DecisionSpace® Geosciences Fundamentals of Geology Volume 2

HALLIBURTON | Landmark

Part Number 200103 Revision B

March 2015

Halliburton | Landmark
10200 Bellaire Blvd., Houston, Texas 77072-5206, USA
P.O. Box 42810, Houston, Texas 77242-2810, USA
Phone: 281.575.3000
Fax: 713.839.2290
Internet: https://www.landmarksoftware.com

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.

Contents

Volume 1

Preface: Fundamentals of Geology

Overview of DecisionSpace Geosciences Courses Preface-1

Leveraging Data with DecisionSpace Geosciences and Dynamic Frameworks to Fill Preface-3

Using Dynamic Frameworks to Fill Preface-3

Design Background of Dynamic Frameworks to Fill Preface-4

The Structure and Content of the Fundamentals of Geology Course . Preface-6

The Workflow of the Course Preface-9

The Dataset for this Class: Heidrun Field Preface-10

Geological Interpretation and Mapping with DFF Preface-11

Chapter Organization and Conventions of this Manual Preface-13

General Organization of Chapters Preface-13

Note on Exercises Preface-13

Personalized Outputs Preface-14

The Conventions of this Manual Preface-15

Chapter 1: DecisionSpace Geosciences Settings

Topics Covered in this Chapter 1-1

Exercise 1.1: Creating Well Lists, Visualizing Well Lists in Map View, and Hiding Wells from a List 1-2

Creating a Well List from the Well Query Tool 1-9

Hide Some Wells from List 1-12

Exercise 1.2: Deviation Survey Point/Position Log 1-14

Exercise 1.3: Display Well Data Along Wellbores in *Map* View 1-18

Exercise 1.4: Well Annotation, Repositioning, and Rotating 1-28

Exercise 1.5: Bubble Mapping. 1-37

Exercise 1.6 (Optional): Creating a New Strat Column with a New Strat Unit 1-48

Importing a Stratigraphic Column 1-55

Exercise 1.7 (Optional): Visualizing GIS Data in DecisionSpace. 1-58

Chapter 2: Well Correlation View

Topics Covered in this Chapter 2-1

Starting a New DecisionSpace Geosciences Session 2-2

Exercise 2.1: Creating a Well Correlation View from a Well List. 2-4

Exercise 2.2: Creating a Well Correlation View from a Section View. 2-7

Exercise 2.3: Creating a Well Correlation View from a Map View 2-10

Exercise 2.4: Creating Well Layouts in a Correlation View 2-13

Creating Custom Well Layouts. 2-13

Exercise 2.5: Adding a Layout to Wells in Correlation View. 2-38

Display a Specific Layout. 2-38

Displaying a Layout in All Wells in Correlation View 2-43

Exercise 2.6: Flattening a Correlation View and Changing Views from TVT to TST 2-47

Changing View from TVT to TST 2-49

Exercise 2.7: Setting View Properties for Well Correlation View. 2-51

Proportionally Spacing or Equally Spacing Wells 2-51

Modify Scale. 2-54

Using Z Range to Clip the Well Correlation View. 2-58

Using Display Range to Clip Well Correlation View. 2-60

Exercise 2.8: Line of Section (LOS). 2-64

Well Projection Detail 2-72

Graphically Modifying a Projection 2-77
Manually Modifying the Projection 2-80
Saving Well Lists from LOS 2-81

Chapter 3: Surface Interpretation and Mapping with Dynamic Frameworks to Fill

Topics Covered in this Chapter 3-1

Exercise 3.1: Selecting and Setting Up Data 3-2

Exercise 3.2: Creating a Surface Pick. 3-8

Exercise 3.3: Mapping Using Dynamic Frameworks to Fill. 3-12

Exercise 3.4: Updating Dynamic Surface Picks. 3-15

Exercise 3.5: Interpreting Surface Picks: Correlation Options. 3-23

Exercise 3.6: Creating a Fault Pick 3-47

Exercise 3.7: Using QC Tools. 3-53

Chapter 4: Mapping with Dynamic Frameworks to Fill

Topics Covered in this Chapter 4-1
The Benefits of Dynamic Frameworks to Fill 4-1

Introduction to Dynamic Frameworks to Fill (DFF). 4-4
Overview. 4-4
Software Defaults in Dynamic Frameworks to Fill 4-5
Gridding Algorithms. 4-6
Conformance Mapping 4-12

Exercise 4.1: Fast Creation of Surface Maps 4-15

Exercise 4.2: Performing On-the-Fly Time-Depth Conversion of Framework Surfaces 4-21

Exercise 4.3: Mapping Surface Picks 4-28

Exercise 4.4: Generating Fault Networks 4-34
Fault Networking 4-42

Exercise 4.5: Framework Editing Tools 4-44

Exercise 4.6: Editing Framework Intersections 4-47
Using the Intersection Editing Tool 4-47
Entering Interpretation Mode 4-49
Using Cursors 4-50

Volume 2

Chapter 5: Interpreting Lithology and Petrophysics

Topics Covered in this Chapter 5-1

Exercise 5.1: Lithological Interpretation 5-3

Overview: Interpreting Petrophysics. 5-15

Exercise 5.2: Using Log Model Editor 5-16

Exercise 5.3: Mapping Petrophysical Attributes in Dynamic Frameworks to Fill 5-22

Chapter 6: Volumetrics

Topics Covered in this Chapter 6-1
Volume Calculation Technologies 6-1
Direct Polyhedral Volume Calculation. 6-2
Fast-Sweep Thickness Extraction 6-2
Slice-Based Volumetric Calculations 6-3
Compartment Technology 6-4
Compartment GeoGrouping 6-4
Advanced Compartment Functionality. 6-5

Exercise 6.1: Creating Compartments 6-9

Exercise 6.2: Volumetric Calculations 6-17

Exercise 6.3: Compartment Display Properties 6-26

Exercise 6.4: Advanced Compartment Construction 6-33

Chapter 7: Final Framework to Earth Modeling

The Topics Covered in this Chapter 7-1

Overview: Integrating Velocity Model with Framework 7-2
The Benefits of Integration 7-2
Creating Earth Modeling Grids. 7-2
Creating Grids 7-3
Looking at Your Grid 7-6

Exercise 7.1: Loading Log Curves and a Framework 7-8

Exercise 7.2: Converting Log Curves to Point Sets 7-12

Exercise 7.3: Analyzing the Data. 7-16

Overview: Stratigraphic Modeling 7-30
PM Grid3Ds and Grid Probes 7-31
Layering Styles and Implications 7-36

Exercise 7.4: Generating a 3D Geocellular Grid 7-39

Overview: Facies Mapping 7-46

Exercise 7.5: Defining Lithotype. 7-47

Exercise 7.6: Seismic Attribute Blocking 7-51

Overview: Well Blocking. 7-54

Exercise 7.7: Well Blocking 7-57

Overview: Earth Modeling: Lithotype Proportion Map Creation 7-62

Exercise 7.8: Creating Lithotype Proportion Maps. 7-64

Appendix A: Geological Mapping

Topics Covered in this Appendix A-1

Understanding Gridding Algorithms. A-2
The Grid and Contour Dialog A-3

Exercise A-1: Creating a Basic Grid. A-5
Customizing Grids A-7

Exercise A.2: Creating Surface Grids. A-18
Basic Contouring A-29
Commonly Used Contour Settings A-32
.The Basic Tab A-32
Contouring Parameters A-34
Output A-37
Lines and Labels Tab A-37

Exercise A.3: Creating Contour Maps A-41
Contour interpretation. A-52
Entering Contour Interpretation Mode A-52

Exercise A.4: Editing Contours A-65

Exercise A.5: Using Inclusive and Exclusive Polygons A-75
Part 1: Polygons A-75
Part 2: Gridding A-80
Using Residual Fit Operations A-85

Exercise A.6: Using the Residual Fit A-89

Appendix B: Wellbore Analyzer

Creating Surface Picks with Surface Grids. B-9

Appendix C: Horizontal Well Correlation

The Topics Covered in This Appendix. C-1
What is Horizontal Well Correlation? C-2
Data Requirements for HWC C-2
Working With HWC. C-3

Horizontal Well Correlation C-4

Exercise C.1: Create a Framework for HWC Workflow C-6

Exercise C.2: Launch and Configure the HWC Workspace C-15

Exercise C.3: Create a Predicted Curve from an Offset Well. C-26

Exercise C.4: HWC Interpretation and Edit Options C-31

Anchor Line at the TD of the Drilling Well C-34

Exercise C.5: Displaying and Generating an Apparent Dip Calculation. . C-44

Optional Workflow 1: Create a Predicted Curve from a Drilling Well . . . C-46

Optional Workflow 2: Create Predicted Curve from Multiple wells C-47

Appendix D: Dipmeter Analysis

How Dipmeter Analysis Works D-1

Using the Dipmeter Analysis Tool D-3

Chapter 5
Interpreting Lithology and Petrophysics

Decision Space Geosciences offers several tools to perform lithological and petrophysical analysis. The final petrophysical results feed directly into the Framework Model. Log Calculator, an essential part of Dynamic Frameworks to Fill, it is also dynamic. This means that if something has to be changed after attribute maps are created in the petrophysical model, the attribute maps will change on the fly. Also, any changes in the structure or any surface pick will trigger updates in the attribute model. Lithological and petrophysical calculations can be performed on a multi-well basis. The unified environment enables a tight integration with Zone Manager, which allows you to include variables that can change by zone or by well in the petrophysical model.

In this chapter you will use Decision Space Geosciences to create lithology curves and strips, as well as have a general overview of log calculator and petrophysical models. For more detailed workflows and descriptions a class is offered specifically covering Petrophysics.

Continuing with the integrated geological workflow, in previous modules you have learned how to create layouts to be used in making interpretations on surfaces within *Correlation* view. In this module you will use those surfaces to define ranges where the petrophysical analysis will be performed within the sandstone reservoir. Finally the attributes that you generate in this workflow will be used to dynamically fill the structural framework.

Topics Covered in this Chapter

By the end of this chapter you will be able to:

- Interpret lithologies and create a litho-facies logs and lithology strips.
- Calculate Porosities.
- Calculate water saturation, bulk volume water and hydrocarbon pore volume and pay.

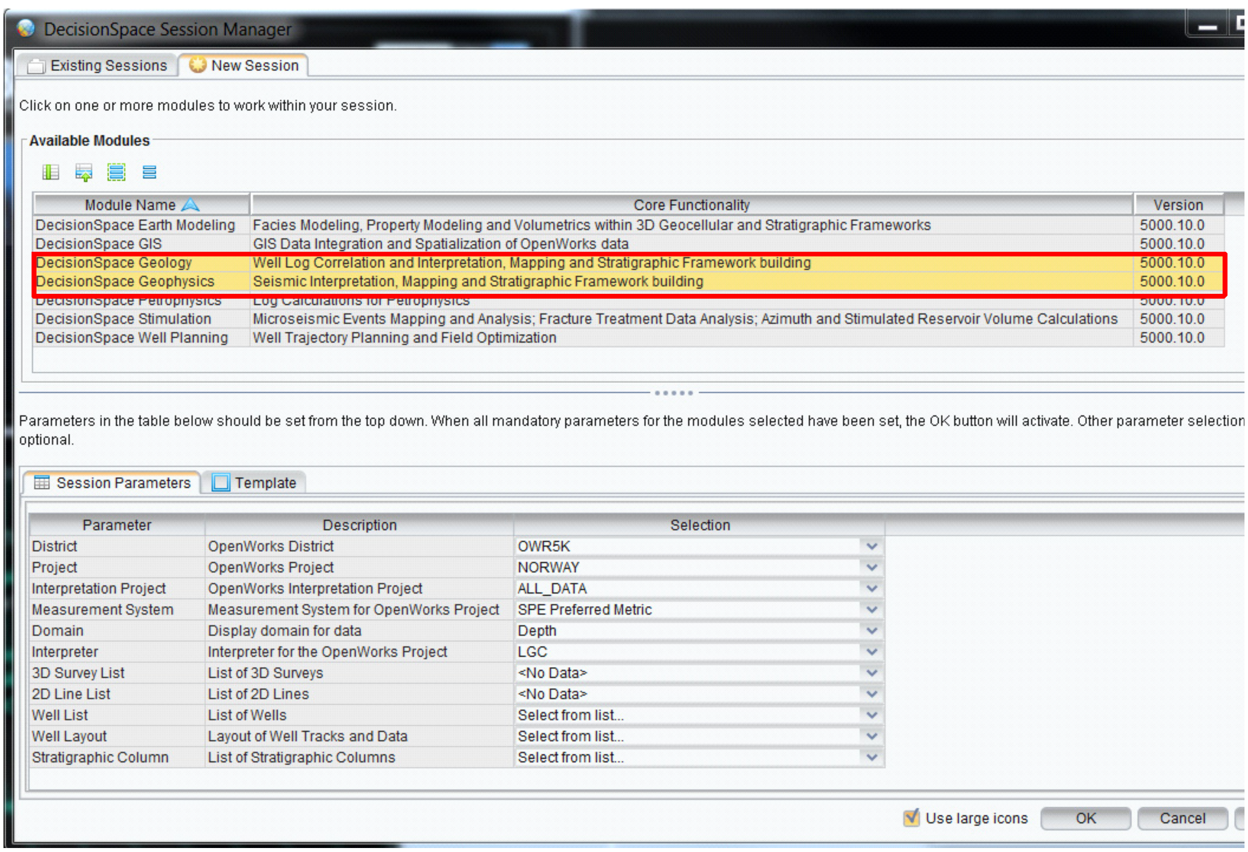
- Test the petrophysical model in wells before applying the equation.
- Create intervals in Dynamic Frameworks to Fill to populate with petrophysical attributes.
- Understand the value of the dynamic nature of frameworks when changing petrophysical attributes.

Exercise 5.1: Lithological Interpretation

The Lithology Interpretation toolkit in the DecisionSpace Geosciences software allows you to interpret rock types along wellbores in response to log curve character and project knowledge.

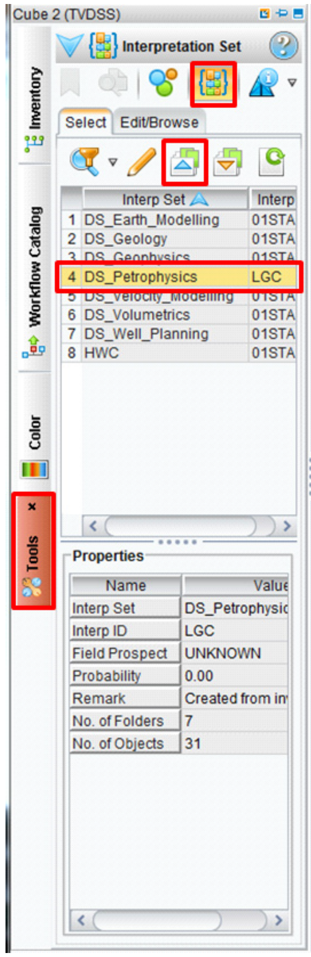
You can interpret lithology in strips (associated with well lists or selected wells) to make precise interpretations that correspond to log, well pick, and well interval data. Lithology strips are displayed in *Section* and *Correlation* views, as well as along well layout images in *Cube* view. You can create custom lithology palettes (for example, a collection of lithology for clastics and a second collection for carbonates relative to your project).

1. Start a new session with the parameters in the shown in the following image, and click **OK**.

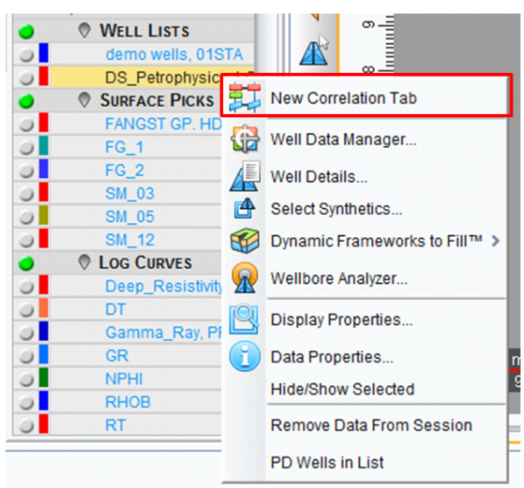


2. In the *Tools* task pane, click the **Interpretation Set** ( ) icon.

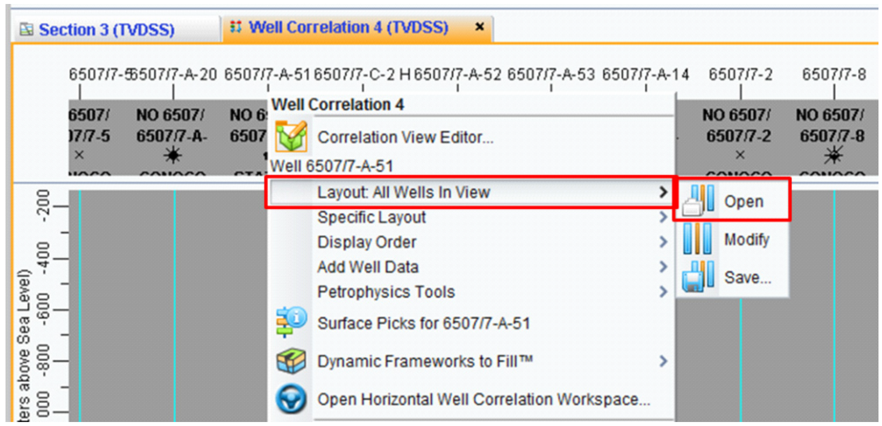
3. Select the **DS_Petrophysics** interpretation set and click the **Load Data To Session** icon.



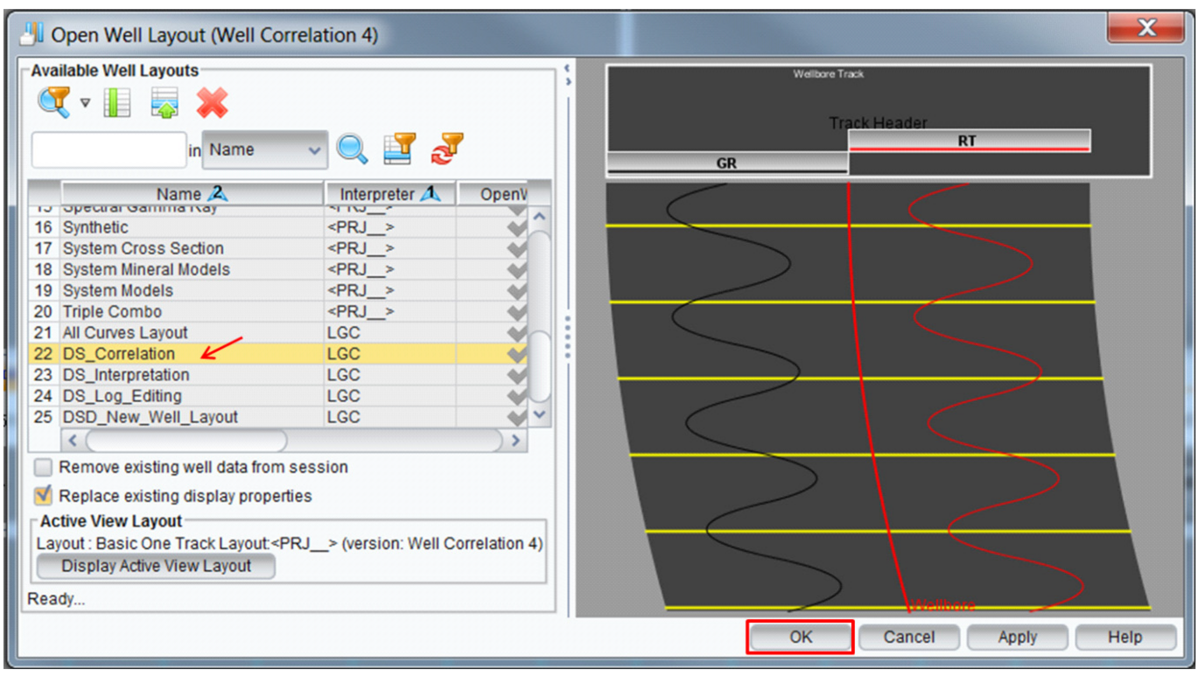
4. In the *Inventory* task pane, put your cursor on the **DS_Petrophysics** well list and **MB3 > New Correlation Tab**. This will open a new *Correlation* view, showing all of the wells within the well list.



5. Maximize your new *Correlation* view and then put your cursor on any of the wells within the view and **MB3 > Layout All Wells In View > Open.**

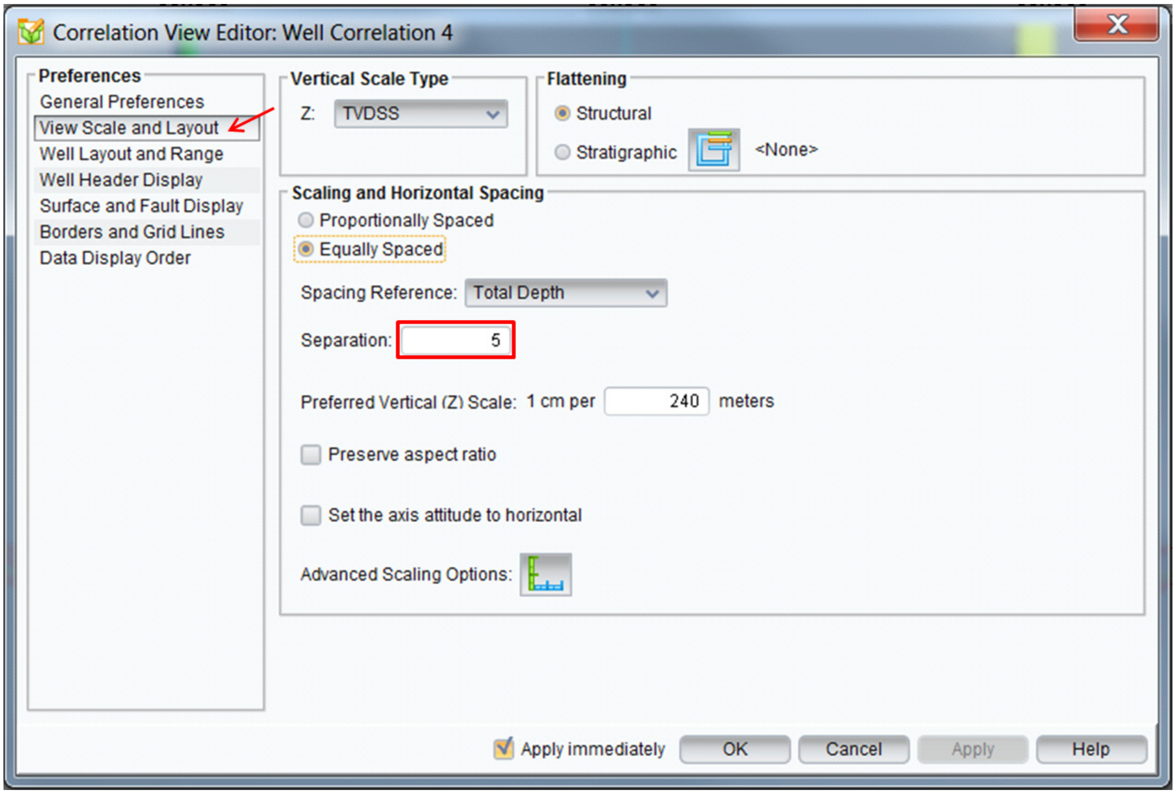


6. The *Open Well Layout* dialog will open. Select the **DS_Correlation** well layout from the list and click **OK**.



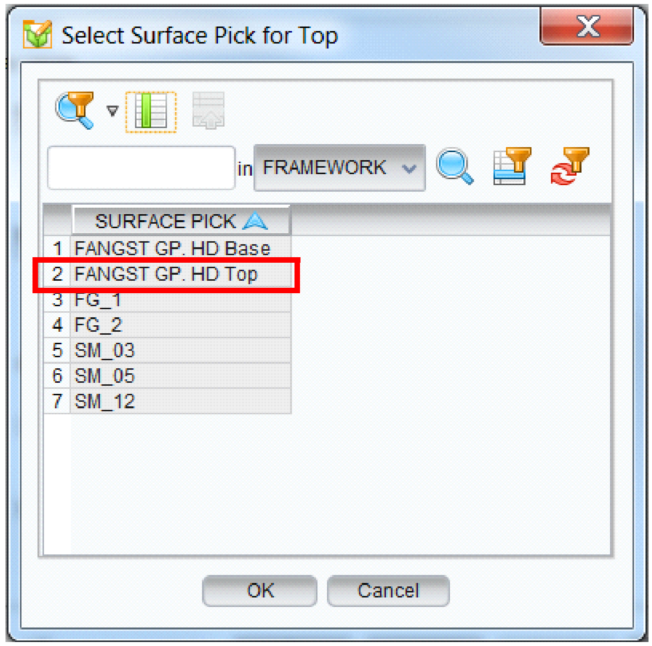
7. In the *Inventory* task pane toggle on all of the **Surface Picks**, except for FANGST GP. HD Base.

8. With *Correlation* view still active, click the **View Editor** ( ) icon to open the *Correlation View Editor*. In the *View Scale and Layout* tab change the *Separation* to **“5”,** and press **<Enter>**.

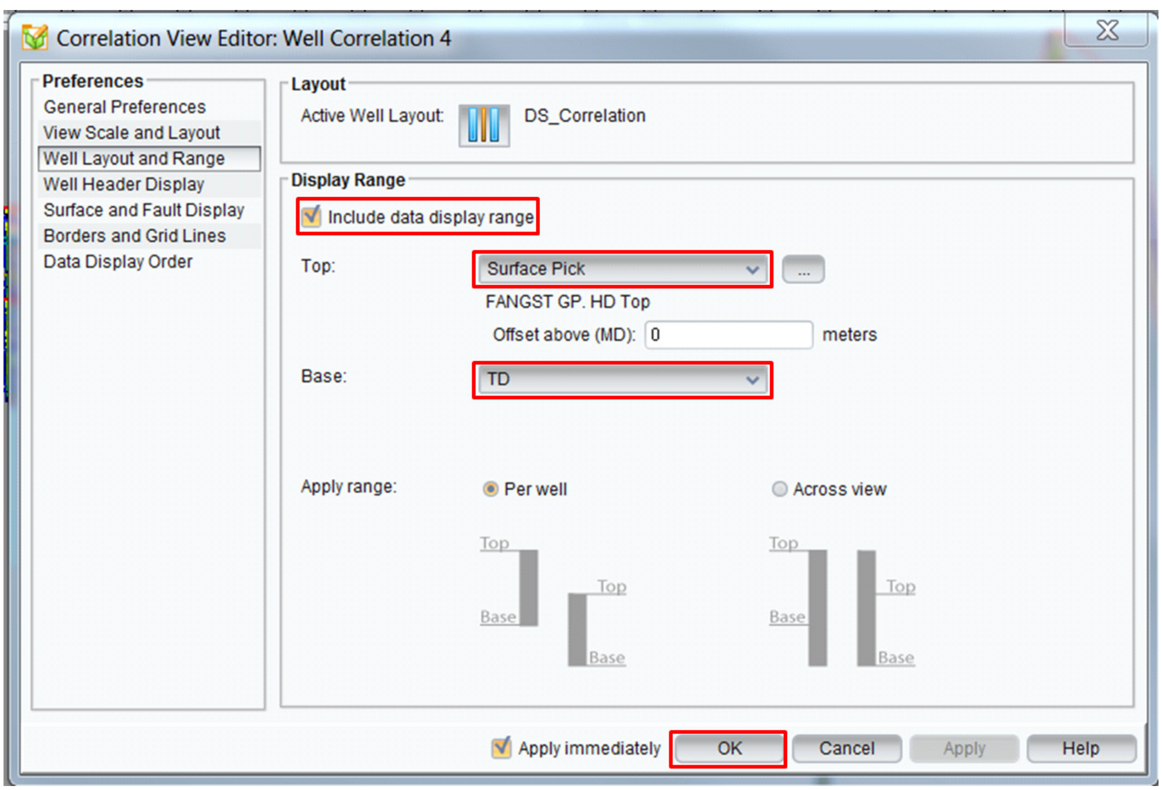


9. In the *Preferences* panel, select *Well Layout and Range*. In the *Display Range* panel, toggle on **Include data display range**.

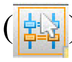
10. On the Top: pull-down menu select **Surface Pick**. The first time you do this the *Select Surface Pick for Top* dialog will open. Select the **FANGST GP. HD Top** and click **OK**.



11. On the Base: pull-down menu of the *Display Range* panel, select **TD**.

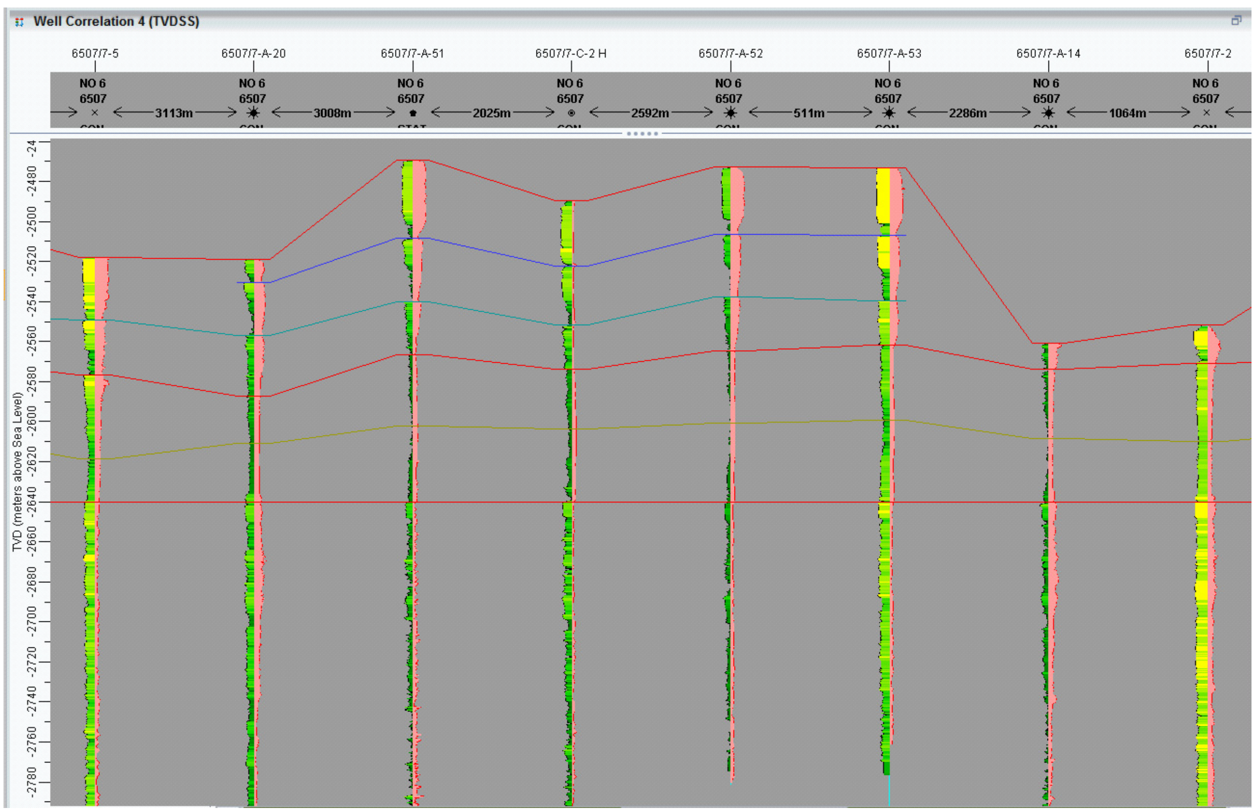


12. Click **OK** to close the *Correlation View Editor.*

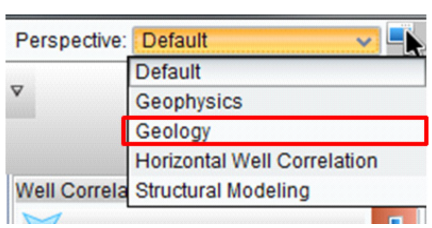
13. While in *Correlation* view, click the **Select Pick for Flattening** () icon and click the **SM_12** surface pick.

> **Note:**
>
> When selecting your pick for flattening, make sure that you click the pick over the well, not the tie line. Zooming may help you make the right selection.

Your Correlation should look similar to the one below.

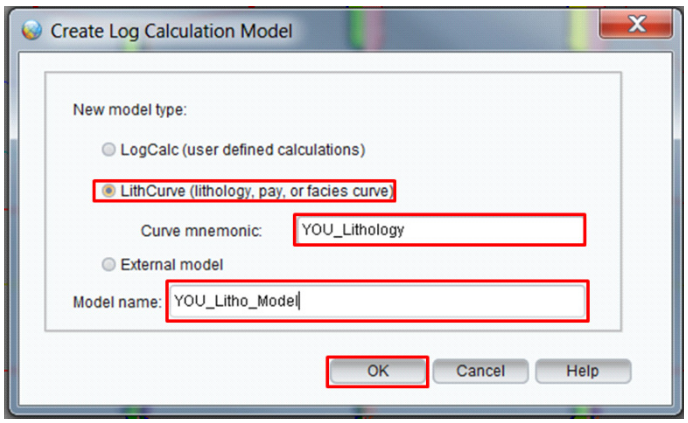


14. On the Perspective pull-down menu, select **Geology.**



> **Note:**
>
> Changing the perspective of a window changes which task panes are displayed. The default perspective shows all task panes, while the Geology perspective displays only task panes associated with geologic workflows.

15. In the *Log Calculator* task pane, click the **Create a new log calculation model** ( ) icon. In the *Create Log Calculation Model* dialog, toggle on **LithCurve (lithology, pay, or facies curve)** and enter **“YOU_Lithology”** in the Curve mnemonic: field. Enter **“YOU_Litho_Model”** in the Model name: field. Click **OK.**



The stratigraphic sequence of the Heidrun Field (Norway) are coarsening-upward, middle Jurassic sandstones controlled by sea level rising and falling. These sandstones were truncated by an Early Cretaceous uplift event. As a result, the top of the youngest producing sandstone is now the base of the Cretaceous, here defined as surface pick FANGST GP. HD Top.

The next producing sandstone is the surface pick FG_2, then FG_1. The surfaces SM have more clay content and are not economical for oil production. For the following exercises, all lithological and petrophysical calculations will be performed starting from the top of FANGST GP. HD Top.

In the next steps, you will use the Gamma Ray log curve to interpret lithology. Note that even though you will use only Gamma Ray for this exercise, the *Log Calculator* in DecisionSpace Geosciences lets you use more than one curve to define lithology.

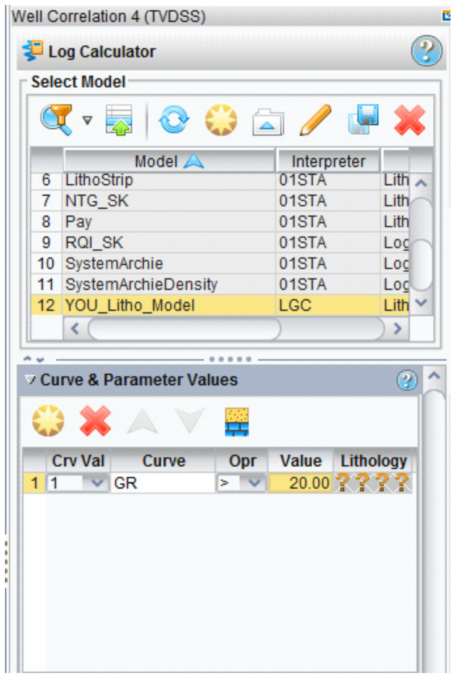
16. In the *Log Calculator* task pane, select model **YOU_Litho_Model** in the *Select Model* panel. In the *Curve & Parameter Values* panel, click the **Add new line** ( ) icon and set the values as follows:

    — Crv Val: **1**

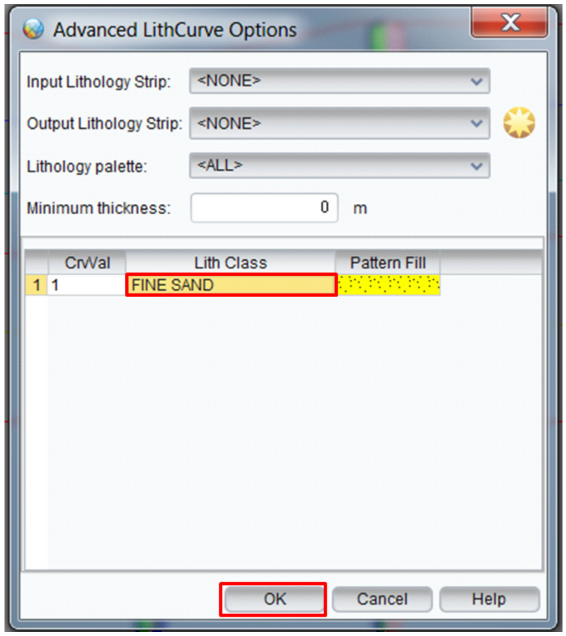
    — Curve: **GR**

    — Opr: >

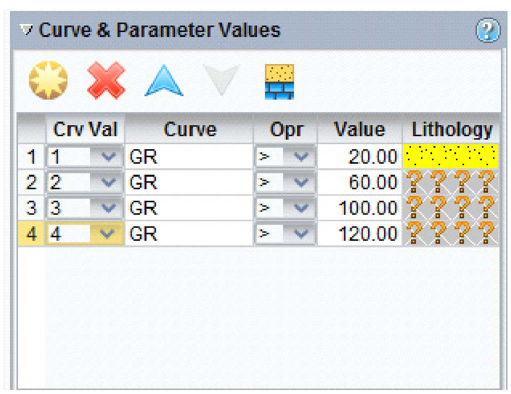
    — Value: Enter **“20”**



17. Click the **Lithology** box. In the *Advanced LithCurve Options* dialog, click **Lith Class** to define Fine Sand. Click **OK.**

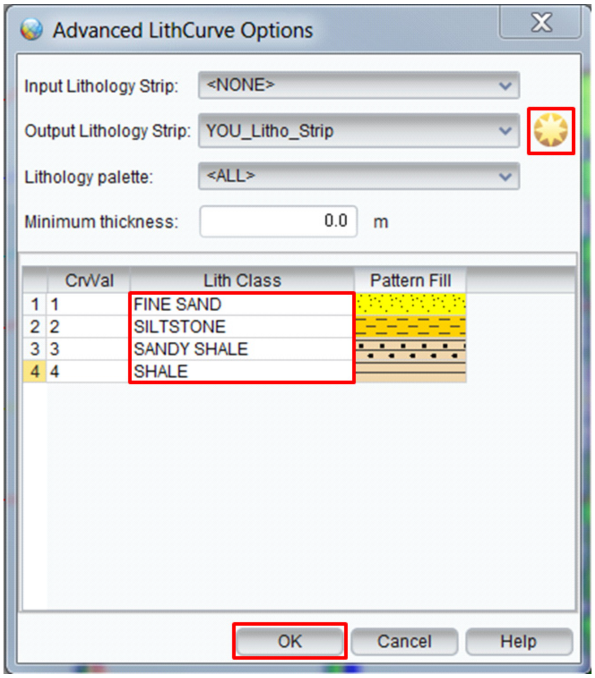


18. In *the Curve & Parameter Values* panel, click the **Add new line** () icon three times. For these new lines, set the parameters as shown in the picture below.

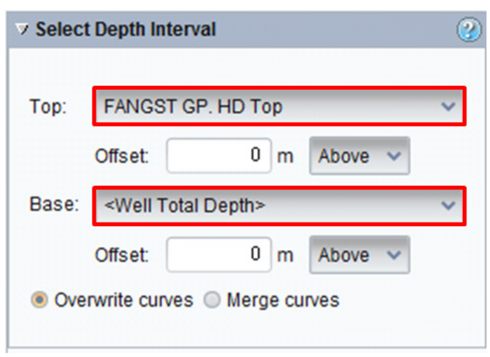


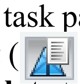
19. Click any of the **Lithology** boxes to open the *Advanced LithCurve Options* dialog, then click the **Create a new Lithology Strip (**  **)** icon. When the *New Output Lithology Strip* dialog appears, name it "**YOU_Litho_Strip**". Click **OK**. This will populate the *Output Lithology Strip* line in the *Advanced LithCurve Options* dialog.

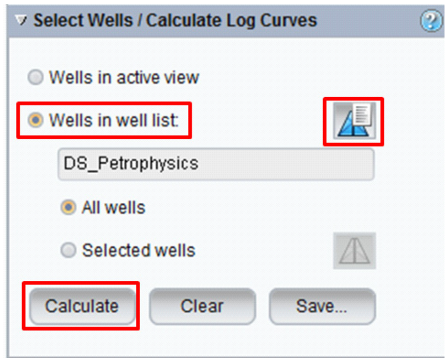
20. Set the parameters for the curve values as shown in the image below, and then click **OK**.



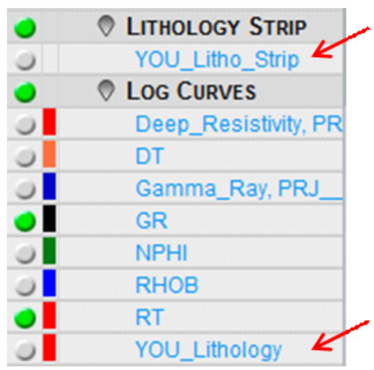
21. To set the interval for which you want the lithology strip calculated, expand the *Select Depth Interval* panel in the *Log Calculator* task pane. In the Top: pull-down menu, select **FANGST GP. HD Top**, and in the Base: pull-down menu, select **<Well Total Depth>**.



22. You also have the option to select which wells you want the lithology strip calculated for. Expand the *Select Wells / Calculate Log Curves* panel in the *Log Calculator* task pane. Toggle on **Wells in well list** and click the **Select well list** () icon to choose the well list **DS_Petrophysics**. Click **Calculate** to calculate both the lithology strip and curve.

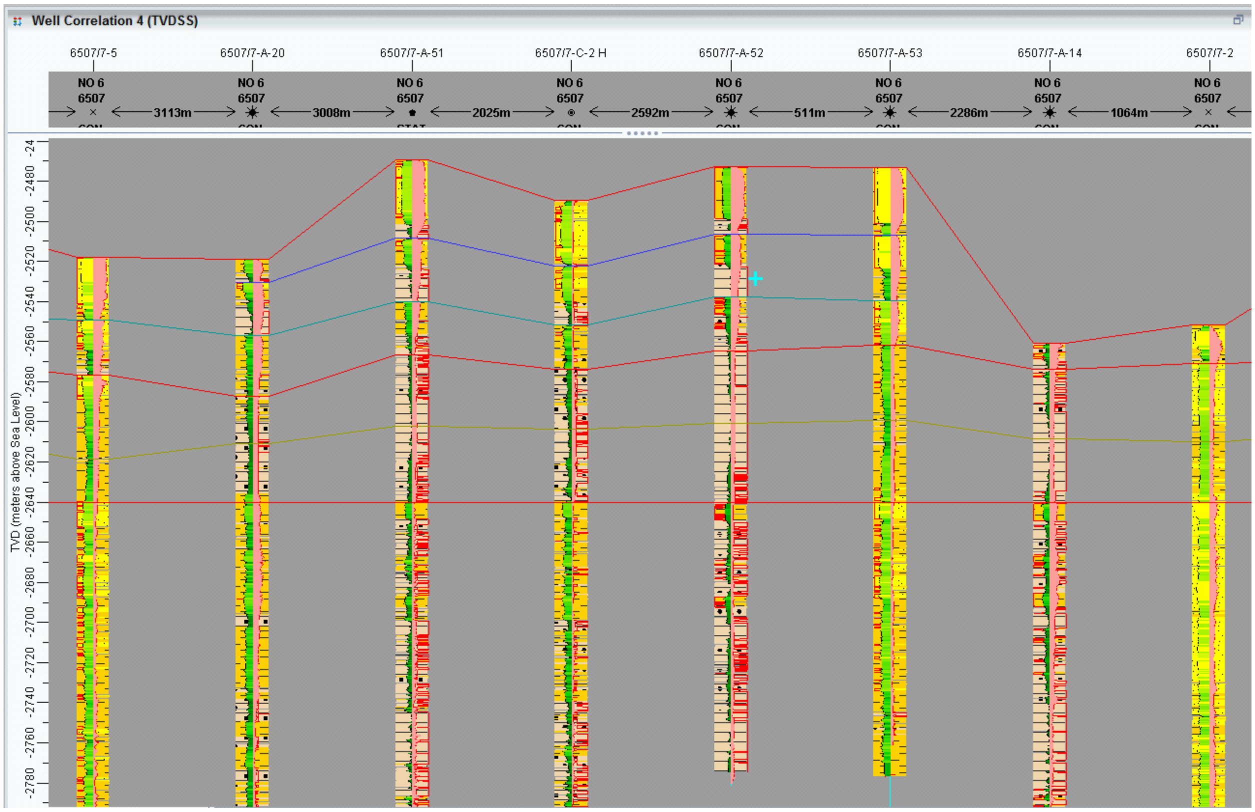


A new category, Lithology Strip, has been added to the *Inventory* with the YOU_Litho_Strip, as well as the YOU_Lithology lith curve added to the Log Curves.



23. Toggle on **YOU_Litho_Strip** to see it in your *Correlation* view.

Your view should look similar to the one below.

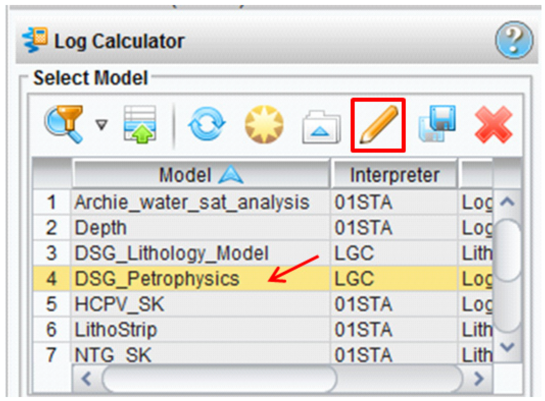


Overview: Interpreting Petrophysics

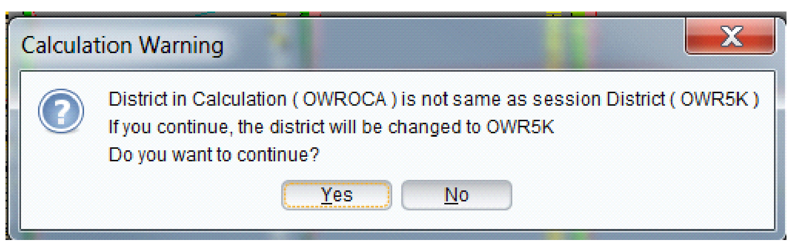
In the following exercise you will calculate average and effective porosities, water saturation, hydrocarbon-pore volume thickness, bulk volume water, and ultimately, define pay, for the reservoir. To do this you will open a previously created petrophysical model to see how to use Log Calculator and observe the calculations that can be made.

Exercise 5.2: Using Log Model Editor

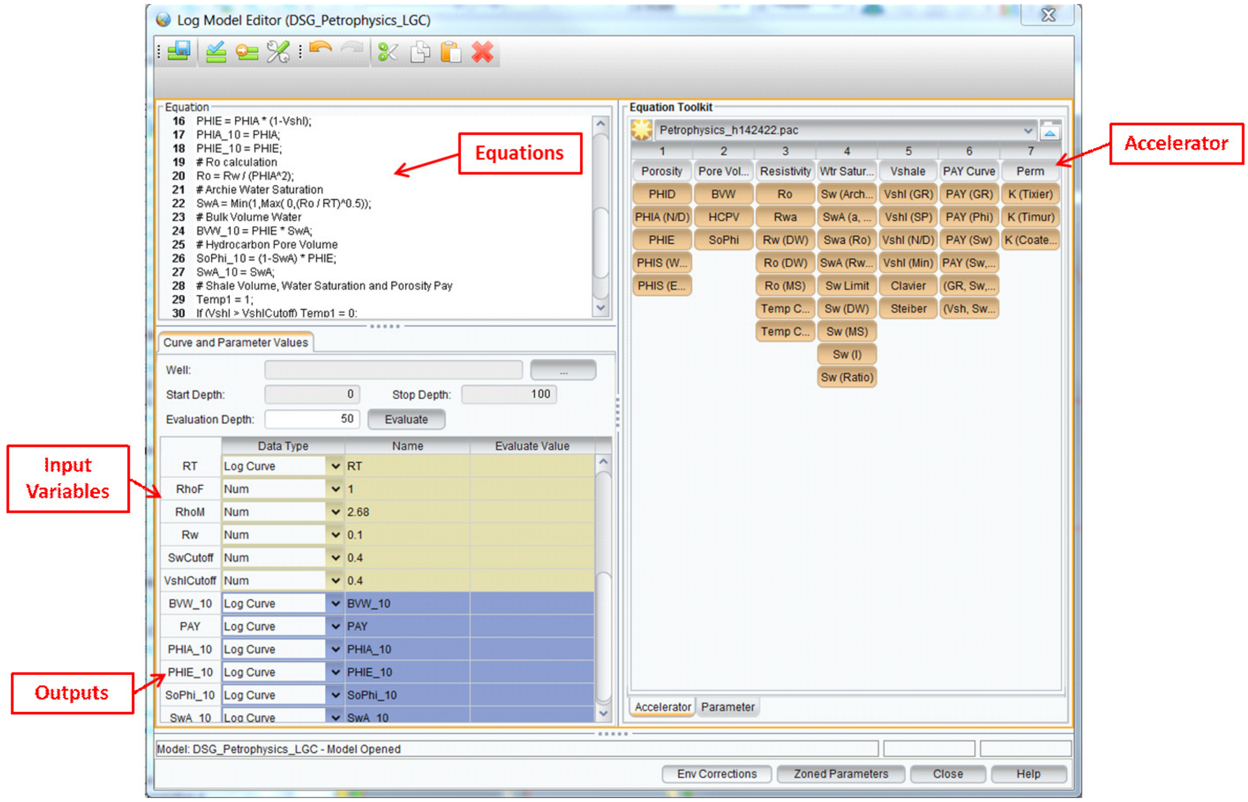
1. In the *Log Calculator* task pane select the **DSG_Petrophysics** petrophysical model, and then click the **Edit the selected model** ( ) icon.



> **Note:**
>
> If a *Calculation Warning* displays, click Yes.



2. The *Log Model Editor* dialog opens. Explore the different portions of the dialog.



The *Equation* panel contains all of the equations that will be used to calculate the designated outputs. It is good practice to make sure that the equations you enter are correctly formatted. To determine if formats are correct, click the Check Equation Syntax ( ) icon. Syntax errors will be identified. When you are satisfied there are no errors, click the Assign Equation Data ( ) icon to populate the *Curve and Parameter Values* tab with all of the input variables and outputs that are specified in the equations.

> **Note:**
>
> To write the equation, you can use the accelerators buttons under the *Equation Toolkit* section or manually type it. The number symbol (#) means an annotation can be used as a title reference, or in some cases to omit any equation line. At the end of every equation line, make sure you add a semicolon (;) to submit the equation in the calculator.

The Accelerator buttons contain equations already created, so you do not have to type them in the Equation box. You can create new accelerators or import existing accelerators as well.

The input variables are a tan row in the *Curve and Parameter Values* tab. You can change how you want the variable defined by a pull-down menu, on which you can choose one of the following: Log Curve, Lithology Strip, Well Interval, Logging Param, Zoned Param, Num, or Undf. Your definition of the variable will dictate what type of value needs to be specified.

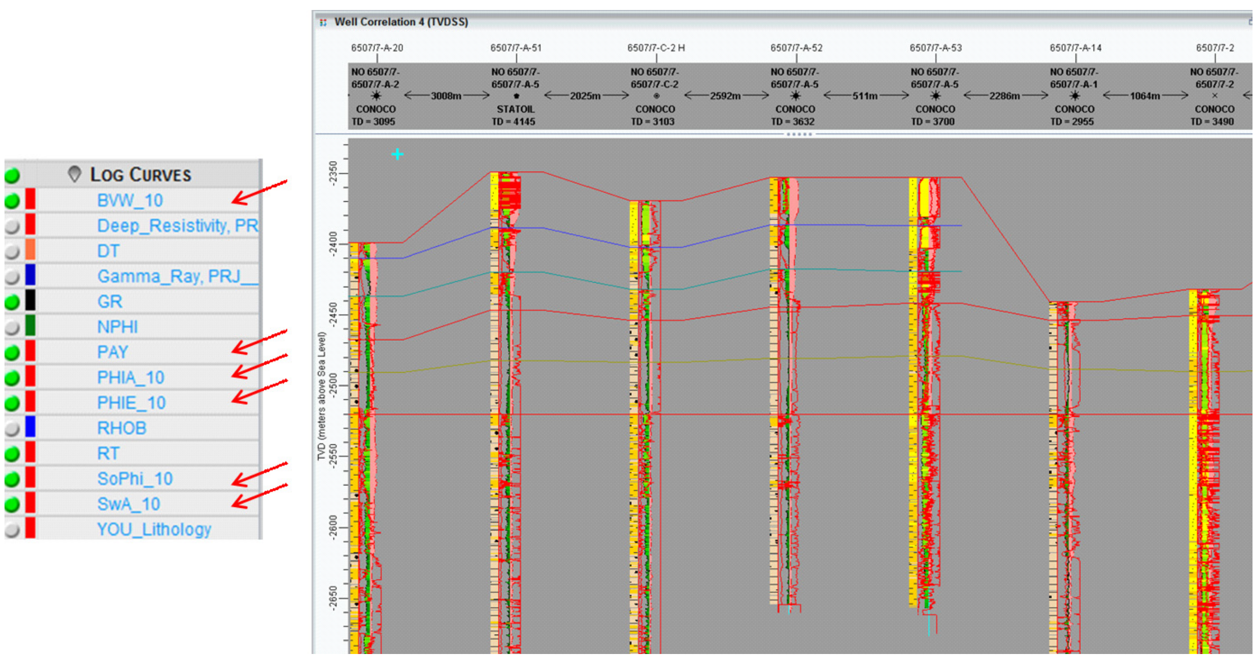
Outputs are a blue row in the *Curve and Parameter Values* tab. From the pull-down menu you can specify whether you want the output value to be calculated as a Log Curve, Lithology Strip, Number, or Undefined.

After you type the equation and define the parameters, you can click the **Save Calculation** ( ) icon to save your work.

3. Click **Close** when you are finished with the *Log Model Editor*.

4. In the *Log Calculator* task pane of the *Select Wells / Calculate Log Curves* panel, ensure that the **DS_Petrophysics** well list is chosen and click **Calculate**.

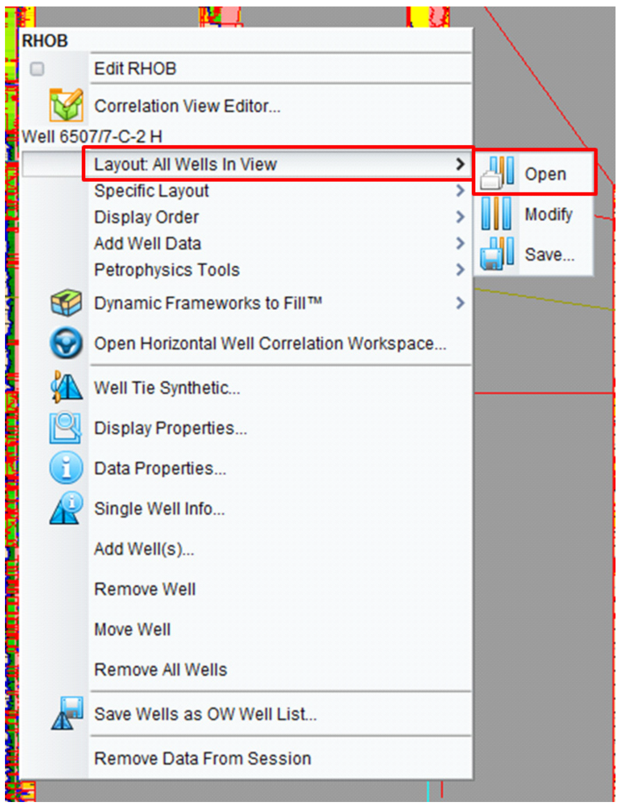
The new curves that were calculated from your Petrophysical model will appear in the *Inventory* and in *Correlation* view.



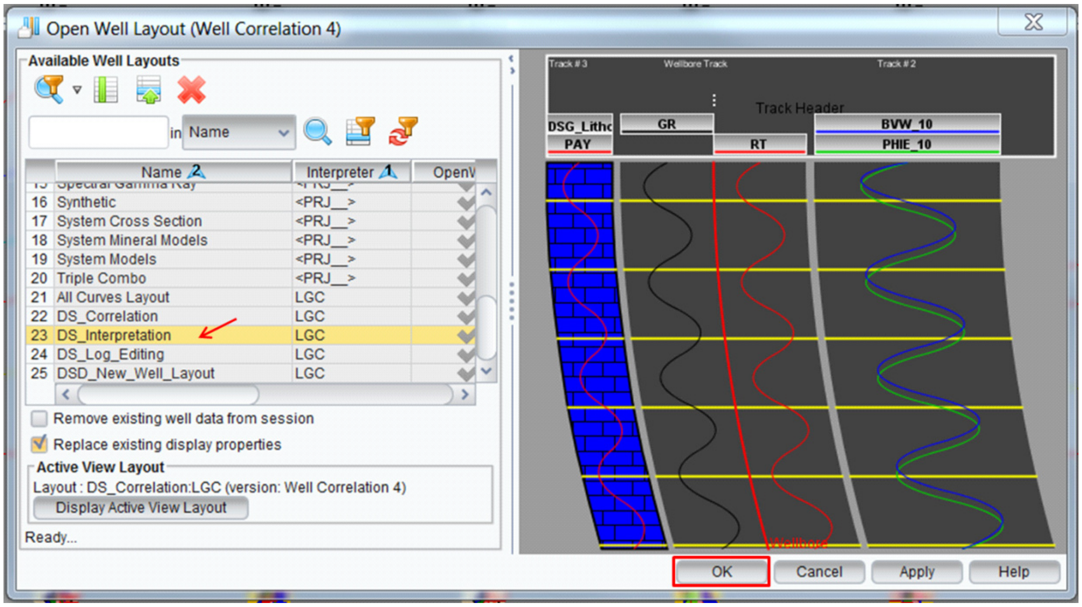
Note:

Do not save the output curves in OpenWorks yet. It is good practice to save the curves to OpenWorks only after you have completed your interpretation.

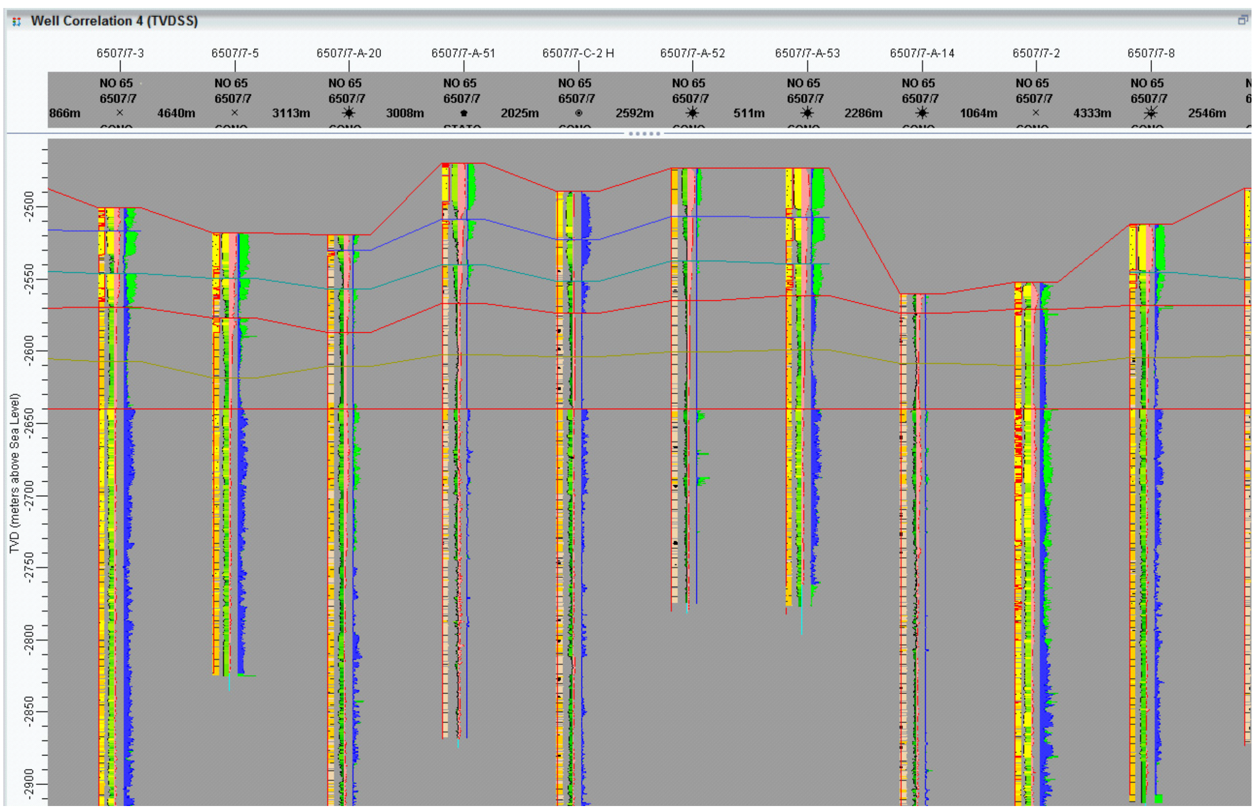
5. To clean up your view you can use a well layout that, for simplification, has already been created. In *Correlation* view put your cursor on any of the wells and **MB3 > Layout: All Wells in View > Open.**



6. In the *Open Well Layout* dialog, select the **DS_Interpretation** well layout and click **OK**.



Your view should look similar to the following image.

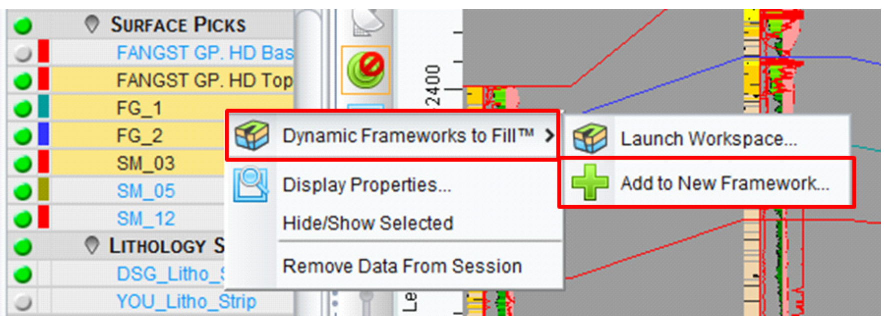


Now that you have calculated water saturation and pay curves you will put them into Dynamic Frameworks to Fill to create attribute maps. Your ability to create attribute maps of these values enables you to compare them and determine the optimum location for drilling.

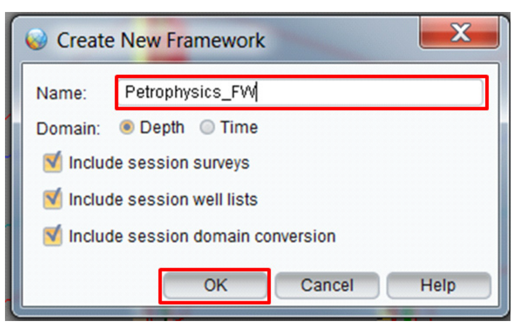
Exercise 5.3: Mapping Petrophysical Attributes in Dynamic Frameworks to Fill

In this exercise you will explore the options that Dynamic Frameworks to Fill provides for including petrophysical attributes in structural models. The dynamic nature of frameworks is also extended to the attributes; this means that any changes in the petrophysical model or new well data will trigger automatic updates to the created interval maps.

1. In the *Inventory* task pane, select the surface picks **FANGST GP. HD Top, FG_1, FG_2,** and **SM_03**, the main surfaces within your reservoir. Put your cursor over the selection, then **MB3 > Dynamic Frameworks to Fill > Add to New Framework.**

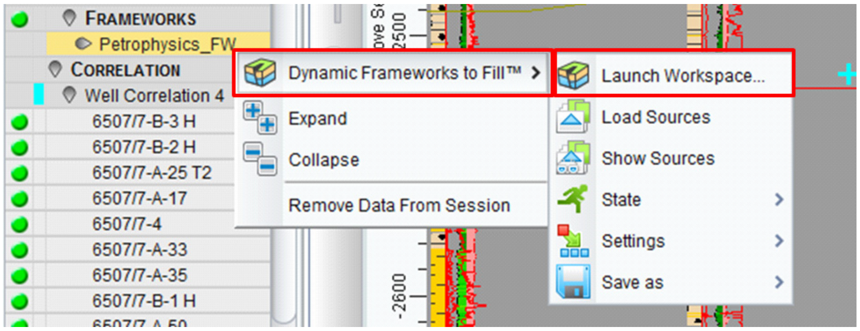


2. The *Create New Framework* dialog opens. In the Name: field, enter **"Petrophysics_FW"** and accept all of the other default settings. Click **OK**.

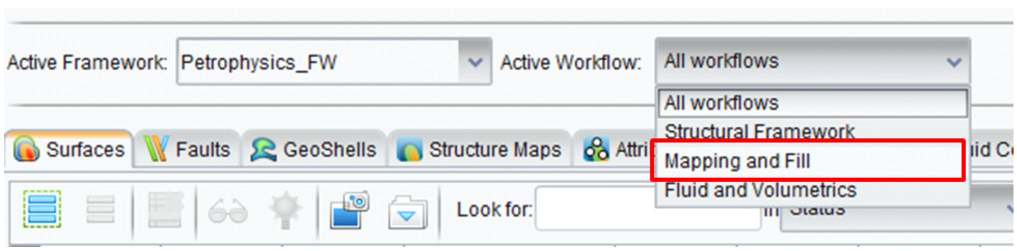


3. In the *Add Sources to Framework* dialog, accept all of the defaults and click **OK**.

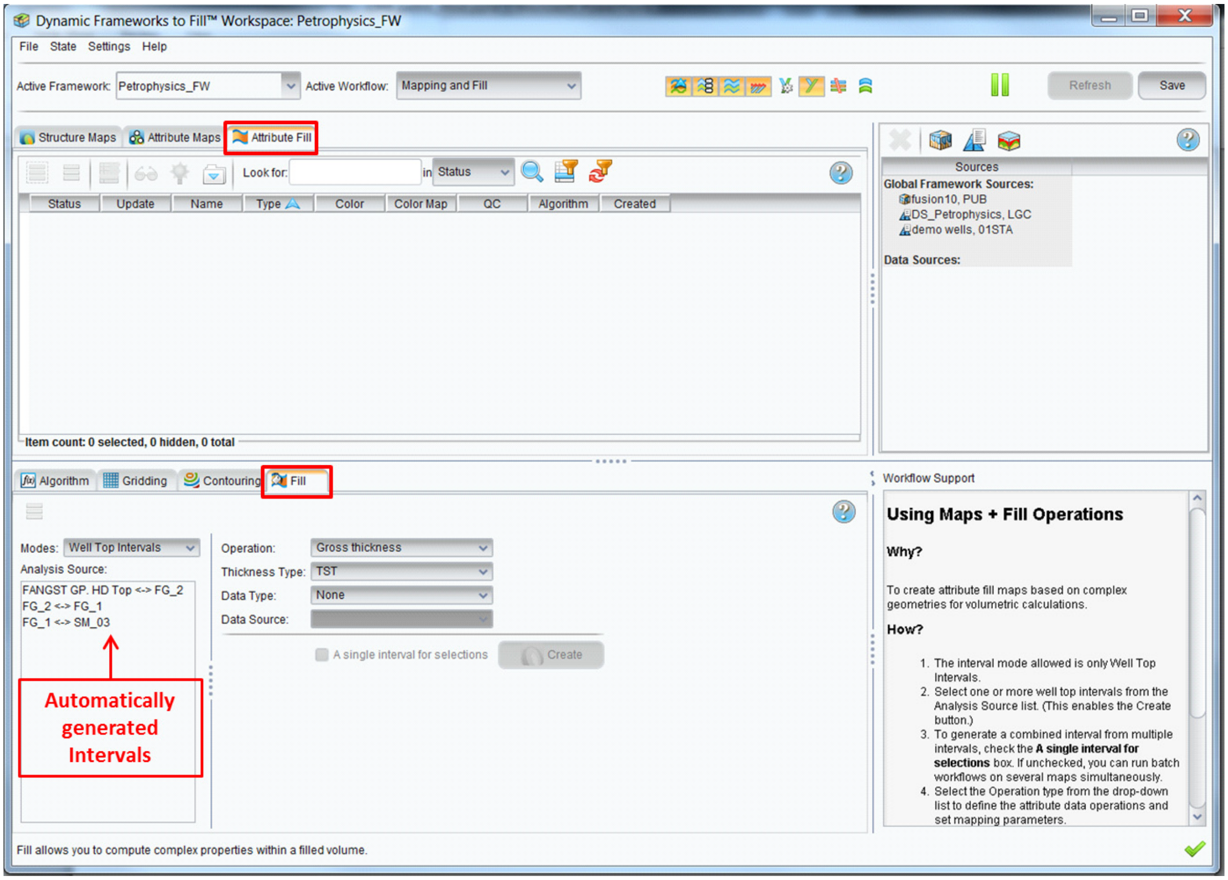
4. In the *Inventory* task pane, put your cursor on **Petrophysics_FW** framework and **MB3 > Dynamic Frameworks to Fill > Launch Workspace.**



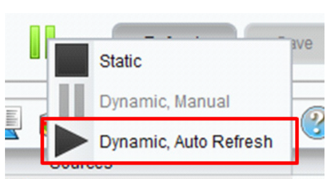
5. In the Active Workflow: pull-down menu on the *Dynamic Frameworks to Fill* dialog, select **Mapping and Fill**. This will reduce the number of object tabs and actions tabs and will leave only those necessary to work with attributes.



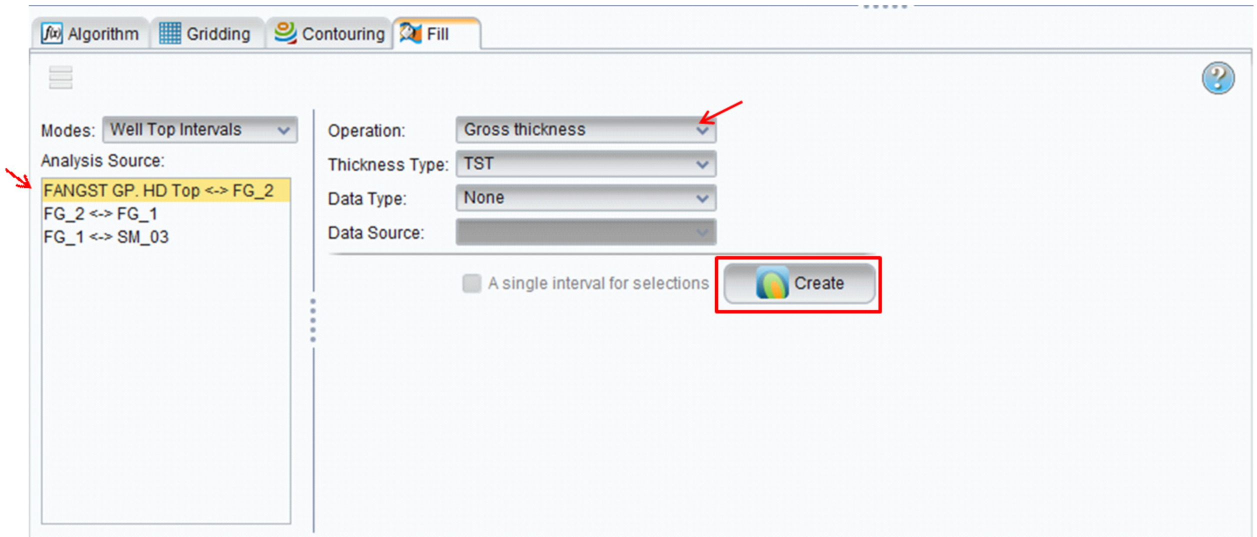
6. Activate the *Attribute Fill* object tab and *Fill* action tab. Notice that Dynamic Frameworks to Fill has automatically generated intervals. The software uses stratigraphic sequential order to define the intervals from the surface picks that you added to the framework.



7. Before you calculate any Fill, change the state of your framework. To do this, click the pause button at the top of the workspace and select **Dynamic, Auto Refresh**. The framework will then update automatically every time there is a change made to it.



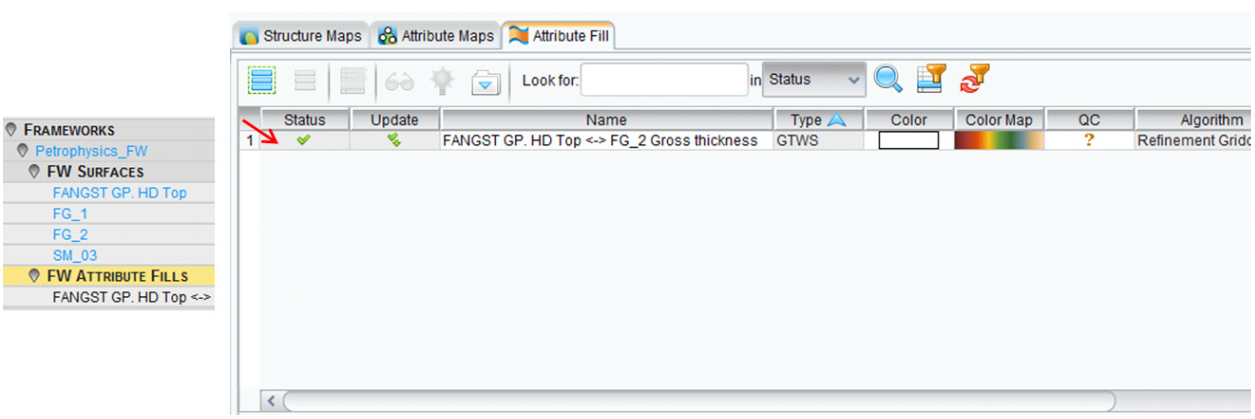
8. Highlight the first Interval, **FANGST GP. HD Top <-> FG_2**. Note that you can execute all attributes operations from here. In the Operation: pull-down menu, select **Gross thickness**. Click the **Create** button.

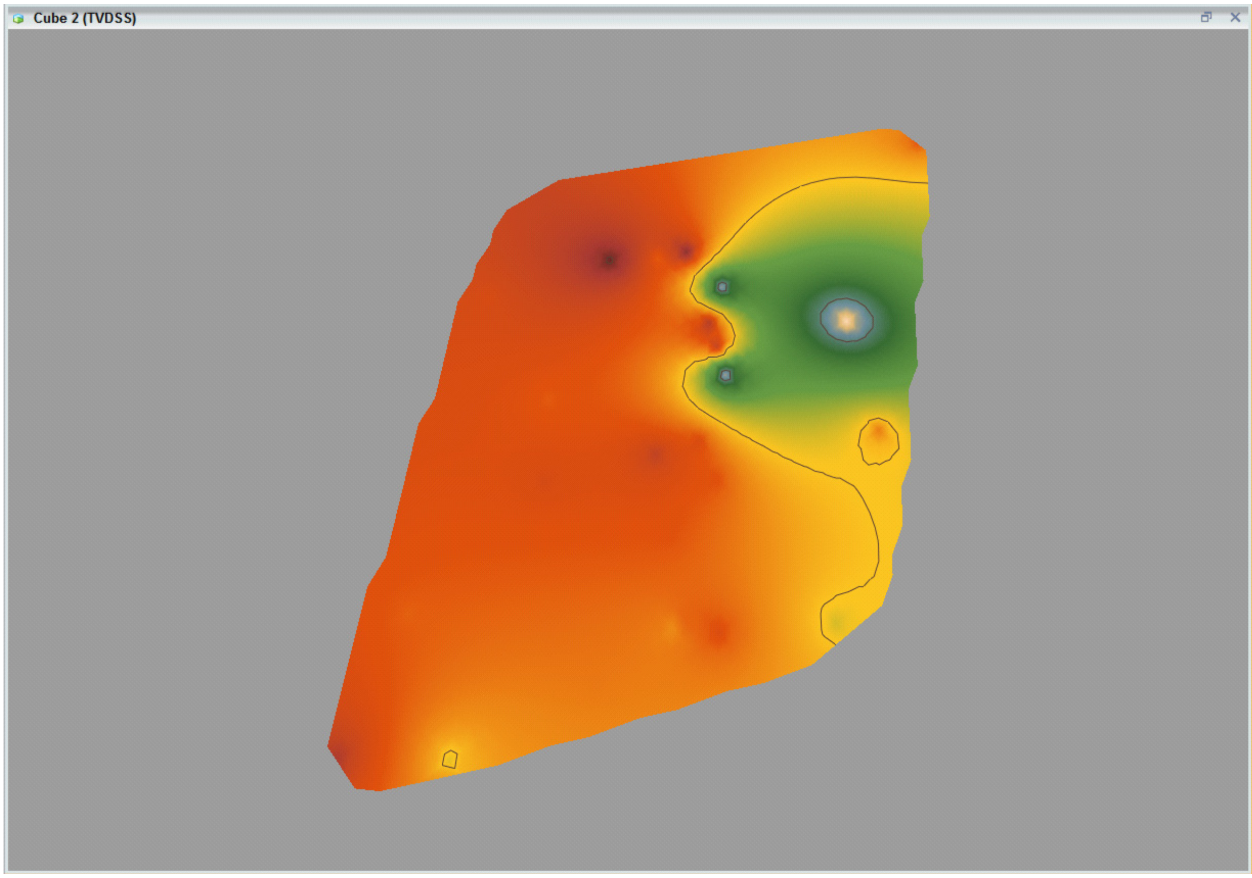


Note:

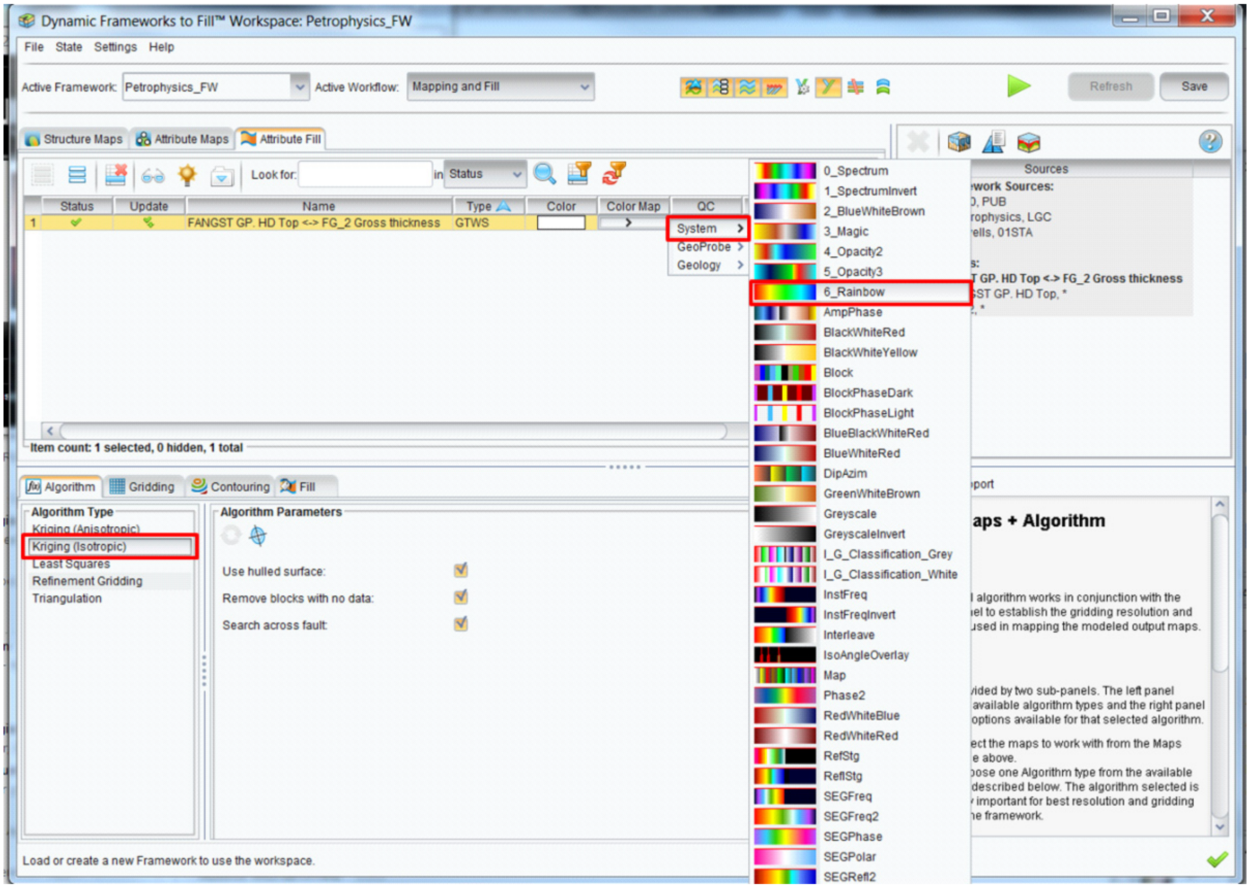
Make sure, if you are using DecisionSpace Geosciences version 5000.10.0.02 or higher, that Surfaces (Well Tops) is selected as the Mode for interval generation, in all of the following attribute fill calculations.

