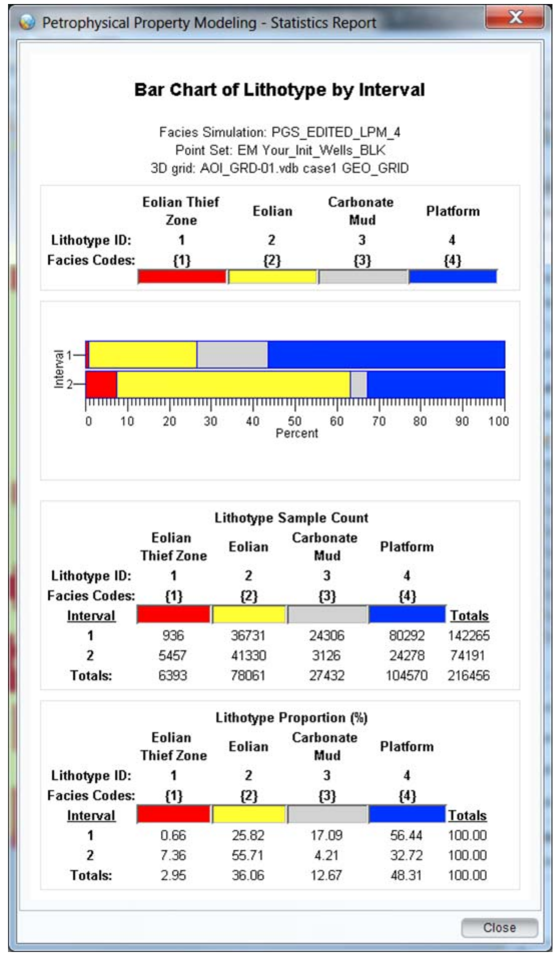

TGS_EDITED_LPM_
1


PGS_EDITED_LPM_
1


Compare the **PGS_EDITED_LMP** simulation 4 to **SIS_Calibrated_Whole_grid_** simulation 4
Probe step:
100.0
PGS_EDITED_LPM_
4
SIS_Calibrated_Whole_grid_
4



To view the statistics of your facies model, click the Statistics button.



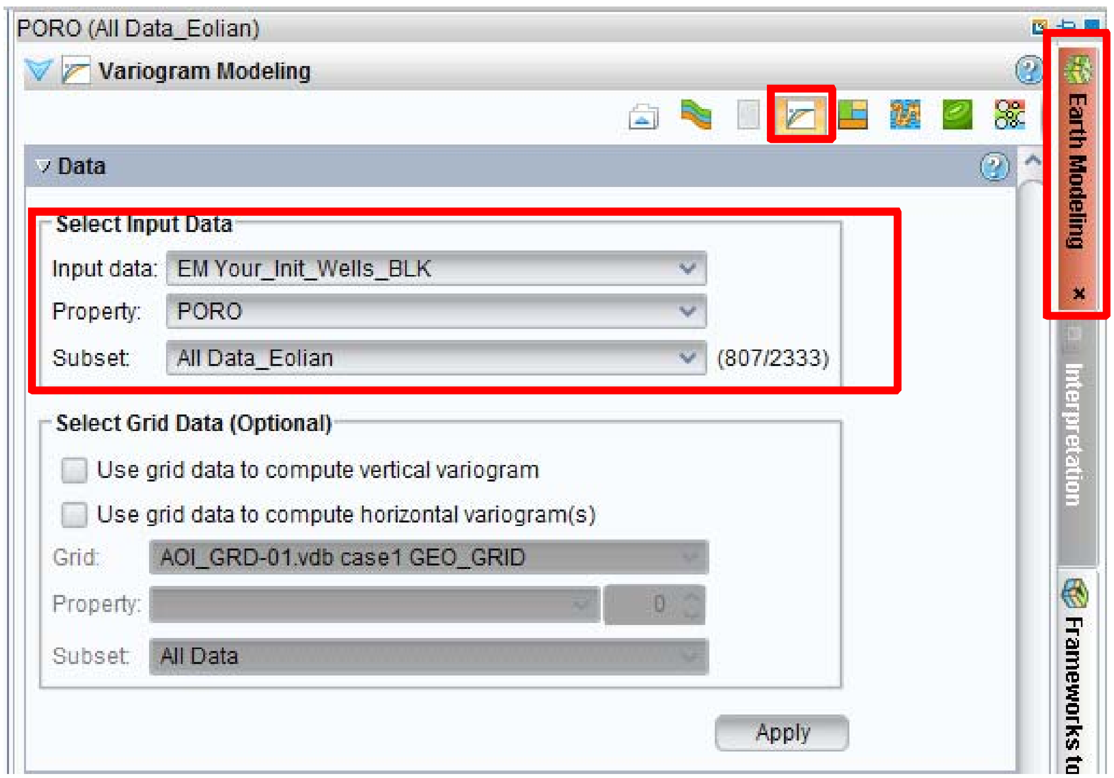
Variogram Modeling: Petrophysical Data

This exercise illustrates a common situation modelers face when computing and attempting to model variograms based on sparse well control. You will use all of the eolian data in the computation.

1. Click the **Variogram Modeling** icon.

2. Enter the input data as shown.

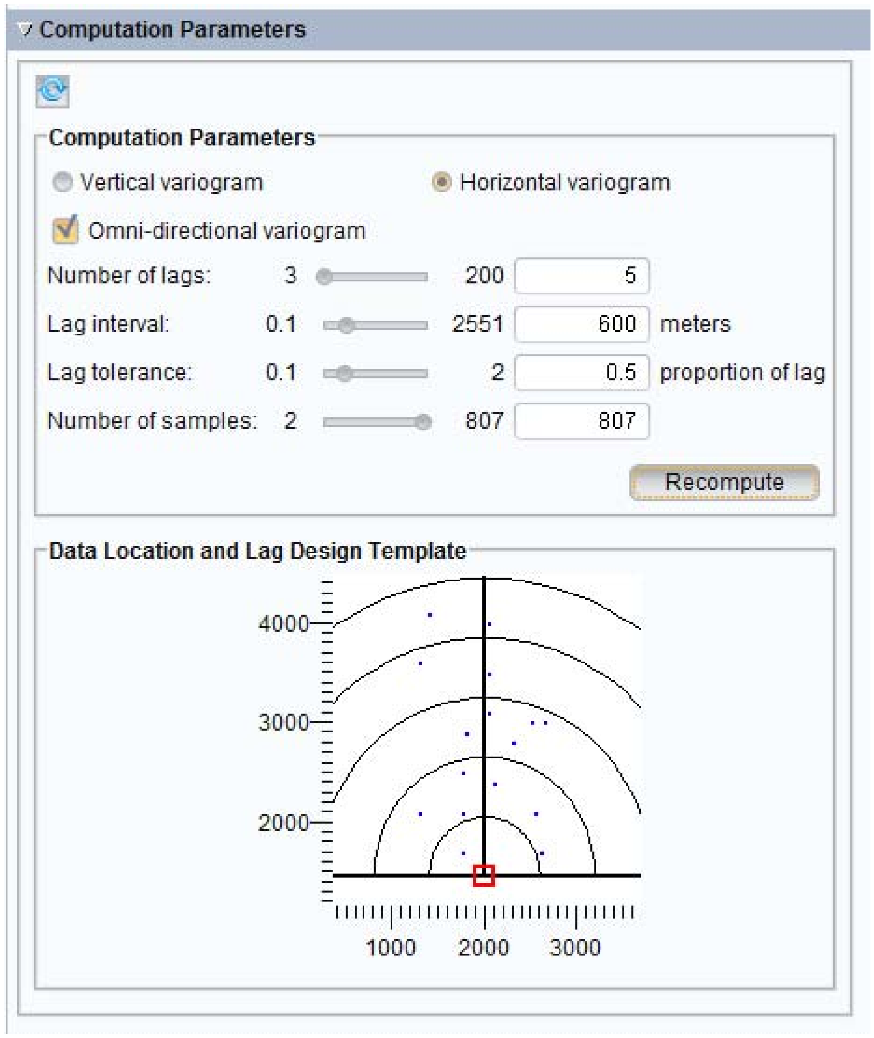
3. Click **Apply**.



4. In the Computational Parameters panel check ON **Omni-directional variogram**.

5. Enter the parameters as shown.

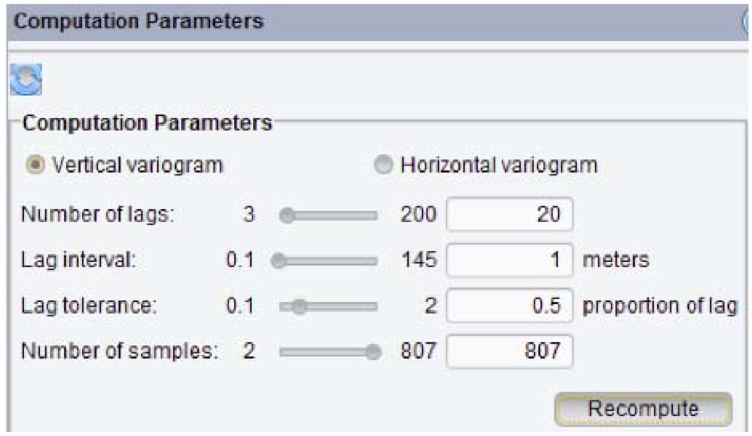
6. Click **Recompute**.

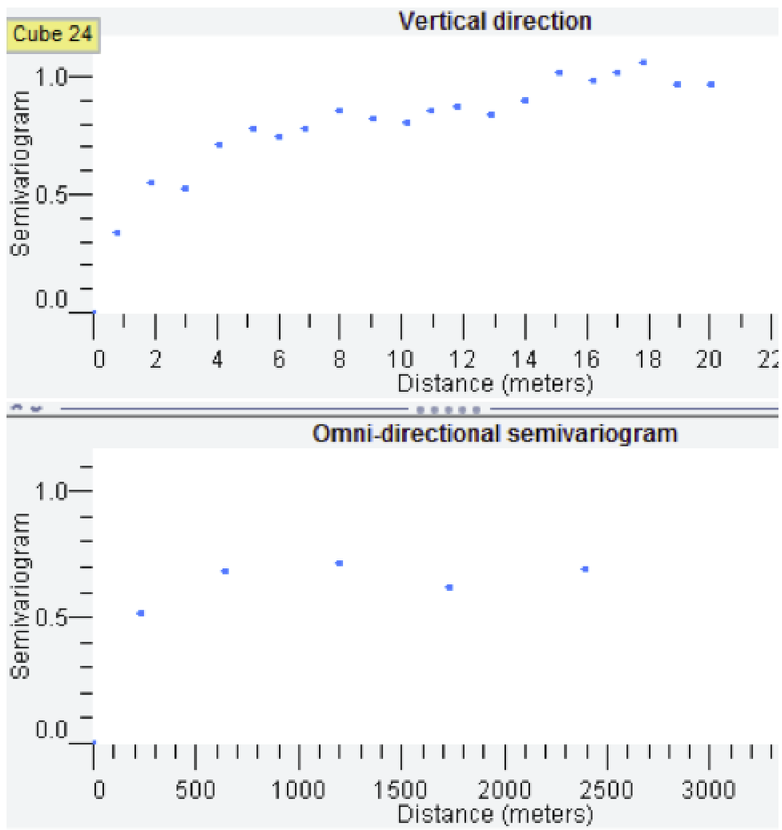


7. Click the **Vertical variogram** option.

8. Adjust the **Number of lags** to an appropriate number as shown in the following image.

9. Click **Recompute**.



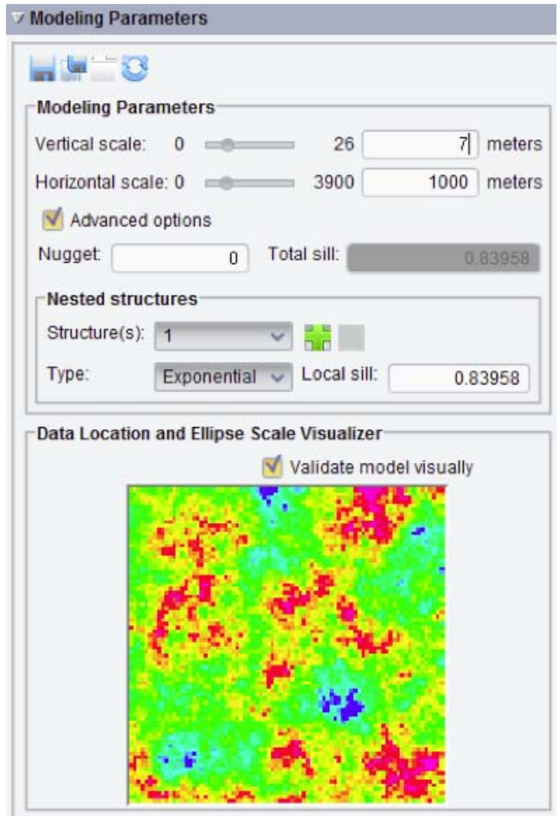


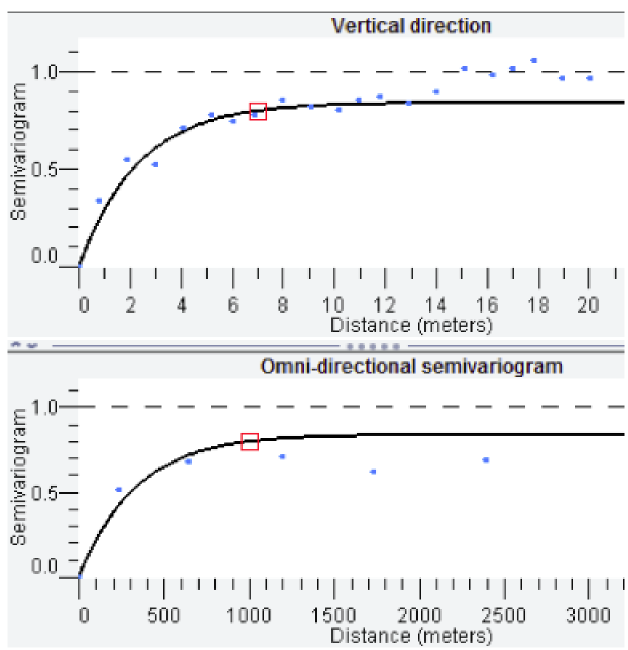
10. Click the **Modeling Parameters** subtask pane.

11. Check ON the **Validate model visually** and the **Advanced options** checkboxes.

12. Change the structure type to **Exponential**.

13. Adjust the vertical and horizontal scales as shown.





This is probably about the best model you could create with the limited data.

You could try other lithotypes but you won’t do much better, so we’ll need to use some ratio of the facies variograms for the horizontal scales when modeling the petrophysical data.

There is no need to save any models here as you will be able to create them interactively in the Petrophysical Modeling task pane, which is the next step.

Petrophysical Property Modeling

Simulate Porosity (Single Realization)

For this exercise you will use facies realization 4 from the PGS_EDITED_LPM facies simulation exercise.

1. Click the **Petrophysical Property Modeling** icon. 

2. Select your blocked pointset. The associated Grid is then automatically entered.

3. Select grid subset as **All Data**.

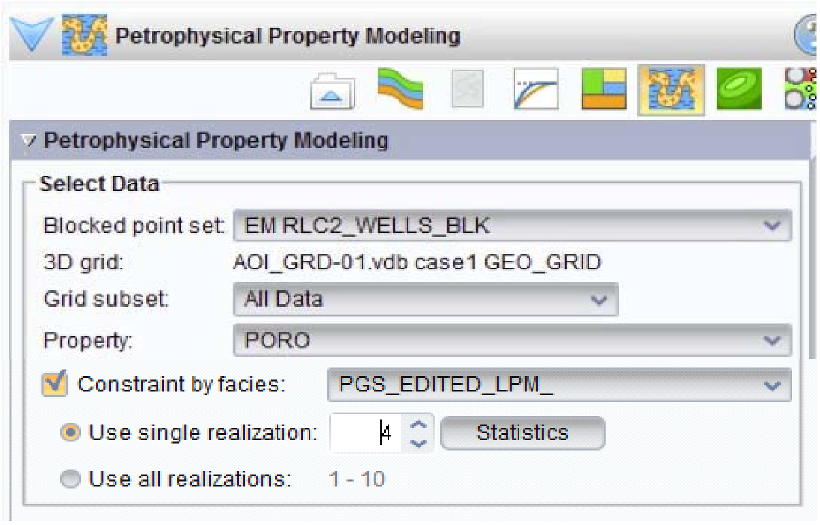
4. For Property, select **PORO**.

5. Use the lithotypes to constrain the modeling, so click the **Constraint by facies** checkbox to select it.

6. For Facies model, select **PGS_Edited_LPM**.

7. Select **Use a single realization**. This is the default for the older versions.

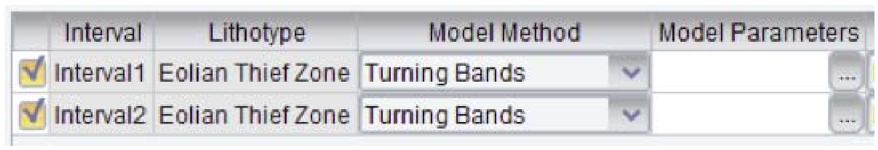
8. Select realization number **4**.



9. In the Modeling parameters, click the **Create New Model** icon.

10. Under Modeling Parameters, select the Lithotype: Eolian Thief Zone.

11. Click the **ellipsis button** under Model Parameters.



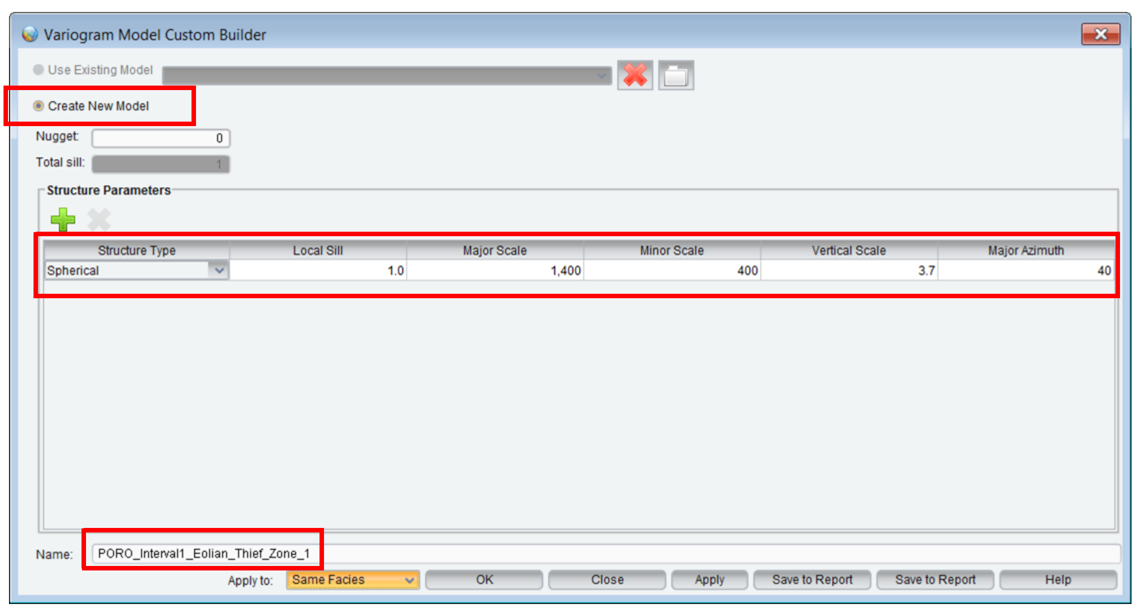
12. In the Variogram Model Custom Builder toggle on the **Create New Model** option.

13. Enter the Structure Parameters as shown.

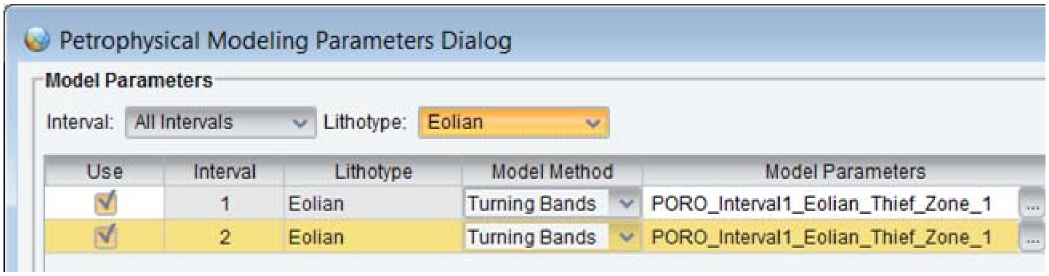
14. Name the model PORO_Eolian Interval1_Thief_Zone_1.

15. Apply to **Same Facies**.

16. Click **OK**.



17. Change the Lithotype to **Eolian** and use the existing **PORO_Eolian Interval1_Thief_Zone_1** variogram.



18. Apply to **Same Facies**.

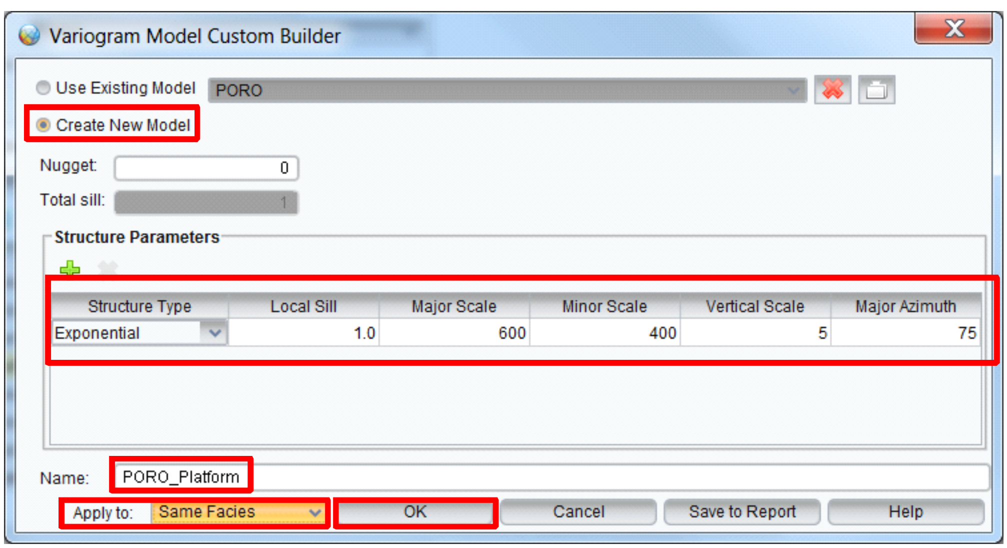
19. Change the Lithotype to **Platform**.

20. Toggle on the **Create New Model** option.

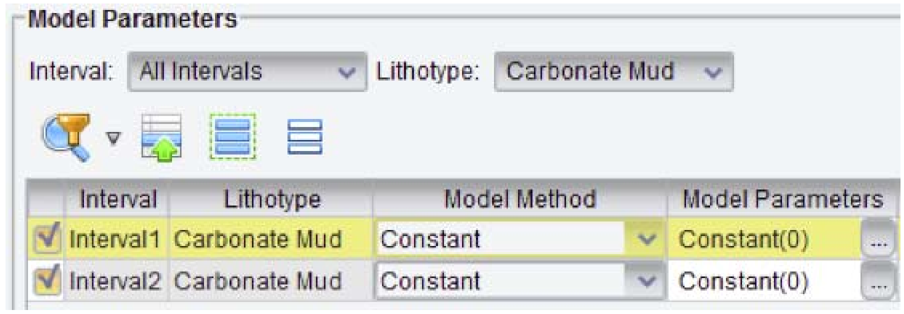
21. Enter the Structure Parameters as shown.

22. Name the model PORO_Platform.

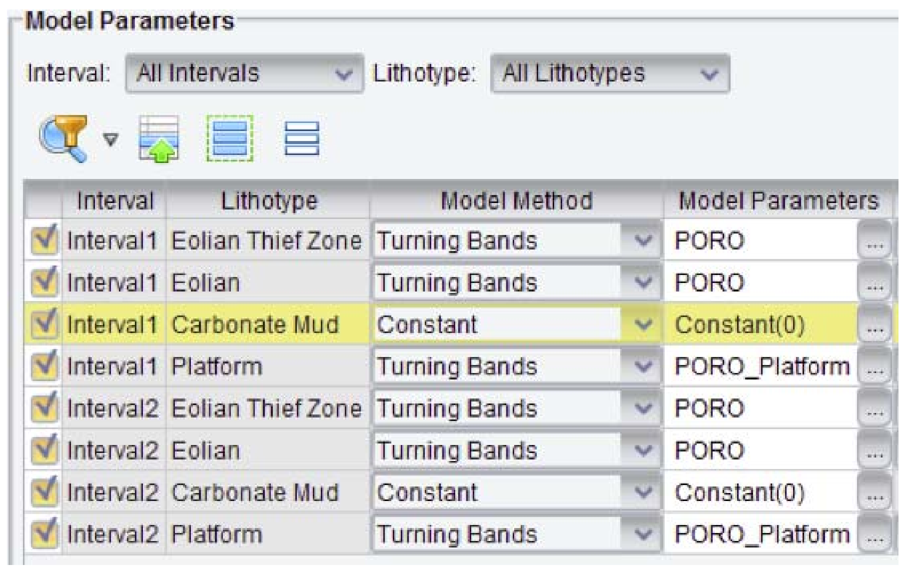
23. Apply to **Same Facies** and click **OK**.



24. For the Carbonate Mud lithotype enter a constant of **0**.



25. Set the Lithotype to **All Lithotypes** to expand the table.

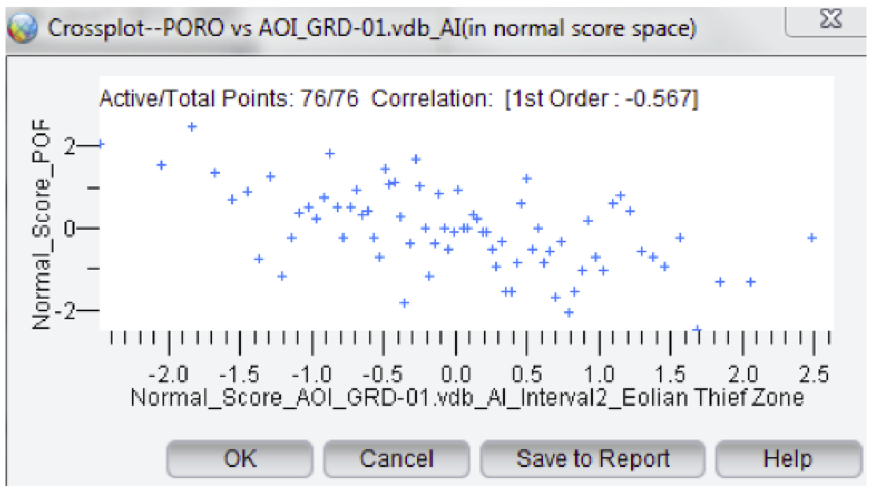


Collocated Cosimulation

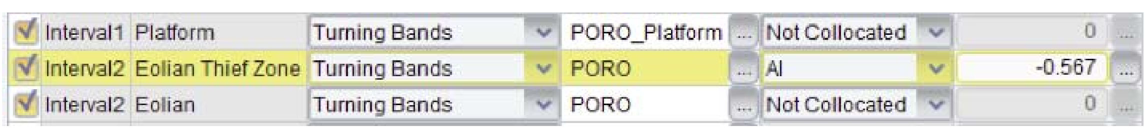
For lithotypes which have a reasonably strong Acoustic Impedance correlation we can use the **collocated option**.

1. For Interval 2 Thief Zone choose **AI** for the Collocated property.

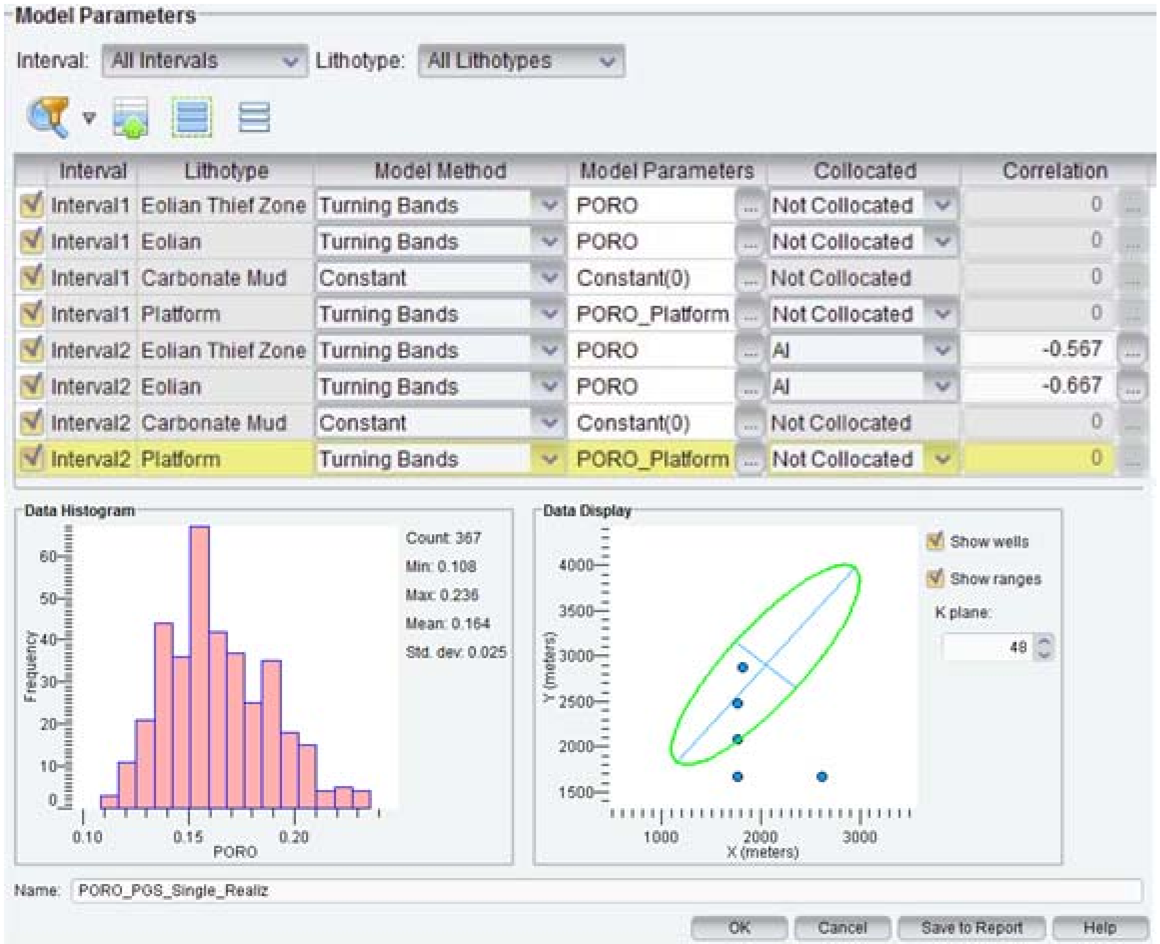
2. Click the ellipsis next to the Correlation to show the cross-plot. Although the correlation is only about -0.57, we will use the AI to assist in the modeling of porosity. Because we will run conditional simulation the data were transformed automatically and the AI property is added to the Blocked point set.



3. Click **OK** on the cross-plot to add the correlation coefficient.



4. Repeat steps 1-3 for Interval 2 Eolian.



5. Type an output name PORO_PGS_LPM_4.

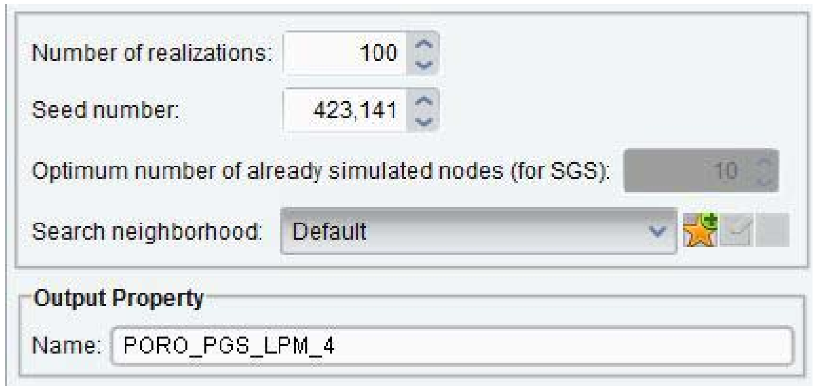
6. Adjust the Number of realizations to **100**.

7. Use the Default **Seed** and **Neighborhood**.

8. The Output name will be PORO_PGS_LPM_4.

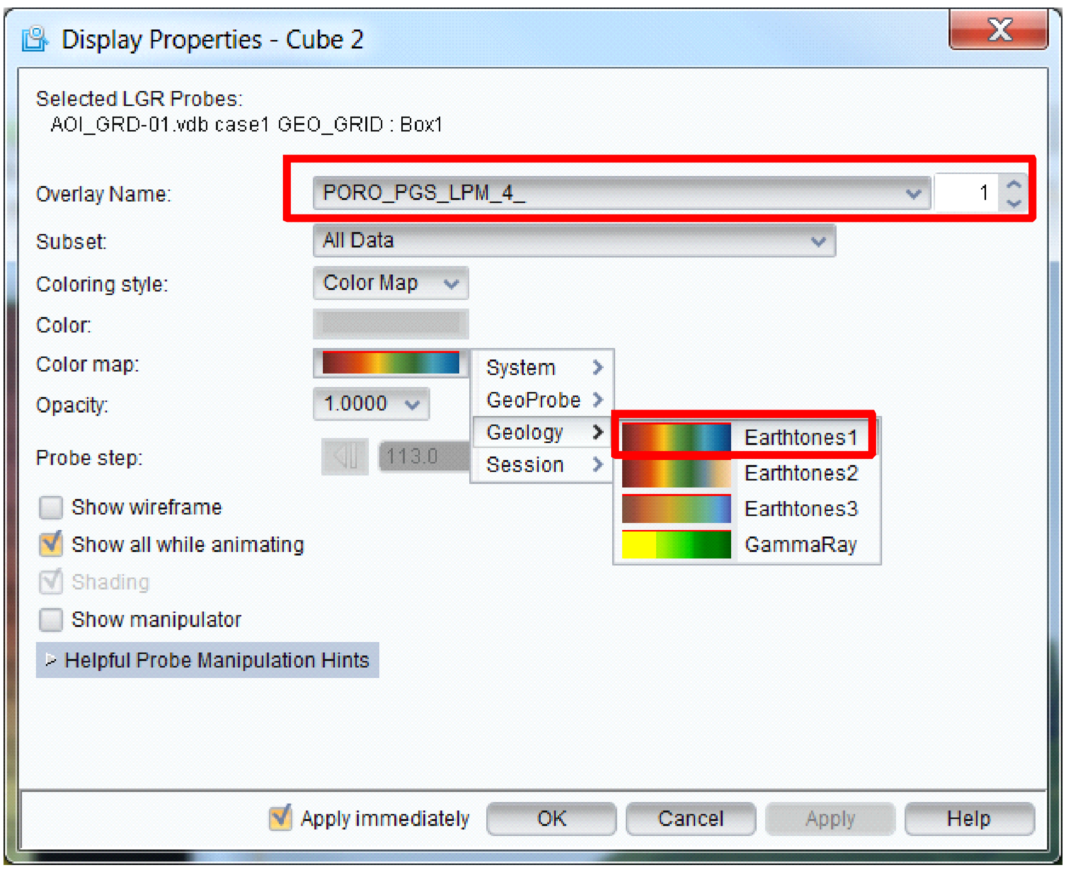
9. Click **Run**.

10. Save the session.



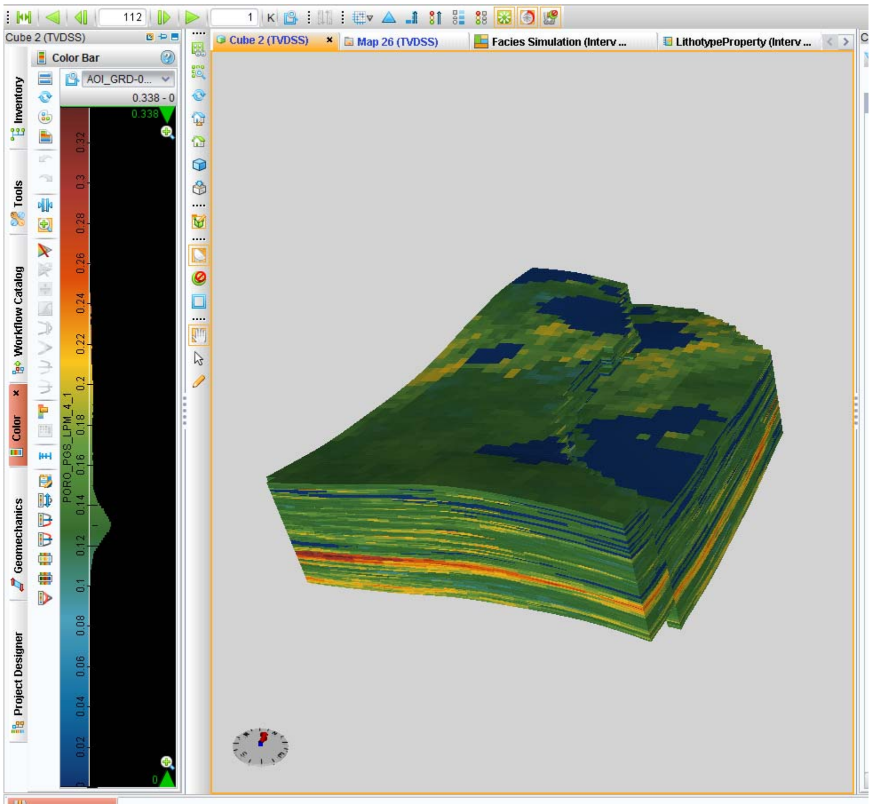
Display the PORO realizations

1. Activate your Cube View.
2. Click MB3 on the 3D grid probe in your inventory and select **Display Properties**.
3. Change the overlay to **PORO_PGS_LPM_4**.
4. Select **Earthtones1** for Color map.
5. Click **OK**.



Your cube view should look similar to the following image.

Use the Display Property Editor Spinner to scroll through the realizations.