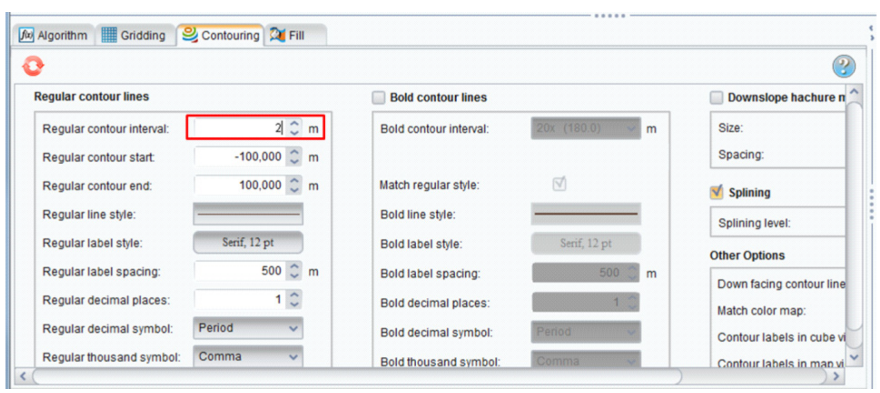
9. The newly created Attribute Fill will appear in both the *Attribute Fill* object tab, as well as in the *Inventory* task pane under Frameworks > Petrophysics_FW > FW Attribute Fills. Maximize ***Cube*** view and toggle on the **FANGST GP. HD Top <-> FG_2 Gross thickness** Attribute Fill.



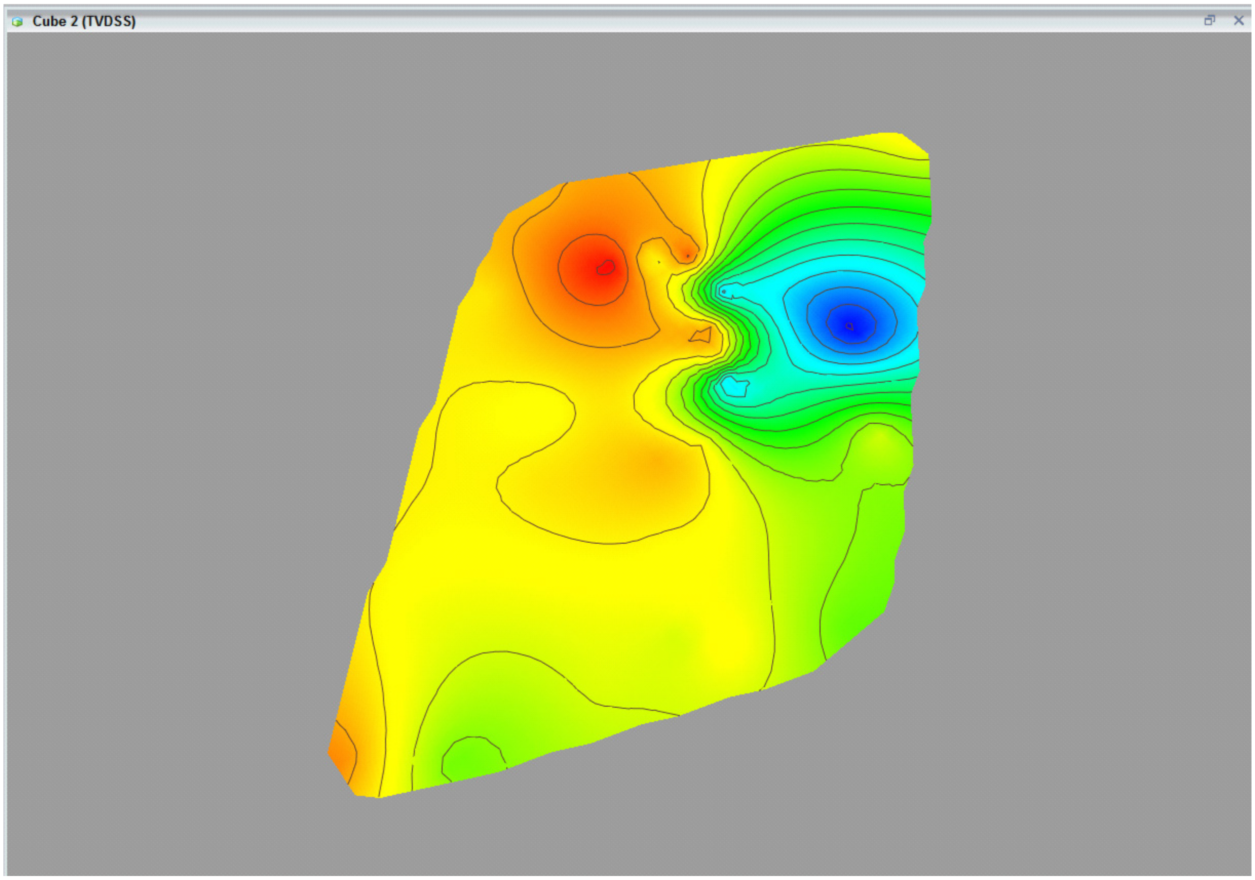


10. Under the *Attribute Fill* object tab in the *Dynamic Frameworks to Fill Workspace*, highlight the attribute **FANGST GP. HD Top <-> FG_2 Gross thickness**. Click the **Color Map** cell and on the menu that is displayed and select **System > Rainbow** color palette. Click the ***Algorithm*** action tab and select **Kriging (Isotropic)**. Click the ***Contouring*** action tab and change the Regular contour interval to **2m**. All changes will happen automatically in *Cube* view.





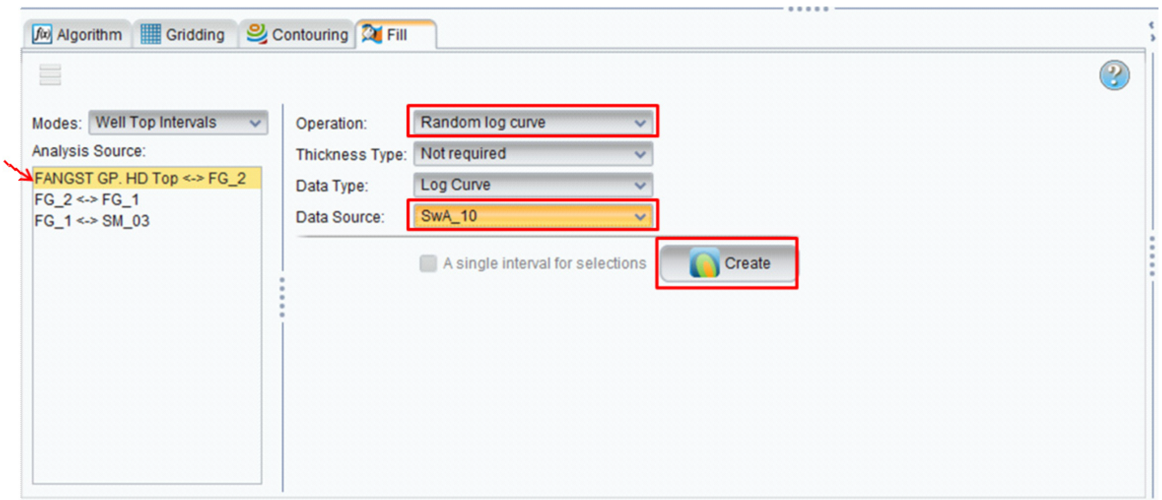
Your view should look similar to the following image.



At this point you have learned how to create attributes between framework intervals, and how to modify the mapping parameters. In the following steps you will use the petrophysical logs to create more attributes and you will see the dynamic updatability of frameworks.

11. Navigate back to the *Fill* action tab for the *Attribute Fill* object tab. In the *Attribute Fill* object tab, click the **Deselect All** ( ) icon to activate the well top intervals in the *Fill* object tab.

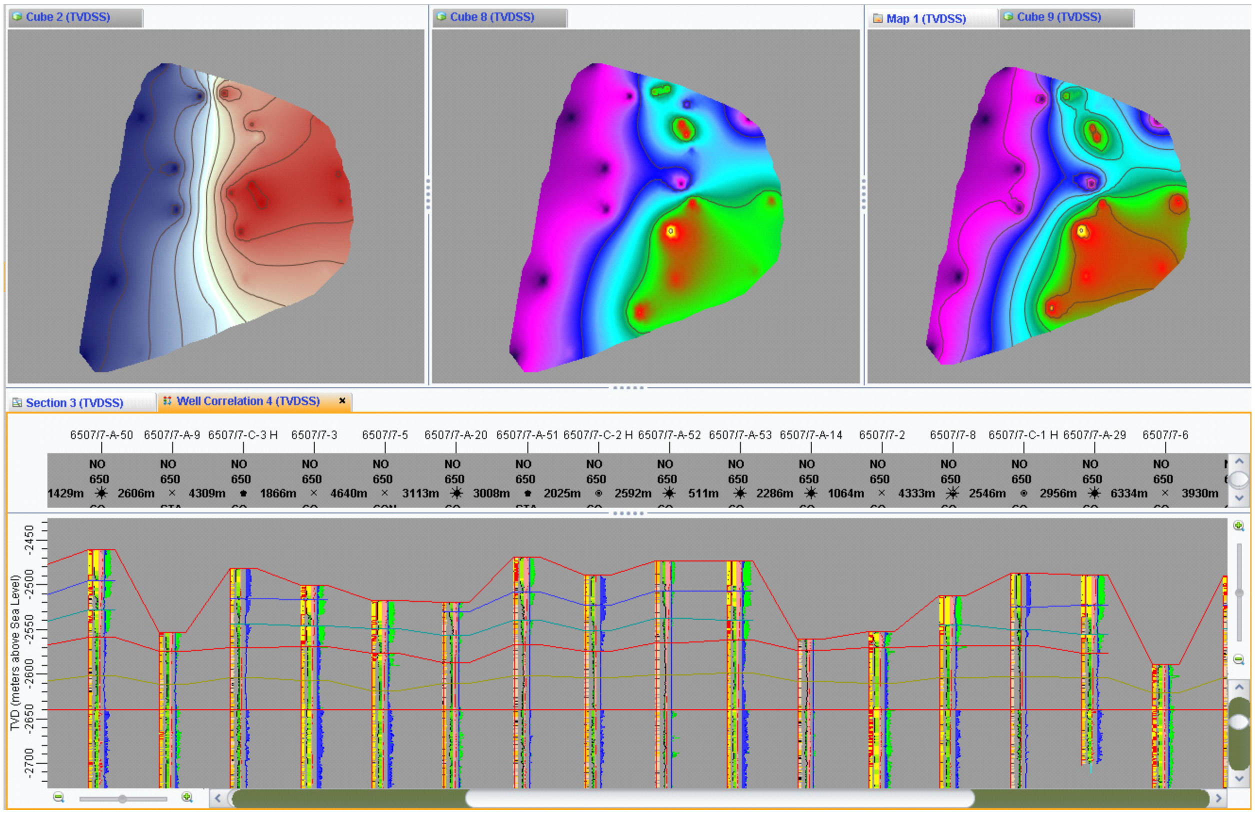
12. Select **FANGST GP. HD Top <-> FG_2** interval in the *Analysis Source* table. On the Operation pull-down menu, select **Random log Curve**, and on the **Data Source** pull-down menu, select **SwA_10**. These actions will create an average map of the calculated water saturation curve. Click **Create.**



13. The new attribute will appear in the *Attribute Fill* tab. Double-click the **Name** column of the attribute just created and change it to **“FANGST GP. HD Top <-> FG_2 SwA_10”.**

14. Repeat the instructions from steps 11 and 12 to create two more attributes for the same interval: **Net to gross ratio** and **Net thickness**. For both attributes, select the **PAY** curve in **Data Source**. Change the **color map** as desired.

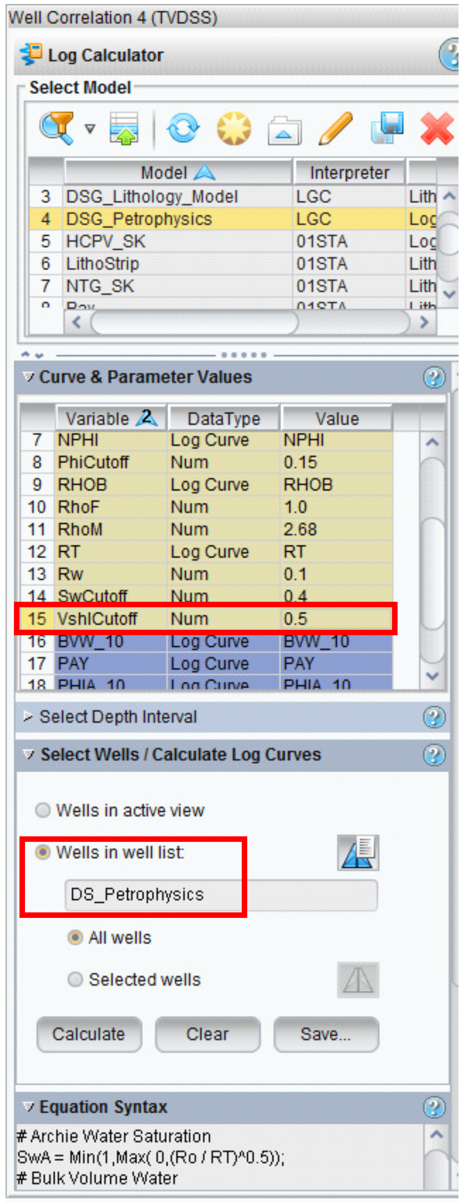
15. Open two more ***Cube* views** and arrange your display as shown in the image below. The *Cube* view on the left displays FANGST GP. HD Top <-> FG_2 SwA_10 (BlueWhiteRed color map). The center *Cube* view displays FANGST GP. HD Top <-> FG_2 Net Thickness (Spectrum color map). The right *Cube* view shows FANGST GP. HD Top <-> FG_2 Net to Gross ratio (Spectrum color map). Move your ***Correlation* view** to the bottom of your display.



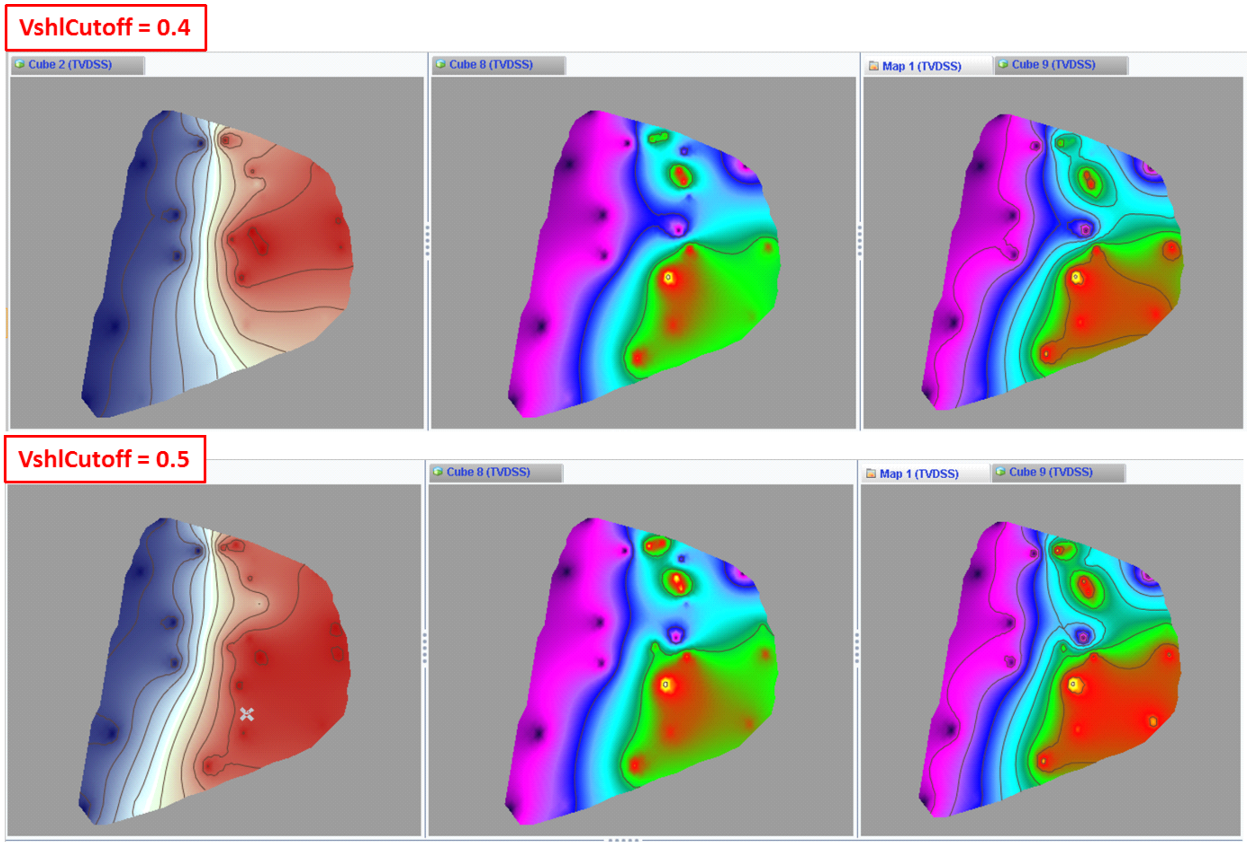
In the above image the three attributes maps are showing a favorable pay at the western portion of the area. Notice low water saturation and higher values of Net Thickness and Net to Gross ratio.

As noted before, any change to your data or the parameters of your petrophysical model will cause your framework to change. You will now adjust some of the parameters within your petrophysical model, to observe the dynamic updateability of Dynamic Frameworks to Fill.

16. Activate *Well Correlation* view and then click the ***Log Calculator*** task pane. In *the Curve & Parameter Values* panel, double-click the current value of **VshlCutoff**, and then enter **"0.5"**. In the *Select Wells / Calculate Log Curves* panel make sure **Wells in well list** is toggled on and that **DS_Petrophysics** is selected. Click the **Calculate** button.


Well Correlation 4 (TVDSS)
Log Calculator
Select Model
Model
Interpreter
3 DSG_Lithology_Model LGC
4 DSG_Petrophysics LGC
5 HCPV_SK 01STA
6 LithoStrip 01STA
7 NTG_SK 01STA
Curve & Parameter Values
Variable DataType Value
7 NPHI Log Curve NPHI
8 PhiCutoff Num 0.15
9 RHOB Log Curve RHOB
10 RhoF Num 1.0
11 RhoM Num 2.68
12 RT Log Curve RT
13 Rw Num 0.1
14 SwCutoff Num 0.4
15 VshlCutoff Num 0.5
16 BVW_10 Log Curve BVW_10
17 PAY Log Curve PAY
Select Depth Interval
Select Wells / Calculate Log Curves
Wells in active view
Wells in well list:
DS_Petrophysics
All wells
Selected wells
Calculate
Clear
Save...
Equation Syntax
# Archie Water Saturation
SwA = Min(1,Max(0,(Ro / RT)^0.5));
# Bulk Volume Water

When you click the Calculate button, all of the Attribute Fill maps will automatically update. Observe the changes made to your maps.



If you have time, create more attributes for the intervals **FG_2 <-> FG_1** and **FG_1 <-> SM_03,** following the given instructions.

17. Select **File > Save Session As.** In the *Save Session As* dialog, enter **"YOU_CH5".**

Chapter 6
Volumetrics

At a series of critical points in the life of a field or a prospect, volumetrics calculations are required. DecisionSpace Geosciences has methods that enable very accurate determinations of gross rock volumes (GRV), as well as stock-tank original oil-in-place (STOOIP) and recoverable hydrocarbons.

Topics Covered in this Chapter

In this chapter you will be learn about:

- Creating sealed spaces (compartments)
- Volumetrics and fluid contacts
- Volumetrics charts
- Creating custom reservoirs

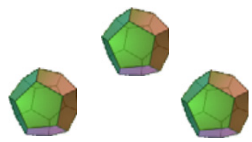
Volume Calculation Technologies

The new volumetric visualization solution is referred to as Visual Volumetrics. This method uses 3D sealed spaces, which are called compartments, to enable visualization of the body in any view. Compartments are analyzed using highly accurate volume calculations. In particular, these technologies represent significant improvements over the traditional slicing method, wherein the body of volume calculation is complex. Three compartment-based volume calculation methods are used:

- Direct polyhedral volume calculation
- Fast-sweep thickness extraction
- Slice-based volumetrics

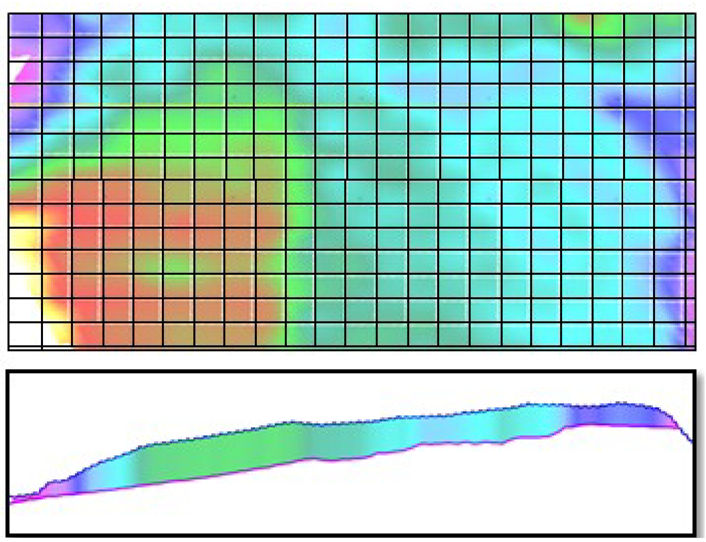
Direct Polyhedral Volume Calculation

GRVs are computed directly from the compartment geometry, which is a significant improvement over traditional slicing techniques. The compartment GRV is calculated through the Direct Polyhedral Volume Calculation technology, which offers a highly accurate deterministic volume calculation. This technique is based on the mathematical definition of a polyhedron volume and the quantity of polyhedrons required to fit into a given compartment geometry.



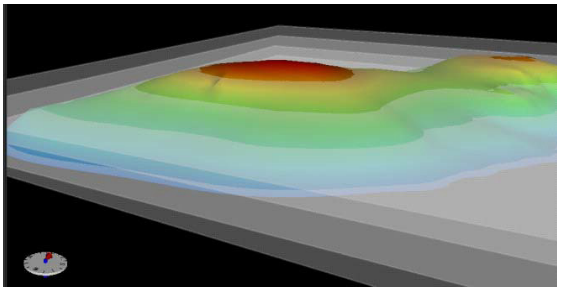
Fast-Sweep Thickness Extraction

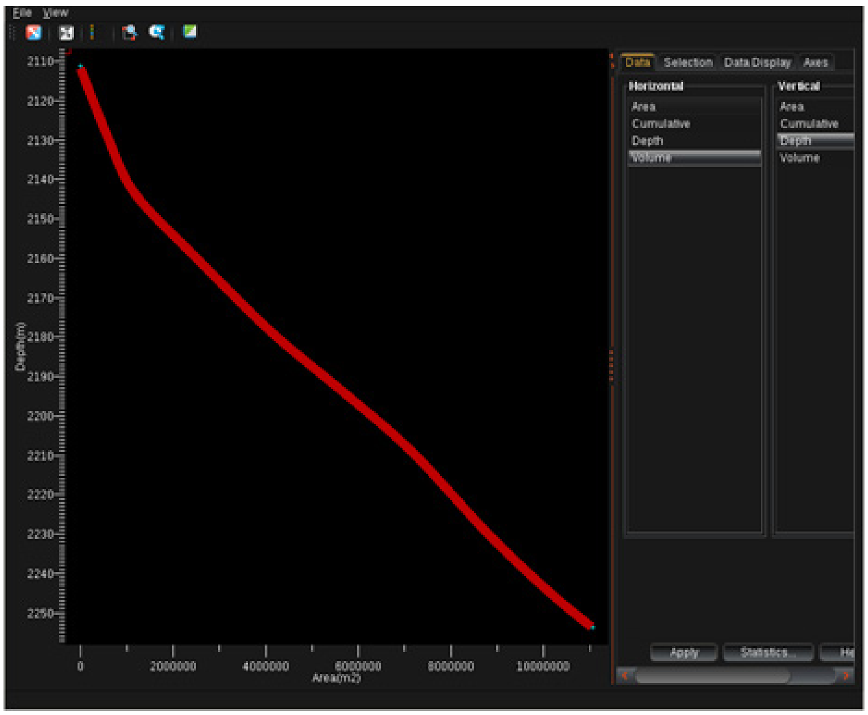
This very fast technique is based on the extraction of a gross thickness grid representing the vertical thickness of a compartment at each grid node. The extracted thickness grid is unrotated. It has a dimension of 1000 x 1000, matching the XY extents of the compartment. Fast-sweep thickness extraction is the underlying volume calculation technique for grid-based volumetrics operations. Gross thickness maps can be multiplied by a constant value or by one or more attribute maps or grids with laterally-varying attributes to produce net thickness grids.



Slice-Based Volumetric Calculations

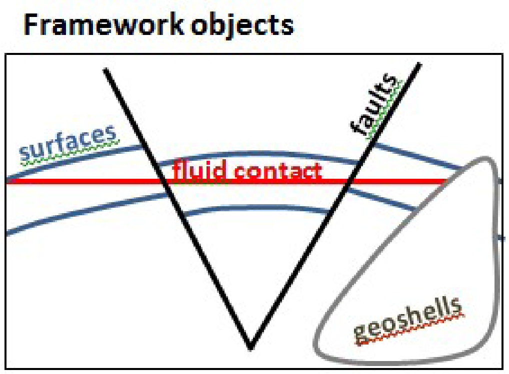
Traditional slice-based volumetric calculations are used for backward compatibility, as well as for plotting (e.g., depth/area graphs). Compartments are sliced horizontally to calculate the volume and surface area of each volumetric slice.

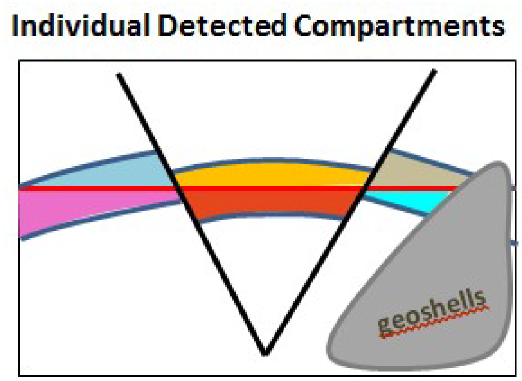




Compartment Technology

Compartments are groups of solid bodies that are triangular-mesh representations of sealed spaces. A sealed space is bounded by framework objects such as surfaces, faults, geoshells, fluid contacts, and the surface boundaries. Compartments are automatically detected and grouped into different categories. They automatically update to any changes of the framework and source data.

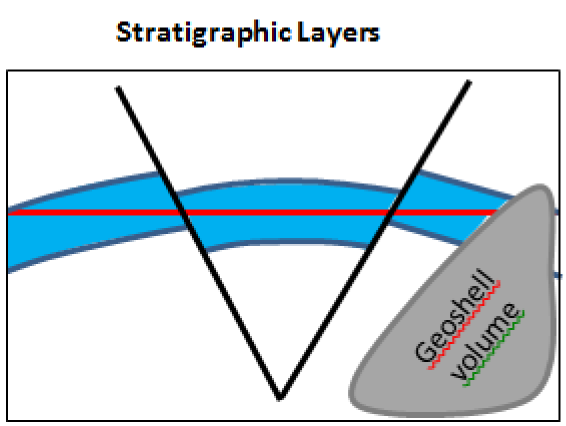




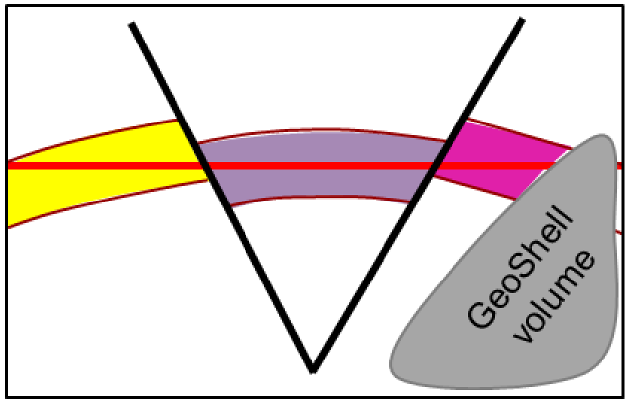
Compartment GeoGrouping

Individual compartments (sealed spaces) are automatically grouped into geological categories defined by the bounding framework objects.

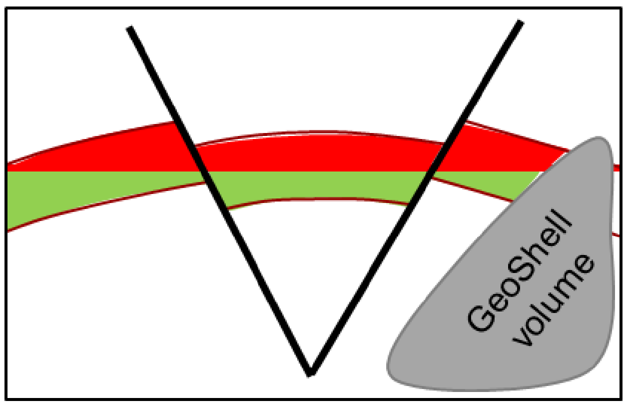
- All individual compartments between two surfaces are grouped into stratigraphic layers.



- All individual compartments between sealing faults are grouped into fault blocks.



- All individual compartments divided by a fluid contact are grouped into fluid layers.



- A geoshell becomes a geoshell volume compartment.

- You can create custom reservoirs manually, by combining multiple compartments or using the overlapping portion of multiple compartments.

Advanced Compartment Functionality

The concept of visual volumetrics is based on the visualization and display of compartments as bodies of volume calculations for improved accuracy and user experience. Several improvements have been made to compartments to optimize compartment functionality and display:

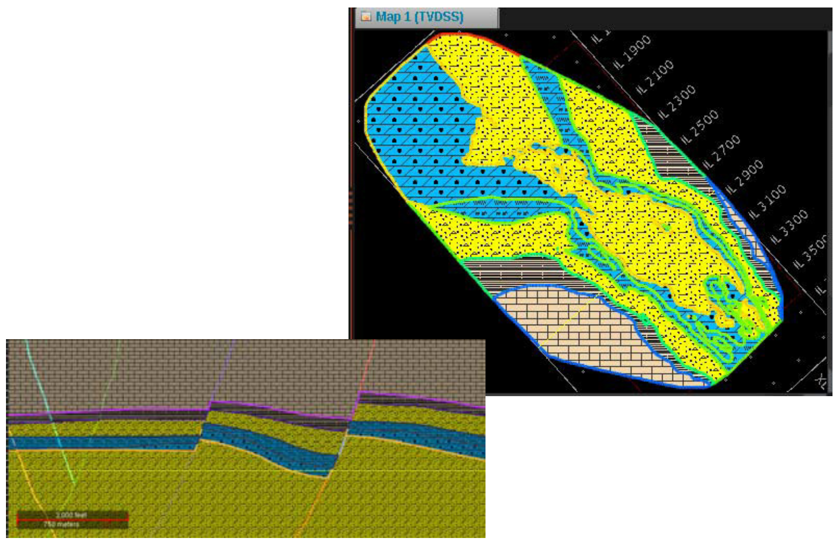
- Color and lithology fills for compartments

- Options for creating custom reservoirs

- Ability to bound a compartment by volume of interest (VOI) polygon and by depth
- Compartment attribute overlays

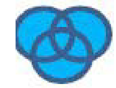
Compartment Color and Lithology Fill

Through the *Display Properties* dialog you can now color-fill compartments in all views. You can also fill compartments with lithology patterns in *Section* and *Map* views.



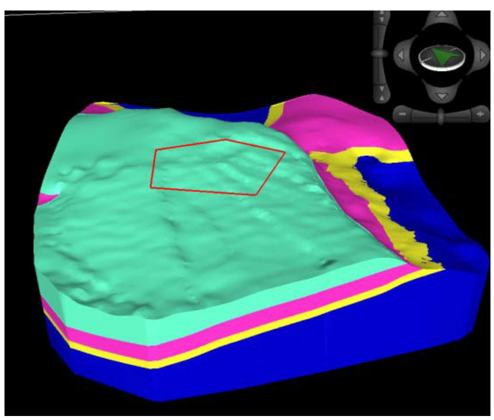
Creating Custom Reservoirs

This feature gives you the option of creating custom reservoirs (compartments) by an intersect and merge operation. The purpose of this feature is to facilitate optimal compartment combinations for volumetric calculations.

1. Merge selected compartments to custom reservoir.	
2. Create custom reservoirs from any intersecting portions of the selected compartments.	
3. Create custom reservoirs from intersecting portions of all of the select compartments.	

Compartments

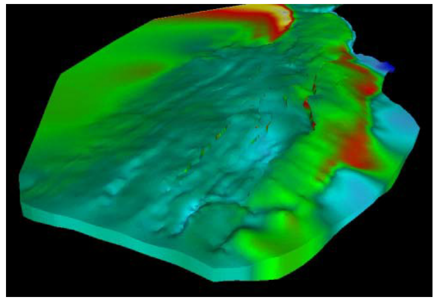
To constrain the compartment model to the desired size using polygons or geoshapers, as well as depth values, you can specify a VOI and Z values in the *Compartments* object tab of the *Dynamic Frameworks to Fill Workspace*.



Section and *Cube* View Attribute Overlay

3D overlay of attributes (seismic attributes, interval fills, and so forth) is available for compartments in *Cube* view and in *Section* view.

This is an analog functionality to the attribute overlay on framework surfaces in *Cube* view.



Exercise 6.1: Creating Compartments

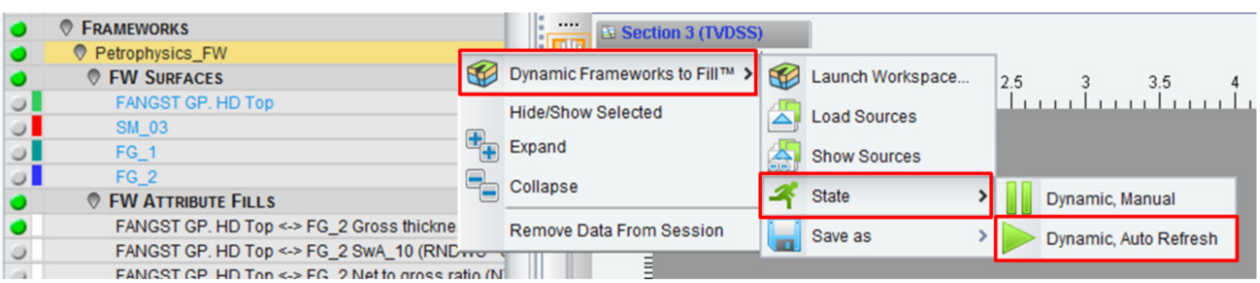
In the previous chapter you ran a petrophysical model to create multiple log curves. You then used those log curves to create attribute fill maps of the FANGST GP.HD Top <-> FG_2 interval. In the following exercise you will learn how to create stratigraphic layer compartments in Dynamic Frameworks to Fill. These compartments will contain the FANGST GP.HD Top <-> FG_2 interval, which you will use for volumetric calculations later in the chapter.

1. If you did not close your session when you finished Chapter 5, continue with step 3, below. If you did close DecisionSpace after you saved your session in Chapter 5, **open** the session now.

You will also need to run the DSG_Petrophysics module again. Although the curves appear in your session, they are not currently populated with data. Curves created in the Log Calculator are in memory until you save them to OpenWorks. Not saving them to OpenWorks will remove the dynamic updatability of any Attribute Fills made with those curves.

2. In the *Log Calculator* task pane select the **DSG_Petrophysics** model, set the Depth Interval to **FANGST GP. HD Top to <Well Total Depth>,** toggle on **Wells in Well list** and set the well list as **DS_Petrophysics**, and then select **Calculate**.

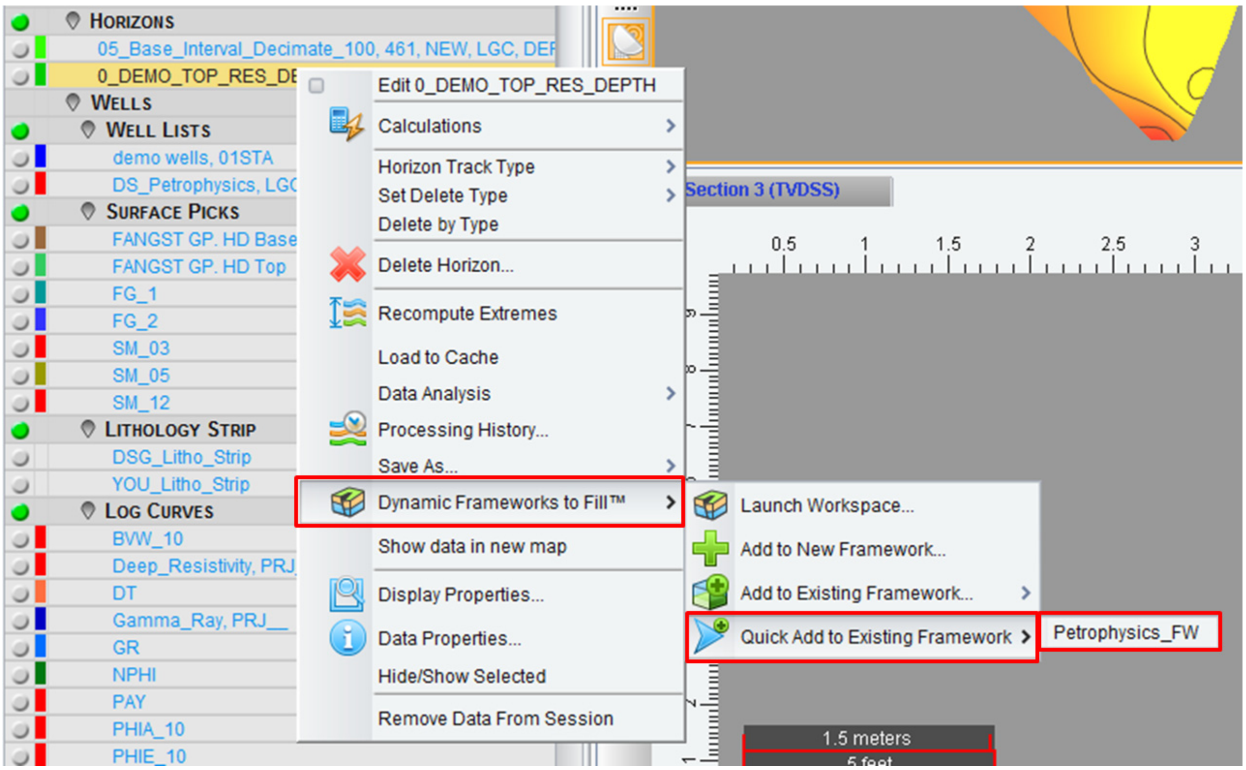
3. In the *Inventory* task pane put your cursor on **Petrophysics_FW** and **MB3 > Dynamic Frameworks to Fill > State > Dynamic Auto Refresh** (if not already refreshed)**.**



Dynamic Frameworks to Fill uses compartments to calculate volumetrics. The compartments are created from sealed framework objects. To ensure consistency in the compartments, you will adjust the grid extents and geological relationships of all of the surfaces in your framework.

In the following steps you will add a seismic surface to your framework as a guiding surface to conform naive framework surfaces.

4. In the *Inventory* task pane put your cursor on **O_DEMO_TOP_RES_DEPTH** and **MB3 > Dynamic Frameworks to Fill > Quick Add to Existing Framework > Petrophysics_FW**. This will add the 0_DEMO_TOP_RES_DEPTH horizon to the framework as a new framework surface.



5. In the *Dynamic Frameworks to Fill* task pane, click the **Launch Framework Workspace Workflow Window (** **)** icon to open the *Dynamic Frameworks to Fill Workspace*.

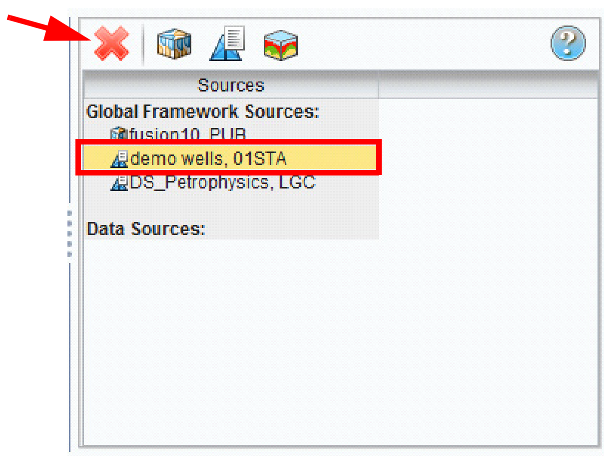
In the *Surfaces* object tab of the *Dynamic Frameworks to Fill Workspace* you can see that the new 0_DEMO_TOP_RES_DEPTH framework surface has been created.

6. If your active workflow is Mapping and Fill from the previous exercise, change it into All Workflows so you can see the *Surfaces* object tab.

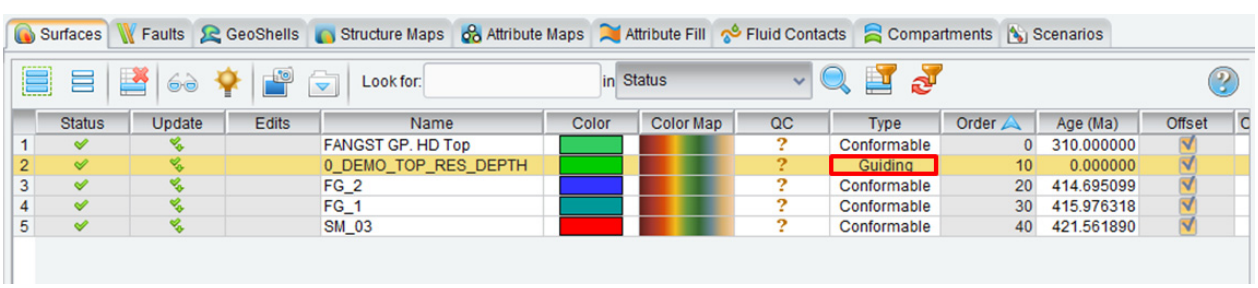
You will be using this as a guide for all of your other framework surfaces. In earlier chapters you added it as a secondary source that was then left as a conformable surface; in this exercise you will change it to a guiding surface. It can then be used when you apply conformance, but it will not be recognized during compartment or interval generation.

Notice also that in global sources you have two well lists: demo wells and DS_Petrophysics. All of the petrophysical calculations were executed only on the wells in the DS_Petrophysics well list. To avoid artifacts, you will remove demo wells from your framework, so you are only working with those wells with petrophysical logs.

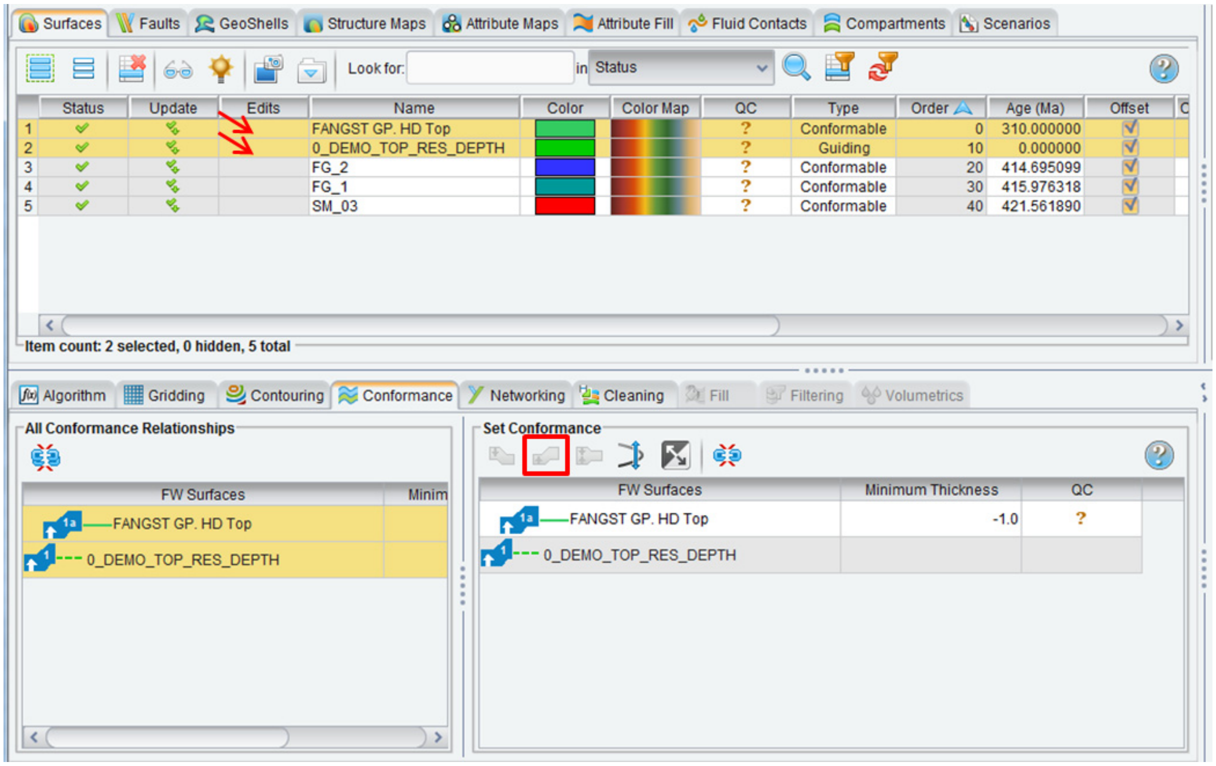
7. In the *Global Framework Sources* panel, highlight the **demo wells** well list, and select the **Delete the selected source** icon. This will remove the well list from your framework and leave only the DS_Petrophysics well list.



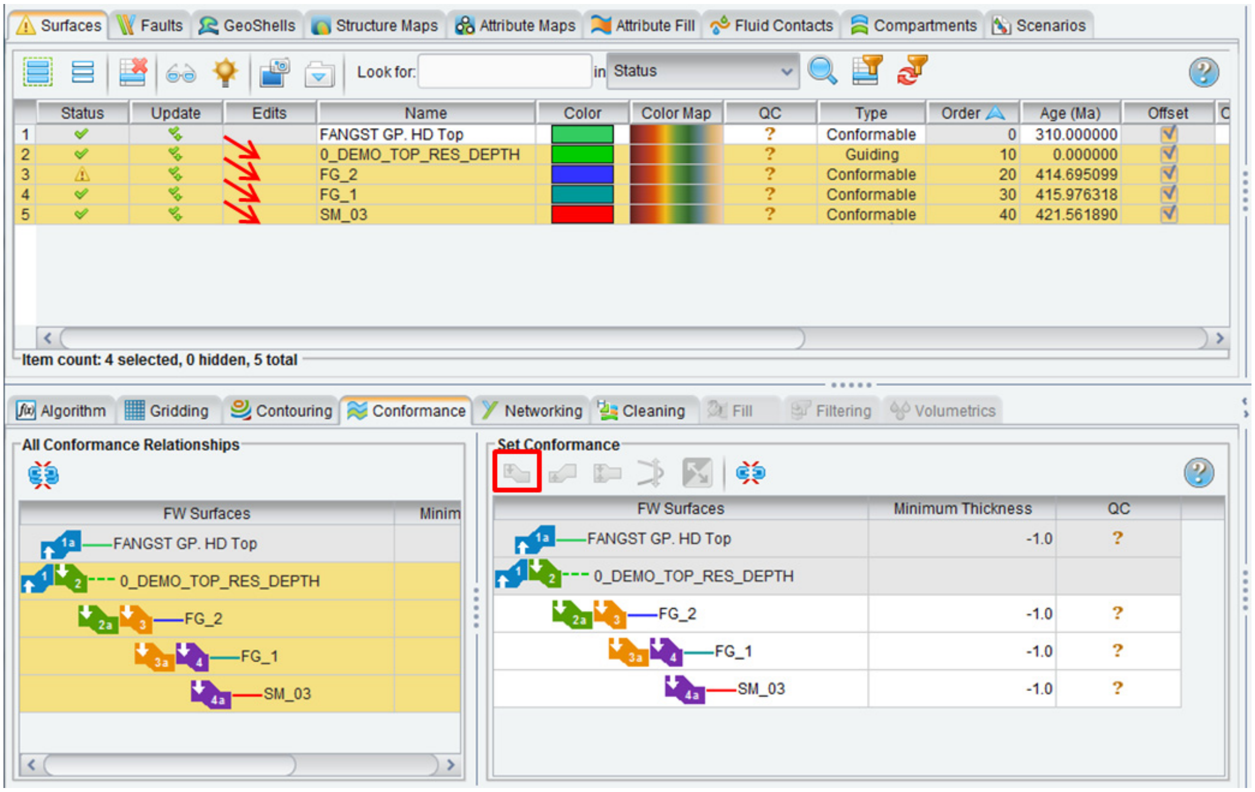
8. In the *Surfaces* object tab select the **O_DEMO_TOP_RES_DEPTH** framework surface. Change the Type to **Guiding**.



9. Select the **O_DEMO_TOP_RES_DEPTH** and the **FANGST GP.HD Top** surfaces. In the *Conformance* action tab, select the **Conform surfaces bottom up (****)** icon to apply a bottom-up conformance relationship.

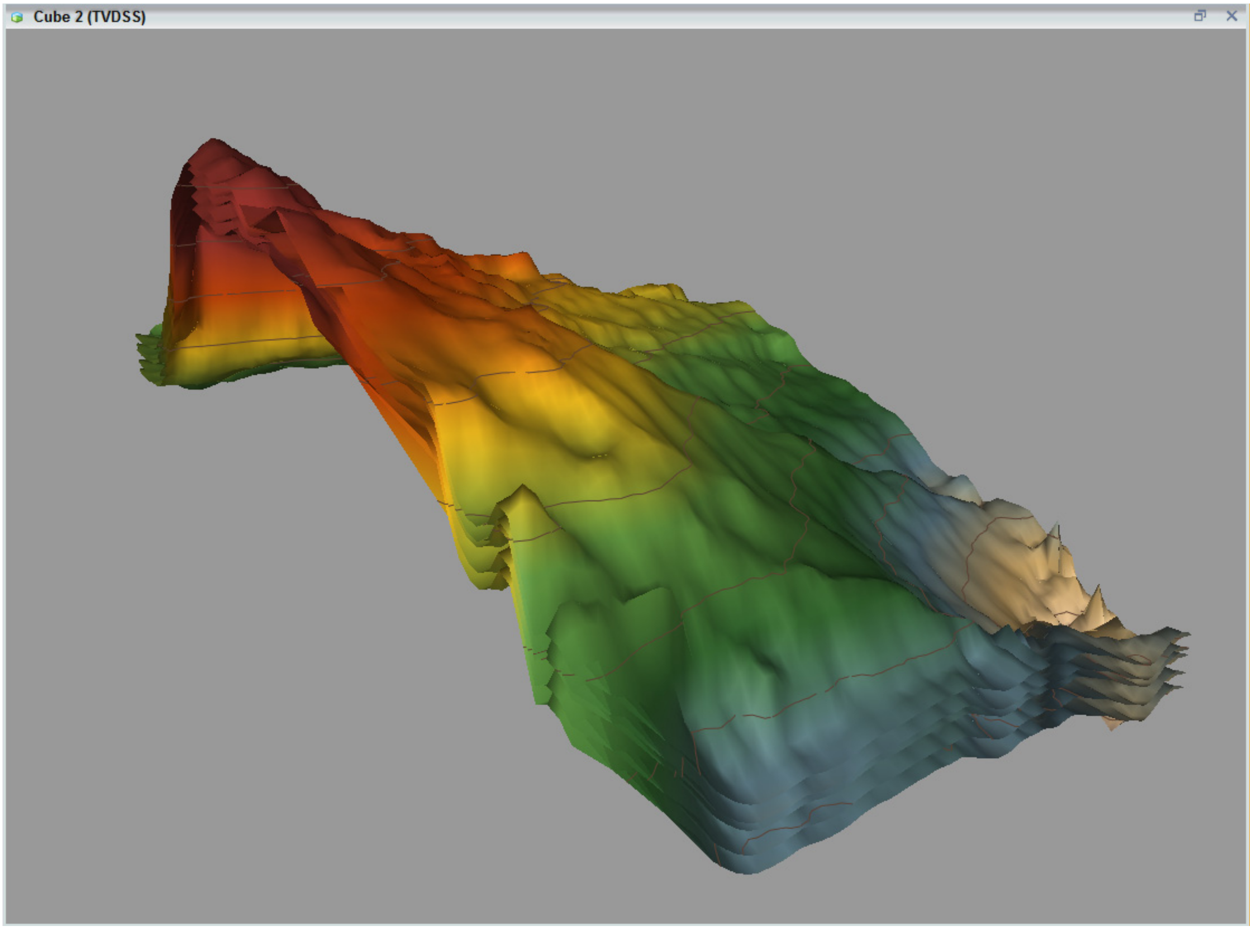


10. Back in the *Surfaces* object tab, select the **O_DEMO_TOP_RES_DEPTH, FG_2, FG_1,** and **SM_03** surfaces. In the *Conformance* action tab, select the **Conform surfaces top down (** **)** icon to apply a top-down conformance relationship.



> **Note:**
>
> If there is a warning message for the FG_2 surface it is due to a minor crossing with another surface. To change this, Minimum thickness can be changed to +1, which disallows the surface crossing.

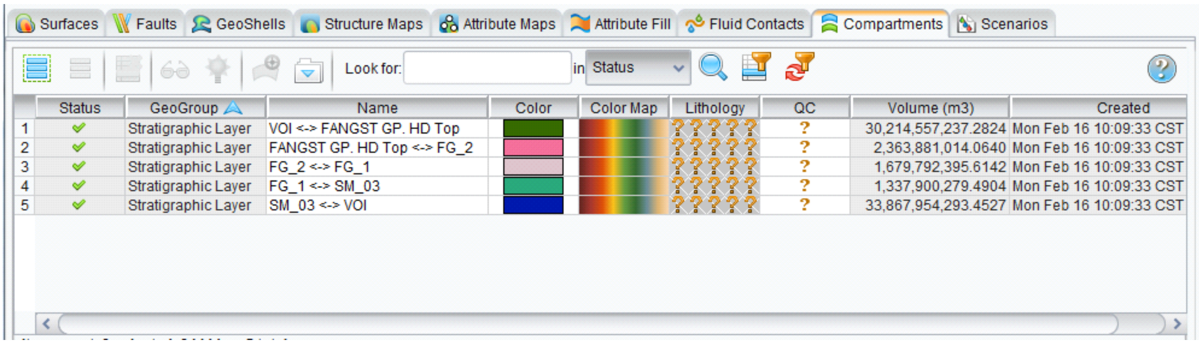
11. In the Z-scale toolbar of your *Cube* view, increase the Z factor to **"5"**, to better visualize your surfaces.

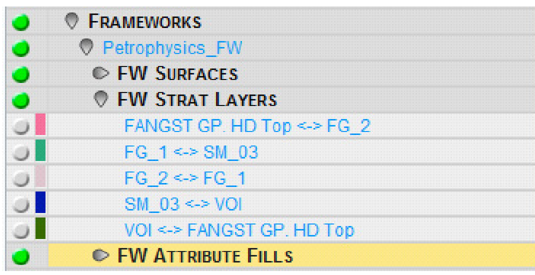


Now that all of the preliminary parameters are set in your framework, you will create compartments.

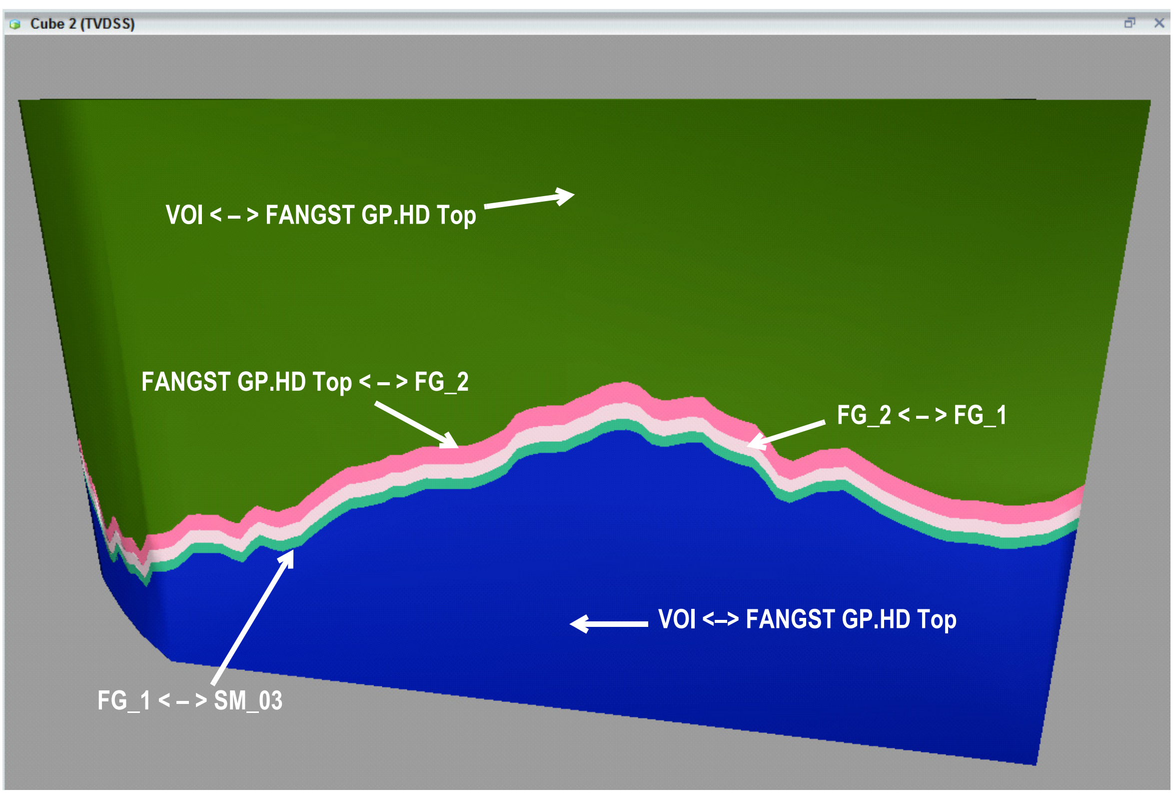
12. In the *Settings* panel at the top of the *Dynamic Frameworks to Fill Workspace*, click the **Compartments** () icon.

The newly created compartments are shown in the *Compartments* objects tab and in the *Inventory* task pane under **FW STRAT LAYERS**.



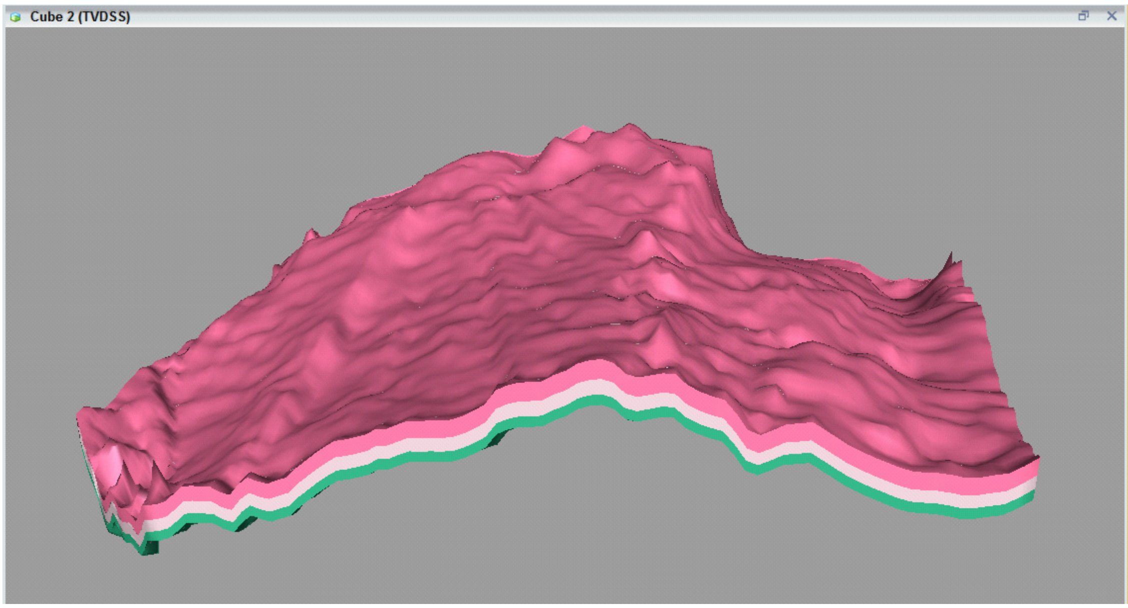


13. In your *Cube* view, toggle on and display only your newly created Framework Strat Layer **compartments**.



The above image shows that five compartments have been created: three within the reservoir (shown as pink, light pink, and light green), one from the top of the seismic volume to the top of the reservoir (dark green) and one from the base of the reservoir to the bottom of the seismic volume (dark blue).

14. In the *Inventory* task pane, toggle off the **VOI <-> FANGST GP. HD Top** and the **SM_03 <-> VOI** compartments, so you are visualizing only the compartments in the reservoir. The following image shows the three reservoir compartments (pink, light pink, and light green).

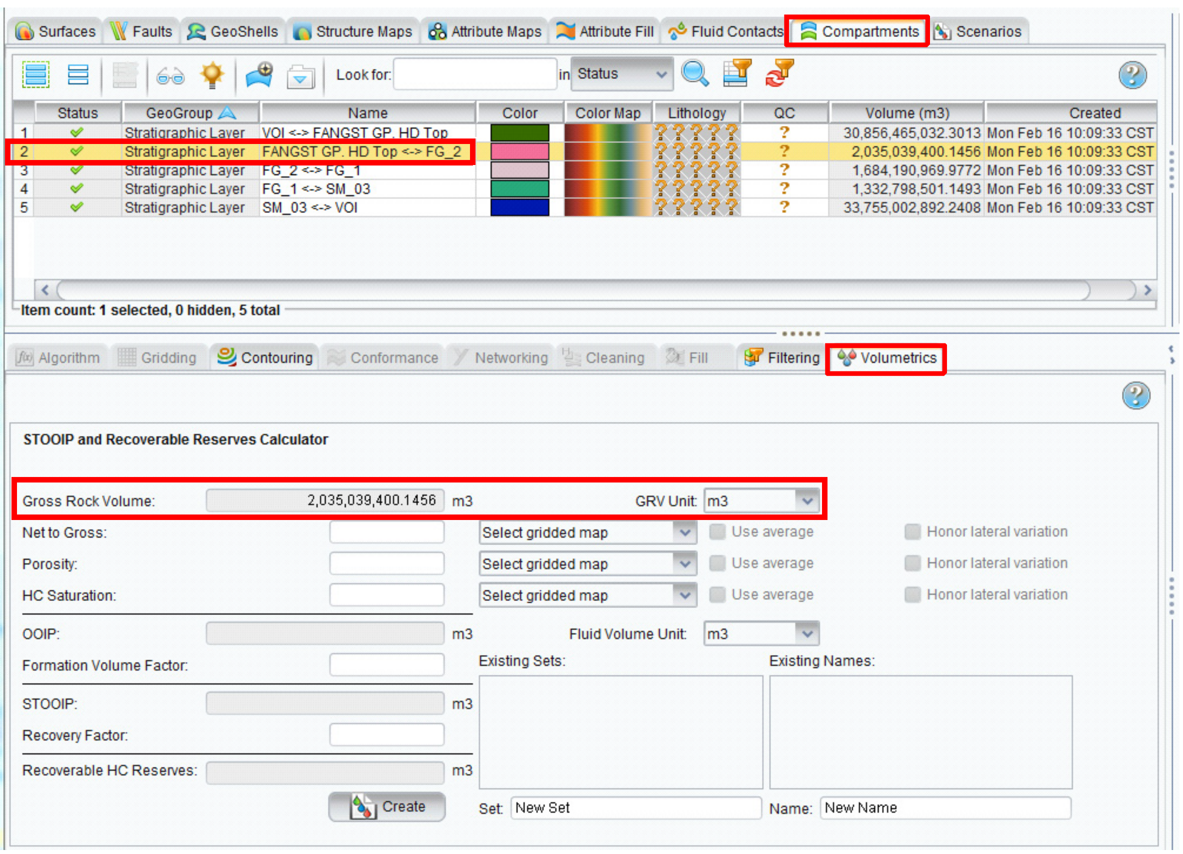


Now that you have finished creating Stratigraphic Layer compartments you will run volumetric calculations on the FANGST GP. HD Top <-> FG_2 compartment. This is from the same interval for which you created your attribute fill maps in Chapter 5.

Exercise 6.2: Volumetric Calculations

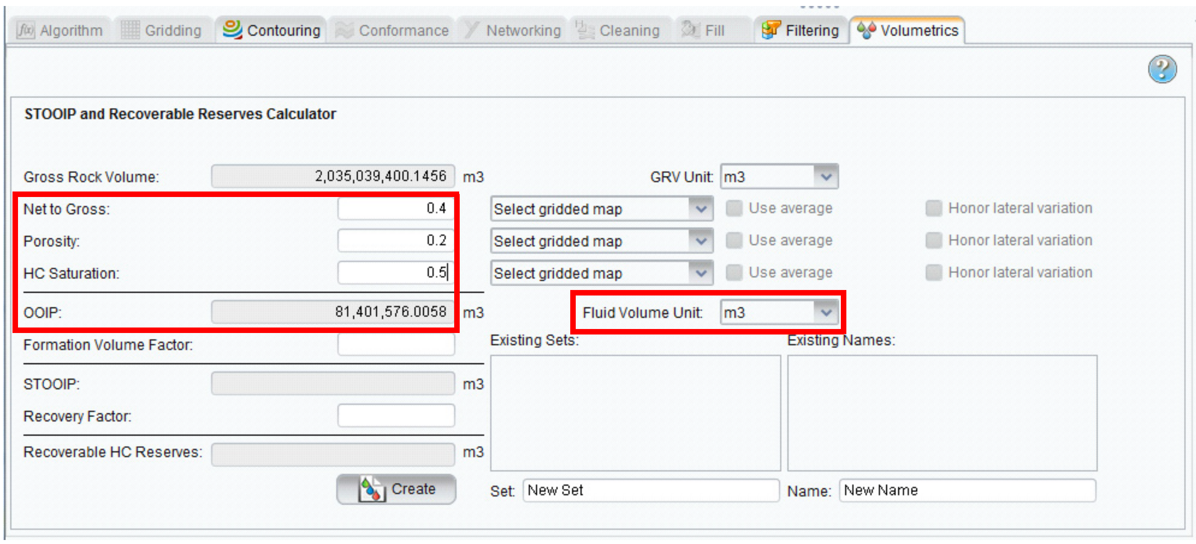
On the *Volumetrics* action tab of the *Dynamic Frameworks to Fill Workspace* you can calculate the Gross Rock Volume, OOIP, STOOIP, and Recoverable Hydrocarbon Reserves. You can run volumetric calculations on any of your compartments by using constant values, or from previously made property maps. In the following steps you will perform a basic volumetric calculation by using both constant values and grids.

1. In the *Compartments* tab of the *Dynamic Frameworks to Fill Workspace*, select the **FANGST GP. HD Top <-> FG_2** compartment, and then select the *Volumetrics* action tab.



Notice in the foregoing image that the Gross Rock Volume: for the selected compartment, FANGST GP.HD Top <-> FG_2, is automatically calculated in the Volumetrics tab. You can change the units in which your Gross Rock Volume: is calculated by selecting it from the GRV Unit: pull-down menu. (See above image.)

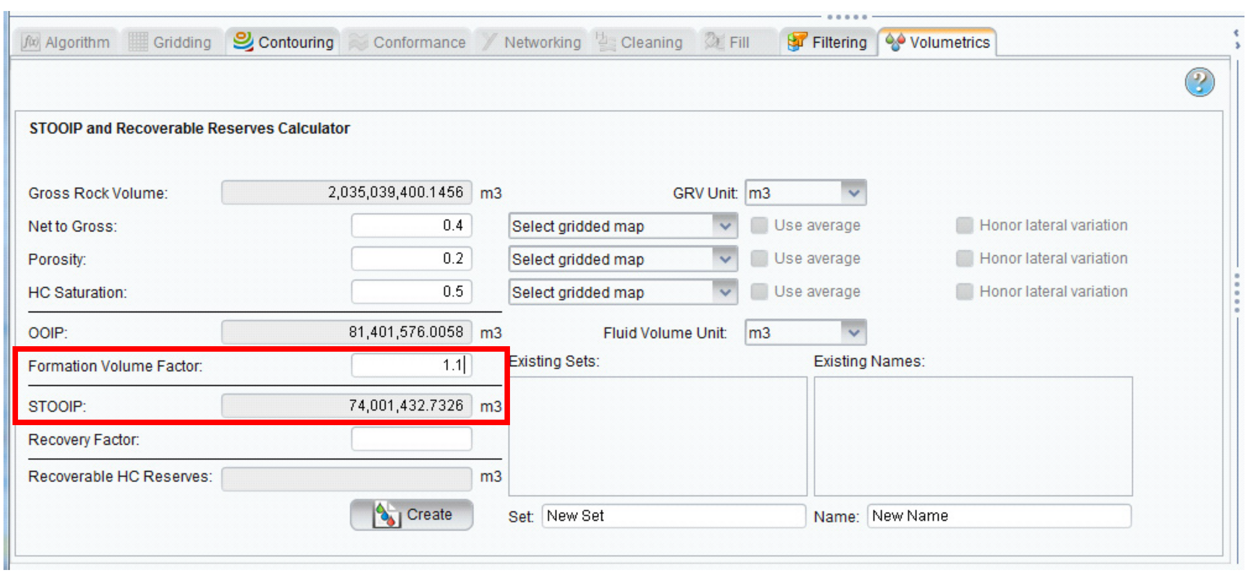
2. In the *Volumetrics* action tab, enter **"0.4"** in the Net to Gross: field. In the Porosity: field, enter **"0.2"**, and in the HC Saturation: field, enter **"0.5"**. After you enter these values the original oil in place (OOIP) value is calculated and displayed.



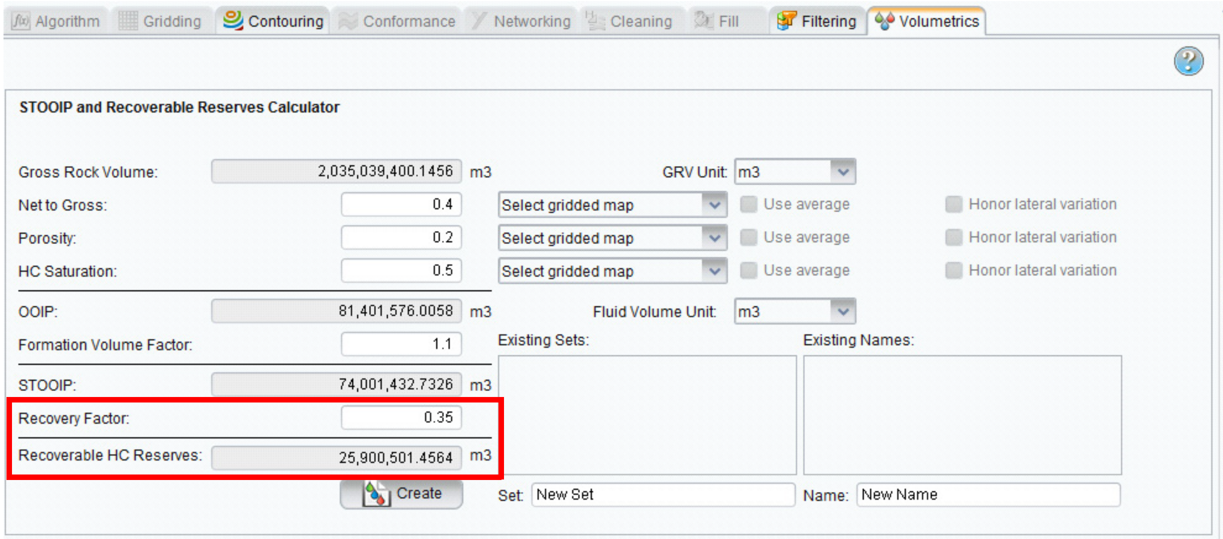
Note:

To change the units in which OOIP, STOOIP, and Recoverable HC Reserves are reported, select the appropriate unit on the Fluid Volume Unit: pull-down menu.

3. In the Formation Volume Factor: field enter **"1.1"**. The stock-tank original oil in place (STOOIP) value is calculated and displayed.

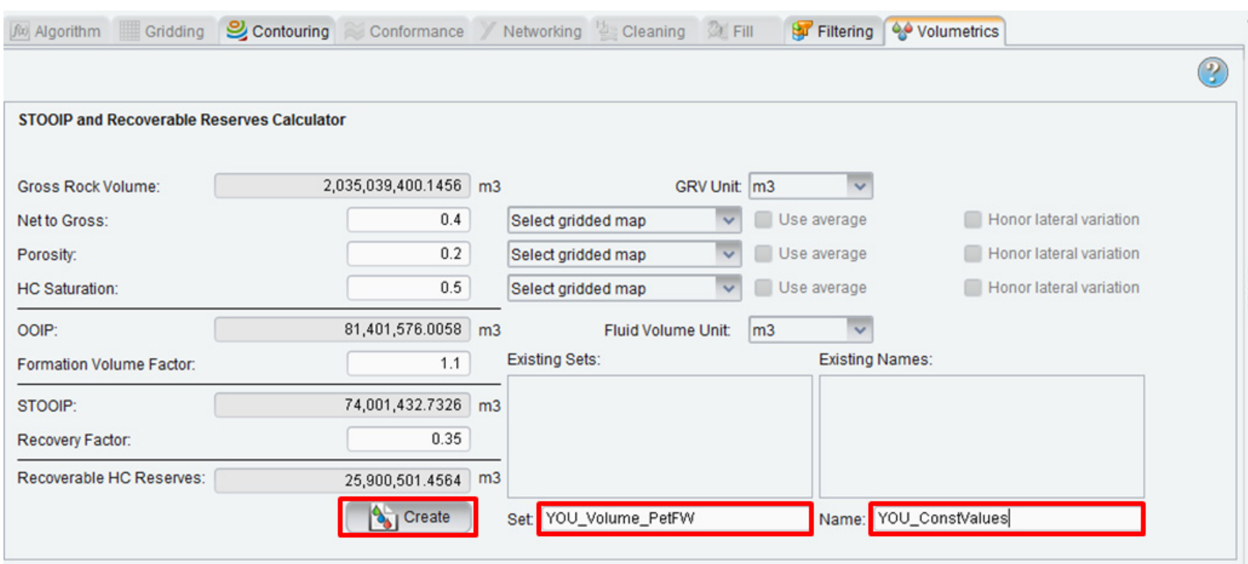


4. In the Recovery Factor field: enter **“0.35”**. The Recoverable HC Reserves will be calculated and displayed, using the information that you entered in the *Volumetrics* tab.

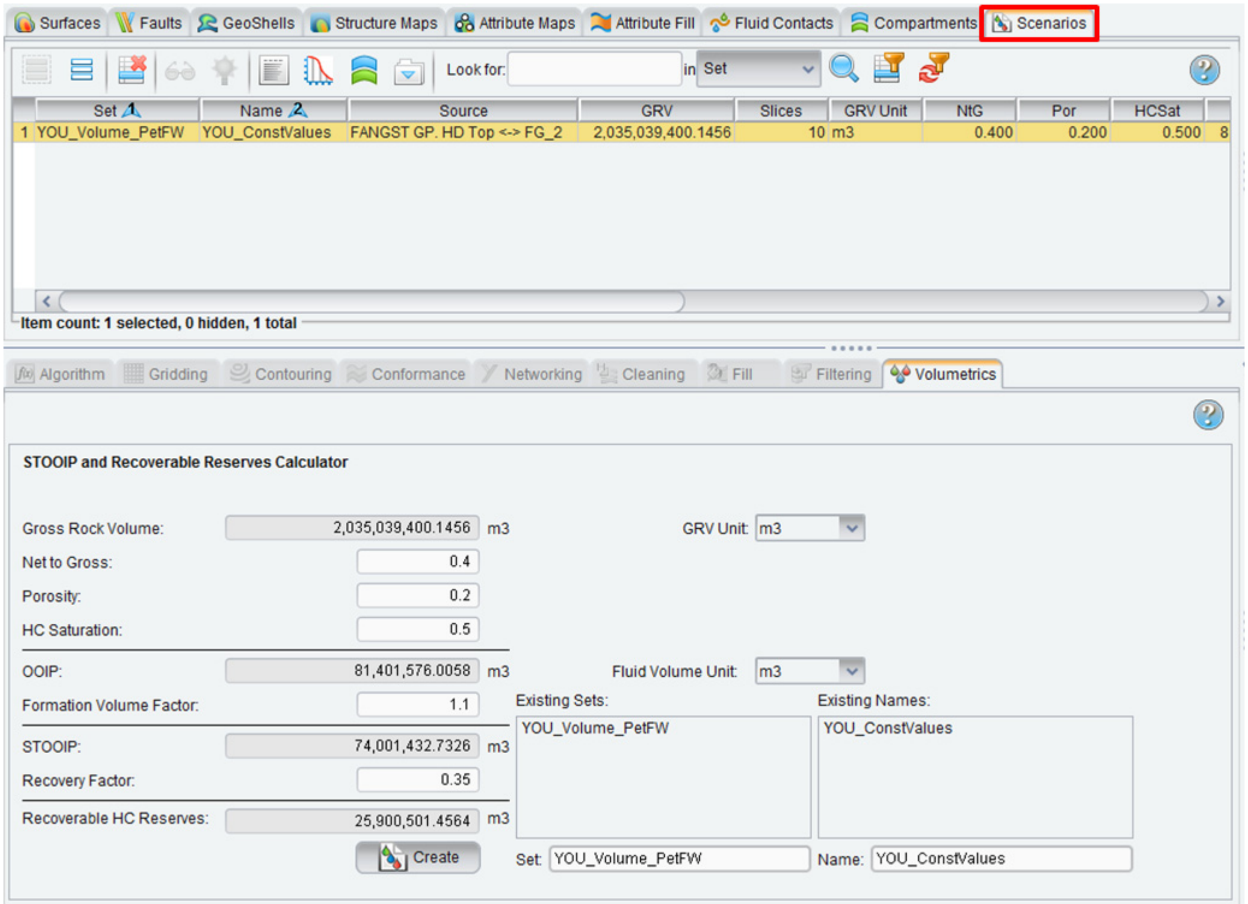


When creating your volumetric calculations you can give a name to the calculation, and also name the set in which the calculation belongs. For example, you may be doing multiple reservoir calculations but you may have values that you consider a best-case scenario as opposed to conservative values that are more likely. You can make both calculations, while associating them with one another by putting them in the same set.