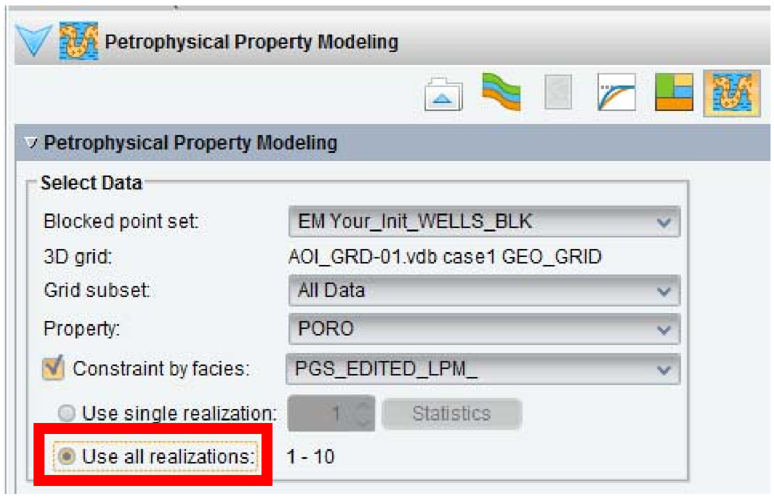


Simulate Porosity (Using All Realization)

1. Go back to Petrophysical Property Modeling and select **PORO**.

2. Constraint by facies: **PGS_EDITED_LPM**.

3. Select **Use all realizations**. For each facies simulated the software will simulate one porosity simulation.



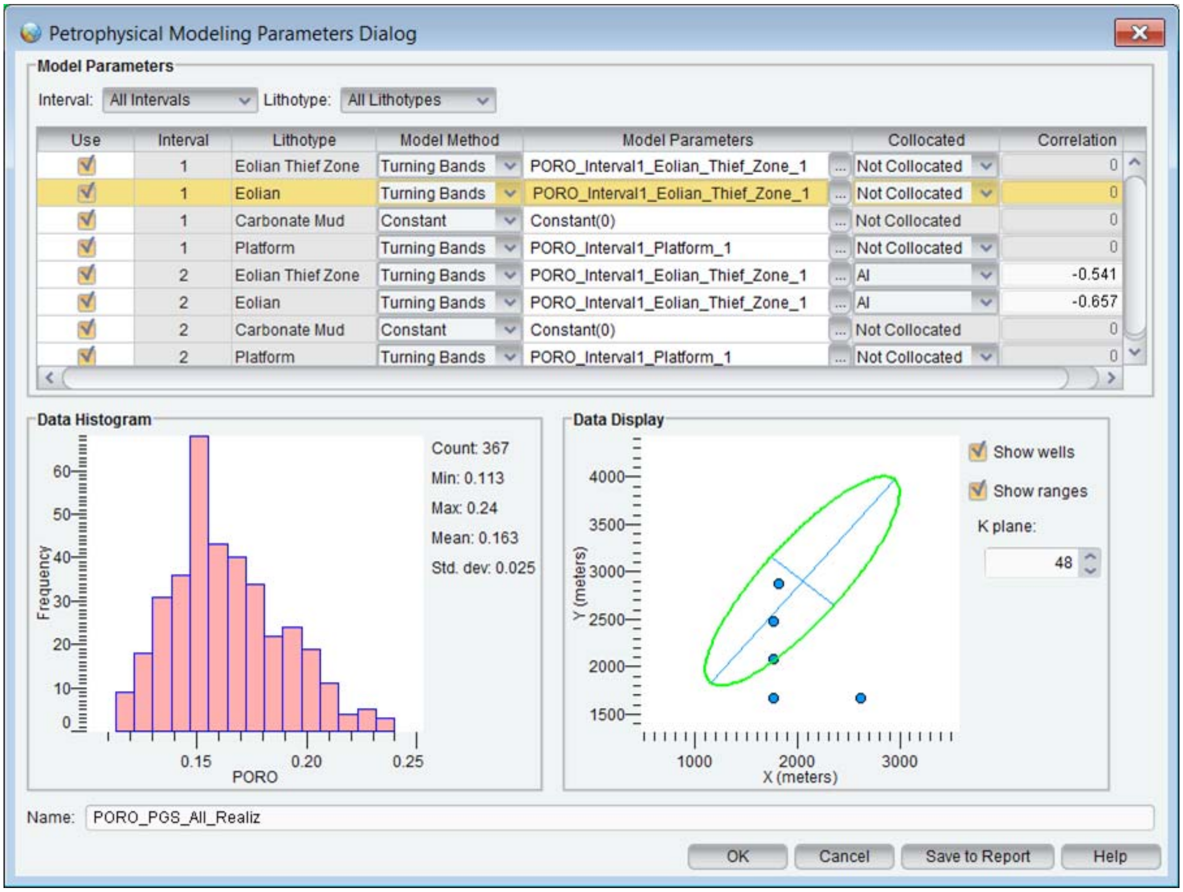
4. In the Modeling parameters, click the **Create New Model** icon .

   A Petrophysical Modeling Parameters dialog box displays.

5. Set the Model Method and model Parameters with the same parameters that you used with single realization, using **Use Existing Model** and remember do the **Collocated** in Eolian Thief Zone and **Eolian** in Interval 2.

6. Give a name PORO_PGS_All_Realiz.

7. Click **OK**.



8. Set the number of realizations to **25**.

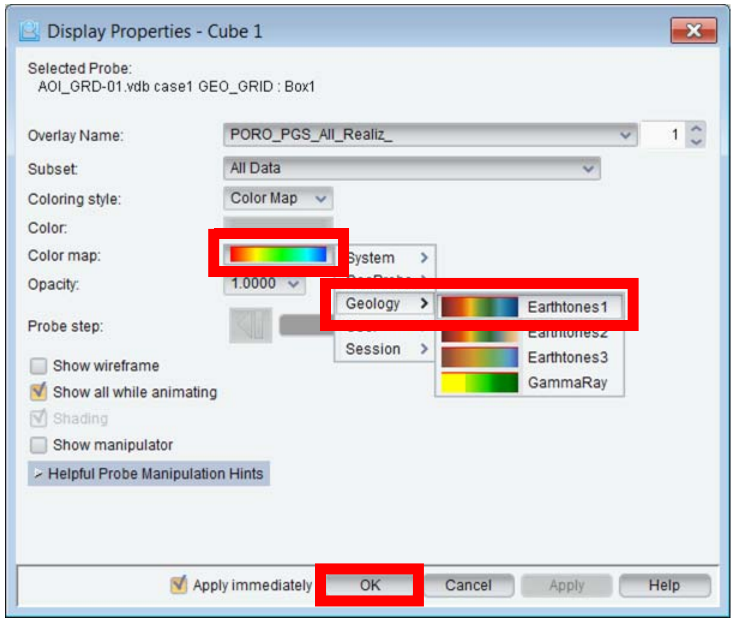
9. Leave the rest of the parameters as default.

10. Leave Output Property Name as PORO_PGS_All_Realiz.

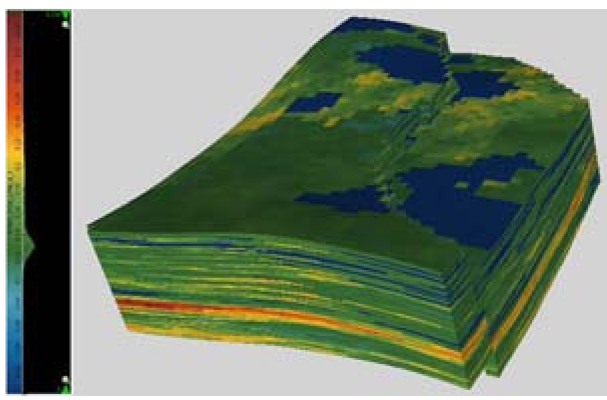
11. Click **Run**.

12. To view the output property on the grid, display the grid in the Cube view using box probe, open the Display Properties dialog box for the probes, change the Overlay Property to **PORO_PGS_All_Realiz**, and use **realization 1**. Change the Color map to **Earthtones1**.

You can run realizations using other parameters to compare the results if you have time.



Your cube view should look similar to the following image.

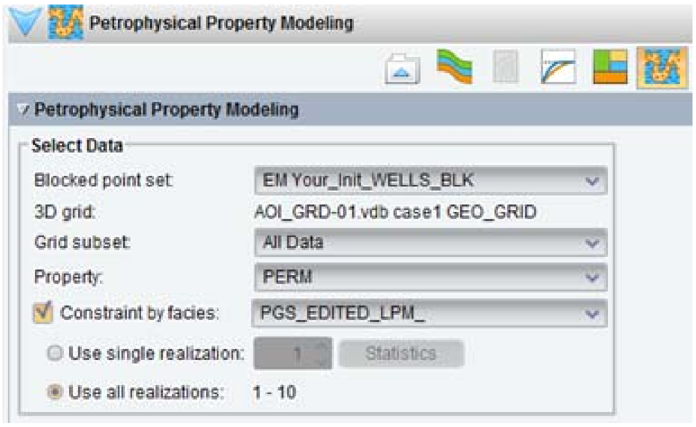


Simulate Permeability

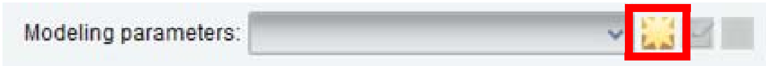
Now go back to Petrophysical Property Modeling and model the permeability data.

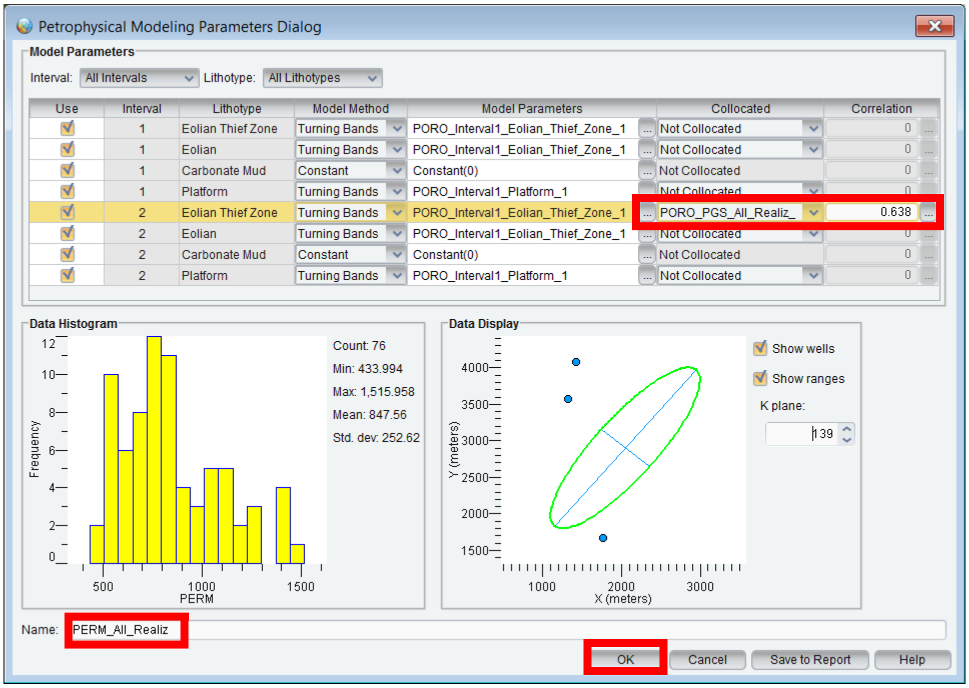
1. In the Petrophysical Property Modeling panel change the property to **PERM**.

2. Use **Constraint by facies** and **Use all realizations**.



3. Create New Model in Modeling parameters and then enter the model methods and parameters as you used in simulation of Porosity.





The only good correlation between porosity and permeability is for the Eolian Thief Zone in Interval 2.

4. Change the Output Name to PERM_All_Realiz and then click **OK**.

5. Notice that number of realization field is grayed out and 25 is displayed. This is equal to PORO realization which we created in previous steps. Since we are collocating PERM simulation with PORO so the software will automatically select equal number of realization for the collocated property and user can't change it.

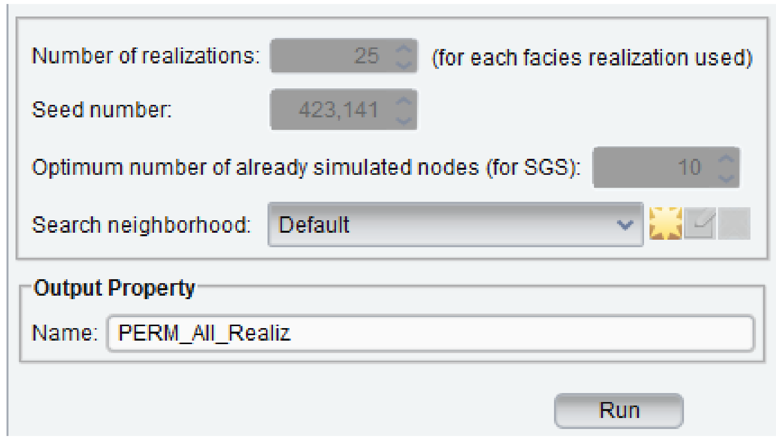
6. Seed number should be 423,141.

7. Search neighborhood default.

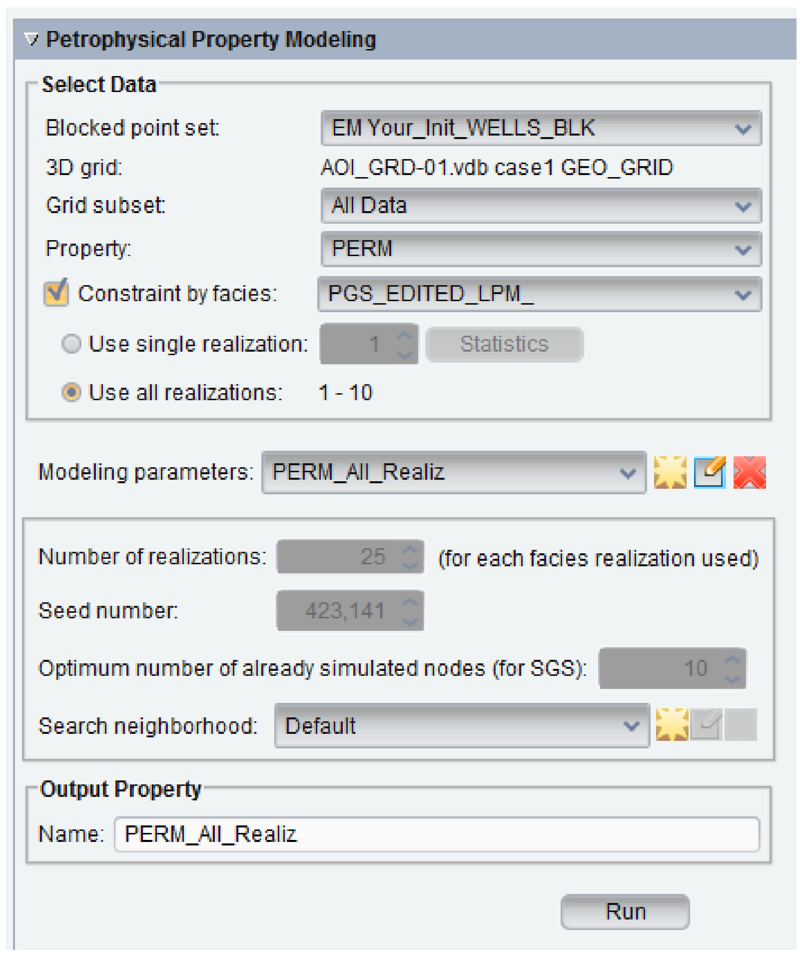
8. Optimum number of already simulated nodes (for SGS): leave default 10.

9. Output Property name: PERM_All_Realiz.

10. Click **Run**.

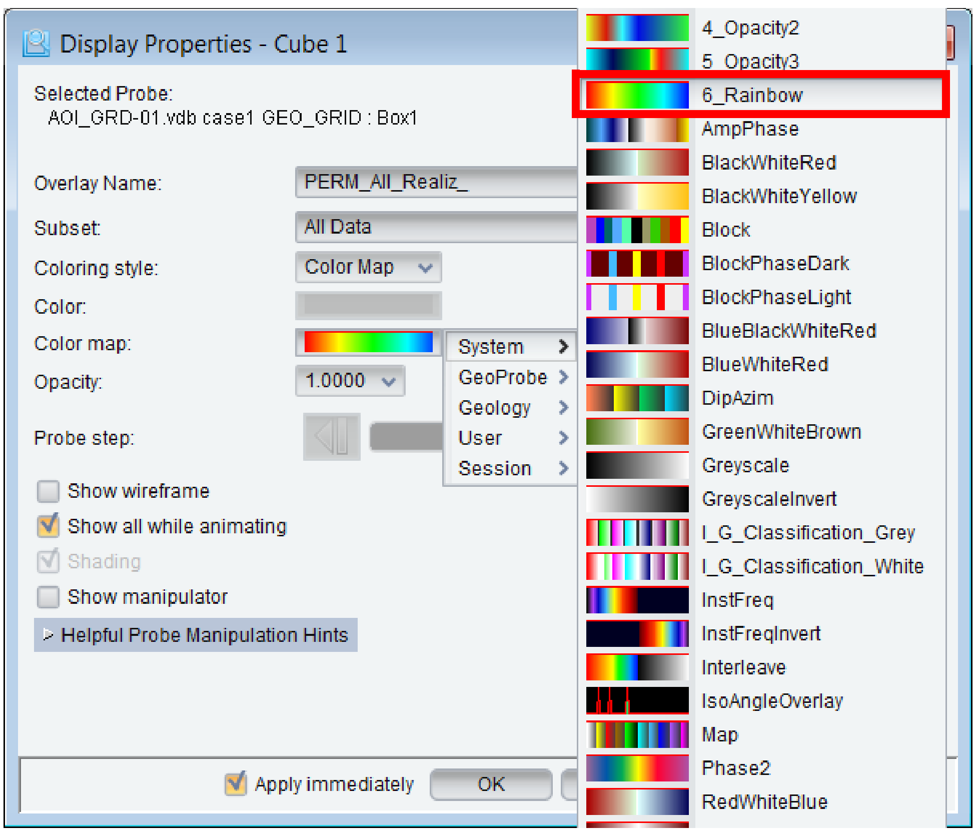


11. Save the session.

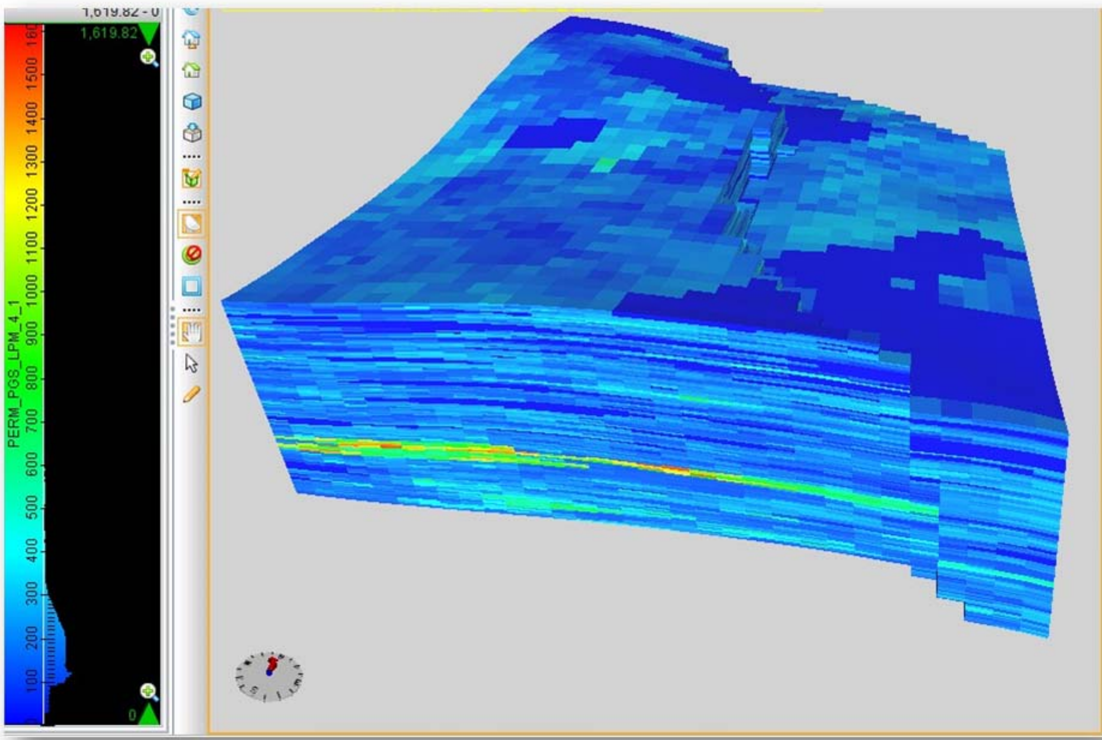


Display the PERM Realizations

1. Activate your Cube View.
2. Click MB3 on the 3D grid probe in your inventory and select **Display Properties**.
3. Change the overlay to **PERM_All_Realiz**.
4. Select **6_Rainbow** for the Color map.
5. Click **OK**.



Your cube view should look similar to the following image.



Create a K-Plane Probe

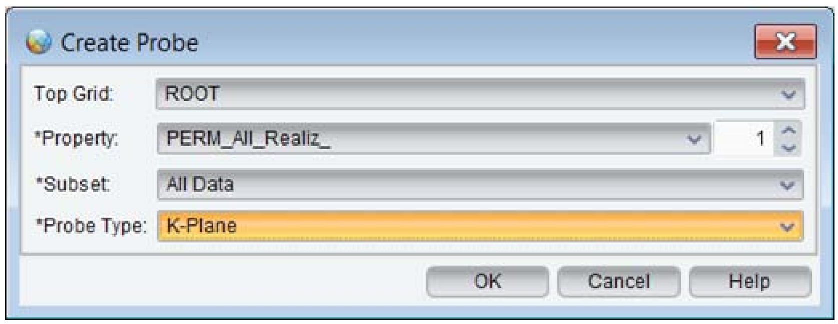
1. Click MB3 on the 3D Grid in the inventory and select **New Probe**.

2. Change the property to **PERM_All_Realiz**.

3. Select **All Data** for Subset.

4. Change Probe type to **K-Plane**.

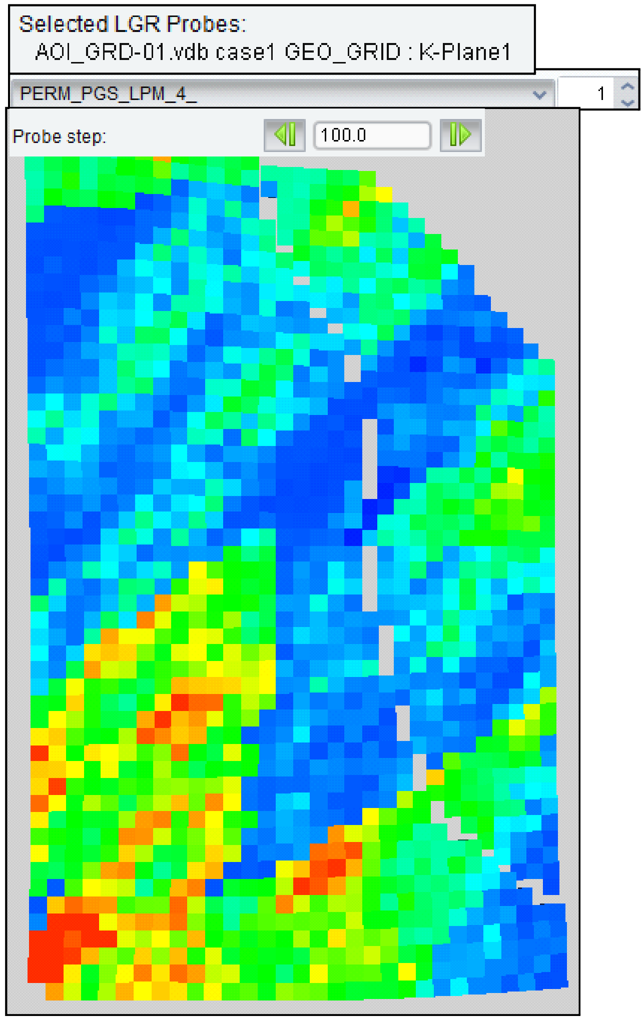
5. Click **OK**.



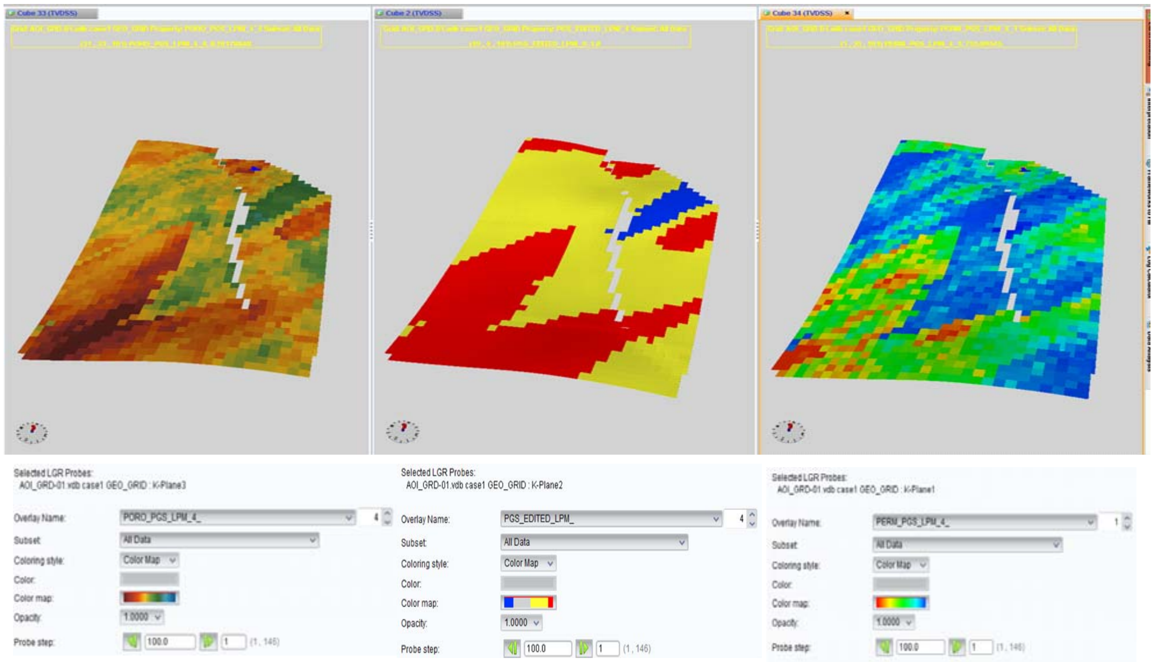
6. Click MB3 on the new K-Plane Probe in the inventory and select **Display Properties**.

7. Change the Color map to **6_Rainbow** and Probe Step to **100**.

Your display should resemble the following image.



By creating multiple probes and multiple cube views you can display your different simulations side by side.



Volumetrics: Leverett J-Function Method

In Post Processing, hydrocarbon resources can be calculated using three different methods:

- Leverett J-functions: This a uses capillary-pressure method to compute water saturation, Sw, and requires porosity and permeability properties on the 3D grid.

- Saturation-Height: This methods requires knowledge of the percent Sw as a function of the height above the fluid contact. The information can be imported via an ASCII flat file with a required format, or can be entered manually through the Design Wizard.

- User Specified: This method requires filling in a table of Sw by interval and lithology.

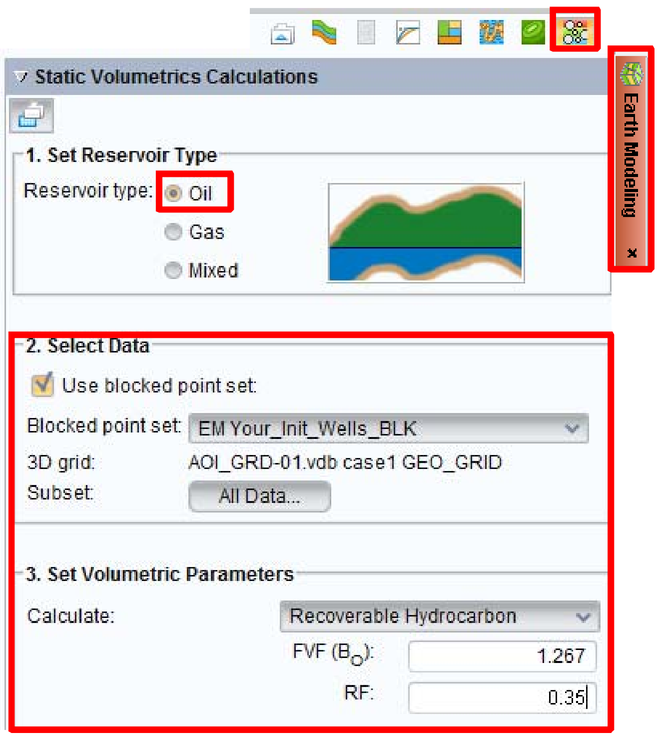
This exercise illustrates how to use the porosity and permeability realizations created in a previous exercise to compute Sw using the capillary-pressure method.

Input Parameters

1. Click the **Post Processing** icon. 

2. In the Static Volumetric Calculations subtask pane set reservoir type to **Oil**.

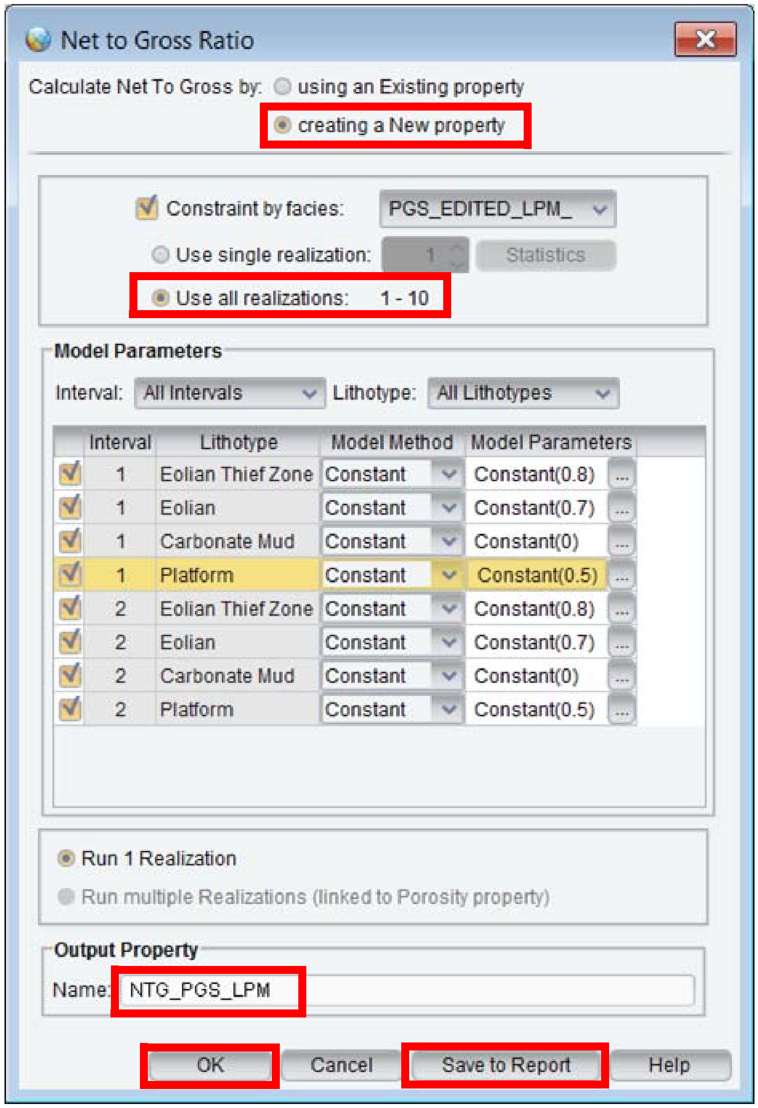
3. Enter the Select Data options and Volumetric Parameters as shown.



Specify Net-to Gross (NTG) Parameters

1. Click **<select method>** for Net to gross ratio.
2. Calculate Net to gross by creating a new property.
3. Check on the **Constraint by facies** checkbox.
4. Select the **PGS_Edited_LPM** facies model and use **all realizations**.
5. Enter **Constant** model method values as shown for each lithotype.
6. Name the output property **NTG_PGS_LPM**.
7. Click **Save to Report**.

8. OK.

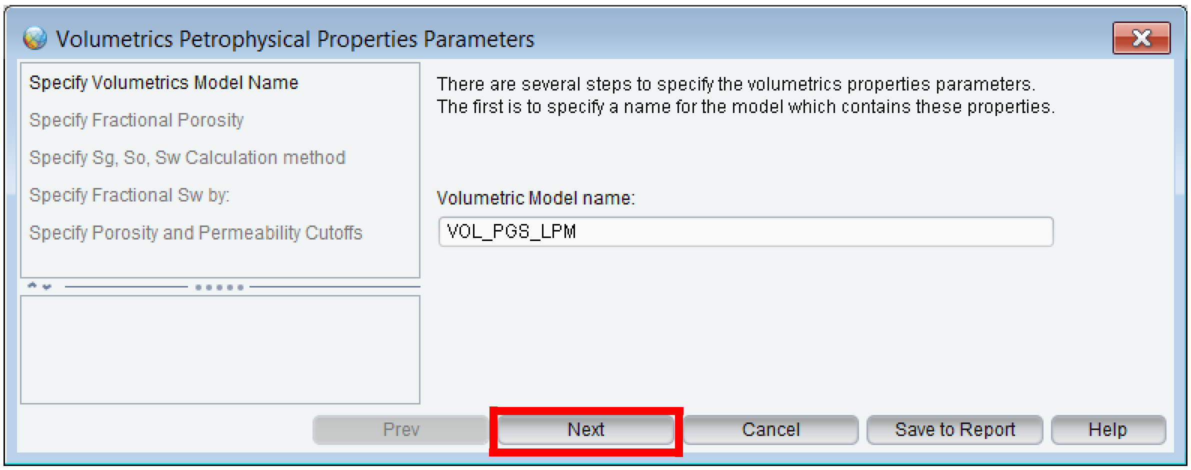

Net to Gross Ratio
Calculate Net To Gross by:
using an Existing property
creating a New property
Constraint by facies:
PGS_EDITED_LPM_
Use single realization:
Statistics
Use all realizations: 1 - 10
Model Parameters
Interval: All Intervals
Lithotype: All Lithotypes
Interval
Lithotype
Model Method
Model Parameters
1 Eolian Thief Zone Constant Constant(0.8)
1 Eolian Constant Constant(0.7)
1 Carbonate Mud Constant Constant(0)
1 Platform Constant Constant(0.5)
2 Eolian Thief Zone Constant Constant(0.8)
2 Eolian Constant Constant(0.7)
2 Carbonate Mud Constant Constant(0)
2 Platform Constant Constant(0.5)
Run 1 Realization
Run multiple Realizations (linked to Porosity property)
Output Property
Name: NTG_PGS_LPM
OK
Cancel
Save to Report
Help

Specify Petrophysical Parameters

1. Click the Petrophysical Parameters: **create new model** icon.

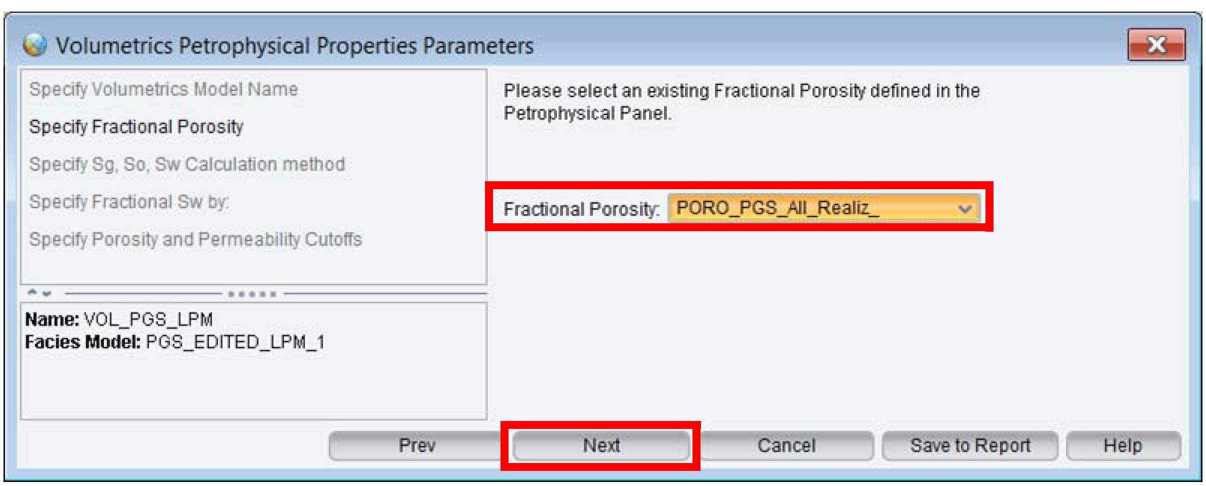
2. Enter VOL_PGS_LPM for the Volumetric Model Name.