5. Change the Set: field to **“YOU_Volume_PetFW”** and the Name: field to **“YOU_ConstValues”**, then click the **Create** button.



The new calculation will appear in the *Scenarios* object tab of your *Dynamic Frameworks to Fill Workspace*.



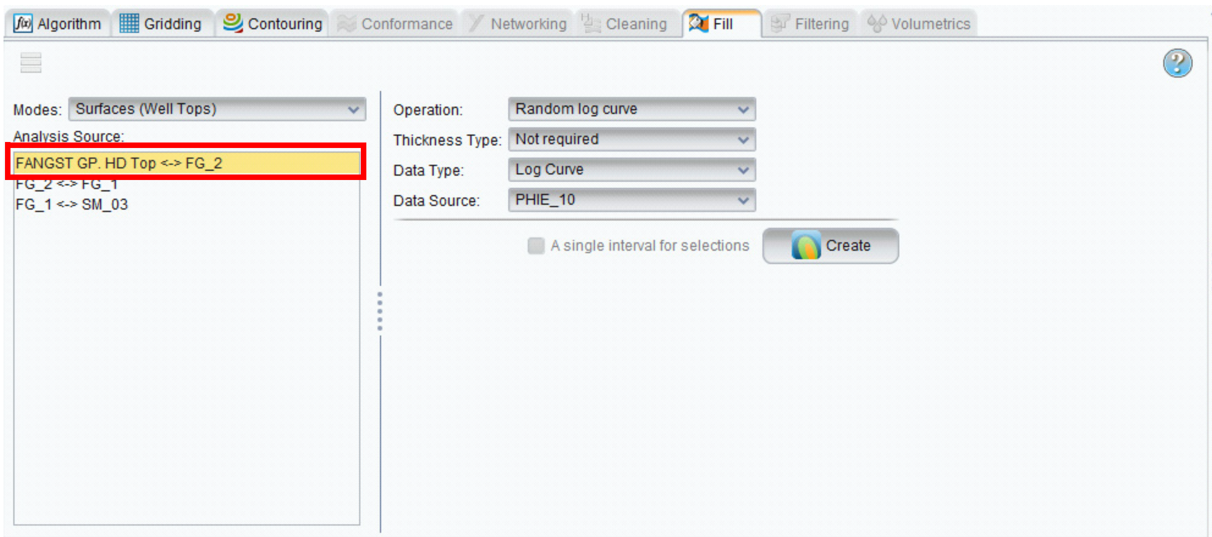
> **Note:**
>
> You can create new calculations from the *Scenarios* tab as well. When a previously selected volumetric calculation is selected the parameters used for that calculation are shown in the *Volumetrics* action tab (as shown in the above picture). You can change any of the parameters and save it with the same, or a different, name.

As noted earlier you can also make volumetric calculations using the maps you've created in your Attribute Fill. Using the maps in your calculations allows you to factor in the variability within your reservoir.

Before continuing with the volumetric calculations, you will create a map of your porosity to use in your volumetrics.

6. In the *Dynamic Frameworks to Fill Workspace,* select the *Attribute Fill* object tab, and then select the *Fill* actions tab (see following image).

7. On the *Analysis Source:* panel of the *Fill* tab, select the **FANGST GP.HD Top <-> FG_2** well top interval. On the Operation: pull-down menu, select **Random log curve**; on the Data Type: pull-down menu, select **Log Curve**; and on the Data Source: pull-down menu, select **PHIE_10.** Click the **Create** button to create the Attribute Fill.

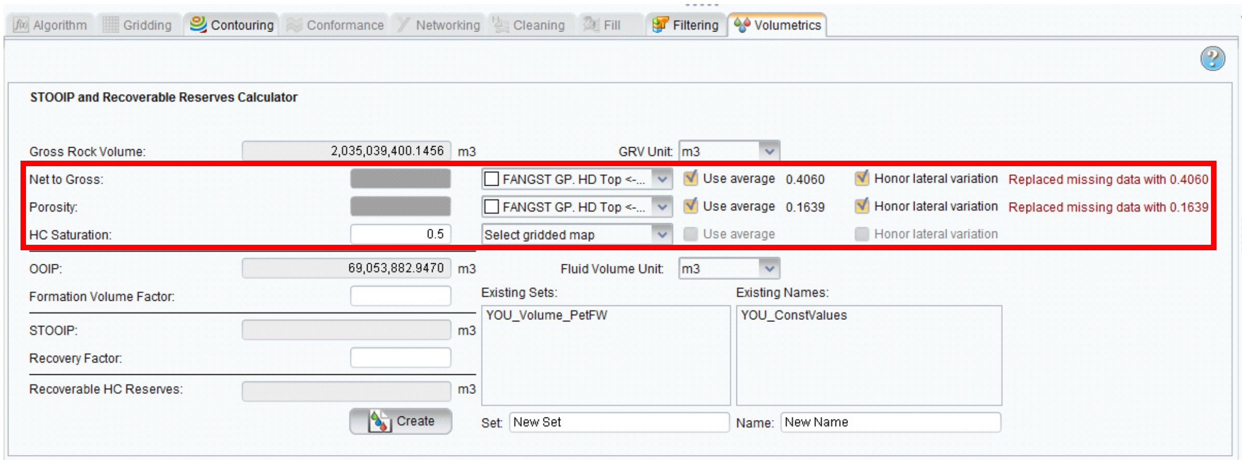


8. Select the **FANGST GP.HD Top <-> FG_2** compartment of the *Compartments* tab, if it is not already selected.

9. On the Net to Gross: pull-down menu, select the **FANGST GP.HD Top <-> FG_2 Net to gross ratio** map. On the Porosity: pull-down menu, select the **FANGST GP.HD Top <-> FG_2 PHIE_10** map; and since you do not have a grid for the HC Saturation:, enter a constant value of **"0.5"**.

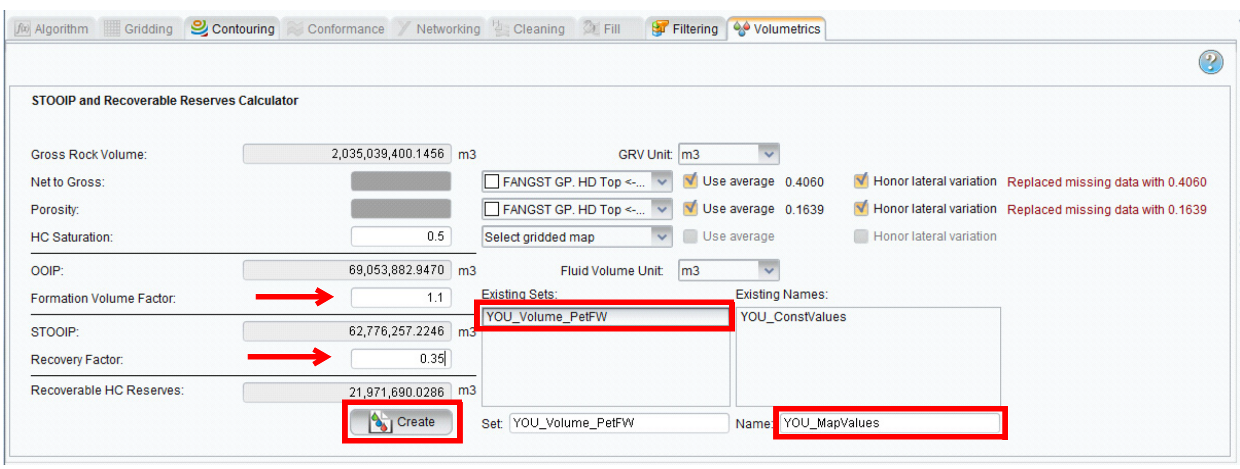
10. Toggle on the **Use average** and the **Honor lateral variation** checkboxes for both the *Net to Gross* and *Porosity.* This designates that the software will use that lateral variation within the maps for

the volumetric calculations, but in areas of null values the software will use the average value from the maps.

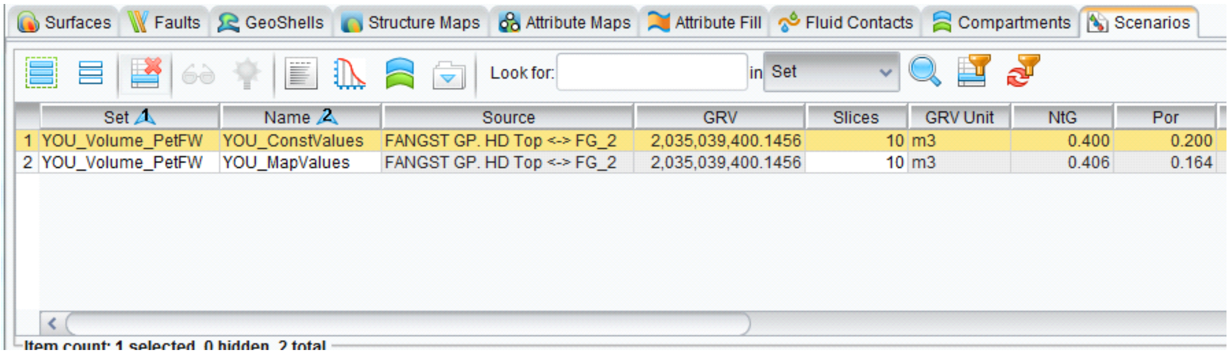


You will keep all of the other numbers the same, as in the previous calculations, so you can see the differences in using maps versus constant values for the parameters in your volumetric calculations. The values are shown in the following image.

11. In the *Existing Sets* sub-panel select **YOU_Volume_PetFW**. This will populate the *Set* field with the already existing set name. In the Name: field enter **"YOU_MapValues"**. Click **Create.**

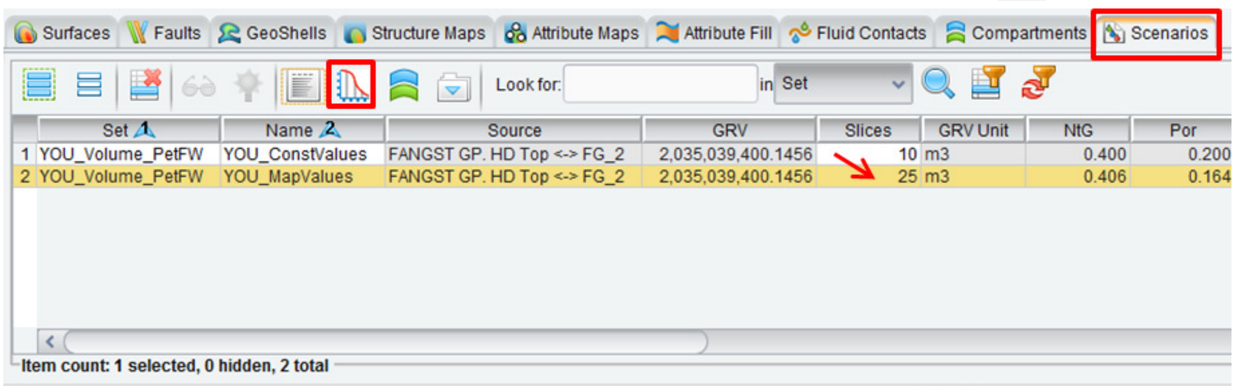


The newly created volumetric calculation will show up in the *Scenarios* object tab. Notice that both calculations are under the same Set, but have unique names.



You can view the results of your calculations in a Scenarios Chart and a Scenarios Report. Both show depth-area pairs and other analyses of the volume, designated by the number of slices that have been set in the *Scenarios* tab. Setting the number of slices for a calculation does not re-run the calculations using the slice method.

12. In the *Scenarios* object tab of the *Dynamic Frameworks to Fill Workspace*, select **YOU_MapValues.** In the *Slices* column enter **"25"** and click the **Volumetric Scenario Chart ( )** icon.

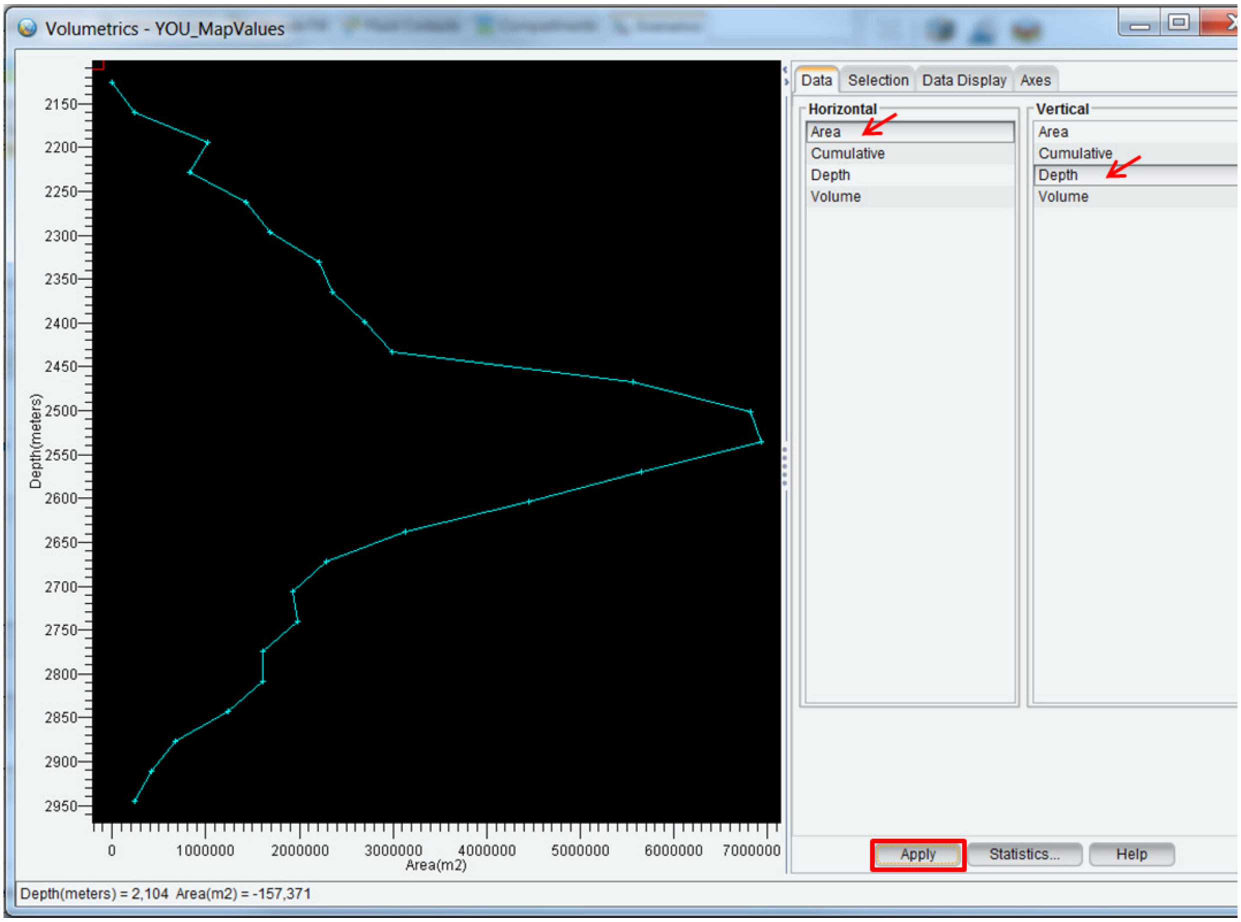


The default plot is Depth (m) versus Cumulative Volume (m^3). The compartment has been divided into 25 equal slices, each slice denoted by a small plus sign. The number of meters per increment is determined by the total height of the compartment, divided by the number of slices entered in the *Scenarios* object tab.

On the right side of the Volumetrics dialog the *Data* tab is selected. On the *Data* tab are two panels, *Horizontal* and *Vertical*. On those panels you can interactively change the horizontal and vertical axes.

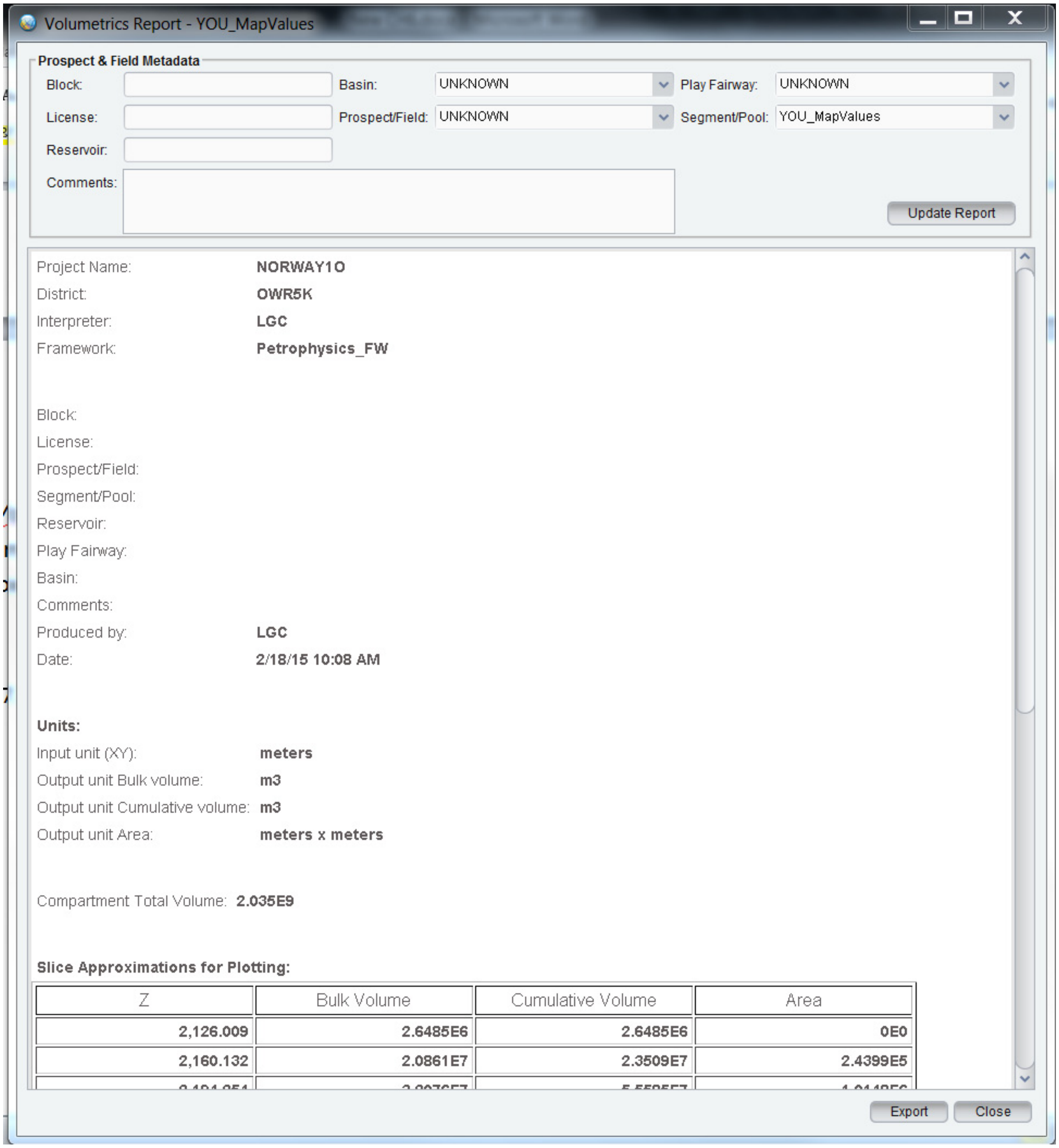
If you hover your cursor over the graph, a red square on the curve displays, which shows the exact point of read-out. The depth and volume at the selected point are displayed in the lower left corner.

In the *Horizontal* panel of the *Volumetrics* dialog, select **Area**. In the *Vertical* panel, select **Depth**. Click **Apply**. Run your cursor along the resulting graph to view the values at a node, and then close the dialog.



13. For a different view of your data, while the **YOU_MapValues** scenario is still selected, click the **Volumetric Scenario Report** () icon.

The *Volumetrics Report* display, showing text fields for entering prospect and field metadata, as well as all compartment calculations and parameters. In addition, the report displays the slice approximations for plotting charts.



> **Note:**
>
> You can export the report to a common delimited *.csv file.

14. **Close** the Report and the Chart.

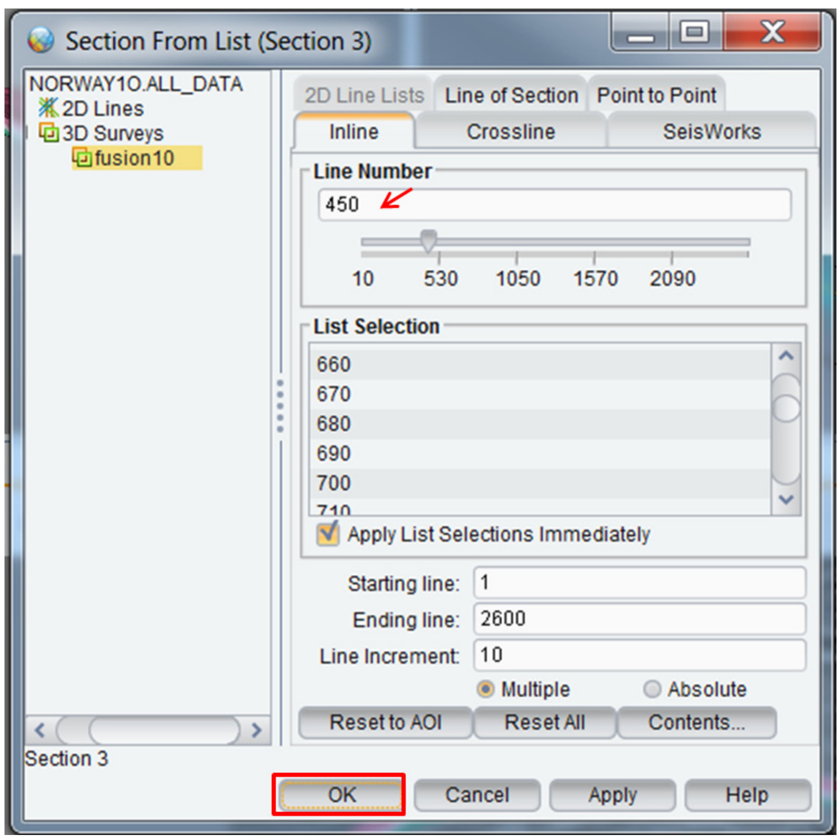
Exercise 6.3: Compartment Display Properties

It is likely that you will want to highlight reservoirs that are in your views. For example, after running volumetric calculations you may have determined which compartments you want to target. Being able to focus on those is important. You may also want to highlight certain characteristics, such as lithology, of the compartments in your framework.

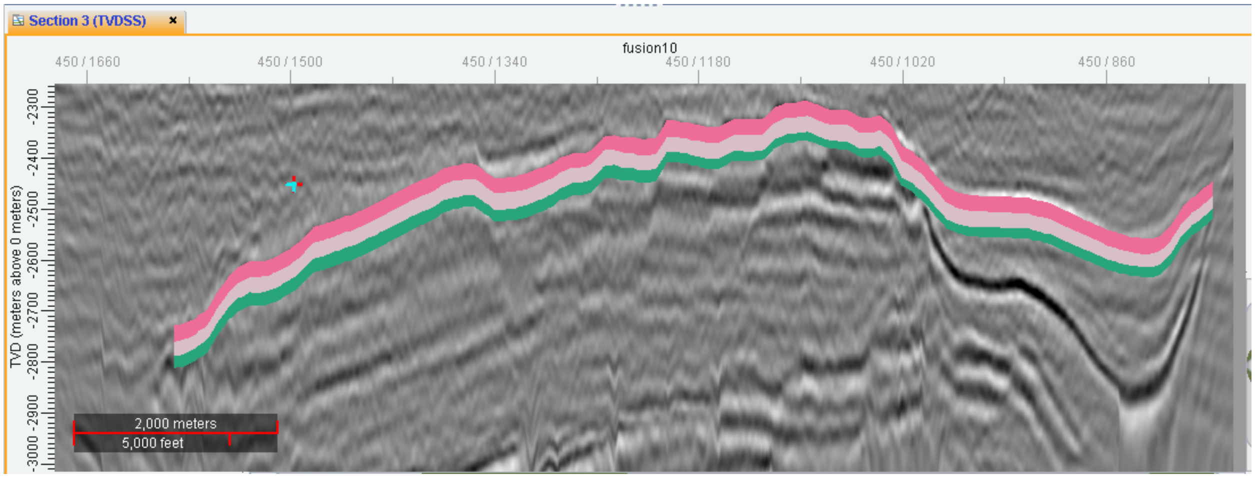
In this exercise you will change the display properties of your compartments in *Map* and *Section* views, to highlight properties of your compartments.

1. With *Section* view active, click the **Section from List (** **)** icon on the side tool bar.

2. In the *Section From List* dialog, select **IL 450,** and then click **OK**.

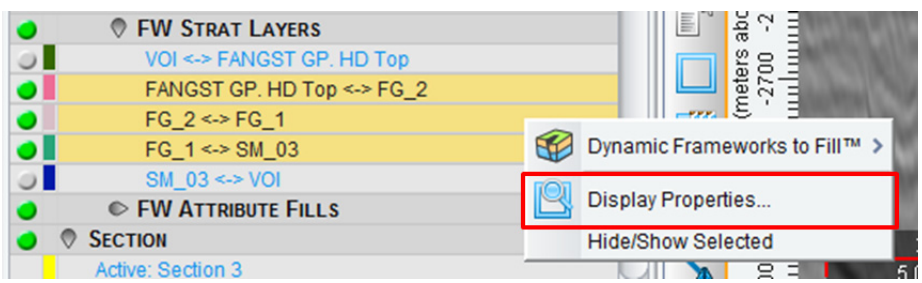


3. With your *Section* view still active: in the *Inventory* task pane toggle on **Depth_full_offset.cmp** on the three FW STRAT LAYERS within your reservoir: **FANGST GP.HD Top <-> FG_2, FG_2 <-> FG_1**, and **FG_1 <-> SM_03.** As with the *Cube* view, change the Z factor to **"5"**. This will enhance the relief in the compartments.

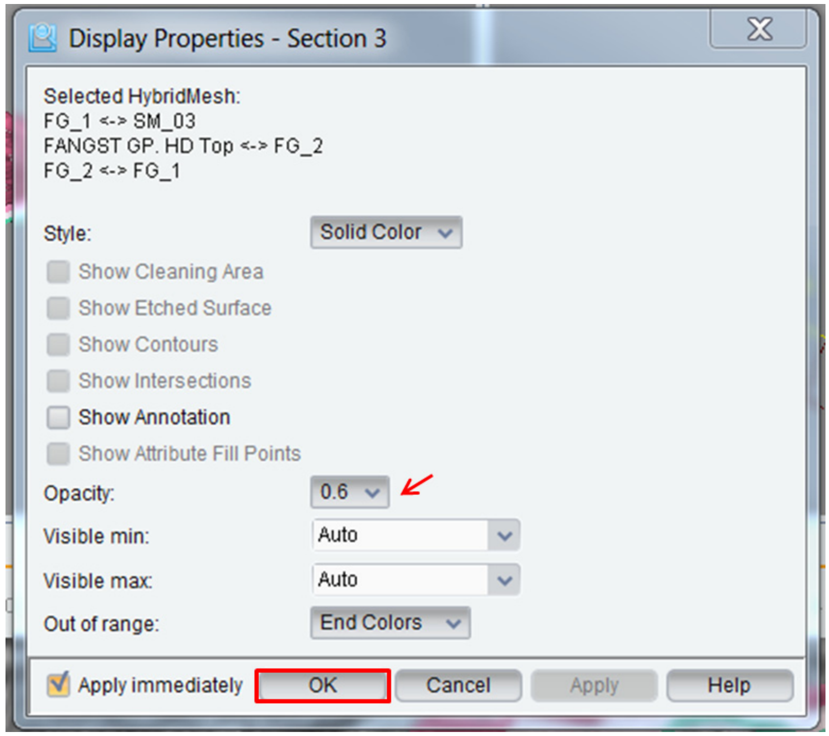


Currently the opacity of the compartments is set to 1. To see how the compartments compare to the seismic, you will need to decrease the opacity.

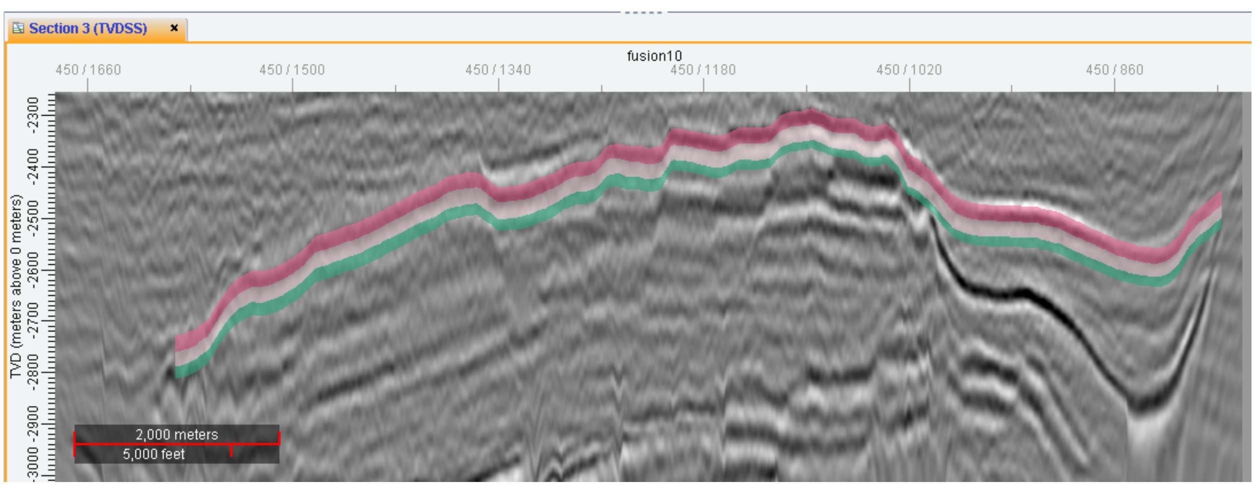
4. In the *Inventory* task pane, select the three displayed compartments, and with your cursor on the highlighted compartments, **MB3 > Display Properties.**



5. In the *Display Properties* dialog change the Opacity to **0.6** and click **OK.**

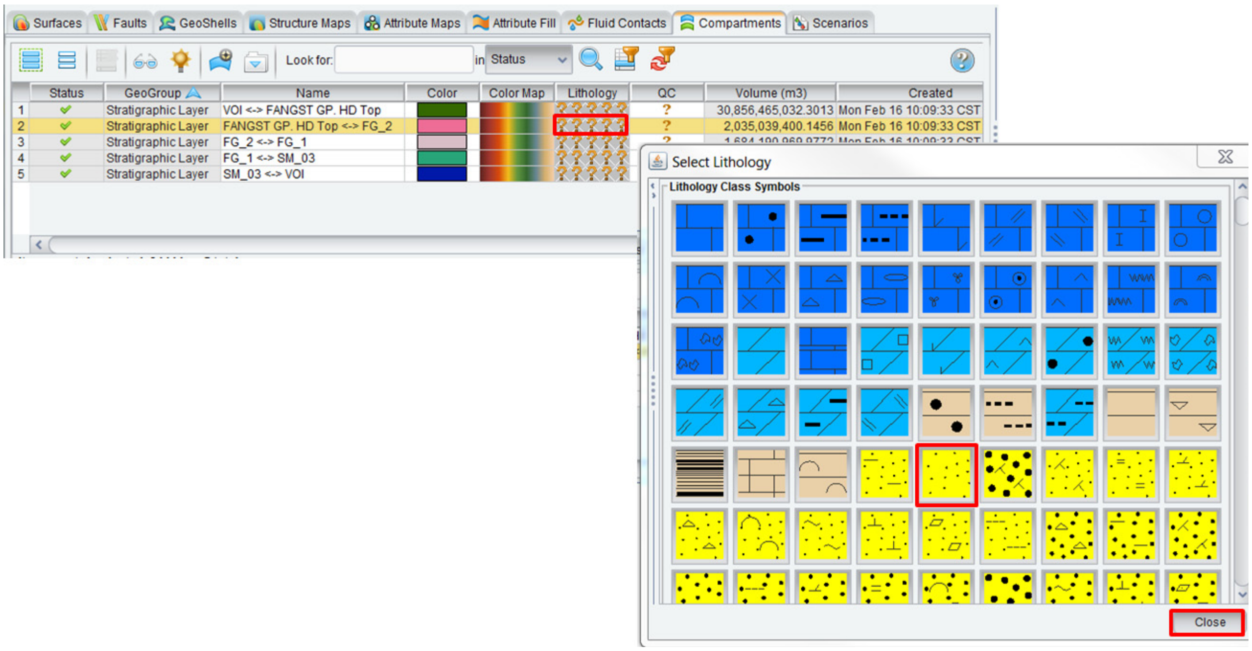


The seismic is now visible behind the compartments; your view should look similar to the one below.



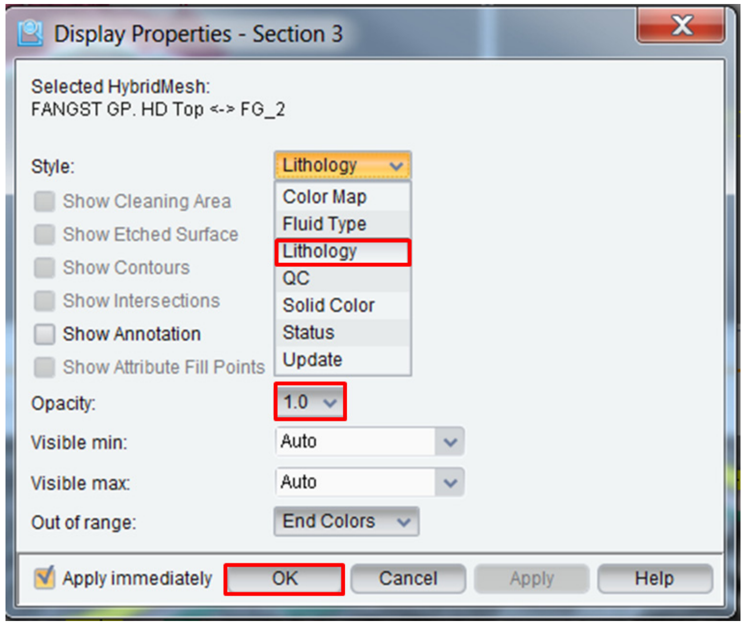
When displaying your compartments you may want to highlight where your lithologies are located. You have many options for changing the display color of your compartments, including the ability to display by lithology in both *Map* and *Section* view.

6. In the *Compartments* objects tab of the *Dynamic Frameworks to Fill Workspace* there are three options (columns) for color: Solid Color, Color Map, and Lithology. Select the **FANGST GP.HD Top <-> FG_2** compartment and click the **Lithology** cell. On the *Select Lithology* dialog, select **Fine Sand**. Click **Close.**

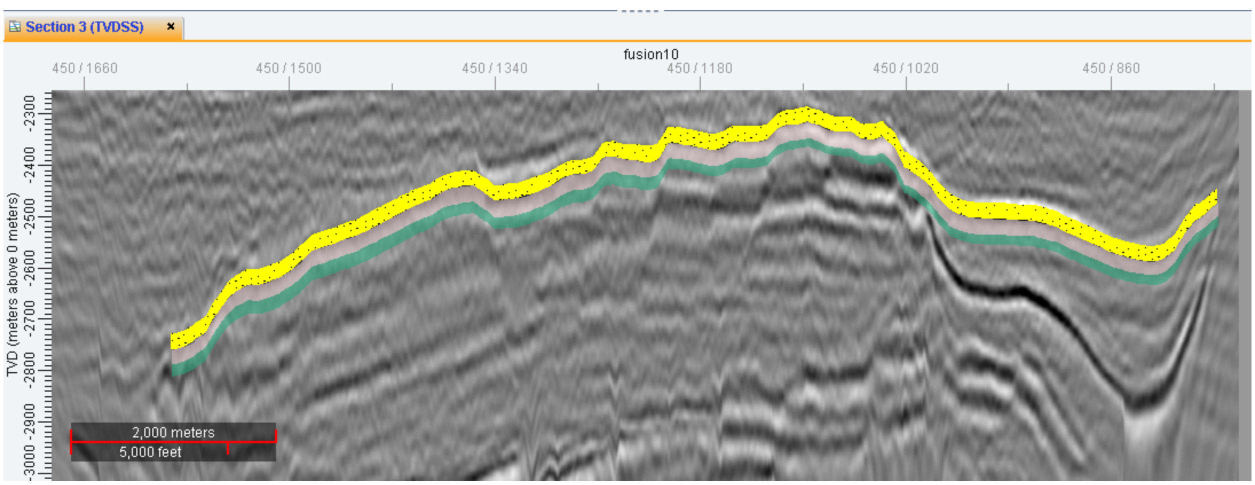


7. With *Section* view active: in the *Inventory* task pane put your cursor on the **FANGST GP.HD Top <-> FG_2** and select **MB3 > Display Properties.**

8. In the Style: pull-down menu of the *Display Properties* dialog notice the options you have, and then select **Lithology.** Then change the Opacity: to 1, and click **OK.**



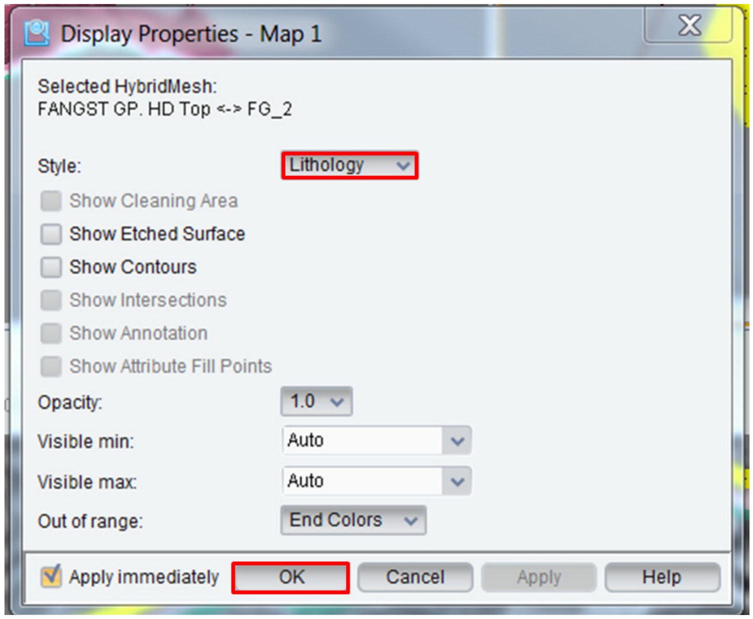
Your view should look similar to the image below.



You can also display by lithology in *Map* view.

9. Activate *Map* view and toggle on the **FANGST GP.HD Top <-> FG_2** compartment. Toggle off all other objects.

10. Put your cursor on the compartment in *Map* view and **MB3 > Display Properties.** In the Style: pull-down menu of the *Display Properties* dialog select **Lithology** and click **OK**.

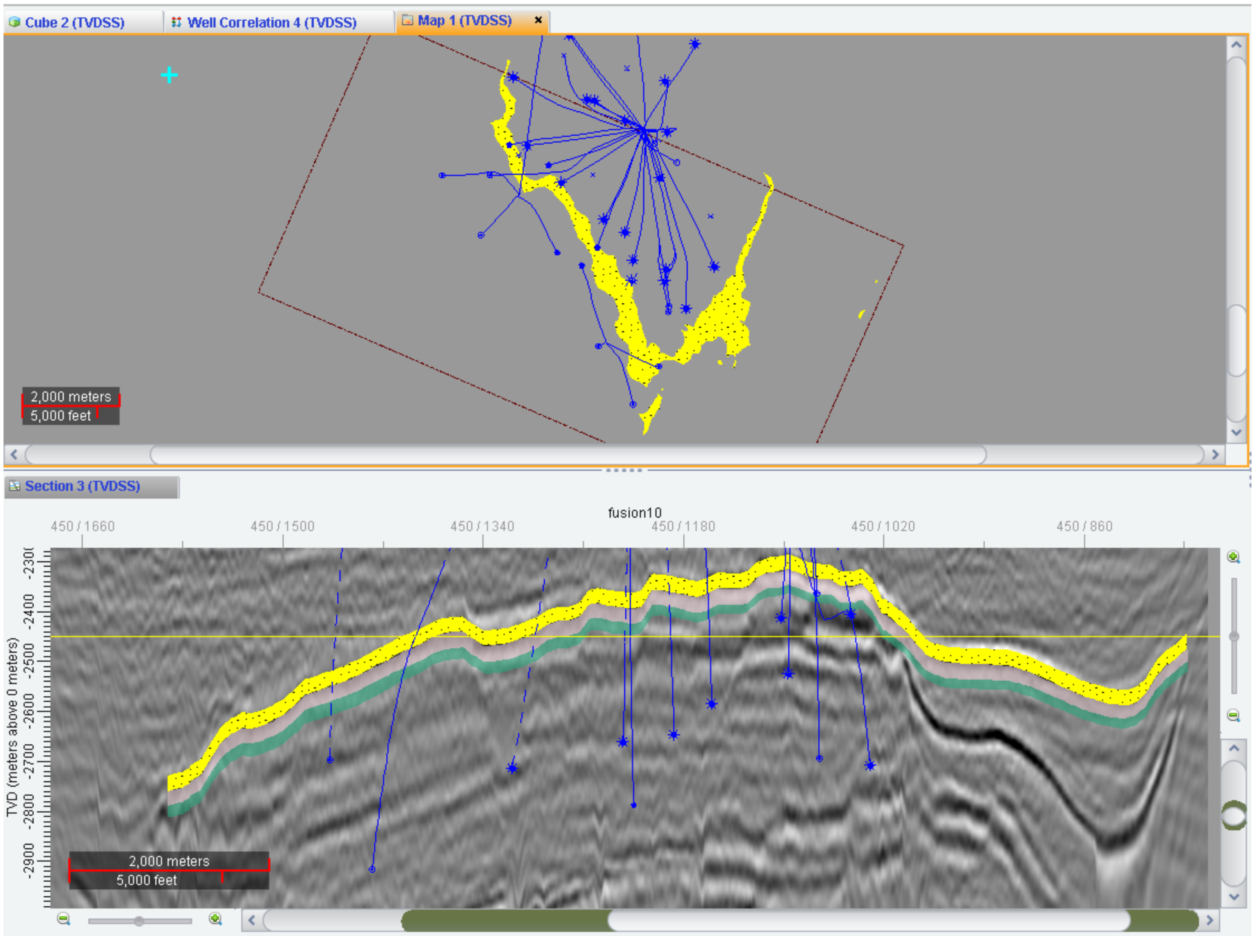


The entire compartment is displayed. Often you want to see the extent of your compartment at a specific depth. You can do this by entering into *Map Intersection View*.

11. With the *Map* view active, select the **Map Intersection View** () icon. In the current domain value field, enter **“-2450”**.

Now only the portions of the FANGST GP.HD Top <-> FG_2 compartment that are at a depth of -2450 are visible. This capability is helpful when you are trying to decide where to drill.

12. To compare how your wells are interacting with your compartments, toggle on the well list **demo wells**.

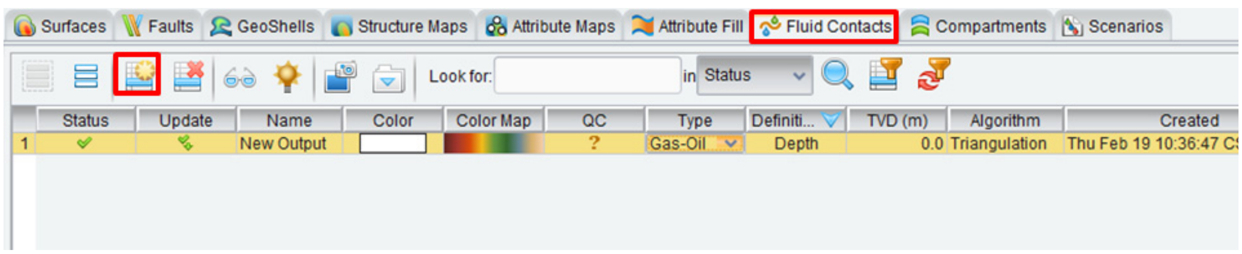


Notice the slice displayed in *Map* view also shows as a reference line in *Section* view.

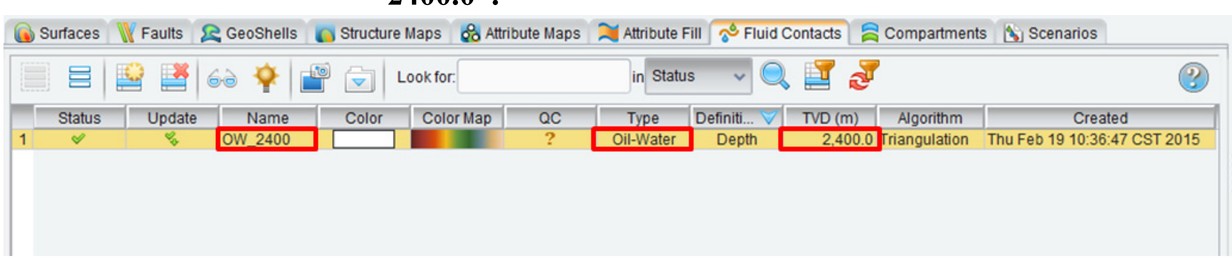
Exercise 6.4: Advanced Compartment Construction

Up until now you have focused on the stratigraphic layer compartments within frameworks. In the following exercise you will learn how to create fluid compartments, and you will also learn how to merge multiple compartments to create a custom reservoir. Being able to combine these compartments is important, especially in structurally complex areas.

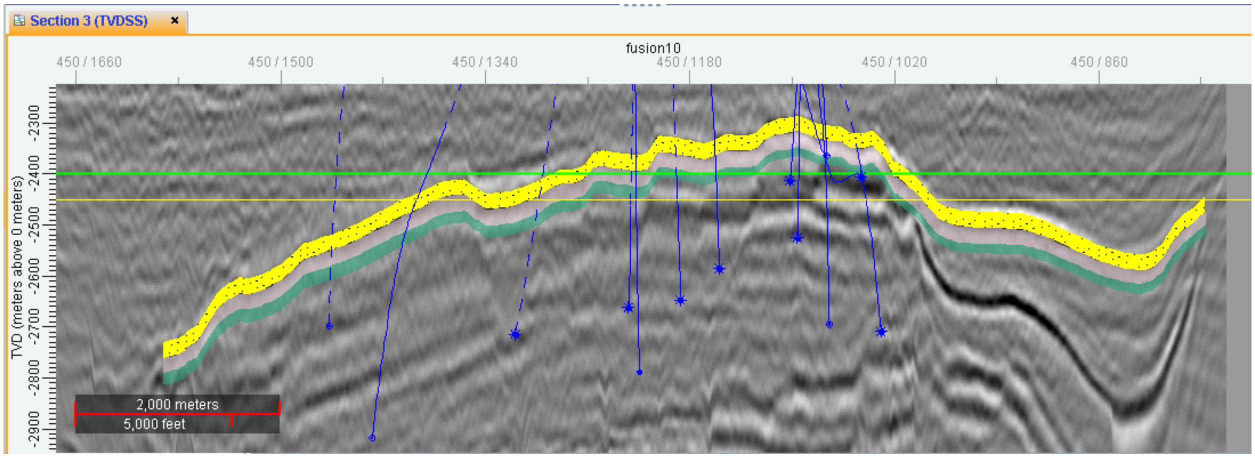
1. In the *Dynamic Frameworks to Fill Workspace*, select the *Fluid Contacts* objects tab, then select the **Add a new model entry** ( ) icon. The new fluid contact will appear in the *Fluid Contacts* objects tab.



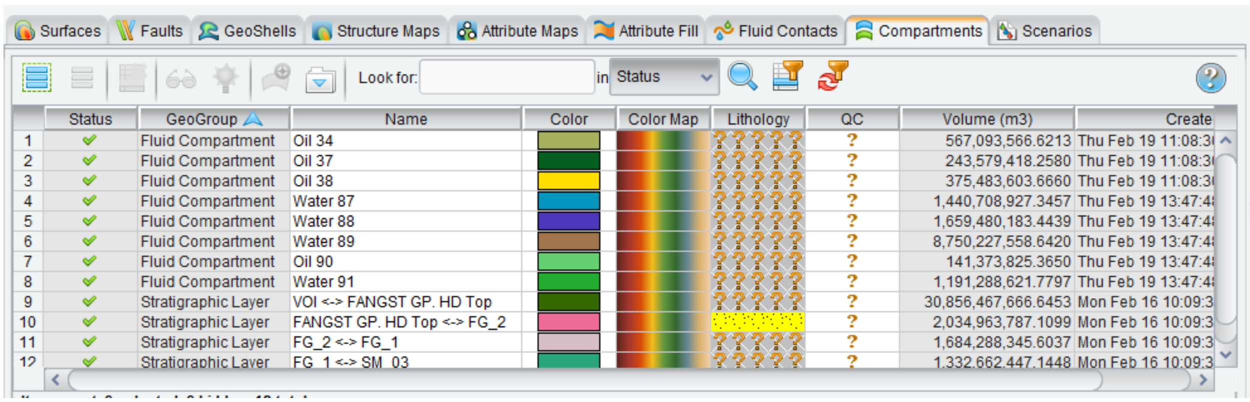
2. You can make all of the changes to the fluid contact in the *Fluid Contacts* objects tab. Change the Name cell to **"OW_2400"**, then change the Type cell to **Oil-Water,** and the TVD (m) cell to **"2400.0".**



3. Activate *Section* view and toggle on your newly created **OW_2400**. Your *Section* view should look similar to the following image.

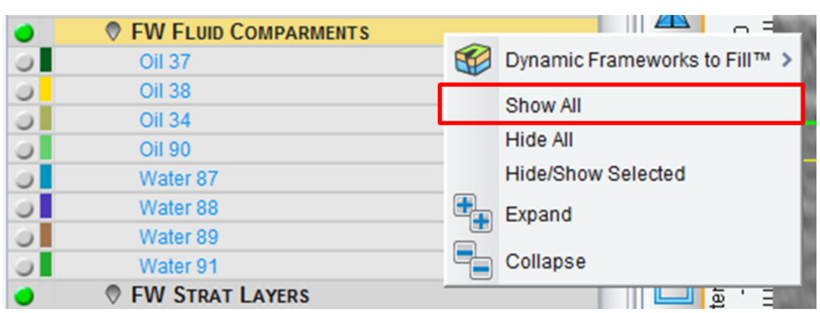


Creating fluid contacts will automatically create fluid compartments. The compartments appear in the *Compartments* object tab of the *Dynamic Frameworks to Fill Workspace*, as well as in the *Inventory* task pane under a newly created category, FW FLUID COMPARTMENTS.

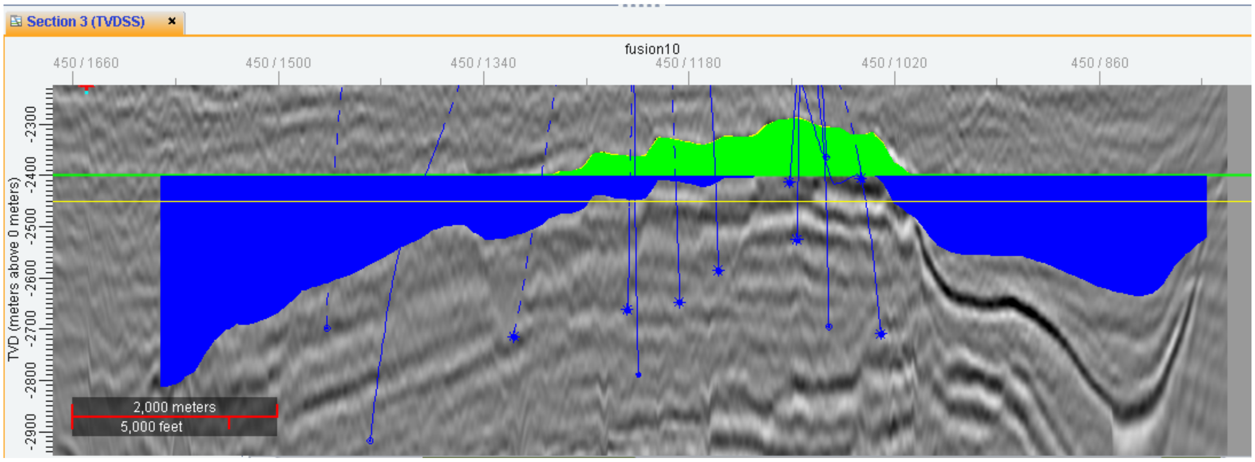


	Status	GeoGroup	Name	Color	Color Map	Lithology	QC	Volume (m3)	Create
1	✓	Fluid Compartment	Oil 34				?	567,093,566.6213	Thu Feb 19 11:08:3
2	✓	Fluid Compartment	Oil 37				?	243,579,418.2580	Thu Feb 19 11:08:3
3	✓	Fluid Compartment	Oil 38				?	375,483,603.6660	Thu Feb 19 11:08:3
4	✓	Fluid Compartment	Water 87				?	1,440,708,927.3457	Thu Feb 19 13:47:4
5	✓	Fluid Compartment	Water 88				?	1,659,480,183.4439	Thu Feb 19 13:47:4
6	✓	Fluid Compartment	Water 89				?	8,750,227,558.6420	Thu Feb 19 13:47:4
7	✓	Fluid Compartment	Oil 90				?	141,373,825.3650	Thu Feb 19 13:47:4
8	✓	Fluid Compartment	Water 91				?	1,191,288,621.7797	Thu Feb 19 13:47:4
9	✓	Stratigraphic Layer	VOI <-> FANGST GP. HD Top				?	30,856,467,666.6453	Mon Feb 16 10:09:3
10	✓	Stratigraphic Layer	FANGST GP. HD Top <-> FG_2				?	2,034,963,787.1099	Mon Feb 16 10:09:3
11	✓	Stratigraphic Layer	FG_2 <-> FG_1				?	1,684,288,345.6037	Mon Feb 16 10:09:3
12	✓	Stratigraphic Layer	FG_1 <-> SM_03				?	1,332,662,447.1448	Mon Feb 16 10:09:3

4. In the *Inventory* of your *Section* view, put your cursor over **FW FLUID COMPARTMENTS** and **MB3 > Show All**. Toggle off all of the other **compartments**.

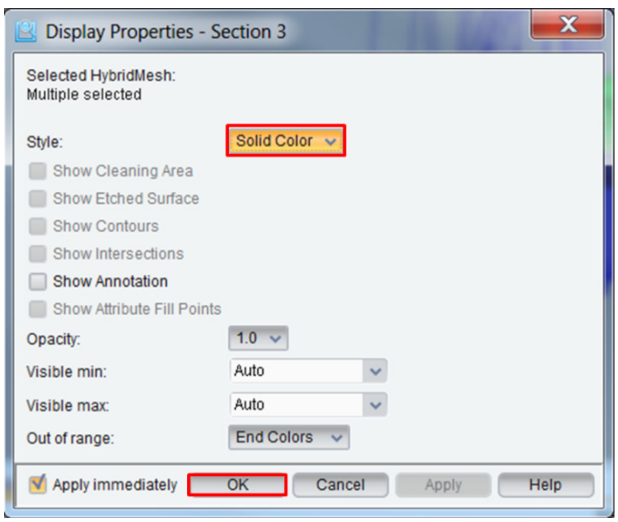


Your compartments should look similar to the ones in the following image.



Currently the compartments are displayed by fluid type. To see all of the compartments you will need to change the Style: to Solid Color.

5. With the *Section* view active, select all of the **FW FLUID COMPARTMENTS** and **MB3 > Display Properties.** In the Style: pull-down menu of the *Display Properties* dialog select **Solid Color**. Click **OK**.



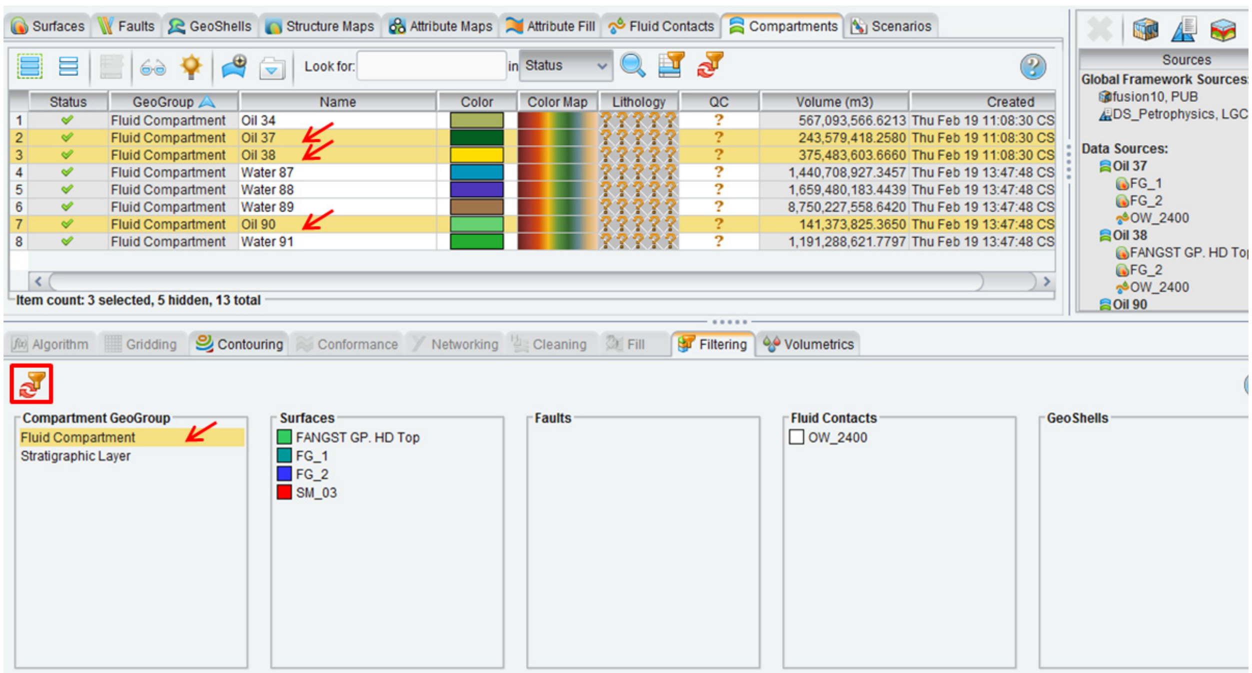
In this exercise eight fluid compartments are created: four oil and four water. You may want to calculate the volume of all of the oil in your reservoir; you can do this by creating custom reservoirs. Within Frameworks you can combine multiple fluid compartments to create a custom reservoir.

In the following steps you will filter your compartments to show only your fluid compartments, and then you will combine them to create a custom reservoir that contains all of the oil in your reservoir.

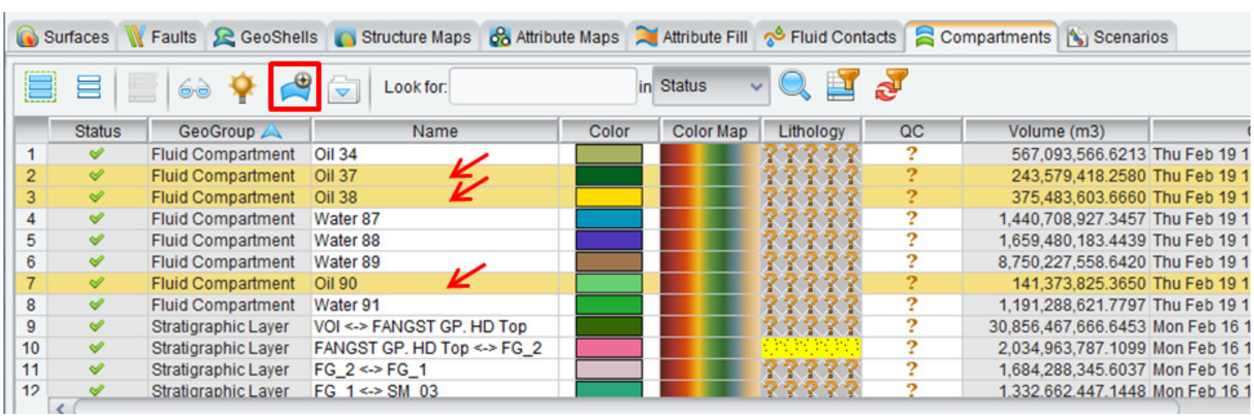
6. In the *Dynamic Frameworks to Fill Workspace*, select the *Compartments* objects tab and then the *Filtering* action tab. Select **Fluid Compartment** in the *Compartment GeoGroup* panel. This will show only the compartments that are classified as Fluid Compartments.

7. Highlight all of the **Oil** fluid compartments in the reservoir, excluding the compartment that is created from the bottom of the reservoir to the top of the fluid contact.

8. Before merging the compartments, select the **Reset all compartment filters** ( ) icon, so you can see all FW Compartments.

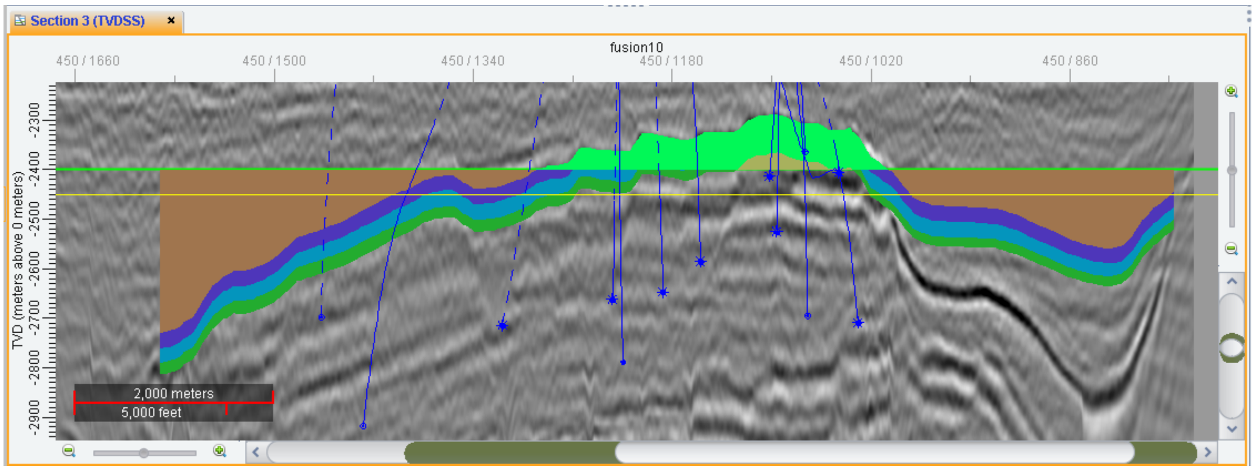


9. Click the **Merge all selected to custom reservoir** ( ) icon.



> **Note:**
>
> You can change the names of any of your compartments in the workspace to better organize your list. To rename the newly created custom reservoir, double-click in the Name: field of the *Compartments* objects tab.

10. Display your new custom **reservoir** in *Section* view.



11. Select **File > Exit** to close your session.

You have learned functionality that varies from interpretation of well picks in *Correlation* view to how to build a complete structural framework and how to perform volumetric calculations within those frameworks. In the next chapter you will learn how to bring your framework into the Earth Modeling module to generate a 3D Geocellular Grid.

Chapter 7

Final Framework to Earth Modeling

The DecisionSpace Geosciences software provides an elegant approach to integrated prospect evaluation and development. The goal of DecisionSpace Geosciences is to help the interpreters quickly and accurately map hydrocarbon-related anomalies. DecisionSpace accomplishes this by integrating a wide range of data types to a narrow range of probable solutions, constrained by geology, geophysics, petrophysics, and reservoir models. Integrating the geological and geophysical interpretation allows you to accurately map and analyze prospects, build sealed structural frameworks and reservoir models, and plan wells in traditional and unconventional scenarios.

The Topics Covered in this Chapter

In this chapter you will learn about:

- The benefits of Framework integration
- Converting log curves to point sets
- Analyzing property data
- Creating an Earth Modeling Grid
- Mapping facies
- Creating Lithotype Proportion Maps

Overview: Integrating Velocity Model with Framework

You can build a DecisionSpace velocity model using a structural framework.

The Benefits of Integration

The framework can contain faults and surfaces, both of which are gridded and truncated at intersections to provide a sealed framework.

The framework model can contain free-floating bodies (GeoShells), such as salt or sand lenses. These GeoShells can be complex bodies, which are created in GeoProbe (from Horizons) or Dynamic Frameworks to Fill (from a Framework Surface). In addition you may also input a set of intersecting top and bottom surfaces as salt bodies in Velocity Modeling.

Moreover, having a velocity model in your framework sources allows you to view the model in either the depth or time domain. Creating an Earth Modeling grid requires a Framework in the depth domain, because the source data used in generating a 3D geocellular model works in depth domain only.

Creating Earth Modeling Grids

The DecisionSpace Earth Modeling software requires a grid before any interpolation methods can be used. This section describes the grids used in this module. When there is a reference to a grid, it refers to a 2D or 3D Earth Modeling grid, as opposed to the Framework grids, OpenWorks surface grids, and SeisWorks 3D horizons.

You can use the DecisionSpace Earth Modeling software to create 2D grids and 3D grids.

About 2D Grids

The 2D grid is called a GridSet. When you save a GridSet, an OpenWorks surface grid with the attribute Z is also created. GridSets are saved to the OpenWorks database.

Each time you add a property, a new surface grid is created. Therefore, the resulting GridSet is a collection of surface grids. In the *Select Session Data* dialog the name of the surface grids created for properties appears as GridSet_Property, the Attribute will be the selected property, and the GeoName will be the name of the GridSet.

About 3D Grids

The output 3D grid is a .vdb, which is a geocellular grid using the VDB storage format. VDB is a Landmark database that is mainly used by reservoir simulators and related software.

The VDB format is a directory tree structure comprising multiple sub-directories and files that contain data of different categories. A .vdb grid can be imported directly into the VIP® or the Nexus® software for reservoir simulation.

You can export .vdb through the Tools > Export 3D Grid command on the *DecisionSpace* menu bar.

GridSets and 3D Grids are listed in the *Select Session Data* dialog. You can access that dialog through the File > Select Session Data command on the main menu bar or the Select Session Data () icon on the toolbar.

Creating Grids

Ideally, the grid is designed in such a way that there are at least four or five grid nodes between each of the measured data locations. For DecisionSpace Earth Modeling, the grid node exists at the gravity center of each cell in 2D and at the gravity center of the voxel in 3D. The grid rotation is based on the geological definition of azimuth, where zero degrees is North. The grid azimuth is defined as 0 degrees plus or minus the rotation value. The absolute value of the grid rotation may not be greater than ± 90 degrees. When the grid rotation angle is changed, the lower right corner of the grid serves as the axis of rotation, with the lower X-axis boundary as the reference edge of the grid.

You can create GridSets from a point set using *Create 2D Grid* in the Property Modeling part. This creates a rectangular 2D grid (these are not .vdb grids). User-defined parameters define the geometry of the basic 2D grid.

In the DecisionSpace Earth Modeling software, 3D grids can be created in three ways:

- Creating a 3D grid
- Using grid geometry
- Loading a grid created outside of DecisionSpace

Creating 3D Grids by Using Create 3D Grid

You can create a rectangular, non-layered, .vdb grid by using the Create 3D Grid command in General Property Modeling part.

User-defined parameters define the geometry of the basic, empty .vdb grid. No layering is involved.

Creating 3D Grids by Using Grid Geometry

You can use Grid Geometry in the Stratigraphic Modeling part. This method uses three sets of inputs.

The first method includes inputting a pre-existing Framework (sealed structural model created using Dynamic Frameworks to Fill) or using the ezModel™ functionality in the GeoProbe® software. A Framework can exist in dynamic or static state. These states will affect the Earth Model.

A dynamic Framework consists of references to input data and an instruction set; when it is loaded, it is rebuilt from the input data and instruction set each time.

Using Dynamic Frameworks to Fill, you can save a dynamic framework as a static framework, which includes all components of the framework frozen in time (no updates are recognized) and saved in their complete state in the database. This ensures that the framework is initialized in Earth Modeling the same way every time. For example, the same number of layers and layer thickness that initially appear in the Stratigraphic Layering table for each interval after you click Initialize will be the same.

The second input includes layering parameters to be used within the stratigraphic grid.

The third input includes user-defined parameters such as cell size and rotation of the grid within the model.

Loading a grid created outside of DecisionSpace

You can load a grid created outside of DecisionSpace. You can load the following model/file/format types stored externally on disk.

- RESCUE (.bin)
- Landmark VDB (.vdb) created using the VIP software or the Nexus software
- ECLIPSE file formats (.GRDECL, .GRID, .EGRID, .FGRID, and .FEGRID)
- Roxar® RMS™ 3D grid format (.roff)
- Paradigm™ Gocad® 3D grid format (.sg)
- PETREL™ 3D grid model (.GRDECL)

You should be aware of potential workflow restrictions when you load external 3D geocellular data. You can review Layering Model Definition in the Help files to understand these restrictions.

Note:

The .grdecl files need to be in an ASCII format (not binary format). They also cannot be touched by a text editor or they will not be usable in DecisionSpace.

When loaded into a session, the 3D grid is saved as an OpenWorks flat file.

Warning:

Do not use this method to load a grid created using the DecisionSpace Earth Modeling software. Those grids are loaded using the *Select Session Data* dialog. Otherwise, you overwrite the grid.

Note:

ECLIPSE grids, Petrel (.GRDECL), Roxar® RMS™ 3D grids (.roff), and Paradigm™ GOCAD® 3D grids are read-only grids. Therefore, no property modeling or other operations done on the grid are saved back to the original file. To use these grids in the DecisionSpace Earth Modeling software and save the results, you must use the menu bar command, Tools > Export 3D Grid, to save the grid as a VDB grid. You can export as a VDB grid before making any modifications, load the new VDB, using Tools > Load 3D Grid, and then work with that new grid, or you can make your modifications to the loaded grid first and then export as a new VDB grid.

Warning:

The software reads a single .GRDECL file, which must contain both the 3D cell geometry (structural model/layers) and the cell properties (such as porosity and permeability). If properties were exported in separate files from ECLIPSE, they should be manually concatenated into a single file with the grid geometry file so the single .GRDECL file contains both the cell XYZ geometry and the cell property values.

Looking at Your Grid

Grids are the main data carriers for DecisionSpace in Earth Modeling. Therefore, you will want to view what is happening inside the grid.

Grid probes Used to Visualize 3D Grids

A box probe is automatically created in *Cube* view when a 3D grid is created using General Property Modeling or Stratigraphic Modeling.

If a grid was loaded using the *Select Session Data* dialog or the Tools > Load 3D Grid command with a *Cube* or *Map* view active, a box probe is created. If a *Section* view is active, a section probe is created.

For information on changing the display properties of a grid probe, see “Display Properties — Probes” in the Help file.

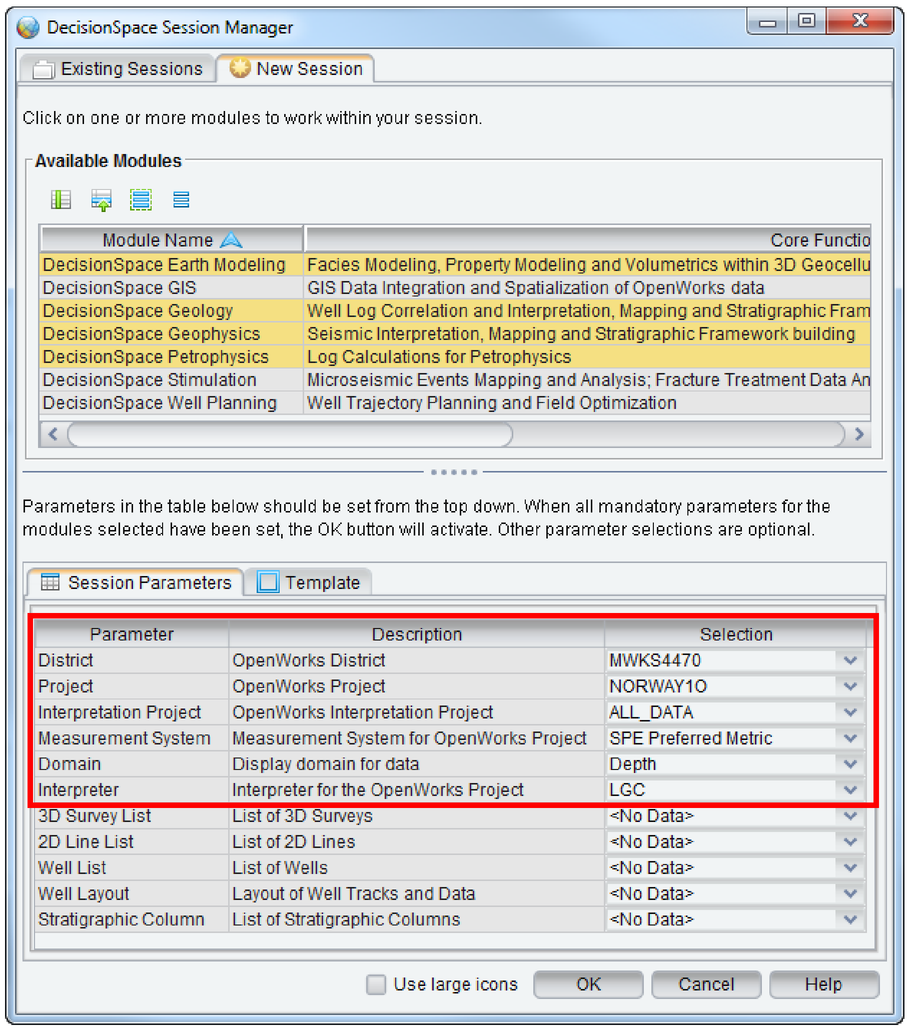
You can create additional probes using the *Create Probe* dialog. All probes for a 3D grid are listed per grid in the *Inventory*. See “Grid Probe MB3 Options” in the Help files for details on the options for viewing a grid probe.

Included is an option to extract a K layer from a .vdb grid and save it as a SeisWorks horizon, an OpenWorks surface grid, or a point set.

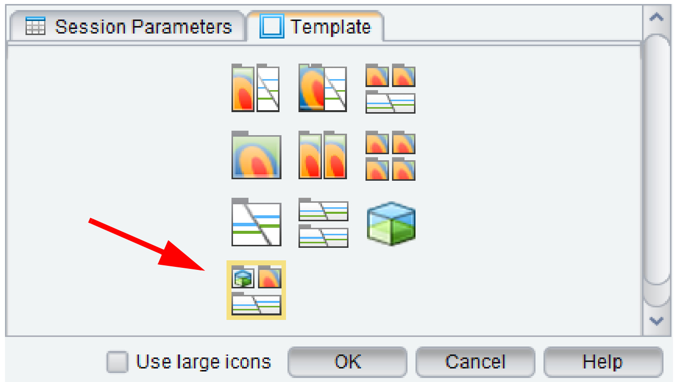
Exercise 7.1: Loading Log Curves and a Framework

In this exercise you will load log curves and a framework from the OpenWorks database.

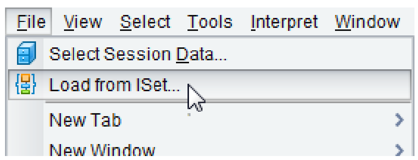
1. On the *DecisionSpace Geosciences Session Manager*, click the ***New Session*** tab, select the highlighted module names in the following image, and make the parameter selections shown in the *Session Parameters* tab.



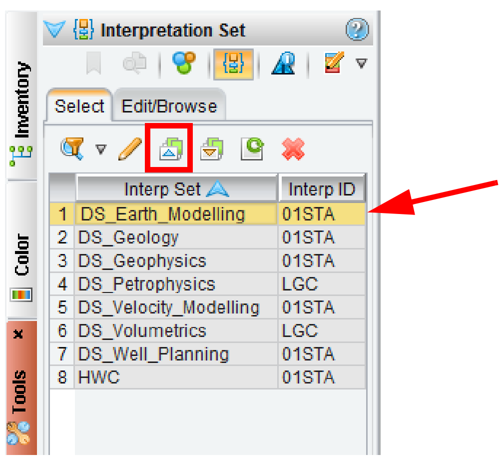
2. In the *Template* tab, select the **Cube**, **Map**, **Section** () icon. Click **OK**.



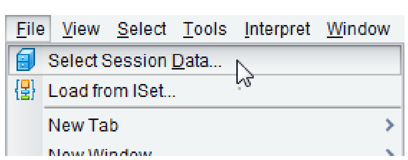
3. Select **File > Load from ISet...**



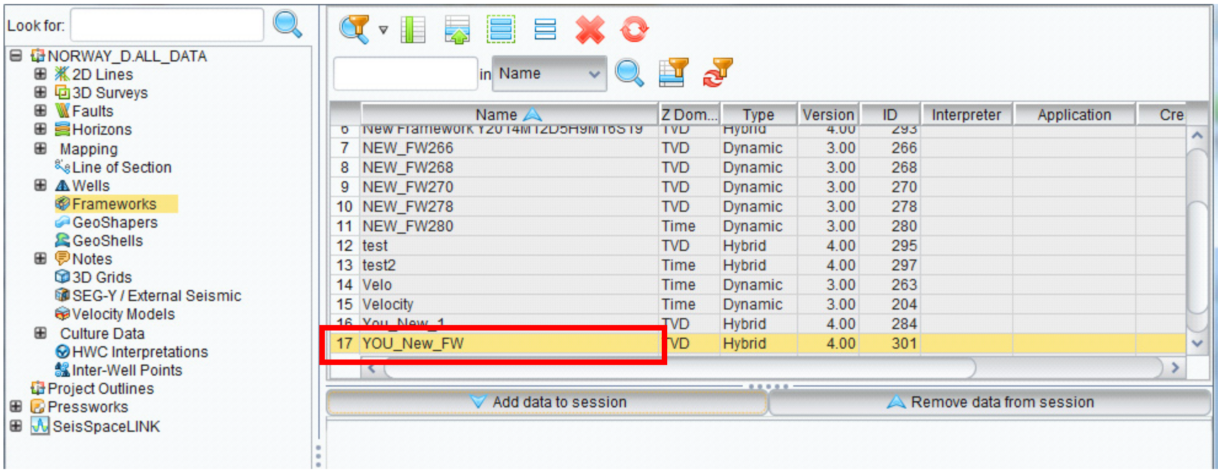
4. In the *Interpretation Set* panel of the *Tools* task pane, select **DS_Earth_Modeling**, then click the **Load Data to Session** () icon.