3. Click **Next**.

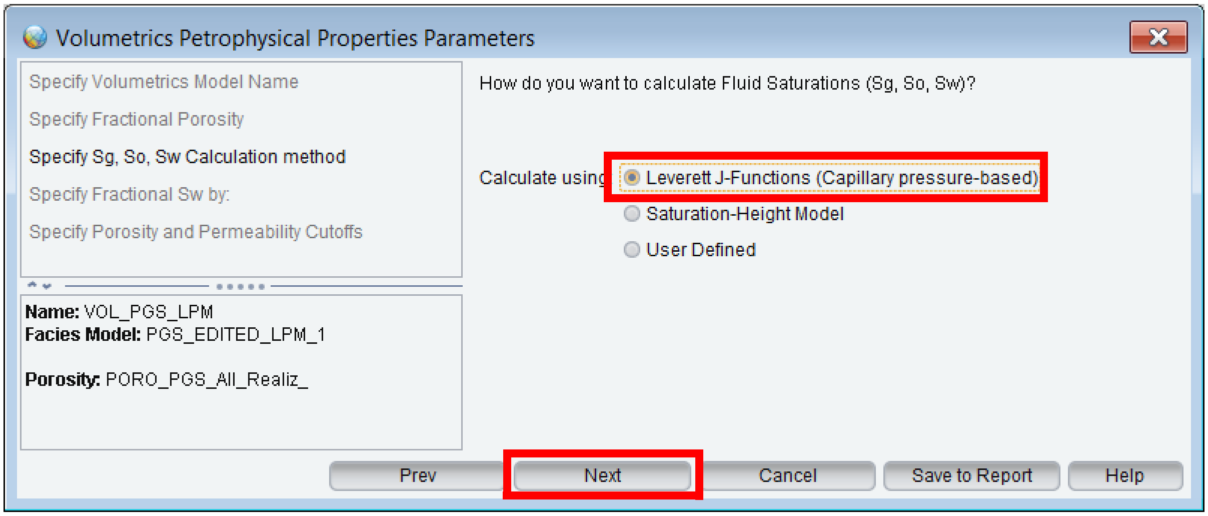


4. Select **PORO_PGS_All_Realiz** for the fractional porosity input.

5. Click **Next**.

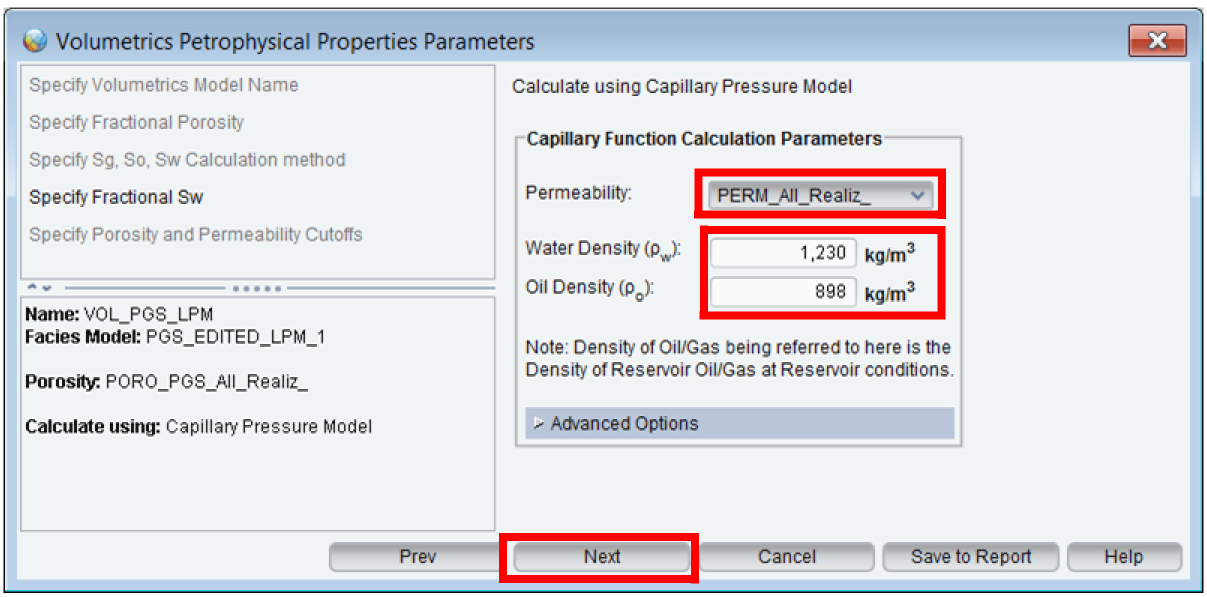


6. Select **Leverett J-Functions** for the method of calculating Fluid Saturations, and click **Next**.

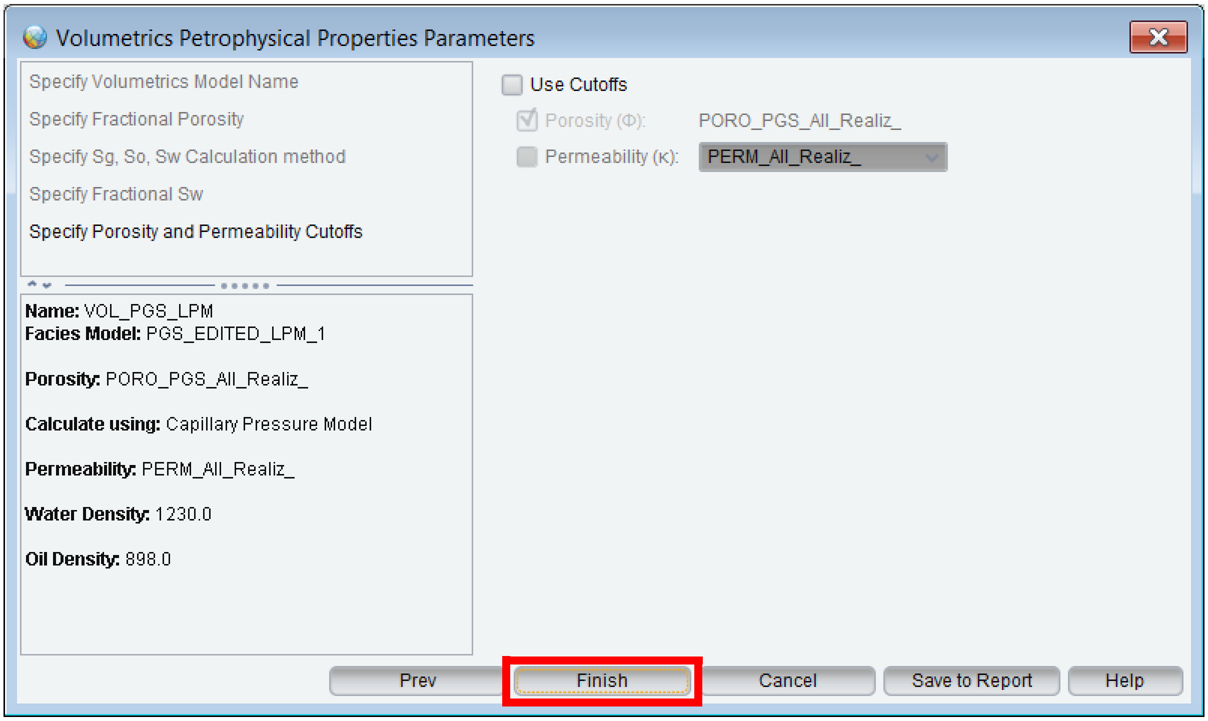


Specify Fluid Densities

1. For Permeability select **PERM_All_Realiz**.
2. Enter Water and Oil densities as shown. These values are typically provided by a reservoir engineer.
   - Water density: 1230 kg/m^3
   - Oil density: 898 kg/m^3
3. For the Advanced Options accept the default values.
4. Click **Next**.



5. Click **Finish** to complete and close the Volumetric Petrophysical Properties Parameters window. No cutoffs are used in the calculations.



Specify Fluid Contacts

1. Click the **Create New Oil Water Contact** icon.

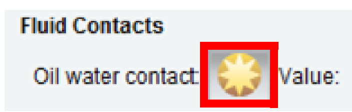
2. Select **Triangular** for the Model Method and enter the values for the

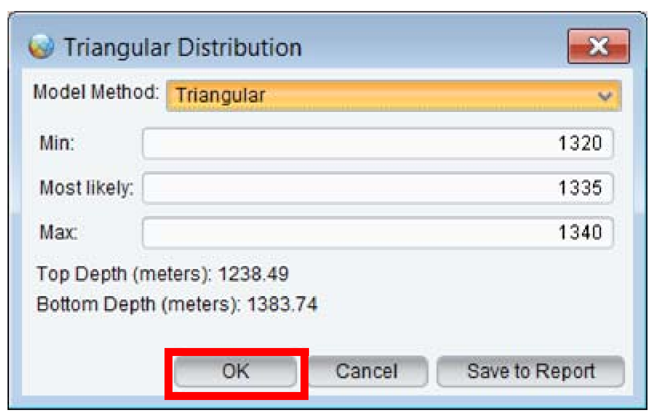
   - Min: **1320**
   - Most likely: **1335**
   - Max: **1340**

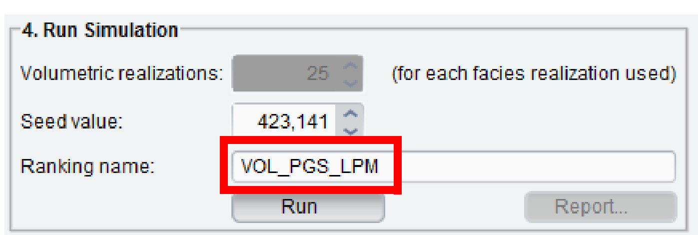
3. Click **OK** to finish and dismiss the window.

4. In the Run Simulation panel. Volumetric realization will be grayed out with the number of porosity realizations, Ranking name will be VOL_PGS_LPM.

5. Click **Run**.







Volumetric Report and Probability Curves

1. Click **OK** to dismiss the Volumetric Calculation completion window.

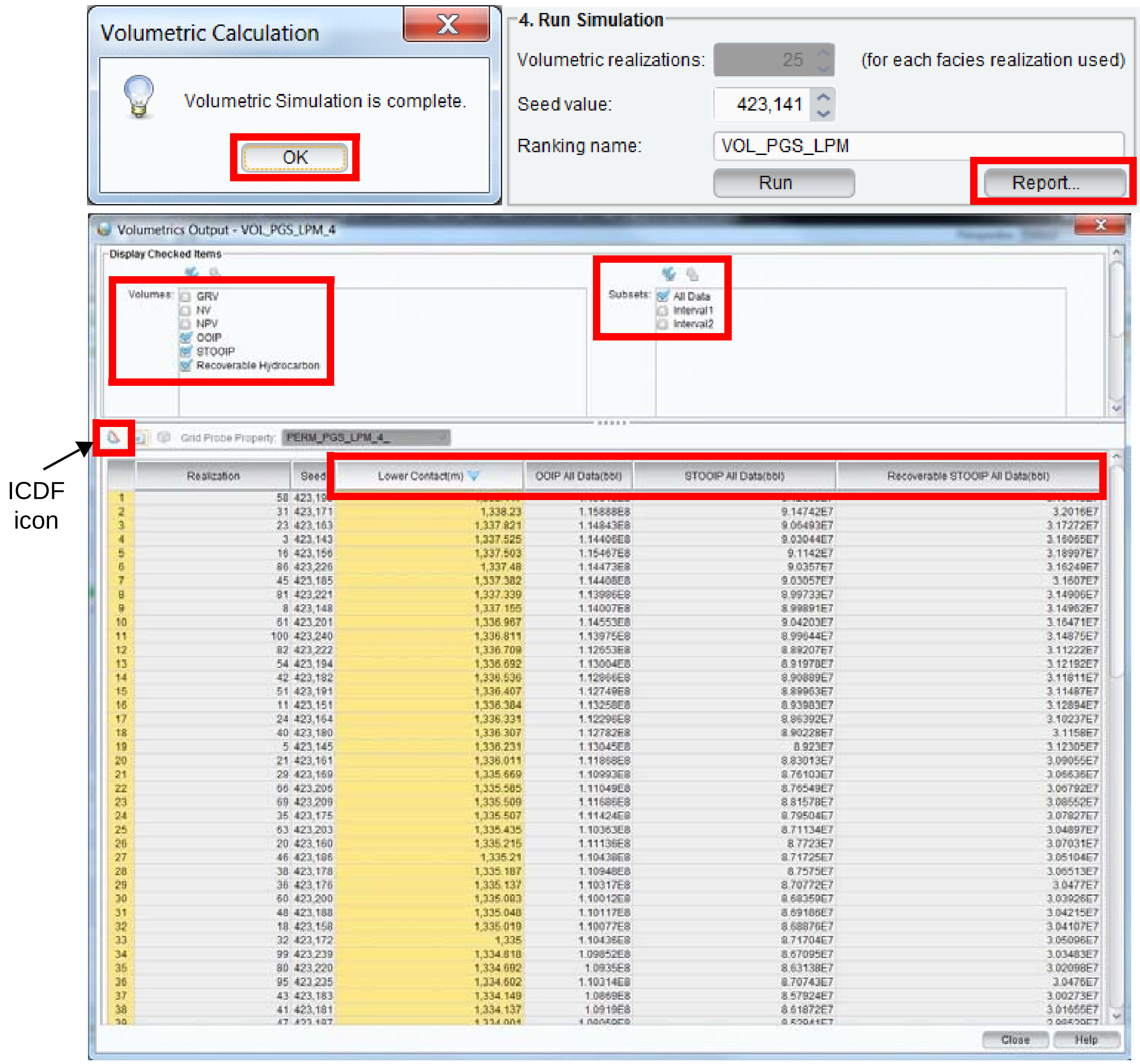
2. Click **Report** to view a report of the results.

3. Uncheck volumes **GRV**, **NV**, and **NVP** in the Display Checked Items panel as well as subsets **interval 1** and **2**.

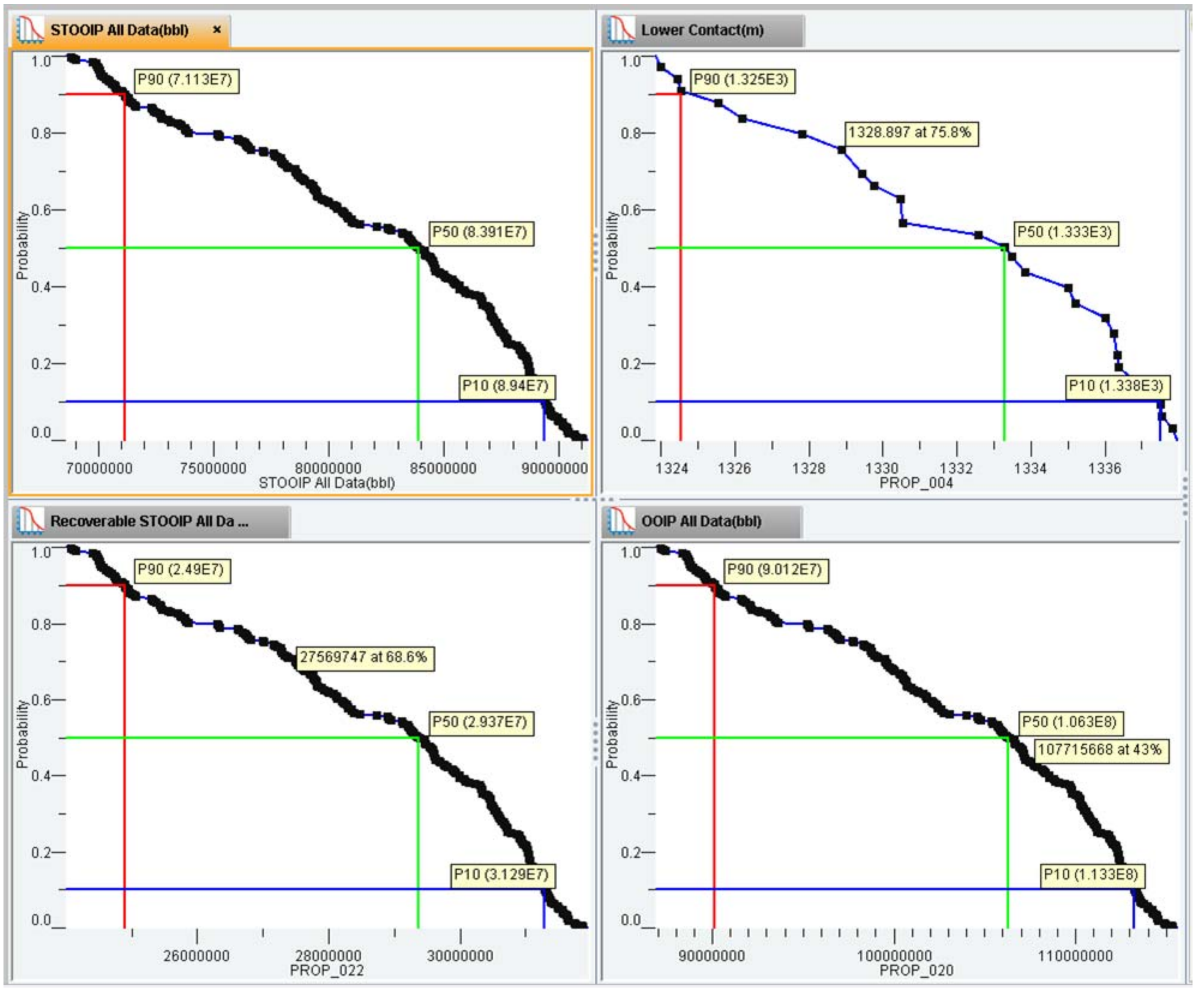
4. Use MB2 to select the **Lower Contact column**.

5. Click the **ICDF** icon (Inverse Cumulative Distribution Function).

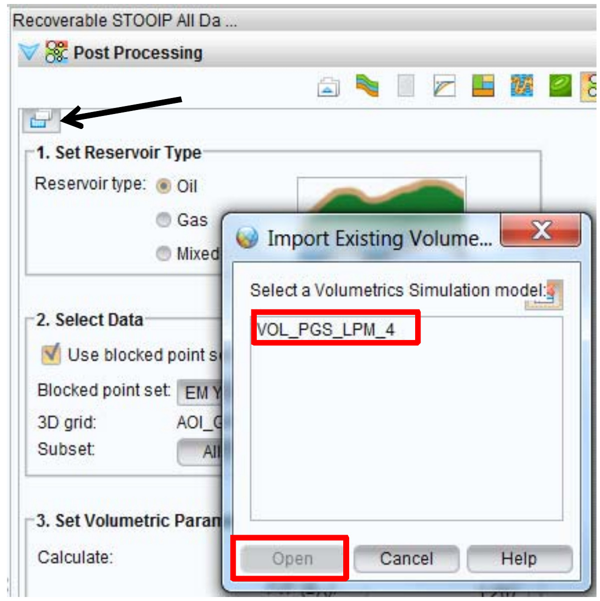
6. Do the same for **OOIP** and **STOOIP**, and **Recoverable STOOIP** and close the window.



The Results of the Inverse Cumulative Distribution Functions should look like the following image.

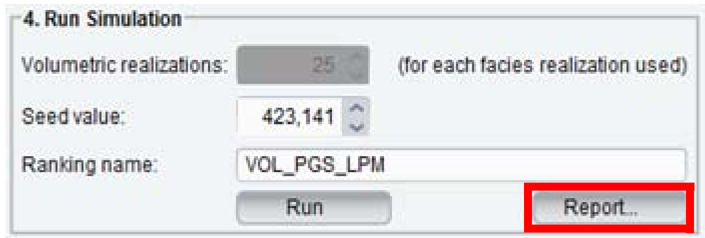


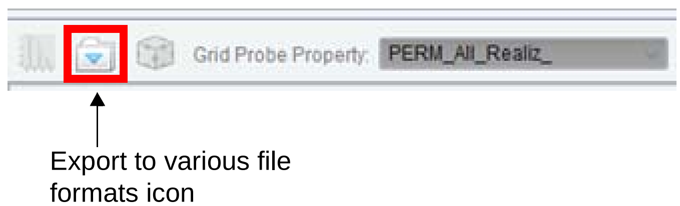
The Open Existing Simulation icon allows you to select a volumetric simulation model previously created when you clicked Run on the Static Volumetrics Calculations panel. At this point we have run only one Volumetric Simulation.



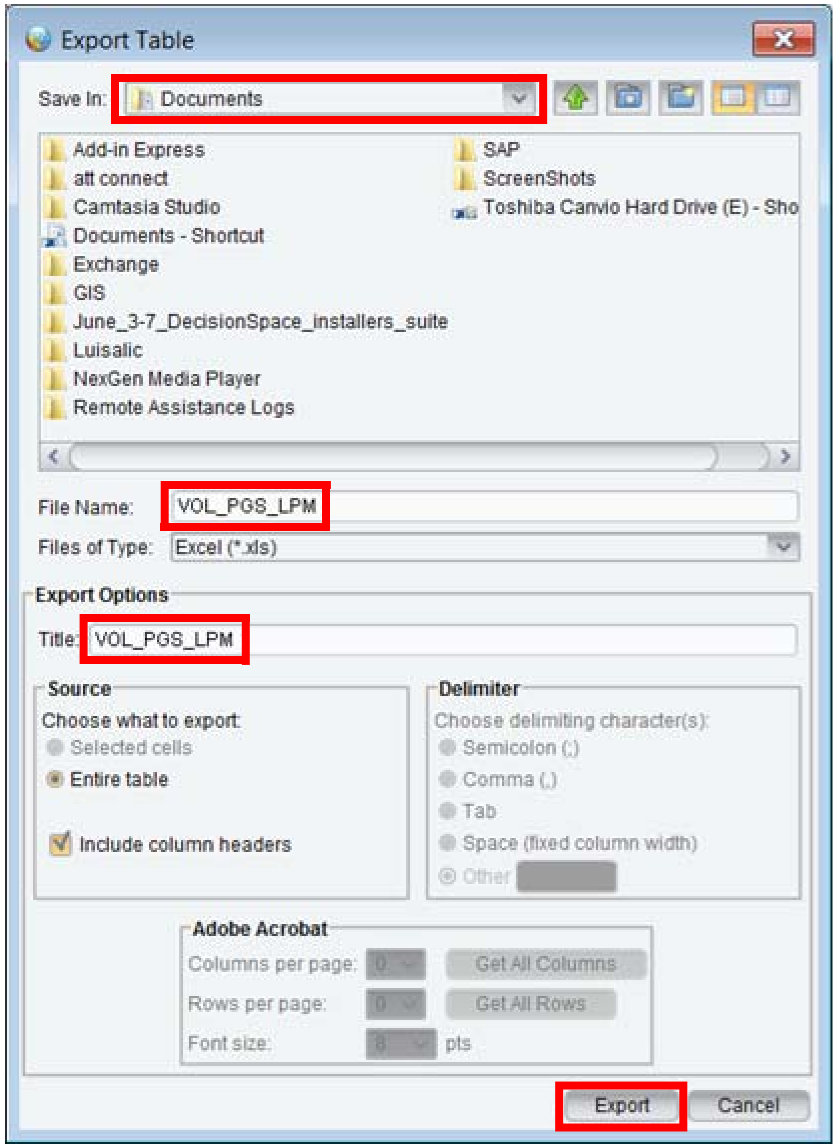
Export Volumetric as an EXCEL File

1. Click the report again.
2. Click the **Export to various file formats** icon.
3. In the Export Table window enter the File Name, Files of Type, and Title and click **Export**.



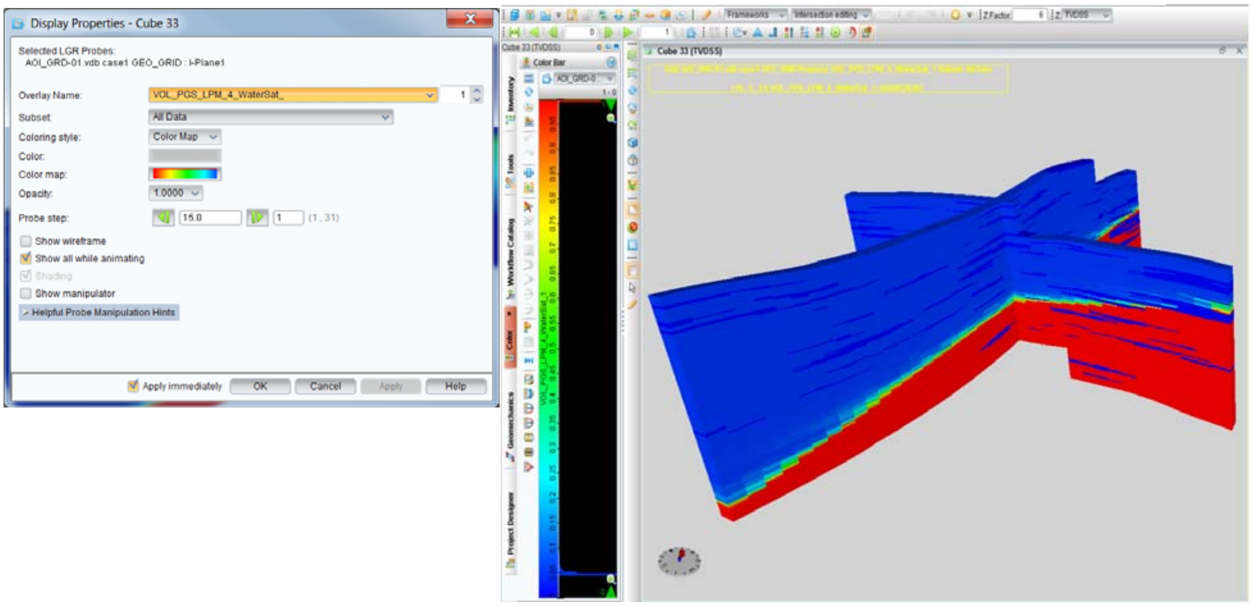


.


Export Table
Save In:
Documents
Add-in Express
att connect
Camtasia Studio
Documents - Shortcut
Exchange
GIS
June_3-7_DecisionSpace_installers_suite
Luisalic
NexGen Media Player
Remote Assistance Logs
SAP
ScreenShots
Toshiba Canvio Hard Drive (E) - Sho
File Name:
VOL_PGS_LPM
Files of Type:
Excel (*.xls)
Export Options
Title:
VOL_PGS_LPM
Source
Choose what to export:
Selected cells
Entire table
Include column headers
Delimiter
Choose delimiting character(s):
Semicolon (;)
Comma (,)
Tab
Space (fixed column width)
Other
Adobe Acrobat
Columns per page:
Get All Columns
Rows per page:
Get All Rows
Font size:
pts
Export
Cancel

Display Water Saturation Results

1. Display an I and J-Plane probe in a cube view.
2. MB3 on the probe and select **Display Properties**.
3. Select the overlay titled **VOL_PGS_LPM_WaterSat**.
4. Explore the data using the I or J-Plane Probe controls.



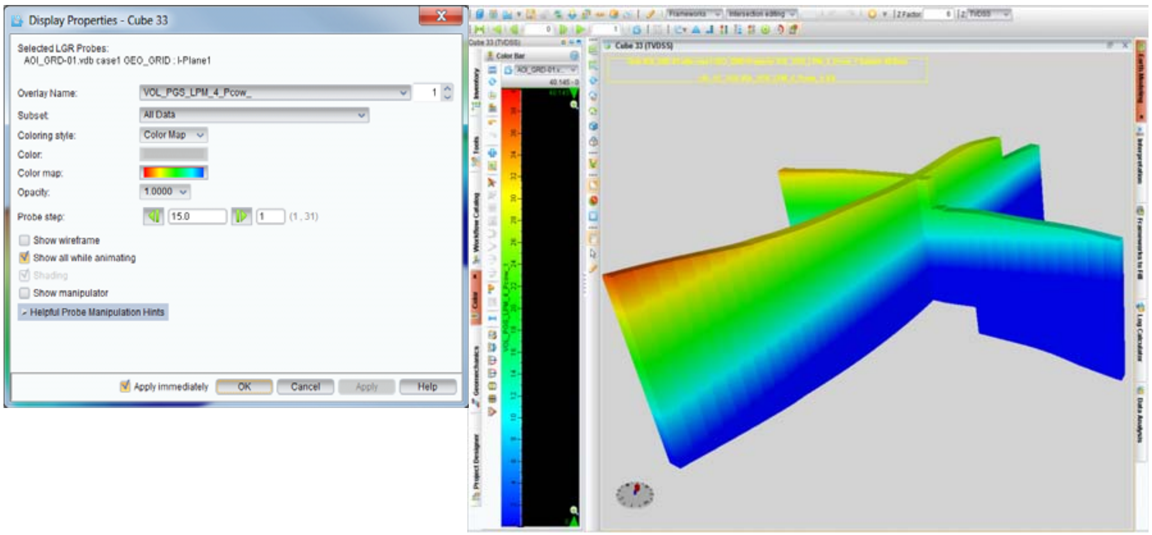
Display Pressure to Oil-Water Contact

1. Display an I and J-Plane probe in a cube view.

2. MB3 on the probe and select **Display Properties**.

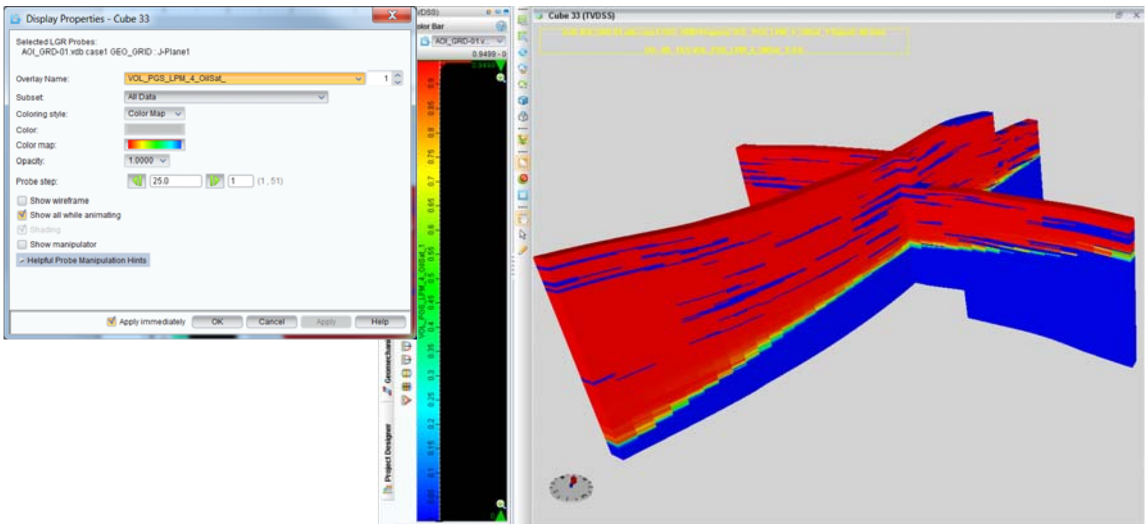
3. Select the overlay titled **VOL_PGS_LPM_Pcow**.

4. Explore the data using the I or J-Plane Probe controls.



Display Oil Saturation

1. Display an I and J-Plane probe in a cube view.
2. MB3 on the probe and select **Display Properties**.
3. Select the overlay titled **VOL_PGS_LPM_OilSat**.
4. Explore the data using the I or J-Plane Probe controls.



Uncertainty Analysis: Summary Statistics

In this exercise you learn how to post process realizations to create summary statistics. Compute Water Saturations Summary Volumes.

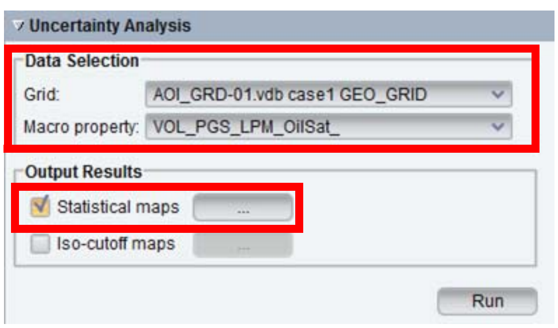
1. Select data for grid and Macro property in the Uncertainty Analysis subtask pane under Post Processing.

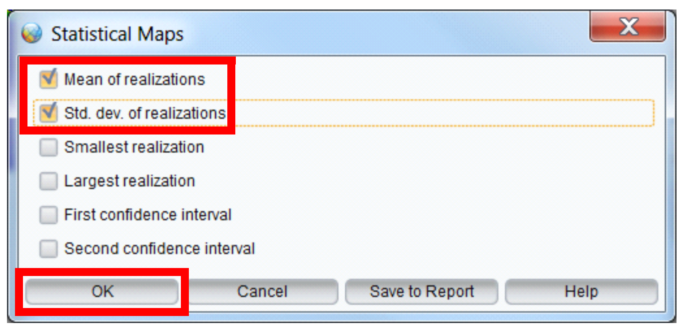
2. Check ON **Statistical Maps** checkbox and click the  button.

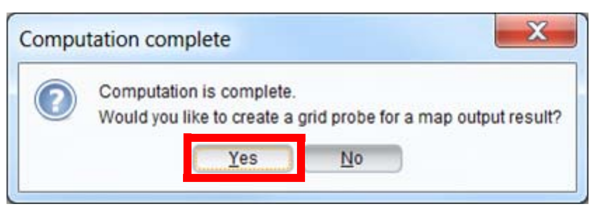
3. In the Statistical Maps window give checkmarks only to **Mean of realizations** and **Std. dev. of realizations**.

4. Click **OK** to dismiss the Statistical Maps window.

5. Click **Run**.

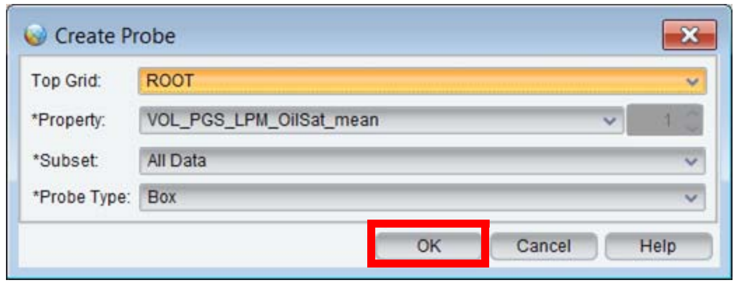




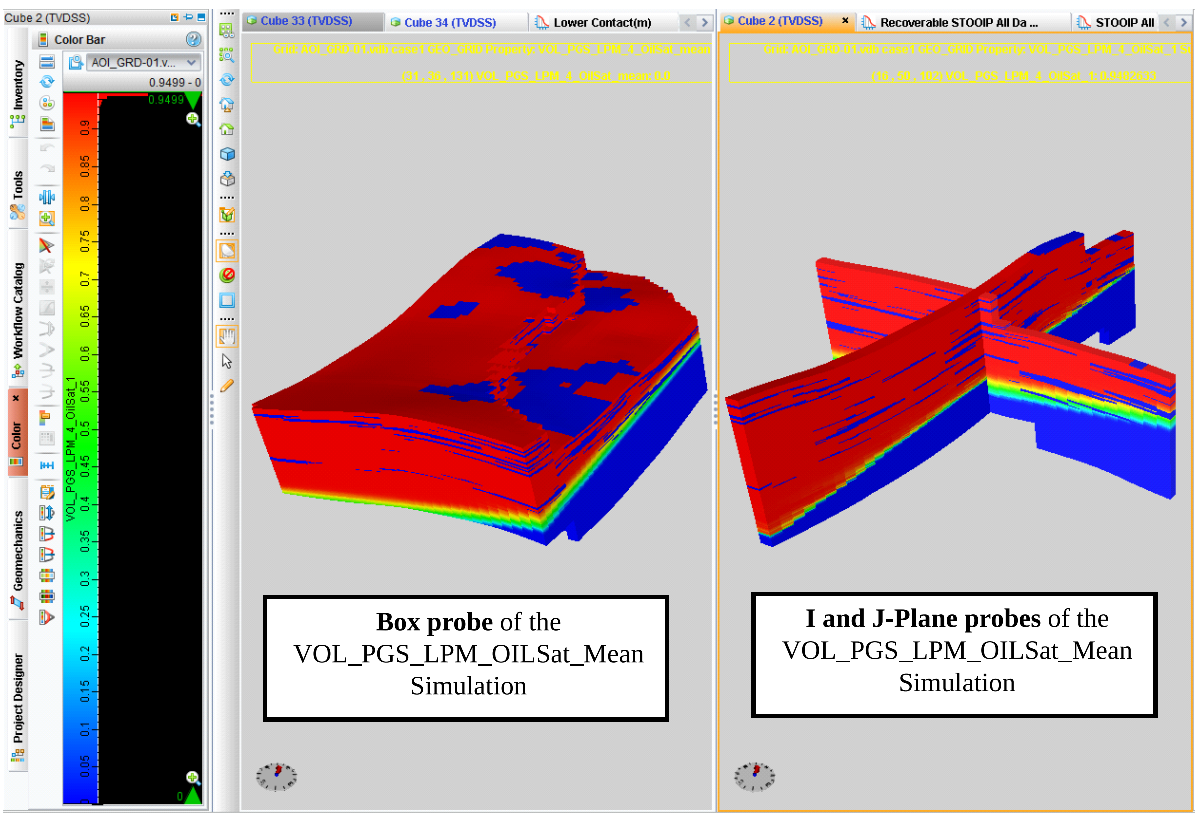


6. Click **OK** to generate a box probe of your volumetric simulation.

You can always change these options later by clicking MB3 on the probe and selecting **Display Properties**.

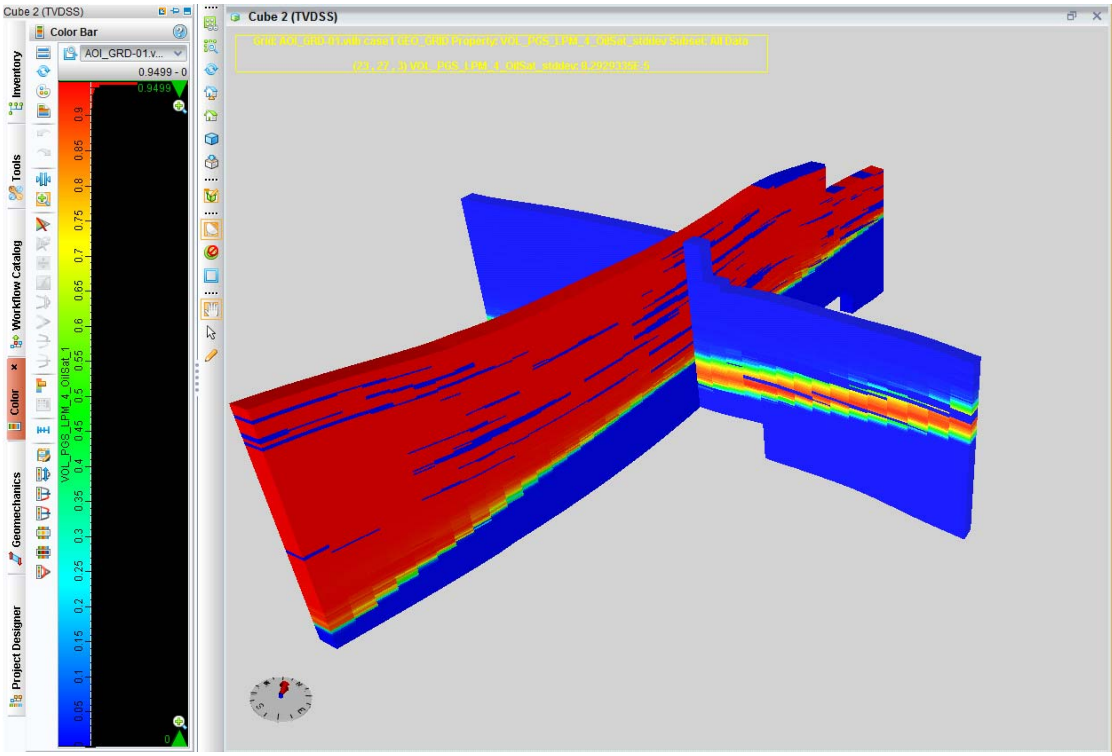


7. Activate a cube view and toggle on the new box probe in the inventory. You can examine the volumetric simulation further by creating different types of probes.