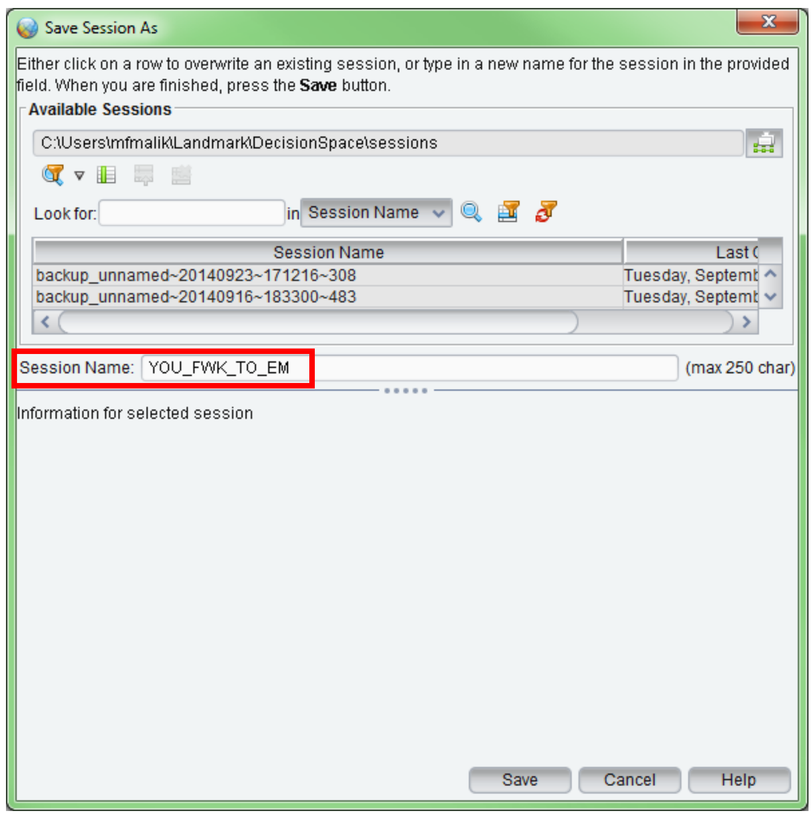
5. Choose **File > Select Session Data.**



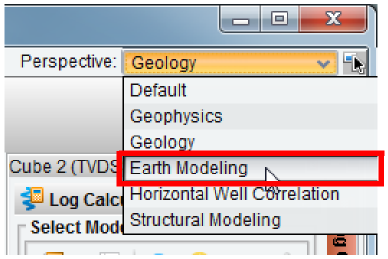
6. Select **Frameworks** in the left pane of the *Select Session Data* dialog. Then select **YOU_New_FW**. Click the **Add data to session** button, then click **OK**.



7. On the menu bar of the *DecisionSpace* dialog, select **File > Save Session.** The *Save Session As* dialog opens. In the Session Name: field enter "**YOU_FWK_TO_EM**" and click **Save**.



8. In the top right corner, on the Perspective: pull-down menu, select **Earth Modeling.** This enables you to view only the task panes associated with Earth Modeling exercises.

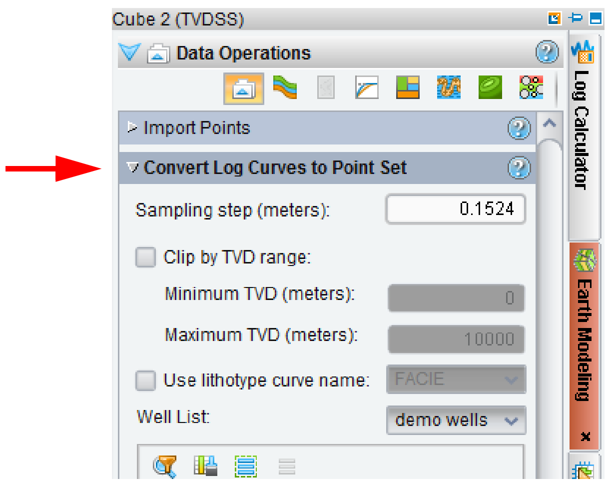


Exercise 7.2: Converting Log Curves to Point Sets

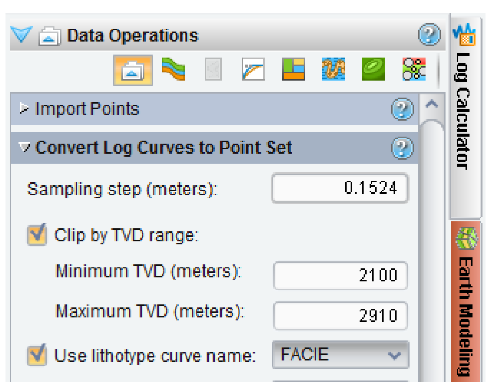
In this exercise, you will use Data Operations in the *Earth Modeling* task bar to convert the log curves to create a point set.

1. Double-click the ***Cube* view** tab to maximize it (avoid the X).

2. In the *Data Operations* panel of the *Earth Modeling* task pane, expand the ***Convert Log Curves to Point Set*** panel**.**

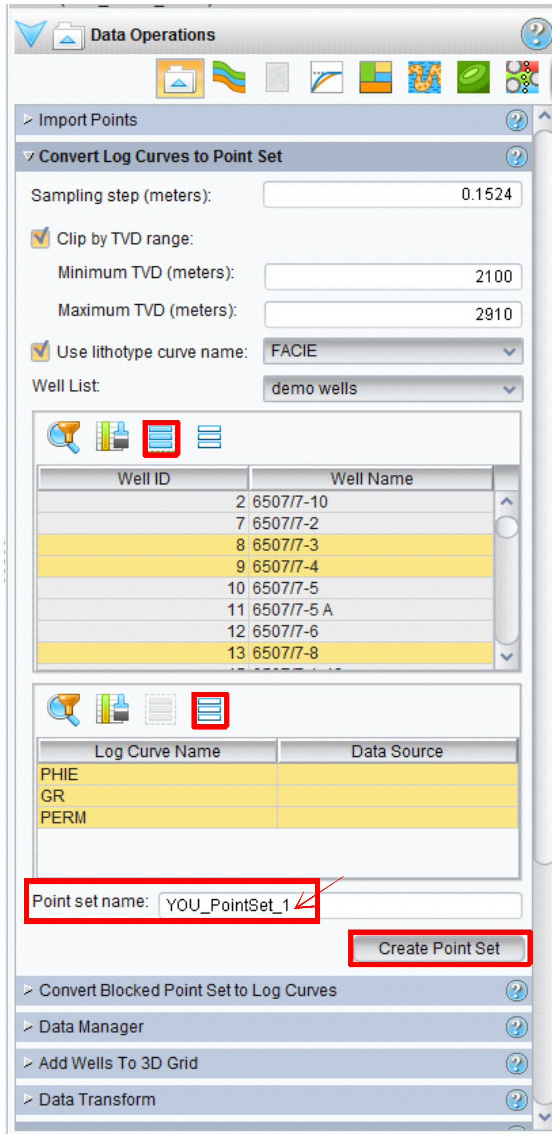


3. In the *Convert Log Curves to Point Set* panel, toggle on **Clip by TVD Range**. In the Minimum TVD (meters): field enter **"2100".** In the Maximum TVD (meters): field enter **"2910"**. Toggle on **Use Lithotype curve name:**.

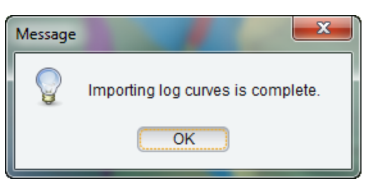


4. Still in the *Convert Log Curves to Point Set* panel, click the **Well List:** pull-down menu and select **demo wells.** Click the **Select All** ( ) icon. All logs associated with the wells in FACIES are listed

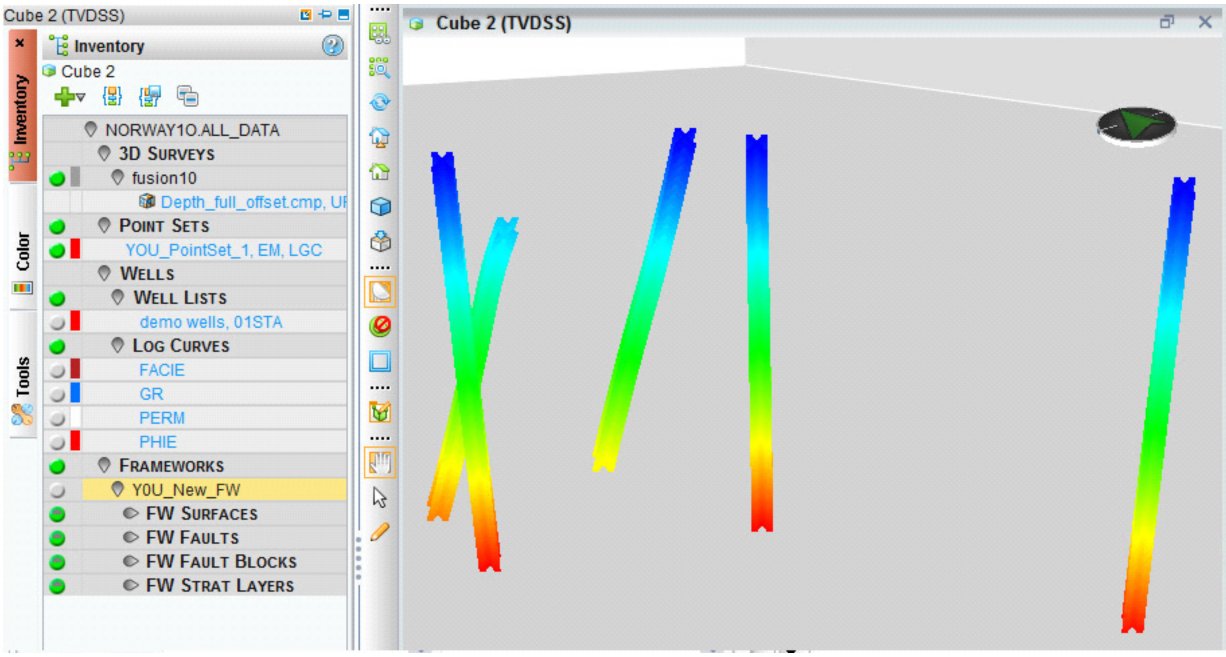
in the lower table with the Log Curve Name and Data Source. Select all of the Log Curve Names and in the Point set name: field, enter **“YOU_PointSet_1”**. Click the **Create Point Set** button**.**



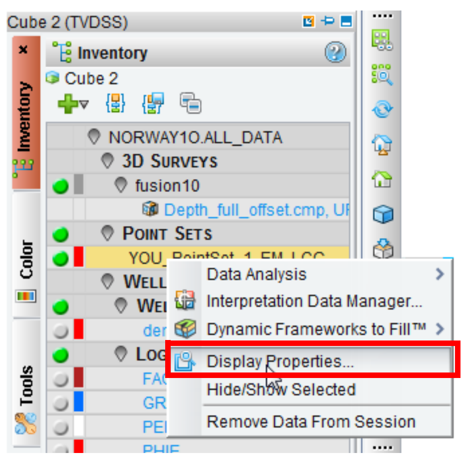
5. In the *Message* dialog, click **OK**. The point set is displayed in *Cube* view.



6. In the *Inventory*, toggle off **everything** except YOU_PointSet_1.



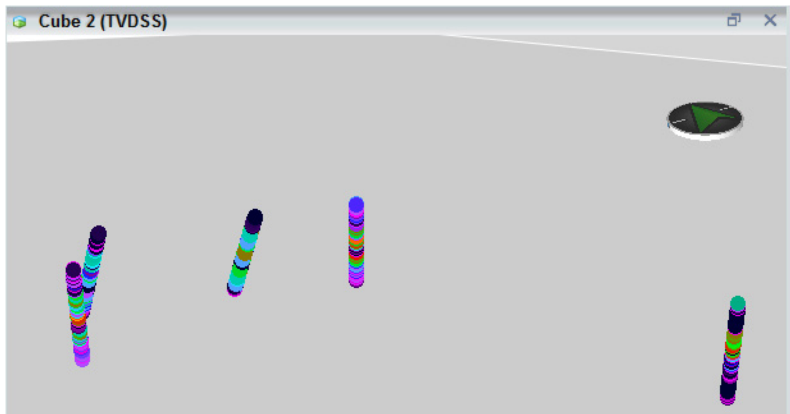
7. In the *Inventory* task pane, put your cursor on **YOU_PointSet_1** and
MB3 > Display Properties.



8. In the *Display Properties* dialog, expand the *Symbols & Annotation* panel and change the Symbol to the **solid circle**. On the Coloring Style: select **Color Map > System > 0_Spectrum**. In the Primary Field, select **PHIE**. Because Apply Immediately is toggled on by default, the changes are applied immediately in the display. Click **OK** to close the *Display Properties* dialog.

Note:

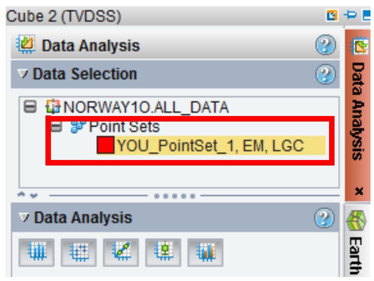
You can also access the *Display Properties* dialog by putting your cursor on an object in the view and MB3 > Display Properties.



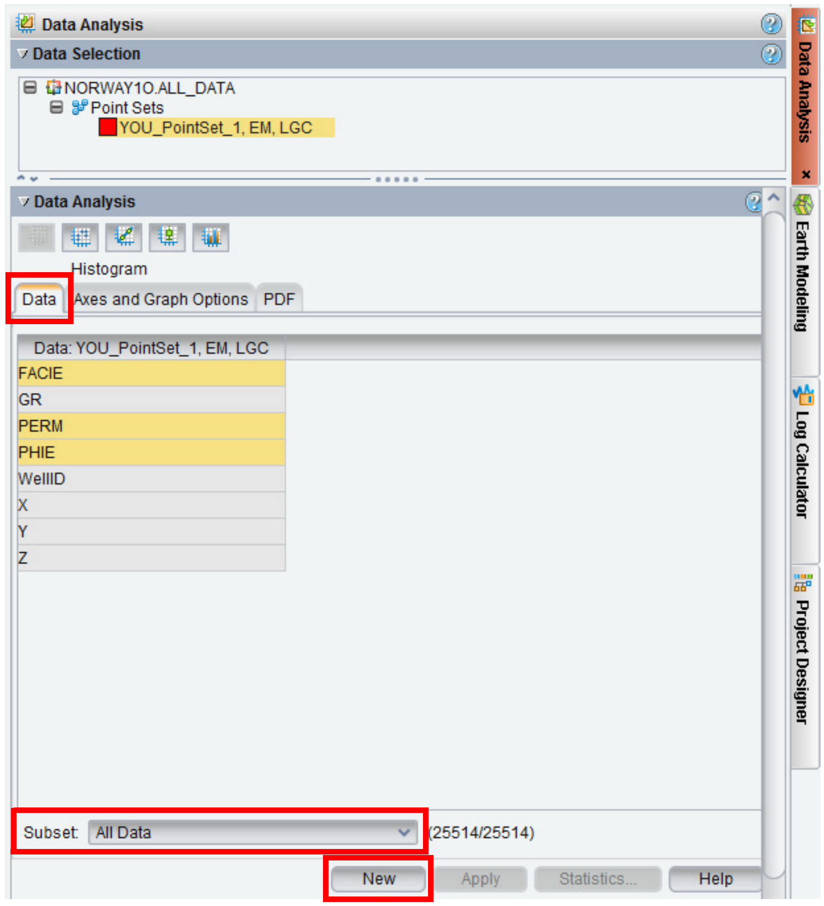
Exercise 7.3: Analyzing the Data

You can edit point sets if they have negative values or if you want to change values. Use the *Data Analysis* task pane to analyze the data using any point set. Then create subsets, filter the point set using the subsets to create a clean point set, and then run data analysis on the clean point set.

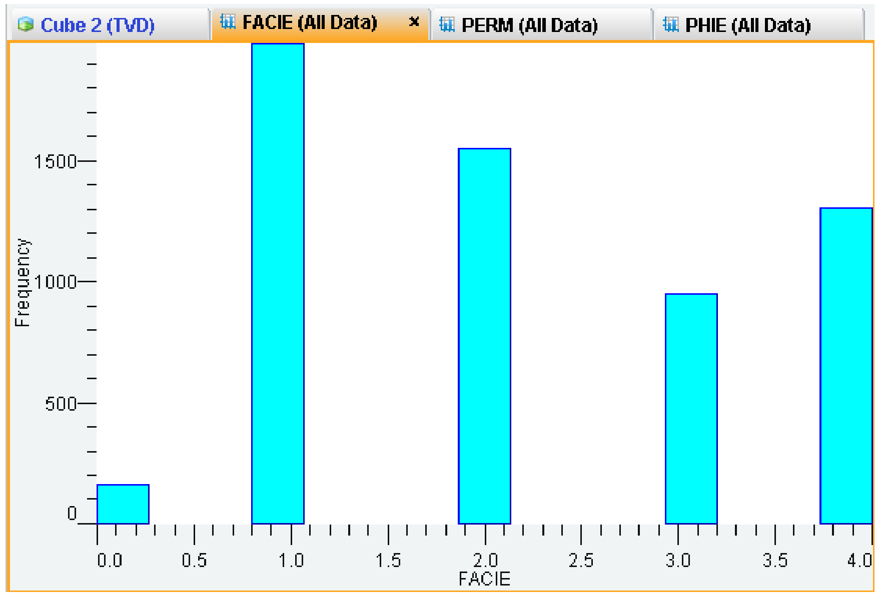
1. In the *Data Selection* panel of *Data Analysis* task pane select the **Point Sets** list, and click **YOU_PointSet_1, EM, LGC**. In the *Data Analysis* panel, select the **Histogram** icon.



2. Click the *Data* tab. On the *Data* tab, select **FACIE**, **PERM**, and **PHIE.** On the Subset pull-down menu, select **All Data**. Click the **New** button to create the new histograms of the data.



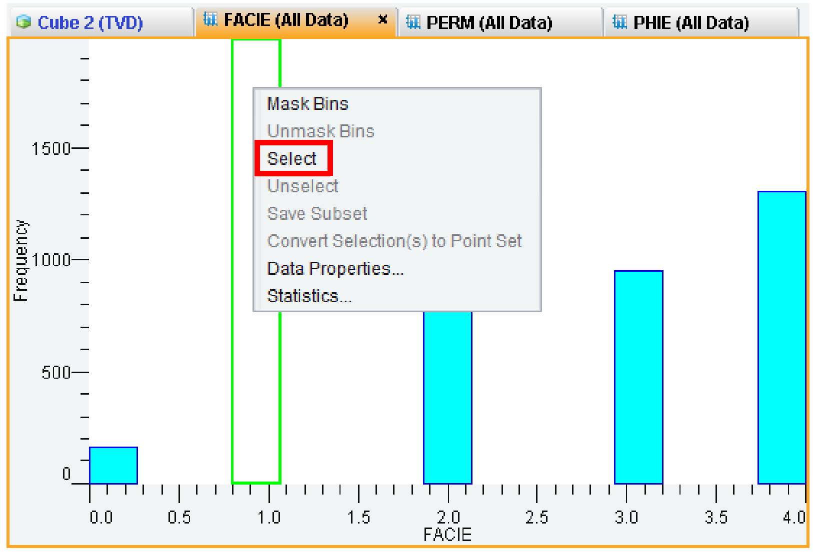
Histograms of the three objects you select in the previous step are on separate tab views. The image below shows the FACIE view.



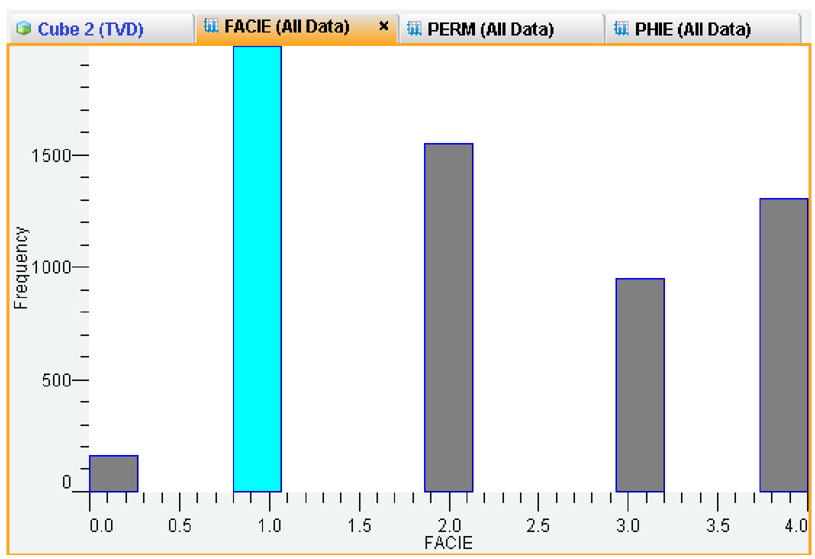
On the FACIE histogram are five facies codes, numbered from 0 through 4.

In this class you are using the Norway data set. A potential reservoir is Fangst sandstones, which were deposited during a low stand of the global sea. The Fangst group can be divided into four clearly recognizable stratigraphic units that, for modeling purposes, serve as basic reservoir units. In ascending stratigraphic order, these are F1, a coarsening-upward sequence deposited on a tidally-dominated shallow-marine shelf; F2, a coarsening-upward sequence deposited on a wave-dominated shallow marine shelf that culminated in upper-shore face deposition; F3, a coarsening-upward sequence deposited in deeper water on a marine shelf; and F4, a fluvial sequence.

3. Select the vertical histogram bar corresponding to the Facies 1 (1.0 on the x-axis), and **MB3** > **Select**.



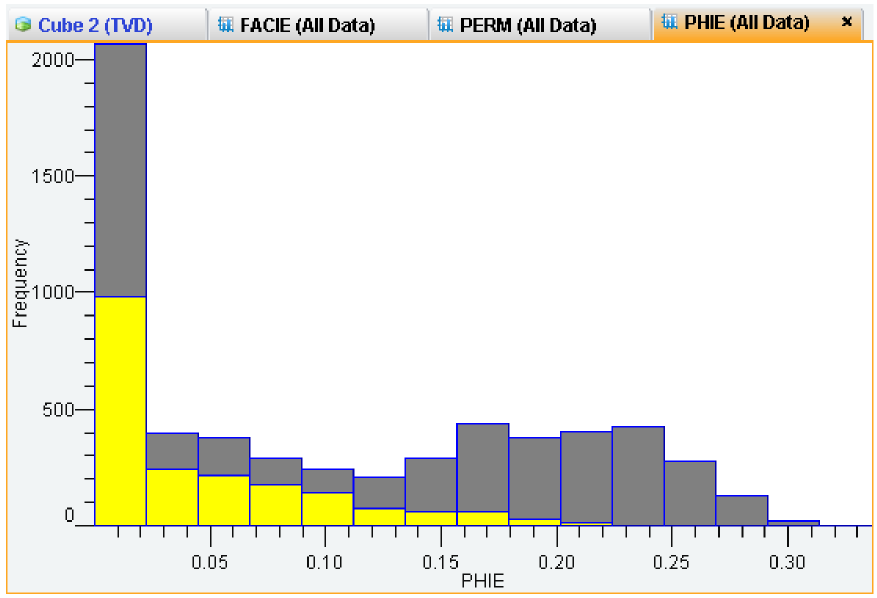
The histogram display changes to show that only Facies 1 is selected, as shown below.

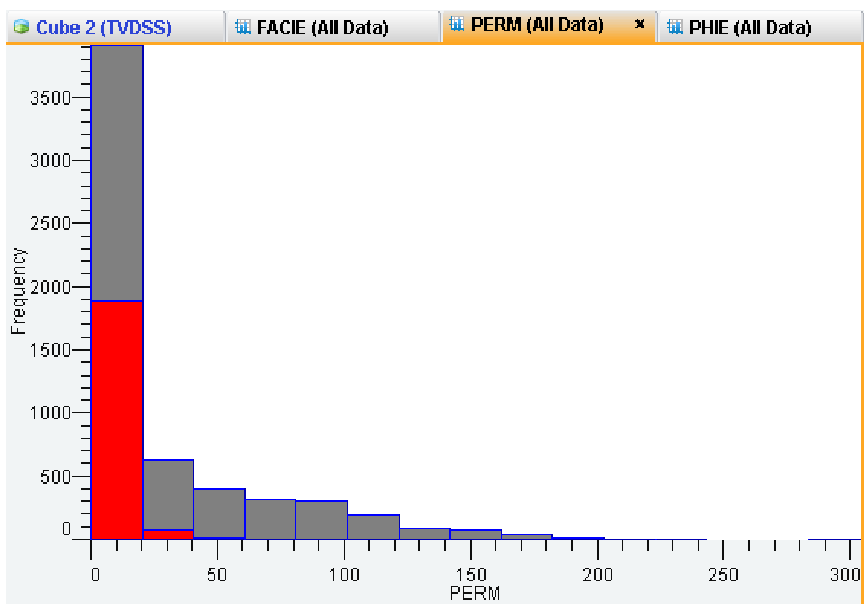


The histogram for PHIE shows that it is updated to display the distribution of these properties in FACIES 1. In the PERM (All Data) tab, the histogram for PERM also displays FACIES 1 in a highlight with all other values grayed out.

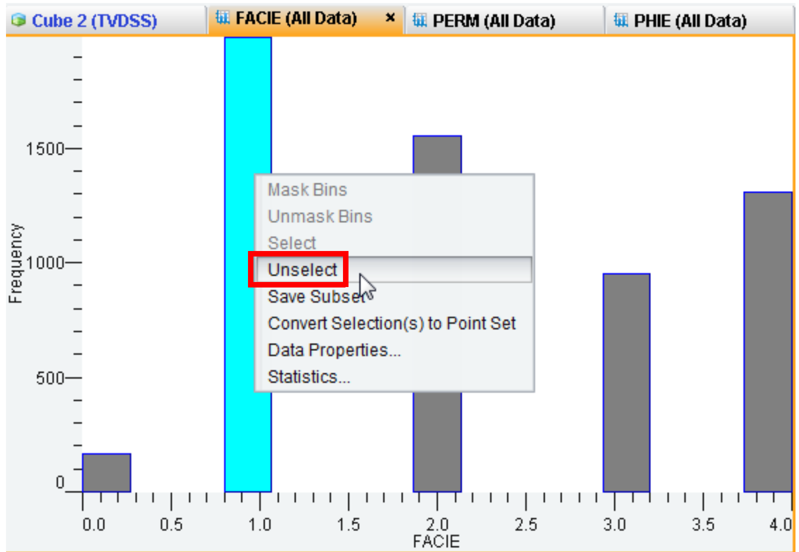
Note:

To easily move through the various Data Analysis tabs, use the left and right arrows on your keyboard.

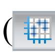

Cube 2 (TVD)
FACIE (All Data)
PERM (All Data)
PHIE (All Data)
Frequency
2000
1500
1000
500
0
0.05
0.10
0.15
0.20
0.25
0.30
PHIE

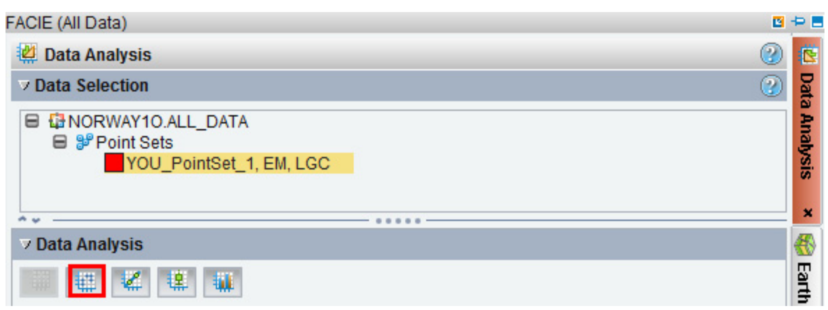

Cube 2 (TVDSS)
FACIE (All Data)
PERM (All Data)
PHIE (All Data)
Frequency
3500
3000
2500
2000
1500
1000
500
0
0
50
100
150
200
250
300
PERM

4. On the *FACIE (All Data)* tab, put your cursor on FACIE 1 and **MB3** > **Unselect**.

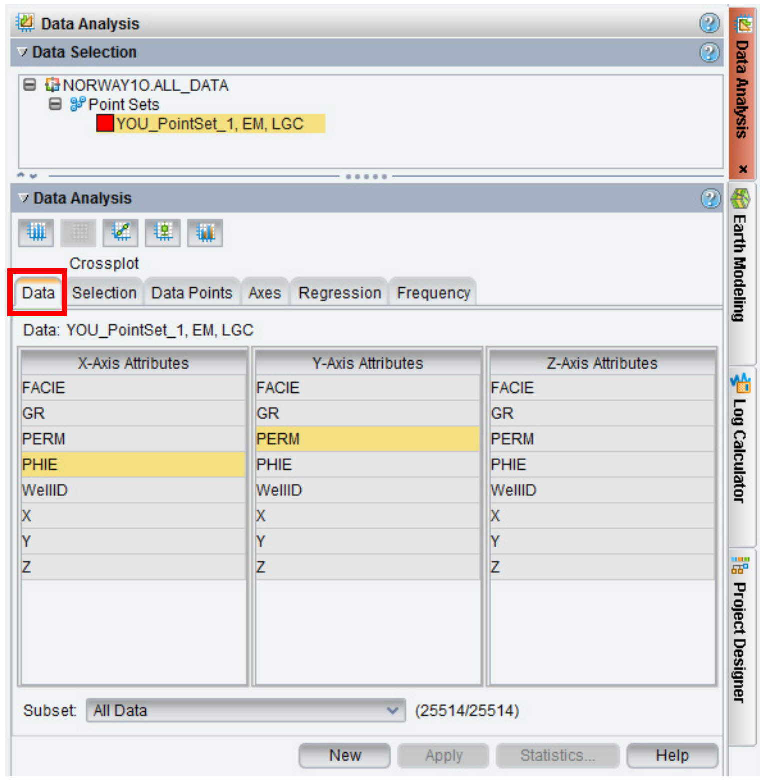


> **Note:**
>
> This selection process works in the same manner for all Data Analysis tabs. You can try it on the porosity and permeability histograms.

5. In the *Data Analysis* panel, click the Crossplot () icon.



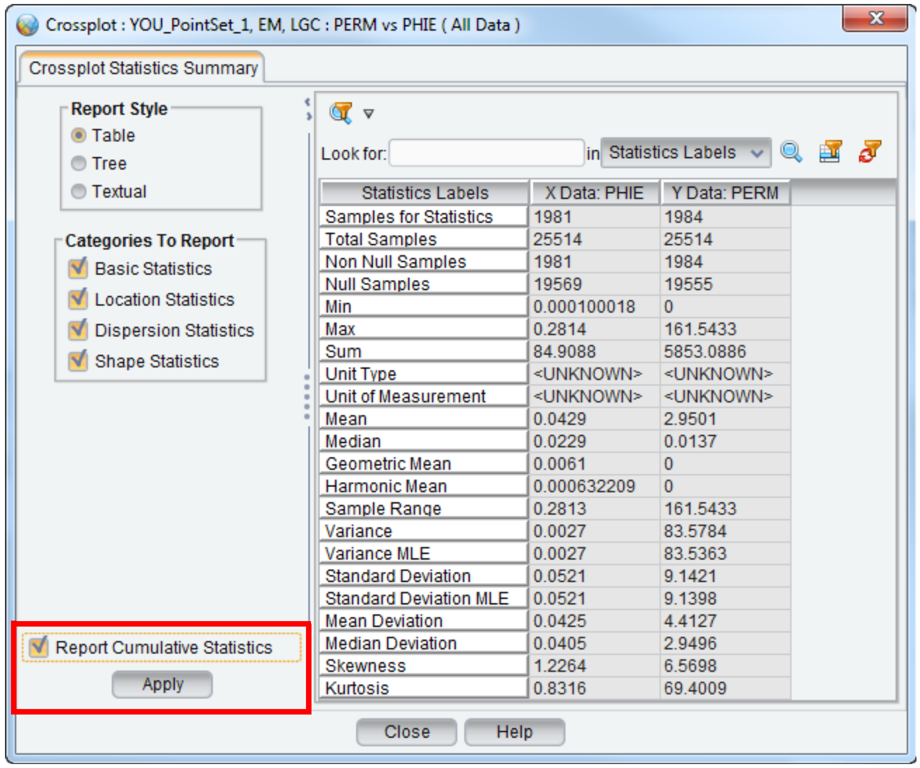
6. In the *Data* tab of the *Data Analysis* panel, select the X-axis Attribute as **PHIE**, the Y-axis Attribute as **PERM**, and in the Subset: pull-down menu, select **All Data**. Click **New**.



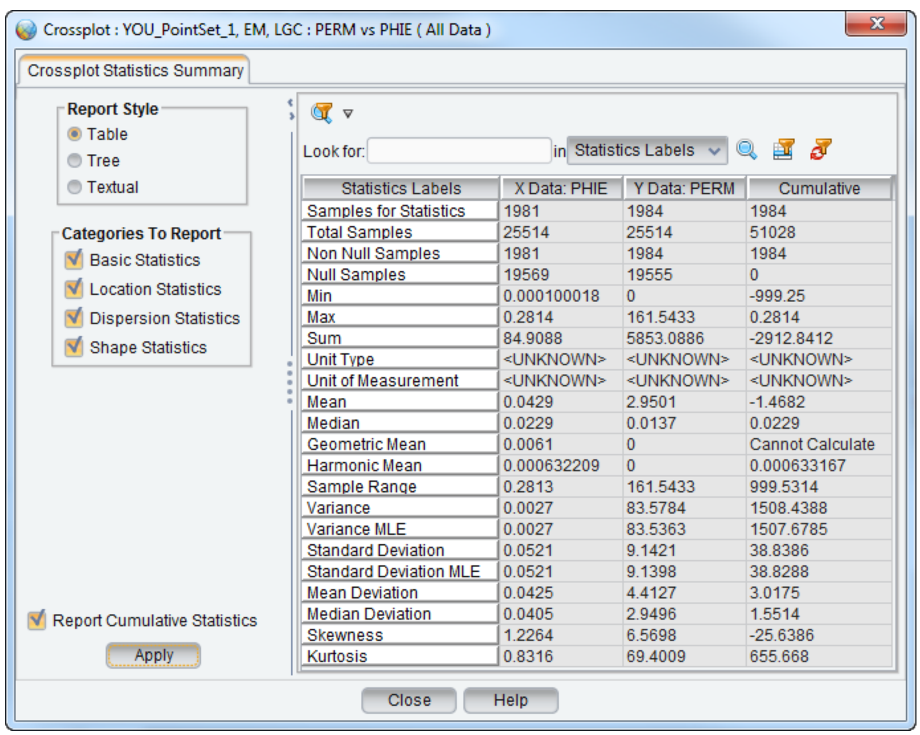
Observe whether there appears to be bad porosity values (less than 0). You can confirm this using Statistics.

7. In the *Data Analysis* panel of the *Data Analysis* task pane, click the Statistic (Statistics...) button.

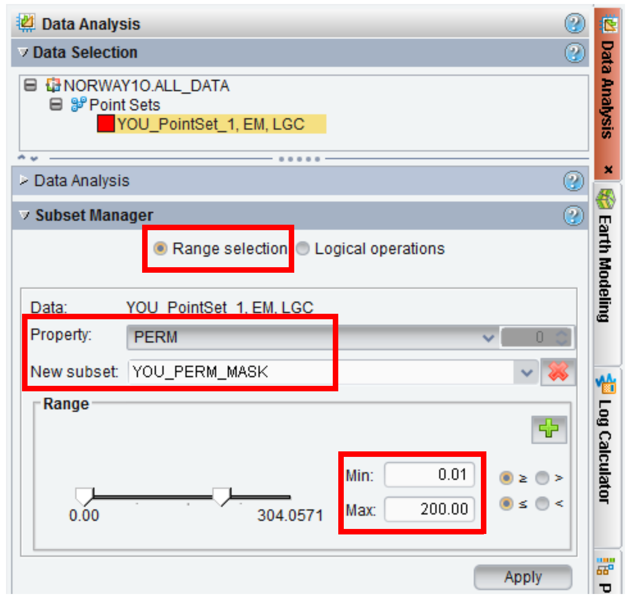
8. In the *Crossplot* dialog, toggle on **Report Cumulative Statistics**.Click **Apply.**



9. A cumulative column has been added to give more statistical information on the PERM vs PHIE crossplot. Click **Close**.

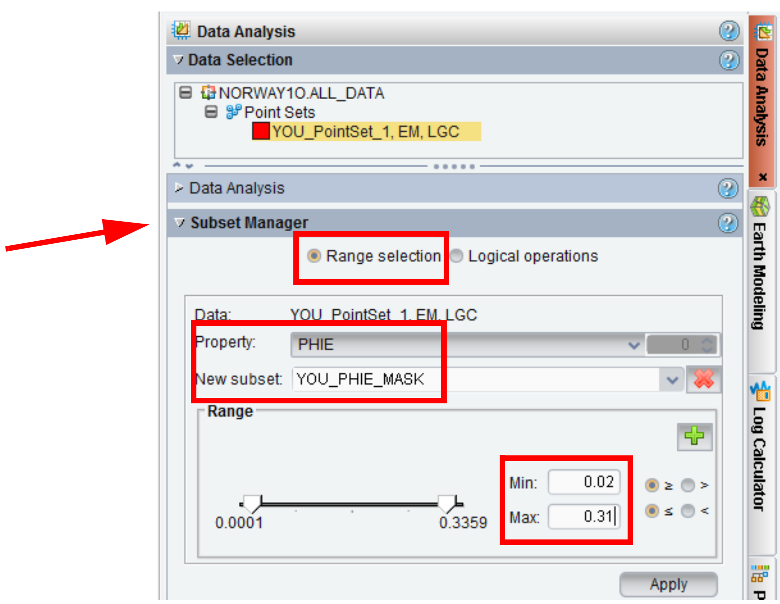


10. In the *Subset Manager* panel of the *Data Analysis* task pane, toggle on **Range Selection.** In the Property: pull-down menu, select **PERM**. In the New Subset: field, enter **“YOU_PERM_MASK”**. In the Range Min: field, enter min **“0.01”** and in the Range Max: field, enter **“200”**. Click **Apply**.

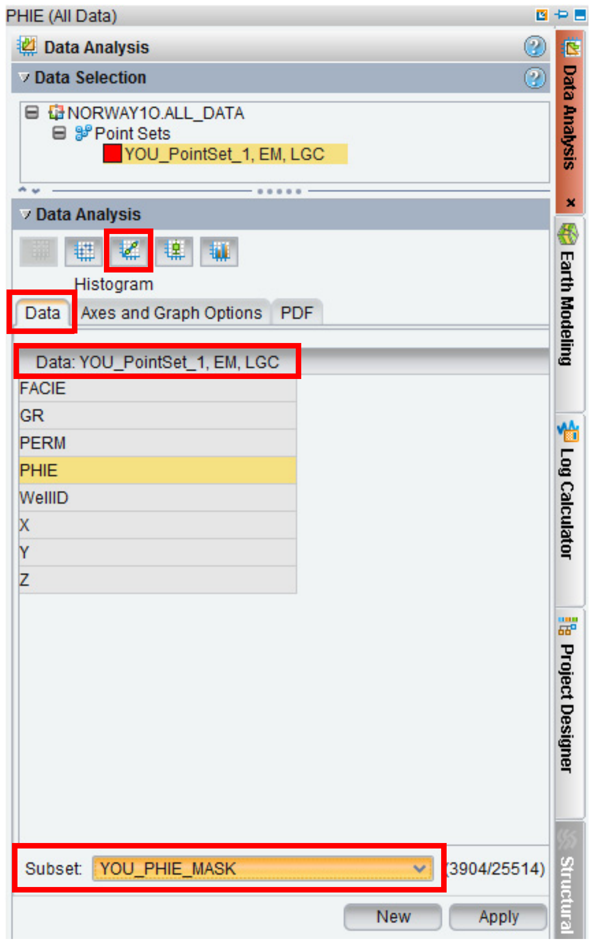


11. A *New Subset Confirmation* message is displayed. Click **OK.**

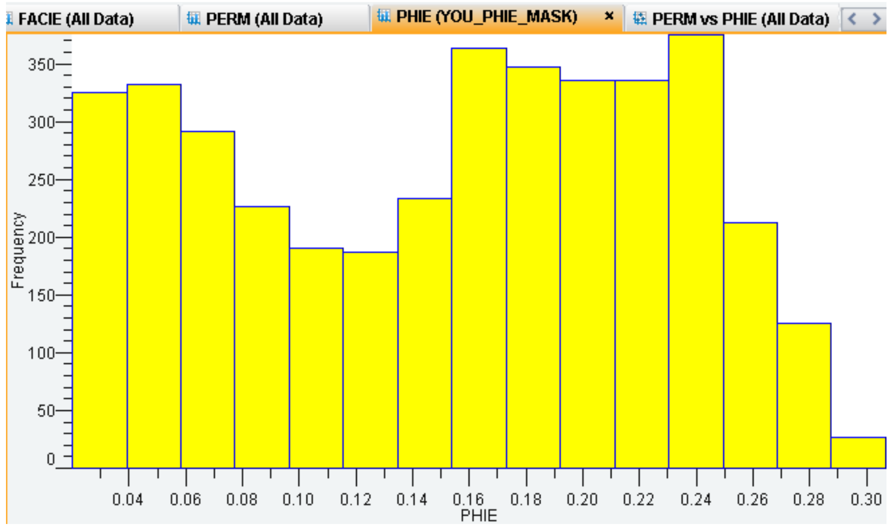
12. In the *Subset Manager* panel of the *Data Analysis* task pane, toggle on **Range selection**. In the Property pull-down menu, select **PHIE.** In the New Subset: field, enter “**YOU_PHIE_MASK”.** In the Range Min: field, enter **“0.02”** and in the Range Max: field, enter “**0.31”** Click **Apply**.



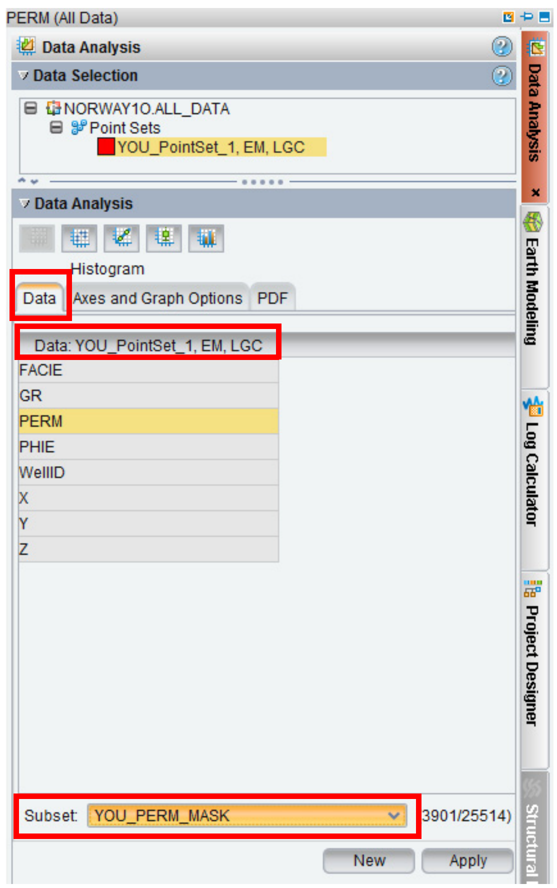
13. Activate your **PHIE (All Data)** tab. In the *Data Analysis* panel of the *Data Analysis* task pane, select the Histogram ( ) icon. In the *Data* tab, on the Subset pull-down menu, select **YOU_PHIE_MASK,** and click **Apply** to update the PHIE histogram.

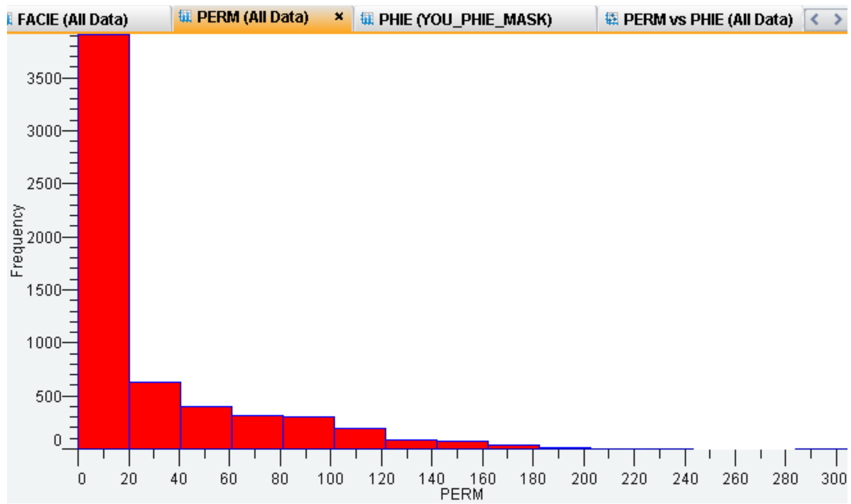


The *Histogram* window shows the PHIE histogram is now within range.



14. In the *Data Analysis* panel of the *Data Analysis* task pane, click the *Data* tab and select **PERM**. In the Subset: pull-down menu, select **YOU_PERM_MASK**, then click **Apply**.

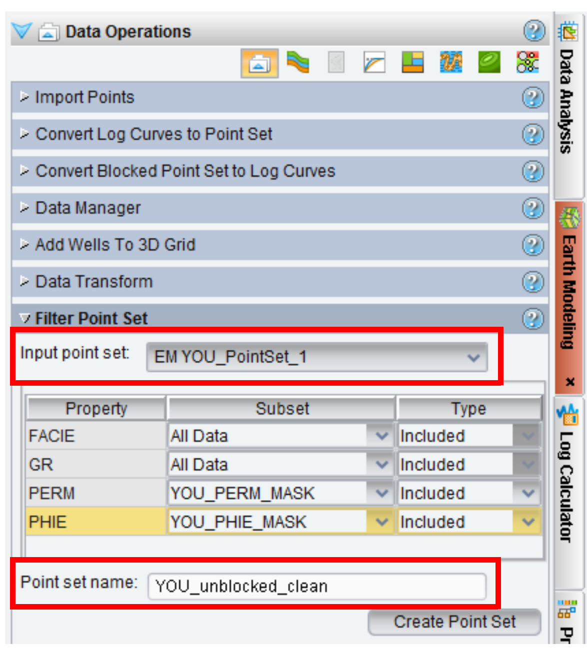




Next, you will create a new point set using these subsets, which are free of outliers.

15. In the *Data Operations* panel of the *Earth Modeling* task pane, expand the *Filter Point Set* panel. On the Input point set: pull-down menu, select **EM_YOU_PointSet_1.** Complete the parameter selections in your *Filter Point Set* panel to match the parameters in the following image, naming your Point Set **"YOU_unblocked_clean"**.

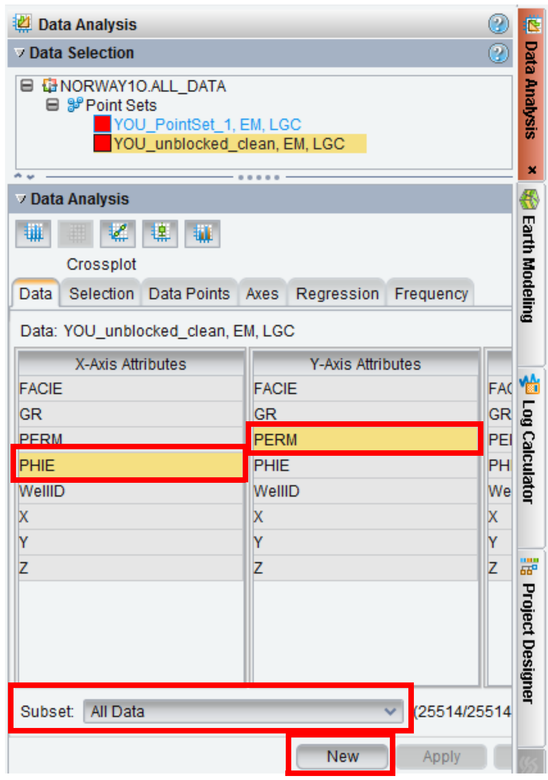
Your *Filter Point Set* panel will look like the following image.



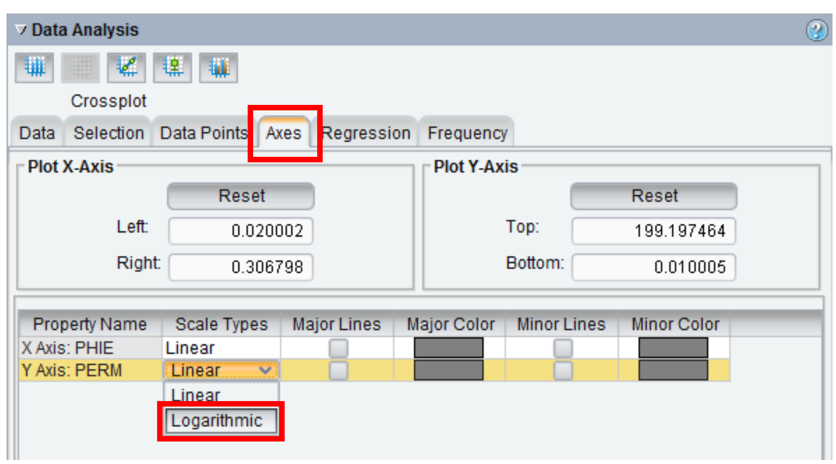
16. Click **Create Point Set.**

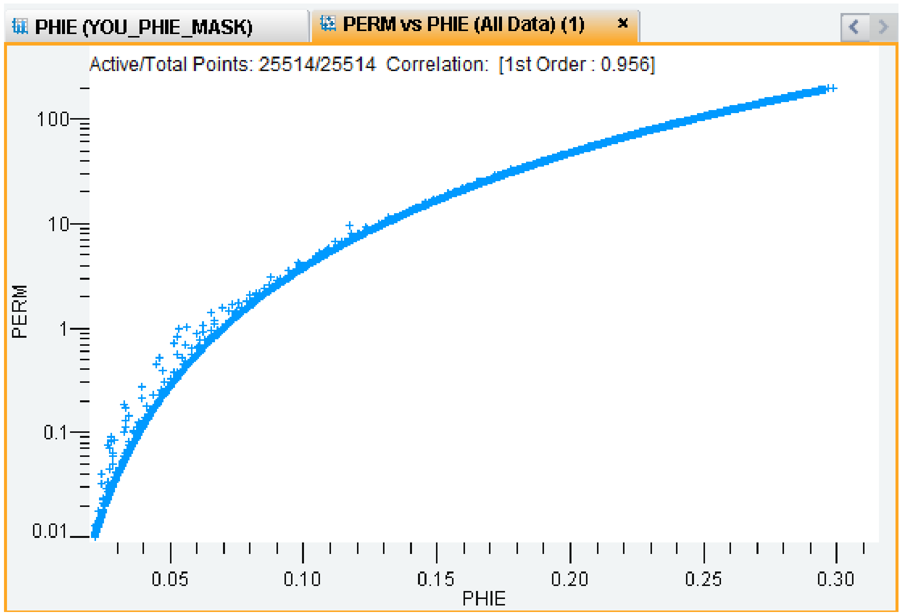
17. When a message box is displayed with the message, “Point set cleaning is complete”, click **OK**.

18. In the *Data Analysis* task pane, expand the *Data Selection* panel. Select **YOU_unblocked_clean, EM, LGC**. In the *Data* tab of the *Data Analysis* panel, select the **Crossplot** ( ) icon. Select **PHIE** as the X-Axis Attribute and **PERM** as Y-Axis Attribute. In the Subset: pull-down menu, select **All Data**. Click **New**.



19. On the *Data Analysis* panel, select the ***Axes*** tab. In the *Y-Axis* Scale Types pull-down menu, select **Logarithmic.** Click **Apply**.




PHIE (YOU_PHIE_MASK)
PERM vs PHIE (All Data) (1)
Active/Total Points: 25514/25514 Correlation: [1st Order : 0.956]
100
10
1
0.1
0.01
PERM
0.05
0.10
0.15
0.20
0.25
0.30
PHIE

Overview: Stratigraphic Modeling

The two major components of a sedimentary model are a stratigraphic model showing internal bedding geometries and a facies model showing the 3D distribution of facies. In stratigraphic modeling, the modeling is performed on an interval-by-interval basis. Within each interval, the task is to identify the depositional environment and primary depositional facies. Because each depositional environment is controlled by physical processes of sedimentation and erosion, there is an internal bedding geometry. These layers act as lines of internal correlation that affect the gathering of statistical information in variogram computations, so they should be recreated as accurately as possible in the model; this ultimately impacts the manner in which the facies and petrophysical properties are distributed within the 3D framework. The facies model provides the template within which you assign petrophysical properties on a facies-by-facies basis.

Once the layering styles are specified for each interval, the well data are resampled at the scale of the layers, and a single property value is assigned to each layer along the wellbore, thus creating a new point set that is used for the final modeling steps. This resampling is termed “blocking” the wells.

The Stratigraphic Modeling tool allows you to build a stratigraphic geocellular grid (PM Grid3D), define lithotypes and assign facies to the lithotypes, and block seismic volumes and well log data. The steps are:

- Grid geometry
- Lithotype definition
- Seismic attribute blocking (optional)
- Well blocking

PM Grid3Ds and Grid Probes

PM Grid3Ds are the 3D grids created by the DecisionSpace Earth Modeling software, using the Create 3D Grid panel in the *General Property Modeling* dialog or the *Grid Geometry* panel in the *Stratigraphic Modeling* dialog. Also, when a grid is loaded (using Tools > Load 3D Grid) a PM Grid3D is automatically created. PM Grid3Ds are visualized using a probe that you can move through the grid in the *Cube* view. If a *Cube* view is active when a 3D grid is created, a box probe is automatically created and displayed in the *Cube* view. The KLayer property is displayed using a default color map. If a *Cube* view is not active when a 3D grid is created, you must select *Cube* view and toggle on the probe in the *Inventory* task pane in order to visualize the box probe in 3D.

The *Display Properties* dialog for PM Grid3Ds allows you to create new grid probes and to change properties and display parameters for existing grid probes.

You can access this dialog by either of the following methods:

- Put your cursor on a PM Grid3D in *Cube* view and MB3 > Display Properties.
- Put your cursor on PM Grid3D in *Inventory* and MB3 > Display Properties.

All probes for the selected PM Grid3D are listed in the *Display Properties* dialog with the Grid name, Property pull-down menu, Subset pull-down menu, and Probe type. Where there are multiple realizations of a property, a combo-box to the right of the Property field is activated, allowing you to select the desired realization. When you place your cursor over the combo-box, the Min/Max Index (minimum realization number and maximum realization number) are shown in a pop-up.

The probe types are:

- **Box —** a hollow cube; what you see are the six exterior faces.
- **I-Plane** — a one-cell-thick plane that moves along the I direction.
- **J-Plane** — a one-cell-thick plane that moves along the J direction.
- **K-Plane —** a one-cell-thick plane that moves along the K direction.

- **X-Plane** — a one-cell-thick plane that moves along the X axis.
- **Y-Plane** — a one-cell-thick plane that moves along the Y axis.
- **Z-Plane** — a one-cell-thick plane that moves along the Z axis.
- **Cut** — a box with a hole in the center.
- **Filter** — a solid cube; can be used to display only cells within a certain range of values by using the *Color* task pane to set a range of values from opaque to transparent. A filter probe differs from a box probe in that the box probe displays only exterior cells, regardless of value with the cells colored by the applied color map.
- **Arbitrary Slice —** creates an arbitrary slice of cell faces.
- **Arbitrary Slice Cell —** creates an arbitrary slice of one cell thickness using the cell closest to a diagonal plane through the slice.

You can show or hide a probe in the view by toggling on or off in the *Inventory* panel. To remove a grid probe, select it in the Active Probes table and click the Remove icon (above the table).

You can change the property or subset of a grid probe by selecting from the pull-down list in the property or subset cell of the table.

MB3 Options

When you MB3 in a grid cell in the table, a pull-down menu opens. Click the associated checkbox to toggle on, or uncheck the box to toggle off, the following options:

- **Manipulator —** displays a red outline around the probe with handles for manipulating the probe when toggled on.
- **Annotation —** displays a yellow overlay window when you click <Shift> <Ctrl> + MB1 on a probe. The window shows the type of probe and the X, Y, Z, or I, J, K ranges, depending on the Type of probe. It also tracks cursor movement by indicating X, Y, Z or I, J, K coordinates as you move or pan a probe.
- **Clip to Probe —** truncates any object in the *Cube* view as soon as that portion of the object is more than one cell width away from the probe that is selected when you click this option. The truncation is performed each time the probe is modified.

- **Fault Cuts —** displays the faces of cells adjacent to faults in the grid. Typically, for each cell adjacent to and on one side of a fault, there is another cell on the other side of the fault facing it.

Selecting Color

Use the parameters below the table to specify the following display parameters:

- **Time Step** — activated only for .vdb files containing timestep information (RECUR data). You can use the slider bar to select a time step or enter a value in the text box. Use the time step buttons on the right to move to the first, previous, next, or last time step.

- **Color —** allows you to toggle on one of two **options**:

  — **Solid** opens the *Color* dialog, allowing you to select a single color for the probe.

  — **Color map** allows you to select from the list of preset color maps. Use the *Color* task pane to edit and save color maps.

- **Opacity —** allows you to specify a level of overall opacity for the displayed probe. Use the pull-down menu to select a value from 1.0 (fully opaque, which is the default) to 0.1 (functionally invisible, or showing a faint outline). To make a Filter probe totally transparent (so certain data values are filtered out) use the opacity editing functionality on the *Color* task pane.

Manipulating a Probe

How you manipulate a probe depends on the probe Type. The information below is provided for reference because a box probe is created when you do grid geometry. You will work with all of these probes in the exercise Grid Probe Manipulation after you perform Facies Simulation and have populated the grid with properties.

During manipulation, a yellow line with arrows on the ends appears, indicating the axis along which the probe is being moved.

Manipulating a Box or Cut Probe

After a Box or Cut probe is selected as the active object (by clicking it), you can resize (shrink or expand) the probe in one direction by interactively pushing a side (face). Press and hold <Shift>, then click the side that you want to push or pull and drag the cursor. To resize in more than one direction, click-and-drag a yellow handle node on the corners. For box probes, you must first MB3 on the probe and select Manipulator to display the outline around the probe.

On cut probes, you can also resize the probe by moving the interior faces using the manipulator handles.

After shrinking the probe, you can move the probe along any axis by pressing and holding down **<z>** and then dragging your cursor up or down, or dragging it left or right. The axis along which the probe moves depends on the face that you click on. The probe moves parallel and perpendicular to your view.

Manipulating a Plane Probe

To move a plane probe in the X or Y or Z direction or in the I or J or K direction (depending on the type of probe), position the cursor over the probe, press and hold <Shift>, and then click-and-drag. The probe type determines the direction in which the probe moves. For example, an I-plane probe moves in the I-direction, so the I values change as it is moved. This is different from panning an I-plane probe (see below), in which case the I value will remain constant and the J and K values will change. To see your X, Y, or Z values or your I, J, or K values, MB3 > Annotation. The cursor coordinates are then displayed in the yellow note box at the top of the display window, and change as you move your cursor.

To resize a Plane probe, put your cursor on the probe and MB3 > Manipulator. This displays a red outline around the probe with yellow handle nodes in the corners. Press <Shift> and click-and-drag a handle to resize the probe.

After the probe has been resized you can pan the probe within the 3D grid by positioning the cursor over any visible portion of the probe (except a handle) and click-and-drag.

> **Note:**
>
> When any side of the probe pushes up against the bounds of the grid, it will appear to resize. This is only because the portion of the subset rectangle that is outside of the grid is not visible. The dimensions of the subset rectangle remain the same. You will work with plane probes in the Facies Simulation exercise.

Manipulating a Slice Probe

Two slice options are available. Arbitrary slice is a slice probe with an arbitrary slice of cell faces. Arbitrary slice cell is a slice one-cell thick that displays the cell closest to the arbitrary slice. By default, two vertical diagonal slices from the corners of the grid are displayed along the K Min plane, i.e., the top of the probe. Green lines with handle nodes on both ends are displayed at the plane min or max, depending on where the points were picked. The default size is the full extent of the input grid.

To create additional slices you can put your cursor on the plane you want to pick on, and MB3 > Set Picking Plane, selecting two points on the plane.

A slice probe is moved and resized by <z>-click-and-drag on a corner handle. When you release the handle, the slice is redisplayed along the plane.

To remove slices related to a probe, put your cursor on the probe and MB3 > Remove. To remove a single slice from the probe, put your cursor on a handle node and MB3 > Remove Slice Plane.

Manipulating a Filter Probe

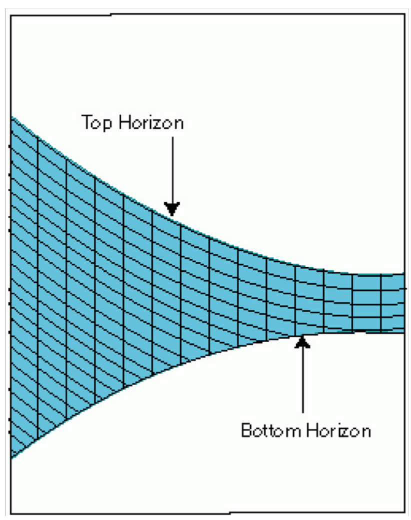
Filter probes cannot be a subset or translated, as can other probes, because filter probes display cells based on their value. The display of cells is controlled by using the *Color Bar* task pane to set a range of values to transparent.

> **Note:**
>
> You cannot use the Transparency slider that is on the *Display Properties* dialog to set transparency for this purpose.

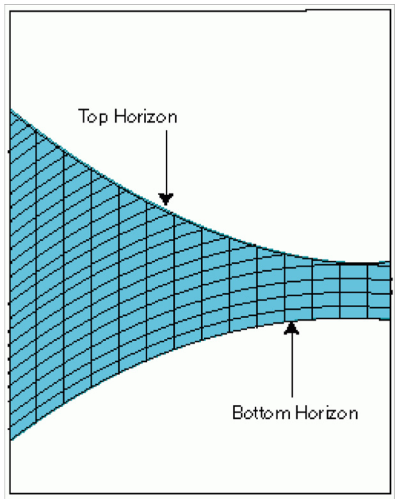
Layering Styles and Implications

In the Grid Geometry step of the *Stratigraphic Modeling* panel, you must specify a layering style for each interval in your structural model. An interval is the space between two surfaces, which may be divided into many layers. There are three possible layering styles:

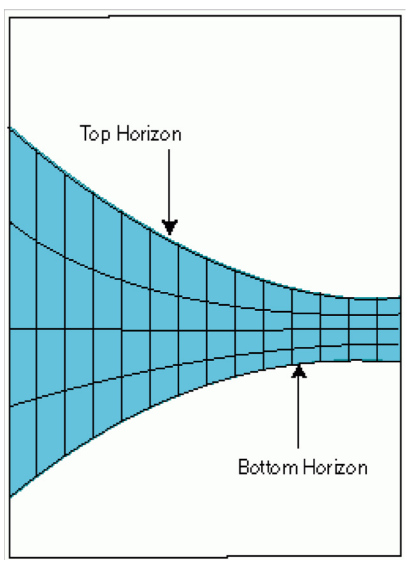
- **Parallel to Top** — cell layers are parallel to the top (shallowest) surface of the interval. This produces cells of constant thickness, as specified by the user, except where the layers onlap or baselap the bottom surface of the interval.



- **Parallel to Bottom** — cell layers are parallel to the bottom (deepest) surface of the interval. This produces cells of constant thickness, as specified by the user, except close to the top surface of the interval, where they are eroded or toplap.



- **Proportional** — cell layers are equidistant between the top and bottom surfaces of the interval. This layering is used when the interval thickens and thins due to compaction or drape, not by erosion, creating the same number of layers (as specified by the user) throughout the interval. It produces thick cells where the top and base are far apart and thin cells when the layers become closer to one another.



For Parallel to Top and Parallel to Bottom, three-layer thickness must be specified. The corresponding value for number of layers is computed from the maximum thickness of the interval and is automatically updated. This will show the maximum number of layers that can be created. The actual number could be slightly lower, depending on your surfaces.

For Proportional layering, the number of layers must be specified. This denotes the maximum number of cells stacked vertically within the interval. The number of layers could be based on the average thickness of the interval, for example.

Regardless of the layering style you choose, the cell walls are always vertical, creating flow-simulation-friendly 3D grids; the cells have vertical or gravity-down cell walls.

Note:

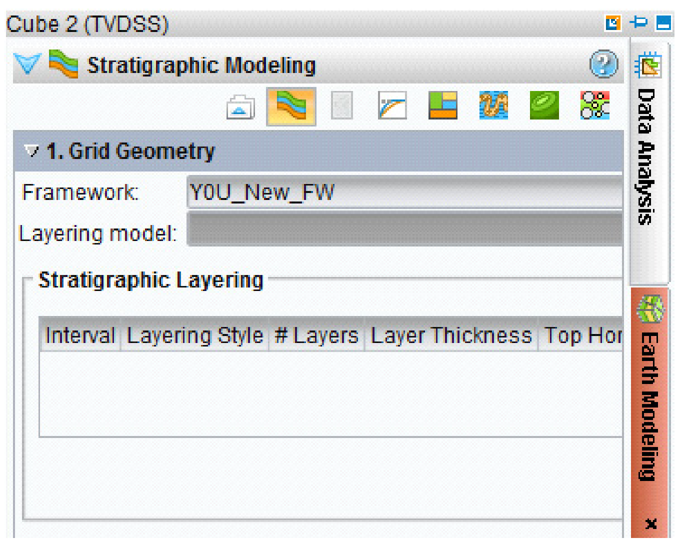
You may want to consult with your reservoir engineer for reasonable layer thicknesses for your particular reservoir, as adding too much detail may lead to extended computation times with little or no benefit.

Exercise 7.4: Generating a 3D Geocellular Grid

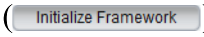
The *Grid Geometry* tab in the *Stratigraphic Modeling* panel allows you to select a GeoModel with its associated layering model and then specify the layering style and number or thickness of layers within each interval. The Top Horizon and Bottom Horizon cells show the bounding surfaces for each interval, and cannot be changed. The Avg Interval Thickness, Min Interval Thickness, and Max Interval Thickness are the average, minimum, and maximum thickness computed between the two horizons of the interval, and are read-only information provided as a guide for the user to specify layer thickness or number of layers.

In the Stratigraphic Gridding section of the panel, you can change the size and areal extent of the selected Framework model where it covers too much area and adjust the rotation based on your knowledge of the Framework.

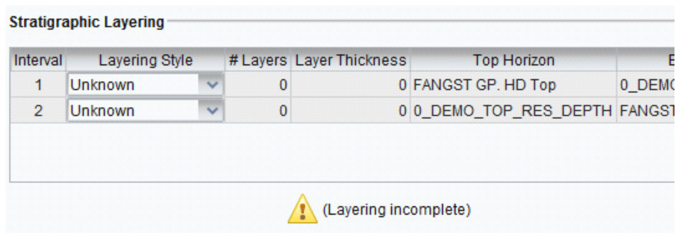
1. On the Earth Modeling task pane, select the **Stratigraphic Modeling** () icon and expand the ***Grid Geometry*** panel.



The Framework model that you loaded in the exercise “Load Log Curves and a Framework” is selected automatically.

2. Click the **Initialize Framework** (

) button.

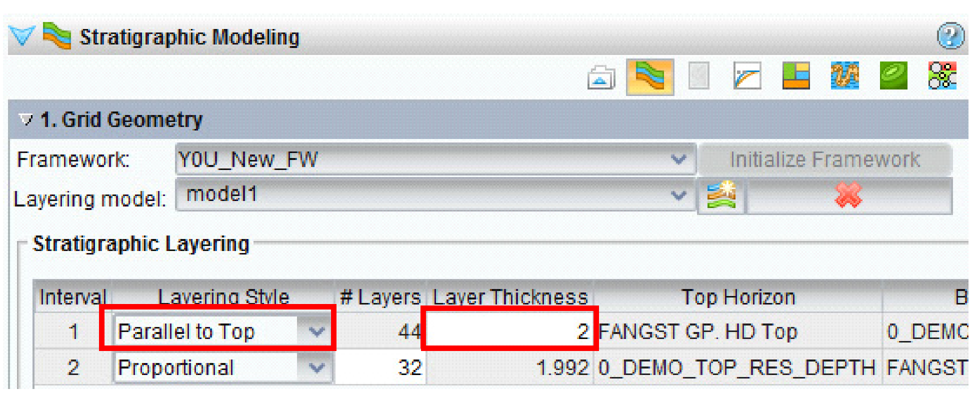
> **Note:**
>
> A warning message will appear in the task pane after you click the Initialize Framework button. This informs you that the layering style options have not been populated. The warning message disappears when the options are populated.
>
> 
>

This sequence must be performed when you build a new grid, to verify that the selected Framework is ready to use for 3D grid building. The Framework must contain more than surface. If the check fails, you will receive a pop-up Warning message that indicates why the software is unable to use the selected Framework. If the check passes, the software computes the layering thickness and populates the panel with the grid geometry for the selected Framework.

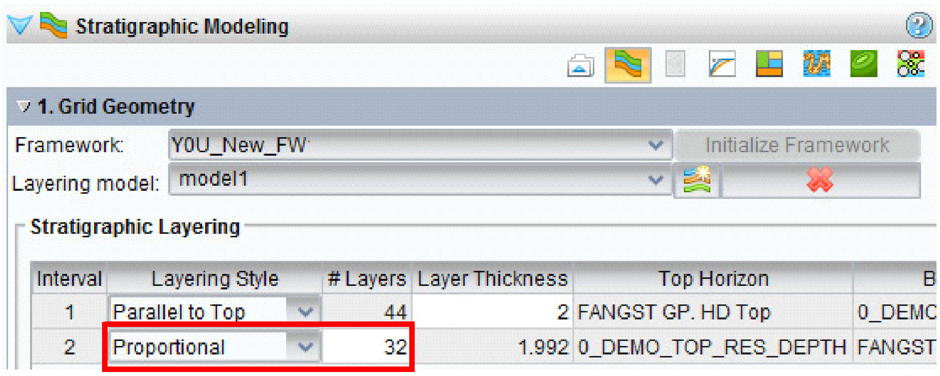
The Stratigraphic Layering table shows the two intervals between the three surfaces, along with the number of layers, layer thickness, top horizon, bottom horizon, average interval thickness, minimum interval thickness, and maximum interval thickness for each interval.

The top half of your *Grid Geometry* panel will look like the image shown below.

3. In the *Grid Geometry* panel, for Interval1, select Layering Style **Parallel to Top**, and in the Layer Thickness cell enter **“2”**.



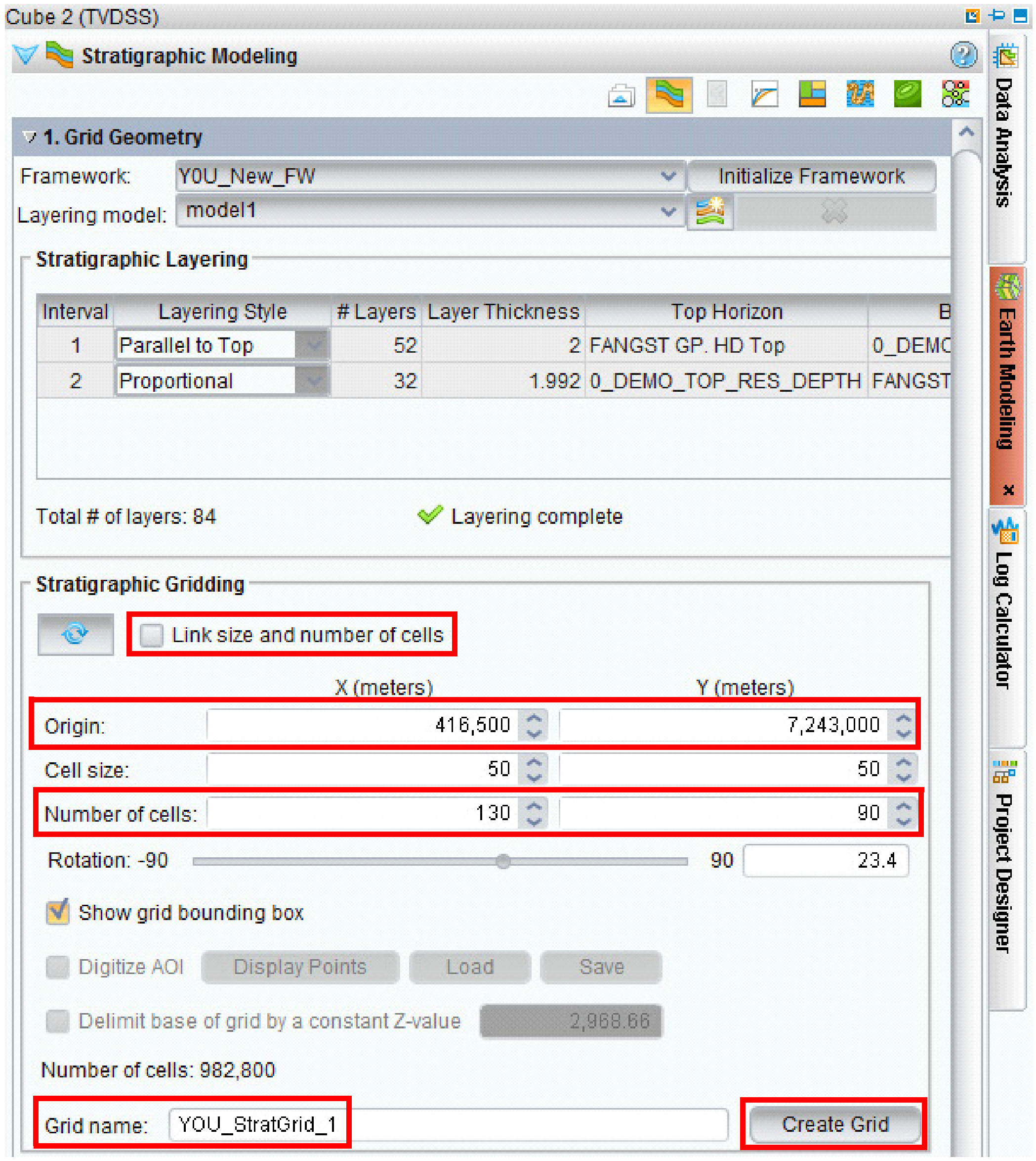
4. For Interval2, select **Proportional** in the Layering Style pull-down menu. In the # Layers cell, enter **"32"**. (Use this value because the average interval thickness for Interval2 is 64.2 and you want layer thickness to be 2.0.) Click **Enter.**



5. In the *Stratigraphic Gridding* panel of the *Grid Geometry* panel, for X Origin enter **"416,500"** and for Y Origin enter **"7,243,000"**. In the Rotation field, enter **"23.4"**. This is the rotation of the fusion10 survey.

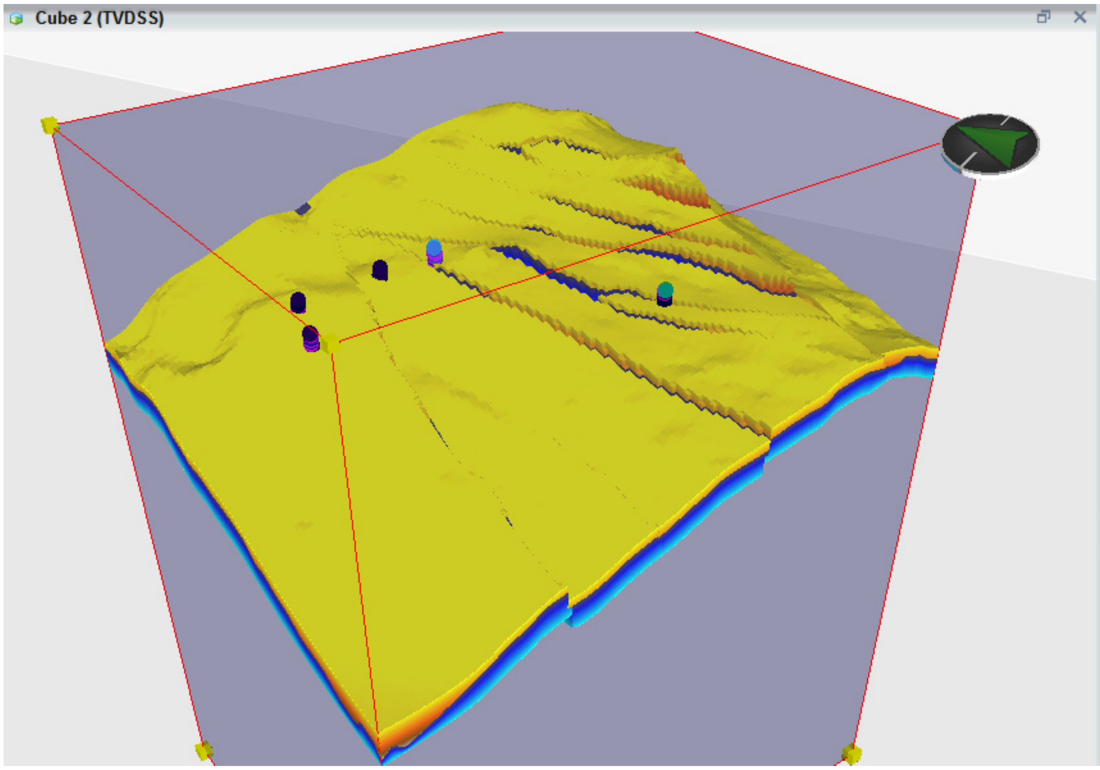
   - Toggle off **Link size and number of cells**, so the Number of cells for X and Y can be changed independently.
   - In the Number of cells: X (meters) field, enter **"130"**, and in the Number of cells: Y (meters), enter **"90"**. Note that the Number of cells changes.
   - In the Grid name: field, enter **"YOU_StratGrid_1"**.
   - Toggle on **Show grid bounding box**.

- Click **Create Grid**.