8. Click MB3 on the J-Plane probe and select **Display Properties**.

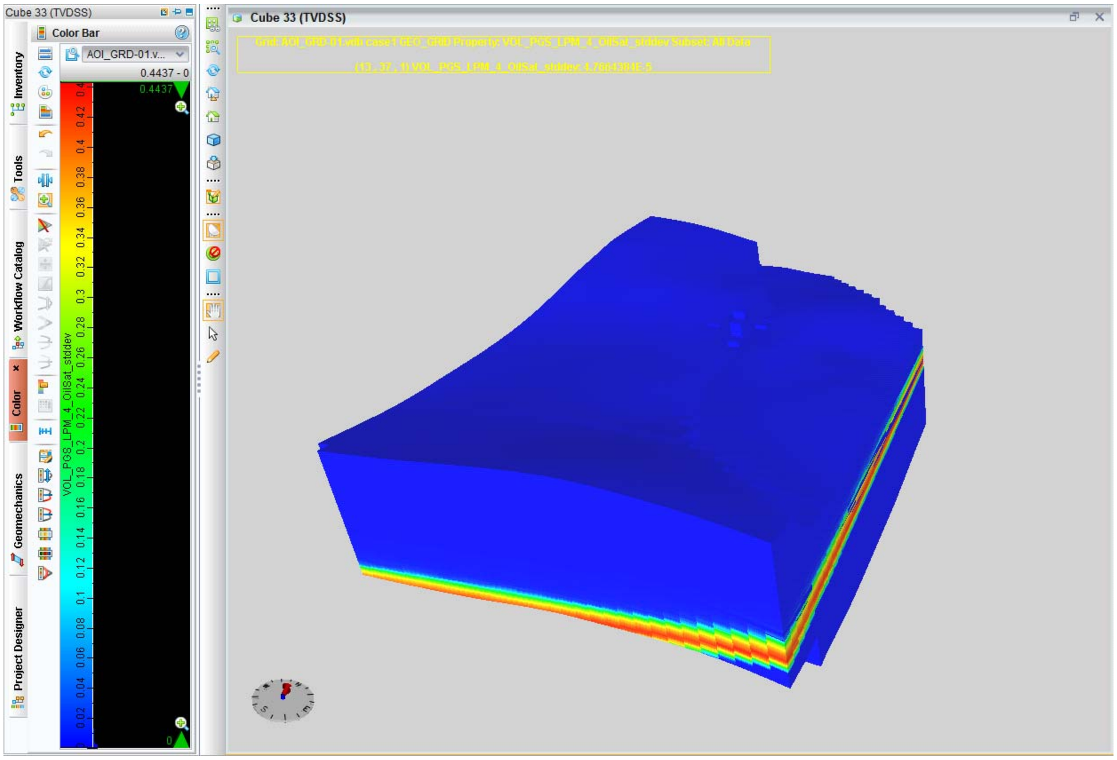
9. Change the overlay to **VOL_PGS_LPM_OilSat_stddev** to display the standard deviation overlay.



10. Toggle off all probes except for one Box Probe.

11. Click MB3 on the Box probe and select **Display Properties**.

12. Change the overlay to **VOL_PGS_LPM_OilSat_stddev**.



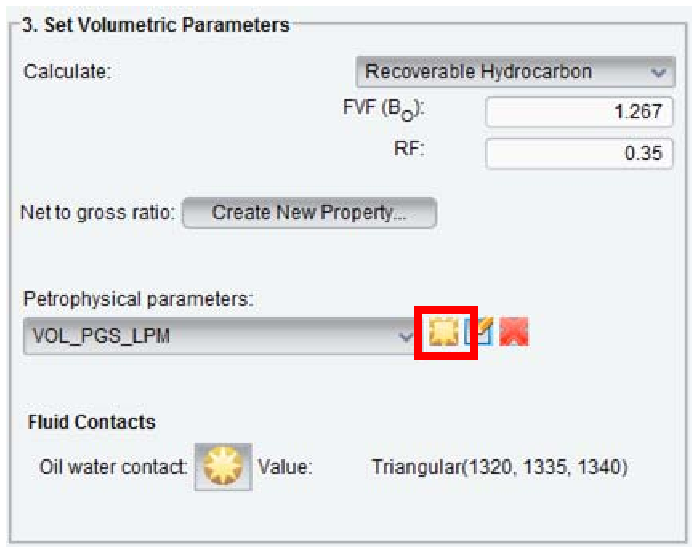
13. Compute summary statistics of other properties if time permits.

14. Save the session.

Volumetrics: Saturation-Height Method

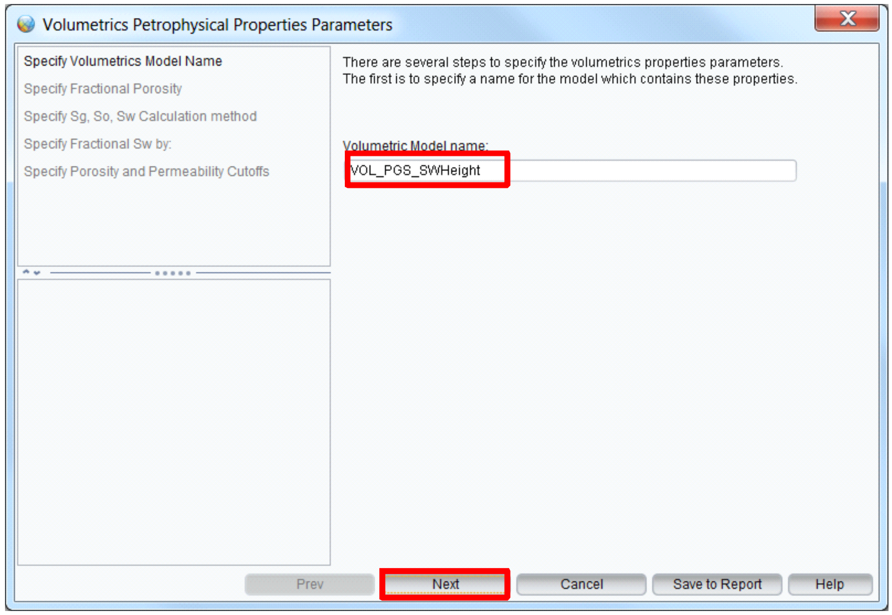
This volumetric computation exercise illustrates how to compute fluid saturations using the Saturation-Height Method. You will use the previous NTG parameters so there is no requirement to revisit that dialog table.

1. Click the **Create New Model** icon in the Set Volumetric Parameters panel.

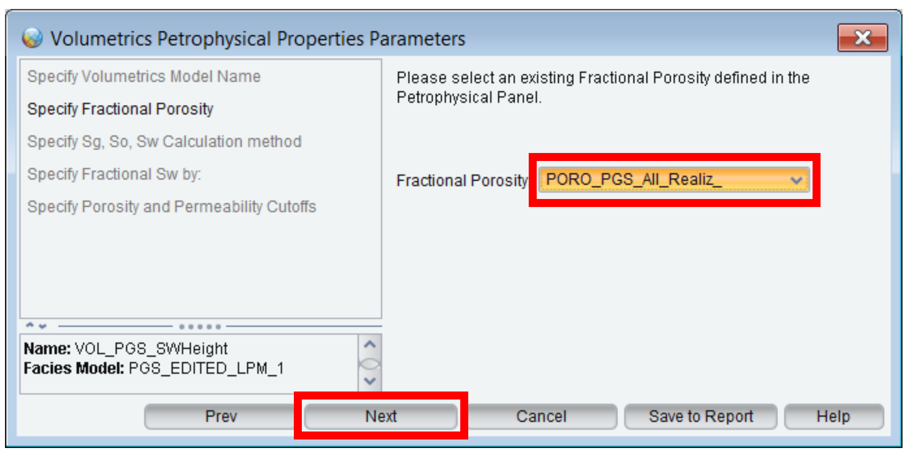


2. Enter the Volumetric Model Name as VOL_PGS_SWHeight.

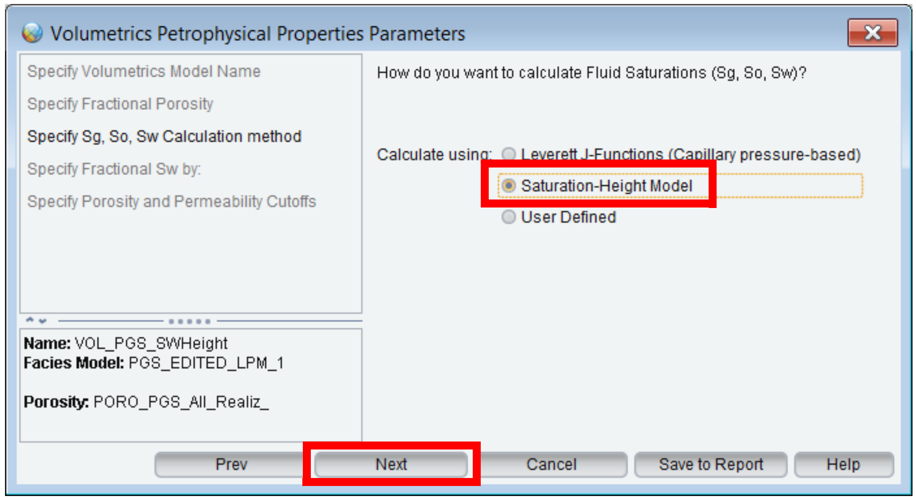
3. Click **Next**.



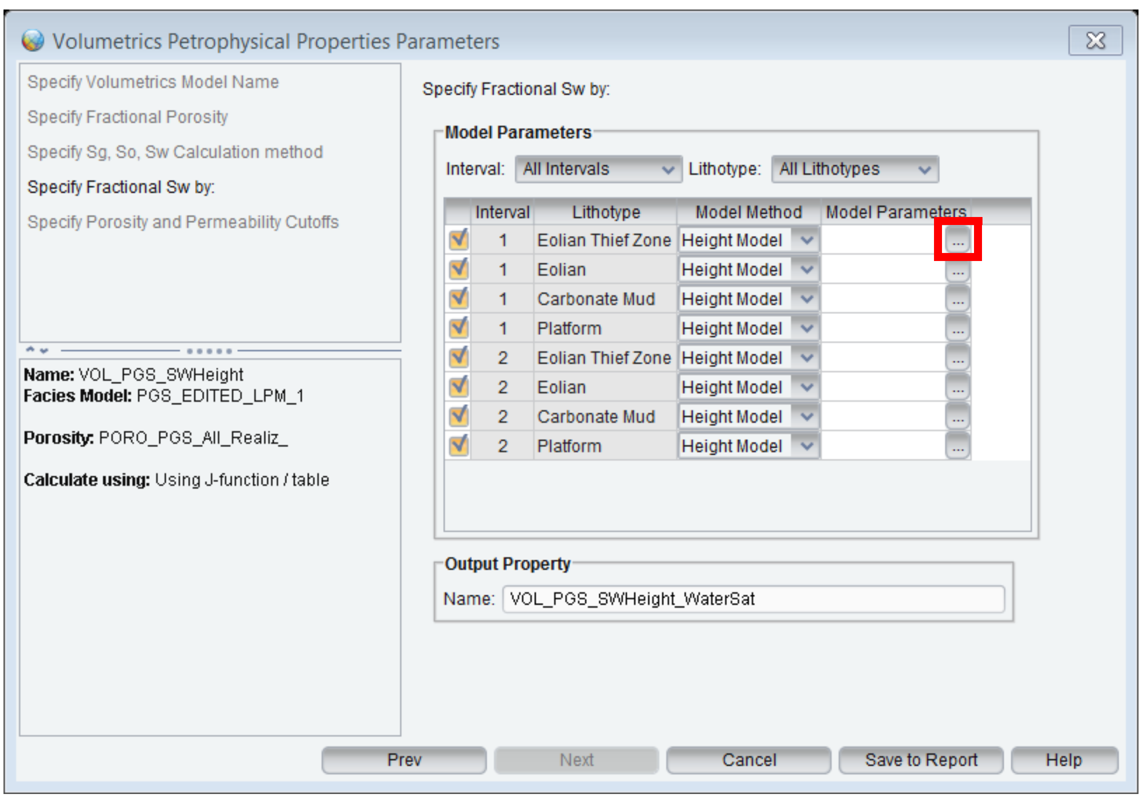
4. In the Fractional Porosity drop-down menu, select **PORO_PGS_All_Realiz** and click **Next**.



5. Select the **Saturation-Height Model** option for calculating fluid saturation and click **Next**.



6. In Model Parameters, click the **Model Parameters** button for the Eolian Thief Zone in interval 1.



7. Select the **Use new Saturation-Height model** option.

8. Name the model SW_Model_1_Siltstone.

9. Click the **Import Saturation-Height model** icon.

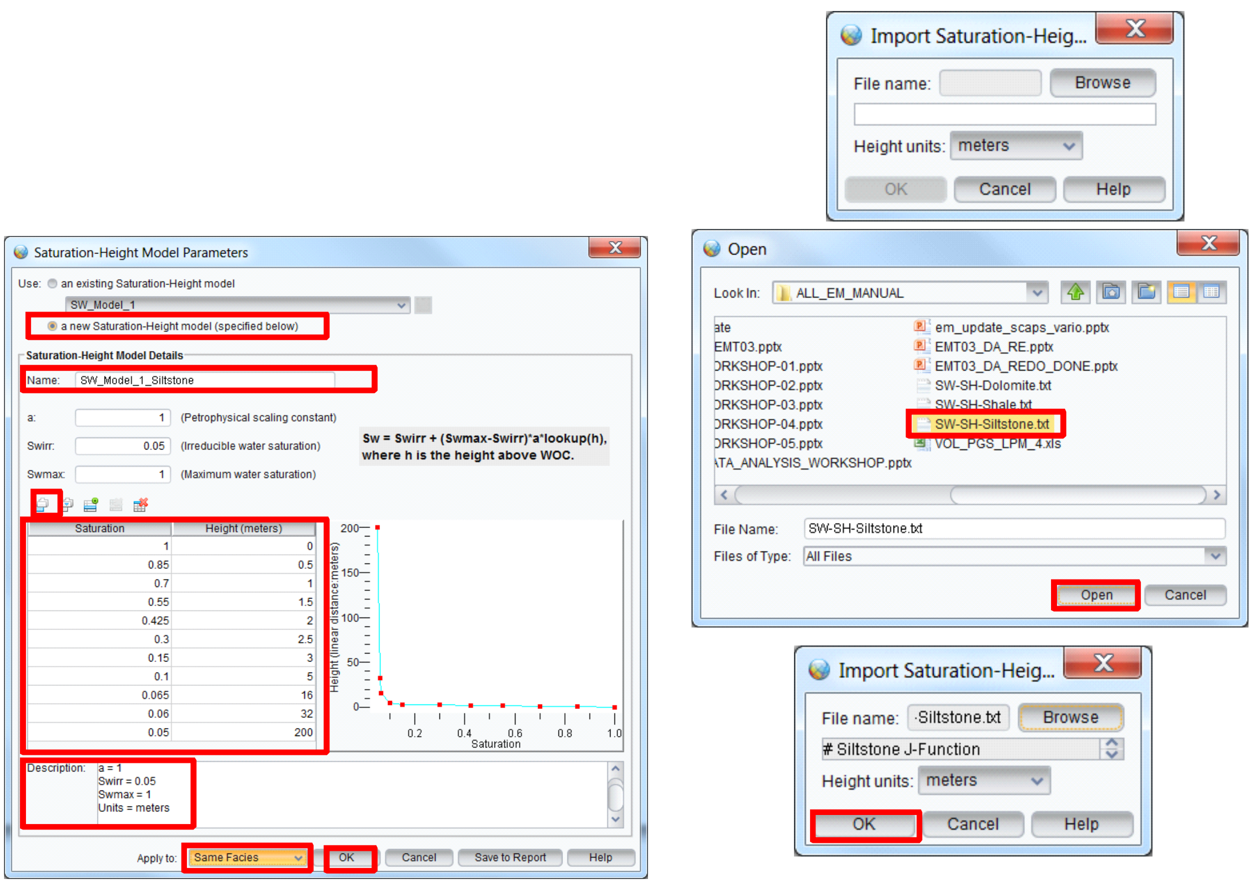
10. Click **Browse** on the Import dialogue box.

11. Open **SW-SH-Siltstone.txt** from the your computer's directory. (Ask Instructor for location of the file.)

12. Change
    - a = **1**
    - Swirr = **0.05**

13. Click **OK** on the Import Dialogue box.

14. Apply to **Same Facies** and click **OK**.

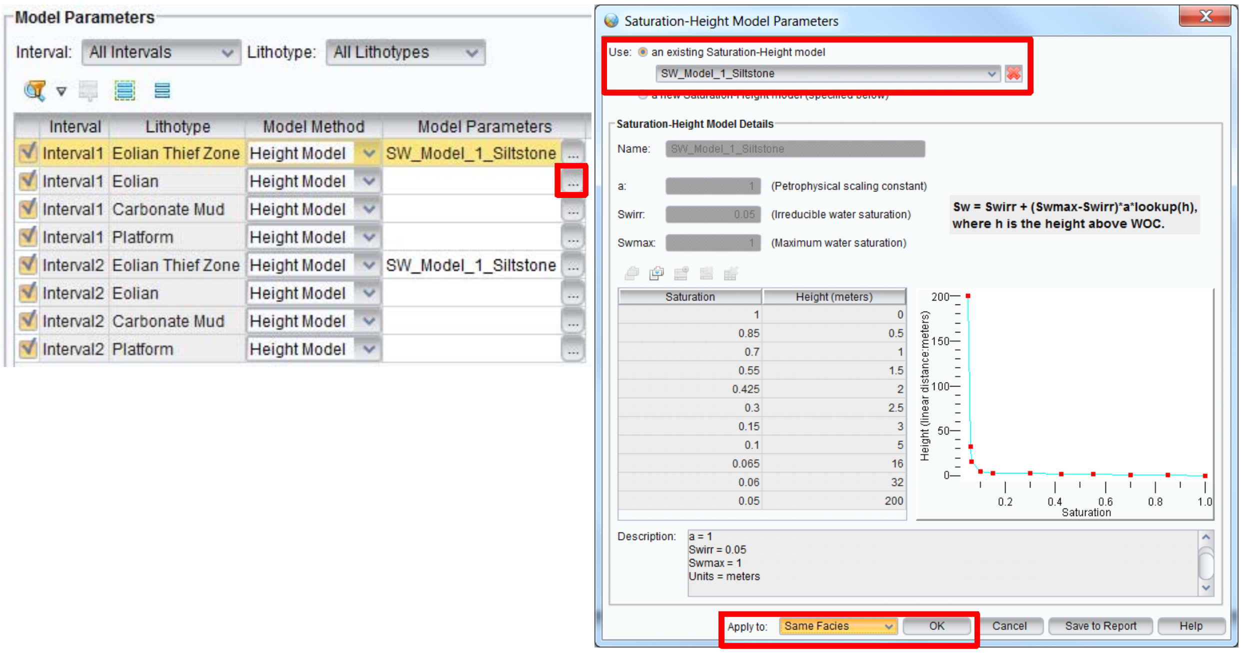


15. In Model Parameters, click the **Model Parameters** button  for the Eolian Lithotype in interval 1.

16. In the Saturation_Height Model Parameters window select **Use an existing Saturation-Height Model**.

17. Select **SW_Model_1_Siltstone**.

18. Apply to **Same Facies** and click **OK**.

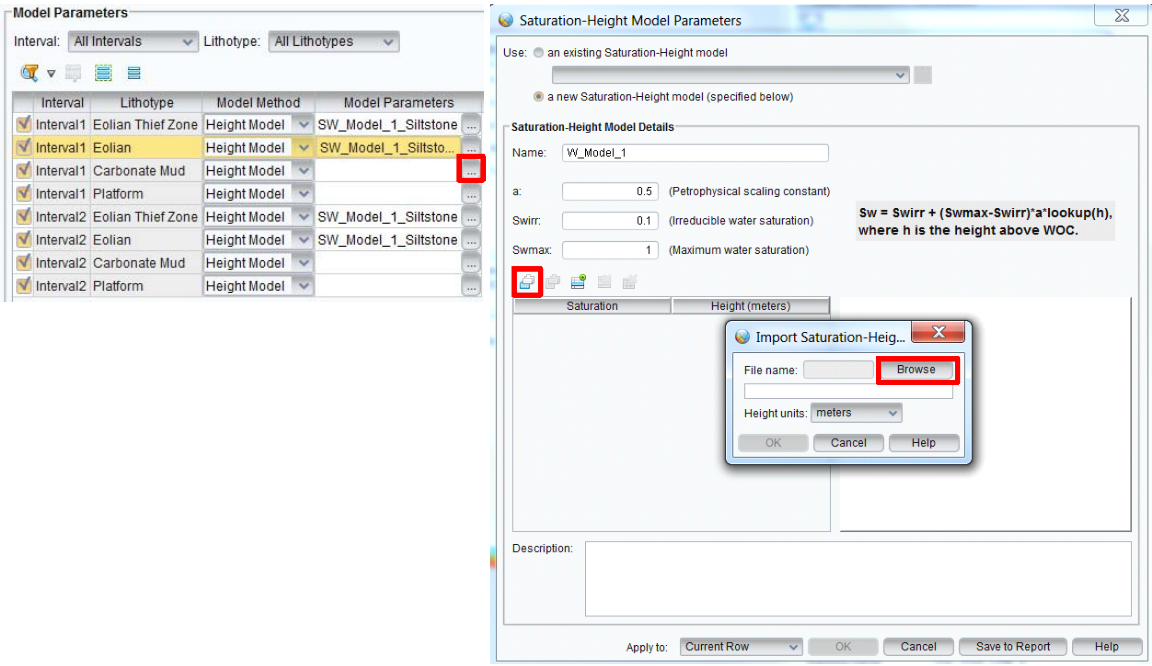


19. In Model Parameters, select the **Model Parameters** button  for the Carbonate Mud Lithotype in interval 1.

20. Select existing new **Saturation-Height Model**.

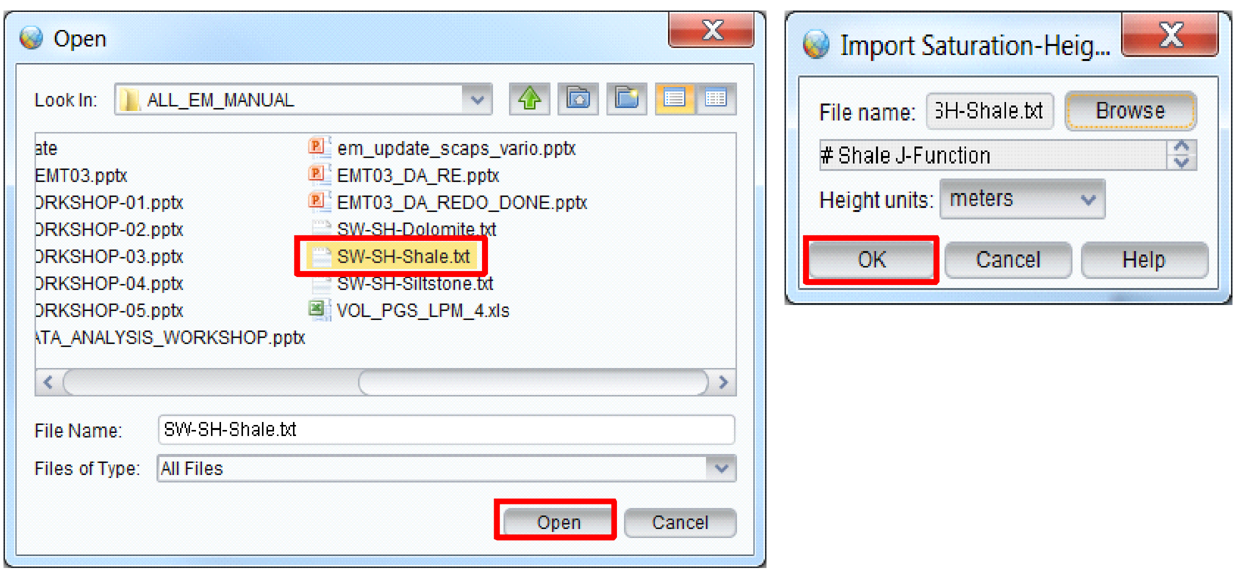
21. Click the **Import** icon.

22. Click **Browse** on the import dialog box.



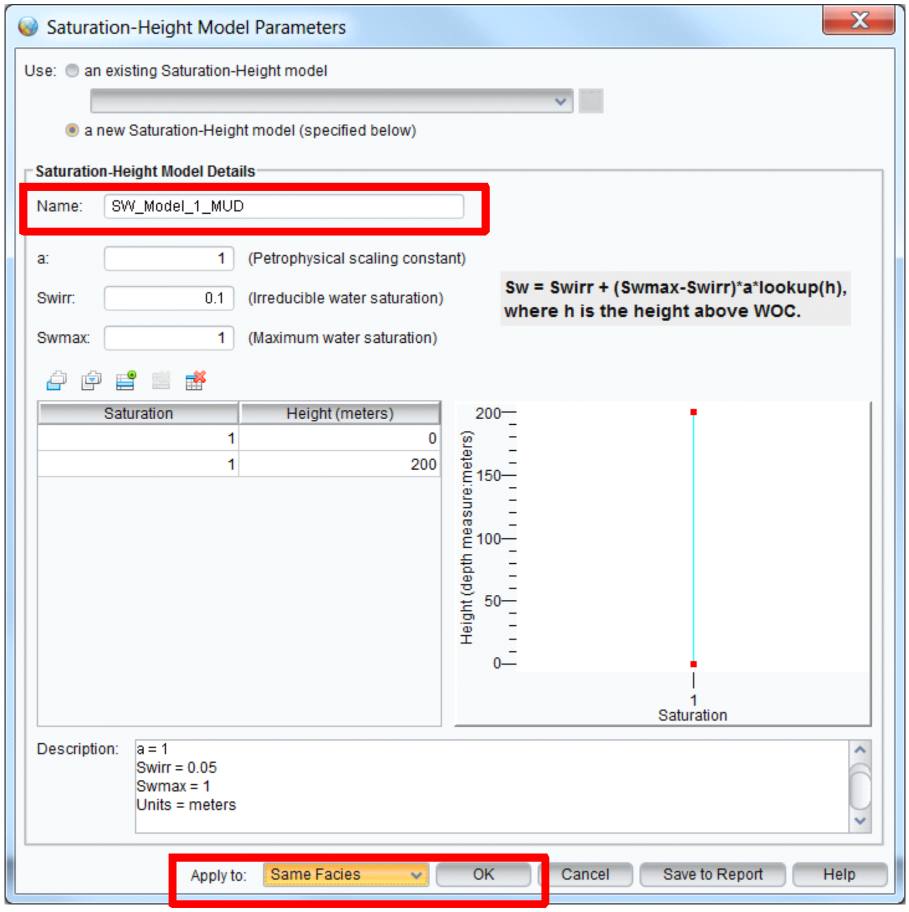
23. Open **SW-SH-Shale.txt**.

24. Click **OK**.



25. Name the model SW Model 1 MUD.

26. Apply to **Same Facies** and click **OK**.



Enter values from this table into the Saturation-Height table as described on the following page for the Platform Lithotype.

Dolomite J-Function
a = 1
Swirr = 0.05
Swmax = 1
Units = meters

Saturation	Height
1.00	0.00
0.99	5.00
0.98	10.0
0.95	15.0
0.90	17.0
0.80	20.0
0.70	22.0
0.60	25.0
0.50	27.0
0.40	31.0
0.30	37.0
0.20	45.0
0.15	55.0
0.10	70.0
0.06	100.0
0.05	200.0

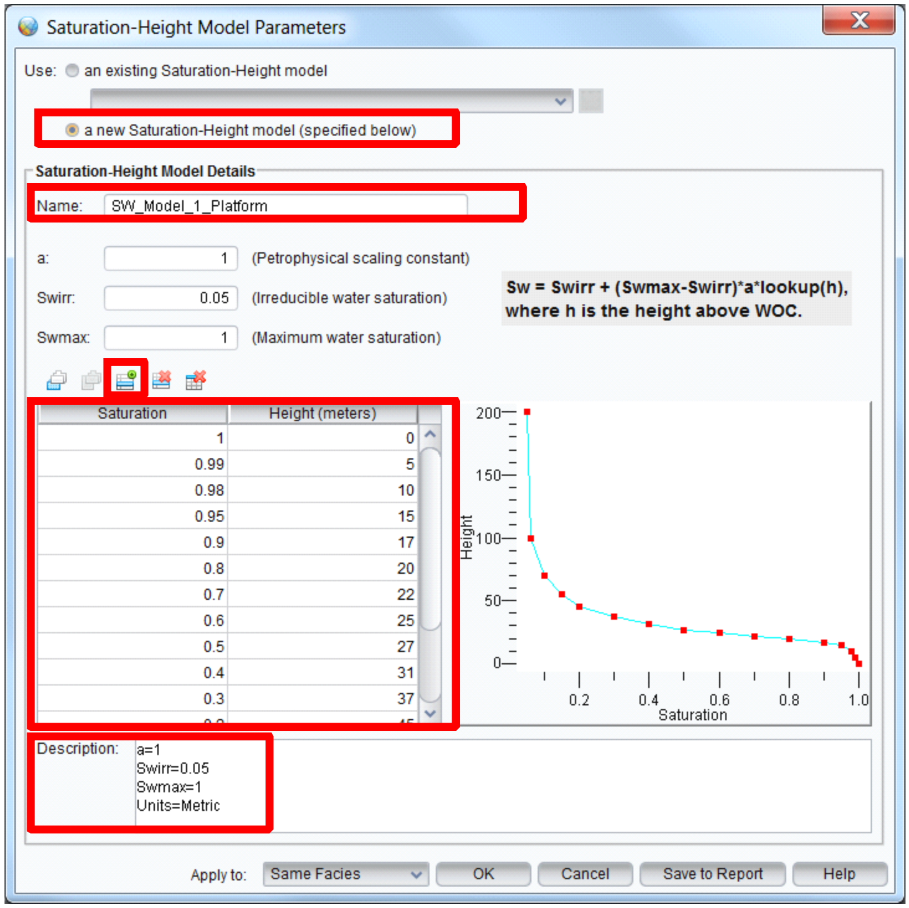
27. Select the Use new Saturation-Height model option.

28. Name the model SW_Model_1_Platform.

29. Use the **Insert a row** icon to create new roles in the table. Add 16 rows.

30. Enter the values shown in the table to create a new saturation-height model.

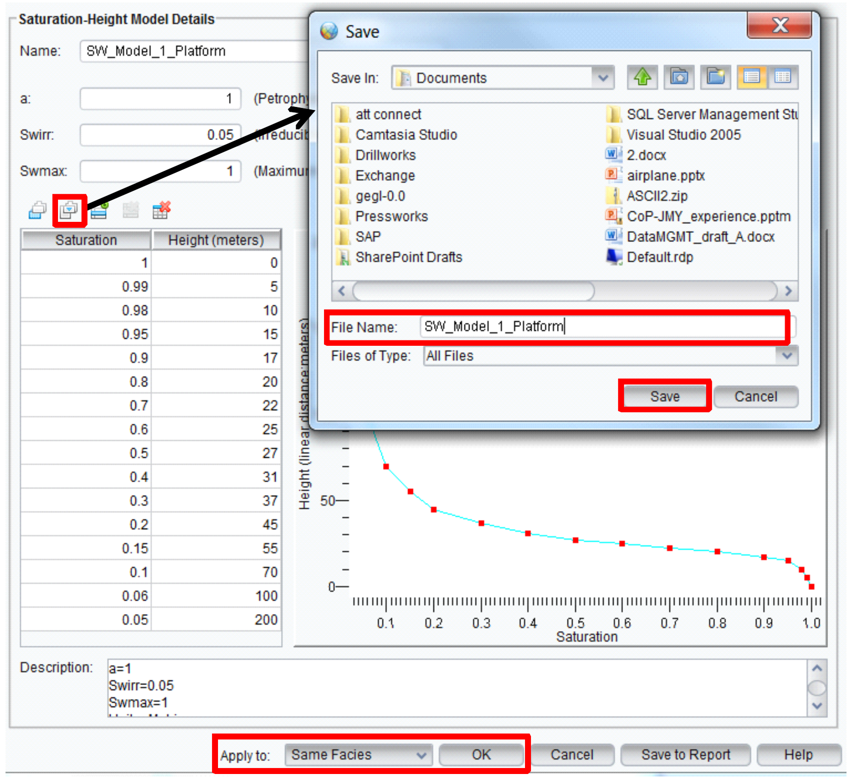
31. Enter a description.



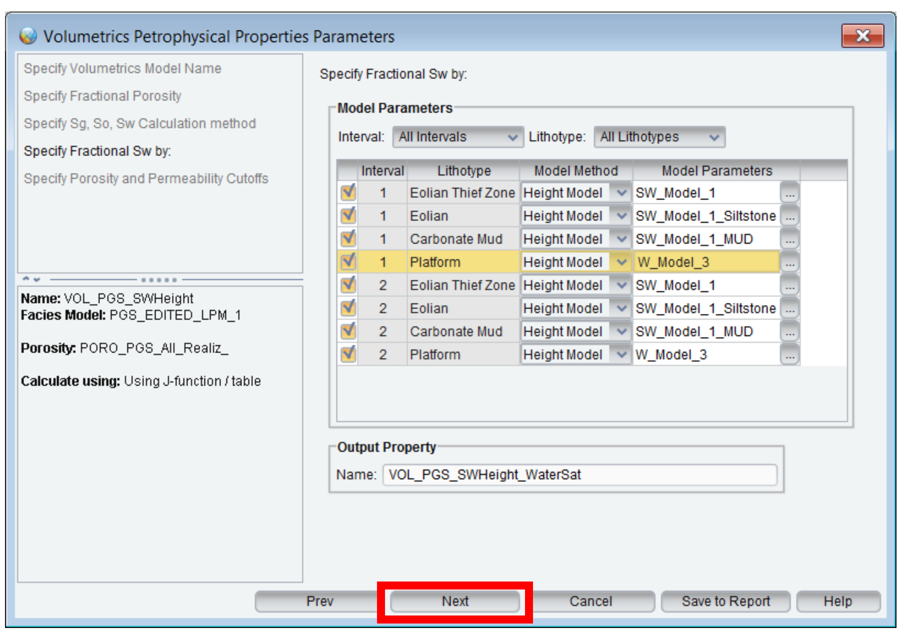
32. Export the Saturation-Height model.

33. Select the Apply to **Same Facies** option.

34. Click **OK**.

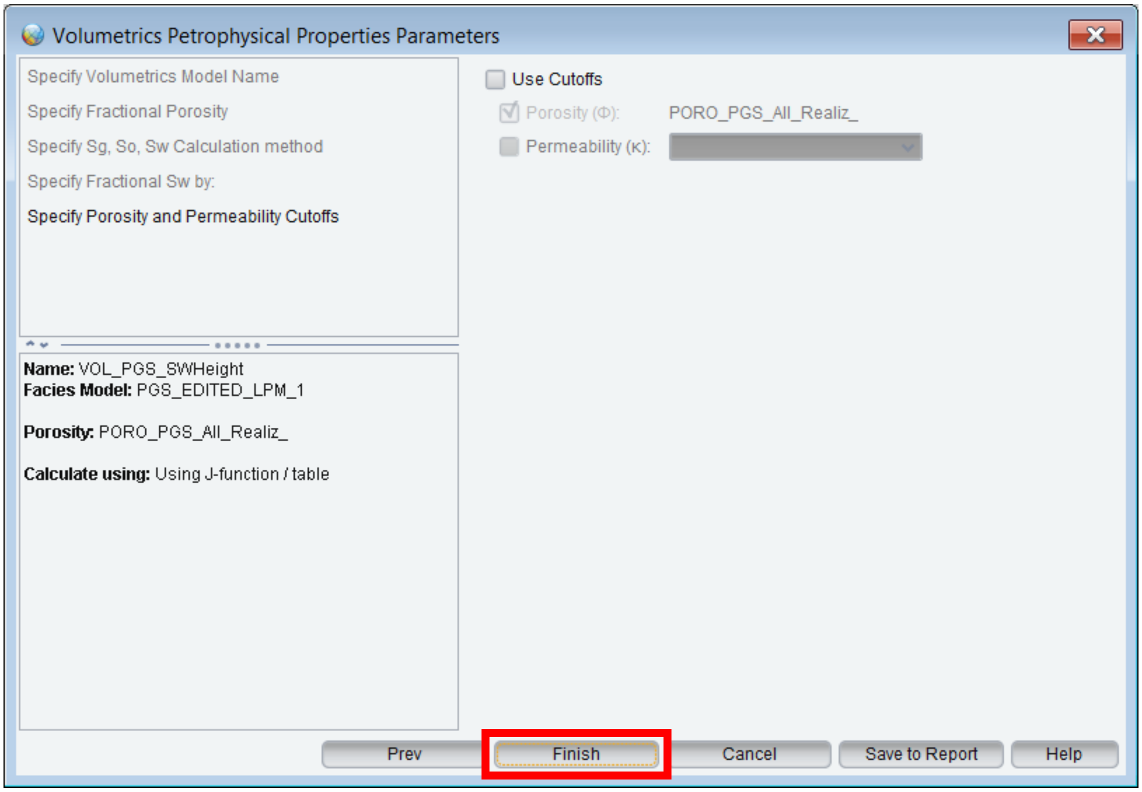


VOL_PGS_SWHeight: Your Petrophysical Properties Volumetric Parameters should match the Parameters below.