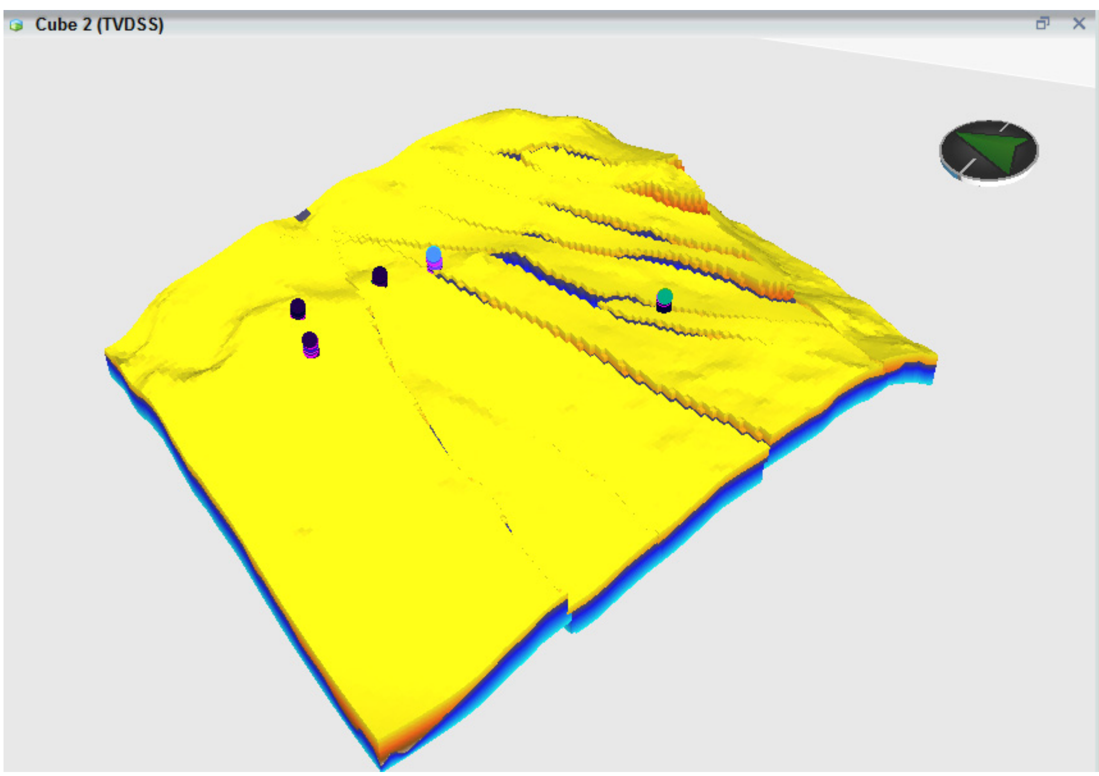


6. On the *Message* dialog, click **OK**.

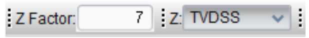
7. Activate and maximize *Cube* view



8. On the *Grid Geometry* panel, toggle off **Show grid bounding box**.

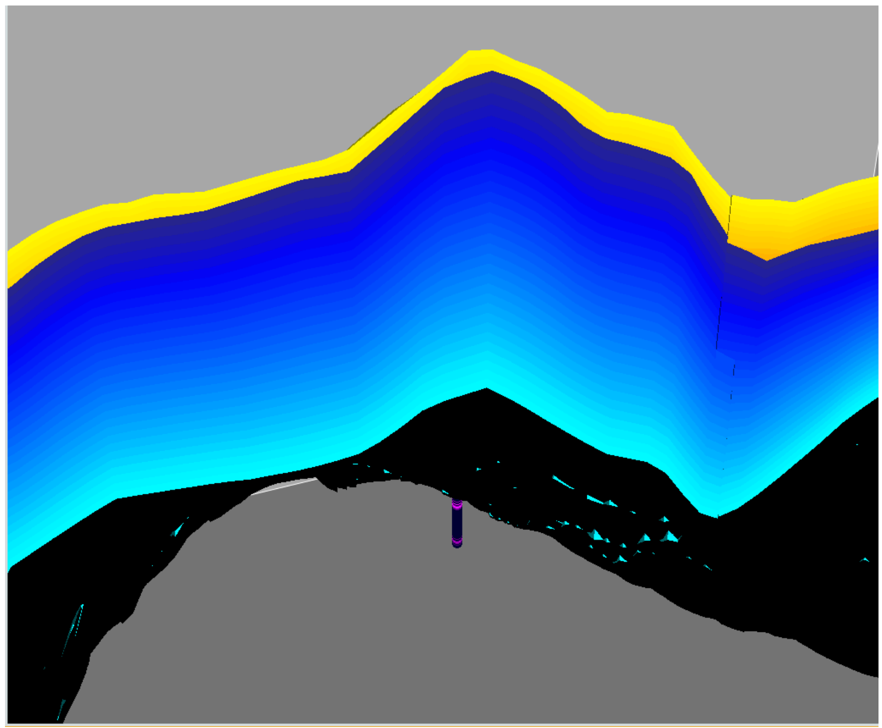


9. In the Z-Factor field, enter **"7"**.

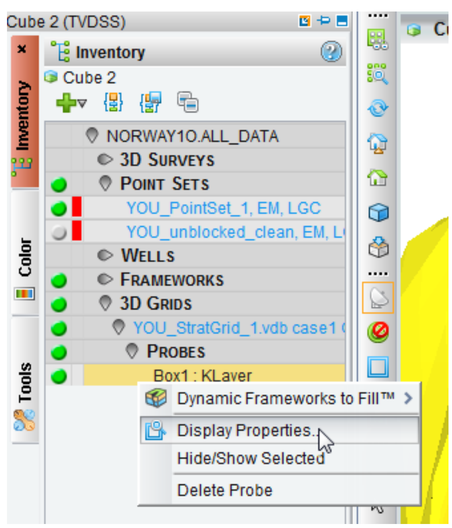


The following exercise will define lithotypes from the facies codes.

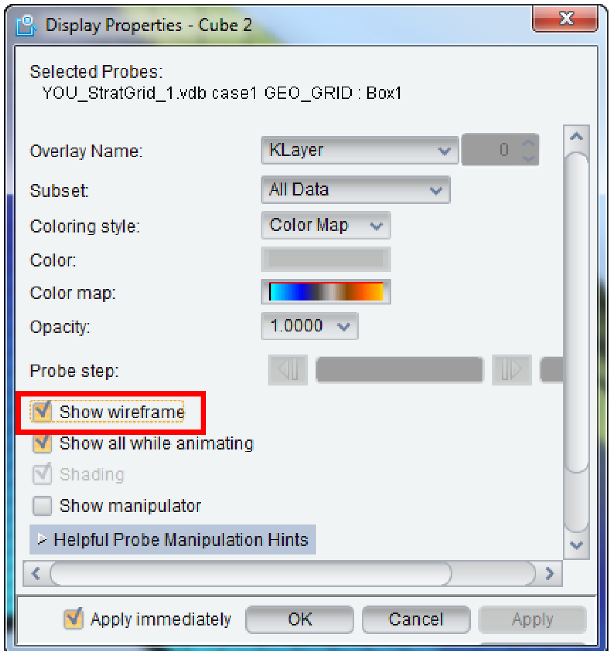
10. Zoom in on your display so you can clearly see the grid layers. Your display should look similar to that shown below.

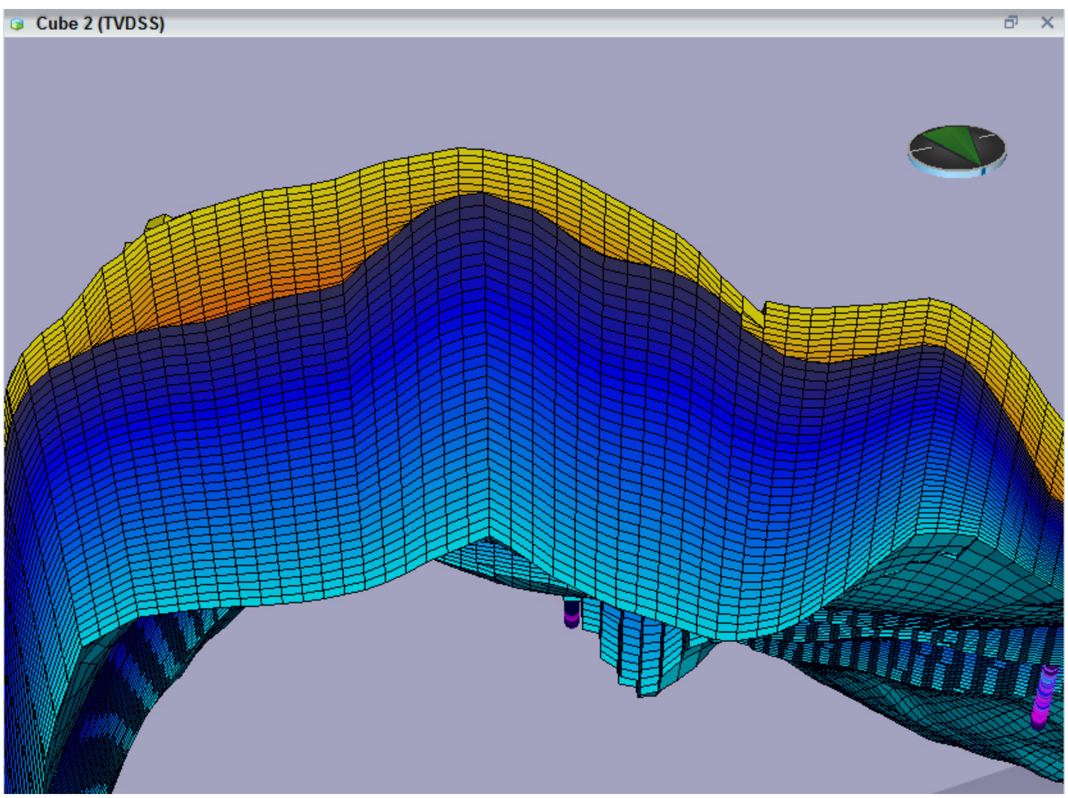


11. In the *Inventory* task pane, put your cursor over 3D Grids Prob **Box1: KLayer** and **MB3 > Display Properties**.



12. In the *Display Properties* dialog, toggle on **Show wireframe**, then click **OK**.





Overview: Facies Mapping

You have an empty geocellular grid. Now you will assign lithotypes to the facies codes in your point set.

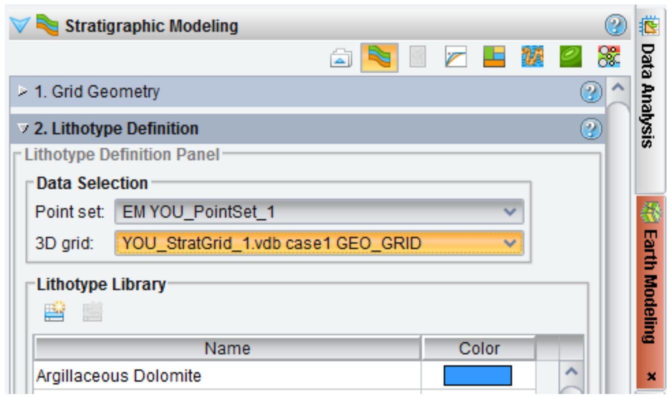
Facies are discrete properties coded by integers. They are assigned integer codes using other applications such as Landmark's StratWorks or PetroWorks® software, or they can be user-defined. To proceed with this step, you need to know the facies types and codes in your data. The Lithotype Definition step is for selecting or creating lithotypes and then to assign the facies codes in your point set to lithotype names, which you select from the Lithotype Library or create in the Active Lithotypes table. A lithotype must be assigned to all facies for the Apply button to become activated.

When a facies code is assigned to a lithotype, it is automatically assigned to that lithotype for all intervals that contain the facies code; for example, if the lithotype Sand is assigned to Facies Code 1, all intervals containing Facies Code 1 will have Sand assigned to it. A maximum of seven lithotypes can be assigned to the facies within an interval; however, if there are multiple intervals with different facies codes, you may have more than seven lithotypes assigned between all the intervals. You can assign a lithotype to multiple facies to group the facies into a common lithotype to be used for the facies modeling.

In the *Earth Modeling* task pane, you completed Step 1 (Grid Geometry) of your stratigraphic modeling. You are now moving onto Step 2 (Lithotype Definition).

Exercise 7.5: Defining Lithotype

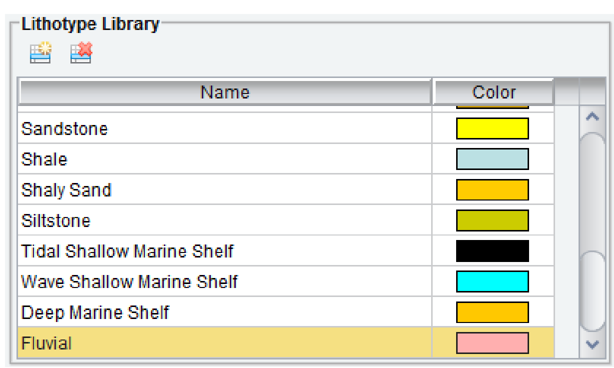
1. In the *Stratigraphic Modeling* panel of the *Earth Modeling* task pane, expand the *Lithotype Definition* panel. Select Point set: "**YOU_point set_1**" and the 3D grid: "**YOU_StratGrid_1**".



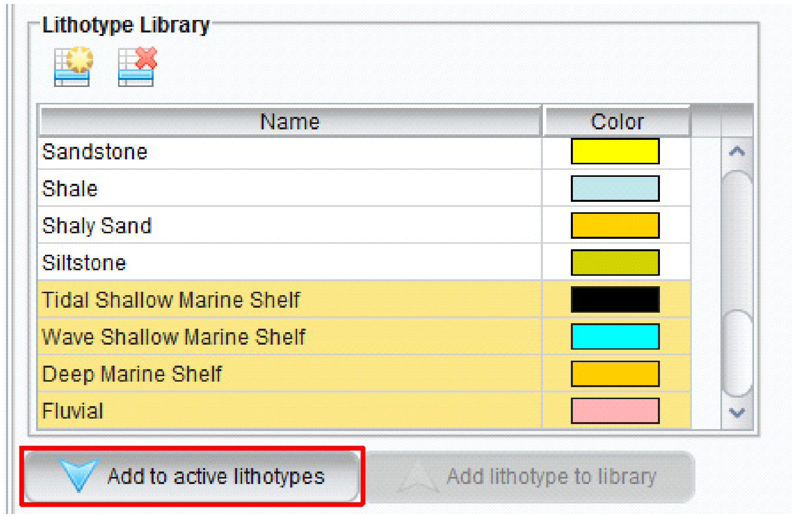
1. In the *Lithotype Library* panel, click the **Add a new Lithotype to your Library** ( ) icon, four times.

2. Enter the following Names, one for each of the new lithotypes.

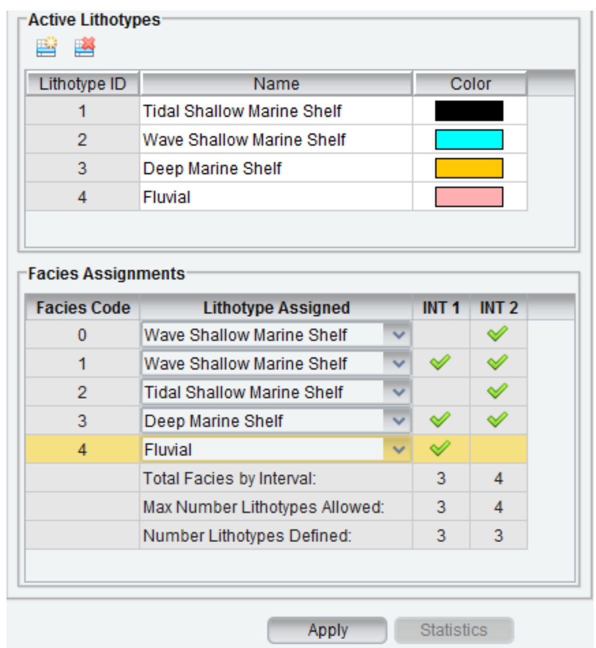
   - Tidal Shallow Marine Shelf
   - Wave Shallow Marine Shelf
   - Deep Marine Shelf
   - Fluvial



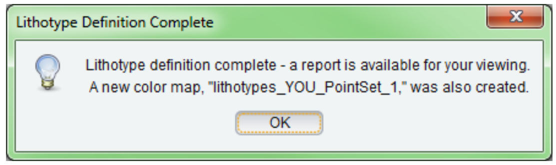
3. Select the four newly created lithologies, then click the **Add to active lithotypes** icon.



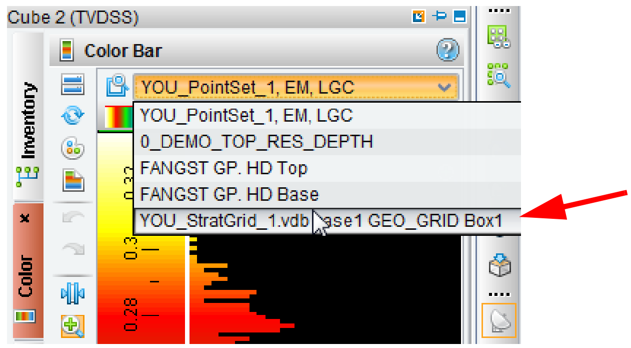
4. In the *Facies Assignments* panel of the *Active Lithotypes* dialog, make the following Lithotype Assignments for each Facies Code. Click **Apply**. In the *Lithotype Definition Complete* dialog, click **OK**.



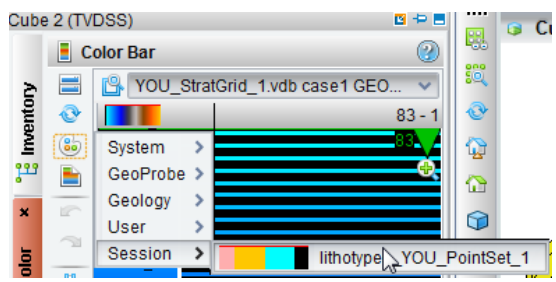
5. The **Lithotype Definition Complete** message box is displayed. Click **OK**.



6. A new color map has been created. In the *Color Bar* panel of the *Color* task pane, select **YOU_StratGrid_1** on the pull-down menu.



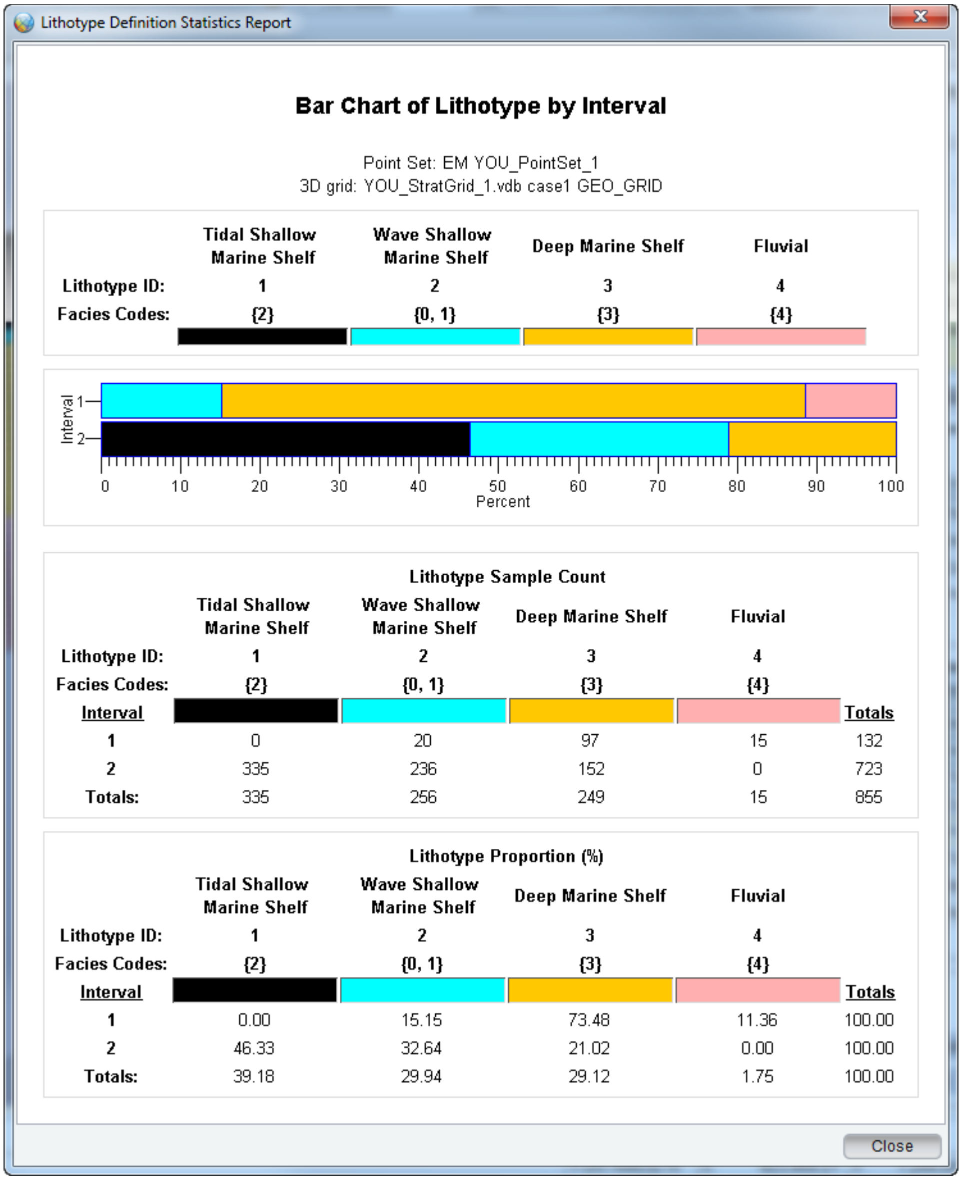
7. Click the **Select Color Bar** () icon, then on the pull-down menu, select **Session > lithotypes_YOU_PointSet_1**.



8. If this point set is used to simulate new lithotype properties on the 3D grid, you should use this same color map to display the grid property in the 3D Grid Probe. On the *Create 3D Grid Probe* panel, this new color map will appear in the Color map pull-down menu.

9. In the *Stratigraphic Modeling* panel of the *Earth Modeling* task pane, expand the *Lithotype Definition* panel. Click the **Statistics** (Statistics...) button to open the report with the lithotype mapping

information. You will see the percentage of total samples for each facies code and each lithotype.



Lithotype Definition Statistics Report

Bar Chart of Lithotype by Interval

Point Set: EM YOU_PointSet_1
3D grid: YOU_StratGrid_1.vdb case1 GEO_GRID

	Tidal Shallow Marine Shelf	Wave Shallow Marine Shelf	Deep Marine Shelf	Fluvial
Lithotype ID:	1	2	3	4
Facies Codes:	{2}	{0, 1}	{3}	{4}

Lithotype Sample Count

	Tidal Shallow Marine Shelf	Wave Shallow Marine Shelf	Deep Marine Shelf	Fluvial	
Lithotype ID:	1	2	3	4	
Facies Codes:	{2}	{0, 1}	{3}	{4}	
Interval					Totals
1	0	20	97	15	132
2	335	236	152	0	723
Totals:	335	256	249	15	855

Lithotype Proportion (%)

	Tidal Shallow Marine Shelf	Wave Shallow Marine Shelf	Deep Marine Shelf	Fluvial	
Lithotype ID:	1	2	3	4	
Facies Codes:	{2}	{0, 1}	{3}	{4}	
Interval					Totals
1	0.00	15.15	73.48	11.36	100.00
2	46.33	32.64	21.02	0.00	100.00
Totals:	39.18	29.94	29.12	1.75	100.00

Close

10. Click **Close**.

Exercise 7.6: Seismic Attribute Blocking

Seismic Attribute Blocking is used to block a 3D seismic attribute (e.g., seismic acoustic impedance) volume onto a grid. The seismic attribute volume must be compatible with the imported geomodel that you are using; i.e., it must be from the same survey used to interpret the horizons and faults used to construct the sealed framework.

There are two Blocking Methods:

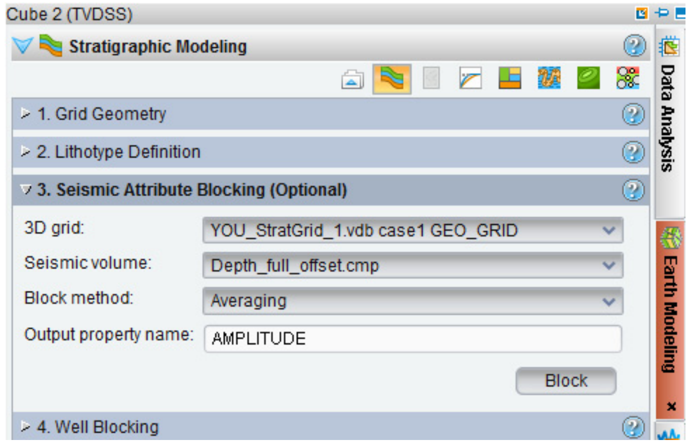
- **Averaging —** iterates through each cell, calculates the arithmetic average of all seismic attribute values that fall within the cell, and assigns the average value to the cell center.

- **Nearest Point —** iterates through each cell, finds the center of the cell, and assigns the seismic attribute value that is closest to the cell center.

By default the seismic attribute name is used as *Output Property Name.* However, you can click in the text field to edit the name, as desired. The output is a property or properties on the grid. To analyze the blocked seismic attributes, select the blocked seismic attributes and view the *Data Analysis Tools.* The output grid with the blocked seismic attributes can be used in Well Blocking. The seismic attributes are mapped to the well locations and that output point set can then be used in collocated co-kriging or co-simulation.

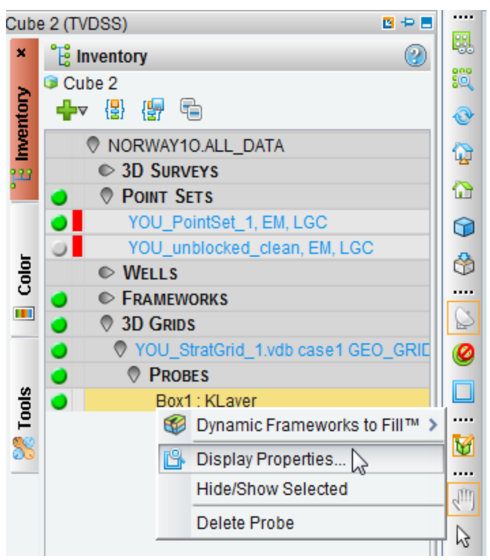
1. In the *Stratigraphic Modeling* panel of the *Earth Modeling* task pane, expand the *Seismic Attribute Blocking (Optional)* panel. 3D grid:, Seismic volume:, and Block method: pull-down menus are

populated. In the Output property name: field, enter **"AMPLITUDE"**, then click **Block**. In the *Message* dialog, click **OK**.

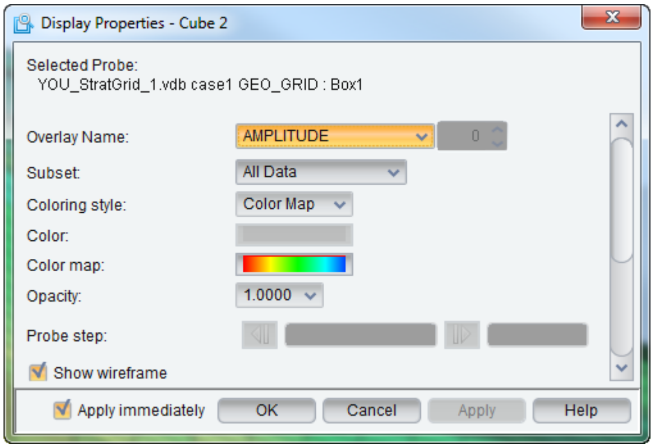


> **Note:**
>
> Only Depth domain seismic data is supported at this time.

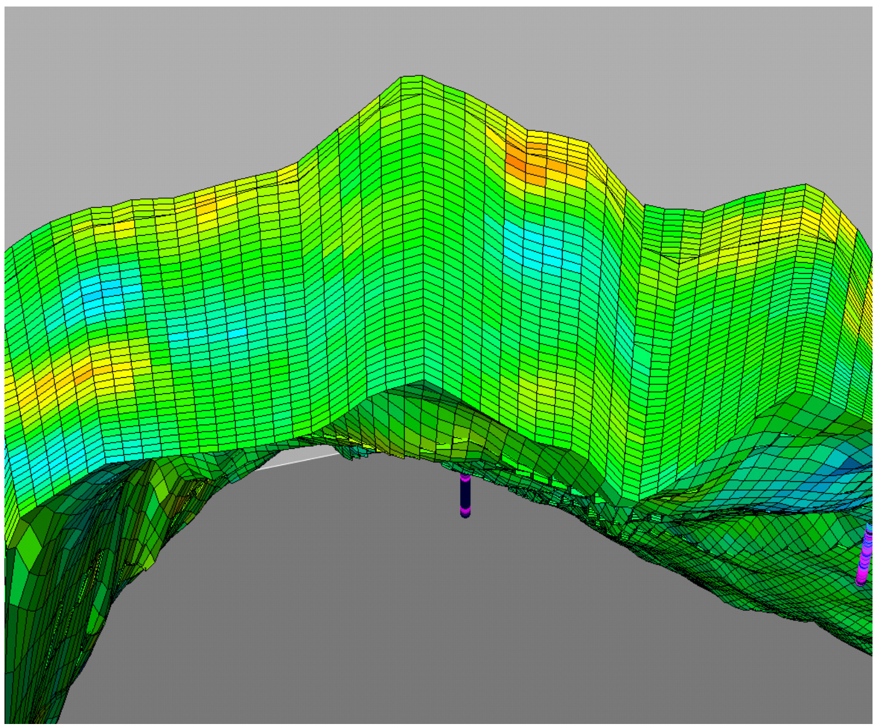
2. In the *Inventory* task pane of the *Cube* view, put your cursor on **Probe Box1: KLayer** and **MB3 > Display Properties**.



3. In the Overlay Name: pull-down menu of the *Display Properties* dialog, select **AMPLITUDE**. Click **OK**.



Your view should look like the view in the following image.



In the next exercise, you will block the well logs from the point set onto the 3D grid and create a new blocked point set.

Overview: Well Blocking

Well Blocking allows you to specify the point set for which you want to block logs, the grid to which the blocked values will be assigned, and the output blocked point set name. To obtain continuous properties, you can select the averaging method, which assigns a value to each grid cell (layer) along the wellbore. Using the selected averaging method, a value is calculated for each property in each cell of the grid by the input point set with a lithology bias. The value is assigned to the gravity center of the cell.

The averaging method chosen is based on the property. For some properties, such as Facies, which are discrete properties coded by integers, the method is set as MOST_OF and uses the most commonly occurring facies code. For continuous properties, you can select from the following options:

- **MOST_OF** — uses the most commonly occurring value (mode). This is the only option for facies codes.
- **ARITHMETIC** — uses the arithmetic mean or average value. This method is typically used for properties such as porosity, saturation, and net to gross, because these are additive values. Because the arithmetic mean is greatly influenced by outliers, this method may not be appropriate for describing skewed distributions. If your data does not vary much, this method can be used to perform a very fast blocking.
- **MINIMUM** — uses the minimum value.
- **MAXIMUM** — uses the maximum value.
- **MID_POINT** — uses the value that falls nearest the gravity center of the grid cell.
- **RANDOM** — selects a value from the continuous property at random.
- **GEOMETRIC** — uses the geometric mean, which, for a set *n*, is the *nth* root of the product of all values. For example, the geometric mean of numbers 2 and 8 is the square root of their product, 16, which is 4. This method is useful for sets of positive numbers that are interpreted according to their product and not their sum, or are exponential. Because it is sensitive to lower

values, this method is normally a good estimate for permeability if it is log-normal distributed.

- **HARMONIC** — uses the harmonic mean, which is the number of variables divided by the sum of the reciprocals of the variables. This is useful for sets of numbers that are defined in relation to some unit. In situations involving rates and ratios, the harmonic mean provides the truest average. Because the harmonic mean of a list of numbers tends strongly toward the least elements of the list, compared to the arithmetic mean, it tends to mitigate the impact of large outliers and aggravate the impact of small ones. Because this method is sensitive to lower values, it is often used with permeability, and gives the effective vertical permeability if the reservoir is layered with constant permeability in each layer. It also works well with log-normal distributions.

By use of a selected method, **Block** assigns a single value for each property in the point set to each cell center in the grid, and outputs a new point set.

Statistics opens a *Well Blocking Report* window with detailed information on the well blocking for each interval and lithotype. The *Proportion Curves* panel at the bottom of the report displays a global proportion curve for each interval; or if there is only one interval, for All Data. This allows you to determine if your layering style is correct before proceeding to Facies Modeling.

Well Blocking automatically creates a lithotype proportion set (LPS), which is included in the naming format. If a *Map* view is active when Well Blocking is done, lithotype proportion curves for All Data, or Interval1 if there are multiple intervals, which is included in the naming format, are automatically displayed in the *Map* view, at the well locations. The well name is displayed next to the proportion curve. The global proportion curve, which is also shown in the *Well Blocking Report*, is displayed in the lower right corner of the *Map* view along with a legend of the lithotypes.

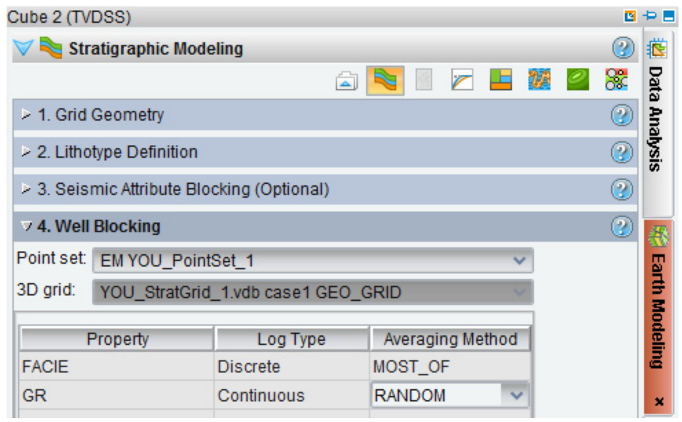
If a *Map* view is not active when you do Well Blocking, the proportion curve set will be automatically displayed when you activate a *Map* view and toggle on **Lithotype Proportion Map Creation** in the *Inventory* task pane.

A *Lithotype Proportions* tab is added to the *Select Session Data* dialog listing the lithotype proportion set, and **Lithotype Proportions** is added in your *Inventory*.

A user color map for the lithotype colors in the blocked point set is also created using the format “lithotypes_[point set_name]”. Displaying a point set using this color map will show where some lithotypes have been blocked away. If this blocked point set is used to simulate new lithotype properties on the 3D grid, you should use this same color map to display the grid property in the grid probe. This new color map will appear in the Color map pull-down menu on the *Display Properties* dialog. The color map is recreated on the fly whenever the selected point set is loaded.

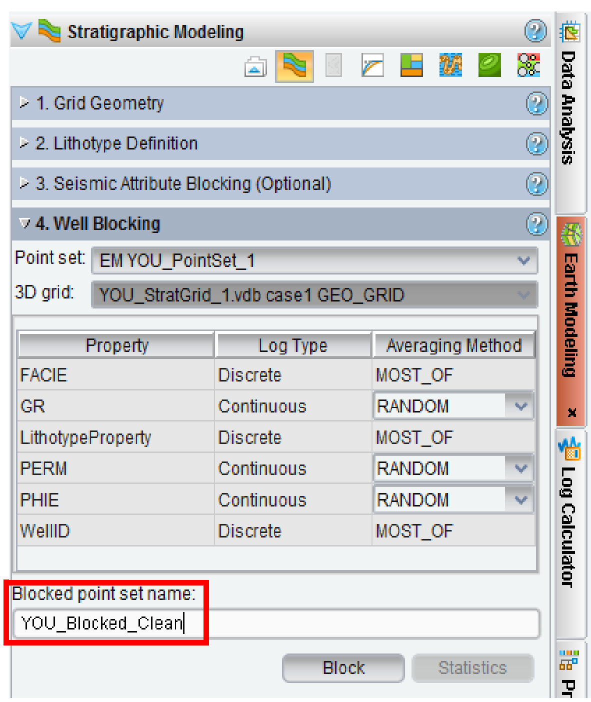
Exercise 7.7: Well Blocking

4. In the *Stratigraphic Modeling* panel of the *Earth Modeling* task pane, expand the *Well Blocking* panel. Select Point set: **EM YOU_PointSet_1**.

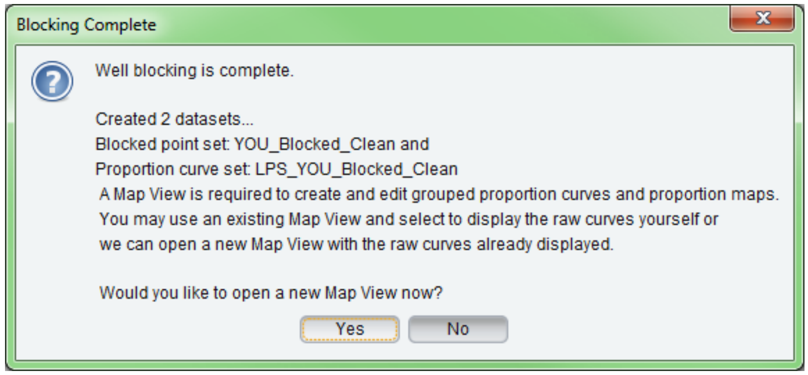


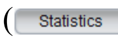
The associated grid is automatically selected. The properties in the grid, with their Log Type and a default Averaging Method, are displayed in the table.

5. In the Blocked point set: field, enter "**YOU_Blocked_Clean".** Click **Block.**



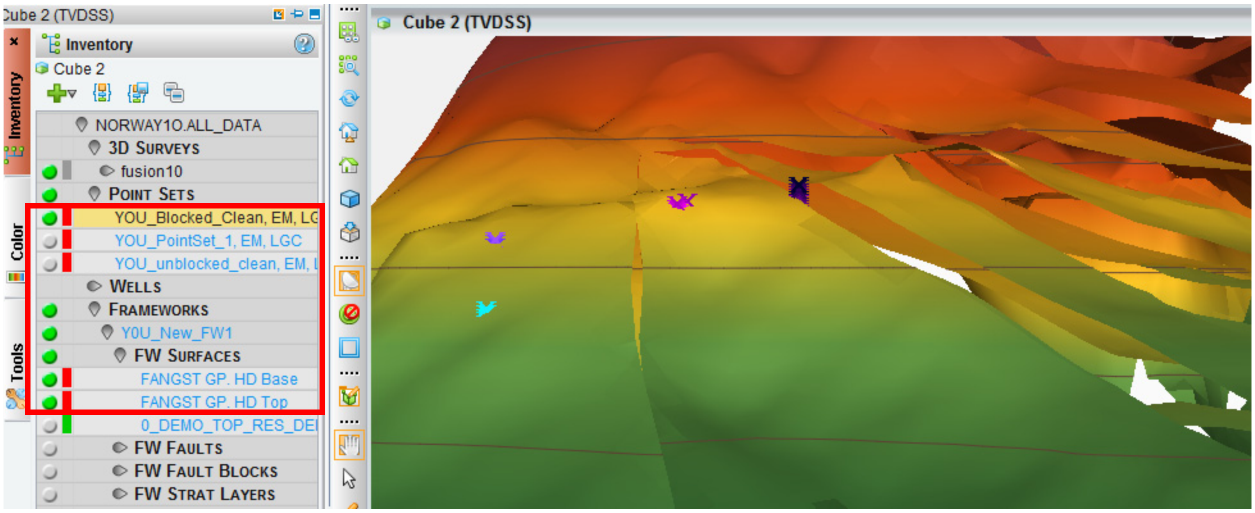
6. In the *Blocking Complete* dialog, click **No.** This message indicates that two data sets were created: the Blocked point set and a Proportion curve set.



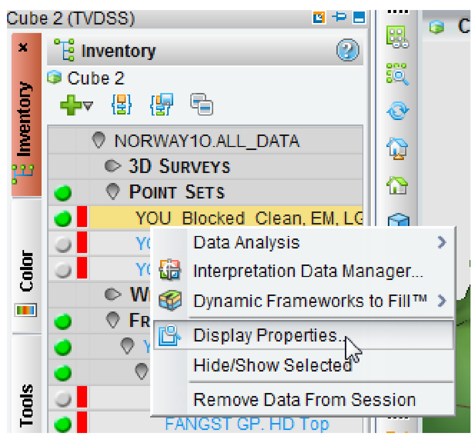
7. In the Well Blocking panel, click the Statistics () button. You will see the number of input and output sample points listed along with information showing the original percentage of each lithotype and the blocked percentage for each interval and

lithotype. The *Proportion Curves* panel displays a proportion curve for each interval. Click **Close**.

8. In the *Inventory task* pane of the *Cube* view, toggle off **all data** except Point Set YOU_Blocked_Clean and FW Surface FANGST GP.HD Top and FANGST GP.HD Base.

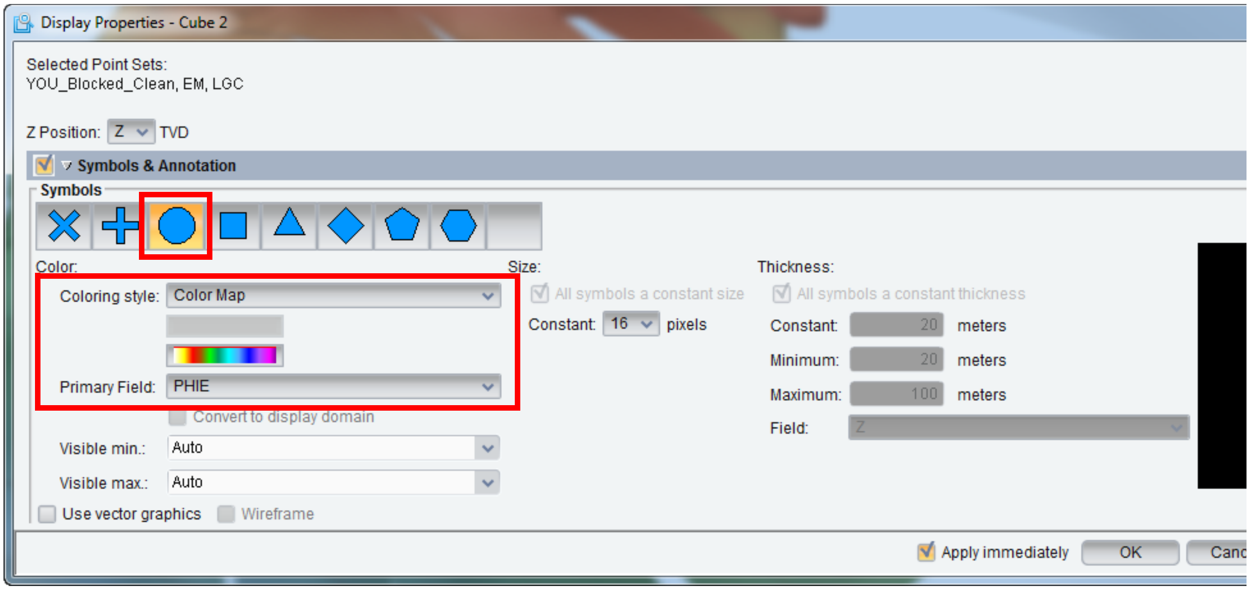


9. In the *Inventory,* put your cursor on Point Set **YOU_Blocked_Clean** and **MB3 > Display Properties**.

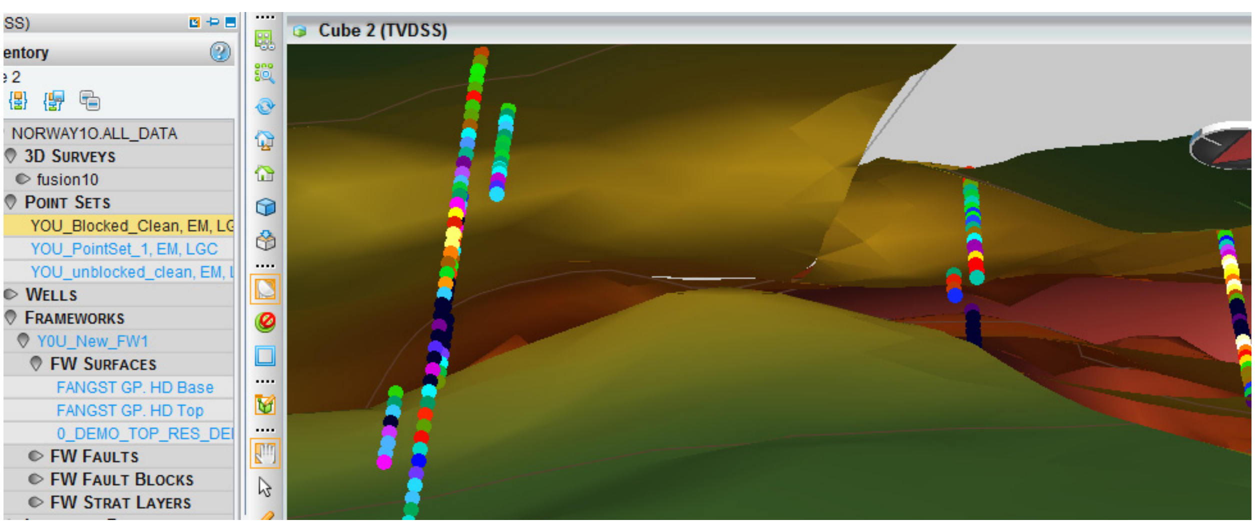


10. In the *Display Properties* dialog, expand the *Symbols & Annotation* panel and select the **circle shape**. On the Coloring style: pull-down

menu, select **Color Map**. On the color system select **0_Spectrum**, and in the Primary Field: pull-down menu, select **PHIE**. Click **OK**.



11. Zoom in on your display so you can see the blocked point set between the top and bottom surfaces. Some of the blocked wells extend from the top to the bottom surfaces and others do not, as is shown below.



The next step is to use the lithotype proportion set, which is created in Well Blocking, to output a lithotype proportion map for use in Facies Modeling.

Overview: Earth Modeling: Lithotype Proportion Map Creation

The *Lithotype Proportion Map Creation* task pane allows you to create grouped proportion curves, apply smoothing to grouped lithotype proportions, and create a lithotype proportion map that will have the same lithotype inputs in Facies Modeling and Simulation. The input is a point set of the blocked facies logs. The logs are grouped using polygons. The lithotype proportion map controls the stratigraphic relationship of the lithotypes.

For each polygon in each interval, a lithotype proportion curve is computed (recomputed proportion curves) and gridded onto the 3D grid at the center of gravity of the cell, with averaged proportion curves placed at the gravity center of the polygon. This accounts for any trends in lithotype across the area both vertically and horizontally. A lithotype proportion map is also created and output to the 3D grid. In the proportion map, a single value is assigned to each cell, using the highest probability for each lithotype.

The *Lithotype Proportion Map Creation* task pane contains three panels:

- *Create Grouped Proportion Curves* panel — enables you to group the proportion curves in the lithotype proportion set created in Well Blocking.
- *Smoothing* panel — enables you to apply smoothing to a selected lithotype proportions group.
- *Proportion Map Creation* panel — enables you to create a lithotype proportion map for the selected lithotype proportion group.

To create grouped proportion curves, you can use regular grouping for all intervals or custom polygons, by interval. If using regular grouping, you only need to do this on one interval, as the polygons are the same for all intervals. If you are using custom polygons, you must create polygons for each interval.

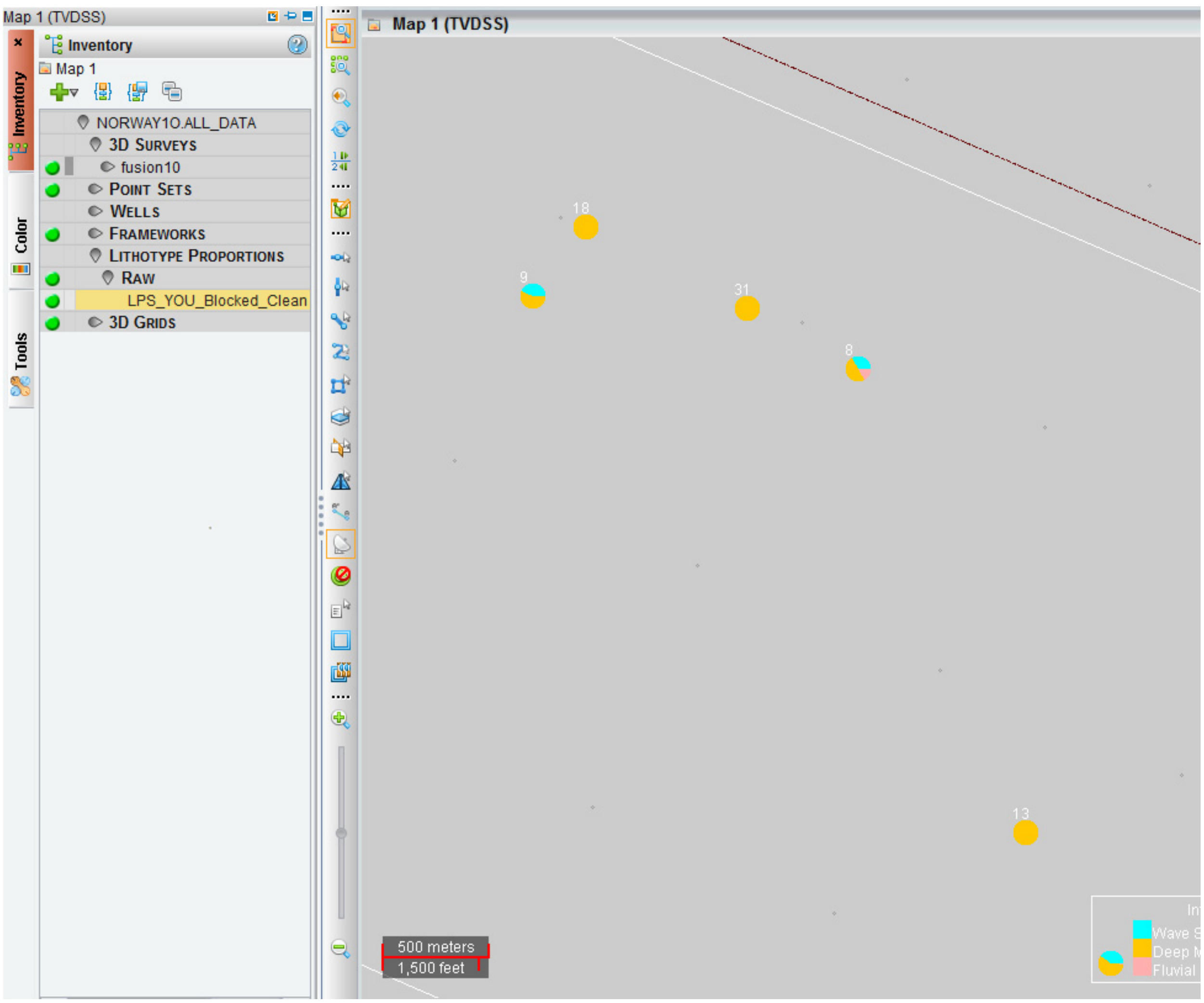
In the steps below, you will use regular grouping for all intervals to compute grouped proportion curves and then create a lithotype proportion map

The *Lithotype Proportion Map Creation* panel can be used only when a *Map* view is active.

Exercise 7.8: Creating Lithotype Proportion Maps

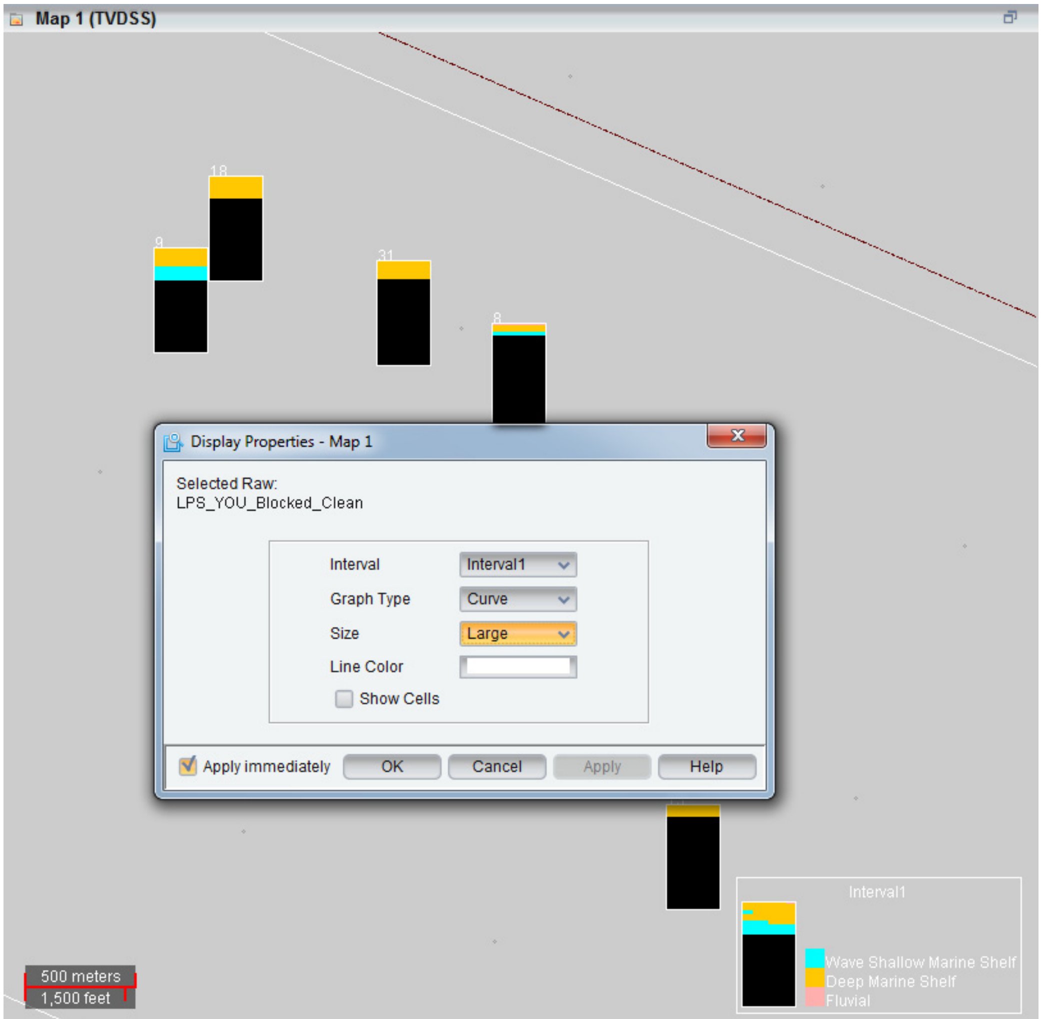
1. In the *Inventory* task pane of the *Map* view, toggle off **all data**.

2. In the *Inventory,* toggle on **Lithotype Proportions LPS_YOU_Blocked_Clean**. This is the lithotype proportion set that was created in *Well Blocking*. The lithotype proportions are displayed as pie charts in the *Map* view (shown below). Well numbers are displayed with each pie chart. A pie chart of the global proportions is shown in the lower right, with a legend showing the Interval number and the lithotypes.

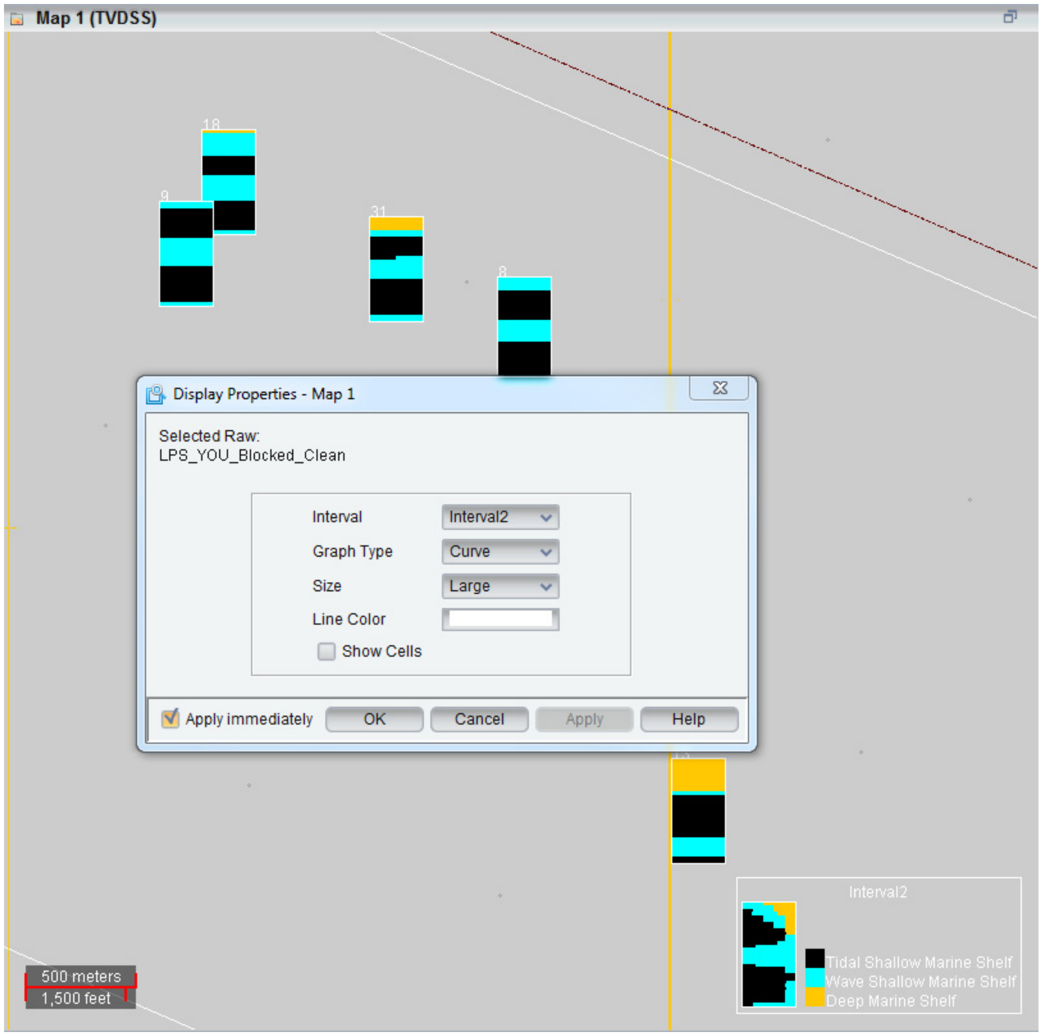


3. With your cursor on a pie chart in *Map* view, **MB3 > Display Properties**. In the Interval pull-down menu of the *Display Properties* dialog, select **Interval2**. In the Graph Type pull-

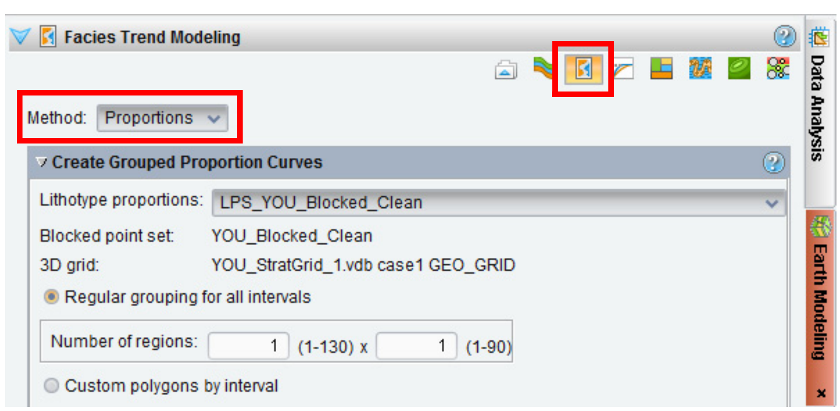
down menu, select **Curve,** and in the Size pull-down menu, select **Large**.



4. Click **OK**.



5. In the *Earth Modeling* task pane, select the **Facies Trend Modeling** ( ) icon. In the Method: pull-down menu, select **Proportions**.

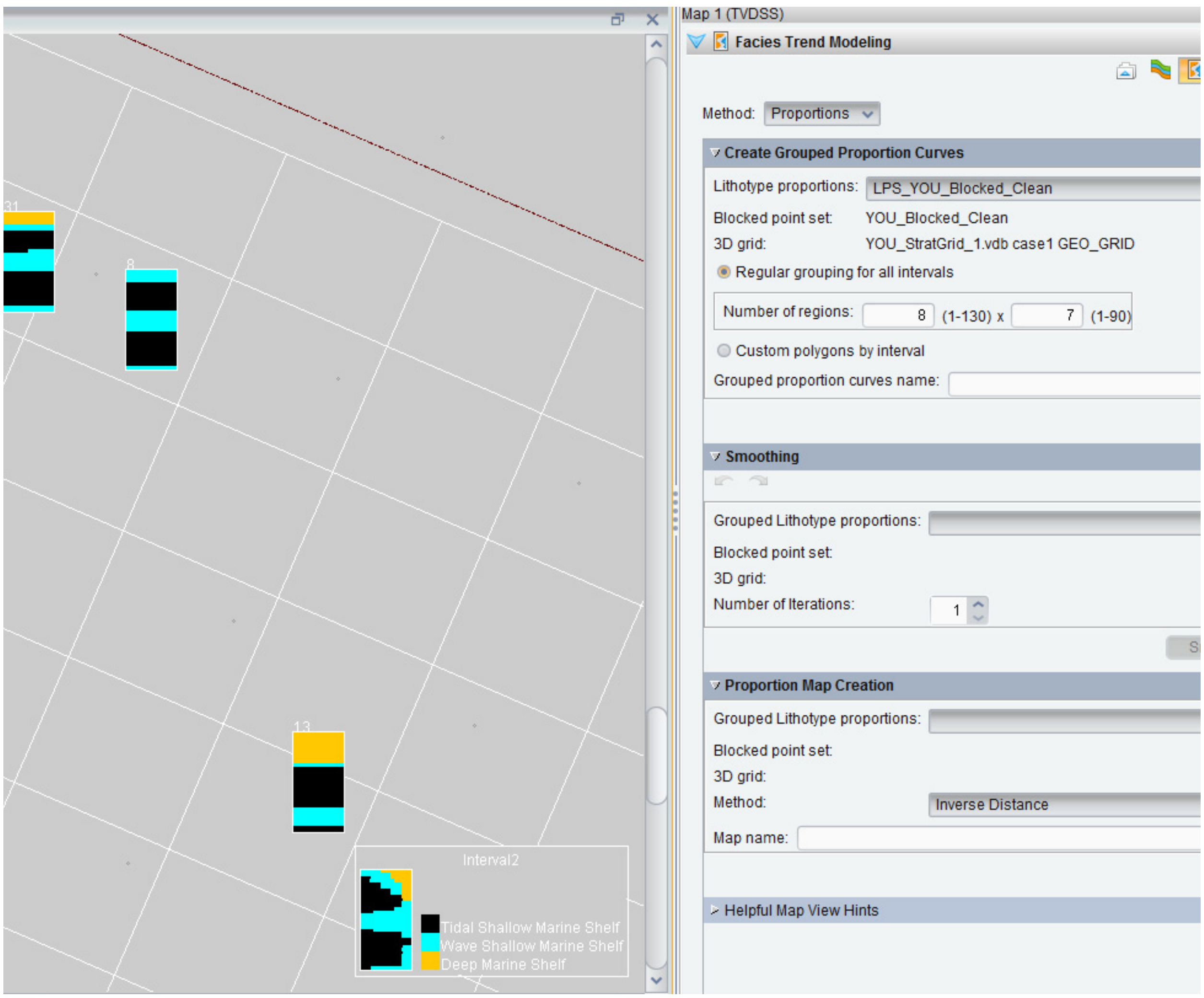


The *Create Grouped Proportion Curves* panel opens by default. The Lithotype proportions that were automatically generated in *Well Blocking* are listed with the Blocked point set and the Grid.

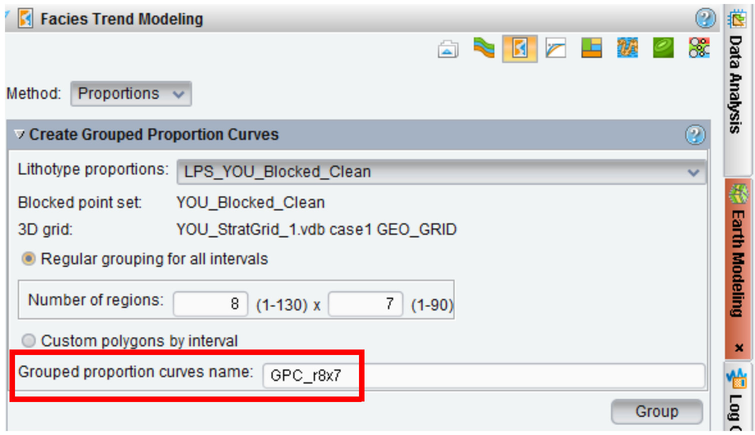
6. In the *Create Grouped Proportion Curves* panel, toggle on **Regular grouping for all intervals** and enter Number of Regions, **"8"** and **"7"**. You are using this number of regions so there is basically one well per cell.

You are not using custom polygons here because they need to be designed for each interval. The instructor will demonstrate how to create a custom polygon and copying, moving, and deleting proportion curves.

As you specify the number of regions, lines are added to the lithotype proportion set in the *Map* view, showing how the regions will be created.



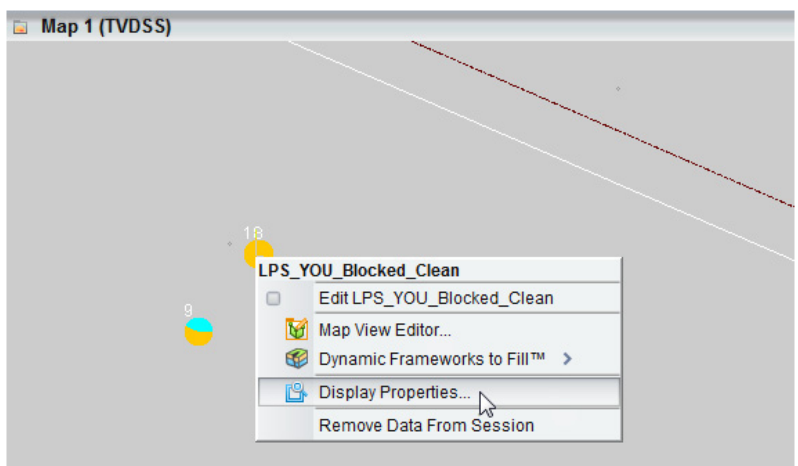
7. In the Grouped proportion curves name: field, enter "**GPC_r8x7**". Click **Group**.



8. In the *Grouping Proportion Curves* dialog, click **OK**.

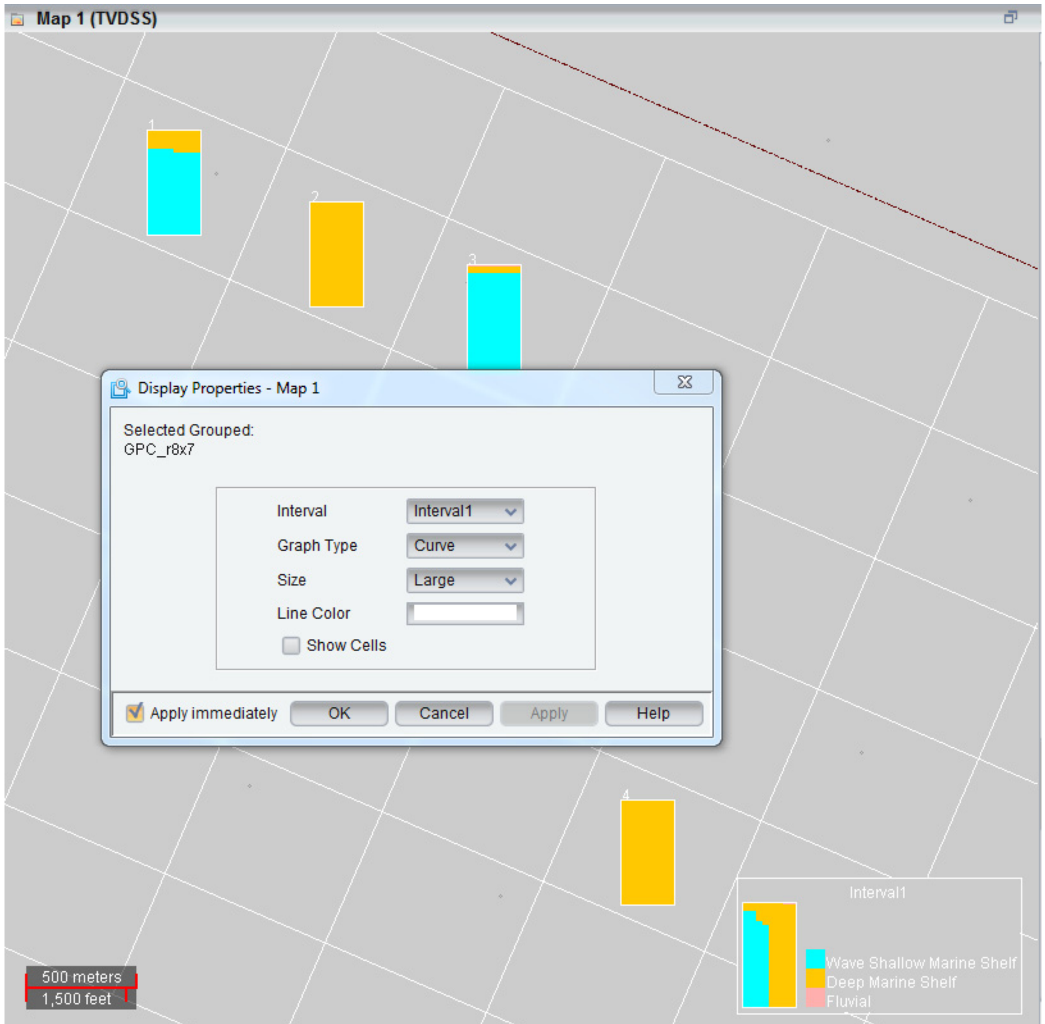
The lithotype proportion map for Interval1 is displayed, using default pie charts.

9. In *Map* view, put your cursor on one of the GPC_r8x7 pie charts and **MB3 > Display Properties**.



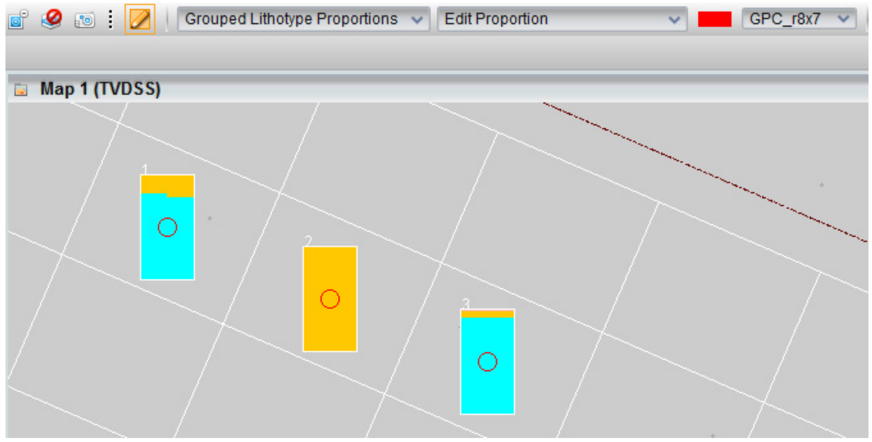
10. In the *Display Properties* dialog, select **Interval1**. In the Graph Type: pull-down menu, select **Curve** and in the Size: pull-down menu, select **Large**.

11. In the *Display Properties* dialog, select **Interval1**. Click **OK**.

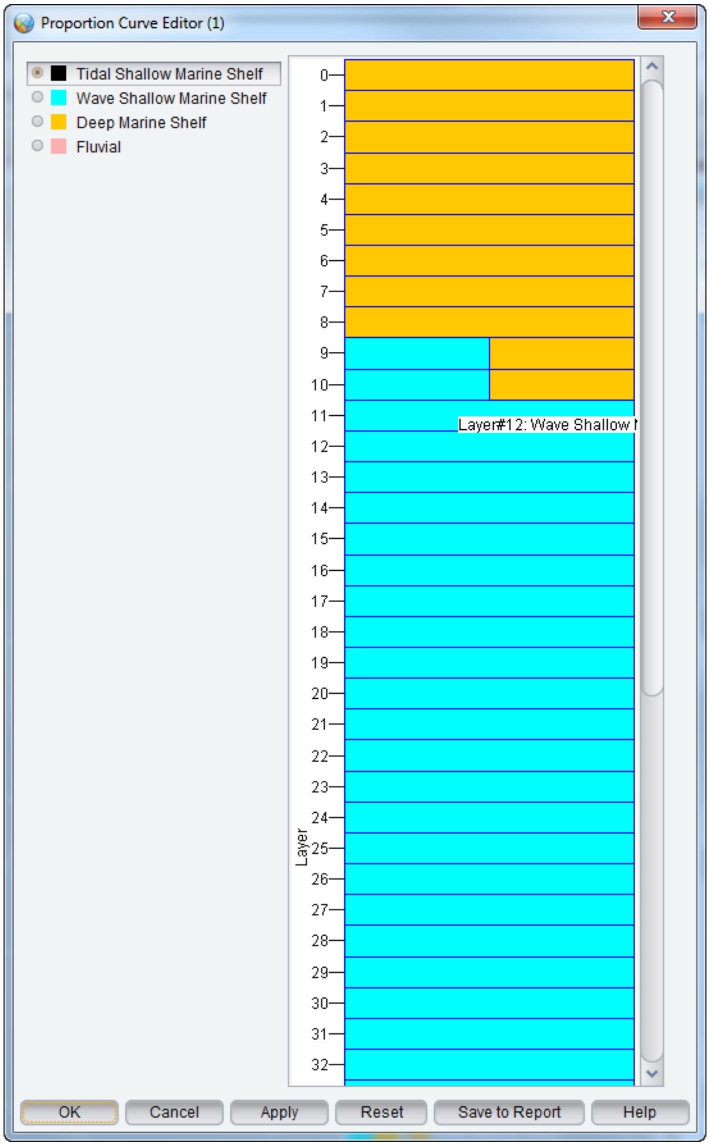


12. In the Interpretation toolbar, toggle on **Interpretation Mode**, select Interpretation data type, **Grouped Lithotype Proportions**, and select Interpretation action, **Edit Proportion**.

You will now see red circles inside each curve chart.



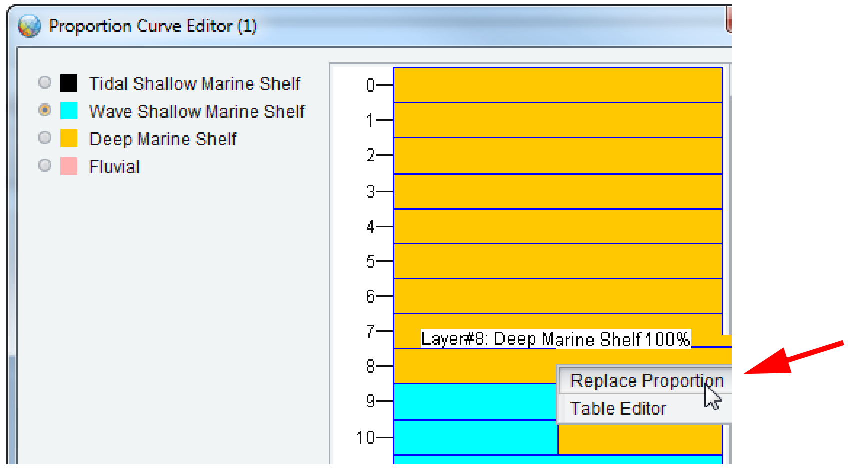
13. In *Map* view, double-click the **red circle** for Curve Chart 1.



The graph on the right indicates the percentage of the lithotypes in each layer. If you hover your cursor over a lithotype color in the graph, the text pop-up shows the layer number and the percentage of that lithotype in that specific layer.

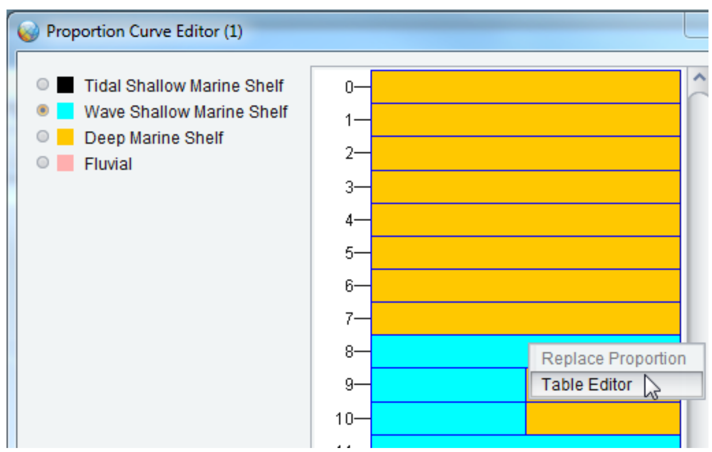
Put your cursor over Layer 8 and note that it contains only the Deep Marine Shelf lithotype. We want to change it so that it is Wave Shallow Marine Shelf.

14. In the *Proportion Curve Editor* dialog, toggle on **Wave Shallow Marine Shelf**, then put your cursor over Layer 8 and **MB3 > Replace Proportion**.

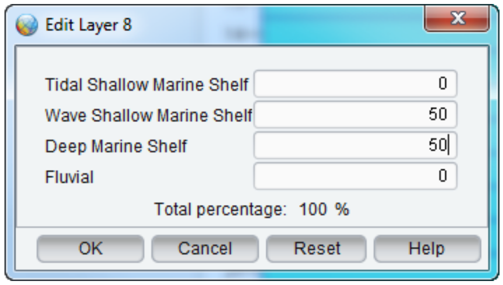


The entire layer is now colored as Wave Shallow Marine Shelf.

15. Next you will edit the same proportion curve using a different technique. In the *Proportion Curve Editor* put your cursor on **Layer 8** and **MB3 > Table Editor**.



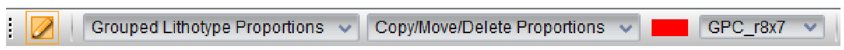
16. On the *Edit Layer 8* dialog, enter **"50"** in the Wave Shallow Marine Shelf field and enter **"50"** in the Deep Marine Shelf field. Click **OK**.



17. Click **OK** to close the editor.

You can use the same technique to change curve proportions.