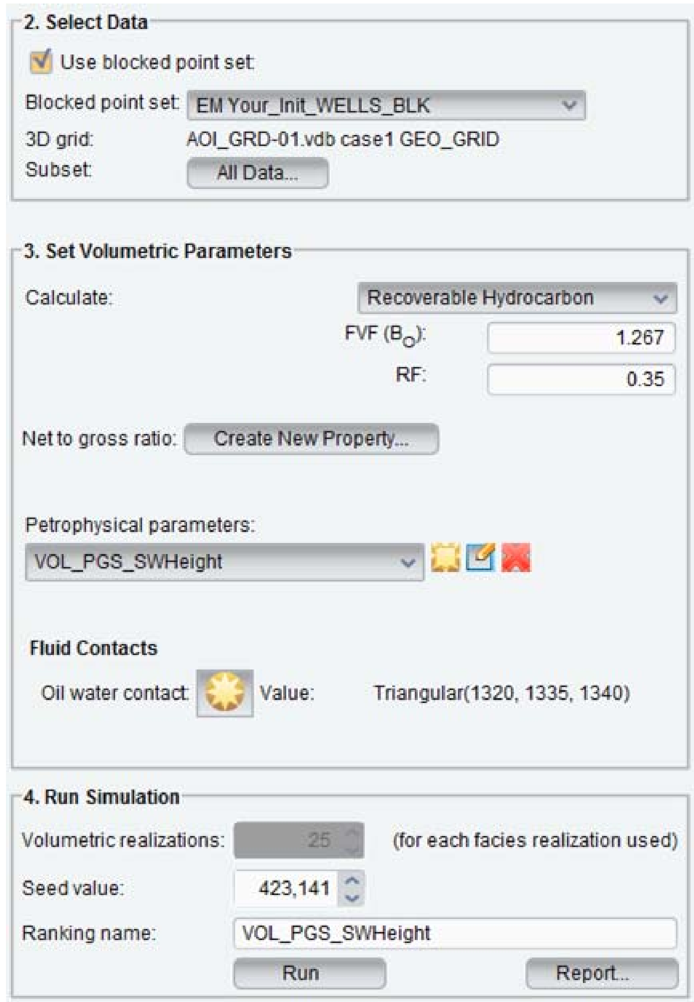


35. Click **Next** to Continue.

36. Click **Finish** to complete the Volumetric Parameters dialog box.



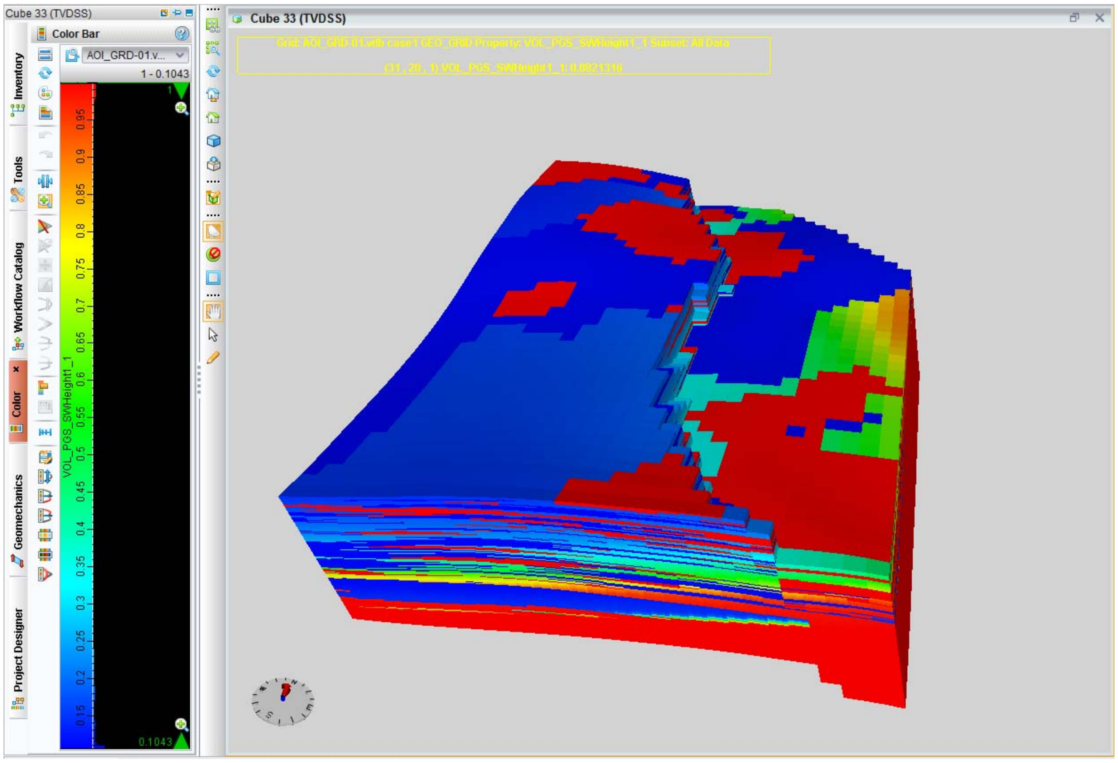
37. For Fluids contacts leave the triangular used before, **VOL_PGS_SWHeight** for the Ranking name and click **Run**.



Display Results

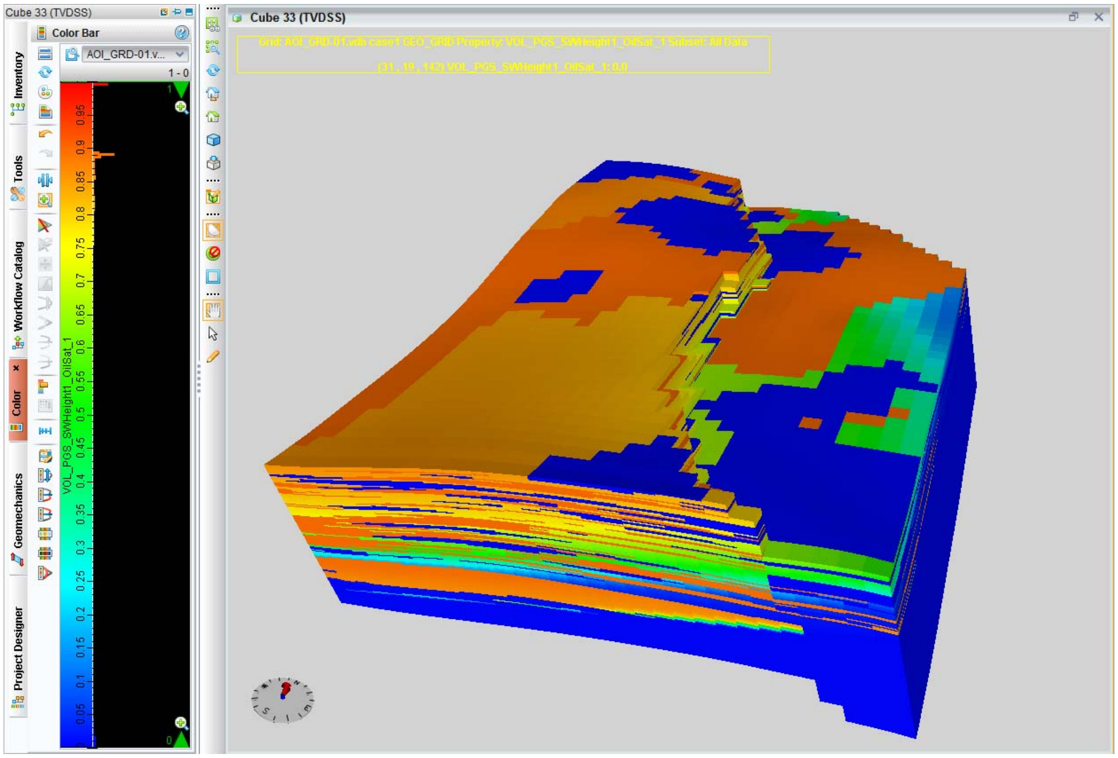
1. Click MB3 on the box probe in a cube view and select **Display Properties**.

2. Change the overlay to **VOL_PGS_SWHeight1_WaterSat**.



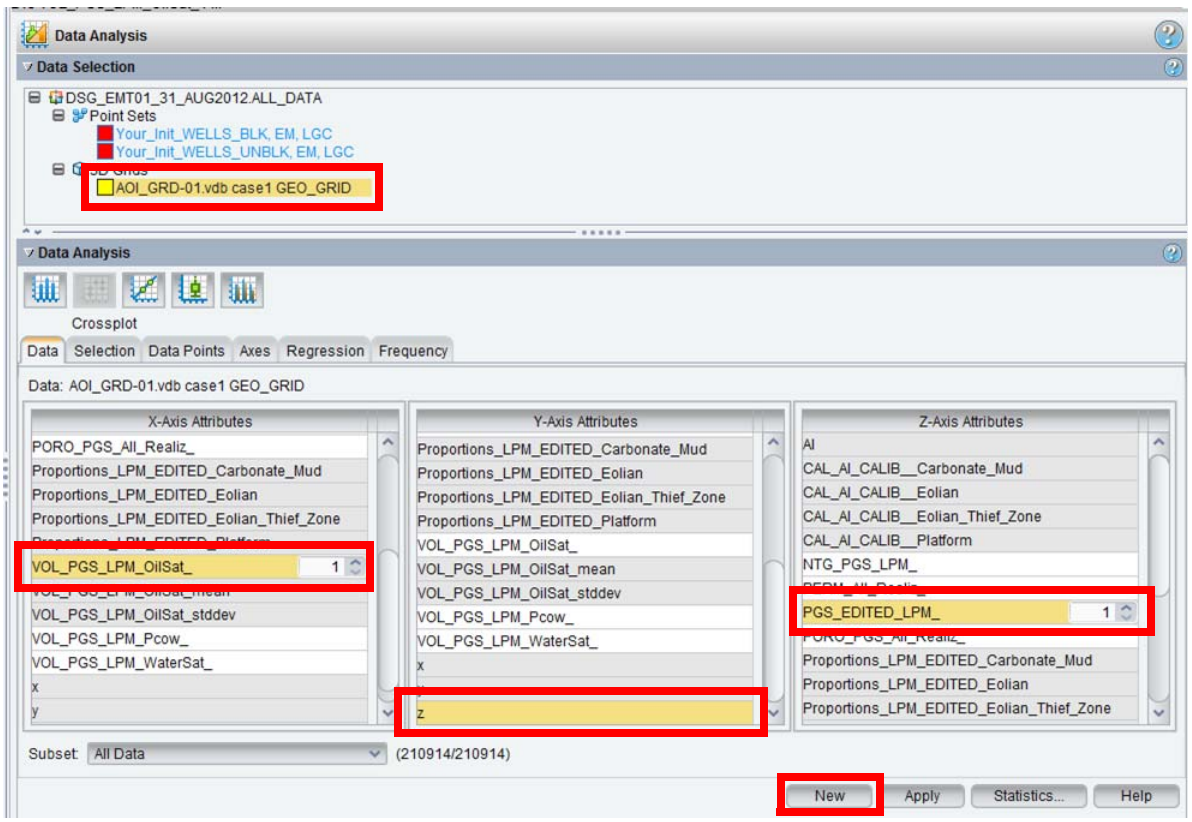
3. Click MB3 on the box probe in a cube view and select **Display Properties**.

4. Change the overlay to **VOL_PGS_SWHeight1_OilSat_**.



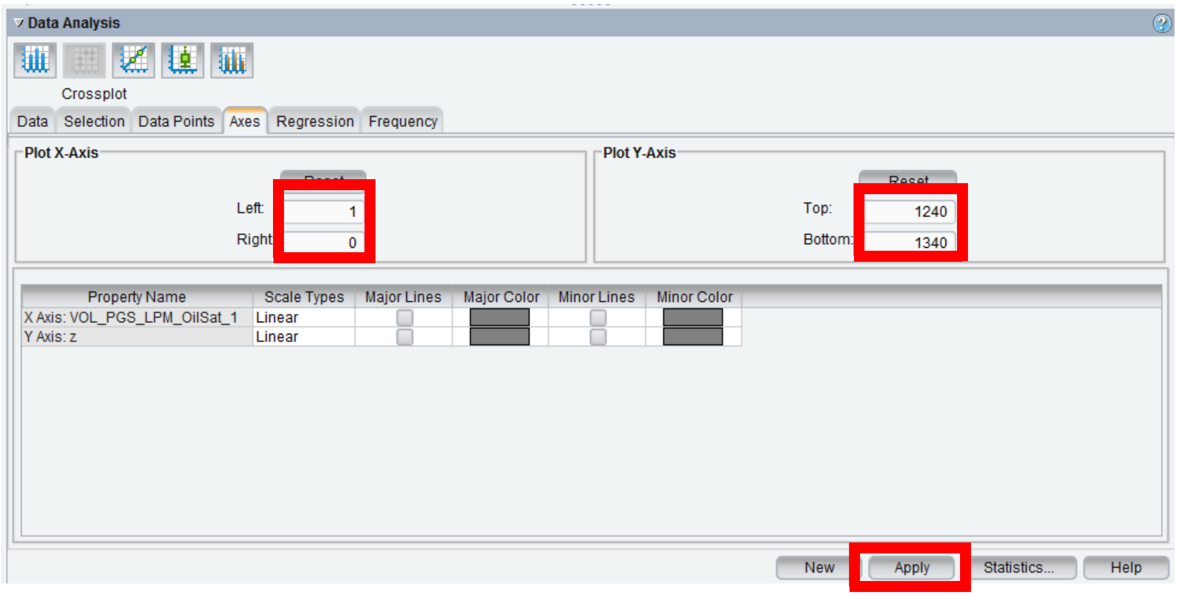
Cross-plot: Oil Saturation vs. Depth

1. In the Data Analysis task pane select the 3D Grid in the Data Selection panel.
2. Click the **Cross-plot** icon.
3. Select **VOL_PGS_LPM_OilSat** #1 for the X Axis, **Z** for the Y axis, and **PGS_Edited_LPM** #1 for the Z axis.
4. Click **New**.



5. Click the **Axes** tab in the Data Analysis panel.
6. Change the X-Axis and Y-Axis Plot values as shown in the following image.

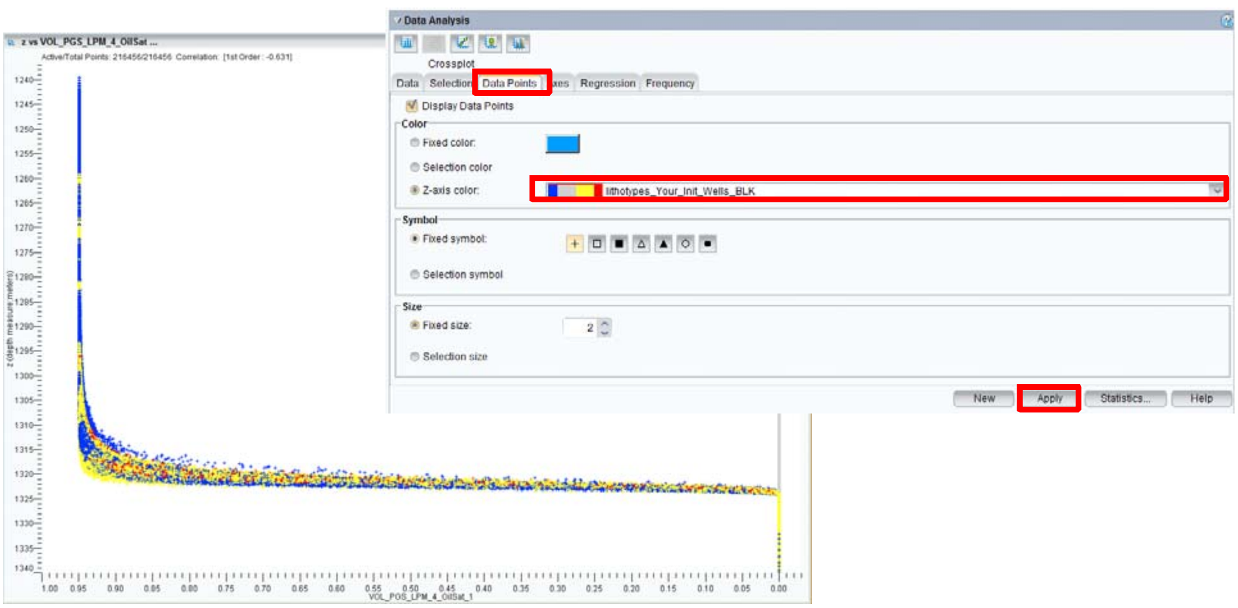
7. Click **Apply**.



8. Click the **Data Points** tab in the Data Analysis panel.

9. Change the Z-Axis color to **lithotypes_Your_Init_Wells_BLK**.

10. Click **Apply**.

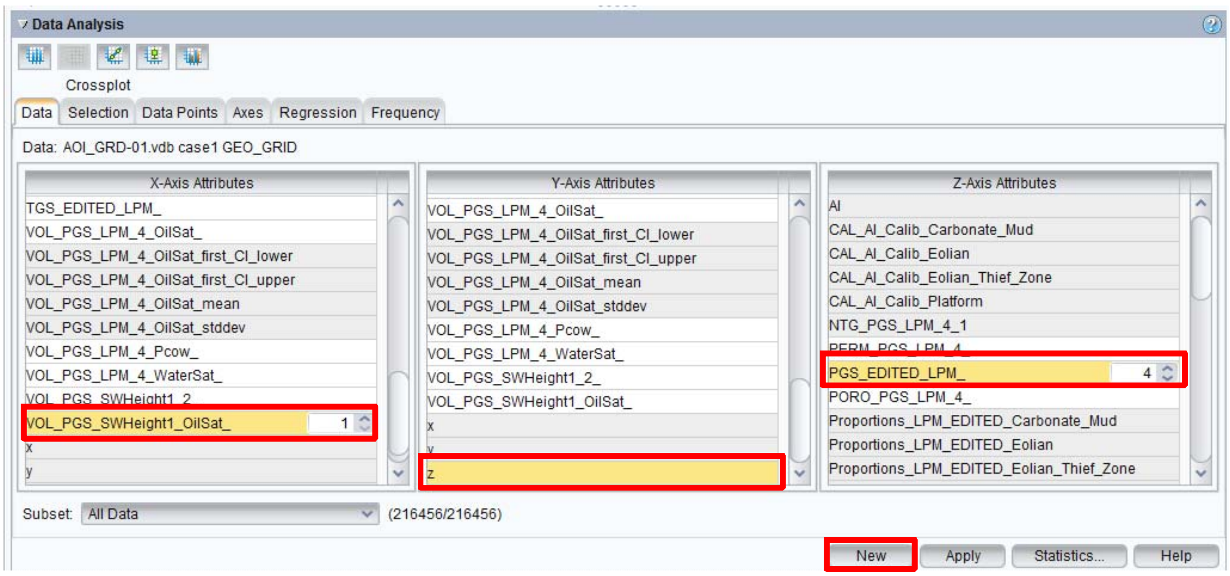


11. Go back to the Data tab and select **VOL_PGS_SWHeight_OilSat #1** for the X-Axis.

12. Select **Z** for the Y-Axis.

13. Select **PGS_EDITED_LPM #1** for the Z-Axis.

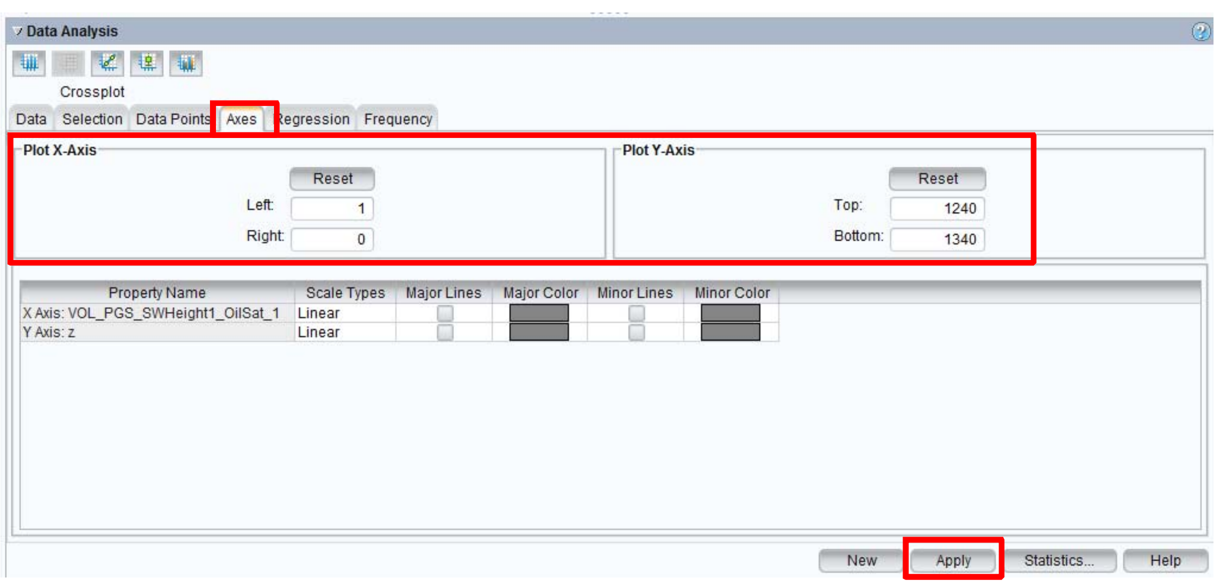
14. Click **New**.



15. Click the **Axes** tab in the Data Analysis panel.

16. Change the X-Axis and Y-Axis Plot values as shown in the following image.

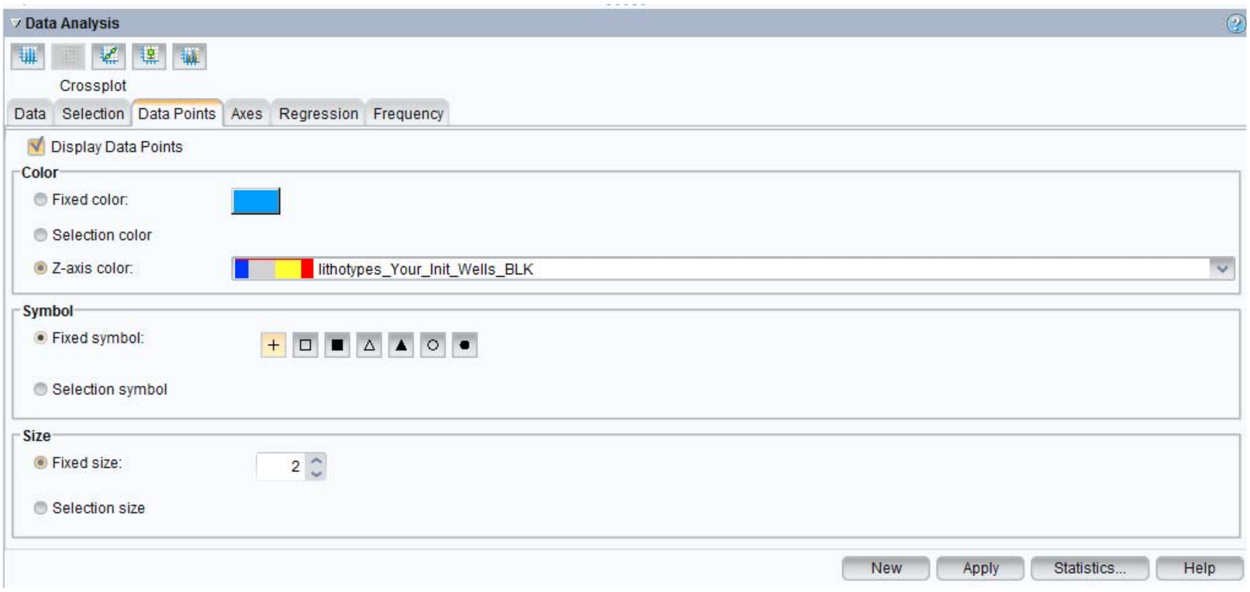
17. Click **Apply**.

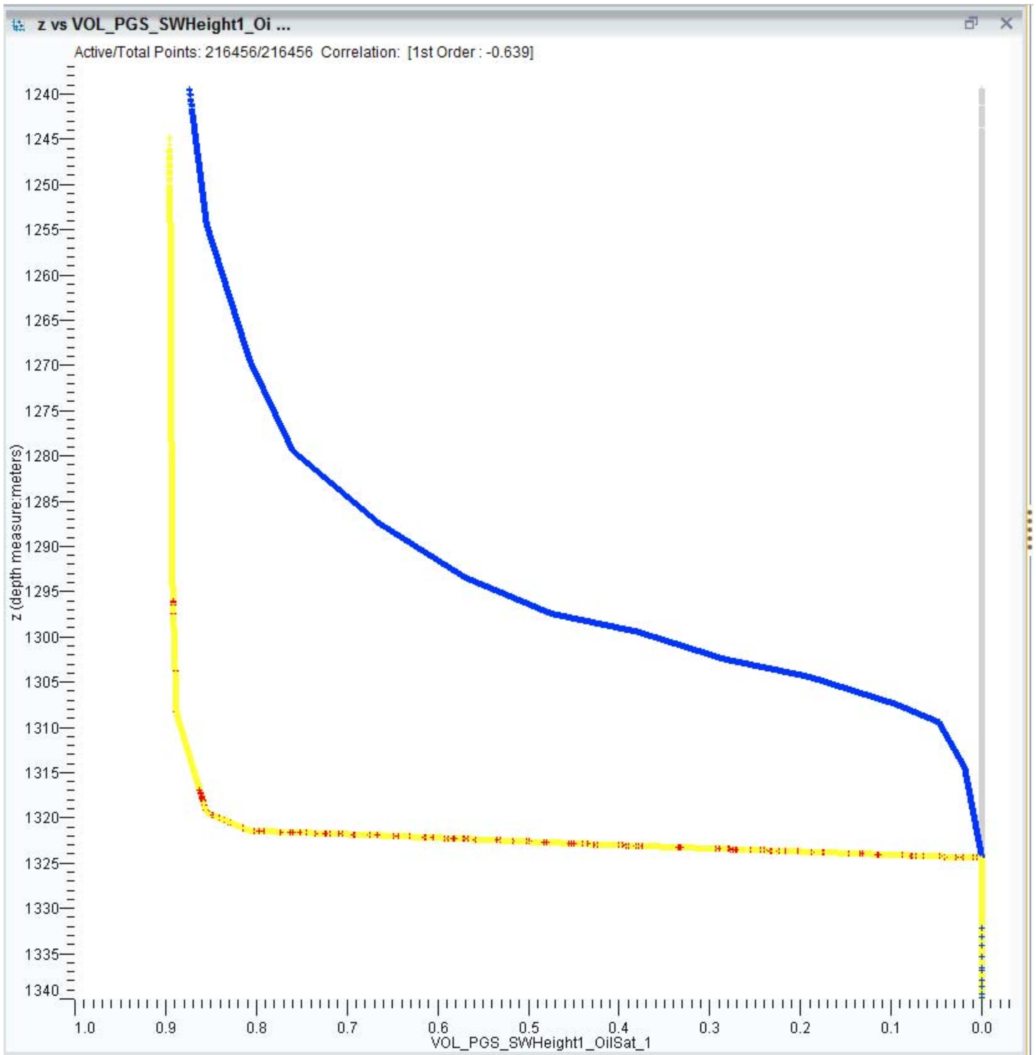


18. Click the **Data Points** tab in the Data Analysis panel.

19. Change the Z-Axis color to **lithotypes_Your_Init_Wells_BLK**.

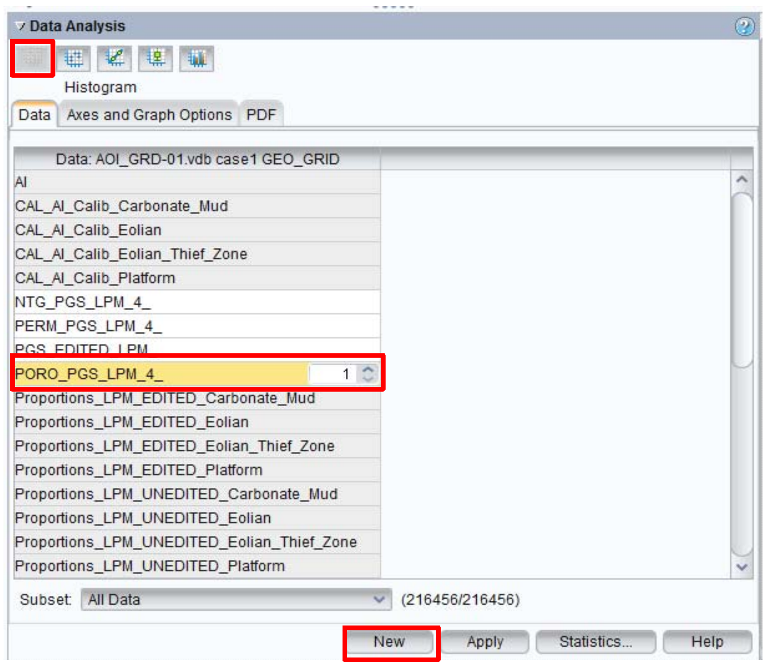
20. Click **Apply**.





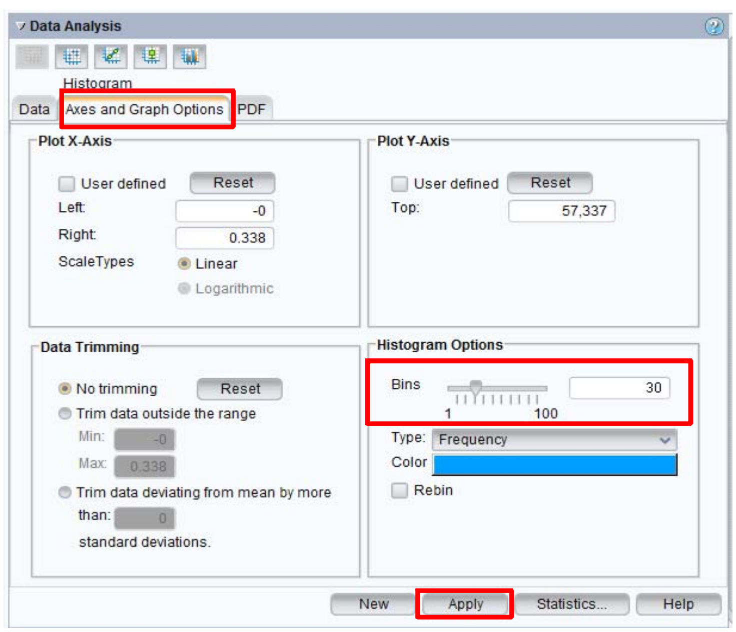
Cross-plot of: VOL_PGS_SWHeight_OilSat #1 for the X-Axis
Z for the Y-Axis
PGS_EDITED_LPM #1 for the Z-Axis

Create a Histogram of the PORO_PGS_LPM_4 Simulation #1 Data

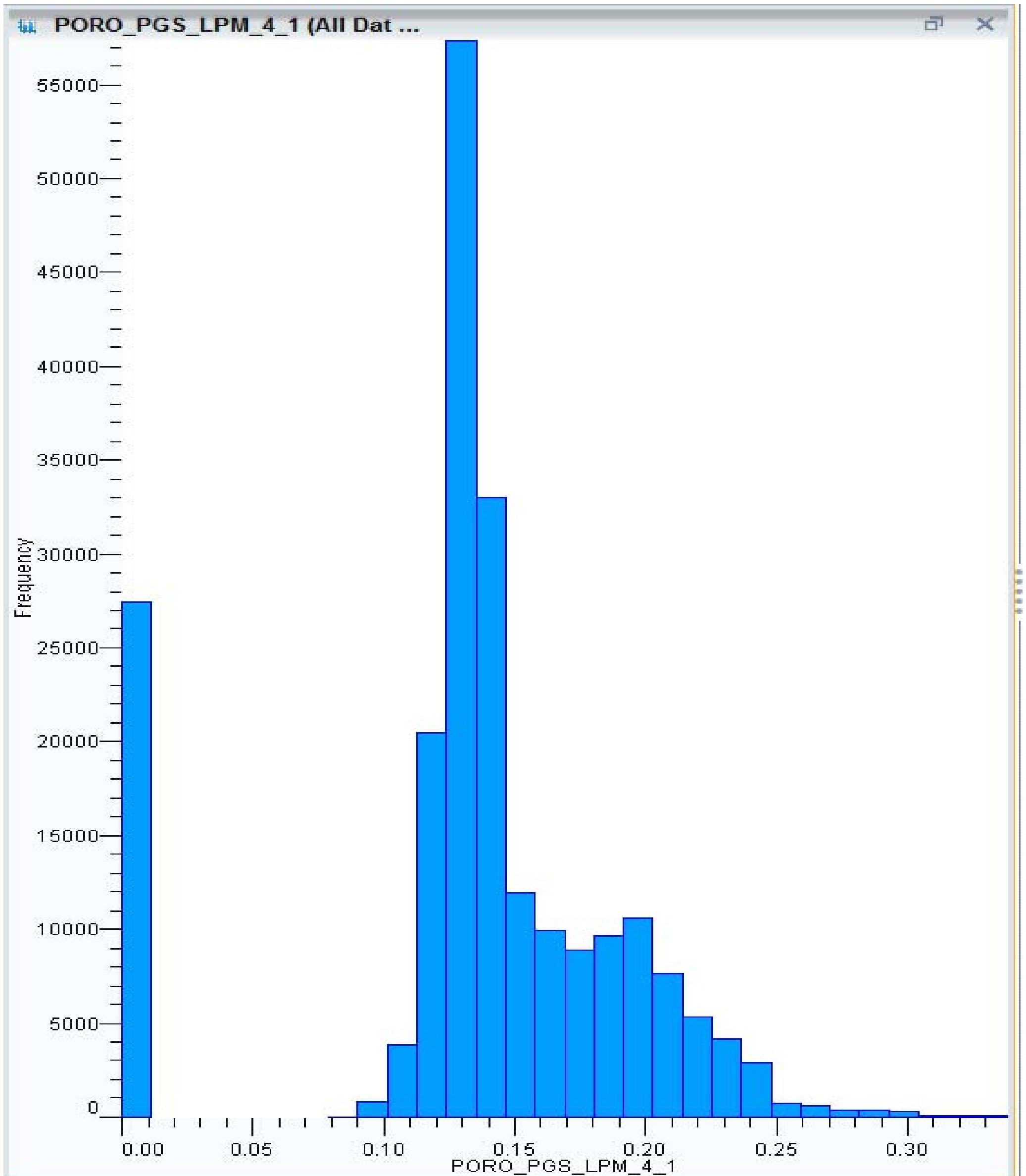


1. Click the **Axes and Graph Option**s tab and change the number of bins to **30** and check Rebin.

2. Click **Apply**.



The resulting Histogram should look like the image below.



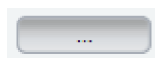
Uncertainty Analysis: Iso-cutoffs

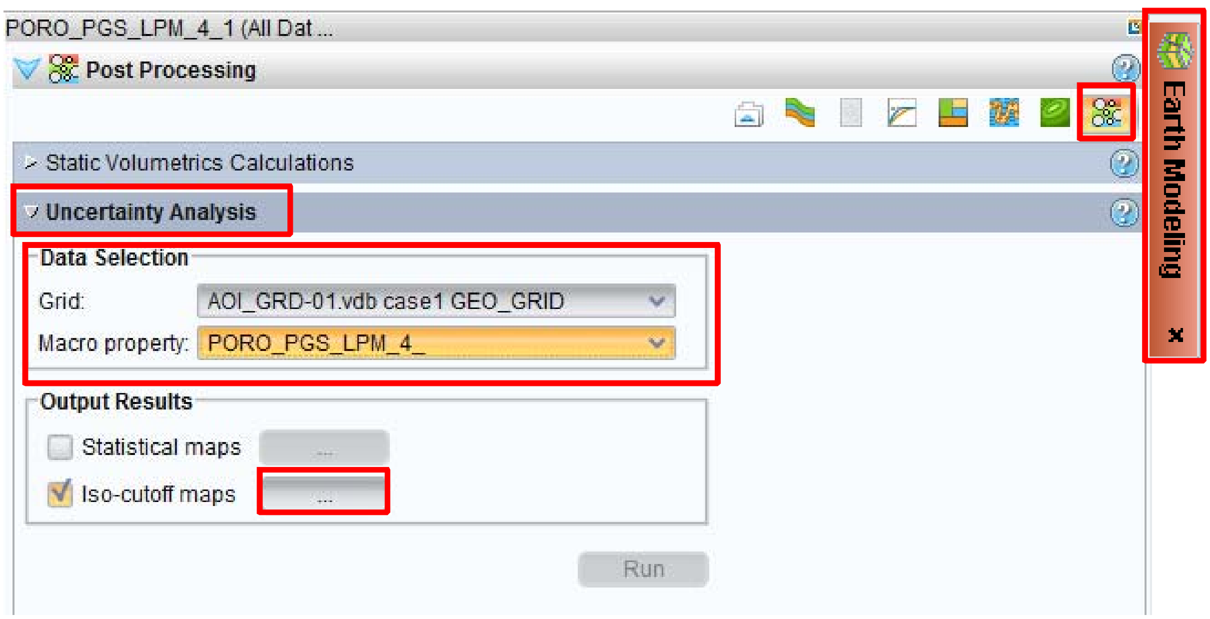
Now we are ready to generate some Iso-cutoff maps in the Uncertainty Analysis subtask pane of Post Processing.

1. Expand the Uncertainty Analysis subtask pane.

2. Enter Grid and Macro property values if not already set.

3. Check on the **Iso-cutoff maps** checkbox and check off **Statistical maps**.

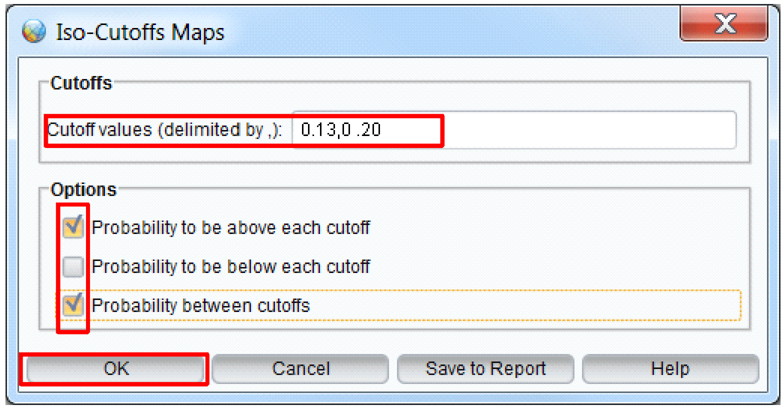
4. Click the  button.



5. Enter Cutoff values of **0.13** and **0.20** separated by commas.

6. Check ON **Probability to be above each cutoff** and **Probability between cutoffs**.

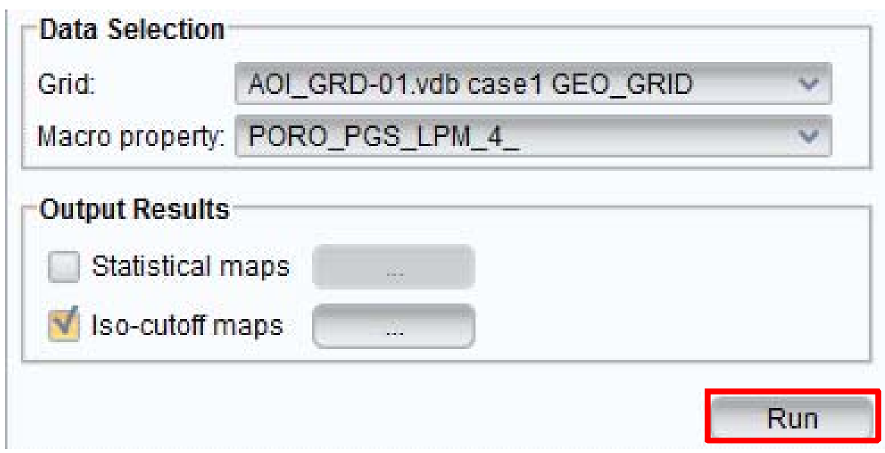
7. Click **OK**.

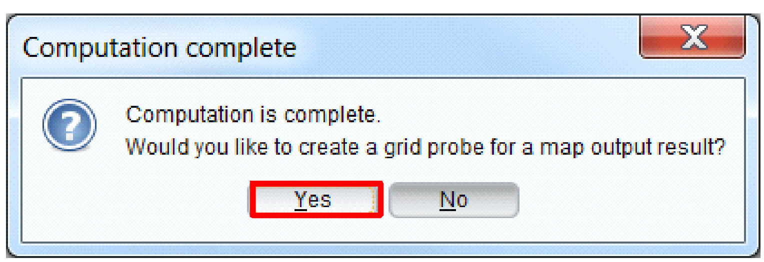


8. Select **PORO_PGS_LPM_4** for the Macro property.

9. Click **Run**.

10. Click **Yes** to Dismiss the Computation complete notification and to create a grid probe to view the output.



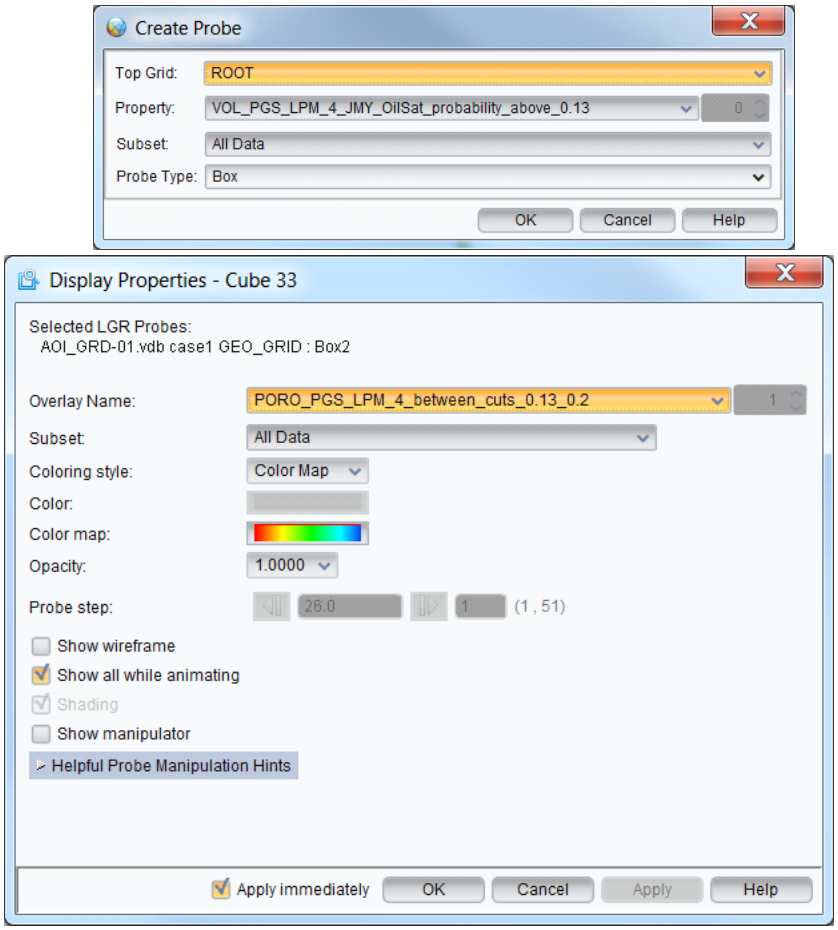


11. Click **OK** in the Create Probe dialog box.

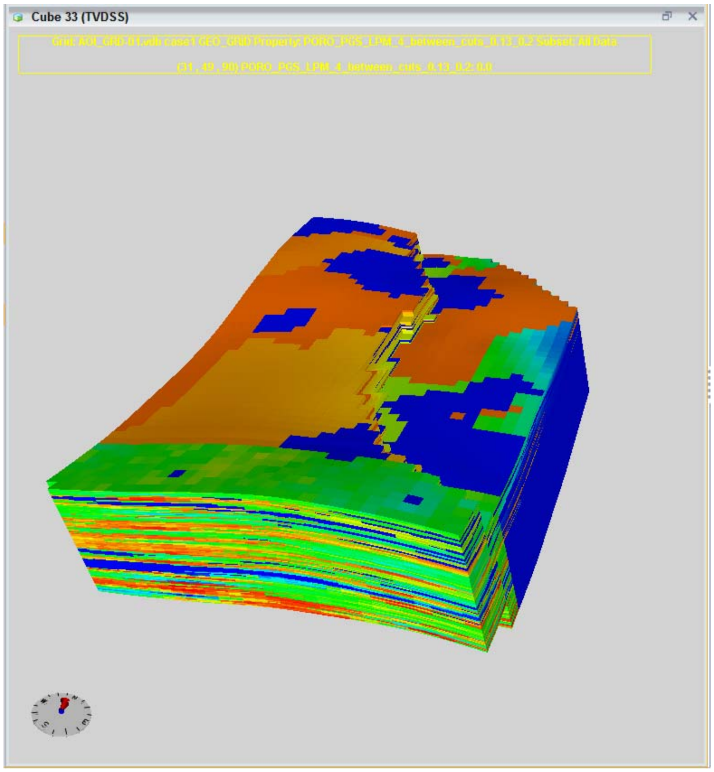
12. Use MB3 on the grid probe and select **Display Properties**.

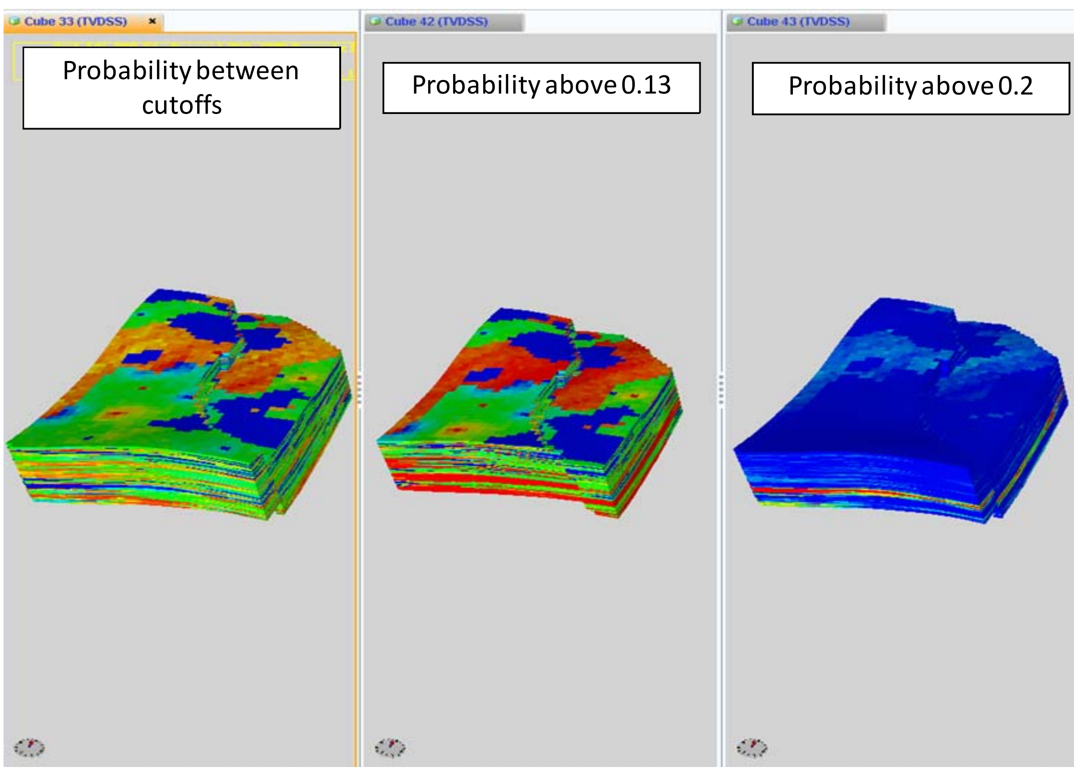
Display the Iso-Cutoff Results

13. Change the overlay to the **PORO_PGS_LPM cutoff between 0.13 and 0.2**.



Box probe with overlay set to the
PORO_PGS_LPM_4_between_cuts_0.13_0.2





Appendix A
Additional Exercises

Create a Project Design and Add to Report

In this exercise, you will create a project design to guide you through the 3D reservoir modeling steps and then save the project design as part of a report. Both of these activities are done using the Project Designer task pane, which contains two task panels: Project Designer and Reporting.

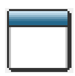
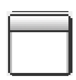
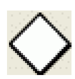
Project Designer

The Project Designer panel on the Project Designer task pane allows you to create a project design that is saved with your session. The project design appears as a diagram in a Project Designer tile in the display section of the DecisionSpace® window. The diagram can contain information on the tools used in the project design as well as activities performed, decisions made, and other text information that you want to save with the project design. Project designs are saved with the session; however, these saved project designs do not contain any status information or report items that may have been added from a Report so that when the session is restored, you will have only the data-less project design outline.

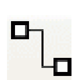
You can export a project design as a template, which can then be reopened in a new session. To export your project design as a template, click the icon on the Project Designer panel, enter a name for the template, and click **OK**. The template is saved as an .xml file to a Workflow Template directory parallel to where your sessions are saved. On Linux, the default is to [home directory]/Landmark/DecisionSpace/WorkflowTemplate. You can open a template for use in a session by clicking on the icon on the Project Designer panel to open the Create Project Design From Template dialog box, selecting the desired template from the list, and then clicking **OK**. You can enter a new name for the project design before clicking OK if desired. If you currently have an active project design, it will be replaced by the newly selected template; so you will be asked to confirm that you want to continue.

The Project Designer display tile is where you create the project design. The Tools used to create a project design are on the left side of the tile as shown below. Click and drag a tool to add it to the project design. You can drag a Tool anywhere within the Project Designer tile. Just click and hold the left mouse button (MB1) and drag the tool box. The Control buttons at the bottom allow you to move and resize the project design boxes, and connect the tools in the project design diagram in the desired order.

The Tools function as follows:

	Adds a Tool Project Design Element, which allows you to select and open an Earth Modeling or Data Analysis tool using the MB3 menu. The boxes are color-coded so that you can easily identify the module from which a task pane was selected. For Earth Modeling, the color at the top of the box will be light blue, and for Data Analysis, it will be a light yellow.
	Adds an activity box, which allows you to enter text describing the activity.
	Adds a decision node, which allows you to enter text describing the decision to be made and assigning Yes or No to a corner of the box from which you will connect other boxes depending on whether the answer is Yes or No.

The Controls function as follows:

	Changes the cursor to the double-sided cursor. When you click in a project design box, yellow handles are added to the corners of the box. Click and drag within the box to move the box. Click and drag a corner handle to resize the box. To move, resize, or apply MB3 actions to multiple boxes simultaneously, click and drag MB1 to draw a rectangle around the boxes. Yellow corner handles will then appear in all boxes within the rectangle. Click outside the rectangle to remove it.
	Activates connector mode. Red squares appear on the project design elements indicating all possible connector points. Click a connector point and drag to connect elements. You must click the move/resize icon to deactivate connector mode.

Each Tool Project Design Element has an icon in the top left corner that is the icon for the selected tool.

The second icon in each Tool Project Design Element and Activity box indicates whether the step has been marked Unassigned, In Progress, or Complete using MB3 (see MB3 menu options below) and can be any of the following:

	Indicates that the step has not been assigned; i.e., has not been started.
	Indicates that the step is in progress.
	Indicates that the step has been completed.

Each Tool Project Design Element and Activity box also contains an  icon, which is the report item counter. The number after the icon indicates how many report items from the Reporting panel have been added to this box. To add a report item from the Reporting panel, click the row in the Reporting table and drag it to the box in the project design. When a report item is added to a box that is currently marked as 'Not Assigned', the box is automatically changed to 'In Progress' (see MB3 options below).

The top right corner of each Tool Project Design Element and Activity box contains a button to collapse or expand all 'children'. All activity boxes and decision nodes between two Tool Project Design Elements are children of the preceding Tool Project Design Element in the project design. The button is activated when an activity box or decision node is connected to from an Tool Project Design Element. Clicking on  collapses the children into the Tool Project Design Element so that they are not displayed in the project design. Clicking on  expands the Tool Project Design Element to show its children.

In addition to the icons described above, an Activity box contains a number in the top left corner if it is connected to a Tool Project Design Element. This is a 'level' number and indicates how far that box is away from its parent Tool Project Design Element. When collapsing an Activity box, all Activity boxes and Decision nodes following it in the project design that have an equal or lower level number. Also, these levels are used to assign outline levels in Reports.

MB3 menu options differ depending on where you click. Clicking MB3 inside a tool project design box accesses the following menu options:

- [*Tool Project Design Element only*] **Assign Task Pane**, which accesses sub-menus for Data Analysis and EarthModeling listing the task panes. Click a task pane name to add it to the tool box.

- [*Tool Project Design Element only after a Data Analysis or EarthModeling tool has been added*] **Open task pane** opens the selected tool in the task pane on the right.

- [*Tool Project Design Element only*] **Report** opens a report containing all report items within that project design box or combining all report items if multiple boxes are selected. This option appears only after a report item has been added by clicking on it in the Reporting panel and dragging it to the box. Report items are formatted in a report using the levels as described above.

- [*Tool Project Design Element only*] **Clear All Report Items** removes all report items from the box. This option appears only after more than one report item has been added to the box.

- [*Tool Project Design Element only*] **Clear Last Report Item** removes the last report item from the box. This option appears only after a report item has been added to the box.

- [*Tool Project Design Element only*] **Add Note** opens the Notes Editor, which allows you to add a note, with attachments if desired. The note is automatically added as an item in the Reporting table.

- [*Tool Project Design Element only*] **View Notes** opens a Notes box listing the available notes and report items for the selected tool box.

- [*Tool Project Design Element only that has been collapsed*] **Hide** appears only after a Tool Project Design Element has been collapsed and allows you to then hide that tool box in the project design. To restore the box to the view, click MB3 > Expand All.

- [*All tool boxes except Tool Project Design Element*] **Enter text** opens a dialog box allowing you to enter the desired text.

- [*Decision Mode tool boxes only*] **Assign Yes to** a connector corner in a Decision Mode tool box to indicate the direction to proceed if your decision is Yes.

- [*Decision Mode tool boxes only*] **Assign No to** a connector corner in a Decision Mode tool box to indicate the direction to proceed if your decision is 'No'.

- **Select with children** adds yellow corner handles to the parent and Tool Project Design Element its children so that operations are performed on all simultaneously. Click the background outside of the selected boxes to deselect.

- **Center** moves the tool box to the center of the Project Designer space.

- **Remove** removes the tool box or connector from the project design.

- **Clone** copies the selected tool boxes and pastes them in the upper left corner of the Project Designer tile, retaining the relative positions and connections of the originally selected tool boxes. If multiple tool boxes are selected and you click one of them, they all remain selected and are all copied and pasted. If multiple tool boxes are selected, but you click an unselected tool box, the previously selected tool boxes are deselected and only the tool box you just clicked is selected, copied and pasted.

- **Complete** colors the tool box green to indicate that this step has been completed in the project design.

- **In Progress** colors the tool box blue to indicate that this step has been started but not completed.

- **Not Assigned** leaves the tool box colored white indicating that this step is neither Complete nor In Progress.

Clicking **MB3** in the background (outside the project design boxes) accesses the following menu options:

- **Background color** opens a Color dialog allowing you to select a color for the Project Designer tile background.

- **Enable conferencing** causes the Project Designer tile to be rendered in such a way that it is visible in commercial conferencing applications.

- **Collapse All** collapses all children into their parent Tool Project Design Element.

- **Expand/Show All** expands all Tool Project Design Elements to show their children and displays any hidden elements.
- **Reset View** centers the elements in the view.
- **Format** is enabled only if more than one tool box is selected. It accesses a sub-menu allowing you to select from the following options:
  - **Size** allows you to resize all selected tool boxes to match the Widest, Narrowest, Tallest, Shortest, Largest, or Smallest.
  - **Align** allows you to align all selected tool boxes Left (along their left sides), Along Vertical (a vertical line through the middle of the boxes), Right (along their right sides), Tops (aligns the tops of the boxes), Along Horizontal, (a horizontal line through the middle of the boxes), or Bottom (aligns the bottoms of the boxes).
- **Snap to Grid** accesses a sub-menu allowing you to select from the following:
  - **Off**, which turns off all snap to grid functionality.
  - **Snap Upper Left Corner to Grid**, which snaps the upper left corner of each tool box to a default Project Designer tile grid line. The centers will generally align to create straight connector lines.
  - **Snap Center to Grid**, which snaps the center of each tool box to a default Project Designer tile grid line. This option will create straight connector lines.
- **Connector Width** allows you to change the width of the connector lines between the boxes using the drop-down list. The change is made automatically as you move your cursor over the list.
- **Arrow Size** allows you to change the size of the connector arrows using the drop-down list. The change is made automatically as you move your cursor over the list.

- **Clone** is enabled only if one or more tool boxes are selected. It copies the selected tool boxes and pastes them in the upper left corner of the Project Designer tile, retaining the relative positions and connections of the originally selected nodes. Selected nodes are indicated by yellow corner handles. Click a tool box to select it, use Ctrl+click to select multiple tool boxes, or click and drag to select a group of tool boxes all at once.

Clicking **MB3** on a connector line accesses the following menu options:

- **Edit Color** opens a Color dialog allowing you to select a color for the connector line.

- **Center** resets the project design display so that the connector is in the center.

- **Remove** removes the connector.

Use MB1+MB3 drag (or your mouse wheel) to zoom in or out on the project design. Use MB2+drag to move the entire project design within the tile. Use MB1+drag to draw a box around portions of the project design, which you can then move using MB2+drag.

After creating a project design, you can add it to a Report as described in the following section.

Reporting

The Reporting Panel allows you to create a report of work done on a project. As you work with the EarthModeling task panes, Command type report items are added to the Report. Report mode, activated by clicking on the Add to Report ( ) icon on the DecisionSpace® Desktop toolbar, allows you to add a snapshot of any task panel, dialog box (note that if Report mode does not work for a particular dialog box, then the dialog box will have a Save to Report button that you can use instead), or display in a tab view to a report simply by clicking on the Add to Report ( ) icon on the DecisionSpace® Desktop toolbar to activate Report mode.

When in Report mode, the cursor appears as a camera and the tab view or task panel under your cursor is highlighted in blue to show the area that will be captured. Click the task panel, dialog box, or tab view to capture it. You will see a shutter action indicating that the snapshot has been taken.

To exit Add to Report mode, you can use any of the following methods:

- Click the icon again
- Click MB3 in the area highlighted in blue and select Exit Report Mode
- Click the Esc key on your keyboard

The captured snapshot appears as a row (report item) in the table on the Reporting panel of the Project Designer task pane. You can then create a report with a default or user-created stylesheet and using all or only filtered report items.

Reports are saved when you save the session. By default, they are saved as .xml files for each user to /OW_PROJ_DATA/[project]/[interpreter]/Report, where the location of OW_PROJ_DATA is specified in your owdir.dat. Reports are project and interpreter dependent, not user or session dependent. Therefore, when you start a new session, you will see all reports created using the same project and interpreter that you selected for your session.

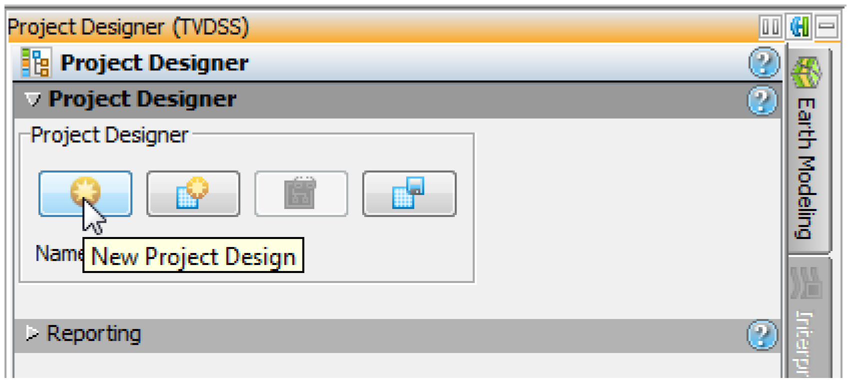
The Report Data table at the top of the Reporting panel shows the **Title** of all report items that have been sent to the report along with the **Type** and the **Date** it was created. You can use the icons above the table to search/filter and arrange the table, select rows, and delete rows in the table. If you delete a row in the table for which the report item has previously been added to a Project Designer tool box, it is automatically removed and the item count for the tool box is updated. **Hide Commands** hides all Command items in the table. This is checked 'on' by default so that all Command items that were automatically created in previous sessions using the same project and interpreter do not appear in the table. To view all Command items, click the checkbox to remove the check mark.The **Note** column allows you to add a note, with attachments if desired, to the report item using the Notes Editor. To open the Notes Editor, click the icon in the Note cell for the report item you want to add the note to.

The button in the **Info** cell for each row opens that a Report Item Information window for that report item using the selected style sheet. You can add a report item to an Tool Project Design Element or an Activity Box in a project design by clicking on the row in the table and dragging it to the box. The report item counter will increase by one. A Report menu option is also added to the MB3 menu for the project design box, which opens a report for that report item.

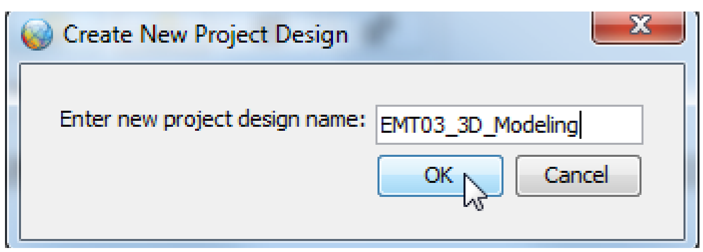
Steps

1. Click the **Project Designer** tab on the right side of the DecisionSpace window to open the task pane and then click the **Project Designer** panel to open it.

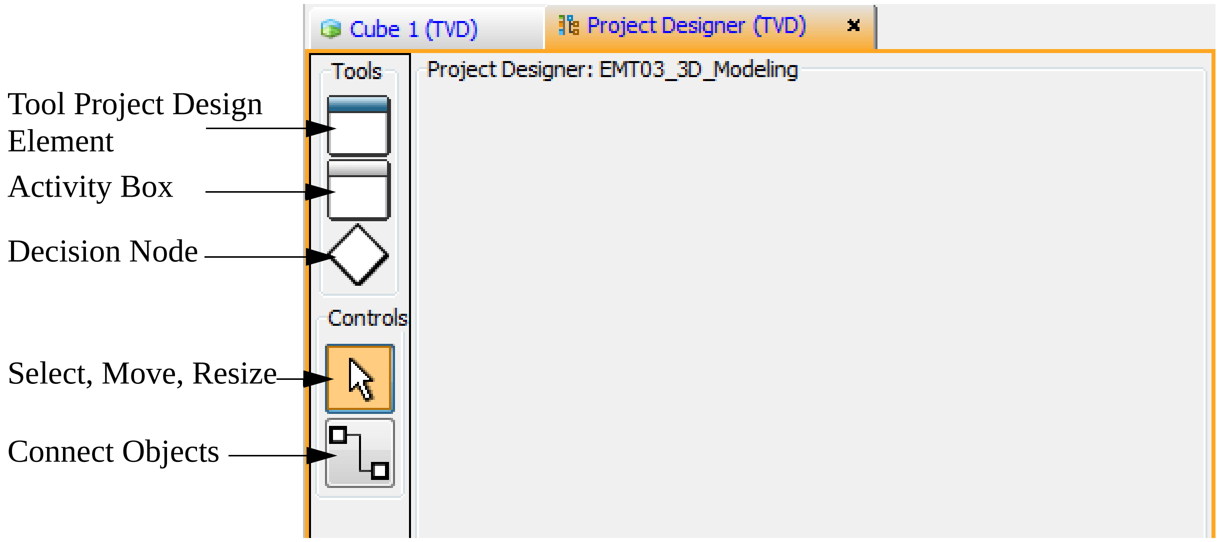
2. At the top of the Project Designer panel, click the button.



3. In the Create New Project Design dialog box, enter the name as EMT03_3D_Modeling and click **OK**.



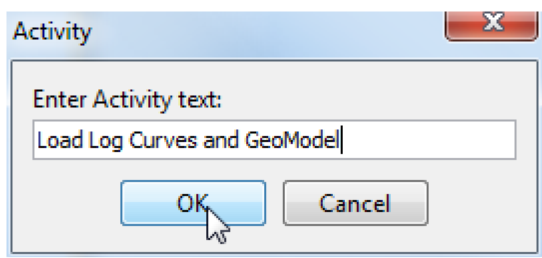
A Project Designer tab appears in the view space. The Tools panel on the left contains the elements that you can use to create the project design.



4. Drag an **Activity Box** into the project design.

5. Click MB3 in the box and select **Enter text**.

6. In the Activity box that appears, enter the text as shown below, and then click **OK**.



If all of the text does not appear, click MB1 in the box, and then click a corner handle and drag to resize the box.

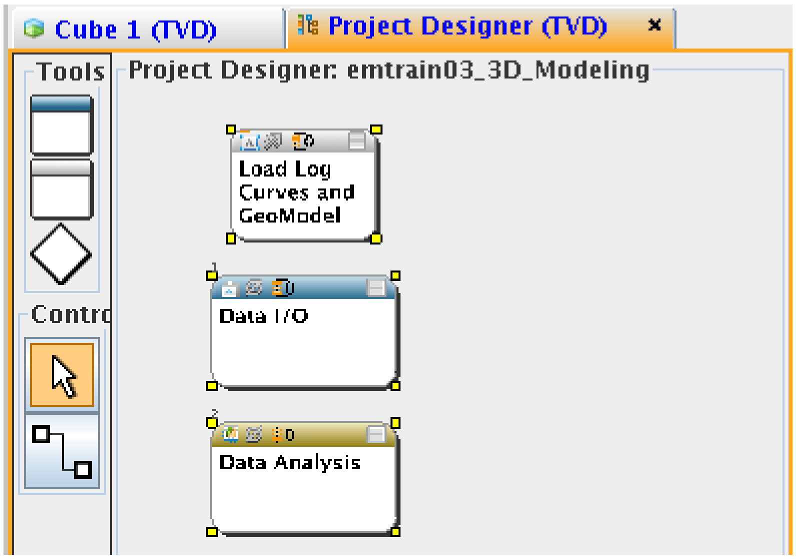
7. Drag two **Tool Project Design Elements** into the project design.

If you click MB3 in the Project Designer background (outside of a box), you will see the option **Snap-To-Grid**. Place your cursor over this option to access a drop-down menu with three options. Currently, Snap Upper Left Corner to Grid is enabled. This is why when you drop the boxes into the project design, they snap to a default grid. To arrange boxes and connector lines without snapping, check **Off**.

8. Click MB3 in the first box and select **Assign Task Pane** > **EarthModeling** > **Data I/O**.

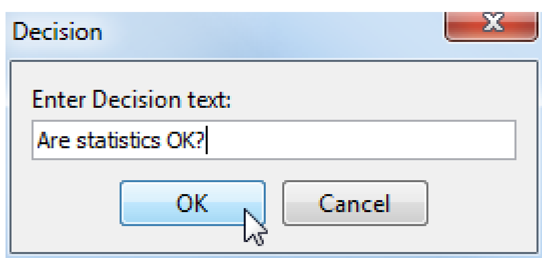
9. Click MB3 in the second box and select **Assign Task Pane > Data Analysis > Data Analysis.**

   At this point, your Project Designer tile should appear as shown below.



   You can use MB1+drag to draw a rectangle around all boxes so that they are selected and then use MB1+drag on a box to move all boxes within the Project Designer tile. Also, with all selected, you can use MB1+MB2, or the middle mouse button alone if it is a scroll wheel, to zoom in or out.

10. Drag a **Decision Node** into the project design below Data Analysis and add the text as shown below.



   As you create new boxes and connect them, you will probably want to turn off Snap-To-Grid so that you can realign the boxes.

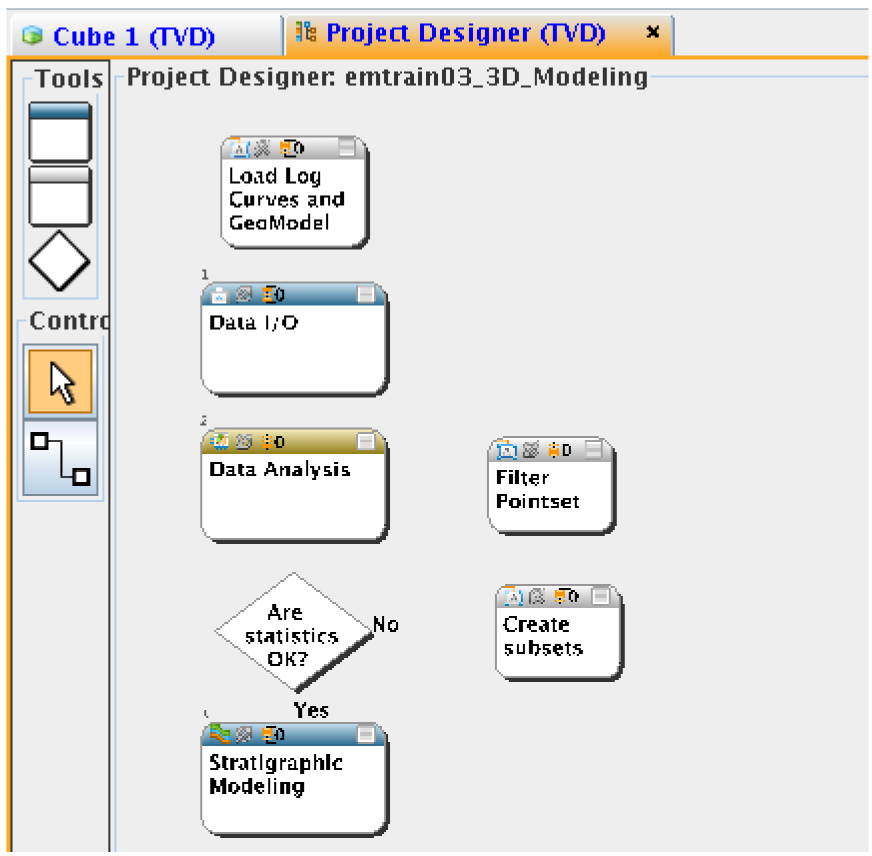
   Use MB3 to add a **No** to the right and a **Yes** to the bottom of the Decision Node.

11. Add an **Activity Box** to the right of the Decision Node and enter the text Create subsets. This is because, if the answer to the question above is No, you want to use the Subset Manager to create subsets and then analyze the subsets.

12. Add another **Activity Box** above the box that you added in the previous step and enter the text Filter Pointset. For this step, you will use the Filter Pointset panel in Data I/O to create a new clean pointset using the subsets.

13. Add another **Tool Project Design Element** below the Decision Node and select **Assign Task Pane** > **Earth Modeling**> **Stratigraphic Modeling**.

    At this point, your Project Designer tile should appear as shown below.



14. Continue adding boxes as shown in the project design in the next step.

    You can align boxes by using MB1 drag to draw a rectangle around a group of boxes and then using the MB3 > Format > Align options to align them as desired.

15. Then click **Connect Objects** on the lower left side of the project design so that it is selected. Red squares appear on each side of the project design elements.

    Use MB1+drag to connect the project design boxes as shown below. You can use MB3 options to remove or change the direction of an arrow.

Create Transforms

The Data Transform panel on the Data I/O task pane of Earth Modeling allows you to transform the data distribution of the selected input data to another type of distribution; the purpose being to approximate a normal distribution in this exercise. This transformed data is then available in Property Modeling. The coefficients of the transform are saved to allow interpolated or simulated results of continuous property data to be back-transformed.

Transform Type allows you to select from the following types (note that this list will be filtered so that only those transform types that can be used for selected Property are shown; for example, if the data has been transformed, only the Back_[transformtype] option will be listed):

- **Normal_Score** transforms the input data distribution into a normal (Gaussian) distribution, which is required when performing simulations. The sill (variance) for normal score transformed data is always one.

> **When to Use Transformed Data**
>
> Normal Score transformation is a prerequisite for conditional simulations. However, Normal Score transformed data should never be used when doing Interpolation in Property Modeling. The Logn, Log10, and Standardize transforms are valid for Interpolation, but not for Simulation.

- **Logn** (natural log) transforms the input data distribution into a logarithm to the base n, where n is an irrational constant approximately equal to 2.718281818459, or in other words the power to which a number x would have to be raised to equal x.

- **Log10** transforms the input data distribution into a logarithm to the base 10 distribution. This is typically used to look for permeability/porosity relationships when comparing them in cross-plots.

Using Log Transforms

Log transforms are typically used to look for permeability/porosity relationships by comparing them in cross-plots. Because porosity data by nature follows a normal distribution and permeability data does not, comparing transformed permeability data against untransformed porosity data in a cross-plot should show a linear correlation.

Log transforms are used in kriging or interpolation where you do not want to use normal score transformed data.

- **Standardize**, sometimes called Z-score conversion, rescales the x axis to a mean of zero and a standard deviation of 1. The resulting histogram will look like the histogram of the original data, the only difference being the X-axis values. A cross-plot of the standardized data versus the original data will show a perfect correlation.

  Standardized data are useful in looking for outliers.

- **Range** rescales the data from 0 to 1. This transform maintains the property relationships and rescales the variances. The output property can be used to grid using interpolation algorithms, particularly in collocated cokriging where the variance ratios could get extreme.

- **Back_Normal_Score** is used to return data that has been transformed to a Normal_Score distribution back to its original distribution.

- **Back_Logn** is used to return data that has been transformed to a Logn distribution back to its original distribution.

- **Back_Log10** is used to return data that has been transformed to a Log10 distribution back to its original distribution.

- **Back_Standardize** is used to return data that has been transformed to a Standardize distribution back to its original distribution.

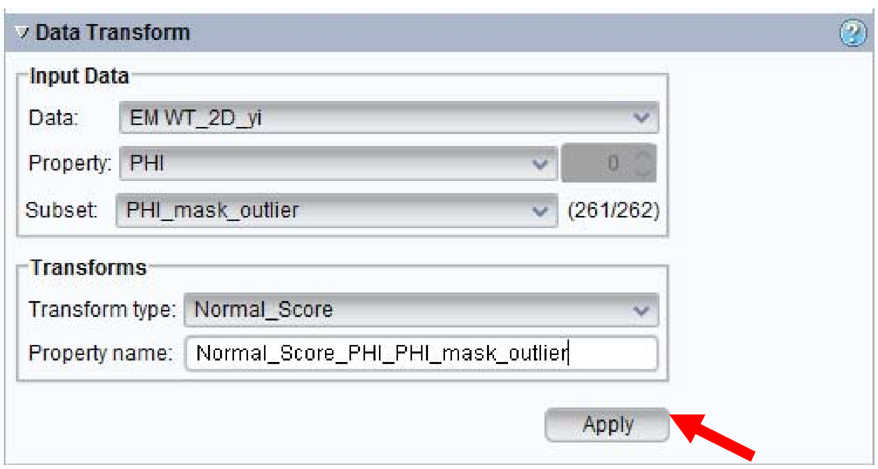
Property name is the name of the property output to the pointset.

Apply transforms the data using the selected type and writes it out as a new property using the Property name that you provide. This Property will then be available from the drop-down list in Property Modeling. *Note*: if you click Apply again using the same parameters, a popup will ask if you want to overwrite the preexisting property. Click 'Yes' to overwrite. Click 'No' to create a new property using the same property name but with a number appended (for example, Normal_Score_PHI_All Data_1).

Because you will be doing a conditional simulation, you need to transform the porosity into a normal distribution using the Normal Score Transform. You could do the transform in Variogram Modeling but for this exercise you will use the Data Transform panel in EarthModeling.

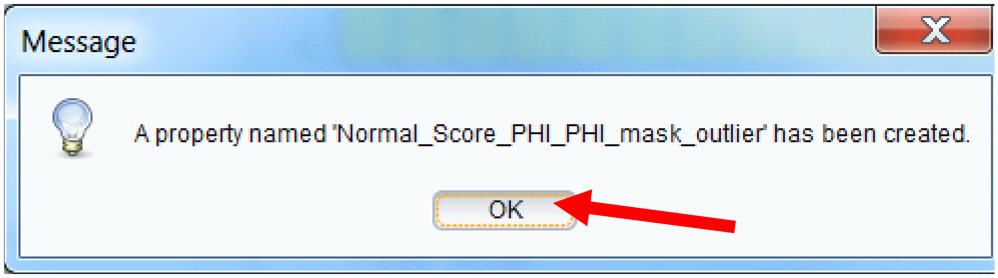
Steps

1. Click the **Earth Modeling** tab to re-open it. The Data I/O task pane should still be open. Click the Import Points panel header to close the panel.
2. Click **Data Transform** to open the panel.
3. Set the parameters as shown below and then click **Apply**.

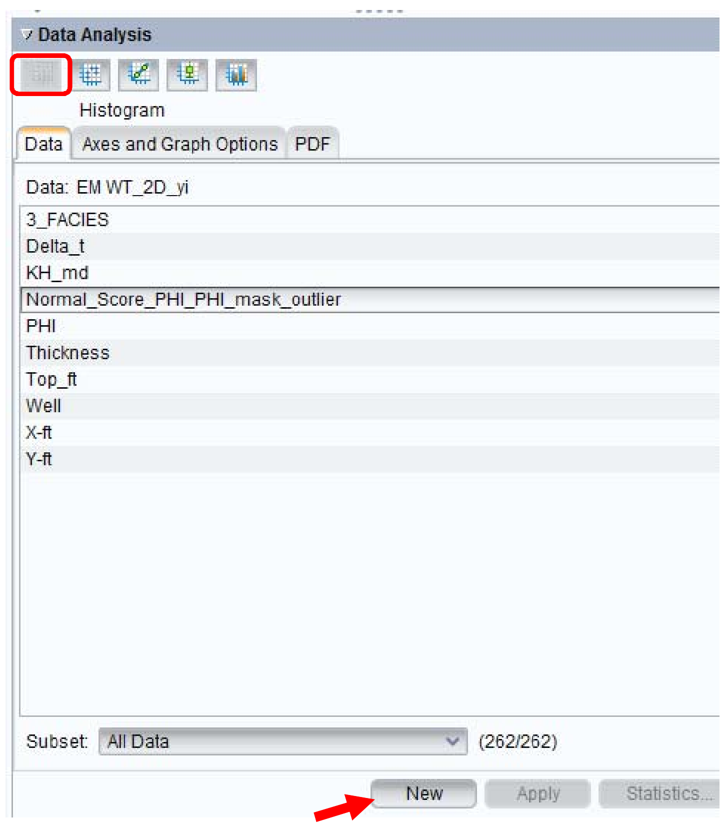


A message dialog box should appear indicating that a new property was created.