18. On the Interpretation toolbar, click the Edit Proportion pull-down menu and select **Copy/Move/Delete Proportions**.

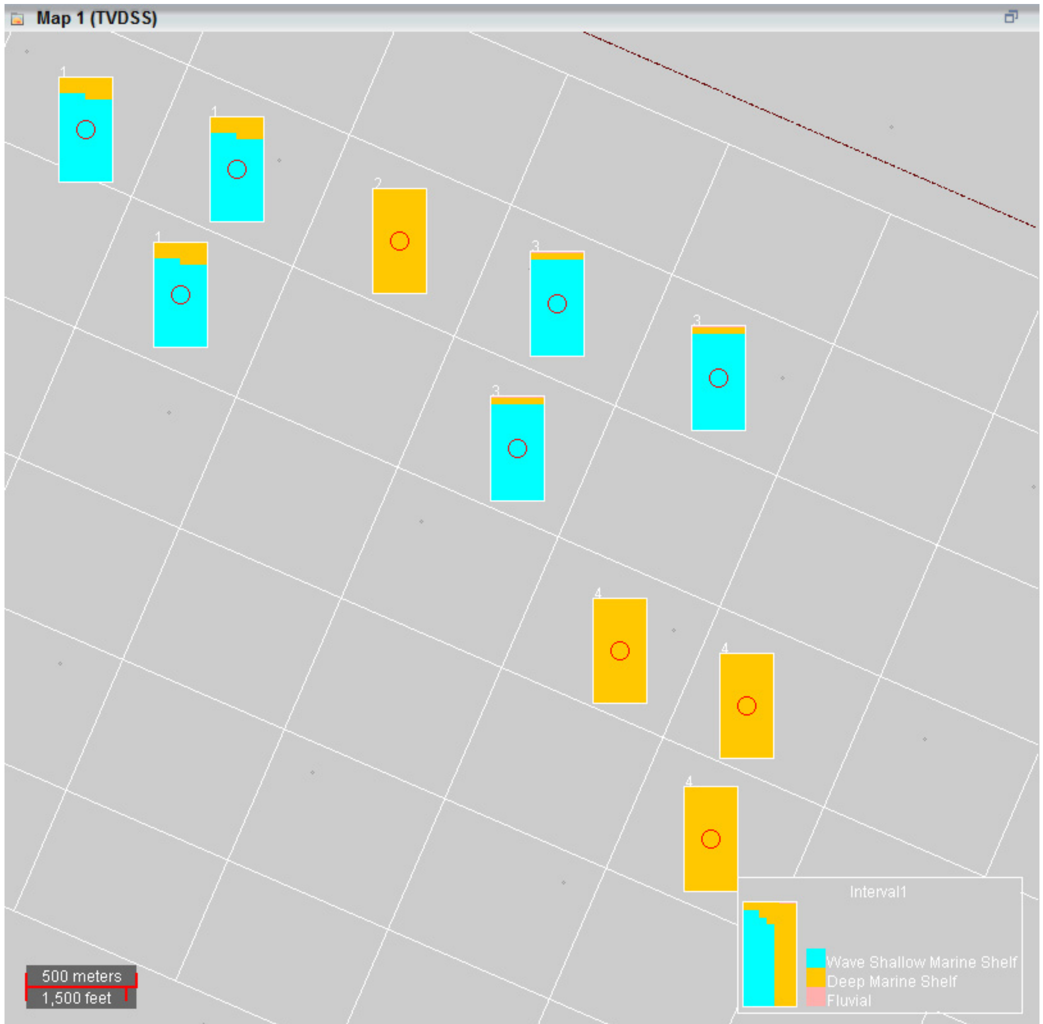


19. Click inside the red circle on curve charts and drag it to copy the curve to two or three new locations around the original curve.

Note:

When the four-headed arrow is displayed, the cursor is ready to drag-and-drop for copying. A circle will also appear, while dragging, to show where the copy is to be placed.

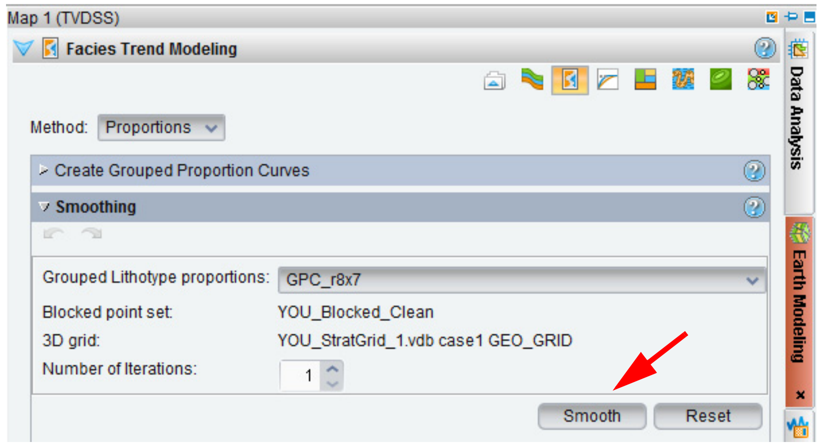
20. Mimic the following **distribution**.



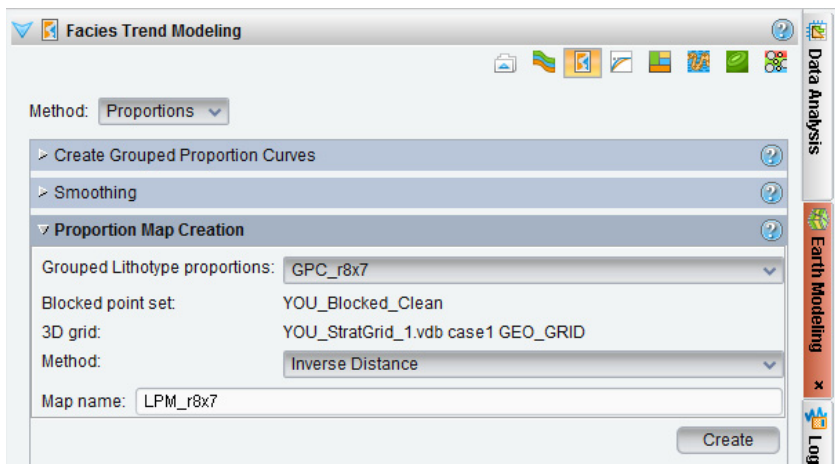
21. Toggle off the **Interpretation Mode** () icon.

22. In the *Facies Trend Modeling* panel of the *Earth Modeling* task pane, expand the *Smoothing* panel. Click **Smooth** to run one

iteration. You will see the display update itself to show the smoothed curves.

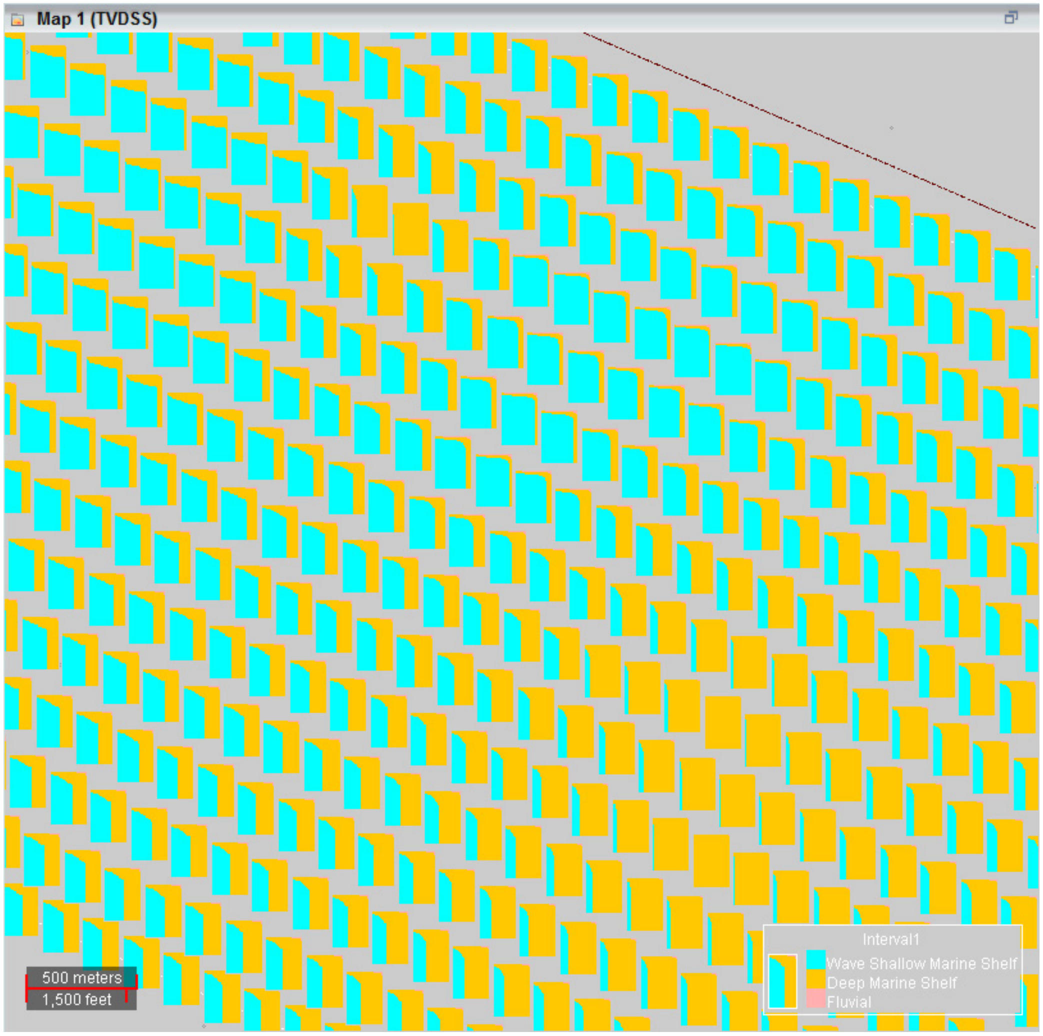


23. In the Map Name: field on the *Proportion Map Creation* panel, enter **"LPM_r8x7"**. Click **Create**.



24. In the *Computing Proportion Matrix* dialog, click **OK**.

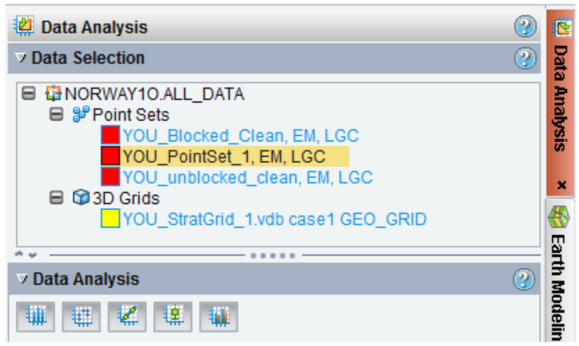
Your display will look like the image shown below.



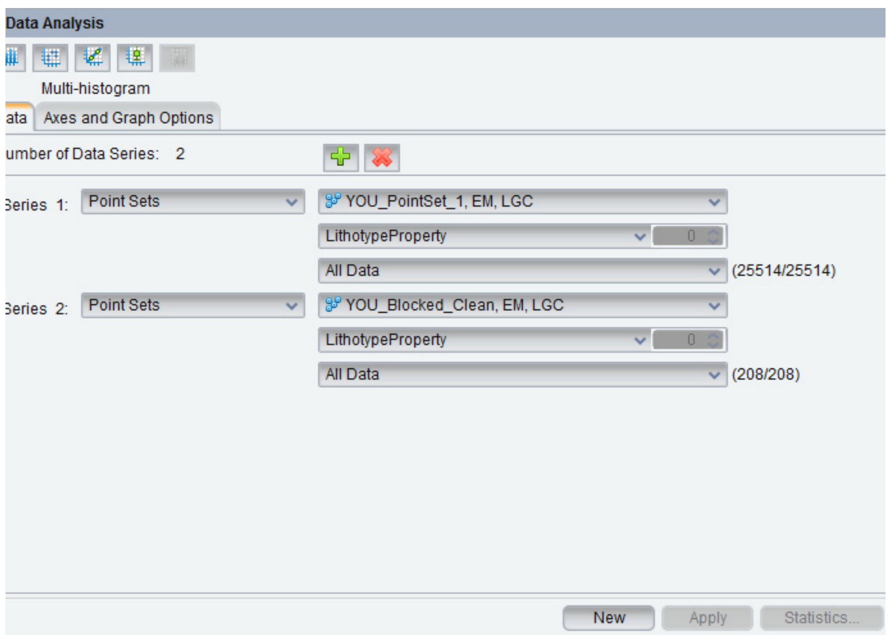
This proportion map (3D volume) will be used with the facies variograms to generate the final facies simulation. Further, you can create more lithotype proportion maps. You use them to help control the facies modeling by creating a second facies model for comparison.

Before proceeding to Facies Modeling, you can use Data Analysis to QC the well blocking by looking at a histogram to compare the lithotype properties in your original unblocked point set against that of the blocked point set.

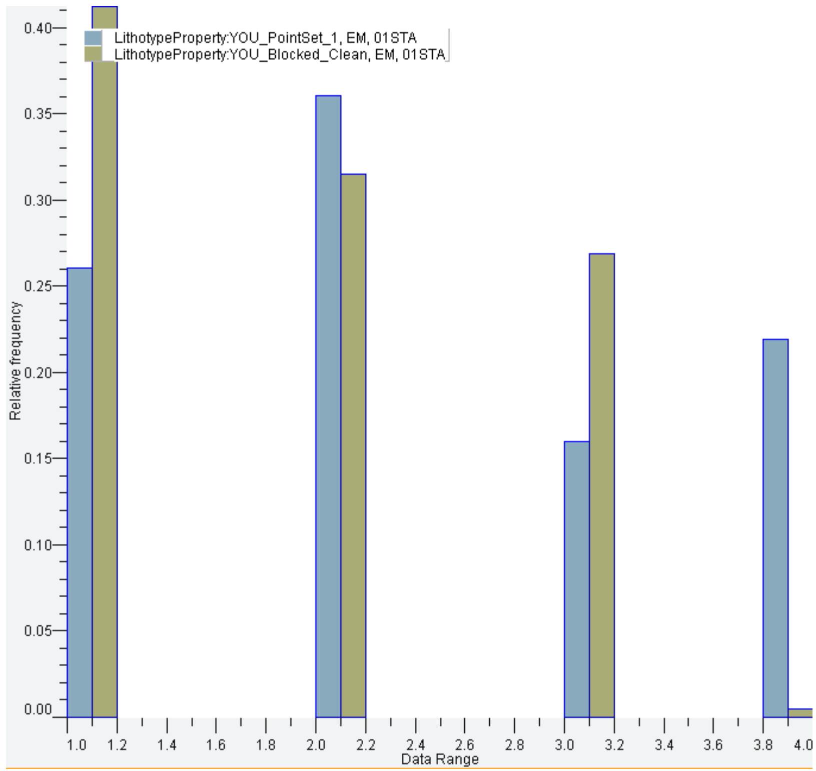
25. In the *Data Selection* panel of the *Data Analysis* task pane, select Point Set **YOU_PointSet_1**. In the *Data Analysis* panel, select in the Multi-Histogram ( ) icon.



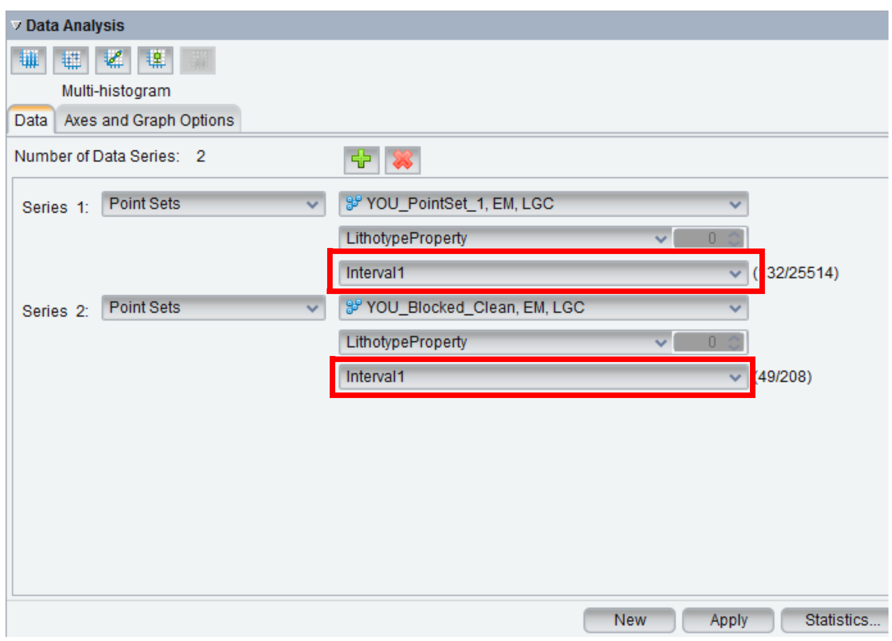
26. Click the **Add series** ( ) icon to add a second series. For Series 1 and 2, select the options as shown below.



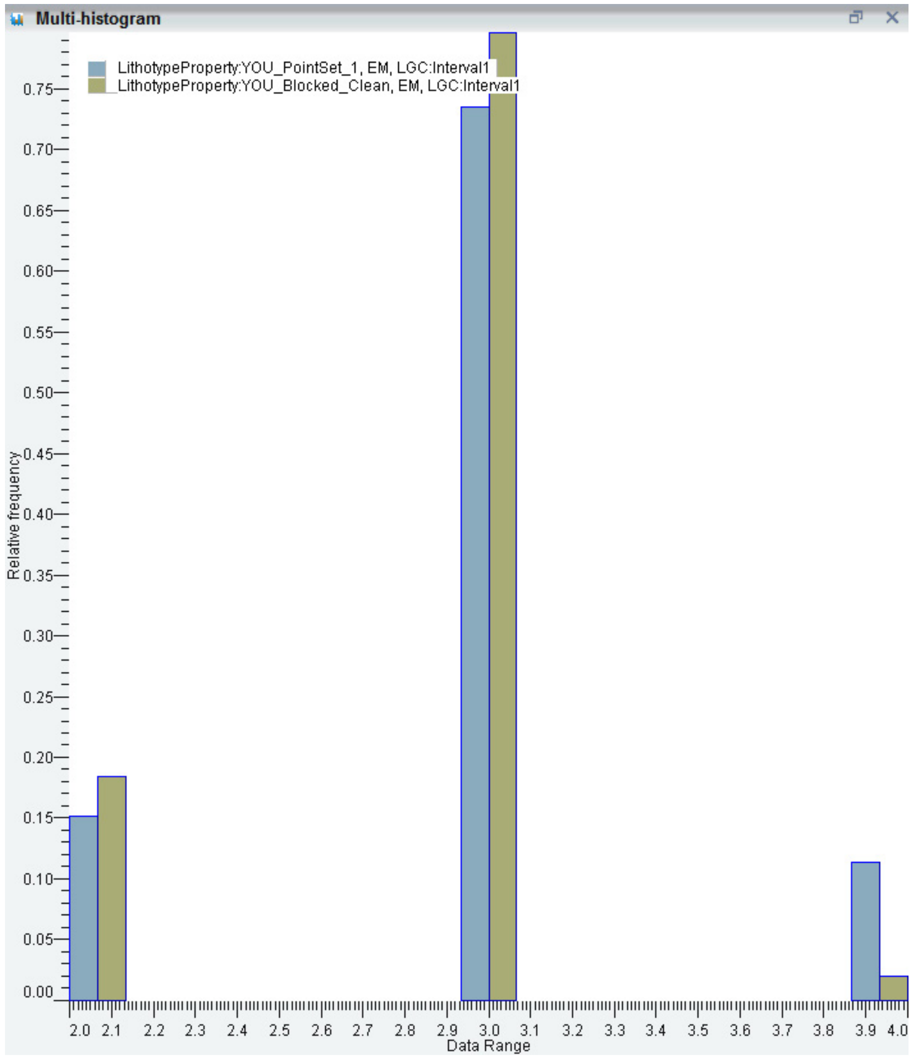
27. Click **New** to display the multi-histogram, showing the relative frequency for each lithology type. Remember that the numbers on the x-axis correspond to the lithology number.



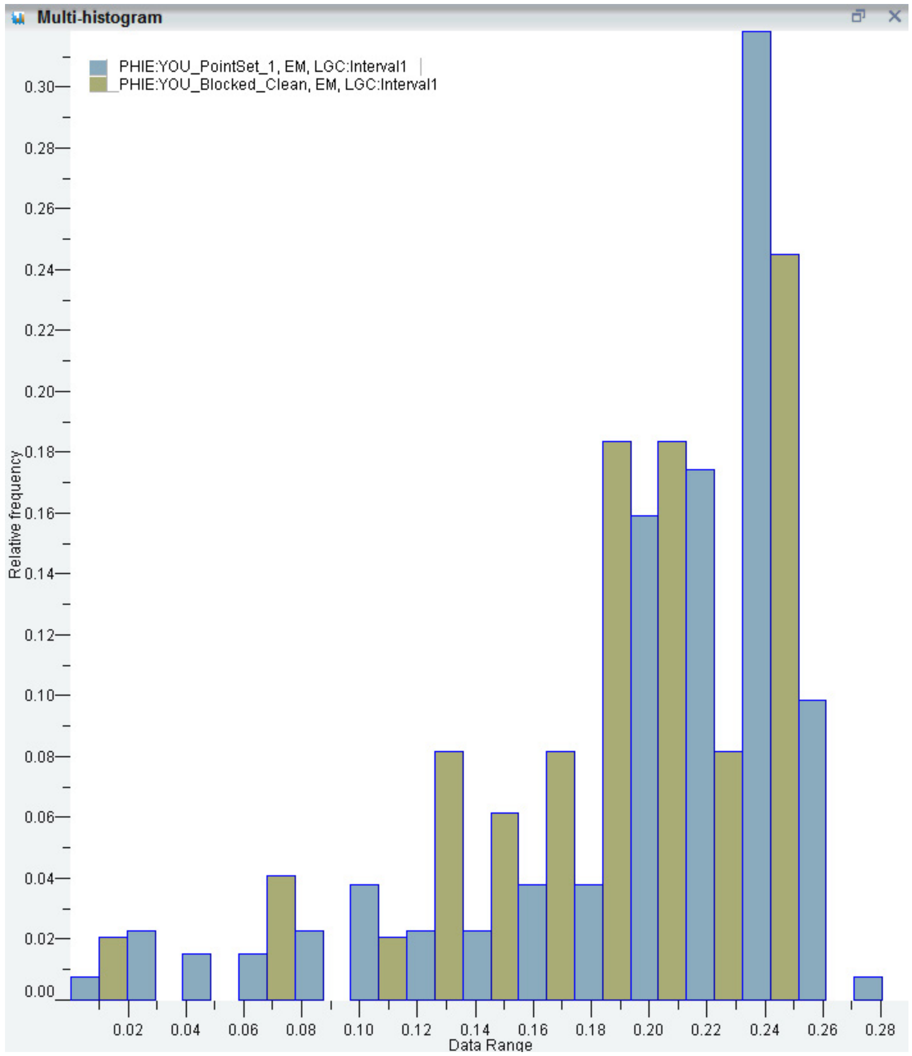
28. On the *Data Analysis* panel, change the subset for Series 1 and Series 2 from All Data to **Interval1** and then click **Apply**.



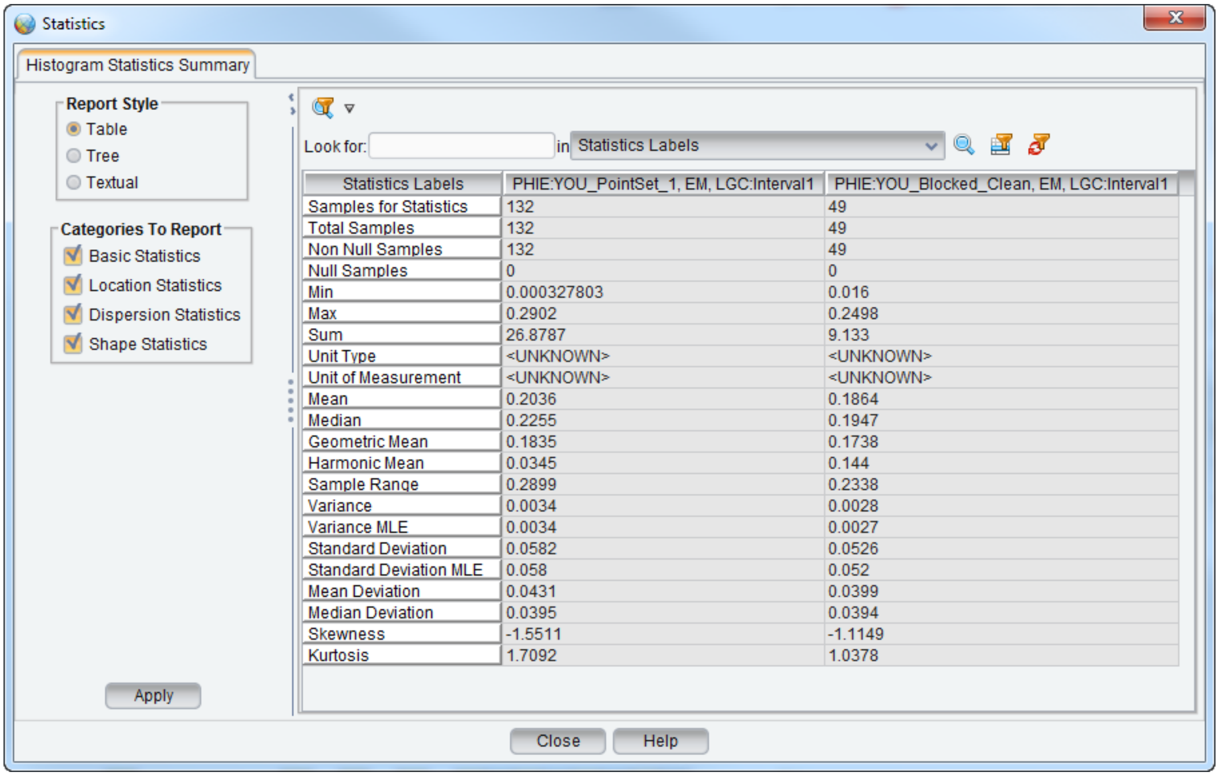
29. The following histogram shows the relative frequencies of the lithotypes in Interval1 for each point set. You can see that these are pretty close, which indicates that the well blocking was successful.



30. In the *Data Analysis* panel, select property **PHIE** for Series 1 and Series 2. Click **Apply**. The results are close.



31. In the *Data Analysis* panel, click **Statistics**. The results for Mean, Median, Variance, and Standard Deviation clearly show the success of the well blocking.



32. In the *Statistics* dialog, click **Close**.

You are now ready to proceed to Facies Modeling and Simulation.

Appendix A
Geological Mapping

The DecisionSpace Geosciences software provides fast and powerful tools for creating surface grids, contour maps, and for performing residual fit operations that honor well pick information.

You can create surface grids from the following data types:

- 2D and 3D horizons (multi-survay selection and prioritization)
- Point sets
- Surface picks
- Polygon/Centerline sets

In the gridding process you can also make selections for the following:

- Static shifts (2D horizons)
- Line lists (2D projects)
- Surveys
- Horizon merging method

With a velocity model selected in Domain Conversion, you can convert the domain of input data items to match the domain of the output surface grid.

Topics Covered in this Appendix

In this chapter you will learn how to:

- Create a surface grid, using the Refinement Gridding algorithm or the B-Spline Gridding algorithm.

- Create a one-step grid and contour map using Auto Contour.

- Create a quick contour map, including fault polygons (Polygon/Centerline sets), as an option.

- Customize contours, using the contouring options.

Understanding Gridding Algorithms

Refinement Gridding is the default gridding algorithm. It is distinguished from the B-Spline algorithm, mainly by these characteristics:

- It enables you to include fault polygons in the grid.
- It is particularly effective at producing a grid that optimizes the appearance of both regional trends and local detail.
- It conducts a series of refinements until the cell size of the target grid is reached, passing the surface model through a bi-harmonic filter to produce a minimum curvature surface.

B-Spline Gridding has these distinguishing characteristics:

- It produces a grid from a large dataset or scattered data more quickly than refinement gridding.
- It does not handle fault polygons.
- It creates a coarse-to-fine hierarchy of gridded control lattices to generate a sequence of bicubic B-Spline functions, whose sum is used to build a surface model that honors the input data. The number of numerical calculations per data point and per grid node is small, allowing rapid surface generation.

Refinement Gridding and B-Spline Gridding share these characteristics:

- They grid very smoothly between points.
- Their grids extrapolate values above and below the actual data values.
- Their trends continue beyond the data points areas that are void of data.

It is best to use Refinement Gridding unless you are working with a very large dataset. With a large dataset, you may want to produce a grid using each algorithm, and then pick the method that gives you the best results.

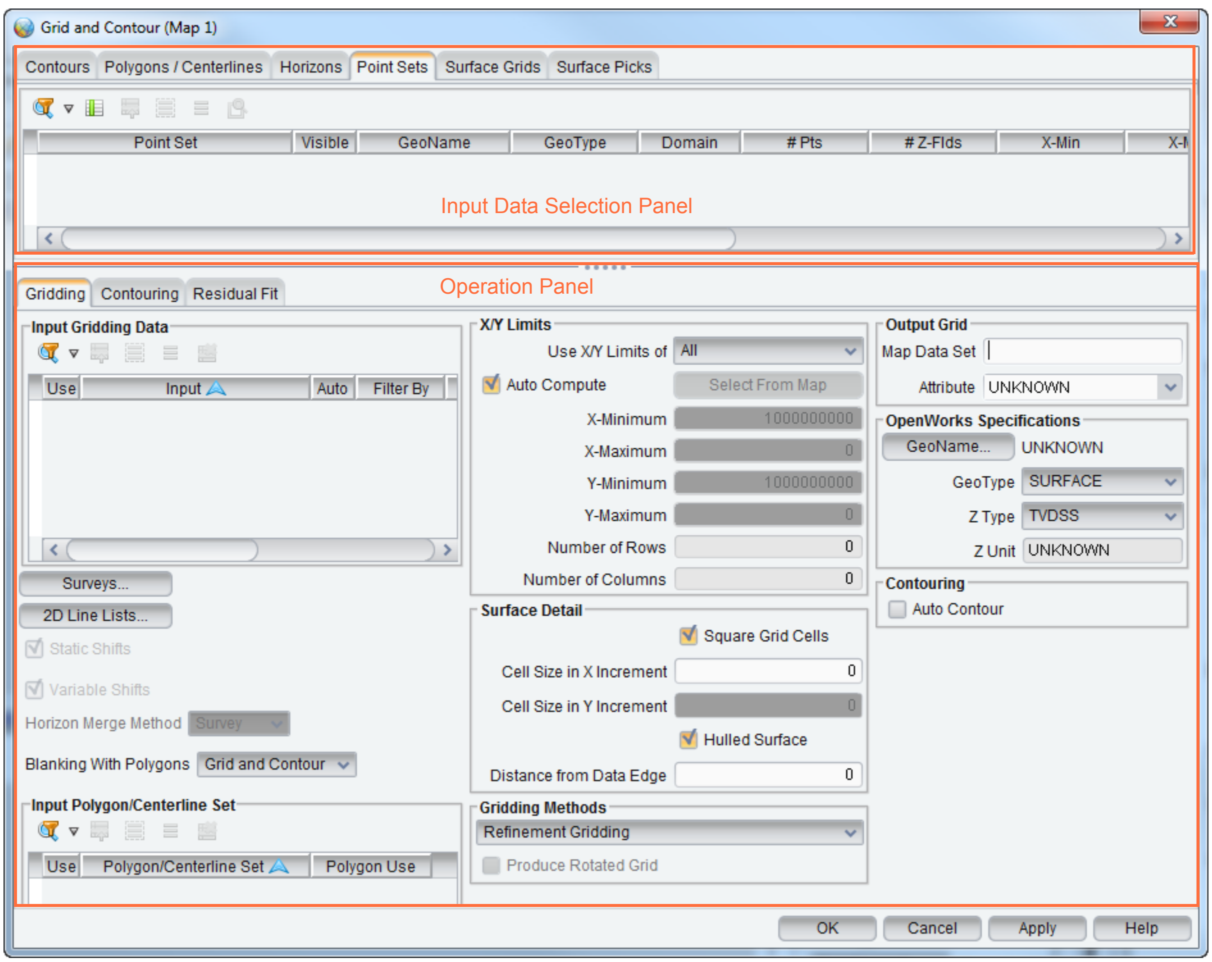
The Grid and Contour Dialog

To open the *Grid and Contour* dialog, select **Tools > Grid and Contour** on the main menu bar. This functionality is available in all views. Surface grids are domain sensitive, which means they display only in the domain in which they were created.

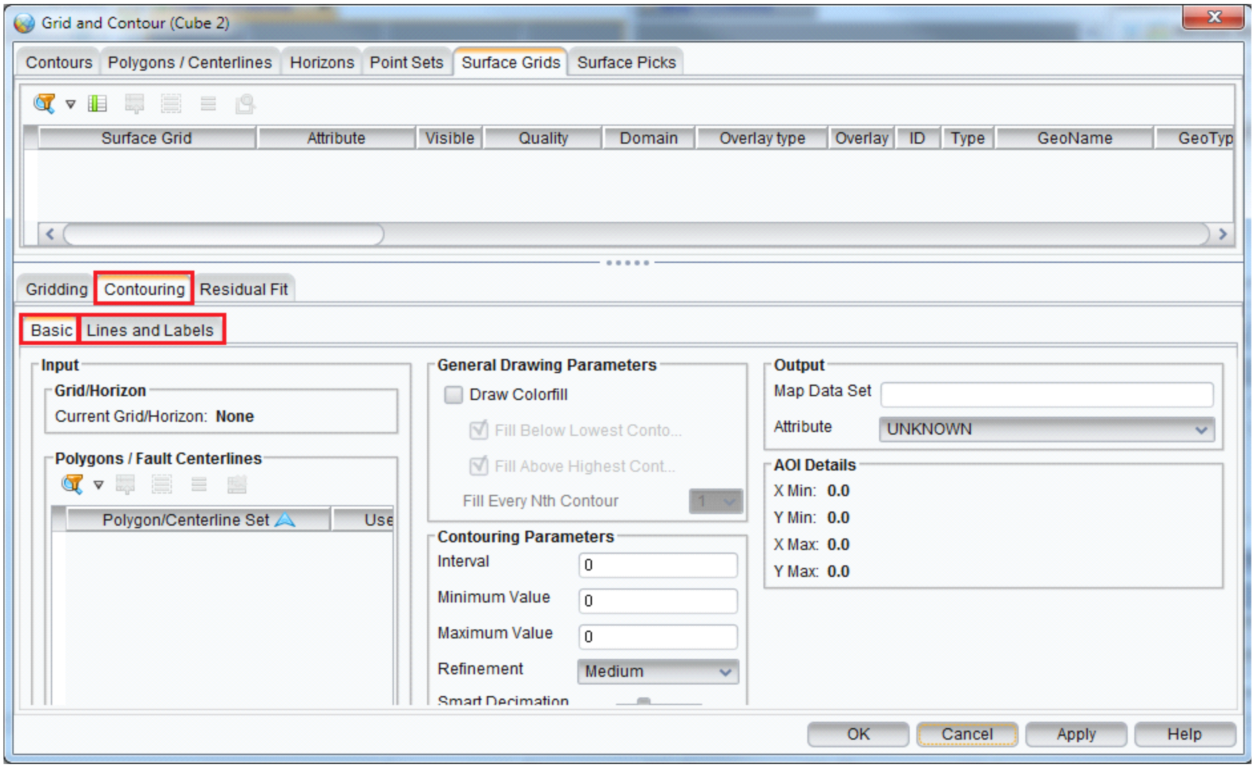
The *Grid and Contour* dialog has two panels:

- The input data selection panel is the top part of the dialog. It contains tabs with data items from the *Inventory* task pane. Click the appropriate tabs to select from lists of data items for gridding and contouring.

- The operation pane is the bottom part of the dialog. On it you select an operation (gridding, contouring, or residual fit), specify the data you want to use to create the selected operation, and set parameters.



On the operations panel, you will click the *Contouring* tab to access its functionality. The *Contouring* tab has two tabs within it: *Basic* tab and *Lines and Labels* tab.



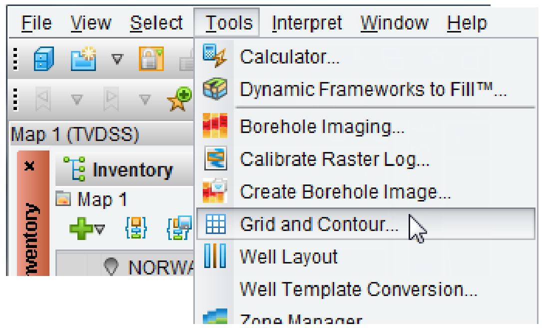
You can use the following data types to create contours:

- Surface grids
- Horizons
- Polygon/Centerline sets

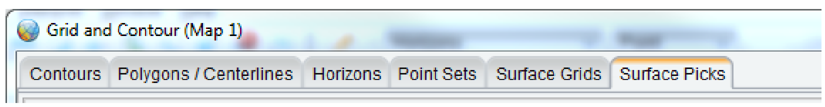
Exercise A-1: Creating a Basic Grid

You can create a simple surface grid with a few steps.

1. On the *DecisionSpace* dialog menu bar, select **Tools > Grid and Contour** to display the *Grid and Contour* dialog.

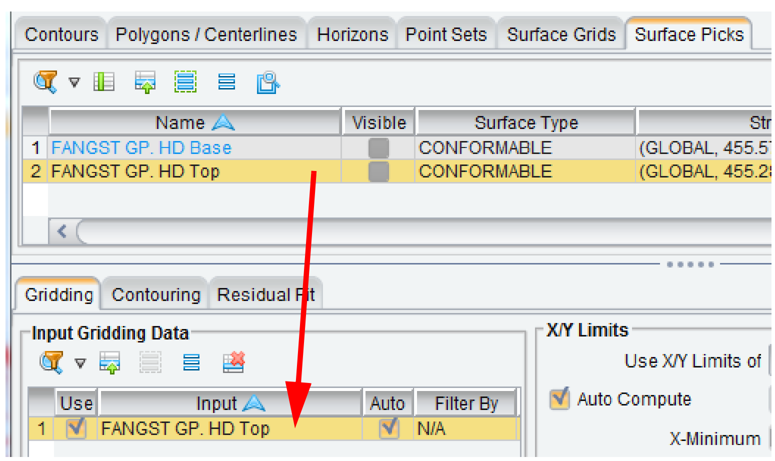


2. Near the top of the *Grid and Contour* dialog, click the ***Horizons***, ***Point Sets***, or ***Surface Picks*** tabs.



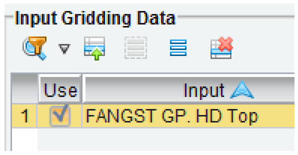
3. In each data category, click a **data item**.

4. The data item you picked moves into the *Input Gridding Data* panel in the Operation panel of the *Grid and Contour* dialog.



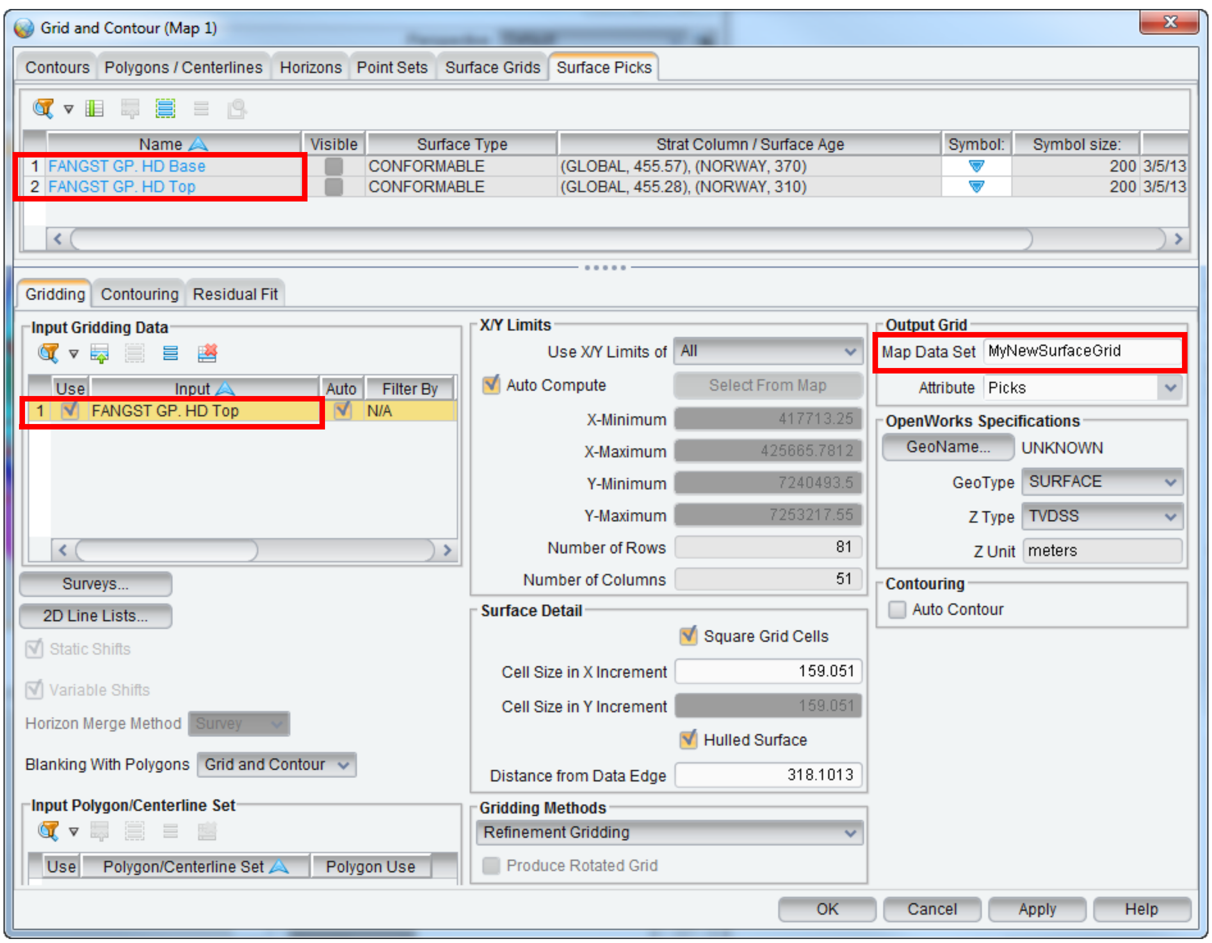
5. Continue selecting data until all the items you want to work with are in the *Input Gridding Data* panel.

6. Review the **data items** in the *Input Gridding Data* panel. If you do not want to use certain items, uncheck the **Use** box associated with the items.



7. Inspect the Map Data Set text field in the *Output Grid* panel. It contains the default name for the new surface grid. This name is the same as the last data item you selected for the *Input Gridding Data* panel. You can enter another name if you wish.

8. Click **Apply** or **OK**.



The name of the new grid appears as a Surface Grid, in the following place:

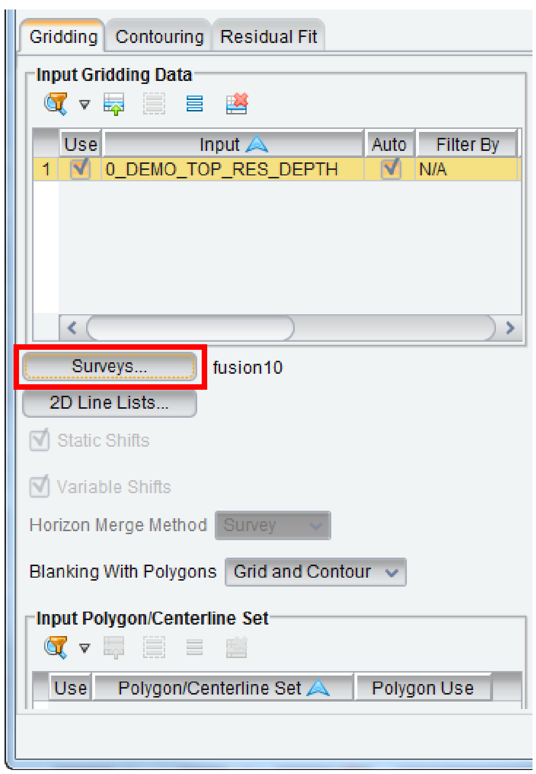
- The *Data Input* panel of the *Grid and Contour* dialog
- The Selected Data list of the *Select Session Data* dialog
- The *Data Properties* tab of the *Contents* dialog
- The *Inventory* task pane

Customizing Grids

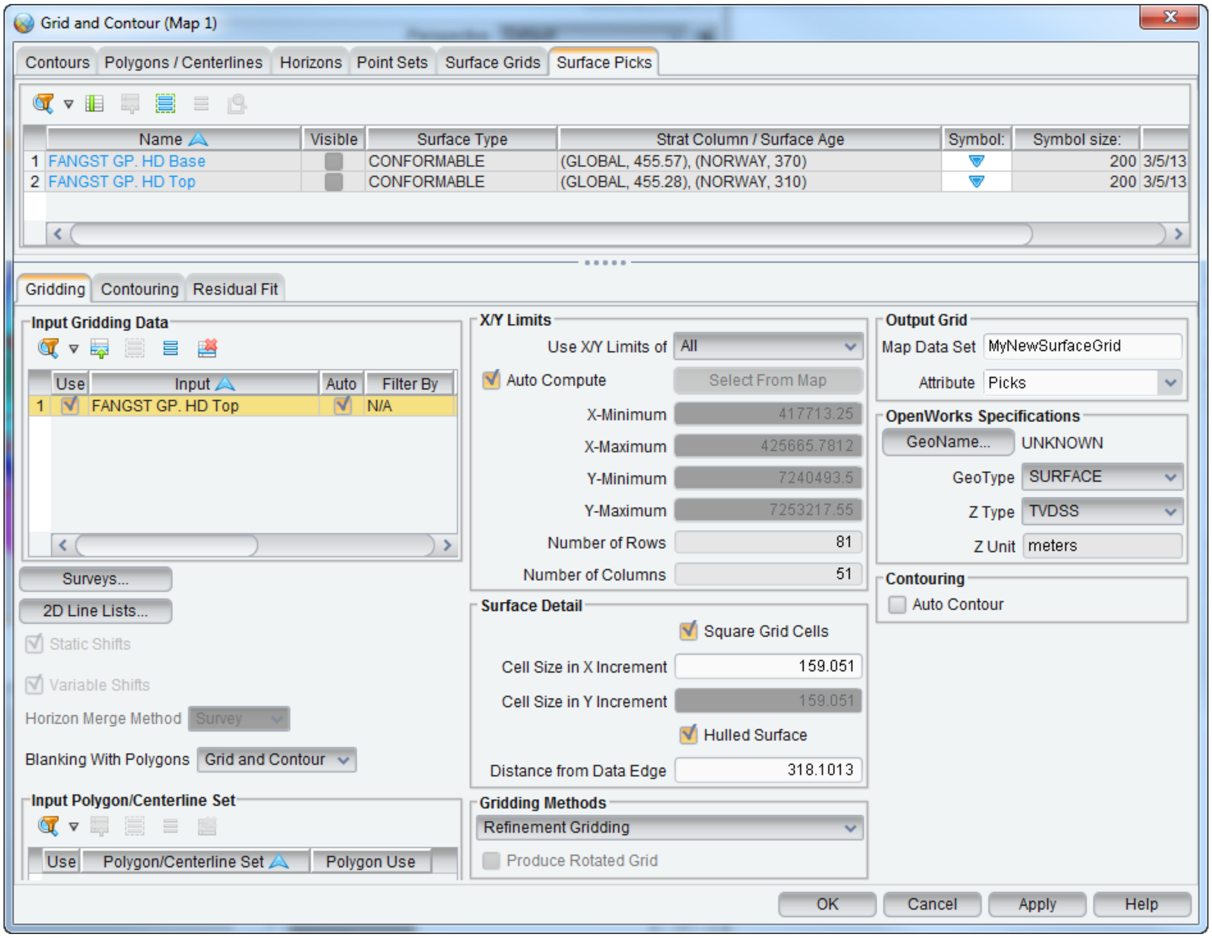
The *Grid and Contour* dialog provides several ways to customize your grid. These are discussed below in the order in which they appear in the dialog.

The Survey Button

If you have chosen horizons that exist in multiple surveys, the Survey button (Surveys...) will be activated. In addition, the names of the surveys that currently contain the data appear associated with the Survey button. To select specific surveys, click the button and make your selections in the *Select Survey(s) for Horizons* dialog.



While you are in the *Select Survey(s) for Horizons* dialog, you can choose areas of interest for multi-survey surface gridding. Follow the instructions below.

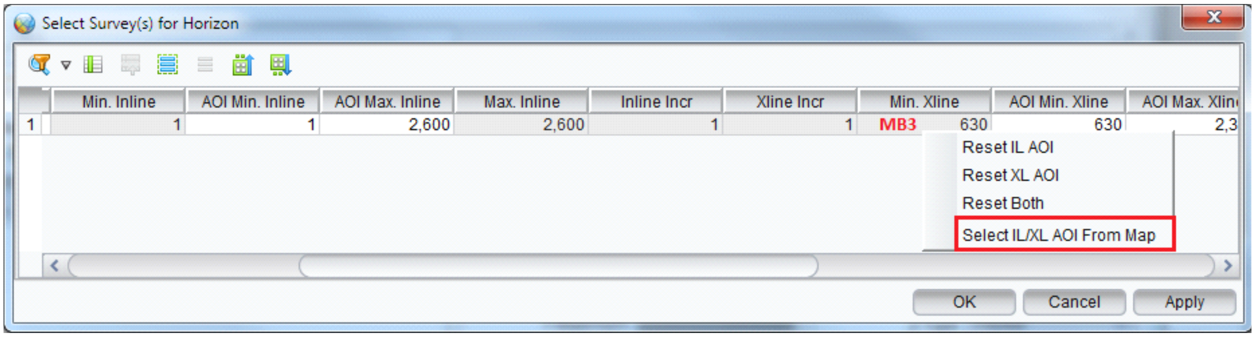


To choose areas of interest for 3D surveys in multi-survey gridding, perform one of the following bulleted workflows in the *Select Survey(s) for Horizons* dialog:

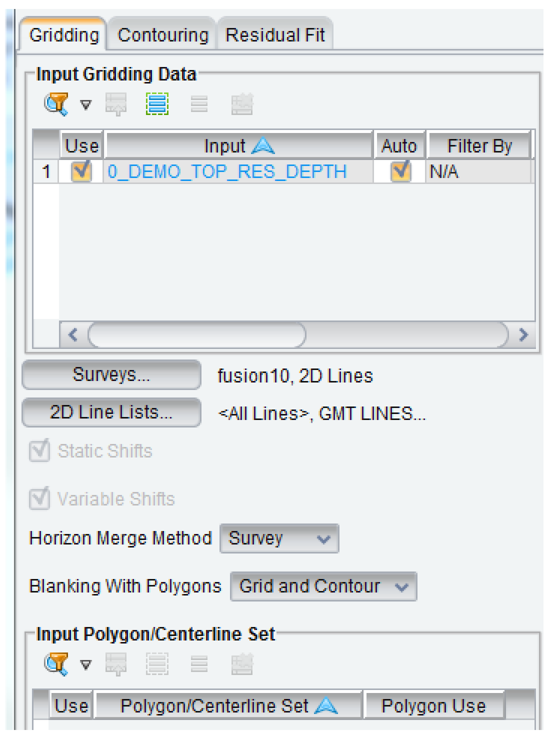
- Edit the following fields:
  - — AOI Min Inline
  - — AOI Max Inline
  - — AOI Min Xline
  - — AOI Max Xline

AOI Min. Inline	AOI Max. Inline	Inline Incr	Xline Incr	AOI Min. Xline	AOI Max. Xline
1	2,600	1	1	630	2,300

- With your cursor on the table, **MB3 > Select IL/XL AOI from Map**, then drag your cursor in the map until you have drawn the area you want. **MB2** to end the selection.



9. For either selection method, click **Apply** or **OK** in the *Select Survey(s) for Horizon* dialog. The AOI changes are reflected in the appropriate columns of the *Select Survey(s) for Horizon* dialog, but are not reflected in the boundaries of the surveys on the *Map* view.



2D Line Lists Button

If you have selected a 2D horizon as the input for gridding, the 2D Line Lists button is active, and the name of the selected list appears next to it. Click the 2D Line Lists button to open the *Loaded 2D Line Lists* dialog. If you want to use other 2D line lists, select them in the dialog and click **Apply** or **OK**.

Static Shifts Checkbox

If you have selected a 2D horizon for input to gridding and have static shifts for the 2D lines, the Static Shifts checkbox will be active. If this box is checked, the static shifts stored with the 2D lines will be applied to the input 2D horizon before gridding.

Variable Shifts Checkbox

If you have selected a 2D horizon as the input for gridding and have static shifts for the 2D lines, the Variable Shifts checkbox will be active. If this box is checked, the variable shifts stored with the 2D lines will be applied to the input 2D horizon before gridding.

Horizon Merge Methods

When you are working with more than one input, you can select the Horizon Merge Method button to enable you choose which and how much data you send to the gridder. The methods are:

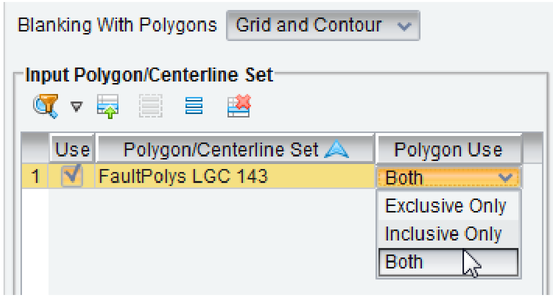
- **Survey** — uses the priority of the surveys listed in the *Select Survey(s) for Horizons* dialog in the *Grid and Contours* dialog, where the areas overlap. This is the default. Survey priorities are listed in the *Select Survey(s) for Horizon* dialog.

- **Proximity** — tests every point in the overlapping area to see if another sample from a higher priority survey exists. If it does not, it is used. Otherwise, the sample is discarded. This feature allows greater flexibility in combining data from multiple surveys, in which the areas to be captured are not rectangular. Using Proximity is especially important in cases where multiple surveys overlap and the available seismic from one is not represented by a rectangle. Without digitizing a polygon for each survey, you can specify a priority and still pick up data from lower priority surveys where no data exists for the high priority survey.

- **All Data** — all data is sent to the gridder regardless of any priority settings.

Blanking With Polygons

When you use this option, grid cells within exclusive polygons are not populated and, if Auto-Contour is toggled on, contours are not drawn inside inclusive polygons. The pull-down menu lets you choose whether to use blanking with only Contours, or with Grid and Contour. The default is Grid and Contour. You can also select None.



Input Polygon/Centerline Set Table

If you want gridding to honor faults, select one or more Polygon/ Centerline sets under the *Polygons/Centerlines* tab at the top of the *Grid and Contour* dialog.

The names of the selected polygon sets appear in the text field Input Polygon/Centerline Set table.

As in the *Input Gridding Data* table, the Use checkbox associated with each name is toggled on by default. Review your data and deselect any item you do not wish to use.

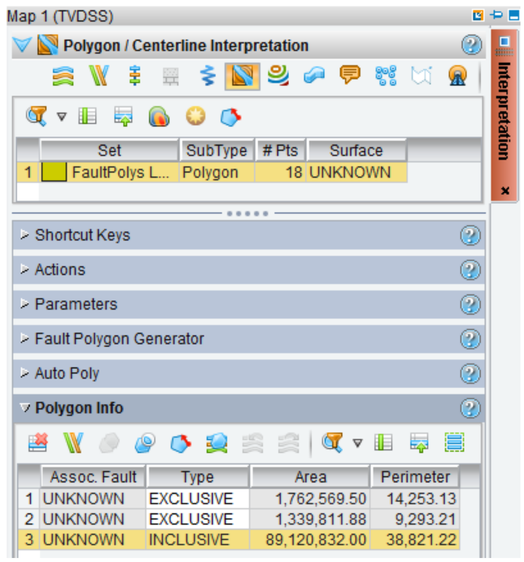
You can choose to use only those polygons marked as an exclusive fault set, only those polygons specified as inclusive, or both. Click in a text field of the Polygon Use column and select one of the following from the pull-down menu:

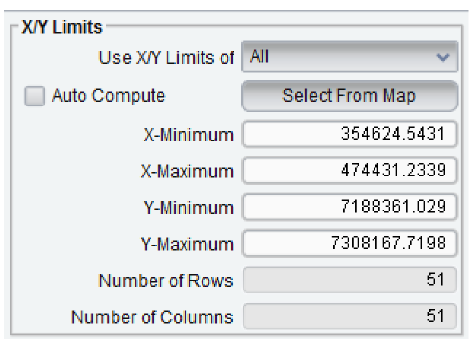
- Exclusive Only
- Inclusive Only
- Both

By default, both inclusive and exclusive faults within the selected polygon sets will be considered in the gridding. You can make this specification in the *Polygon/Centerline Interpretation* task pane.

Note:

A polygon is set as Inclusive or Exclusive in the *Polygon/Centerline Interpretation* task pane (under Polygon Info). It is possible to have Inclusive and Exclusive polygons in the same Polygon Set.





X/Y Limits

To change the area of interest for gridding, enter new values in the *X/Y Limits* panel of the *Gridding* tab. The X-Minimum, X-Maximum, Y-Minimum, and Y-Maximum values specify the grid area. (If you use a horizon to create a rotated grid, you cannot change the values.)

You can also use the *Select From Map* button to choose X/Y Limits. This option lets you easily define an area in the *Map* view. Follow this procedure:

1. Toggle off **Auto Compute** in the *X/Y Limits* panel and click the **Select From Map** button.

2. Click in the **map** and drag a **rectangle shape**.

3. When the rectangle is complete, **MB2** to apply these coordinates in the *X/Y Limits* panel.

   - **Pointset** — If you select an input point set, the computed X/Y Limits values that appear by default encompass the input data points and expand the area of interest slightly in all directions.

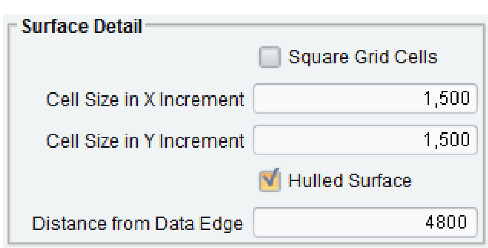
   - **Horizon: Unrotated Grid** — If you use an input horizon to create an unrotated grid, the default X/Y Limits values define an orthogonal bounding box around the part of the horizon that contains data, with the area of interest expanded slightly.

   - **Multiple Inputs** — If you use multiple items in the Input Gridding Data panel, X/Y Limits define an orthogonal box around all the data, with the AOI expanded slightly.

   - **Horizon: Rotated Grid** — If you use an input horizon to create a rotated grid, the default X/Y Limits values reflect the x,y limits of the 3D survey. The Produce Rotated Grid option is active only when you have selected one 3D survey.

Number of Rows and Number of Columns

The *X/Y Limits* panel in the *Gridding* tab reports the number of cell rows and cell columns in the output grid. These values appear in the read-only Number of Rows and Number of Columns text fields. You cannot edit these values directly, but if you change the grid cell size, the Number of Rows and Number of Columns values are updated automatically.



Surface Detail

To change the grid cell size and proportions, enter new values for the following options in the *Surface Detail* section of the *Gridding* tab:

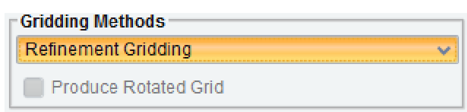
- **Square Grid Cells checkbox** — By default, this option is toggled on. To change from square grid cells to rectangular cells, deselect this checkbox.

- **Cell Size in X Increment field** — To change the default X increment value for each grid cell, enter a new value in this text field. The default dimensions of grid cells are calculated, based on the extents of the input data.

- **Cell Size in Y Increment field** — To change the default Y increment value for each grid cell, enter a new value in this text field.

- **Hulled Surface checkbox** — To reduce extrapolation effects that can appear in some grids, you can create a hull encompassing the input data by toggling on Hulled Surface. A data hull is an imaginary line drawn around the data points used in gridding. Grid nodes outside the hull are set to null.

- **Distance from Data Edge** — If you toggled on Hulled Surface, review the Distance from Data Edge value. The default value is an input-dependent value (approximately three cell sizes wide). If you toggle on Hulled Surface, the Distance from Data Edge setting expands the data hull boundary away from the data to include additional grid nodes and enable some extrapolation beyond the actual input data.



Gridding Methods

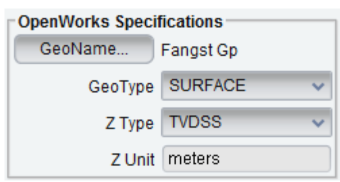
The default gridding method is Refinement Gridding, which produces a grid that honors both regional and local trends. The second option is B-Spline Gridding, which works well with a very large dataset.

For a grid created from a 3D horizon, the default output is gridded orthogonally. To produce a grid that is aligned with the 3D seismic survey, toggle on Produce Rotated Grid.

Using a Customized Grid Name

To use a custom name for the output grid, enter a new name in the Map Data Set text field in the *Output Grid* panel. The grid name identifies the grid for display purposes, during contouring, and in OpenWorks database.

You can also select an attribute from the Attribute pull-down menu in the *Output Grid* panel.



OpenWorks Specifications

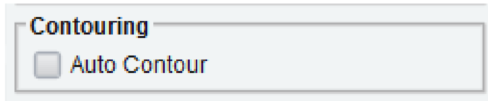
To change the default Geo Name and Geo Type values, make changes as shown in the *OpenWorks Specifications* section of the *Gridding* tab:

- **Geo Name** — Click the **Geo Name** button and choose from the resulting *Select Geo Name* dialog. A geoname is usually the name of a geological formation associated with the grid (such as a surface pick or unit).

- **Geo Type** — Click the **Geo Type** arrow and select an option from the pull-down menu. The Geo Type is a categorical term used to represent the geological formation associated with the grid (such as SURFACE, STRAT_UNIT, or FAULT).

- **Z Type** — Click the **Z Type** arrow and select an option from the pull-down menu. The Z Type is the domain or type of measurement used to express Z values (TWT, TVDSS, TVD, MD, or OTHER) for the grid data. The output grid Z Type defaults to the domain of the view from which the *Grid and Contour* dialog was launched. You can select data items for input to gridding that are not of the same domain (Z Type) as the output grid.

- **Z Unit** — These are read-only values based on the Z Type setting. Z Unit values identify the units of measure for grid Z values, such as feet, meters, seconds, ms, or UNKNOWN.



Using Auto Contour

This option lets you generate a grid and a contour map in one step. To do this, toggle on **Auto Contour**.

The output contour set will take on the same Map Data Set, Attribute, Geo Name, and Geo Type as the corresponding output grid.

If you use the Auto Contour option and want to customize the display parameters for the contours, you can make another contour set in the *Contouring* operation tab or edit the appearance of the auto contour in the *Interpretation Contour* task pane. You cannot customize the contouring settings for the initial Auto Contour operation. To create a follow-up contour map, change one or more of the optional settings in the *Contouring* tabs, then click **Apply** or **OK** to generate another contour map.

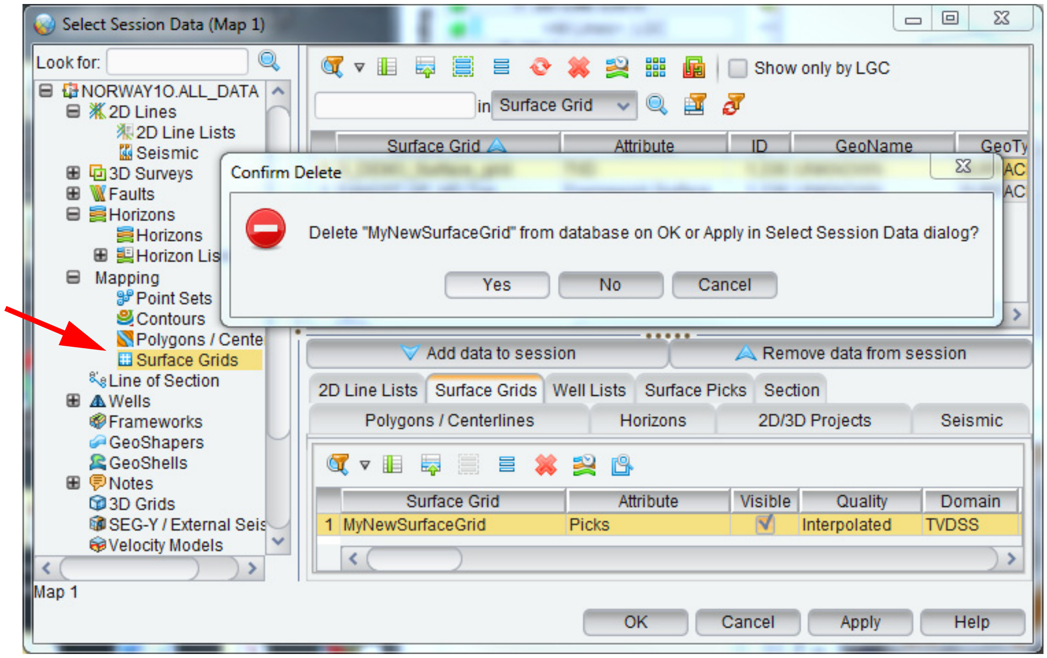
Deleting Grids

To delete a grid, follow these steps:

1. Select the **surface grid** in the *Select Session Data* dialog. The Delete selected grids icon becomes available.

2. Click the **Delete selected grids** ( ) icon. A dialog appears asking you if you want to delete the grid from the database. Click **Yes**.

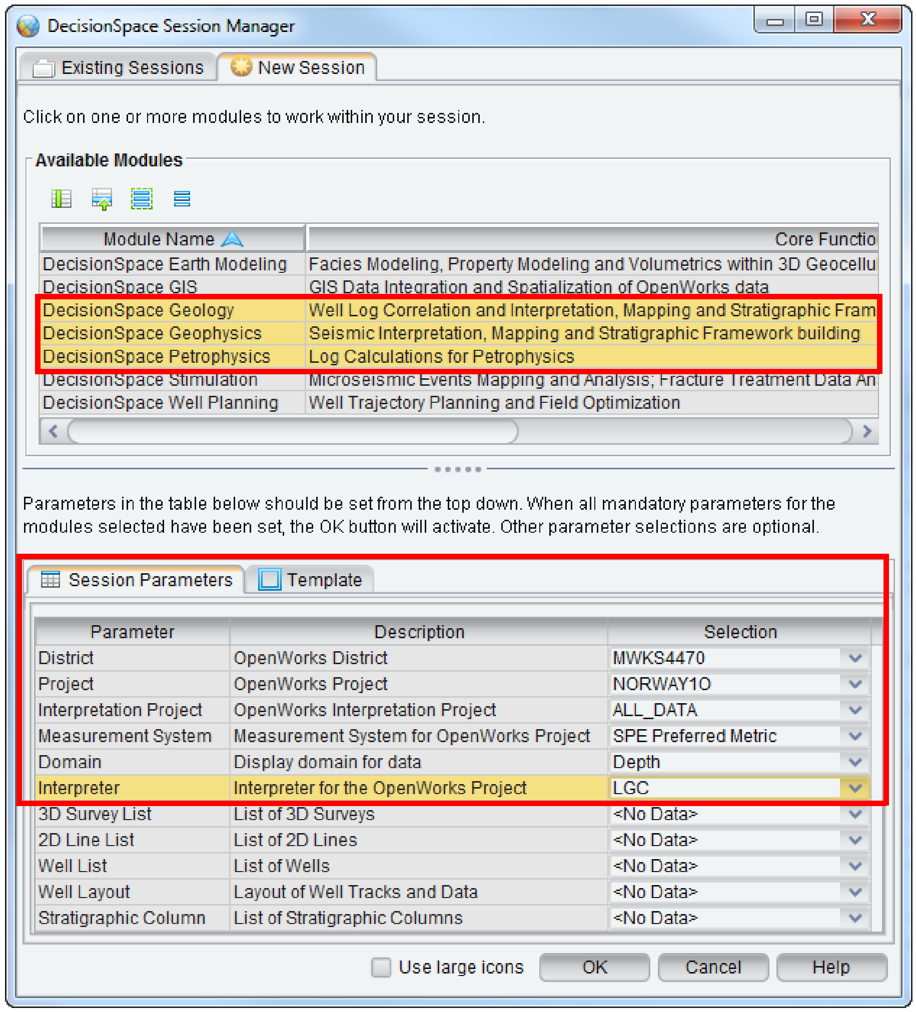
The surface grid will be deleted from the OpenWorks database.



> **Note:**
>
> You may delete only surface grids that are yours or that were created by an interpreter ID with public access. Attempting to delete the surface grids of others will produce an error message.

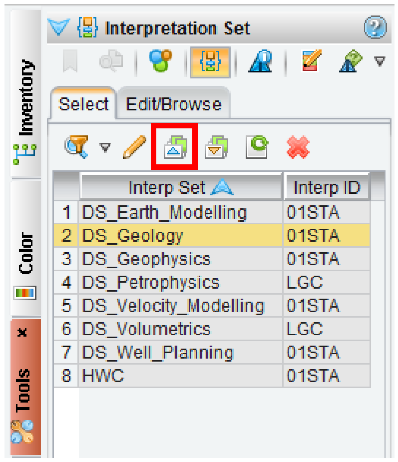
Exercise A.2: Creating Surface Grids

1. On the *New Session* tab of the *DecisionSpace Session Manager* select modules and parameters, as shown in the following image, then click **OK**.



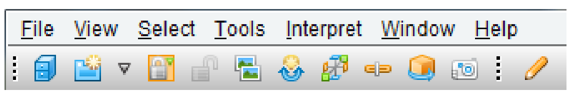
2. On the *Tools* task pane click the **Interpretation Set** ( ) icon to activate it. On the Interp Set list, select **DS_Geology** and click the

Load Data to Session ( ) icon.

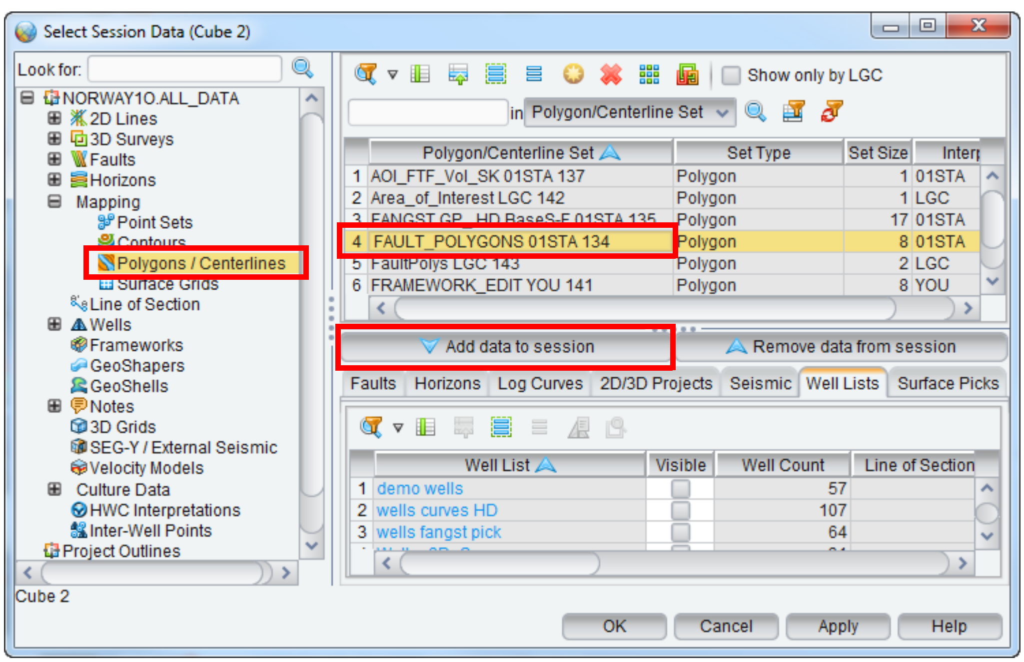


All of the data in DS_Geology ISet will load to the session.

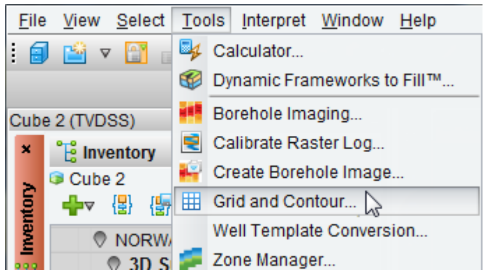
3. On the Horizontal toolbar of the *DecisionSpace* dialog, click the **Select Session Data** ( ) icon to open the *Select Session Data* dialog.



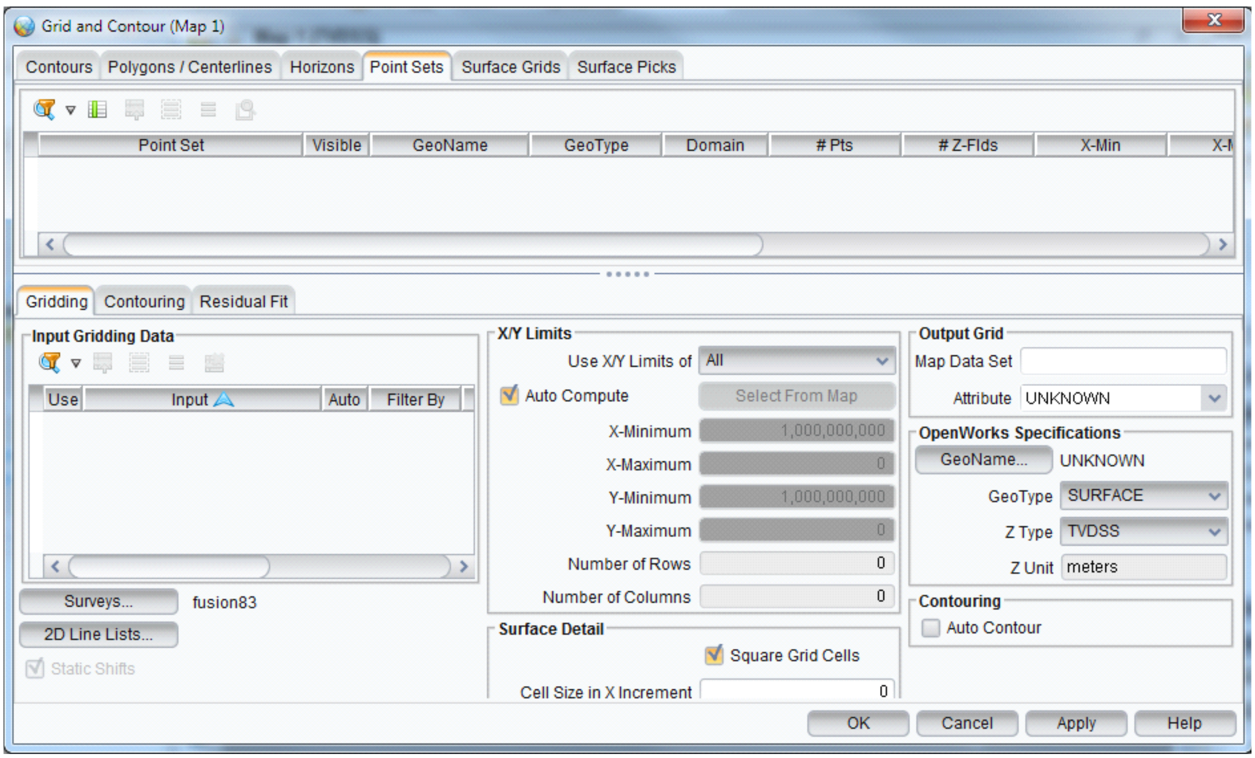
4. In the left task pane of *Select Session Data* dialog, click **Mapping > Polygons / Centerlines**. In the right upper panel, select **FAULT POLYGONE 01STA 134** and click the **Add data to session** button. Click **OK**.



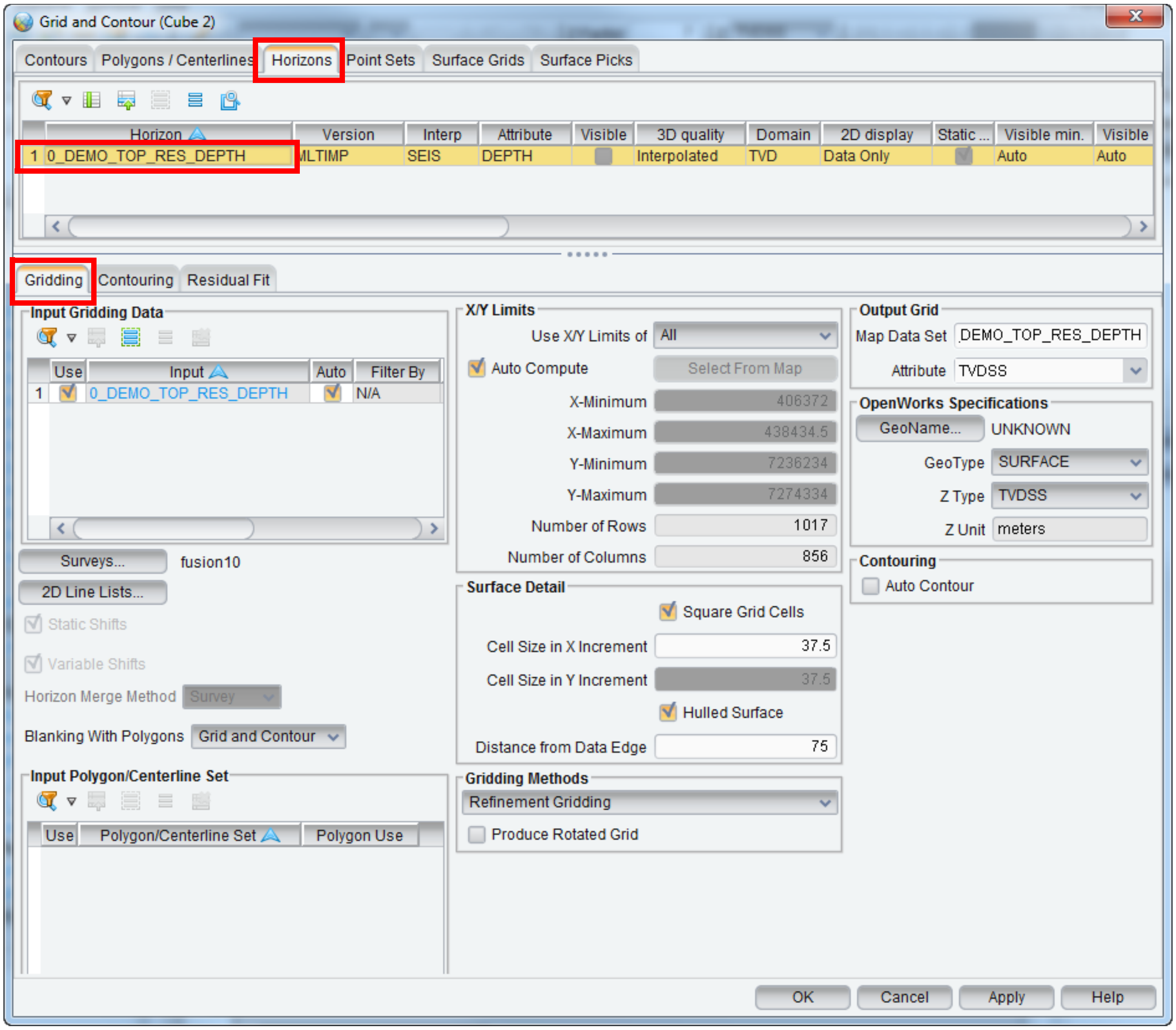
5. On the *DecisionSpace* dialog menu bar, select **Tools > Grid and Contour...**.