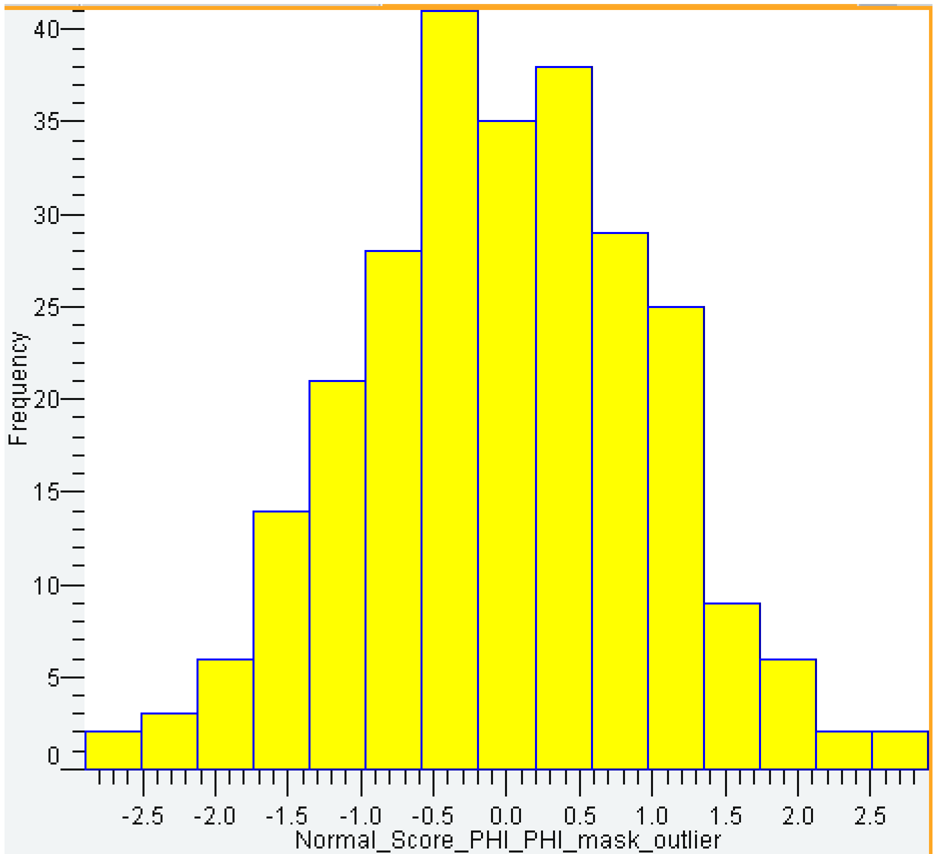
4. Click **OK**.



5. Go back to the Data Analysis task pane and display a histogram of **Normal_Score_PHI_PHI_mask_outlier**.

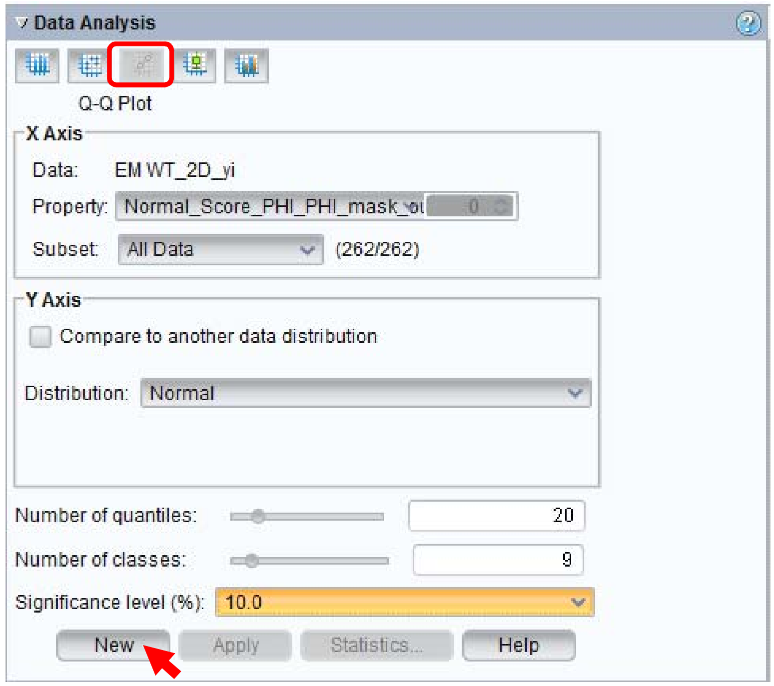


Your histogram should appear as shown below.

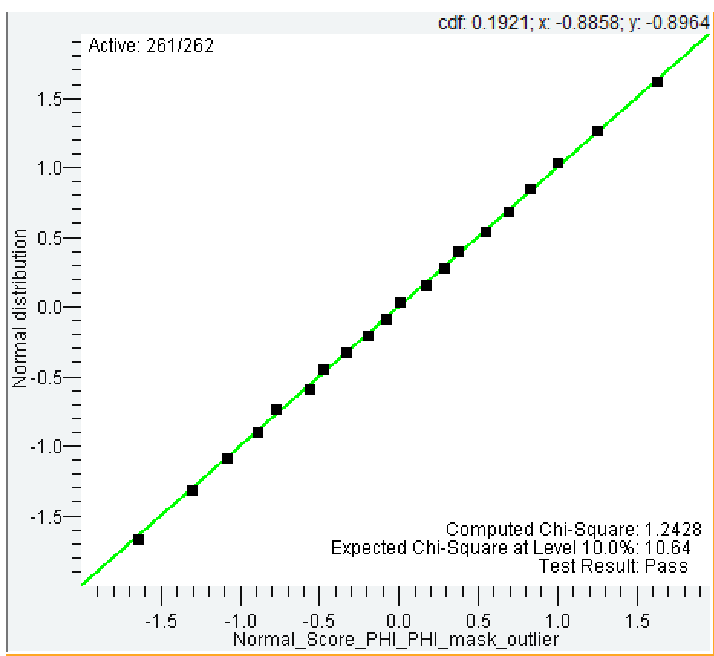


This histogram shows a normal distribution. You can see that the units are in terms of standard deviation. The mean value is 0 and X axis values are now in terms of standard deviation (0.5, 1.0 etc.).

6. Display a **Q-Q Plot** to see that it falls along the perfect correlation line.



Even though you are selecting the subset here as All Data, when you display the Q-Q Plot, because the mask outlier subset was active when you perform the normal score distribution, the resulting Q-Q Plot only has 261 active data points.



With Data Analysis and Data Transform completed, you are now ready to do Variogram Modeling. Before proceeding to the next exercise though, save the session.

7. Select **File > Save Session as...** on the DecisionSpace menubar.

8. In the Save Session As dialog box, enter the **Session Name** as [your_initials]_2D_thru_da.

9. Click **Save**.

10. You can remove any data analysis display tabs before continuing.

Exercise — Collocated Cokriging

In the previous exercises, you learned how to apply a geostatistical interpolation technique to a single attribute. You will now use a secondary variable in the kriging process by using the Collocated Cokriging technique with permeability (KH) as the primary variable and the previously gridded porosity (PHI) as the secondary variable.

Steps

1. On the Interpolation panel, set the following parameters:

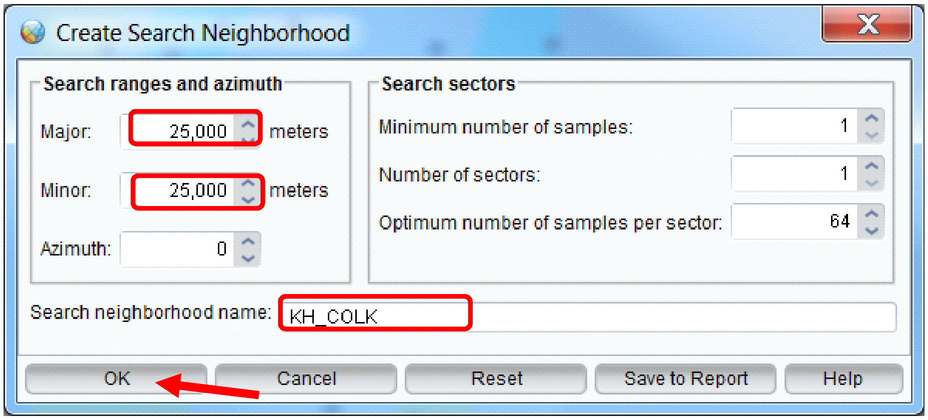
   - Property: **KH_md**
   - Interpolation type: **Collocated Cokriging**
   - Collocated input property: **PHI**
   - Collocated grid property: **PHI_Ordinary_Krig**
   - Variogram: **PHI_nested_aniso**

   > **Note:**
   >
   > We use the variogram from the secondary data as it typically has many more data values than the primary data. In this example, we borrow the variogram model for KH_md from PHI. The PHI variogram is also used to create the cross-variance model between KH_md and PHI using the correlation coefficient as the sill.

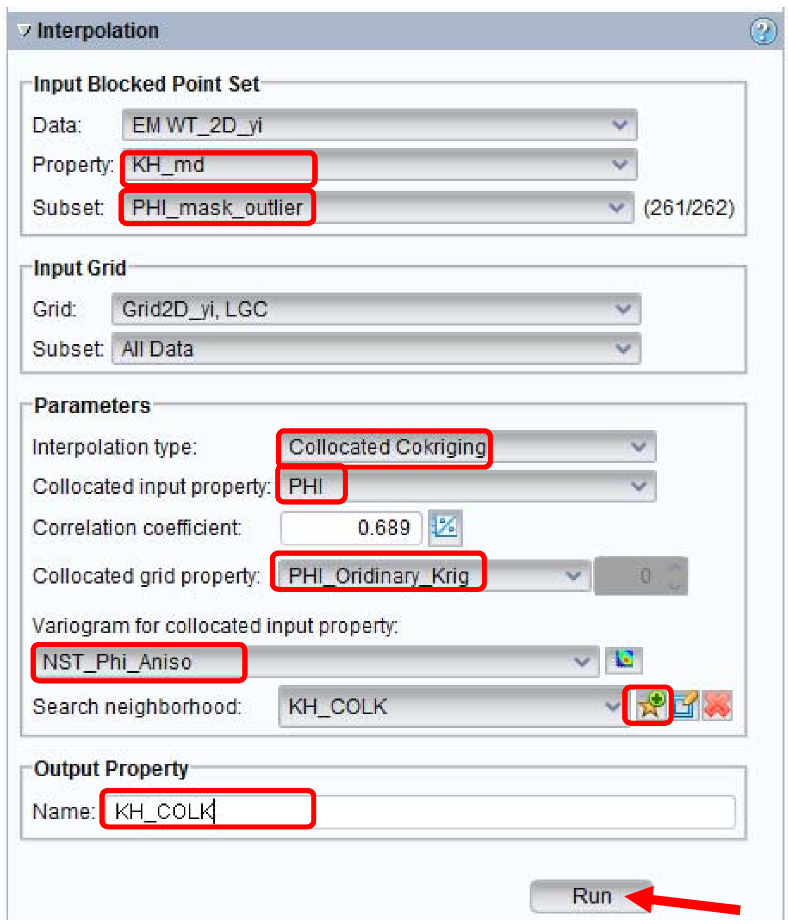
2. Create a Search Neighborhood using the following parameters:

   - Major: **25,000**
   - Minor: **25,000**
   - Optimum number of samples per sector: **64**
   - Search neighborhood name: **KH_COLK**

3. Enter the Output Property **Name** as **KH_COLK**.

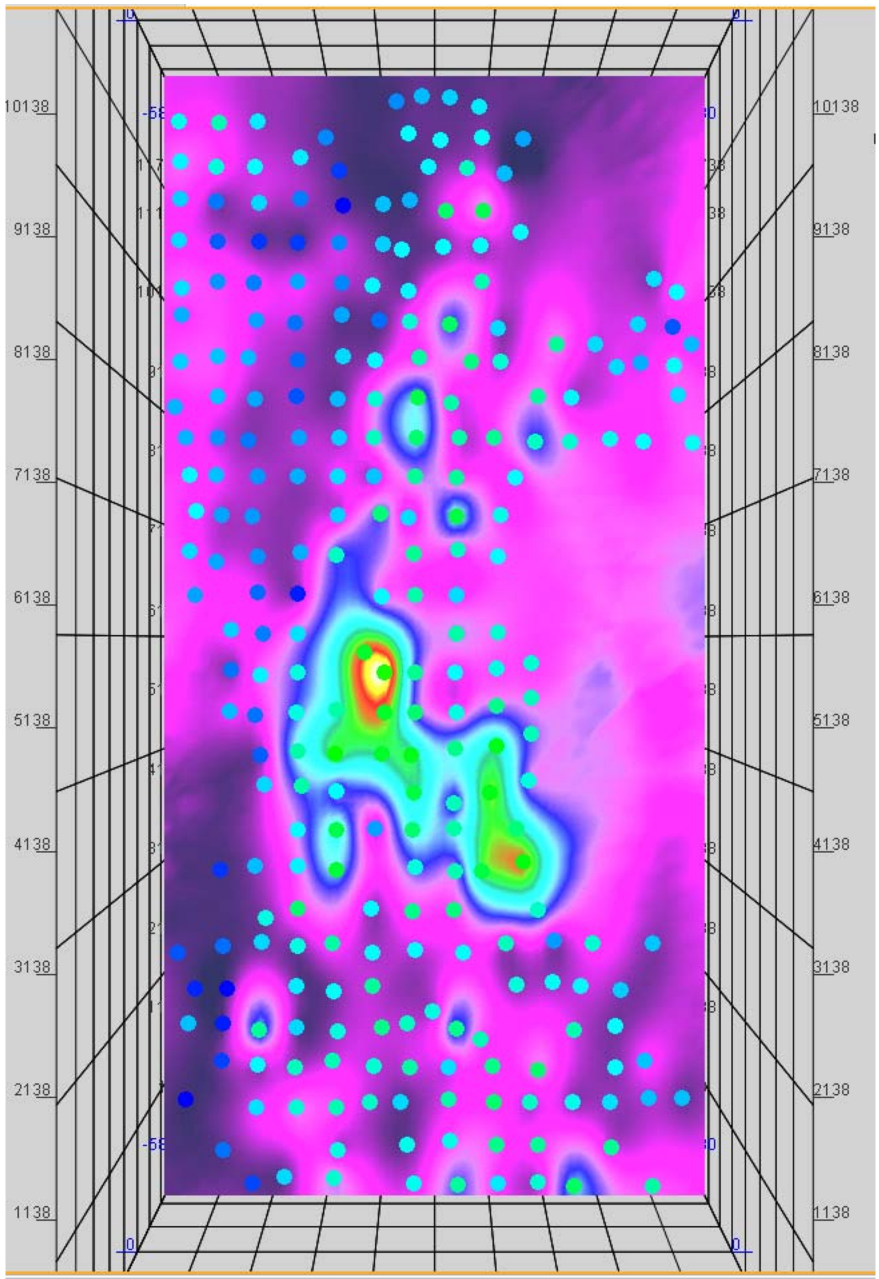


4. Click **Run**.



5. Using the Display Properties dialog, display the property **KH_COLK**.

Your display should appear approximately as shown below.



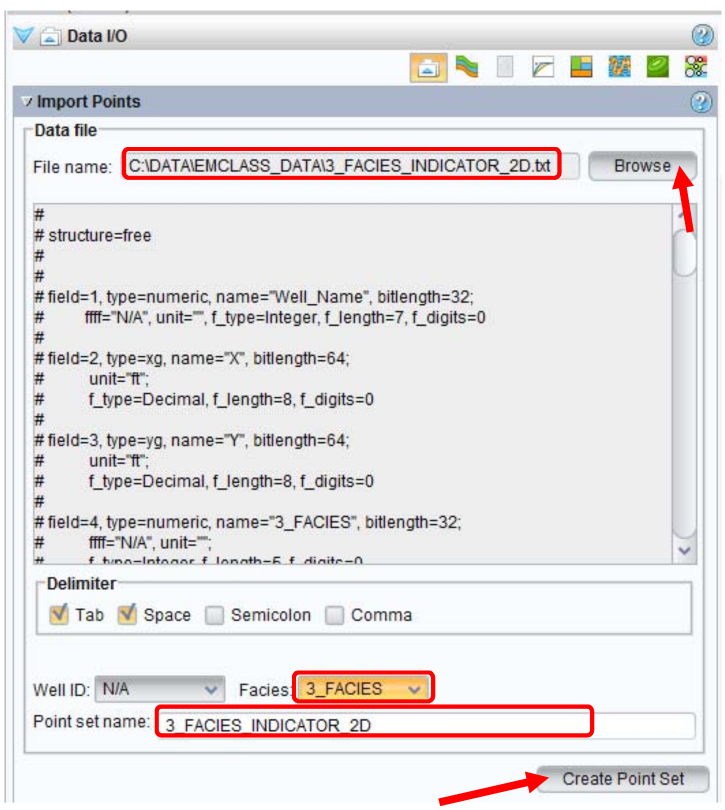
Indicator Kriging

In this exercise, you create a pointset from a flat file containing an indicator, do Data Analysis to determine the most frequently occurring facies, create a variogram model, create a 2D grid, do Indicator Kriging, and then display the output property on the grid.

The environment in the data you will use has a rotation of approximately 30 degrees north of east and is a bar system. It has a major scale of approximately 3500 meters and a minor scale of 1500 meters. This will not be clear from the experimental variogram, which is typical, and stresses the fact that you must know the geology of your data.

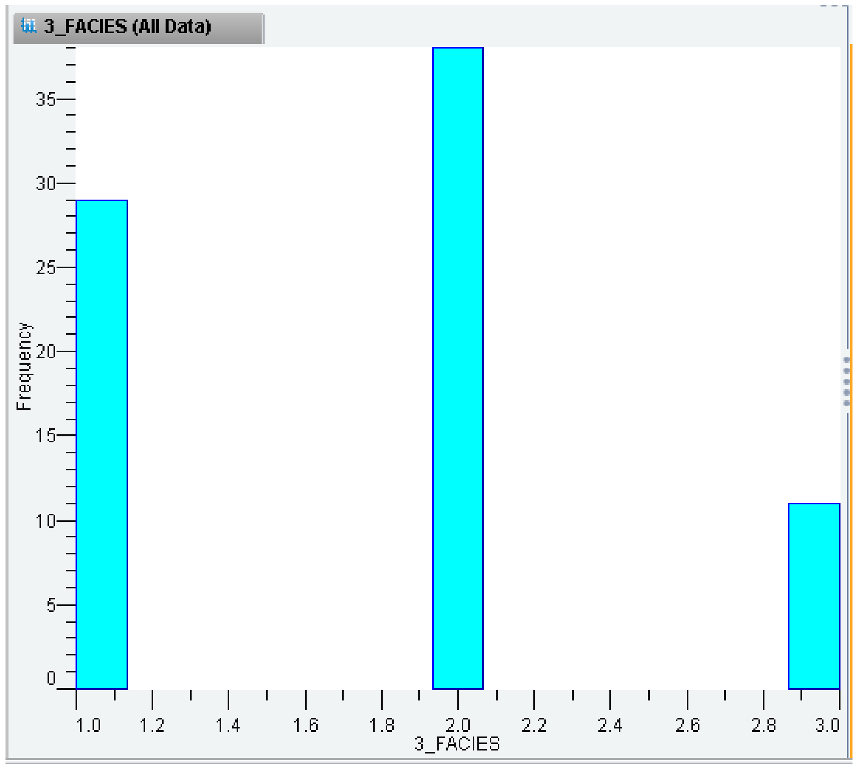
Steps

1. Open **Data I/O** and then the **Import Points** panel.
2. Browse to, select, and open **3_FACIES_INDICATOR_2D.txt**.
3. Set the parameters as shown below and then click **Create Pointset**.



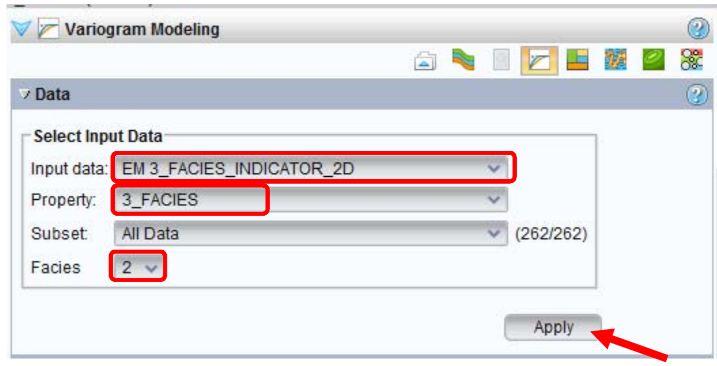
4. Open **Data Analysis** and create a histogram using **3_FACIES_INDICATOR_2D** for **3_FACIES** using **All Data**.

5. Your histogram should appear as shown below.

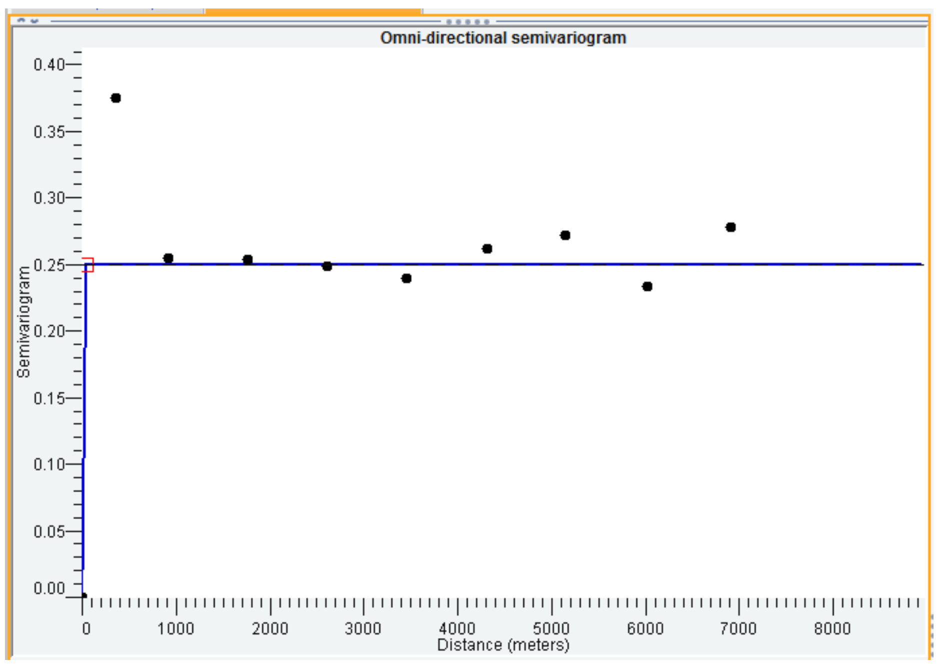


The histograms are displayed on the X axis at 1.0, 2.0, and 3.0 to identify the three facies. Only the major facies is modeled in variogram modeling and that model is applied to the other facies. You can see that Facies_2 is the most frequently occurring. Therefore, you will use Facies_2 in variogram modeling.

6. Open **Variogram Modeling**, set the parameters on the **Data** panel as shown below, and click **Apply**.

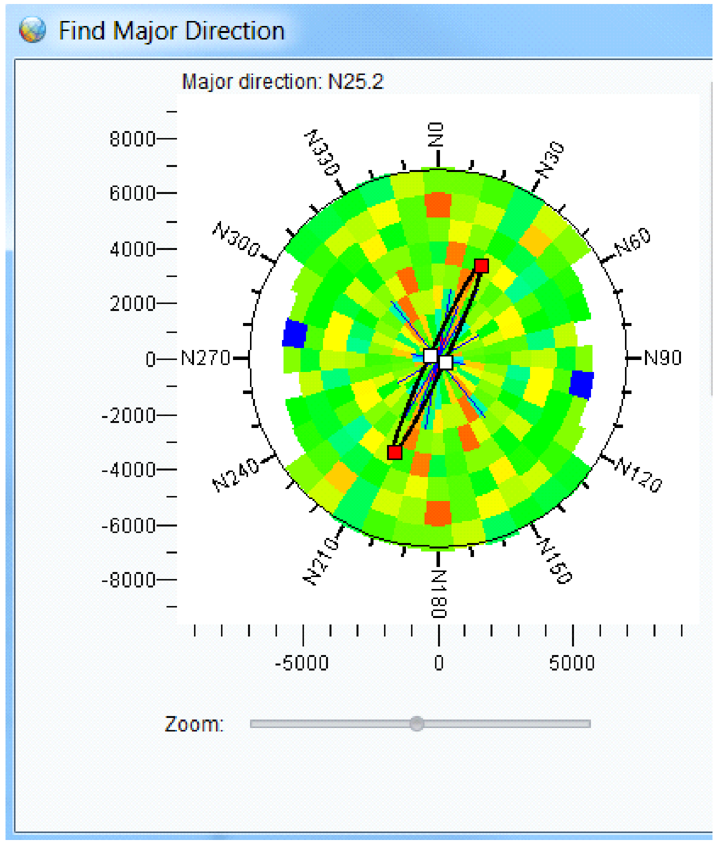


Your experimental variogram will appear as shown below.



7. Open the Computation Parameters panel, check off **Omni-directional variogram**. (Note that Major and Minor direction variograms are automatically computed and displayed.)

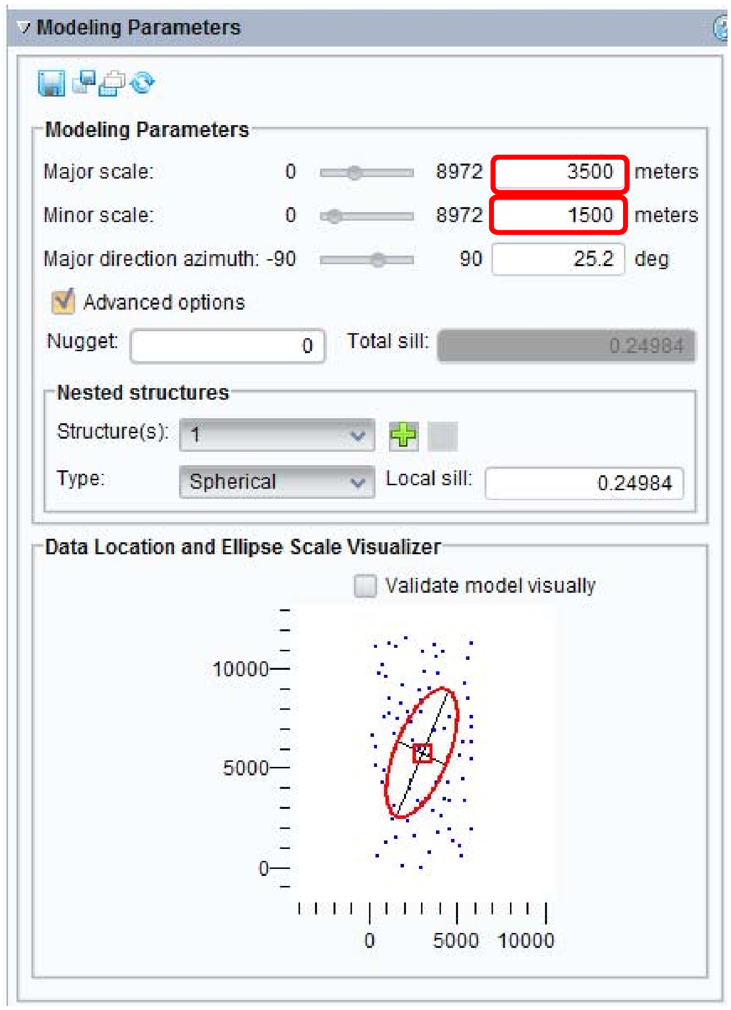
Then compute a **Variogram Map** using the default parameters, you will see that the ‘smart defaults have computed the Major direction azimuth almost perfectly.



8. Click **Recompute** on the Computation Parameters panel to compute the omni_directional variogram.

9. On the **Modeling** panel, change the Major scale to **3500** and Minor scale to **1500**.

   Your Data Location and Ellipse Scale Visualizer should appear approximately as shown below.



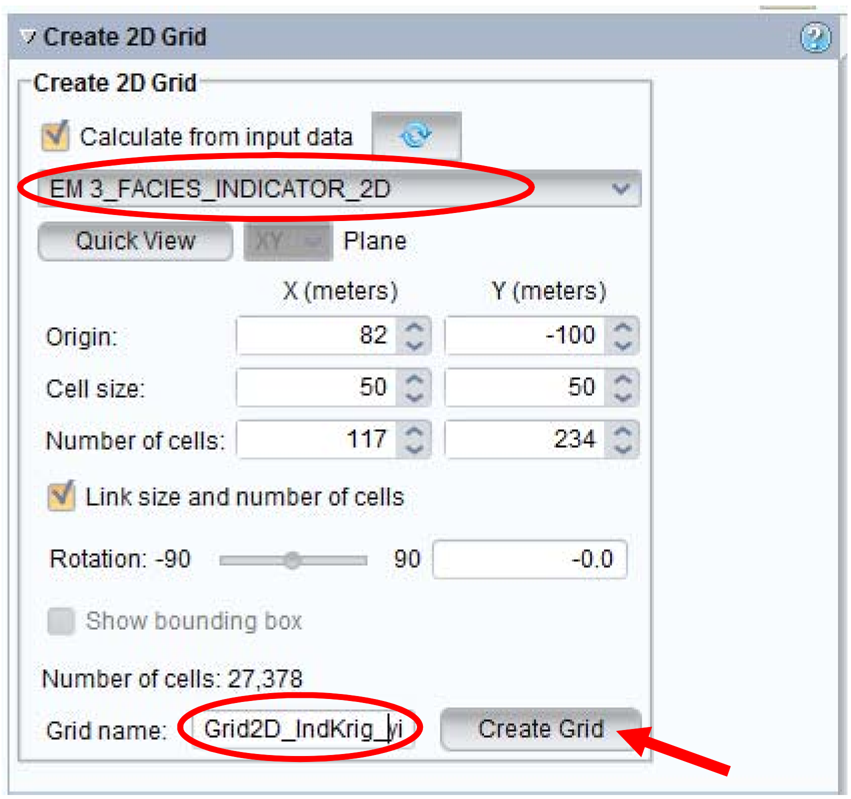
10. Click the icon at the top of the Modeling panel and save the variogram model as indicator_variogram.

11. Open **General Property Modeling** to **Create 2D Grid**.

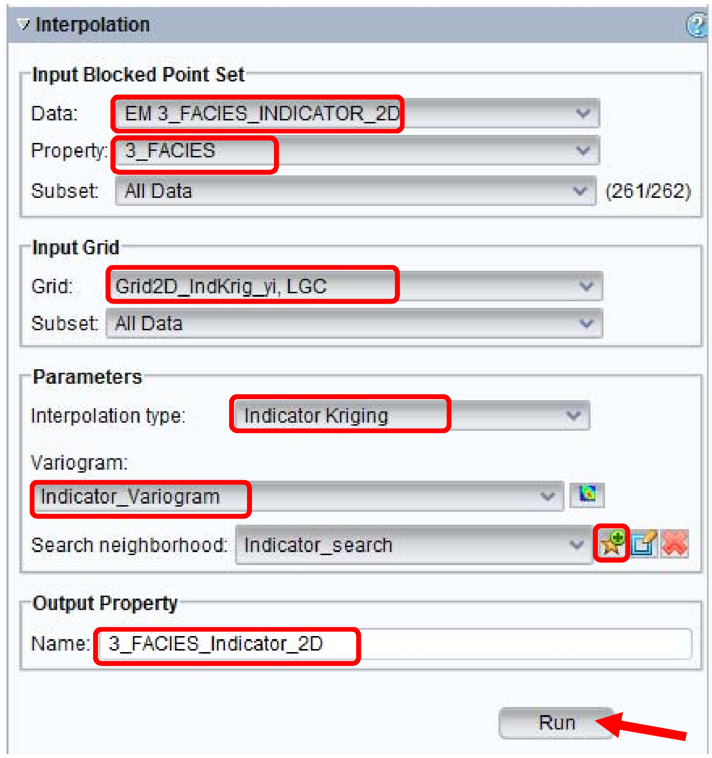
12. Select **EM 3_FACIES_INDICATOR_2D**.

13. Enter the **Grid name** as Grid2D_indkrig_yi.

14. Click **Create Grid**.



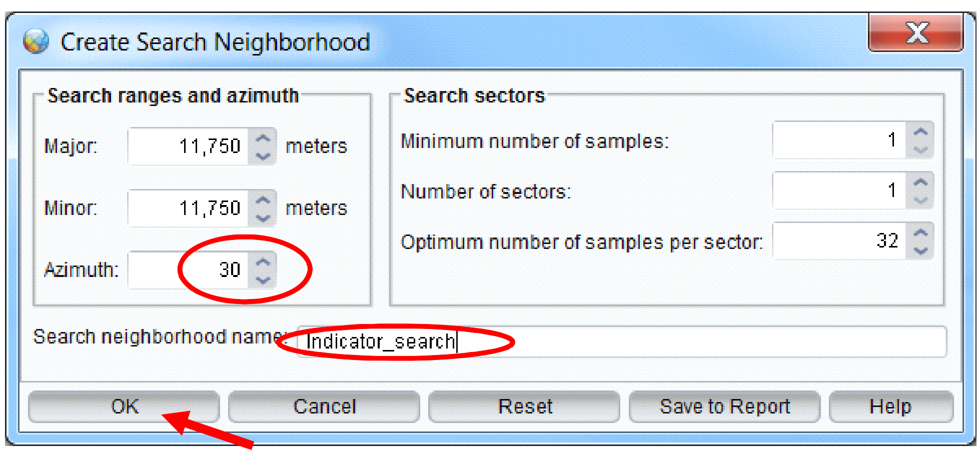
15. On the **Interpolation** panel, set the parameters as shown below.



16. Create a **Search Neighborhood** using all of the defaults except Azimuth. Change the **Azimuth** to **30** to align with the major direction of continuity.

17. Enter an Output Property Name.

18. Click **Run**.

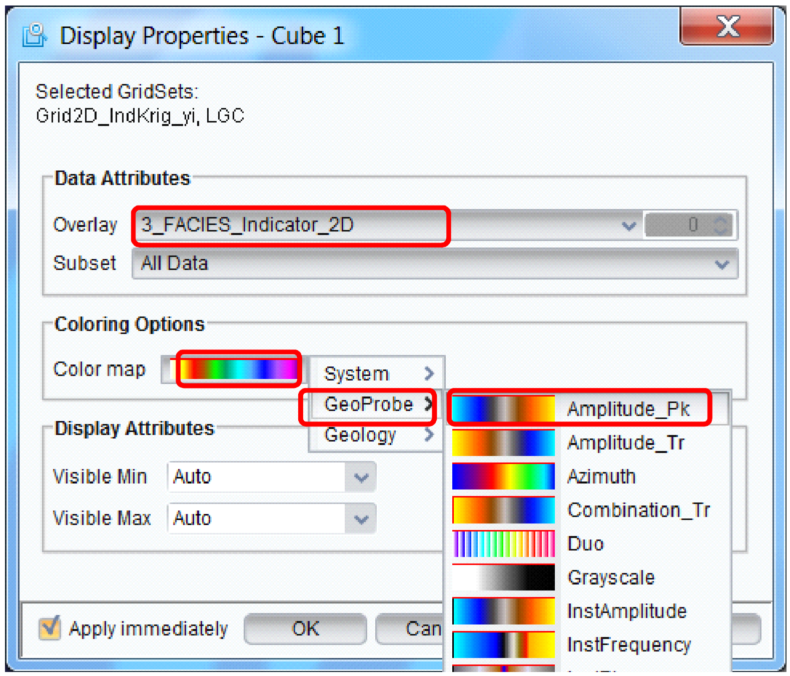


19. Open your Inventory and deselect the checkbox for **Grid2D_yi** so that it is hidden in the Cube view. Select the checkbox for **Grid2D_indkrig_yi** to display it in the Cube view.

20. Click MB3 on **Grid2D_indkrig_yi** and select **Display Properties** to open the Display Properties dialog box for that gridset.

21. Change **Overlay** to the output property from Indicator Kriging.

22. Change the color map to **GeoProbe > Amplitude Peak**.

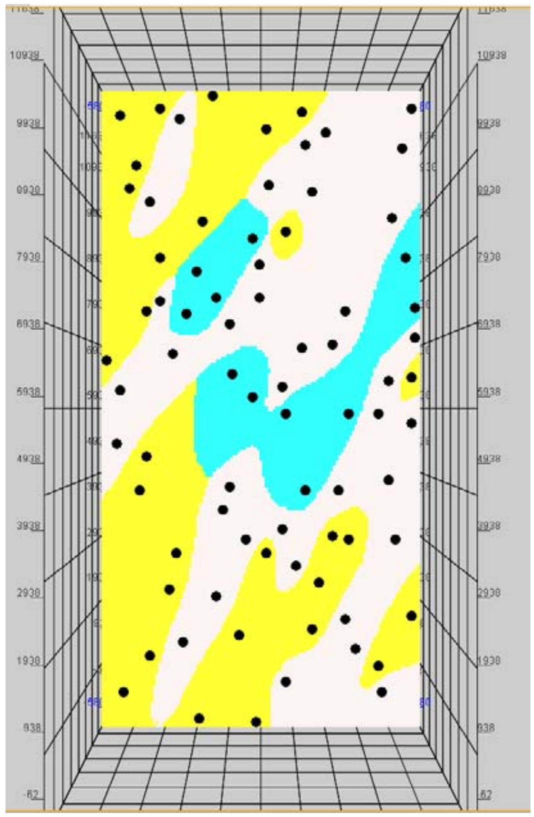


23. Hide your **WT_2D_yi** pointset in the view and display **3_FACIES_INDICATOR_2D**.

24. Change the Symbol for 3_FACIES_INDICATOR_2D to the **solid circle**, the Coloring Style to **Solid Color**, and the Color of the symbol to **black**.

25. Click the **View From Top** icon ( ) on the display toolbar and then the **View All** icon ( ).

    Your display should appear as shown below.



Flow Simulation

DecisionSpace® Earth Modeling extends the geoscientist's workflow with fast and efficient flow simulation modeling using Landmark's next generation reservoir simulator Nexus. Although classically a Reservoir Engineer's tool, flow simulation is also applicable to the Earth Modeler's workflow, allowing the ranking and assessment of produced petrophysical models according to the dynamic effects of petrophysical property controlled flow (hydraulic) barriers, system tortuosity and areal sweep potential for a specified injector. This workflow enables pre-screening of the earth model prior to use in history matching of existing production wells or the planning process for new wells.

The Flow Simulation feature allows the Earth Modeler to model single-phase flow on the earth model grid prior to upscaling.

The following are necessary inputs for the Flow Simulation workflow:

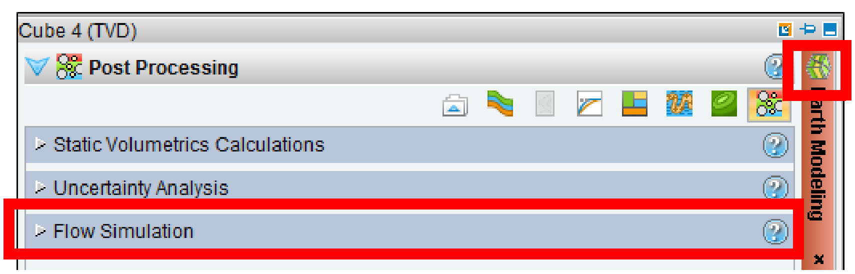
- A VDB grid (pre-upscaled; GEO class data input is permitted exclusively; and possesses less than 16.7 million cells)
- The grid containing properties of porosity and permeability
- Defined simulation wells

Exactly one well must be selected to be an injector.

At least one well must be selected to be a producer, and no more than five (5) wells may be made producers.

Accessing Flow Simulation

1. The Flow Simulation tool is accessible from the Earth Modeling tab in the Post Processing panel.

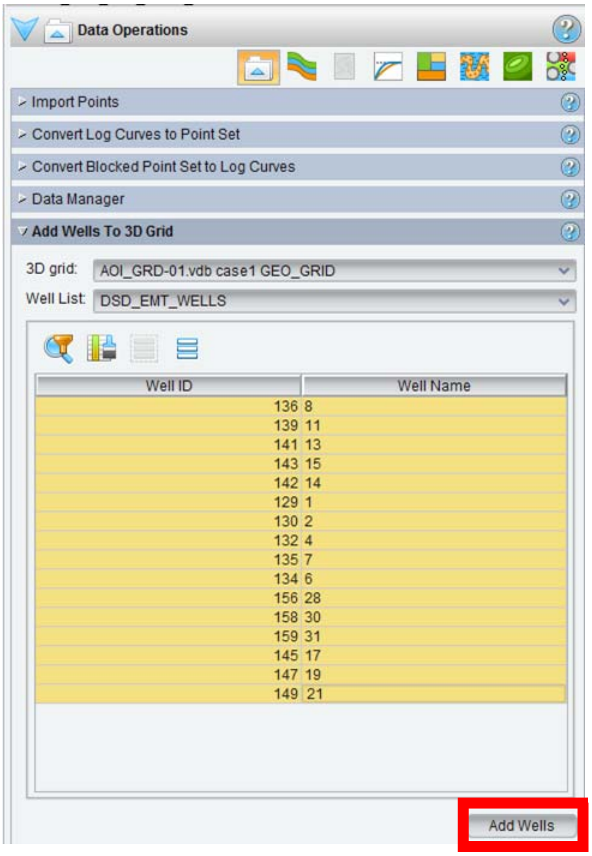


   Wells from OpenWorks should be added to the VDB of interest before attempting to use the Flow Simulation tool.

2. In the Earth Modeling tab click the **Data Operations** icon and then select **Add Wells to 3D Grid**.

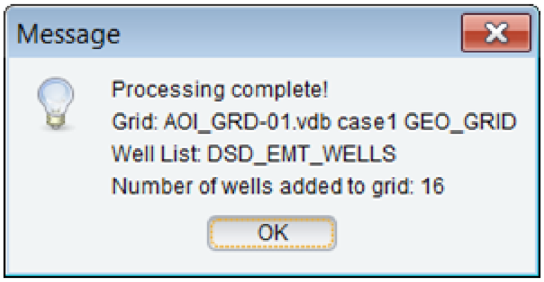
3. In the Add wells to 3D Grid panel, select the grid **AOI_GRD-01.vdb** from the drop-down list and select the well list **DSD_EMT_WELLS**.

4. Select all the wells available in the list and then click **Add Wells**.



A confirmation dialog box displays.

5. Click **OK**.

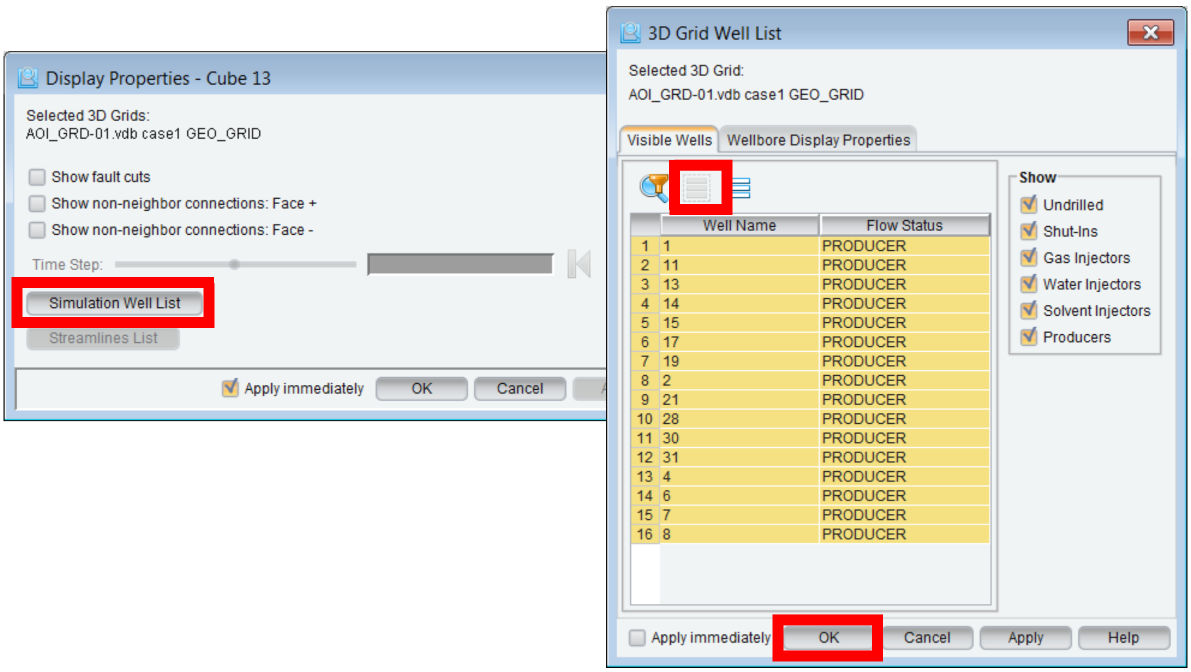


6. In a cube view display the 3D grid with PORO_PGS_All_Realiz property as Overlay.

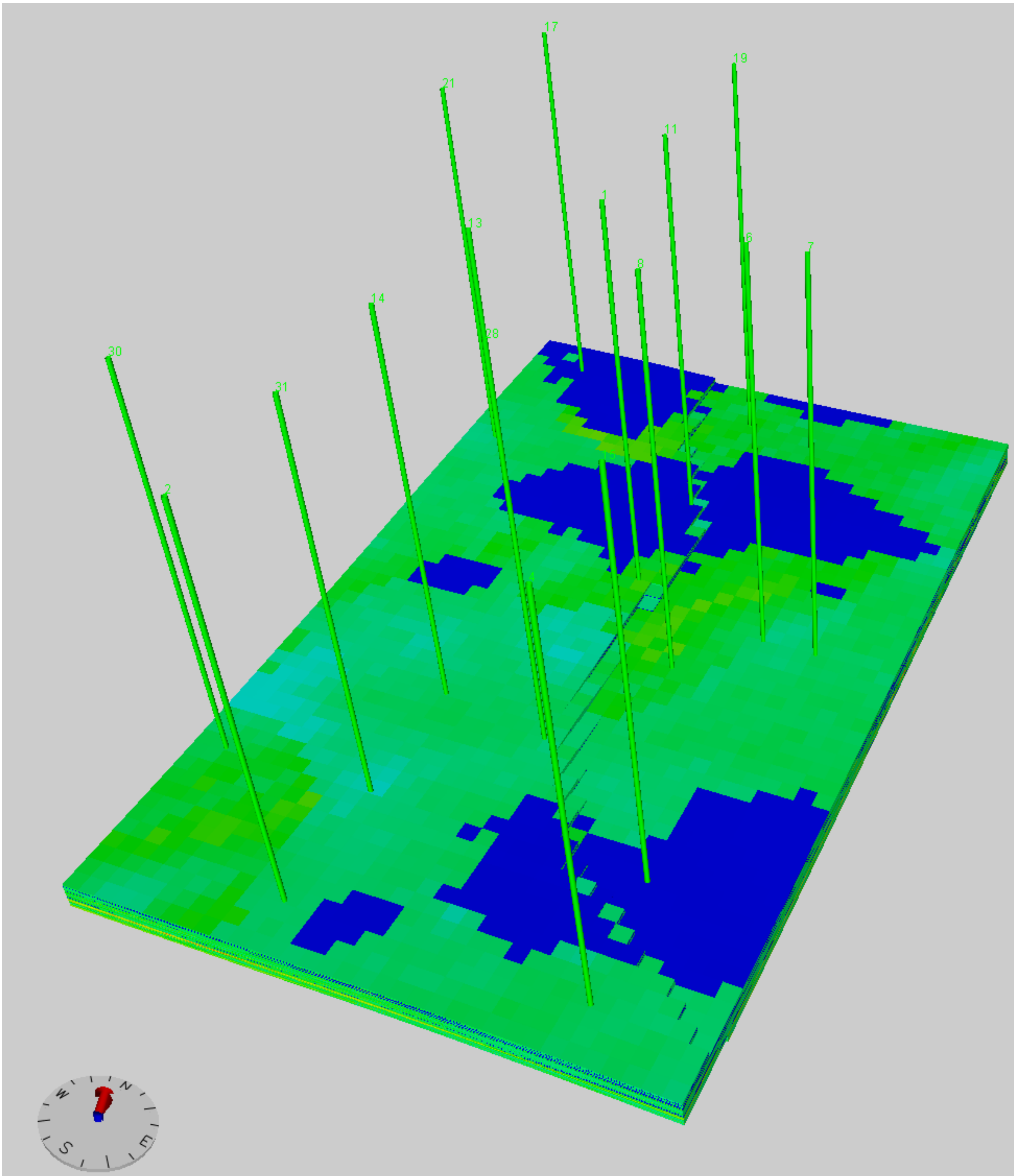
7. Click MB3 on **AOI_GRD-01.vdb** and select **Display Properties**.

8. In the Display Properties dialog box, click **Simulation Well List**.

9. In the Visible Wells tab, click the **Select All** icon and then click **OK** in both dialog boxes.



In the cube view you will notice that the wells are displayed as a Green tube indicating that the wells are ready to be defined as producers or injector in the flow simulation workflow.



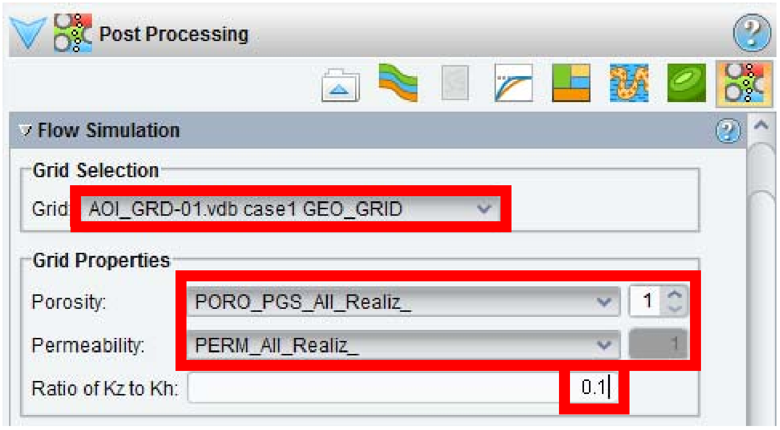
Flow Simulation Input Parameters

1. Select the **Flow Simulation** panel in Earth Modeling tab.

2. In the Grid Selection section, select **AOI_GRD-01.vdb** for Grid.

3. In the Grid Properties section, select **PORO_PGS_All_Realiz_** and **1** for Porosity and **PERM_All_Realiz_** for Permeability.

4. Enter 0.1 for Ratio of Kz to Kh.

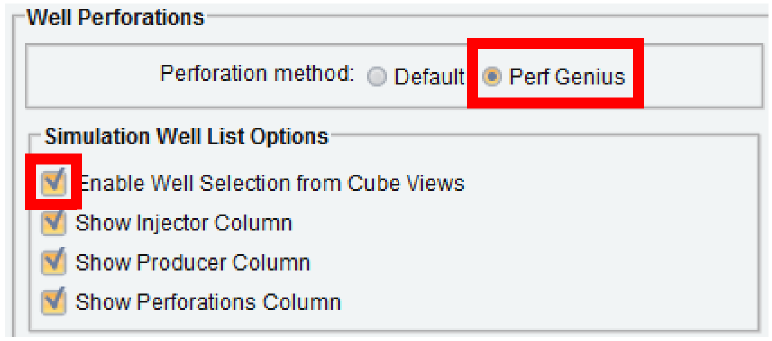


Note

Ratio of Kz to Kh field, specify a ratio for permeability in the vertical direction (Z) relative to the horizontal direction (XY). The Flow Simulation feature assumes that the permeability volume that is selected expresses permeability in the X direction and the feature is also programmed to assume that permeability in the Y direction is equal to that in the X direction. The specification of a ratio of Kz to Kh is to provide Nexus, as a part of the simulation process, with scaled Z direction permeability according to the input X direction permeability property. Typical values for this ratio is 0.1 but 1.0 is also widely used.

5. Select the Perforation method **Perf Genius**.

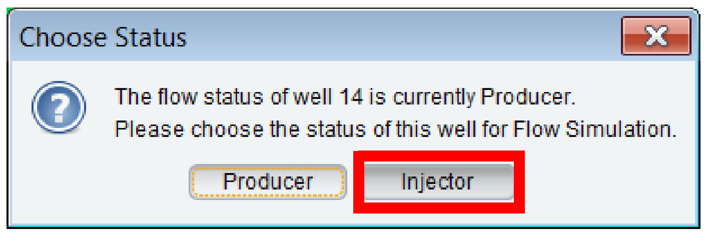
6. Select the **Enable Well Selection from Cube Views** checkbox.



You will notice that your mouse pointer has turned into Plus sign and ready to select injector and producer for simulation.

7. Click MB1 on the well **14**.

A Choose Status dialog box displays.



8. Select **Injector** as the status for this well.

Notice that in the simulation well list Well # 14 is highlighted as and Injector with a down-pointing blue solid triangle.

9. Similarly select wells **13**, **15**, **28**, **30** and **31** and make them producers by clicking MB1 in the cube view.

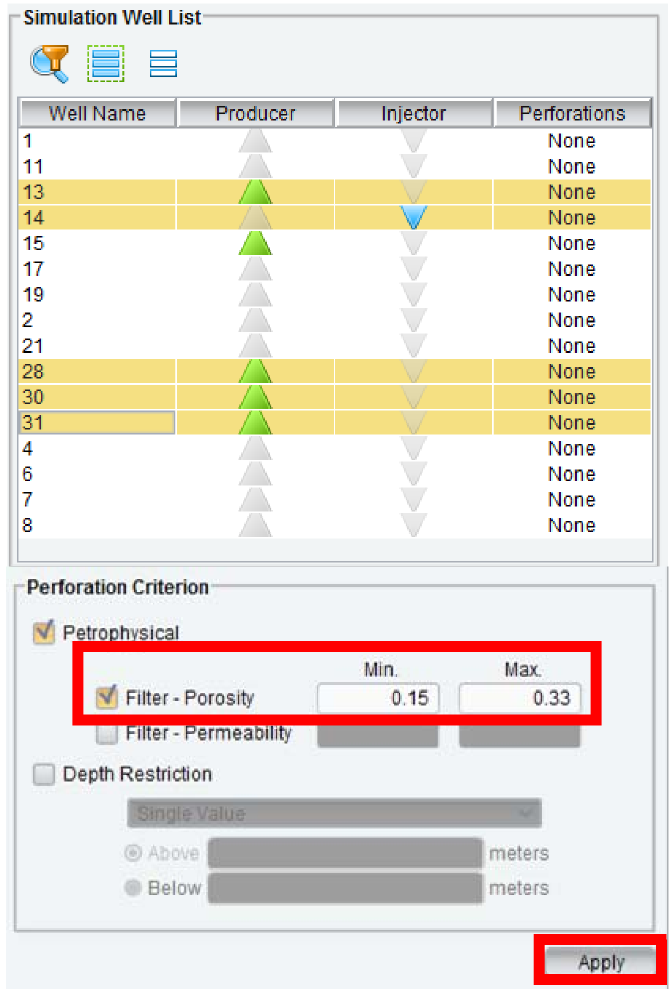
Notice that in the simulation well list these wells are highlighted as producers with a up-pointing green solid triangle.

Alternatively, you can select Injector and Producer from the Simulation well list directly by clicking MB1 on each triangle.

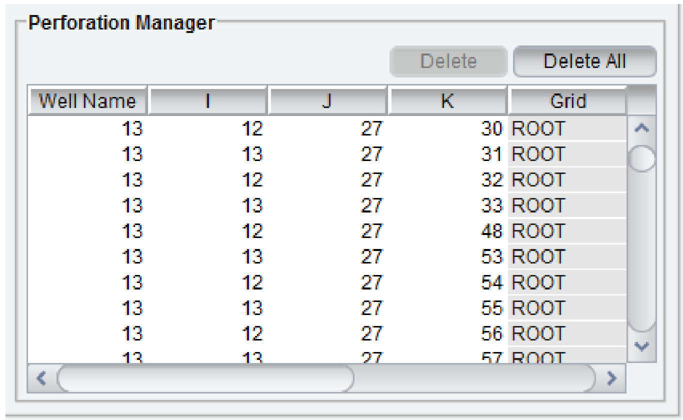
10. Select the wells using Ctrl+MB1 in the Simulation Well List.

11. In Perforation Criterion select the **Filter - Porosity** checkbox and type 0.15 for Min. and 0.33 for Max.

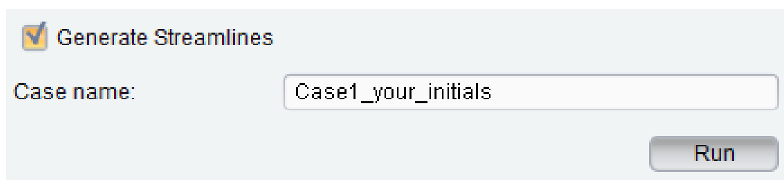
12. Deselect **Filter - Permeability** and **Depth Restriction** and then click **Apply** to generate all the possible perforation.



Perforation Manager is populated with all possible perforation intervals for the selected wells. Individual perforation intervals may be removed from the Perforation Manager (i.e., not included in the flow simulation) using the Delete button or the entire perforation interval list may be removed by selecting Delete All.



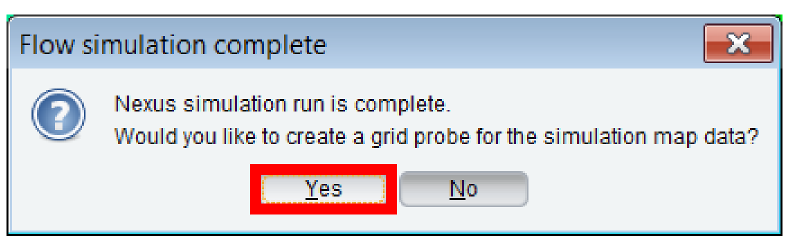
13. Select the **Generate Streamlines** checkbox, type Case1_your_initials as the Case name, and then click **Run**.



Once the flow computation is completed, the results will be available in the dynamic data class RECUR of the named case in the VDB, stored in the 3D Grids folder in inventory tree.

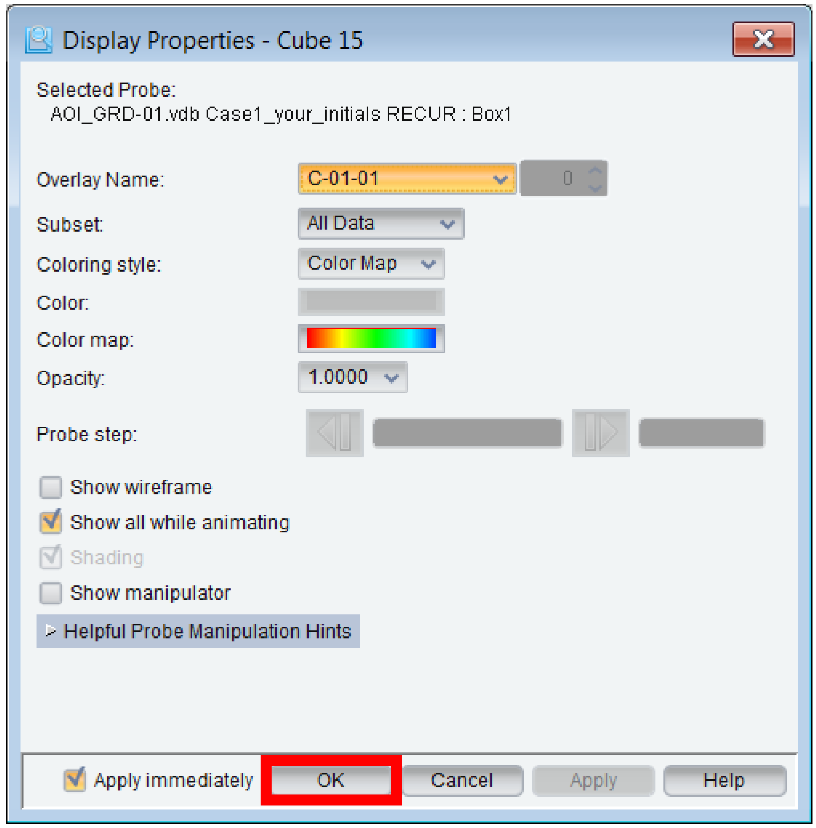
A Flow simulation complete dialog box displays.

14. Click **Yes**.

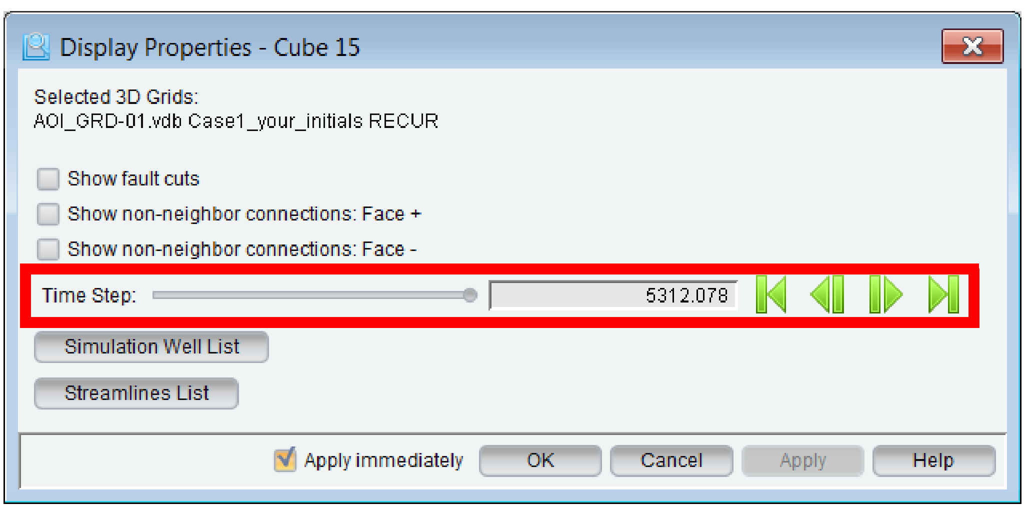


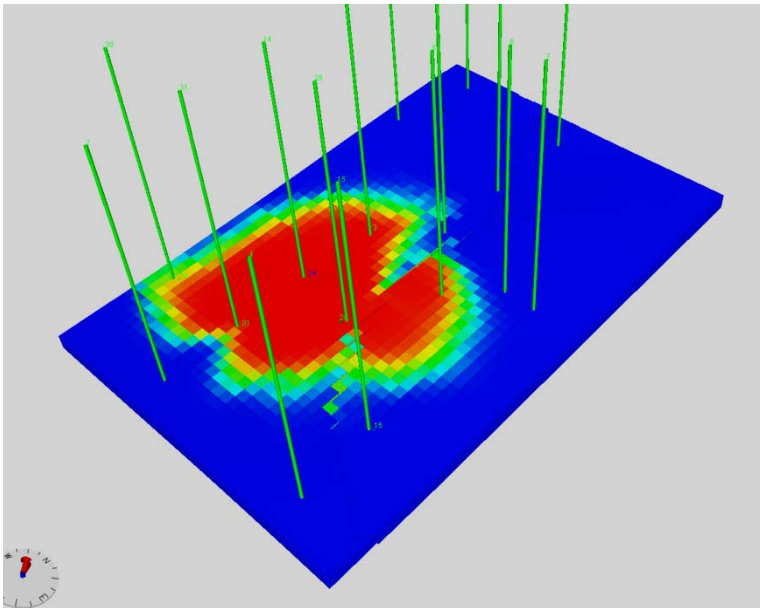
15. Display the RECUR grid in a cube view; click MB3 on the **3D Grid** and select **Display properties**.

16. In the Overlay Name drop-down menu, select **C-01-01**, which is the tracer concentration and then click **OK**.



17. In the Inventory tree > Display Property Editor on the RECUR box probe and then use the Time Step player.





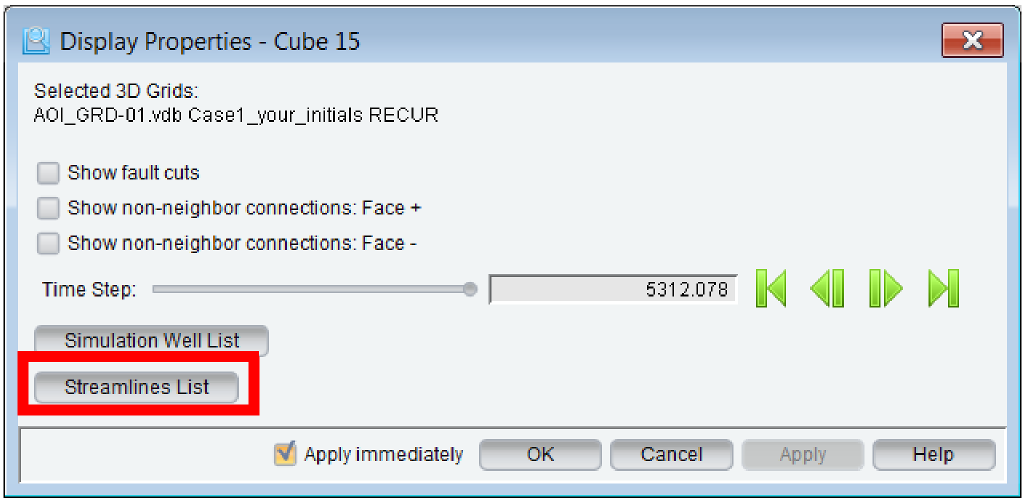
Streamline Visualization

Streamline data can only be viewed in the Cube view -independently or with other dynamic or static property volumes. Streamline visualization in the DecisionSpace suite allows you to:

- Visualize injector-producer pair groupings
- Use time of flight to understand displacement efficiency
- Enhance the interpretation of rock property effects on resulting fluid flow simulation

1. Click MB3 on the **RECUR** .vdb to open its Display Property dialog box.

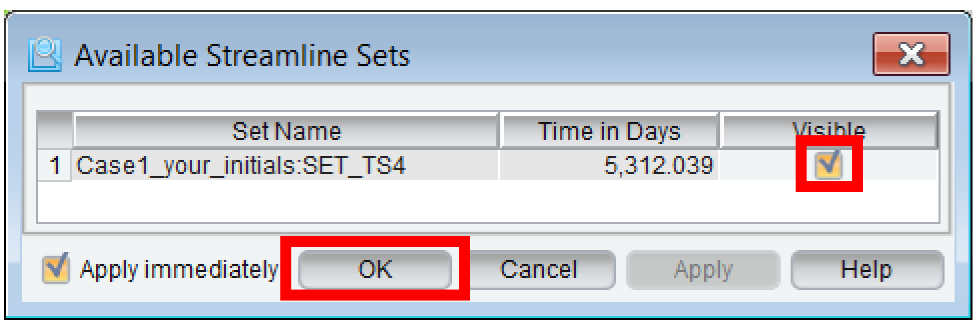
   In the VDB grid if streamline data exists, the Streamlines List button will be active.



2. Click **Streamlines List** to view available streamlines.

   A list of available streamlines displays.

3. Select the **Visible** checkbox for the streamlines and then click **OK**.



4. Return to the cube view and turn visibility off for the box probe.

   You will notice that Streamlines are now available in the cube view.

5. In the cube view, click MB3 on the Streamlines and select **Display Properties**.

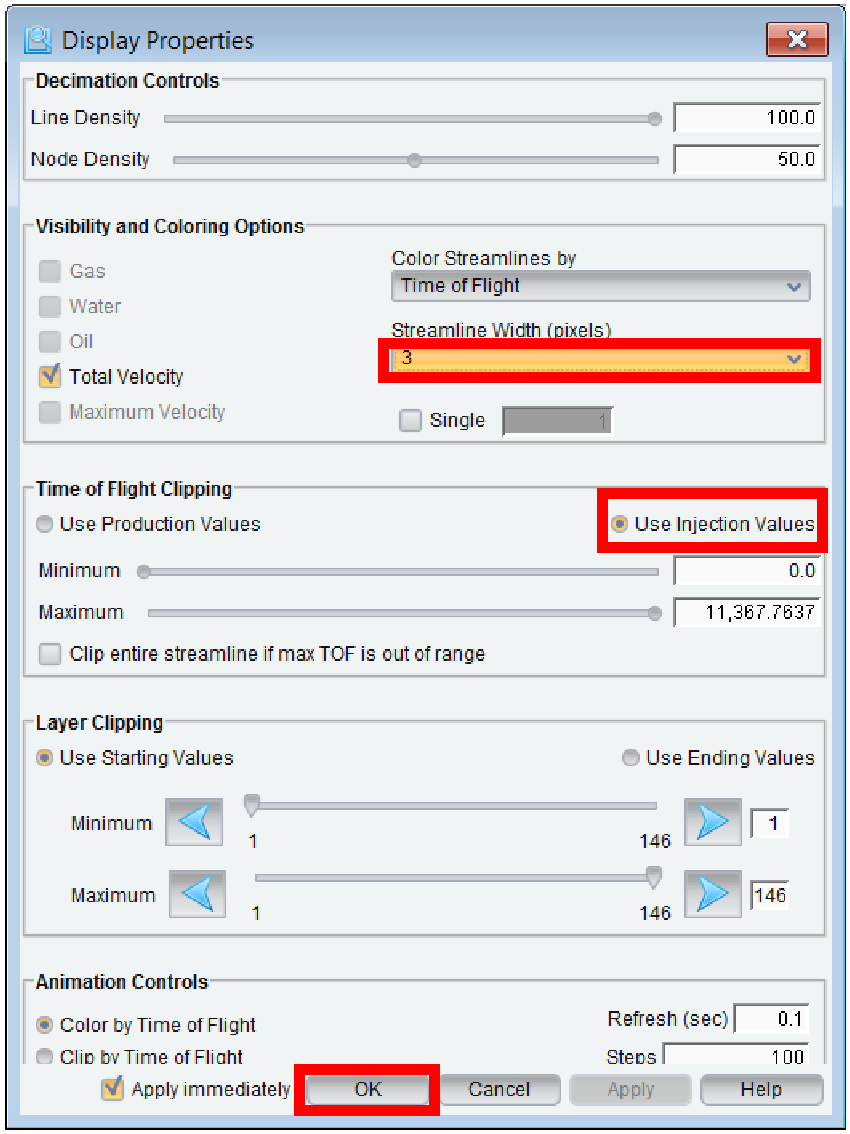
The display properties for streamlines consist of the following visualization controls:

<table>
<tr><td>1.</td><td>Decimation Controls</td><td colspan="2">These options allow you to manipulate the data density of the display. Decimating by line reduces the number of lines displayed. Decimating by node reduces the number of nodes (or data points) displayed.</td></tr>
<tr><td>2.</td><td>Visibility and Coloring Options</td><td colspan="2">These options allow you to specify the data to be displayed along the streamlines by checking the boxes.</td></tr>
<tr><td rowspan="13">3.</td><td rowspan="13">Color Streamlines</td><td colspan="2">Allows you to specify how the current color map gets applied to the streamlines. Following options for Color Streamline:</td></tr>
<tr><td>Time of Flight</td><td>The color range to display fluid displacement over time.</td></tr>
<tr><td>Common Producer</td><td>The color range to identify streamlines that terminate at a common producer. For example, if you have a single injector and multiple producers, the software identifies the streamlines that terminate at the producers and displays them with different colors.</td></tr>
<tr><td>Common Injector</td><td>The color range colors to identify streamlines that originate at a common injector. For example, if you have multiple injectors and a single producer, the software identifies the streamlines that originate at the injectors and displays them in different colors.</td></tr>
<tr><td>Producer-Injector Pairs</td><td>The color range colors to identify streamlines that are common to an injector-producer pair.</td></tr>
<tr><td>Saturation SW</td><td>The color range colors to identify water saturation values along the streamlines.</td></tr>
<tr><td>Saturation SG</td><td>The color range colors to identify gas saturation values along the streamlines.</td></tr>
<tr><td>Saturation SO</td><td>The color range colors to identify oil saturation values along the streamlines.</td></tr>
<tr><td>Starting Layer</td><td>The color range colors to identify streamlines that start in the same layer</td></tr>
<tr><td>Ending Layer</td><td>The color range colors to identify streamlines that end in the same layer</td></tr>
<tr><td>ID colors</td><td>Identifies individual streamlines by their streamline IDs. (Each streamline has a unique ID. For example, if the streamline set contains 20 streamlines, the unique IDs will range from 1 to 20.)</td></tr>
<tr><td>Streamline Width</td><td>Sets the display thickness of the streamlines in pixels.</td></tr>
</table>

		Single Streamline	Displays a single streamline at a time. To use this option, select the checkbox to check it 'on' and enter the ID of the streamline that you want to view. If you do not know the ID, you can determine the ID range for the entire streamline set by enabling the ID option (above) and consulting the color editor.
4.	Time of Flight Clipping	These options filter the time-of-flight data. For example, if you specify minimum and maximum values to be 50,000 to 70,000, the software displays only streamlines or parts of streamlines that have time-of-flight values between 50,000 and 70,000. You can perform the clipping based on the production values or the injection values. Toggling between these two options tends to reverse the visualization effect, since high production values generally correlate to low injection values, and vice versa. Use Production Values performs the clipping on the time-of-flight production values. Use Injection Values performs the clipping on the time-of-flight injection values. • Minimum sets the minimum time-of-flight value to be displayed. • Maximum sets the maximum time-of-flight value to be displayed. Clip entire streamline if max TOF is out of range eliminates partial streamlines from the display.	
5.	Layer Clipping	Streamlines can be clipped (eliminated from the view) by using either the layer the streamline begins or ends in.	
6.	Animation Controls	Streamlines can be animated using their Time of Flight data.	

6. In the Display Properties of streamline increase the width of streamline to **3** under Visibility and Coloring Options.

7. Select the **Use Injection Values** radio button under Time of Flight Clipping and then click **OK**.



To animate streamlines in the Cube view, they must first be selected so that the VCR controls become enabled.

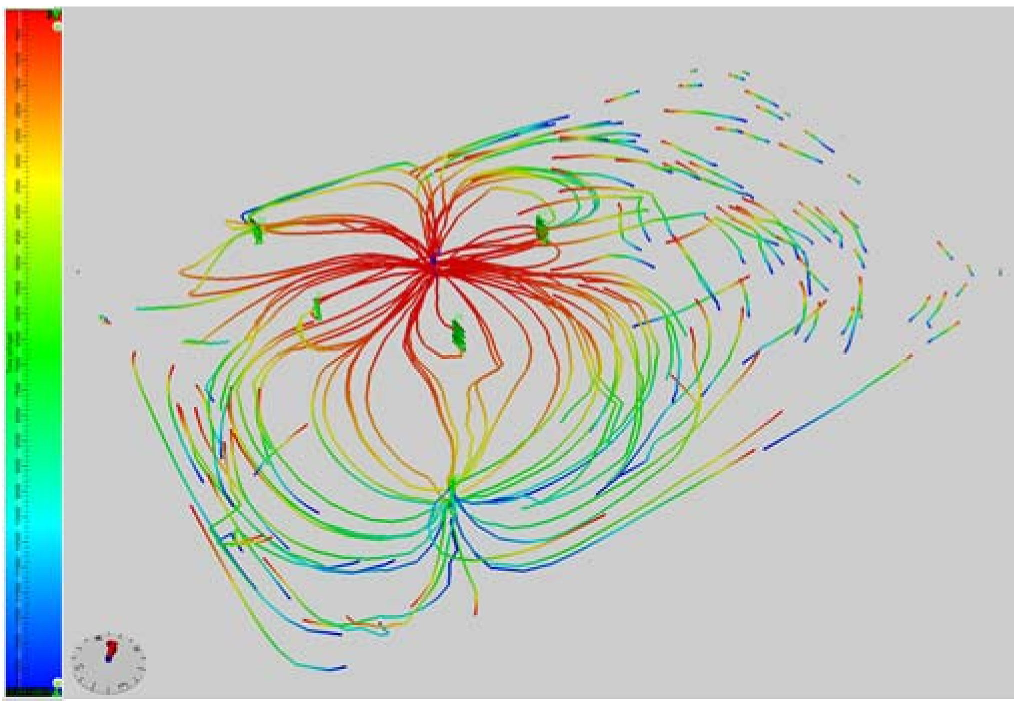
8. Switch the active cursor mode from Pan/Zoom/Rotate to **Select/Drag** (the Arrow cursor) to select streamlines in the Cube view.

9. Click anywhere on the streamlines in the Cube View.

   The VCR controls should become enabled.

10. Click **Play** of the VCR.

11. You will notice fluid movement along the streamlines in the cube view.



Streamlines Visualization