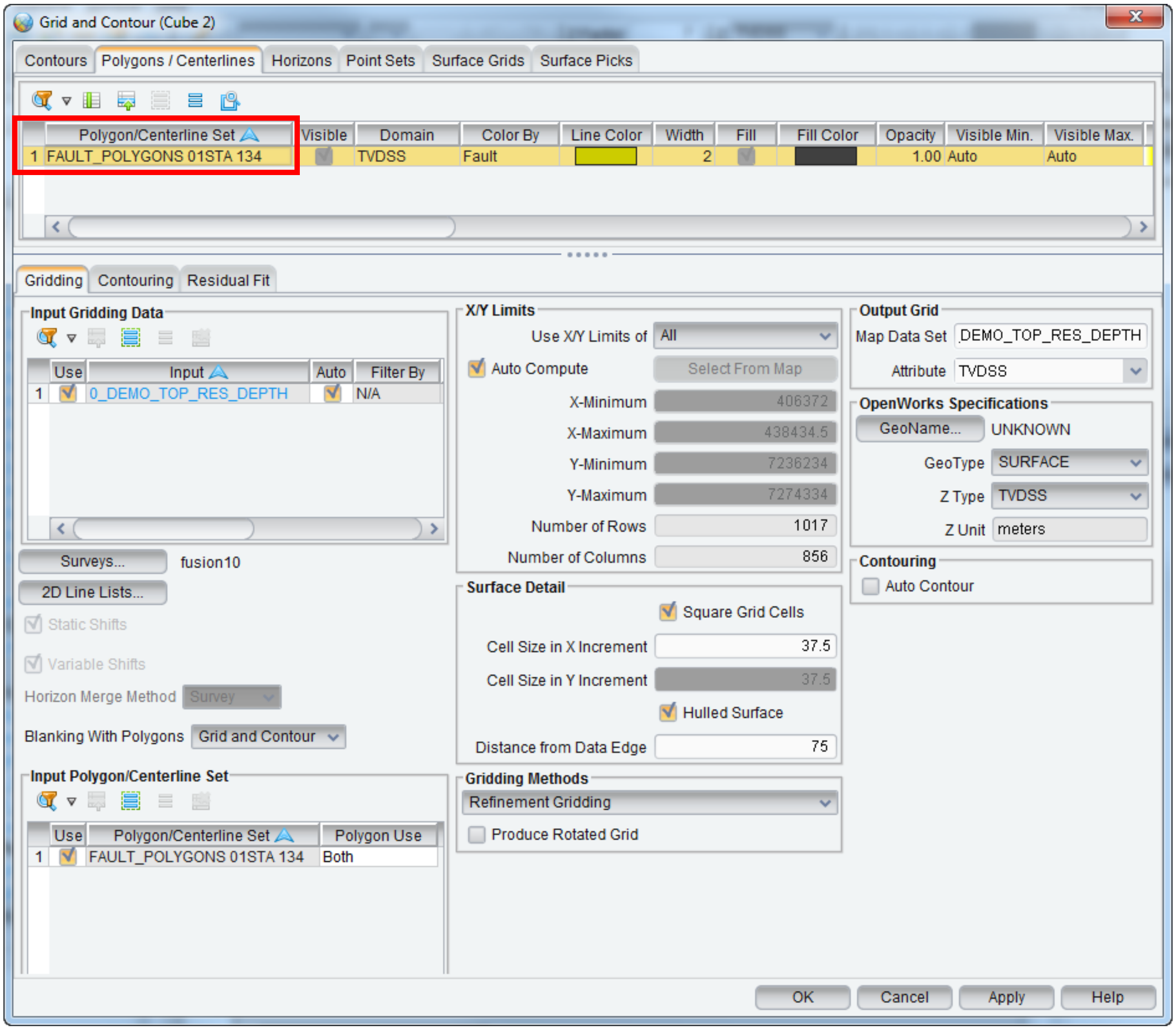
The *Grid and Contour* dialog opens.



6. In the *Grid and Contour* dialog, select the ***Gridding*** operation tab then select the ***Horizons*** input data tab. Select **0_DEMO_TOP_RES_DEPTH**.


Grid and Contour (Cube 2)
Contours
Polygons / Centerlines
Horizons
Point Sets
Surface Grids
Surface Picks
Horizon
Version
Interp
Attribute
Visible
3D quality
Domain
2D display
Static ...
Visible min.
Visible
1 0_DEMO_TOP_RES_DEPTH
MLTIMP
SEIS
DEPTH
Interpolated
TVD
Data Only
Auto
Auto
Gridding
Contouring
Residual Fit
Input Gridding Data
Use
Input
Auto
Filter By
1 0_DEMO_TOP_RES_DEPTH
N/A
Surveys...
fusion10
2D Line Lists...
Static Shifts
Variable Shifts
Horizon Merge Method
Survey
Blanking With Polygons
Grid and Contour
Input Polygon/Centerline Set
Use
Polygon/Centerline Set
Polygon Use
X/Y Limits
Use X/Y Limits of
All
Auto Compute
Select From Map
X-Minimum
406372
X-Maximum
438434.5
Y-Minimum
7236234
Y-Maximum
7274334
Number of Rows
1017
Number of Columns
856
Surface Detail
Square Grid Cells
Cell Size in X Increment
37.5
Cell Size in Y Increment
37.5
Hulled Surface
Distance from Data Edge
75
Gridding Methods
Refinement Gridding
Produce Rotated Grid
Output Grid
Map Data Set
DEMO_TOP_RES_DEPTH
Attribute
TVDSS
OpenWorks Specifications
GeoName...
UNKNOWN
GeoType
SURFACE
Z Type
TVDSS
Z Unit
meters
Contouring
Auto Contour
OK
Cancel
Apply
Help

7. On the *Polygons/Centerlines* input data tab, select **FAULT_POLYGONS 01STA 134**.



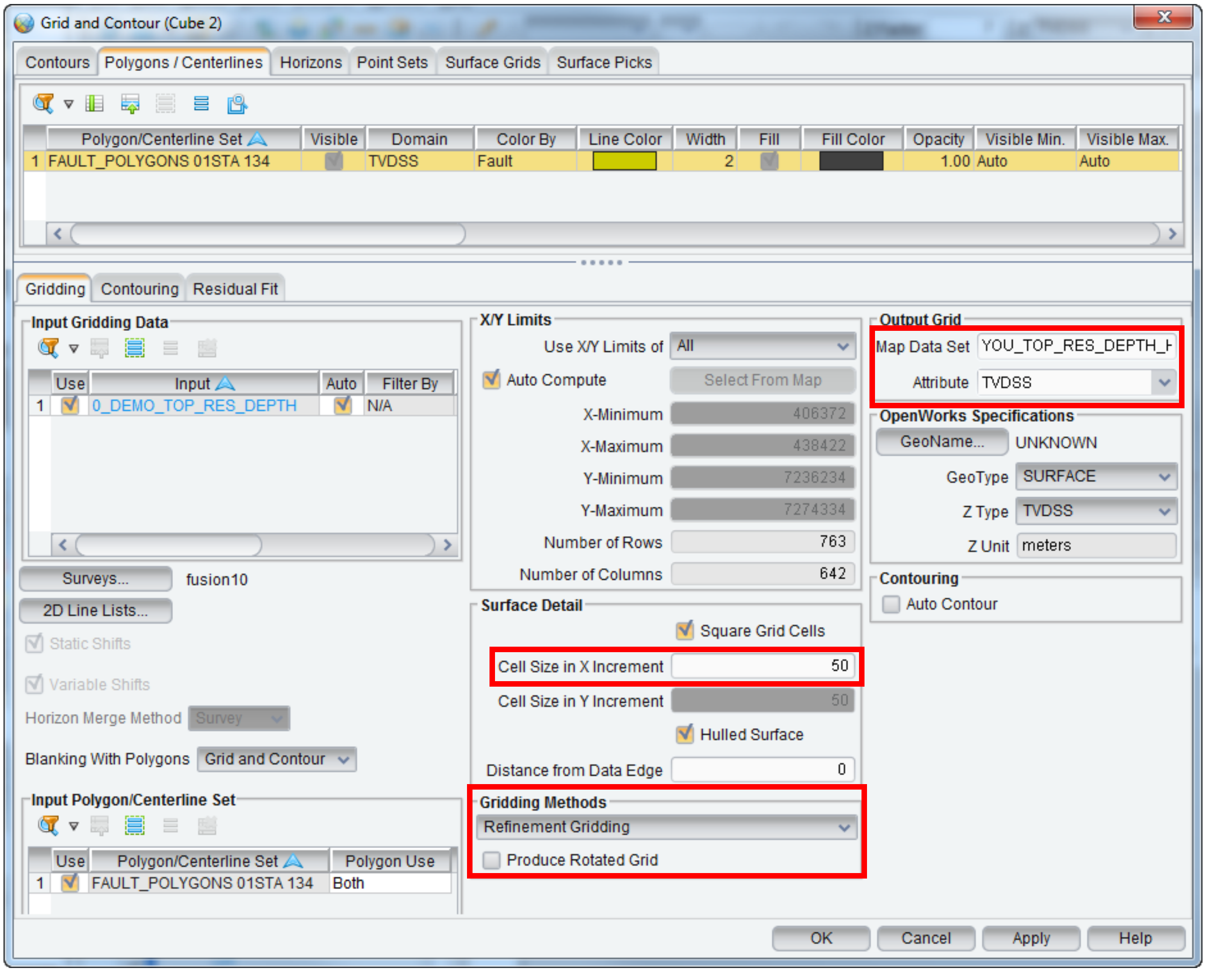
8. On the *Gridding* tab of the *Grid and Contour* dialog make following changes.

    - In the *Surface Detail* panel, enter **"50"** in the Cell size in X Increment text field. Leave **Hulled Surface** toggled on**.** Enter **"0"** in the Distance From Data Edge text field.

    - In the *Gridding Methods* pull-down menu, accept the default selection, **Refinement Gridding.** Toggle off **Produce Rotated Grid**.

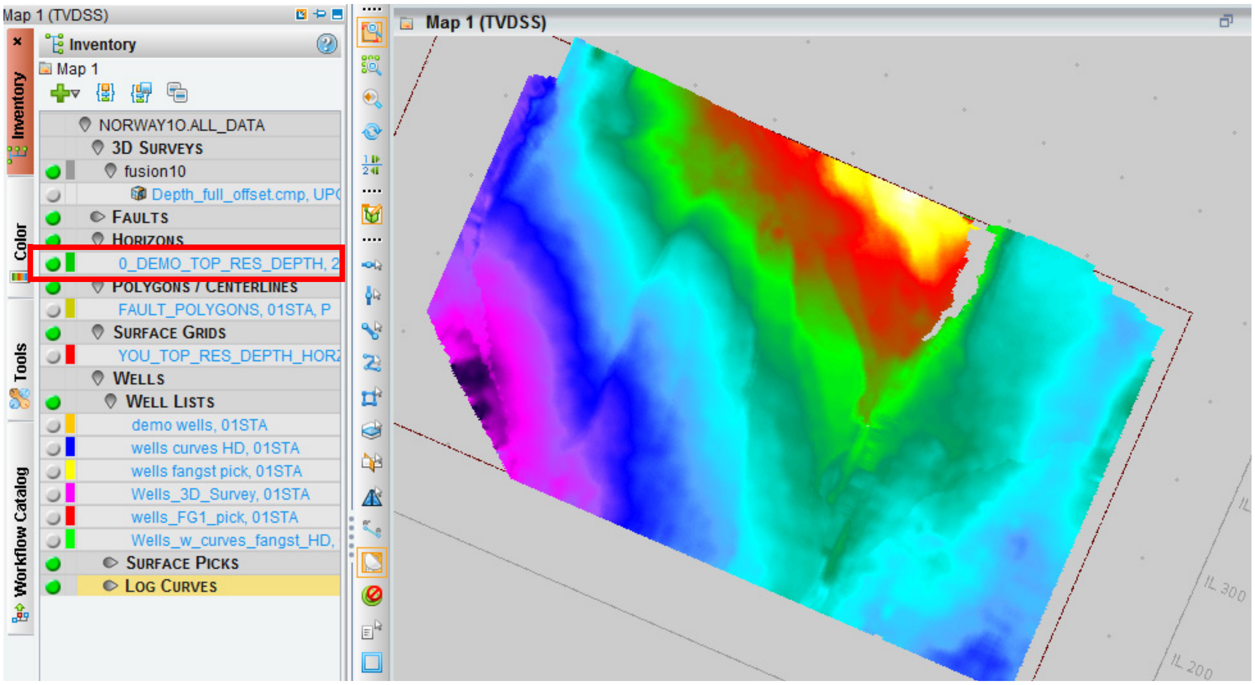
    - In the *Output Grid* panel, enter **"YOU_TOP_RES_DEPTH_HORZ"** in the Map Data Set text field. On the Attribute pull-down menu, select **TVDSS**.

Click **OK**.

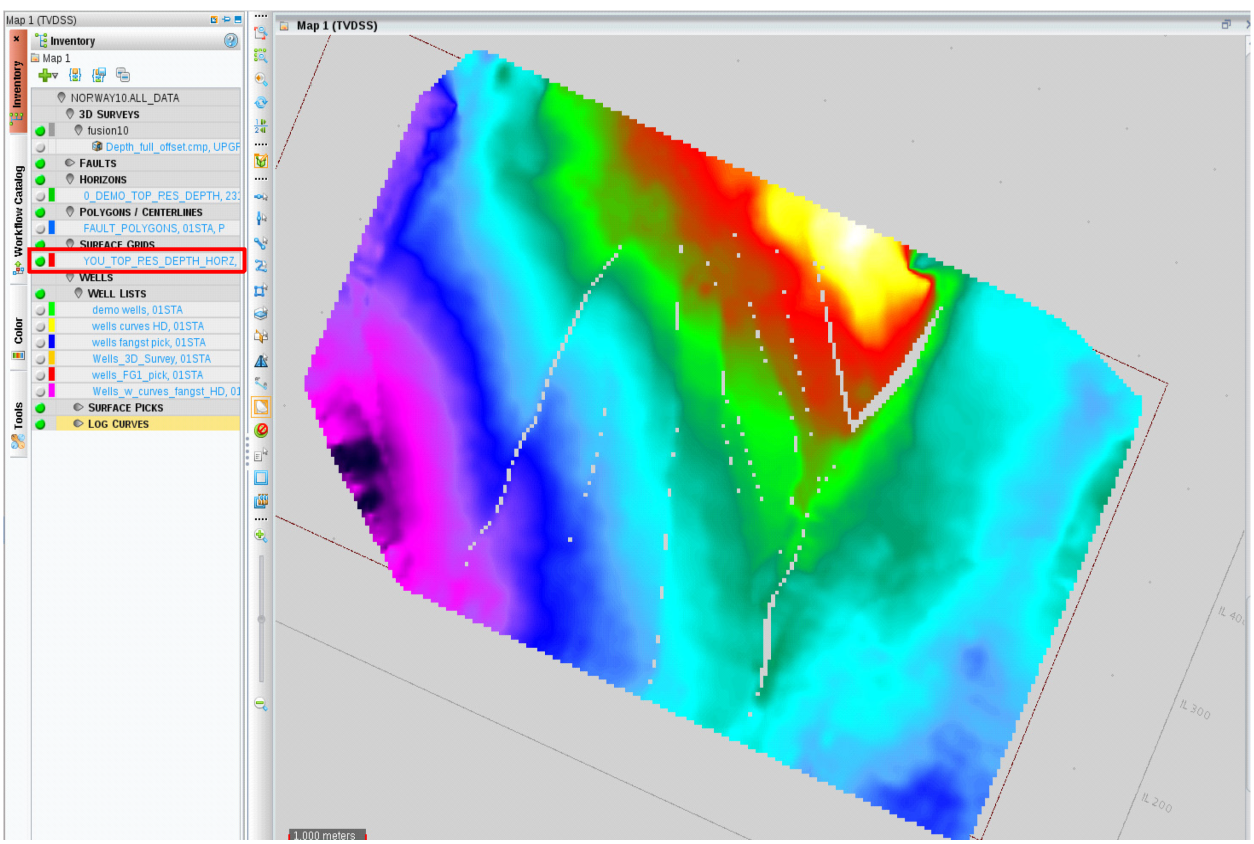


The new surface grid has now been saved to the OpenWorks database.

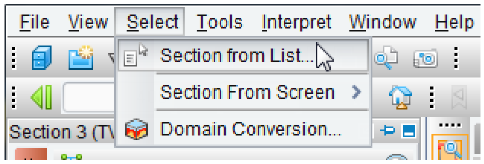
9. On the *Inventory* panel of the *Map* view, toggle on **0_DEMO_TOP_RES_DEPTH**.



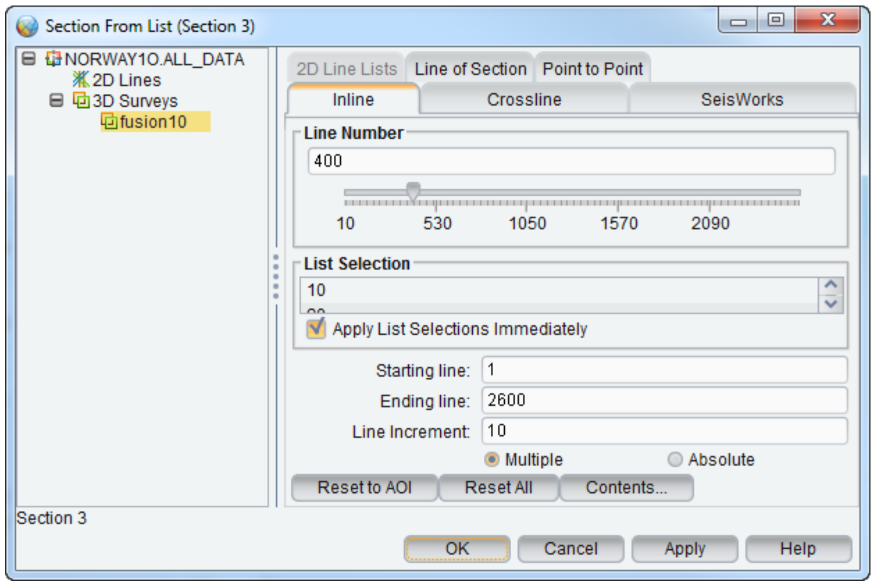
10. In the *Inventory* task pane, toggle on the surface grid **YOU_TOP_RES_DEPTH**. Observe the difference.



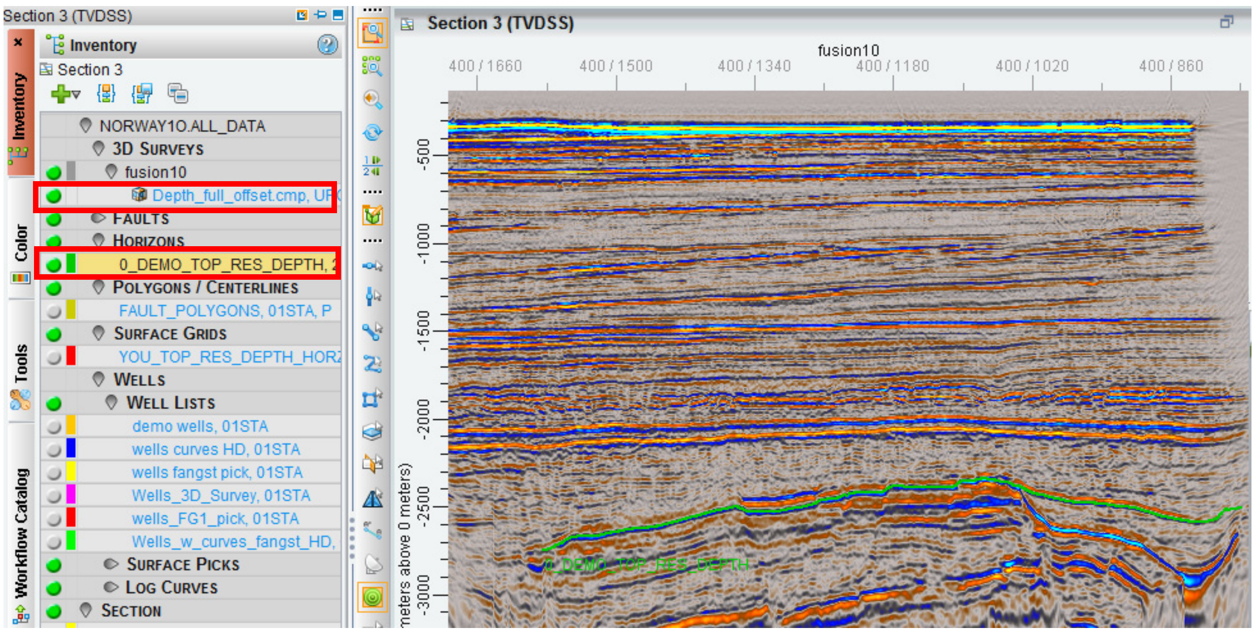
11. In the *Section* view, choose **Select > Section from List...** to open the *Section from List* dialog.



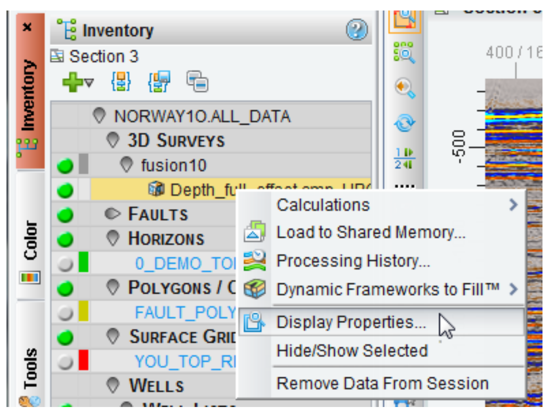
12. In the left task pane of the *Section from List* dialog select **3D Survey > fusion10.** On the *Line Selection* panel of the *Inline* tab, select **400.** Click **OK**.



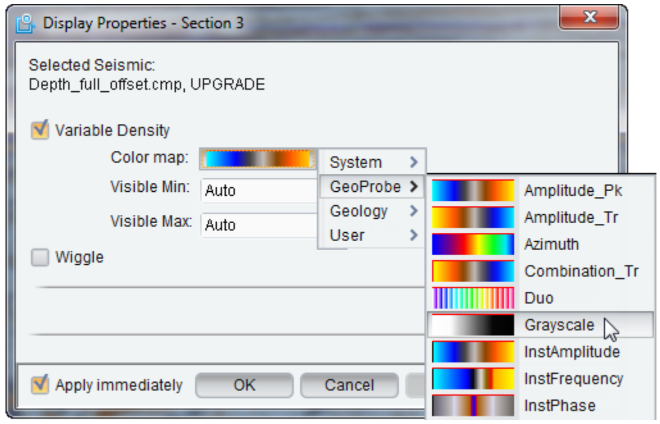
13. In the *Inventory* task pane of the *Section* view, toggle on **Depth_full_offset.cmp** and horizon **0_DEMO_TOP_RES_DEPTH**.



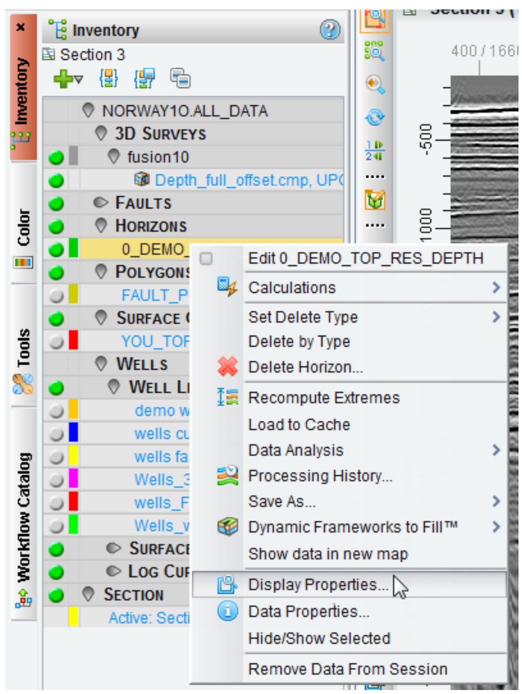
14. In the *Inventory,* put your cursor on **Depth_full_offset.cmp** and **MB3 > Display Properties**.



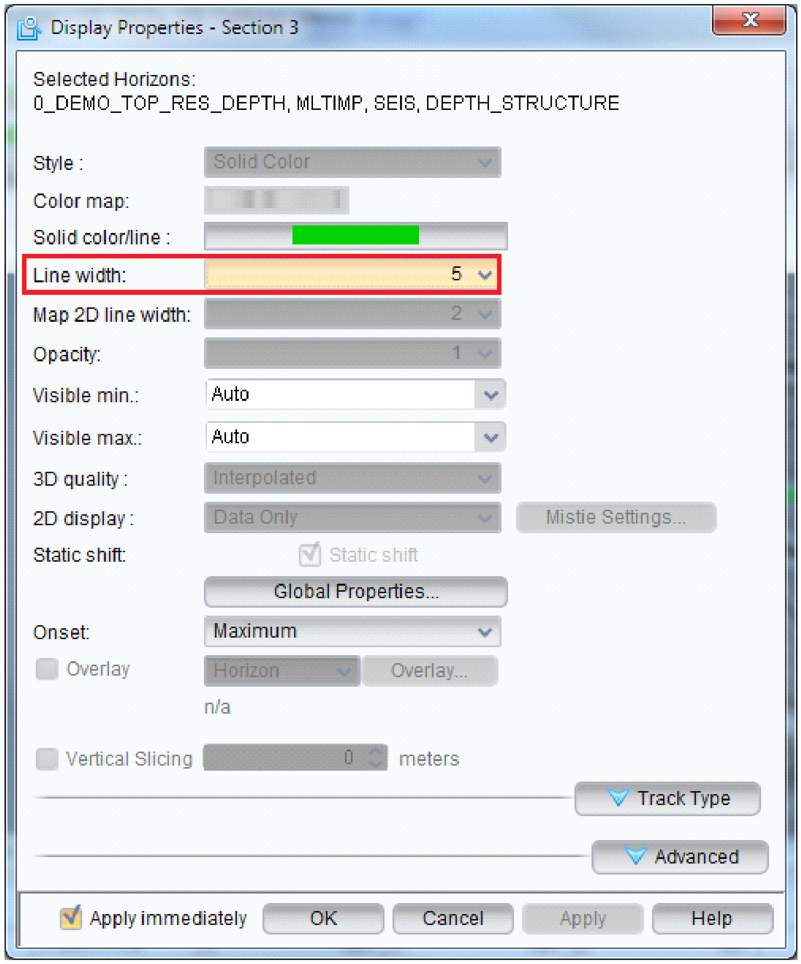
15. In the *Display Properties* dialog select **Color map > GeoProbe > Grayscale**, and click **OK**.



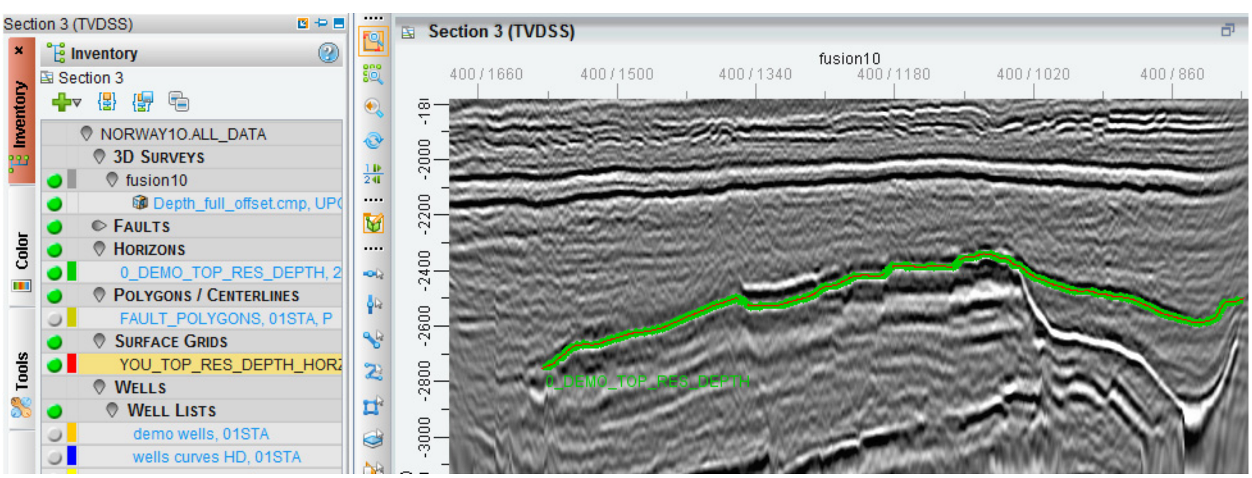
16. On the *Inventory* task pane, put your cursor on **0_DEMO_TOP_RES_DEPTH** and **MB3 > Display Properties.**



17. In the *Display Properties* dialog, select **5** on the Line width: pull-down menu and click **OK**.



18. In the *Inventory* task pane, toggle on the surface grid **YOU_TOP_RES_DEPTH_HORZ**.



Basic Contouring

1. On the *DecisionSpace* main menu, select **Tools > Grid and Contour.** The *Gird and Contour* dialog opens.

> **Note:**
>
> The horizon or surface grid can be new (created in the *Grid and Contour* dialog) or added to the session through the *Select Session Data* dialogue.

2. Select the *Contouring* operation tab.

3. Select an item from the *Horizons* or *Surface grids* input data tab.

4. Optionally, you can choose the *Polygon/Centerline* input data tab and select a set.

5. Select the default contouring settings.

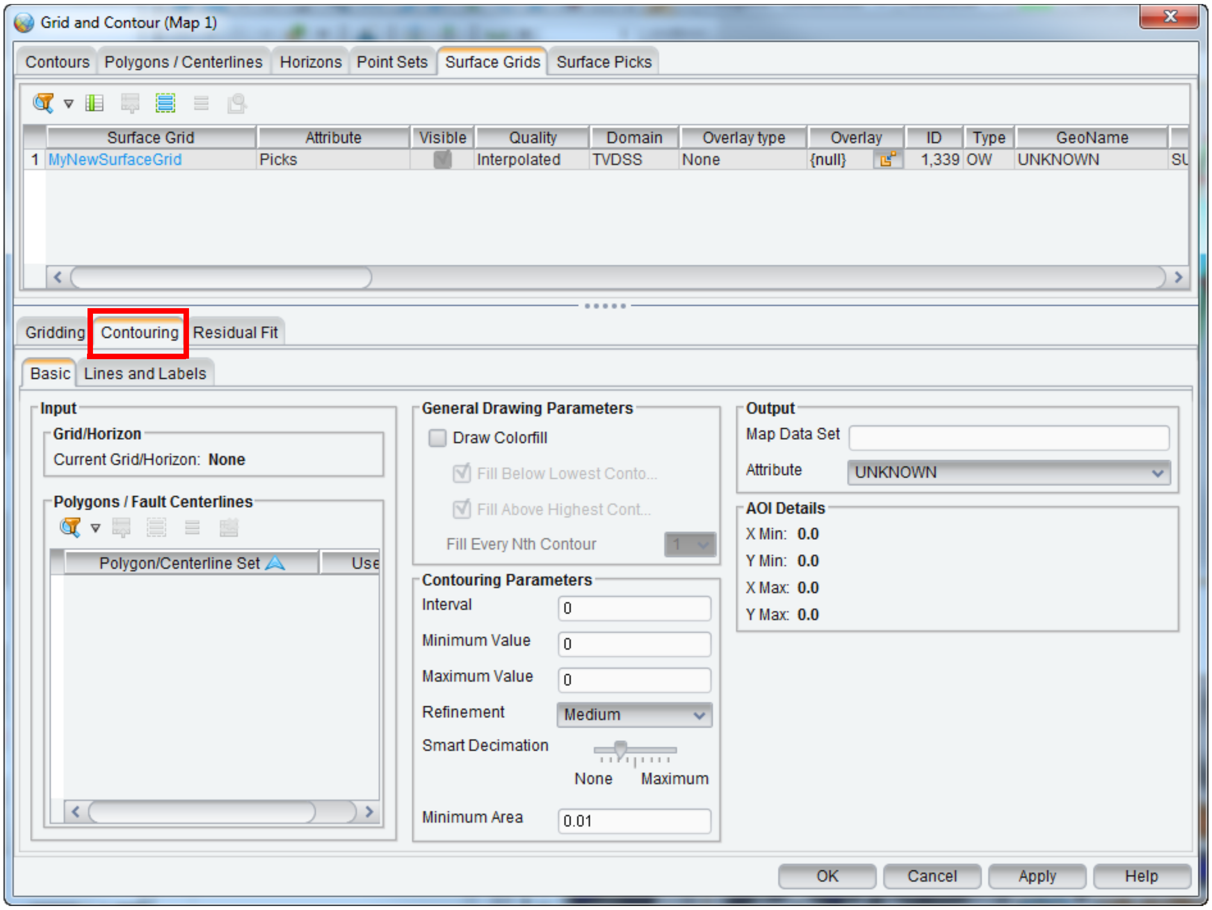
6. In the *Contouring* operation tab, click **Apply** or **OK**.

7. Examine the contour map. If necessary, adjust the settings in the *Contour Interpretation* task pane or create a new contour map.

To contour, you must select a horizon or surface grid from the input data selection panel. To contour with regards to fault offset, an associated Polygon/Centerline set must exist and be selected from the input data selection panel. All other settings are optional. If you do not specify any other options, the application uses pre-set or calculated default values for the other settings.

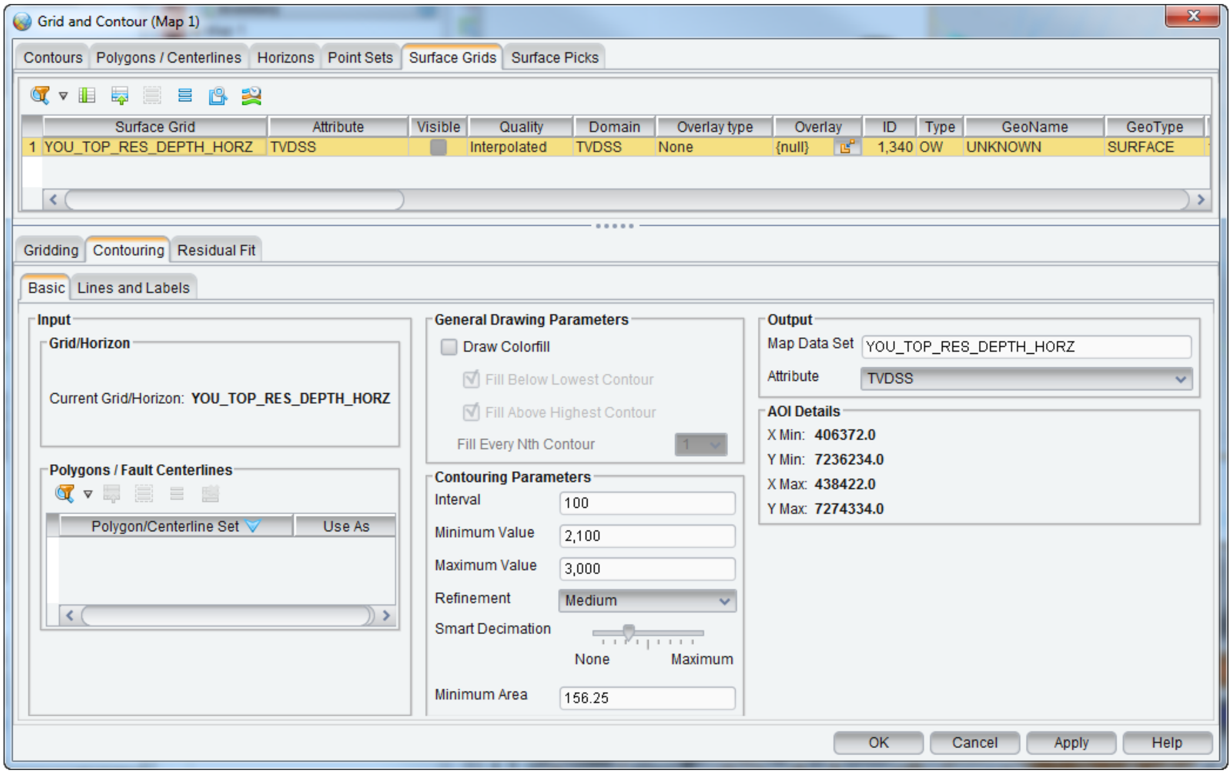
To select a filename to use as input, follow these steps.

8. Click the ***Contouring*** operation tab in the *Grid and Contour* dialog.



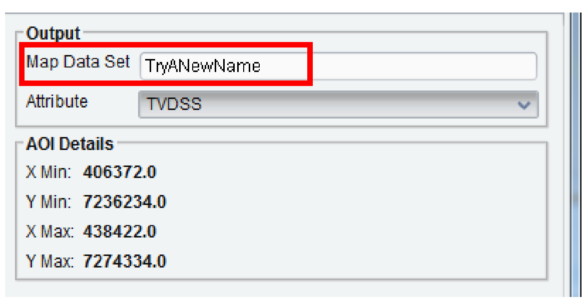
9. In the *Input Data Selection* panel of the *Grid and Contour* dialog, click the *Horizon* tab or the *Surface Grid* tab, and then select **Horizon** or **Surface Grid** from the list.

> **Note:**
>
> When you select a horizon or surface grid for contouring, values associated with the data item automatically appear in the AOI Details panel on the right side of the *Contouring* operation tab. These values cannot be edited.



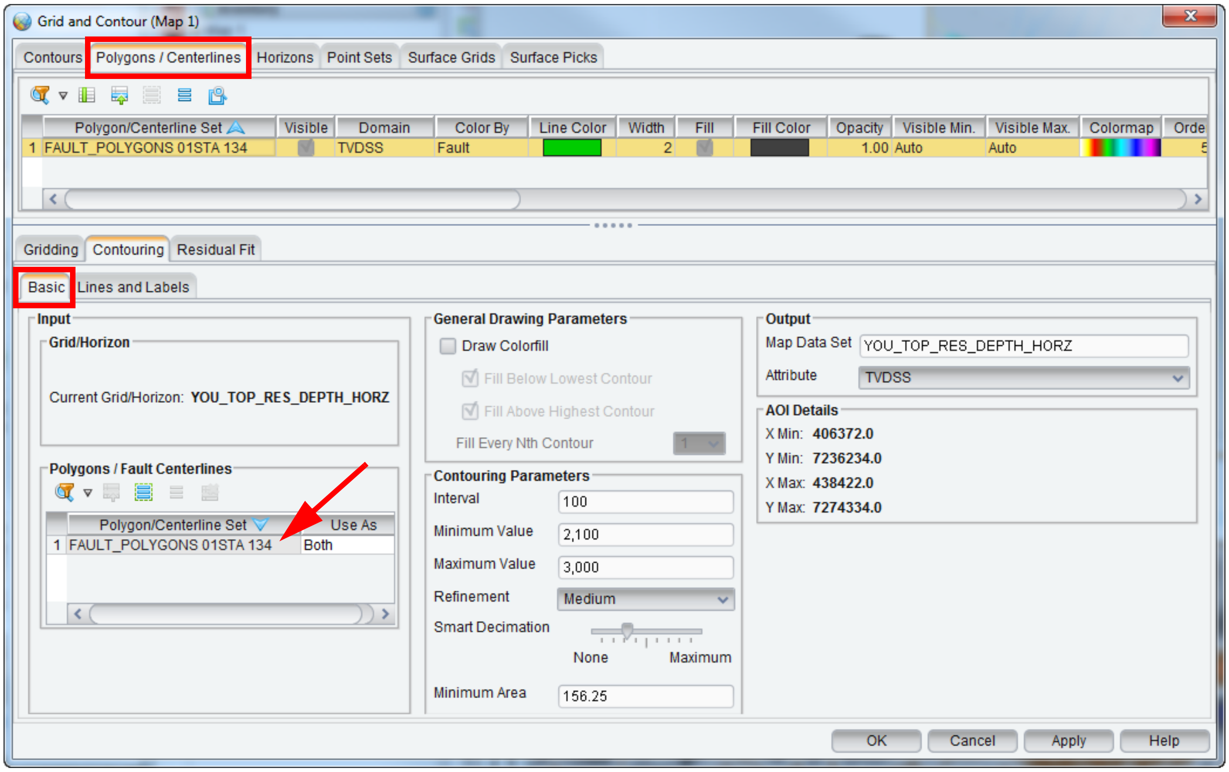
The horizon or surface grid name populates the Grid/Horizon text field in the *Contouring: Basic* tab. The program calculates default values for several parameters, based on the selected data.

Optional: The name of the new contour appears in the Map Data Set text field. By default, it is the name of the Grid/Horizon you chose. If you wish to change the default name, enter a new name over the current name.



10. To include polygons/centerlines in the contour map, click the ***Polygons/Centerlines*** tab in the *Input Data Selection* pane of the *Grid and Contour* dialog. Then select a polygon/centerline from the list.

The name of the selected polygon set appears in the *Polygons/Fault Centerlines* panel of the *Basic* tab.



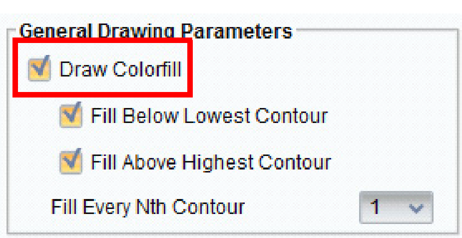
11. Accept the other parameter defaults and click **OK**.

Commonly Used Contour Settings

The *Contouring* tab in the *Operation* pane of the *Grid and Contour* dialog comprises a *Basic* tab and a *Lines and Labels* tab.

The Basic Tab

The *Basic* tab comprises several parameters.



General Drawing Parameters

To create a Colorfill Contour Map at the same time that you create a contour map, toggle on Draw Colorfill. The *Contours and Contour Maps* (colorfill) will appear as two items in all displays (*Inventory, Select Session Data,* and *Contents Data Properties* tab).

Contour Maps (colorfill) are not saved in the database or in session files.

Fill Below Lowest Contour

This option is dimmed until you toggle on Draw Colorfill. Then, if you want to add color fill below the lowest contour level, toggle on Fill Below Lowest Contour. The software applies color fill between the lowest contour line displayed and the lowest point on the map.

Fill Above Highest Contour

This option is dimmed until you toggle on Draw Colorfill. If you want to add color fill above the highest contour level, toggle on Fill Above Highest Contour. The software applies the color fill between the highest contour line drawn and the highest point on the map.

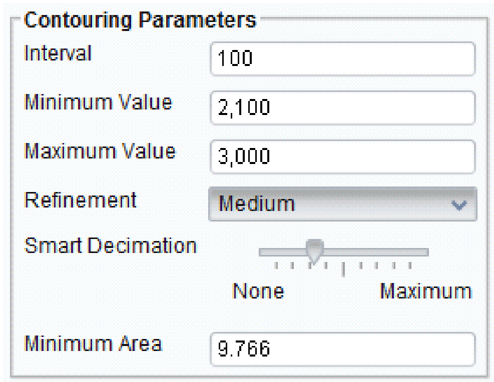
Fill Every Nth Contour

By default, each contour is filled with a unique value from the associated color map, as reflected by the Fill Every Nth Contour default value of one.

To change the number of contours filled with a single color value, select a new value in the *Fill Every Nth Contour* pull-down menu. For example, if you select 2, the application is set to use a single color to fill two consecutive contour intervals.

Contouring Parameters

To set the interval between contour lines, enter a value in the Interval field. The Interval value is expressed in surface units. For example, if map surfaces are displayed in meters and the Interval value is 5, the interval (or vertical distance) between contour lines is 5 meters.



The application calculates the default value for the interval, based on the Z value range of the input data.

- For a typical map, the interval is designed to display contours at standard, round-numbered Z values, such as every 100 feet.
- For a map with a wide range of Z values, the interval between contours may be larger.
- For a map with a very narrow range of Z values, standard Z values may not be applicable.

The Interval field in the *Basics* tab is connected to the Interval field for Regular Contours in the *Line and Labels* tab. If you change the Interval value in the *Basics* tab, it updates to the new value for Regular Contours in the *Lines and Labels* tab. The Interval cannot be adjusted in the *Line and Labels* tab. Regular Contours Interval, Min, and Max will always be grayed out. The Interval value for Hachure, Minor, and Major Contours updates proportionately.

Setting the Z Value Range for Contour Lines

To set the Z value range for contour lines, type new values in one or both of these text fields:

- The Minimum Value text field
- The Maximum Value text field

These settings define the lowest and highest Z values that can be contoured. The settings are expressed in Z units, the measurement units used for depth in the current map.

For a typical map, the default values are the Visible Min. and Visible Max. values for the selected data file, which are rounded to an appropriate value.

Determining the Contour Smoothness

To change the options that affect the smoothness of the contour lines, use the following functionality:

- Refinement
- Smart Decimation

Choosing a Refinement Setting

To select a refinement setting, click the Refinement button and select from the pull-down menu. The Refinement setting controls the amount of smoothing applied to the contour lines. The coarser the refinement level, the fewer the nodes at sharp corners. You can use any of five refinement options, ranging from Very Coarse (no refinement) to Very Fine (maximum refinement).

In most cases, the default Refinement setting is Medium. The Refinement setting does not determine the number of nodes used to create the contour. It determines how the nodes are spread along the contour.

When to Change the Refinement Level

In most cases, you will adjust the refinement level after you create an initial contour map and notice a problem or want to make a comparison map. You may want to adjust the refinement level in these circumstances:

- For an overly long execution time, try a coarser setting.

- For jagged contour lines, use a finer setting.

    — If the dataset is rough (the values of adjacent data points vary widely at a given scale), you need a finer Refinement setting to avoid jagged contours.

— If the dataset is relatively smooth (so the data follows a smooth arc when viewed in a cross section), you do not need a fine Refinement setting.

— If a contour line slightly misses a known target (such as a well), try a finer setting.

- For polygons/centerlines included in the contour map, you may need to use a finer Refinement setting. As a general thumb, the refined grid cell should be less than two pixels wide when polygons/centerlines are included.

Working with Smart Decimation

Adjust the Smart Decimation slider to intelligently simplify contour lines by eliminating points along the straighter sections of the contours. This process increases rendering performance and makes contour editing less tedious.

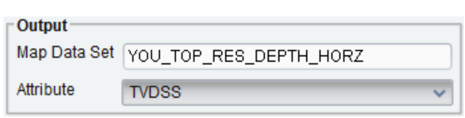
Move the slider to a high value to increase rendering speed. Note, however, that higher values also tend to produce rougher contours, thus reducing the integrity of the contour lines.

Minimum Area

The Minimum Area value specifies the size of the smallest contour polygon that can be created. This value is expressed in surface units squared (such as square feet or square meters).

The default minimum contoured area is based on the size of the input grid cells. The default minimum contoured area becomes smaller as the size of the refined grid cell decreases. The default value is designed to eliminate contour polygons that are a fraction of the area of a refined grid cell to minimize erroneous bull’s eyes.

To retain more small contoured areas, decrease the Minimum Area value.



Output

The default output for the contour set is the Map Data Set; that is, the name of the input grid or horizon and an Attribute. Click in the Attribute field to select a different attribute from the pull-down menu.

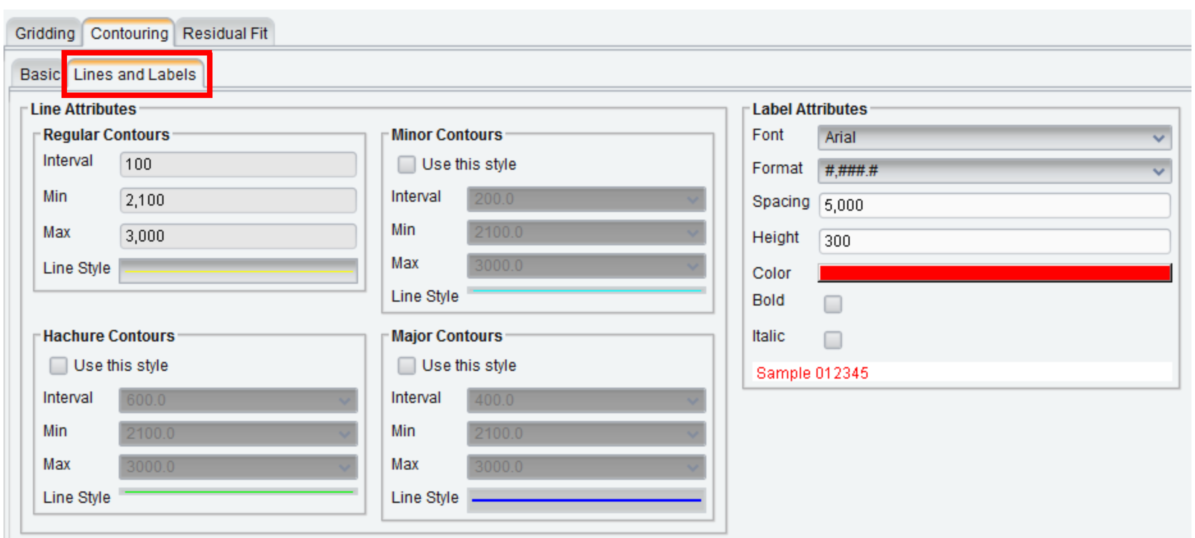
After you have created the contour, its name and associated Attribute appear in the *Inventory* task pane, *Select Session Data* dialog, and *Contents Data Properties*.

About the AOI Details Panel

When you select a surface grid or horizon for contouring, values associated with the data item automatically appear in the AOI Details panel. You cannot edit these minimums and maximums.

Lines and Labels Tab

On the *Lines and Labels* tab you can specify values that affect the appearance of your contour map.



To display the tab, click the *Contouring* operation tab at the bottom of the *Grid and Contour* dialog, then click the *Lines and Labels* tab. The tab turns orange to show it is active. The *Lines and Labels* tab contains panels for each kind of contour line.

- **Regular Contours** — Every contour in the contour set will take on the Regular setting, but can be overridden by Major, Minor, and Hachure lines.

- **Hachure Contours** — These contours have perpendicular hatch marks, indicating dip. This line type overrides Regular lines.

- **Minor Contours** — These contour lines override Regular and Hachure line types.

- **Major Contours** — These override all other line types.

Only the Regular Contours are turned on by default. To use any of the other kinds of contours, toggle on Use this style in their respective panels. You can use any combination of line styles at any time.

Setting the Interval

If you change the Interval in the *Basic* tab, the new value also appears for Regular Contours in the *Lines and Labels* tab. At the same time, the Interval values for Hachure, Minor, and Major Contours update proportionately. You can override these Interval defaults. You cannot change the Regular Contour Interval value in the *Lines and Label* tab. To change the Interval in the *Lines and Labels* tab, type in a new value or use the pull-down menu.

You can choose only the multiples of the Regular Contour interval.

Min and Max Values

The Min and Max values in the *Lines and Labels* tab work in a way similar to the Intervals. If you change the Minimum or Maximum Value in the *Basic* tab, the new value immediately appears in all the Min or Max fields for all the line styles in the *Lines and Labels* tab.

To change a Min or Max value in the *Lines and Labels* tab, follow the steps below:

1. Click in the **Min** or **Max** text field. A pull-down menu appears, showing values that are within the minimum and maximum values in increments determined by the Regular Contour Interval.

2. Choose one of the values and click **Apply** or **OK**.

Line Style

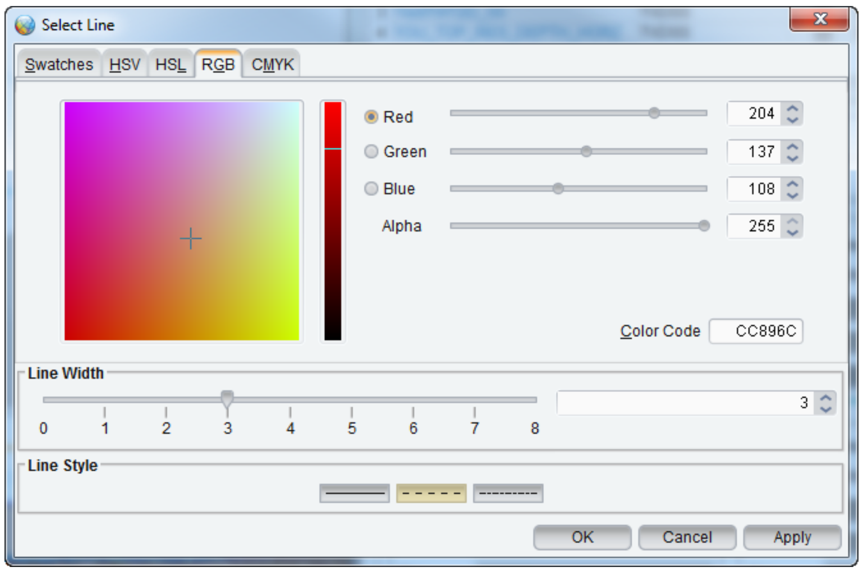
You can change the Line Style for all the contour types. To change the Line Style:

1. Click in the **Line Style bar** for any contour type. A *Select Line* dialog opens. Click a **new color** from the Swatches, HSV, HSL, RGB, or CMYK tabs.

2. Move the Line Width **slider** or click the **arrow** in the text field next to it to change contour line width. Your selection is instantly reflected in both areas.

3. The line style default is a solid line, but you can also select a dashed line or a dot-dash line. Click in the appropriate **box**.

4. In the *Select Line* dialog, click **OK** to see the changes appear next to Line Style.



Label Attributes

Settings for Label Attributes affect all contours. You can change the following:

- **Font** — By default, the Font button is set to Arial. To change it, click the arrow in the text field and select a new font from the pull-down menu.

- **Format** — You can select how many decimals appear in the annotation and whether commas will divide numbers in the thousands. Select from the pull-down menu.

- **Spacing** — You can change the amount of spacing in the selected measurement between labels. If you are using metric, a label will appear every 5,000 meters along each contour line. Select from the pull-down menu.

- **Height** — The height, in the selected measurement, of each label. If using metric, each label will appear 300 meters tall. Select from the pull-down menu.

- **Color** — To change the default label color, click the color bar.

  — When the *Choose Color* dialog appears, click a new color from Swatches, HSV, HSL, RGB, or CMYK tabs.

  — Click OK.

The new color appears in the Color bar.

- **Bold** — Check the box to make labels appear in boldface.

- **Italic** — Check the box to make labels appear in italics.

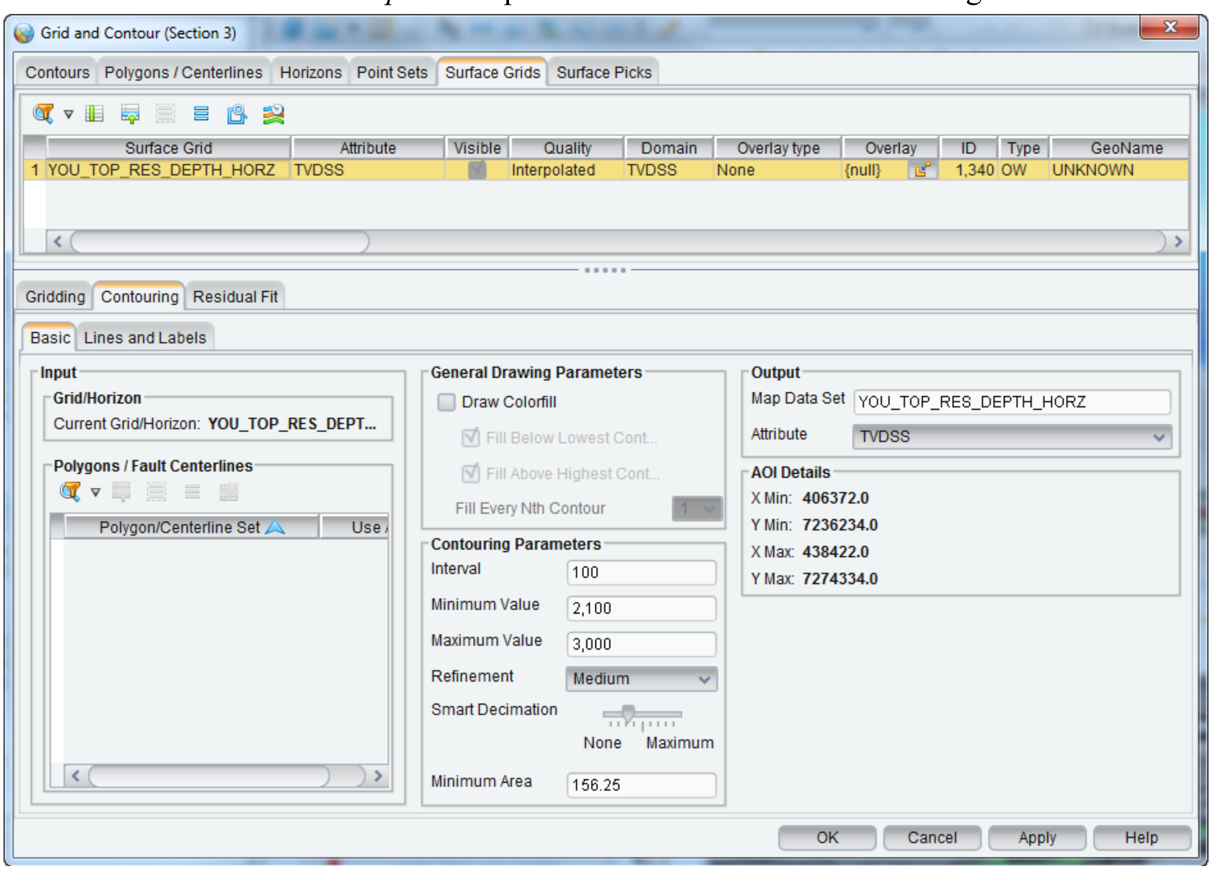
Exercise A.3: Creating Contour Maps

DecisionSpace Geosciences software enables you to create contours from surface grids or horizons. Using surface grids as input typically produces smoother results. You can create sets in the *Grid and Contour* dialog.

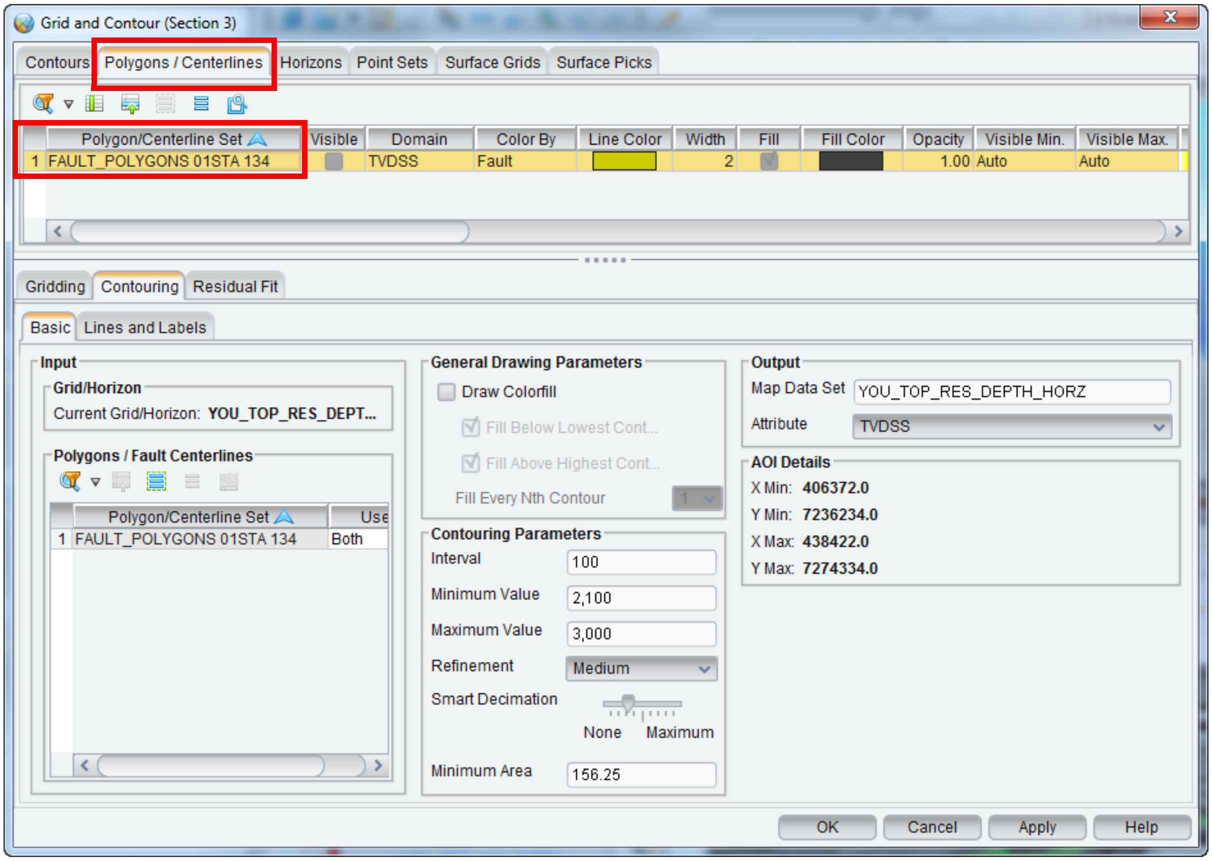
DecisionSpace Geoscience surface grids and contour sets are automatically saved to the OpenWorks database.

1. Ensure the *Grid and Contour* dialog is still open. In the *Grid and Contour* dialog, select the *Contouring* operation tab.

2. Click the *Surface Grids* tab in the *Input Data Selection* panel of the *Grid and Contour* dialog. Select the surface grid you saved in the previous exercise (**YOU_TOP_RES_DEPTH_HORZ**). The name of the selected grid is now visible under Grid/Horizon in the *Operation* panel of the *Grid and Contour* dialog.



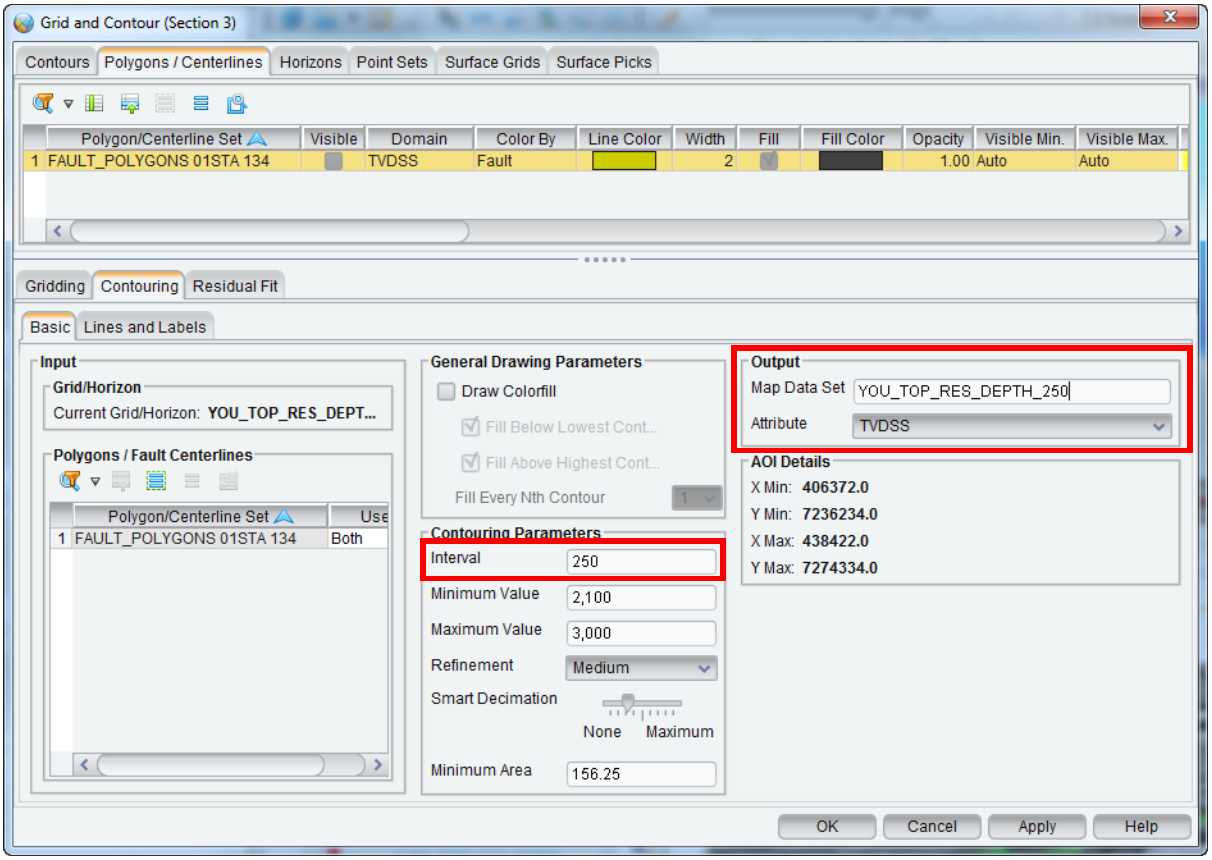
3. Click the ***Polygons/Centerlines*** tab of the *Input Data Selection* panel, on the *Grid and Contour* dialog. Select the **FAULT_POLYGONS 01STA 134** polygon set.



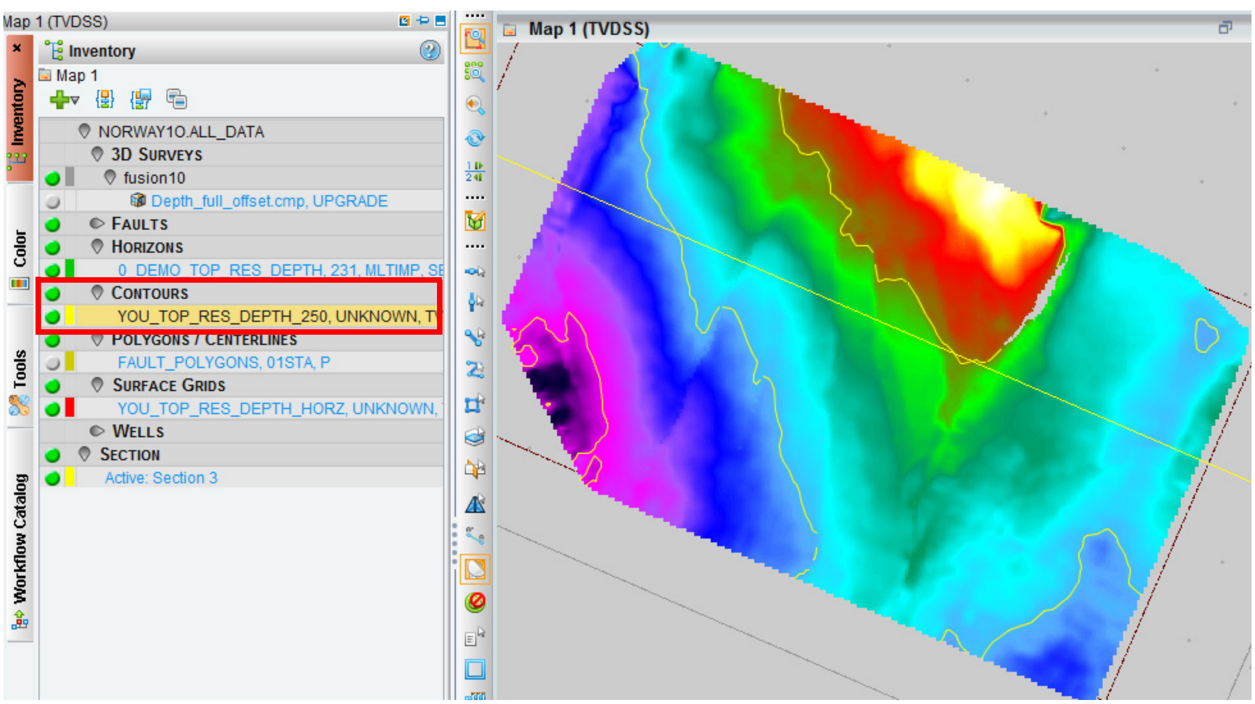
Now make the following changes in the *Contouring* tab of the *Grids and Contour* dialog:

- In the Contouring Parameters panel, enter "**250**" in the Interval field.
- In the *Output* panel, enter "**YOU_TOP_RES_DEPTH_250**" in the Output Data Set field.

Keep all other default settings and click **Apply**.



4. In *Map* view, toggle on Contour **YOU_TOP_RES_DEPTH_250**.

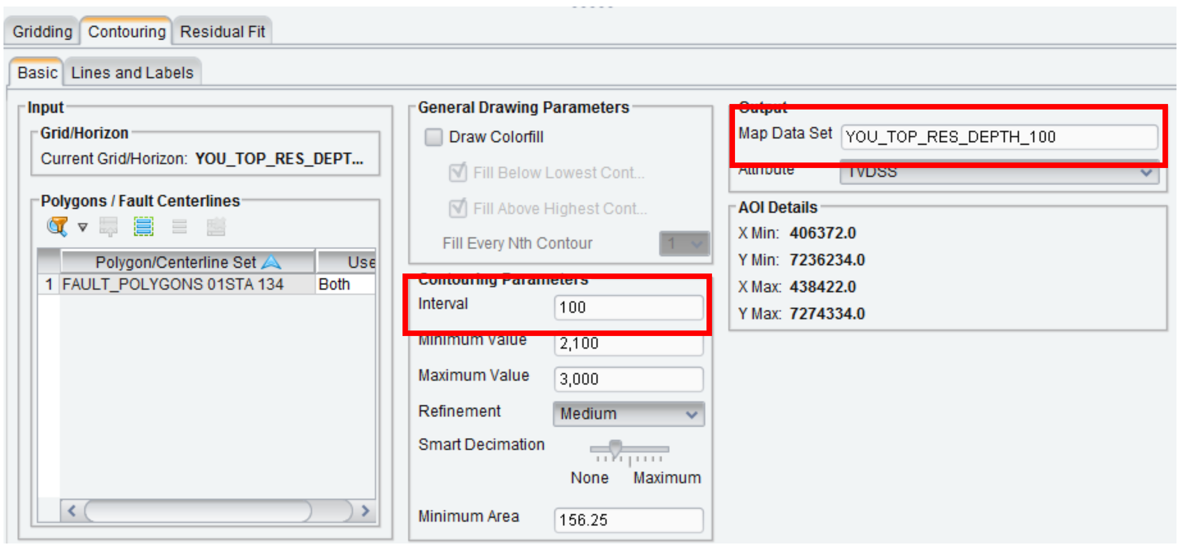


5. In the *Contouring* operation tab of the *Grid and Contour* dialog, set the following:

   In the *Basic* tab:

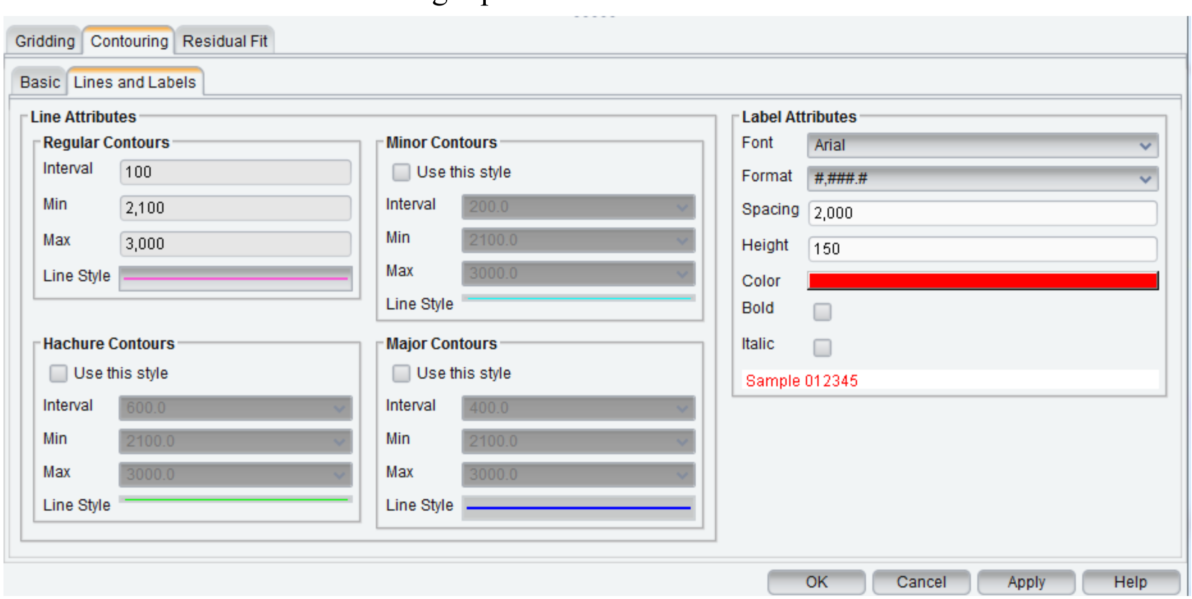
   - In the *Contouring Parameters* panel, enter **"100"** in the Interval text field. Leave the Minimum Values, Maximum Values, and Refinement settings unchanged.

   - In the *Output* panel, enter **"YOU_TOP_RES_DEPTH_100"** in the Map Data Set text field.



In the *Lines and Labels* tab:

- In the *Line Attribute* panel, change the **Regular Contours** color to light pink and line width 2.



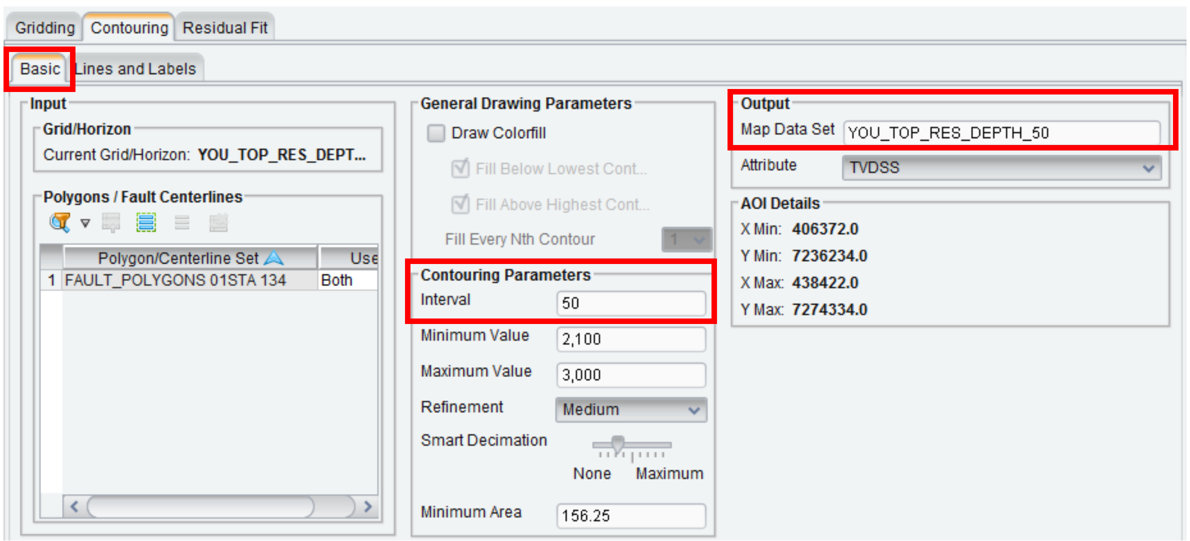
- Click **Apply**.

6. Modify the existing setting to create a new contour set.

   On the Basic tab:

   - In the *Contouring Parameters* panel, enter **"50"** in the Interval text field.

   - In the *Output* panel change the Map Data Set text field to **"YOU_TOP_RES_DEPTH_50"** and click **Apply**.

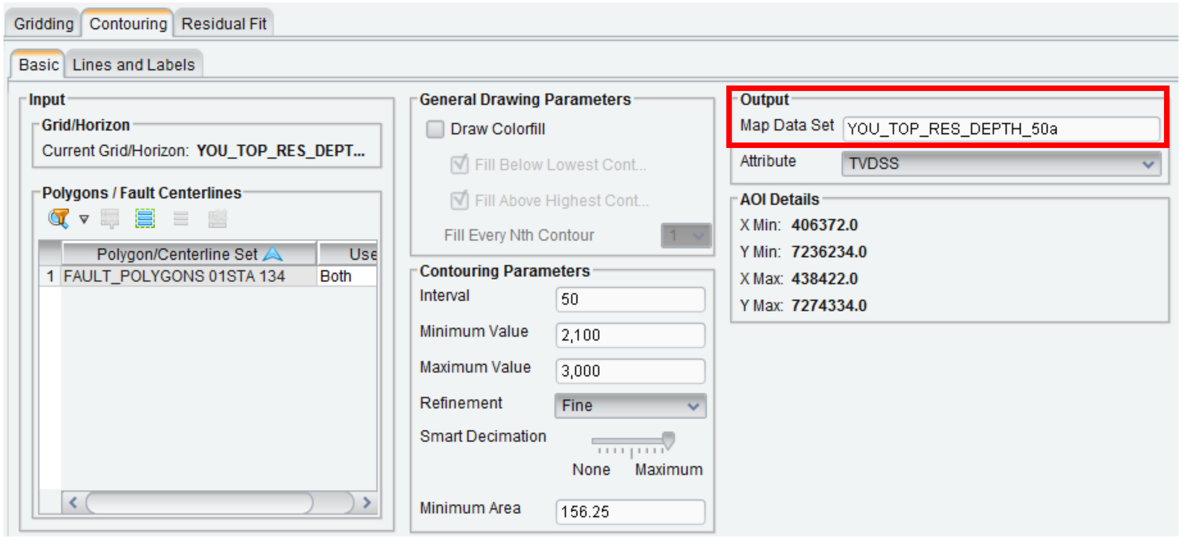


7. In the *Operation* pane of the *Grid and Contour* dialog, select the *Contouring* tab. Change the following.

   In the *Basic* tab:

   - In the *Contouring Parameters* panel, enter **"50"** in the Interval text field.

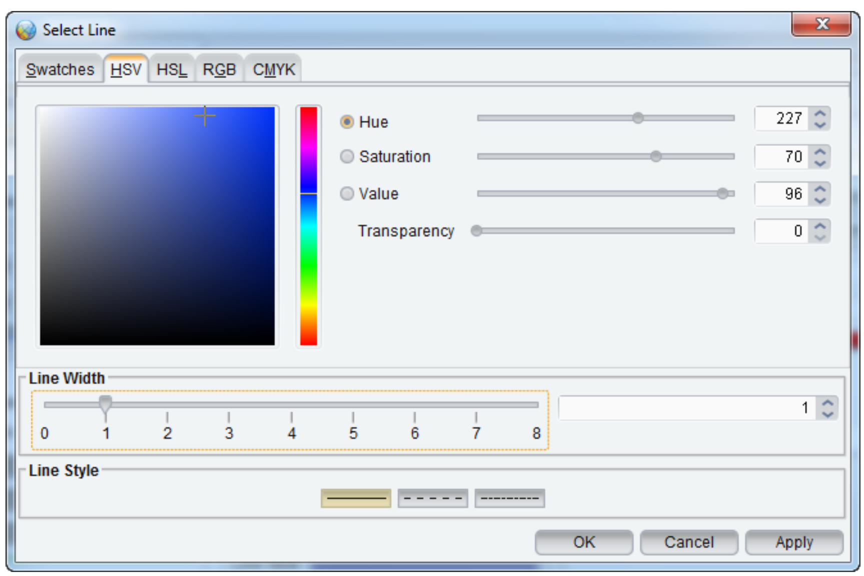
- In the *Output* panel, enter “**YOU_TOP_RES_DEPTH_50a”** in the Map Data Set text field.



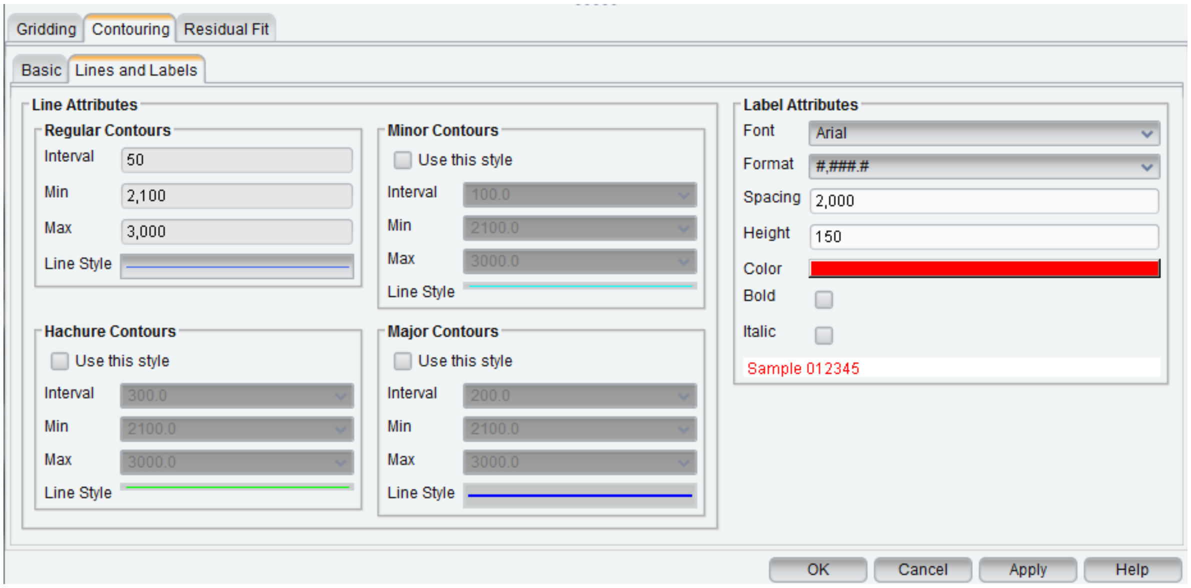
On the *Lines and Labels* tab:

8. In the *Line Attributes* panel, change the Regular Contours color to something different (**pink-red** or **blue**).

9. In the *Regular Contours* panel, select Line Style **blue** with line width **1**.



10. Click **Apply**.

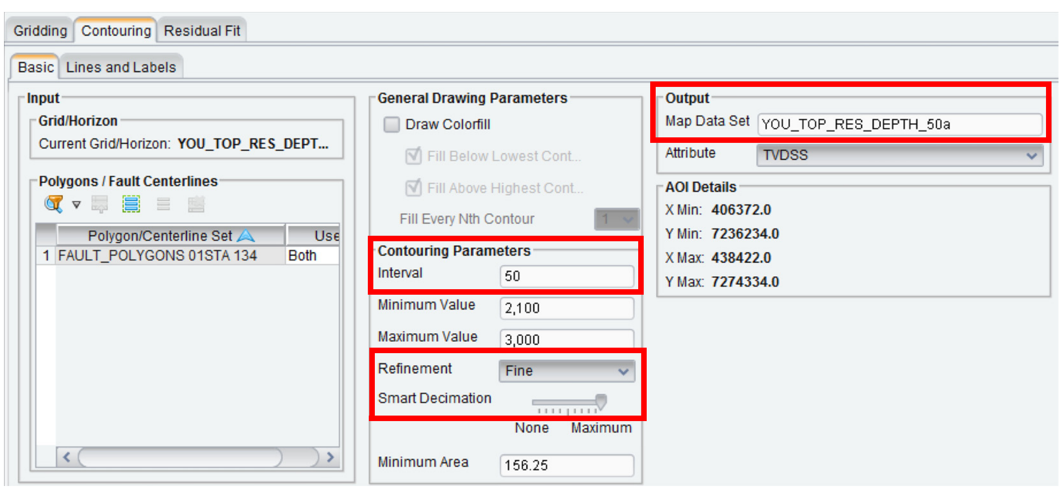


11. In the *Operation* panel of the *Grid and Contour* dialog, select the ***Contouring*** operation tab. Change the following.

    On the *Basic* tab:

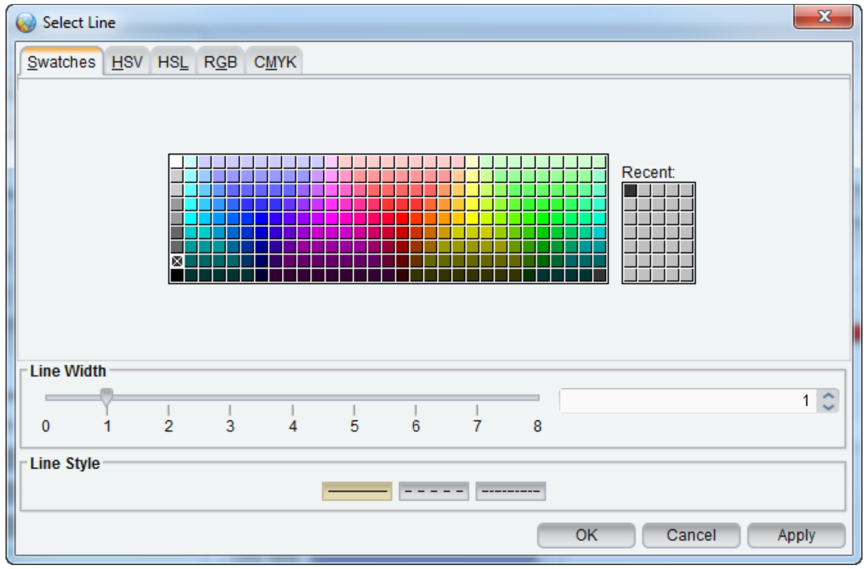
    - In the *Contouring Parameters* panel, enter **"50"** in the Interval text field. On the Refinement pull-down menu, select **Fine** and set Smart Decimation to **Maximum**.

    - In the *Output* panel, enter **"YOU_TOP_RES_DEPTH_50a"** in the Map Data Set text field.



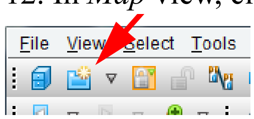
In the *Lines and Labels* tab:

- In the *Line Attribute* panel, change the Regular Contours color to Line Style **dark gray** with line width **1**.

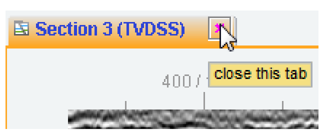


- Click **Apply**.

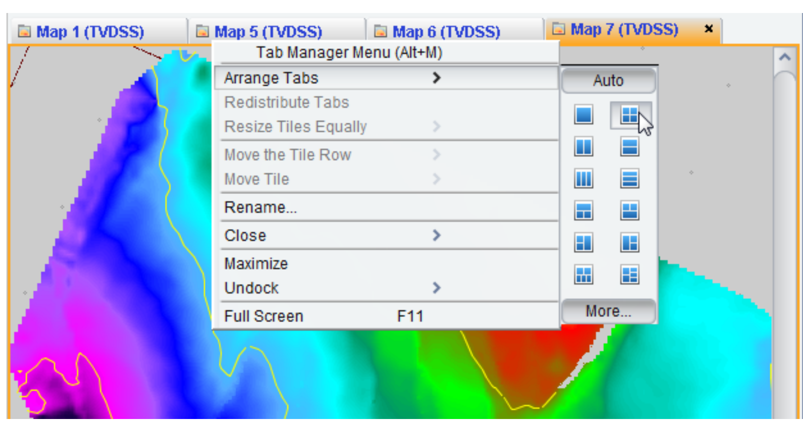
12. In *Map* view, click the **New Tab** ( ) icon three times.

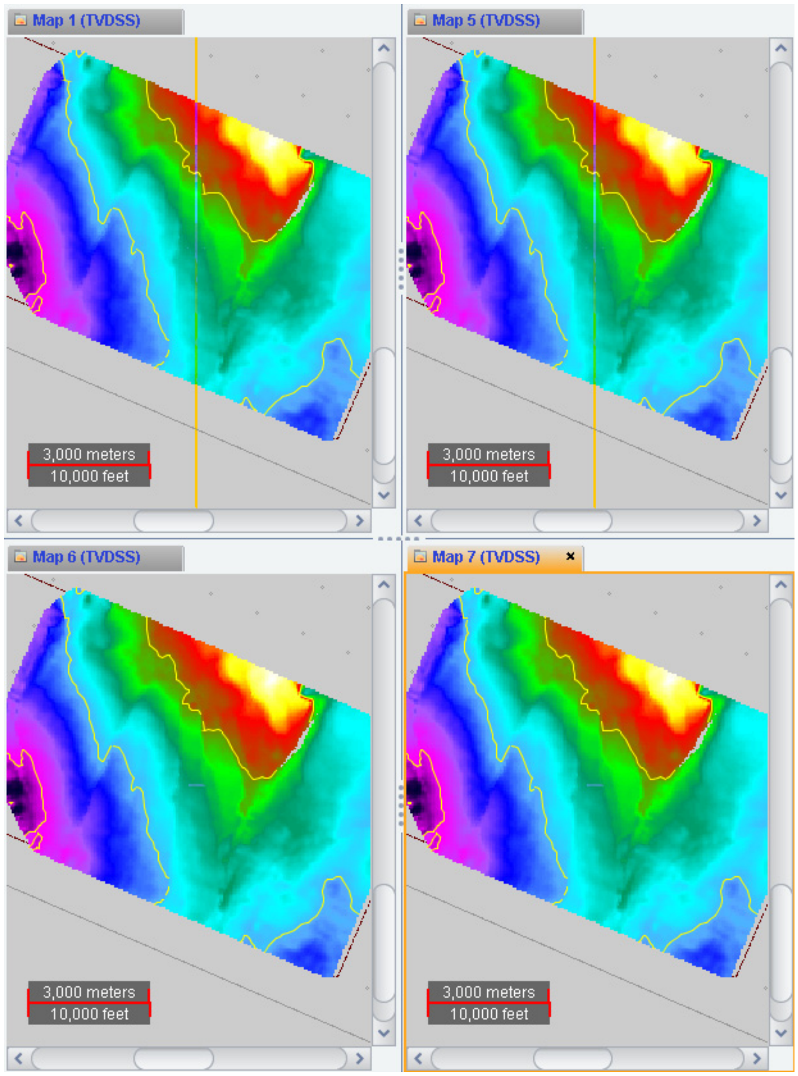


13. Close the *Cube* and *Section* views.



14. With your cursor on the *Map* view tab, **MB3 > Arrange Tabs.** Select the four-panel (2x2) display ( ).

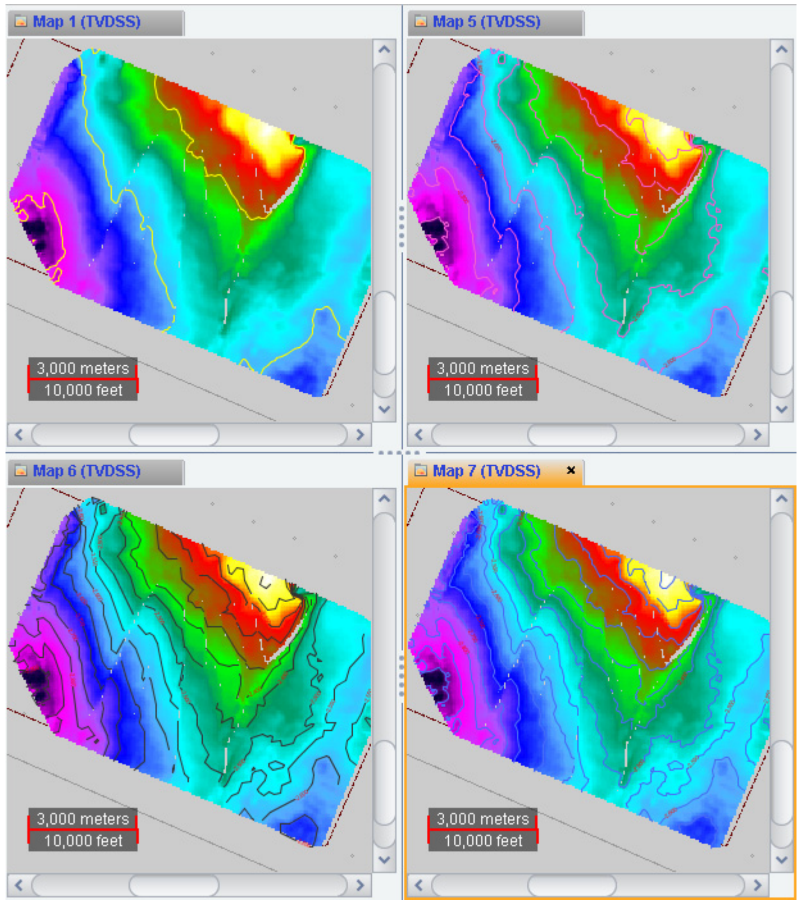




15. Make the following changes in each *Map* view.

- In the top left *Map* view, toggle on surface grid **YOU_TOP_RES_DEPTH_HORZ** and contours **YOU_TOP_RES_DEPTH_250**.
- In the top right *Map* view, toggle on surface grid **YOU_TOP_RES_DEPTH_HORZ** and contours **YOU_TOP_RES_DEPTH_100.**
- In the bottom left *Map* view, toggle on surface grid **YOU_TOP_RES_DEPTH_HORZ** and contours **YOU_TOP_RES_DEPTH_50a**. This contour set was created using maximum decimation and fine refinement.

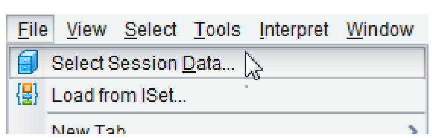
- In the bottom right *Map* view, toggle on surface grid **YOU_TOP_RES_DEPTH_HORZ** and contours **YOU_TOP_RES_DEPTH_50**.



Compare the results of the contour maps you made. Maximize or zoom in on any map you want to examine more closely.

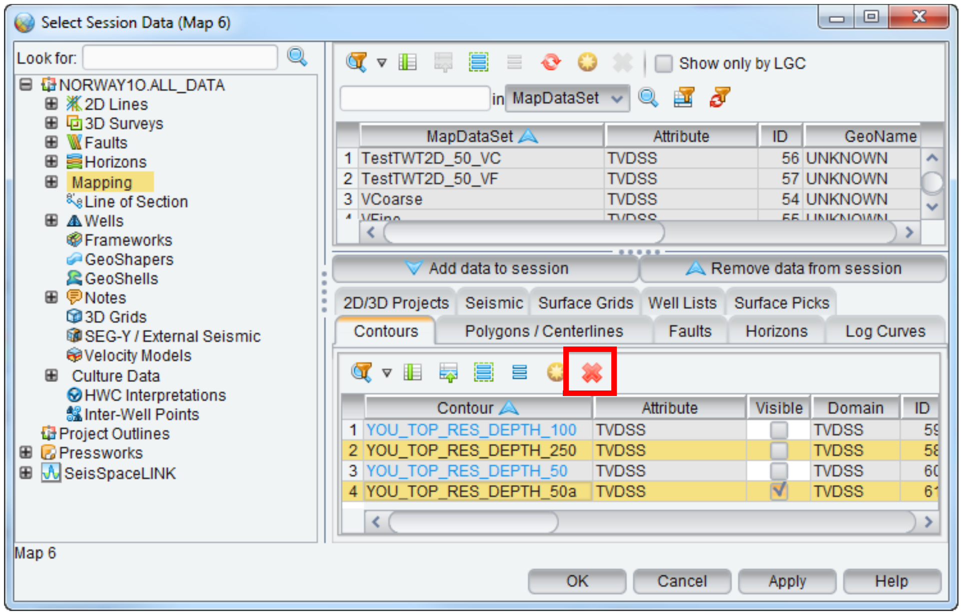
16. **Close** three of the *Map* views.

17. Select **File > Select Session Data.**

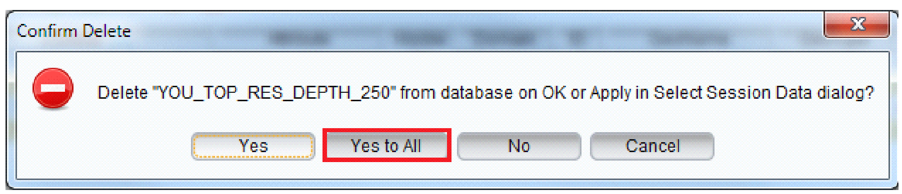


18. In the *Select Session Data* dialog, click the ***Contours*** in-session tab, then select **YOU_TOP_RES_DEPTH_250** and

YOU_TOP_RES_DEPTH_50a. Click the **Delete Selected Contour Set** ( ) icon.



19. A *Confirm Delete* dialog opens; click ***Yes to All***.



20. In the *Select Session Data* dialog, click **OK.**

Note:

This command removes the data from the session and deletes the files from the database, not just the session. You can also remove the following types of data in this manner:

— Surface grids
— Faults
— Horizons
— Contours

Contour interpretation

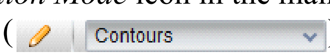
You will perform contour interpretation only in *Map* views. One of its major features is the ability to update surface grids from edited contours. The *Contour Interpretation* task pane enables you to perform these activities:

- Select and deselect contours on a map.
- Select a contour line by elevation.
- Create contour sets by means of a dialog.
- Manually add contours directly to a map.
- Edit contours.
- Update surface grids from edited contours.
- Save contours to and retrieve contours from the OpenWorks database.
- Change line and label attributes.
- Use shortcut keys for convenience.

The following sections describe these options.

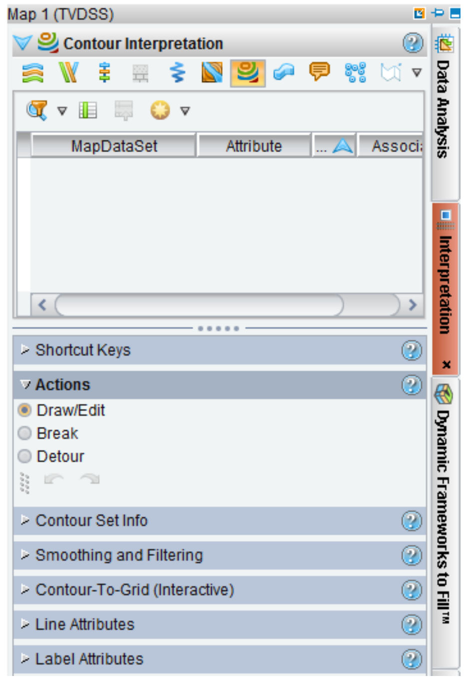
Entering Contour Interpretation Mode

You can enter contour interpretation by either of the following methods:

- Click the *Interpretation Mode* icon in the main tool bar and select the **Contours** (

)on the pull-down menu.
- On the *Interpretation* task pane, select the **Contour Interpretation** () icon.

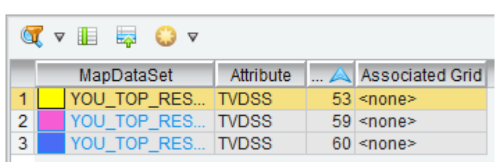
The *Contour Interpretation* task pane is displayed.



The *Contour Interpretation* task pane contains the following panels, which are discussed in detail the following paragraphs:

- Table of contour sets
- Shortcut Keys
- Actions
- Contour Set Info
- Smoothing and Filtering
- Contour-To-Grid (Interactive)
- Line Attributes
- Label Attributes

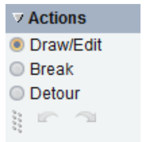
Table of Contour Sets— All contour sets toggled on in the current view are listed in this panel and are available for interpretation. When you create a new contour set, it is automatically added to the list.



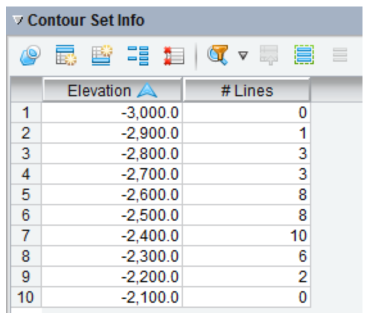
	MapDataSet	Attribute	...	Associated Grid
1	YOU_TOP_RES...	TVDSS	53	<none>
2	YOU_TOP_RES...	TVDSS	59	<none>
3	YOU_TOP_RES...	TVDSS	60	<none>

Shortcut Keys — Provides a quick reference for shortcut keys.

Actions — Includes toggles for the modes Draw/Edit, Break, and Detour, plus the Undo and Redo icons.

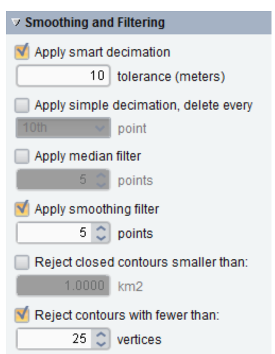


Contour Set Info — Contains a table showing one entry for every Z value in the contour set and a count of how many lines are at that Z value. When you are in contour interpretation mode and you select a row, the software graphically selects all of the contours for that Z value.

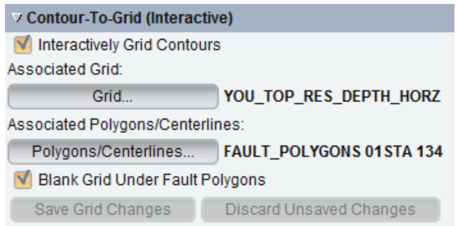


	Elevation	# Lines
1	-3,000.0	0
2	-2,900.0	1
3	-2,800.0	3
4	-2,700.0	3
5	-2,600.0	8
6	-2,500.0	8
7	-2,400.0	10
8	-2,300.0	6
9	-2,200.0	2
10	-2,100.0	0

Smoothing and Filtering — Various options for decimating, smoothing, and filtering out contour lines.



Contour-To-Grid (Interactive) — By default, Interactively Grid Contours is toggled off in this panel. When you toggle it on, each contour edit triggers a regridding of the affected area in the associated surface grid. If that surface grid is already displayed, it is redrawn when the regridding is completed.



To associate a contour set and its underlying grid, click the Grid button and select it from the resulting *Choose Grid* dialog.

Line Attributes — Supports the four types of lines listed below. Regular is selected by default. Use the pull-down menu to select another kind of line.

- **Regular** — Every contour in the contour set will take on the Regular setting, but can be overridden by Major, Minor, and Hachure lines.
- **Major** — Overrides all line types.
- **Minor** — Overrides Regular and Hachure line types.
- **Hachure** — These lines have perpendicular hatch marks that indicate dip. This line type overrides Regular lines.

You can set the following for each of the above kinds of lines:

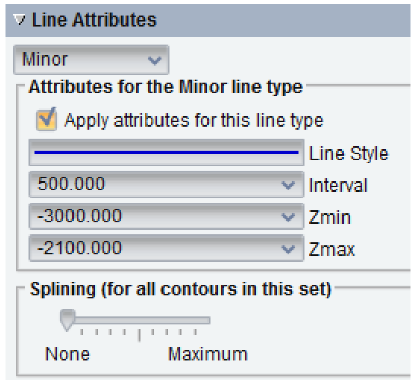
- **Line Style** — Includes color, style, and width. Click in the Line Style box to open a dialog wherein you can select new settings.
- **Splining (for all contours in this set)** — The higher the value, the more smoothing will be applied to the contour lines.

You can set the following for Major, Minor, and Hachure lines:

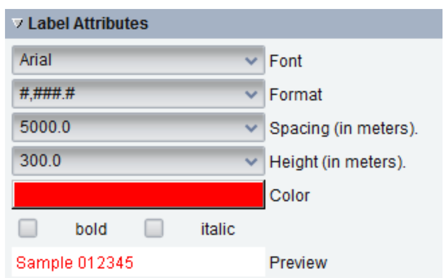
- **Apply attributes for this line type** — Indicates whether to display the selected line type.
- **Interval** — The contour interval to use for the selected line type.

- **Zmin** — The starting Z value for which the selected line type will be applied.

- **Zmax** — The ending Z value for which the selected line type will be applied.



Label Attributes



In the *Label Attributes* panel, you can choose the following commands from four pull-down menus:

- **Font** — Provides a pull-down menu wherein you can select a typeface for contour annotation.

- **Format** — Offers a pull-down menu wherein you can choose the number of decimals and whether to use a comma for values in the thousands for the contour annotation.

- **Spacing (in feet or meters)** — Lets you adjust the spacing between labels on the *Map* in feet or meters per contour.

- **Height (in feet or meters)** — Allows you to change the size of the annotation according to how many feet or meters tall the typeface will appear. Adjust Height and Spacing together for the best annotation results.

- **Color** — Click the Color bar and from displayed palette, choose a new color for the labels.

- **Bold or Italic**— toggle on Bold or Italic to show those in the display annotation.

- **Preview** — Displays a preview of how your annotation will appear (disregards spacing and height).

Selecting and Deselecting Contours on the Map

Various colors and symbols on contours and their meaning:

- **Red symbols** — This contour line is available for editing, but is not currently selected.

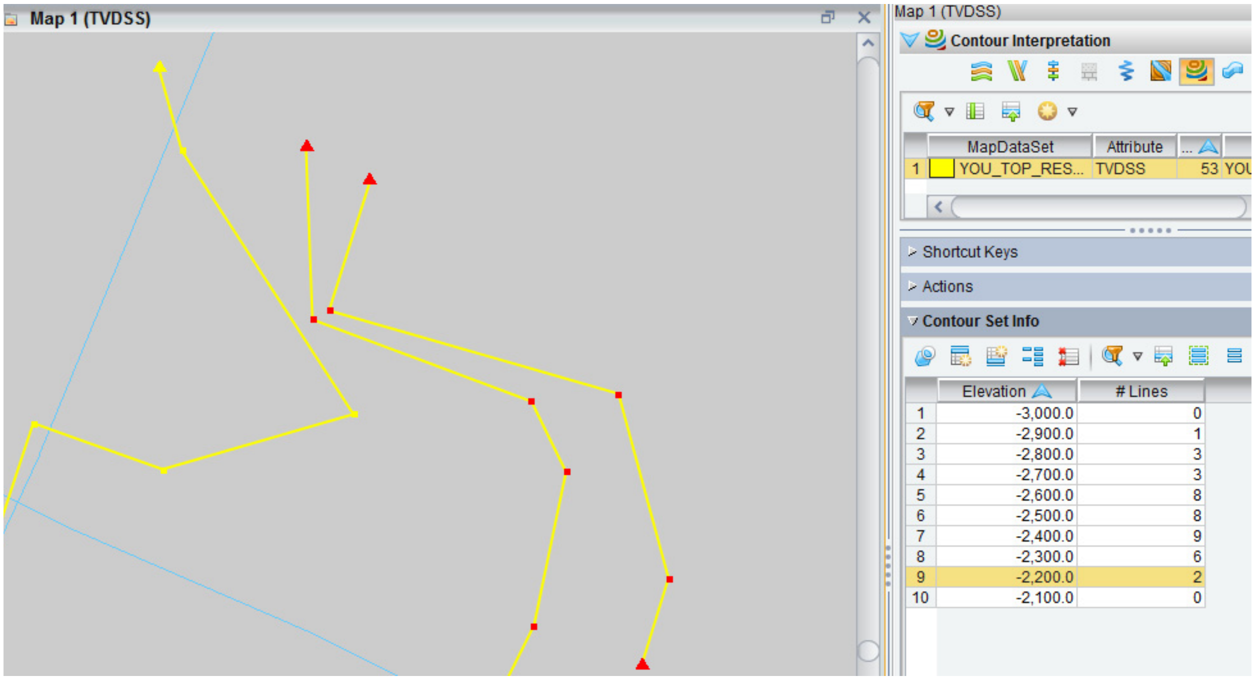
- **Yellow symbols** — This contour line is active and ready for editing.

- **Filled in square** — node/handle that has been digitized and is a set point until moved or deleted.

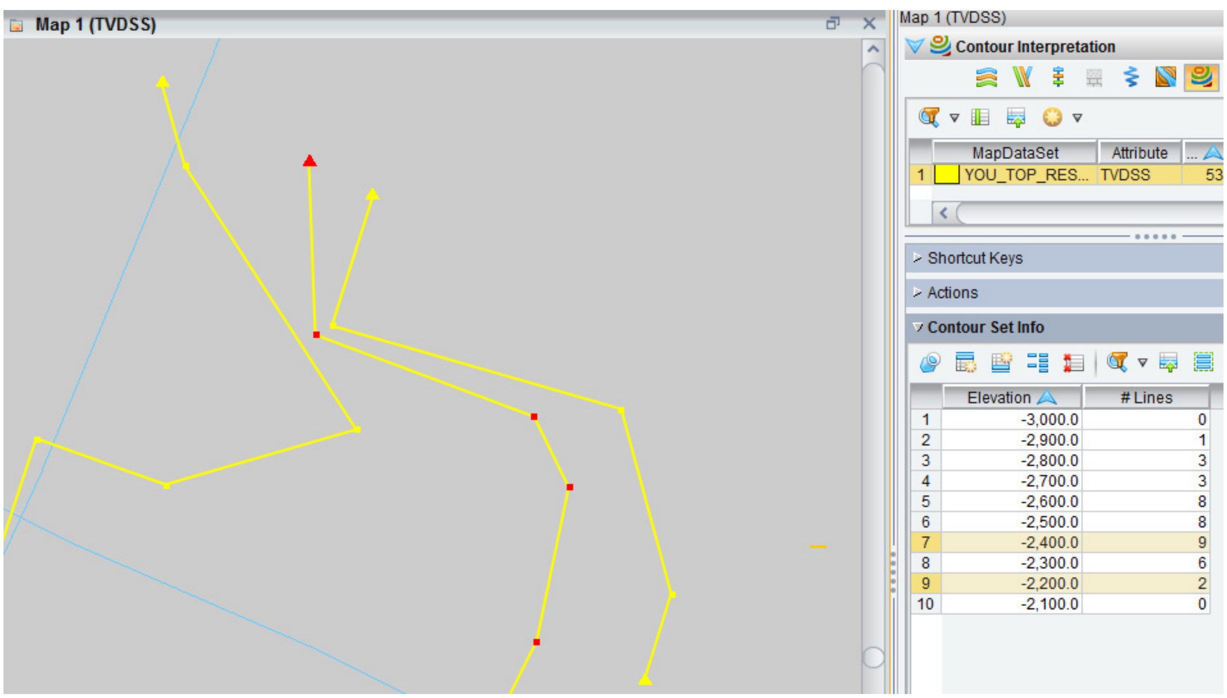
- **Filled in Triangle** — The end of a contour line. Can be grabbed to create free-hand extension of line or to be dragged to another triangle to join two contour lines

Whenever you select a contour line for editing, its nodes turn yellow. To select a contour line, follow one these options:

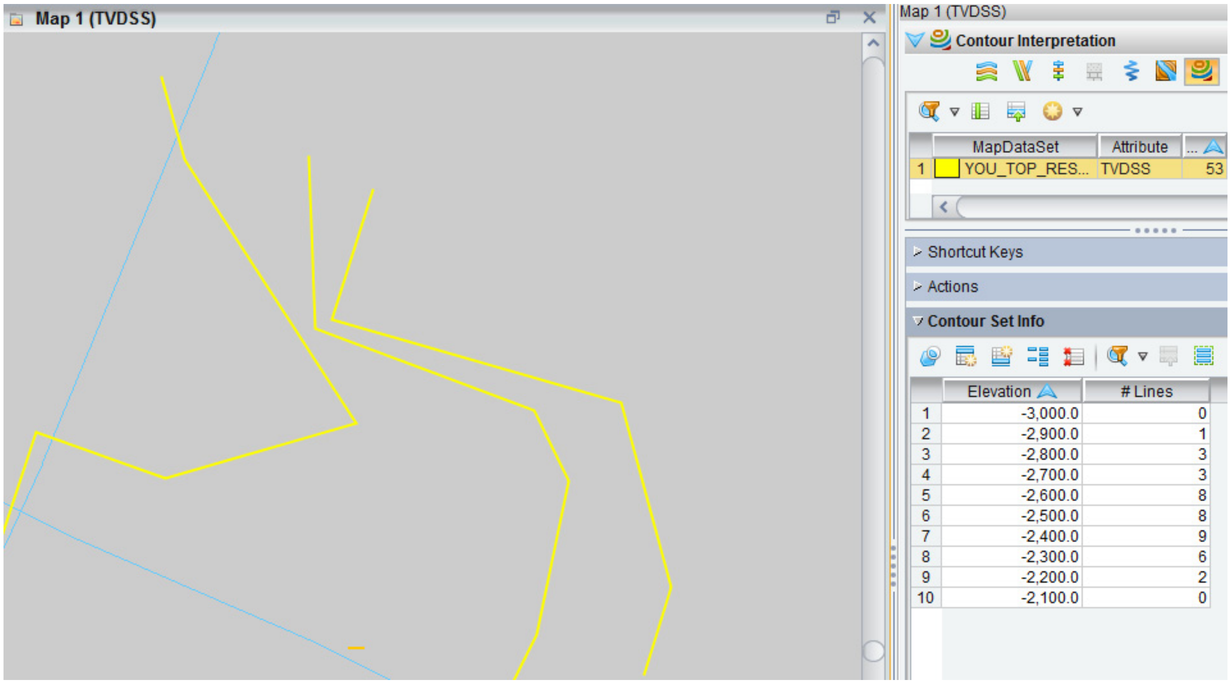
1. On the *Contour Set Info* panel of the *Contour Interpretation* task pane, select a row to make all contours at the Z-value active.



2. In *Map* view, click a **contour line**; to select more than one line at one time, put your cursor on a contour line and <**Ctrl**>+**MB1**.



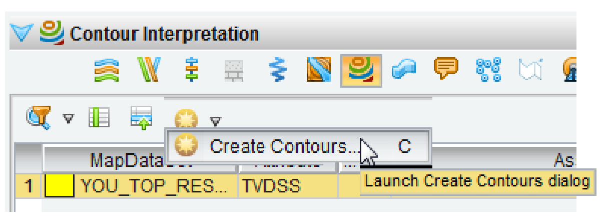
3. To deselect all contours, in *Map* view **MB2** away from the contours.



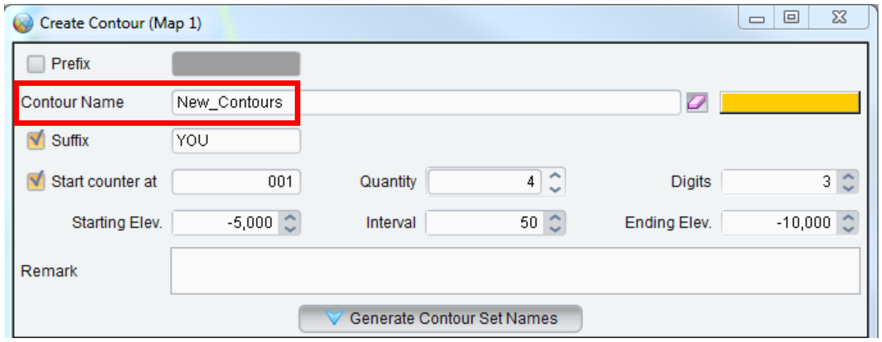
Creating Contour Sets

DecisionSpace Geosciences enables you to create new contour sets that you can manually populate.

1. In the *Contour Interpretation* task pane, click the **Create Multiple Contours** icon (the white arrow beside the Create New Contour).

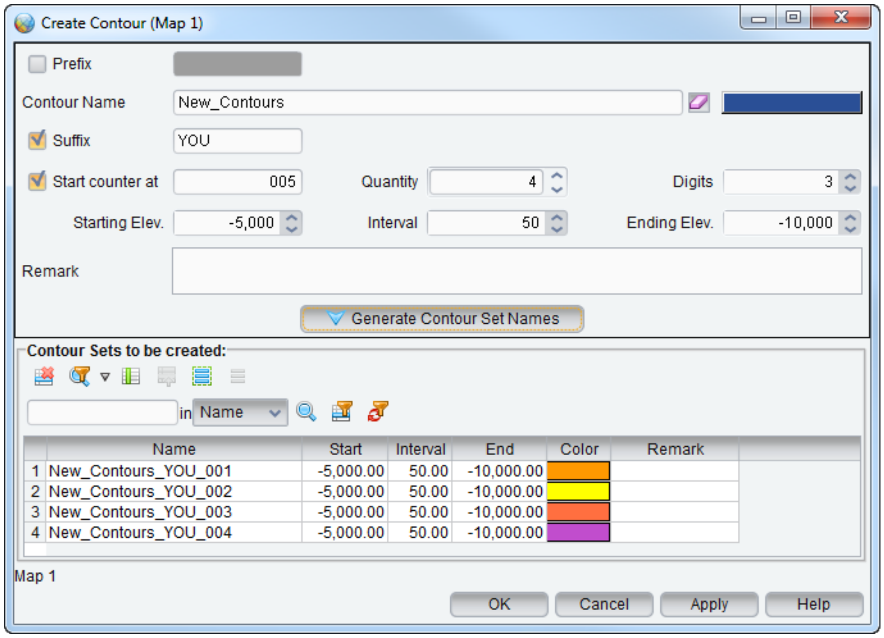


2. In the *Create Contour* dialog, make the following changes.



- In the Contour Name text field enter **"New_Contours"**.
- Toggle on Suffix to use the session Interpreter ID. You can change this field as you wish.
- If you want to create more than one contour set, toggle on **Start counter at**. When you check this box, the Start contour at and Quantity text fields become active. Start counter at adds a consecutive numerical suffix to however many contour sets you specify in the Quantity field. You can specify the number of digits you want the suffix to contain by using the up and down arrows in the Digits text field.
- Accept or change the values for the **Starting Elev. (Elevation)**, **Interval**, and **Ending Elev. (Elevation).**
- Click the **Generate Contour Set Names** button.

- The names of the new, and empty, contour sets appear in the bottom half of the dialog box. Click **Apply** or **OK**.



These new contour sets have been saved to the OpenWorks database and are automatically loaded to your session.

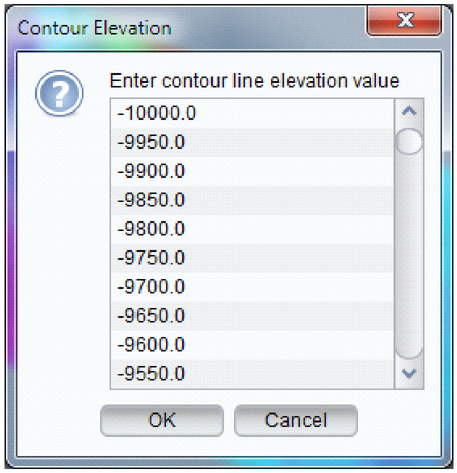
Adding Contours Manually

To manually add a contour, follow these steps.

1. Click a **place** on the *Map* view where you want to add the first contour line. As you click other points (nodes), the software draws a straight line from one node to the next.

2. **MB2** to end the contour line. To end the contour lines and enclose the contour (create a bull's eye), **<Ctrl>+MB2** while adding nodes.

3. The *Contour Elevation* dialog appears. Choose a **Z value** from the list of choices.



> **Note:**
>
> If you select a row in the *Contour Set Info* panel before drawing a new contour, the elevation value on that row will automatically be assigned to the new contour.

4. To discard all nodes from the new contour line, press **<Esc>** before you close the contour line.

Editing Contours

You can edit contours in the following ways:

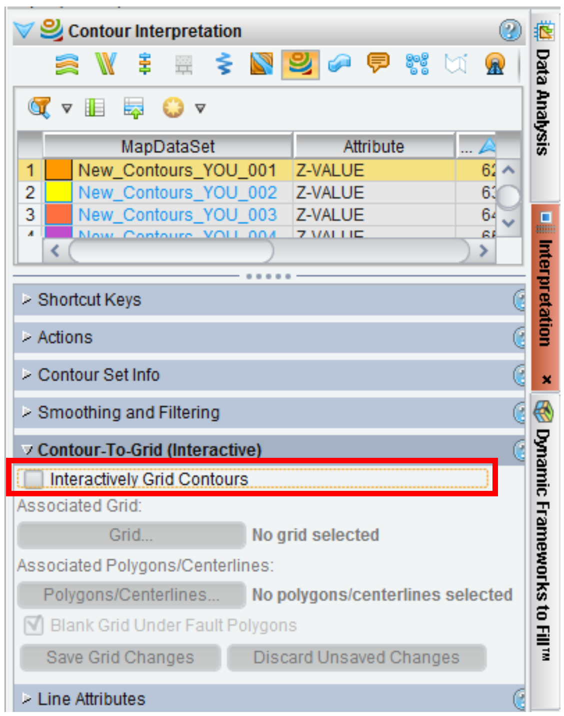
- **To move a node** — Click a node and drag it.
- **To delete a node** — Put your cursor on the node and MB2.
- **To add a node** — Click on a contour line, between nodes.
- **To drag the entire line** — anywhere on a contour line or node, press <Shift> + MB1 and drag to the desired spot.
- **To delete a line** — MB2 between nodes on a line.
- **To close a contour** — Drag an end node (triangle) and release it on top of the other end node (triangle).

Updating Grids from Edited Contours

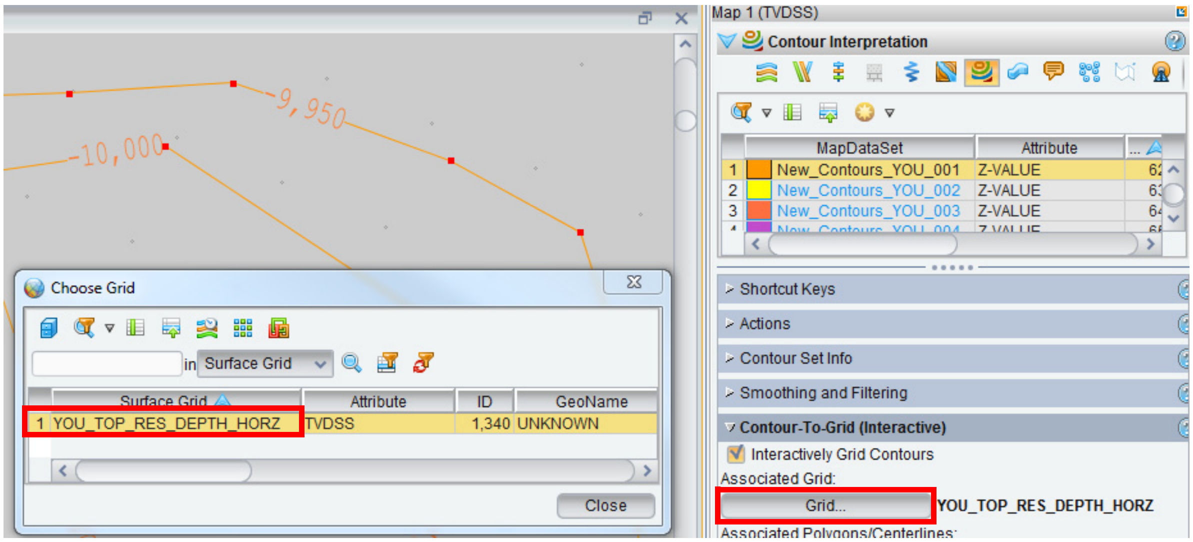
You can update an OpenWorks surface grid every time you edit a contour. If the surface grid is already displayed, the application redraws it when the regridding is complete.

To use this functionality, you must associate a surface grid with the contour set. Complete the contour-to-grid procedure, as follows:

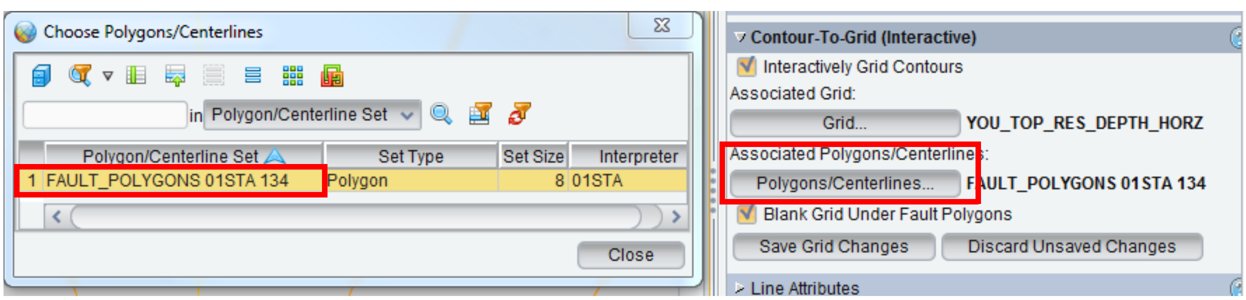
1. In the *Contour to Grid (Interactive)* panel of the *Contour Interpretation* task pane, toggle on **Interactively Grid Contours**.



2. On the *Contour-To-Grid (Interactively)* panel, click the **Grid...** button.The *Choose Grid* dialog appears. It lists all of the surface grids you have in your session. Select a **surface grid** and click **Close**.



3. You may also apply your fault polygons when regridding. Click the Associated Polygons/Centerlines, **Polygons/Centerlines...** button. Select **FAULT_POLYGONS**, then click **Close**.



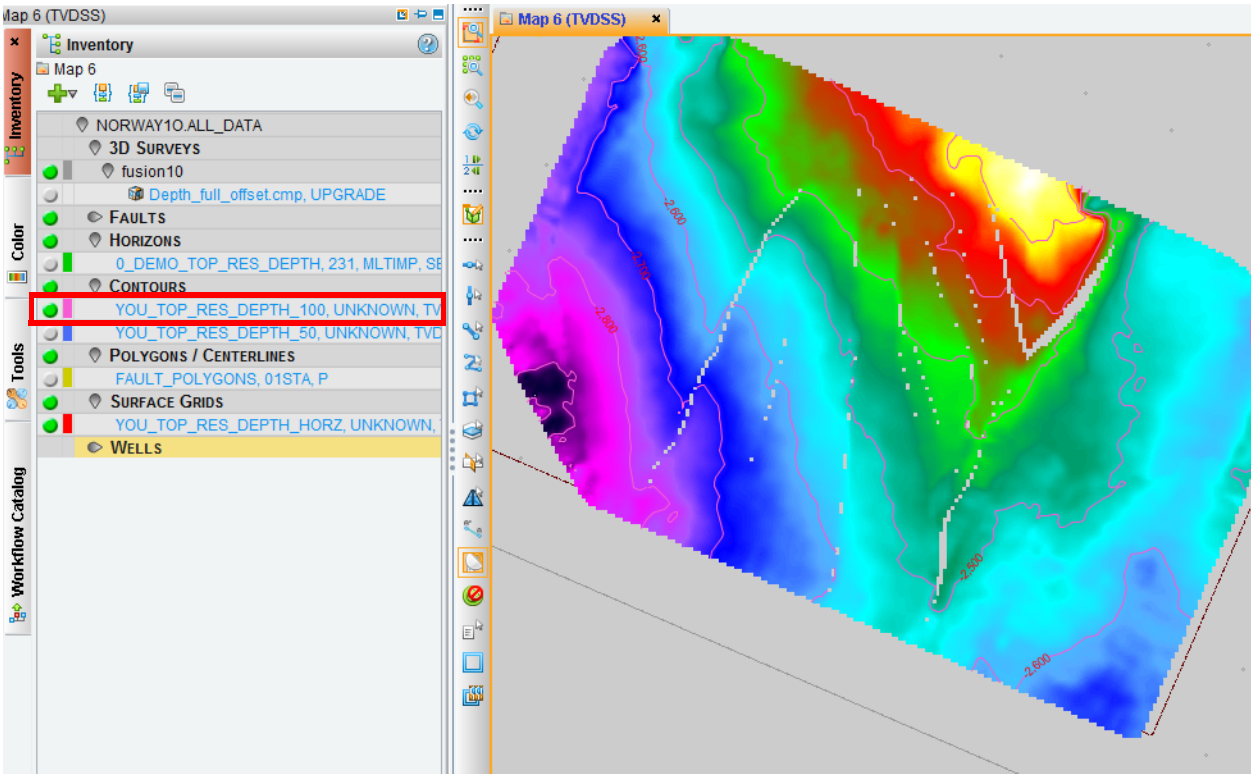
4. Begin to edit a contour. With the first click of your cursor, the Save Grid Changes and Discard Unsaved Changes buttons become available. Click **Save Grid Changes** or **Discard Unsaved Changes,** as needed.

If you undo contour edits back to the beginning, the software restores the original contours.

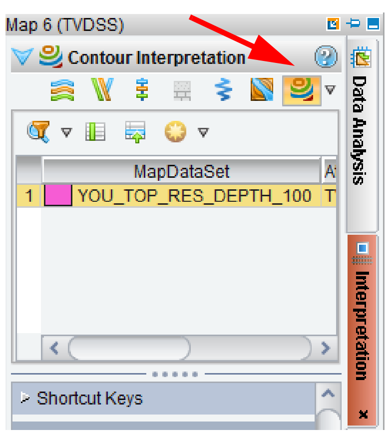
> **Note:**
>
> If you click the Undo icon partway through contour interpretation, the software moves the contour node to its previous location. The grid is updated with a new contour-to-grid operation during the Undo operation only if Interactively Grid Contours is toggled on.

Exercise A.4: Editing Contours

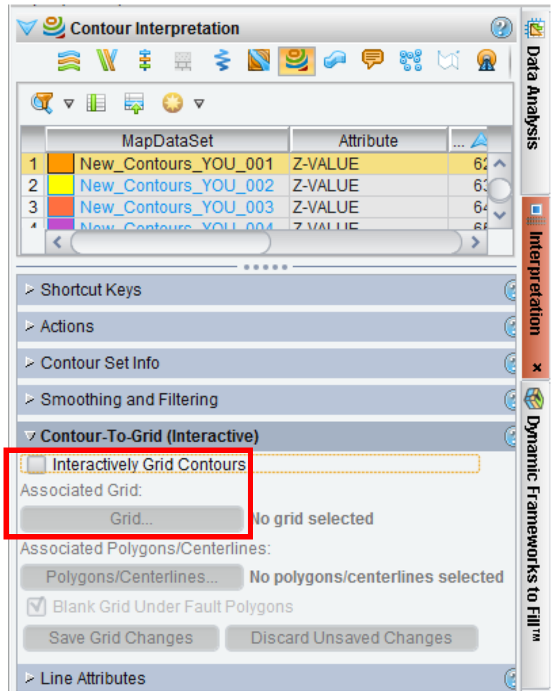
1. In the *Inventory* task pane of the *Map* view, toggle on the contour **YOU_TOP_RES_DEPTH_100**. Toggle off all other contours.



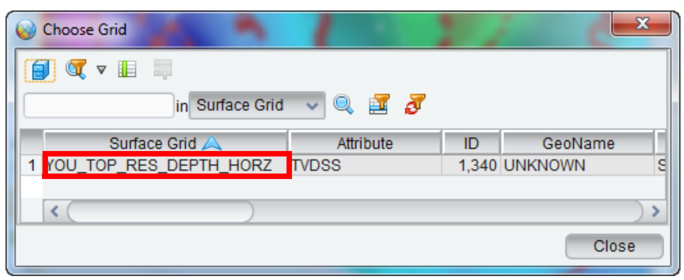
2. In the *Interpretation* task pane, click the **Contour Interpretation** () icon.



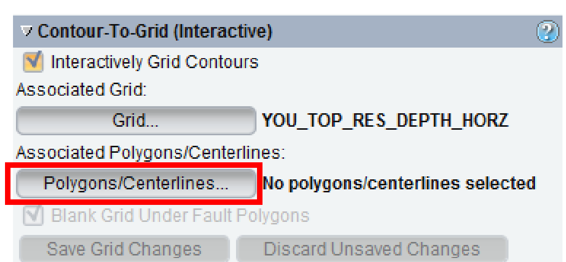
3. In the *Contour-To-Grid (Interactive)* panel, toggle on **Interactively Grid Contours.** Click the Associated Grid: **Grid** button.



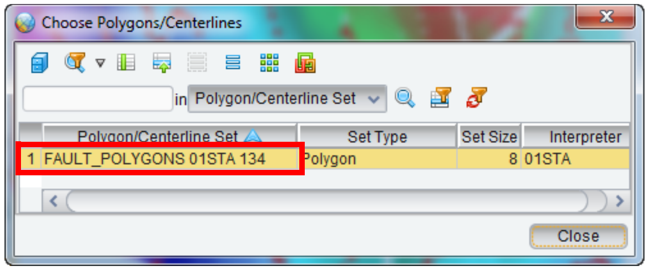
4. In the *Choose Grid* dialog, select **YOU_TOP_RES_DEPTH_HORZ** and click **Close**. The grid will update as you edit your contours.

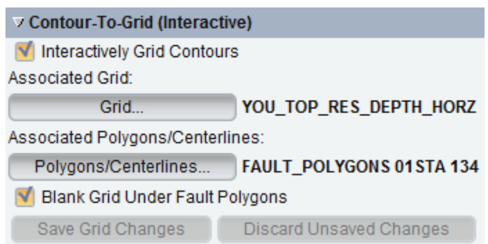


5. On the *Contour-To-Grid* panel, click the **Polygons / Centerlines** button.

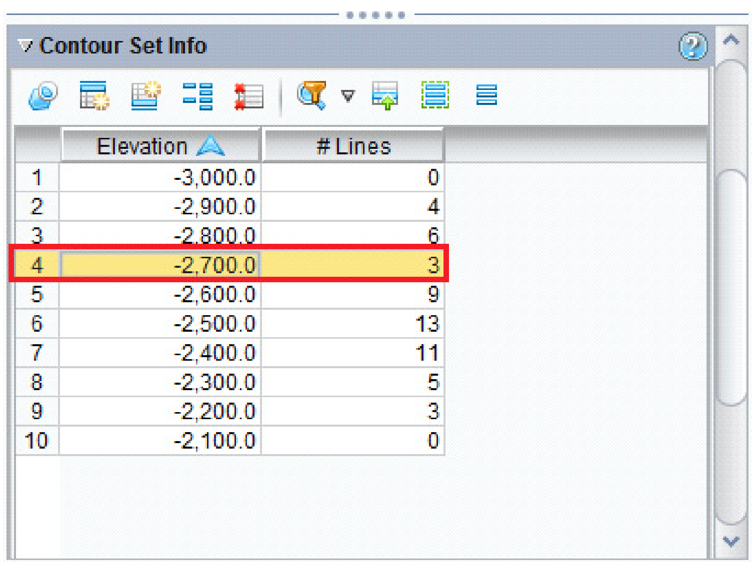


6. In the *Choose Polygons/Centerlines* dialog, select **FAULT_POLYGONS 01STA 134** and click **Close**. The selected polygon set will now be used to update the grid as you edit your contours.

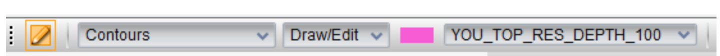




7. In the *Contour Set Info* panel select the entry for elevation **-2,700**.

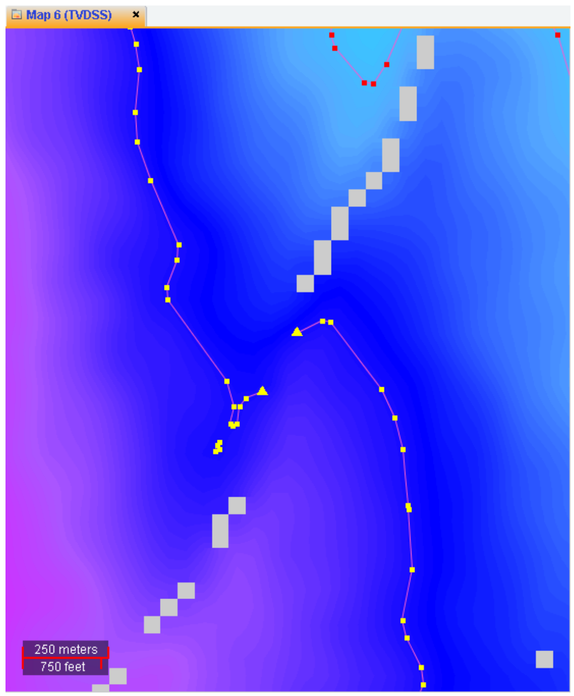


8. In the horizontal toolbar, toggle on **Interpretation Mode** ( ). .

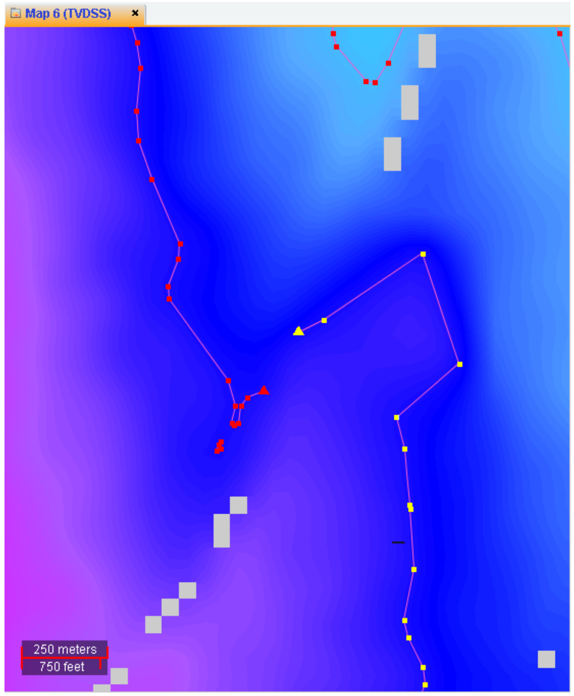


> **Note:**
>
> Note that in *Map* view all contour lines have red nodes except for the elevation you selected in the prior step (these have yellow nodes).

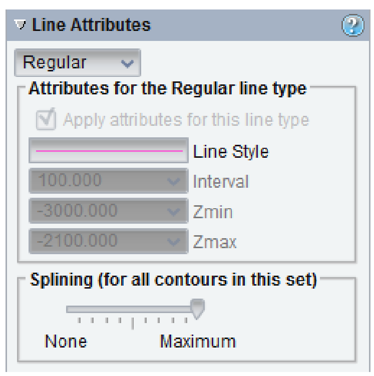
9. Zoom in to the **intersection** of the 2,700 contour line and the western-most fault.

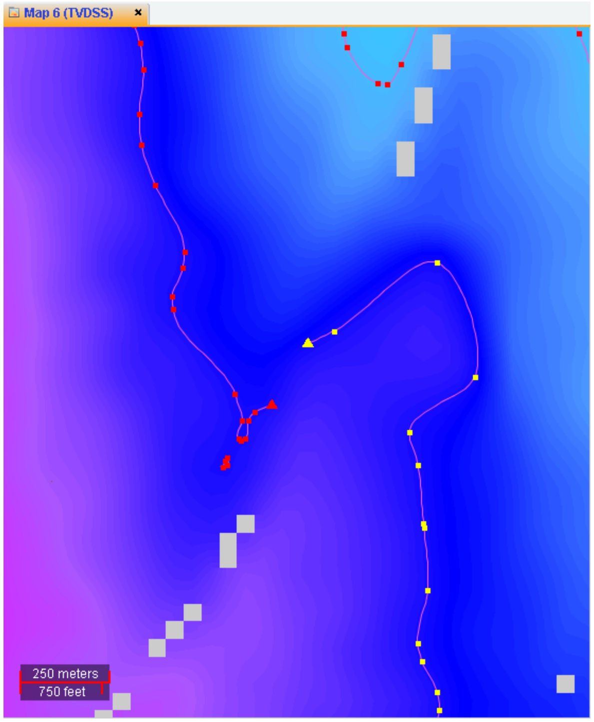


10. Click-and-drag a node to a new location. Notice the regridding that occurs when you move the node.



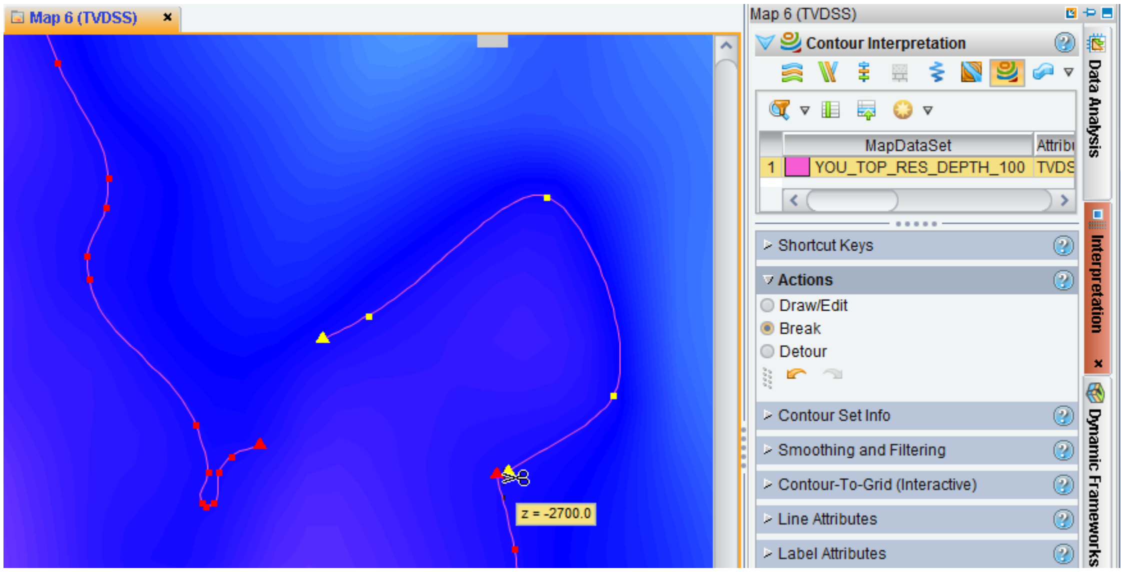
11. In the *Line Attributes* panel, set the Splining (for all contours in this set) to **Maximum**. This makes the contours curved, not angular.



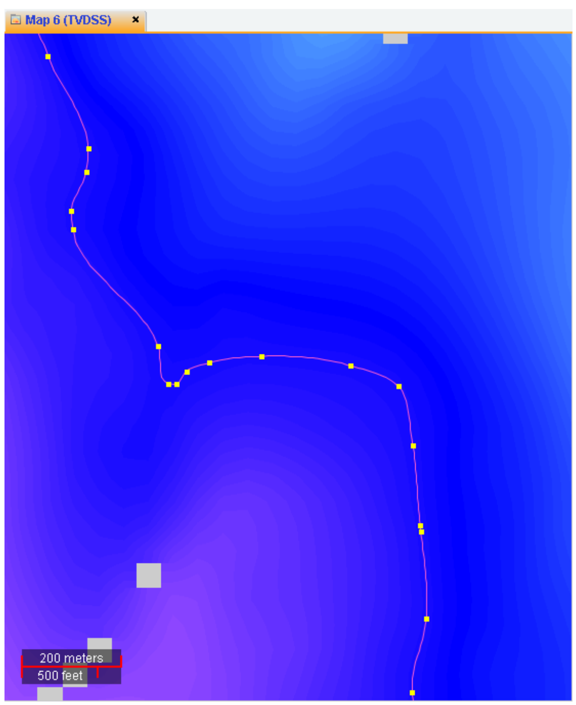


- **MB2** on a node to delete it.
- **MB2** on the contour line between nodes to delete it. This is useful for removing closed-loop bull's-eyes.
- **MB1** on the contour to add a node between existing nodes.
- **<Shift> + MB1** to move the entire contour.

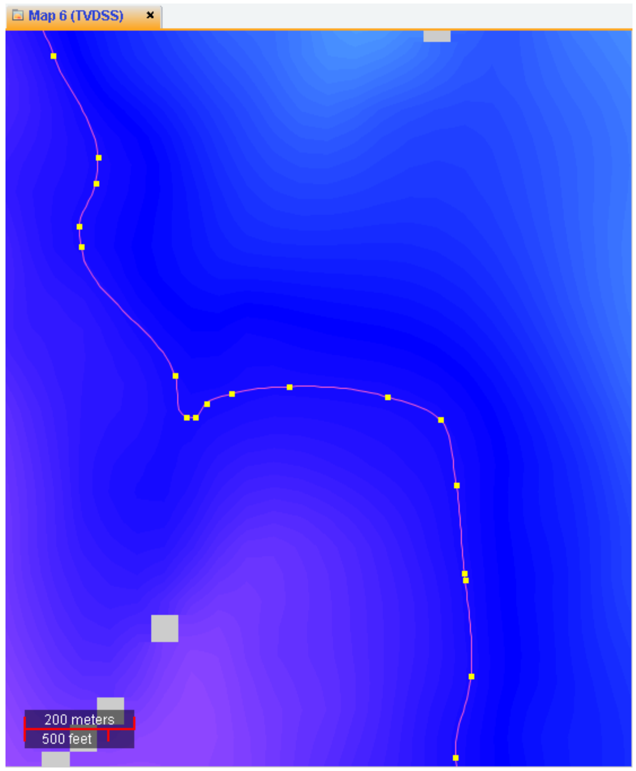
12. In the *Actions* panel, toggle on **Break** to cut a contour line into two contour lines.



13. In the *Actions* panel, toggle on **Draw/Edit.** Select the end point of one contour line (a triangle), then digitize a few points until close to the end point of another contour line. With your cursor over the end point of the other contour line (a triangle), note that the cursor has turned into a + sign. Click to join these two contour lines.

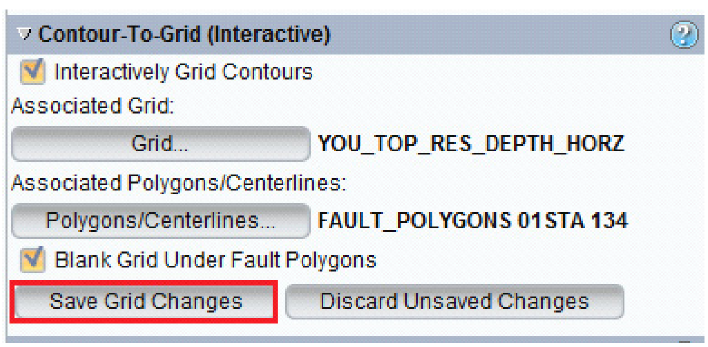


14. Use the foregoing methods to **contour** until you are happy with the appearance.

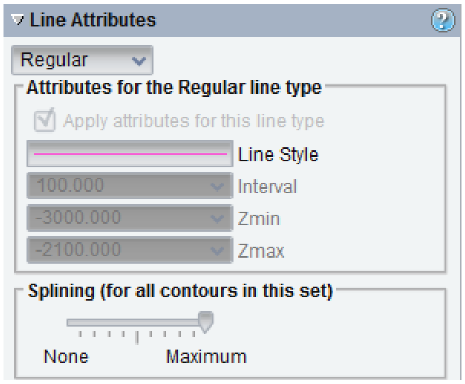


15. In the horizontal toolbar, toggle off **Interpretation Mode (**  **)**.

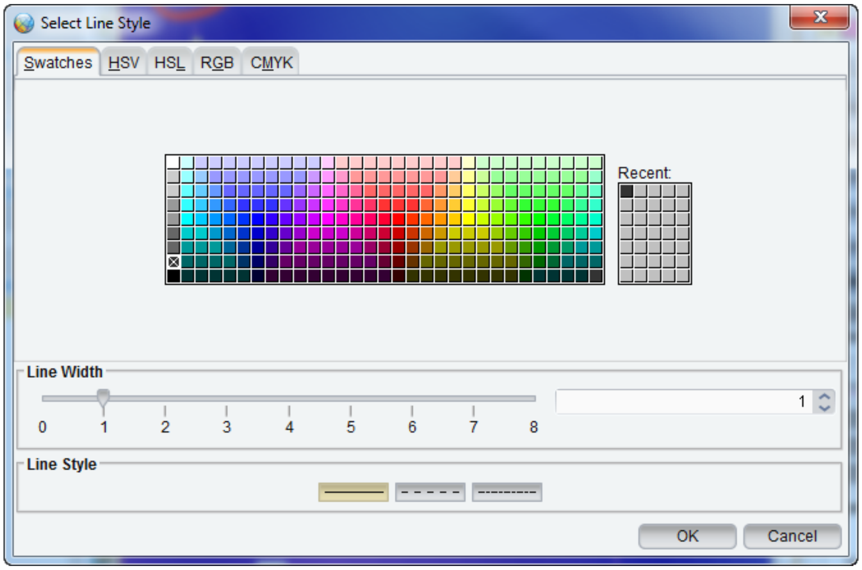
16. In the *Contour-to-Grid (Interactive)* panel, click the **Save Grid Changes** button to commit the grid updates to the database.



17. In *Line Attributes* panel, click the **Line Style** button.



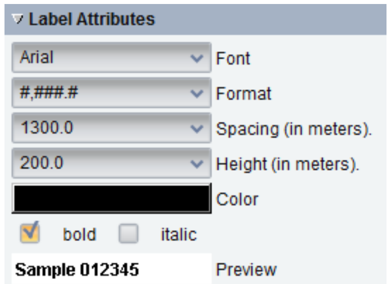
18. In the *Select Line Style* dialog, choose a **dark gray** and click **OK**.

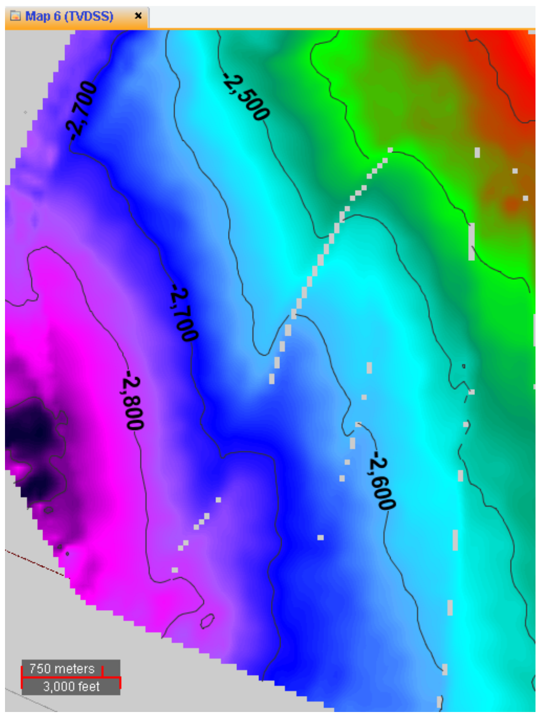


19. In the *Label Attributes* panel make the following selections.

- Set Spacing (m) to **1300.**
- Set Height (m) to **200**.
- Toggle on **Bold**.

- Set Color to **black**.



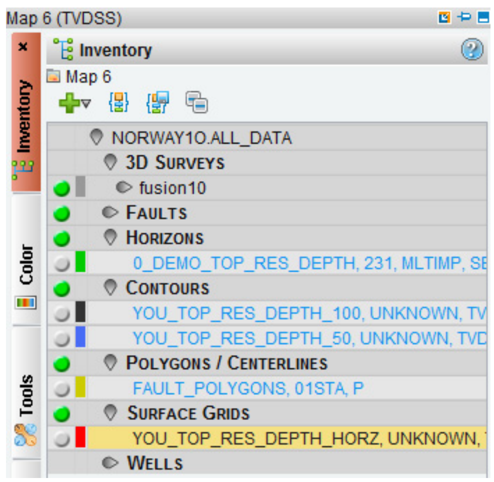


Exercise A.5: Using Inclusive and Exclusive Polygons

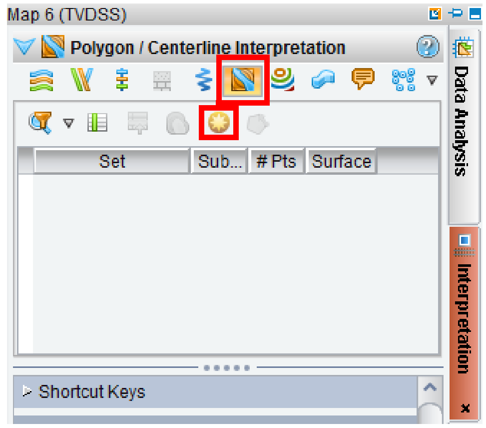
This exercise will illustrate the usefulness of inclusive polygons in mapping. An inclusive polygon allows you to use only data that falls within the constraints of a polygon, and applies to several tools, including ezTracker Plus and Grid and Contour. An exclusive polygon nulls the data that falls within the bounds of the polygon.

Part 1: Polygons

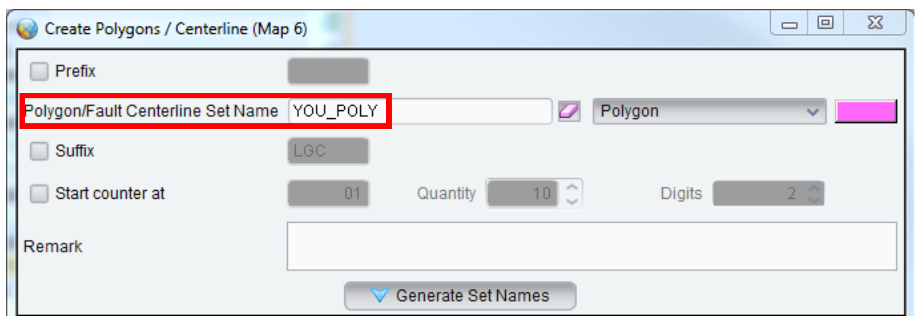
1. In the *Inventory* task pane of the *Map* view, toggle off all **surface grids**, **contours**, and **polygons**.

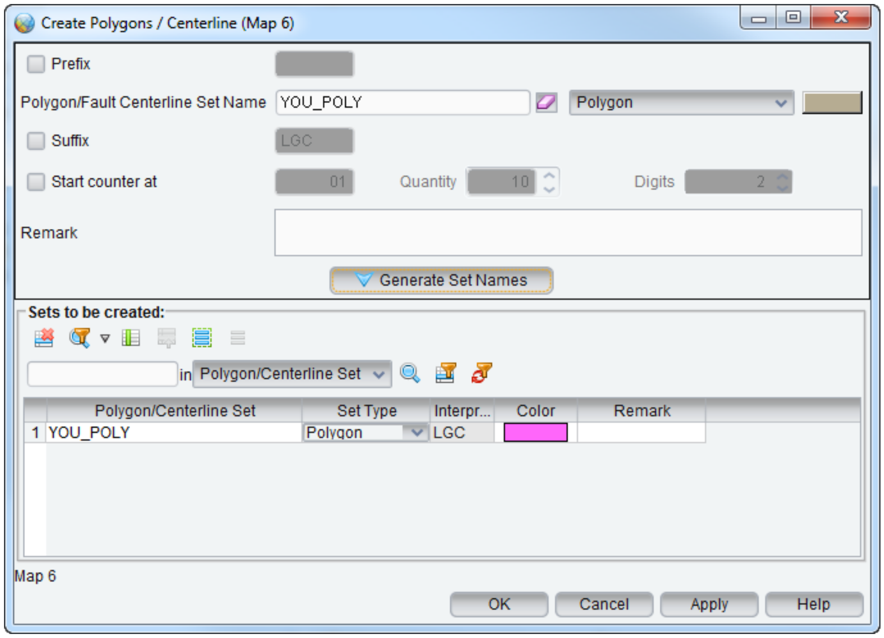


2. In the *Interpretation* task pane, select the **Polygon/Centerline Interpretation** () icon, then click the **Launch Create Polygons/ Centerlines** dialog () icon to open the *Create Polygons/ Centerline* dialog.



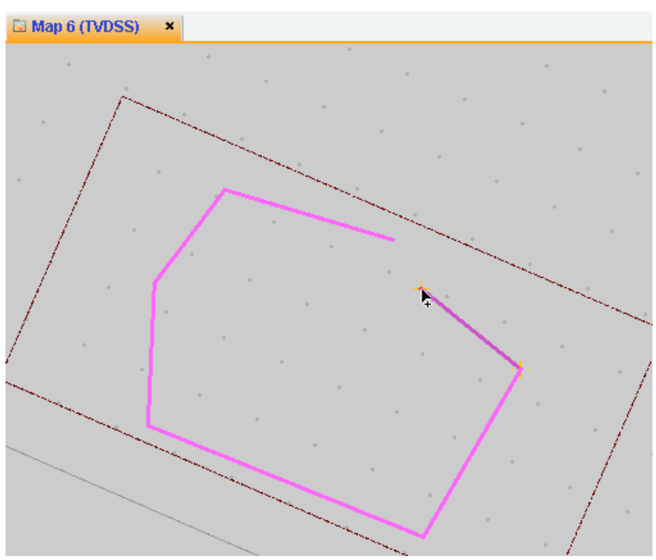
3. In the *Create Polygons/Centerline* dialog, enter **"YOU_POLY"** in the Polygons/Fault Centerlines Set Name text field, then click **Generate Set Name** button to create a polygon that is visible in the *Sets to be created:* panel of the *Create Polygons/Centerline* dialog. Click **OK**.



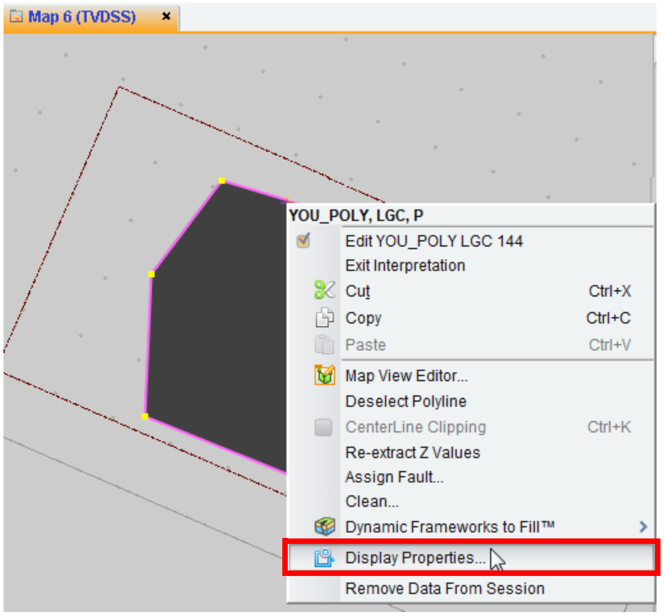


> **Note:**
>
> To edit the Polygon/Centerline Set name, Set Type, Color, and Remark, first click Generate Set Names.

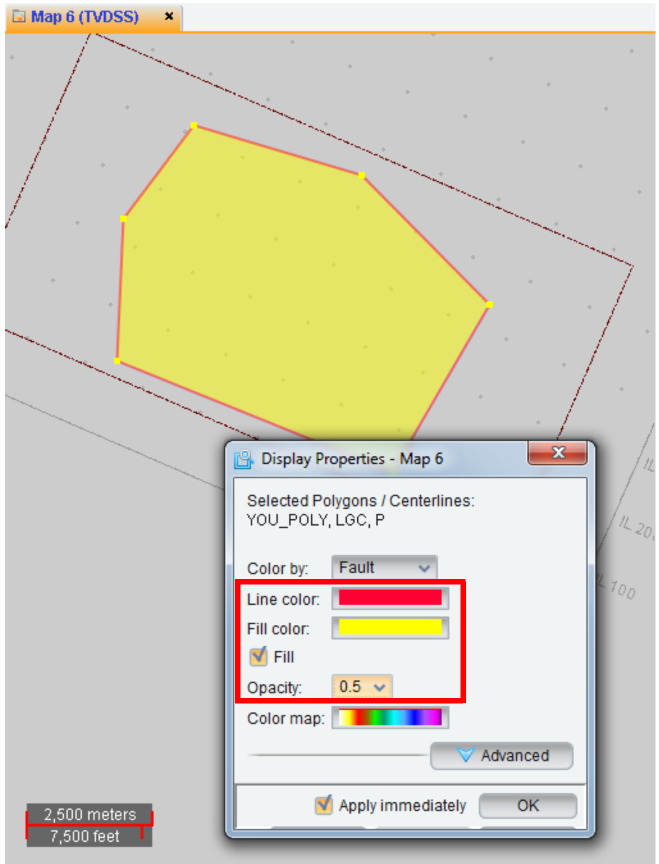
4. On the horizontal toolbar, click the **Interpretation Mode** ( ) icon. Click along a path to digitize a polygon in your *Map* view. **MB2** to end or close the polygon.



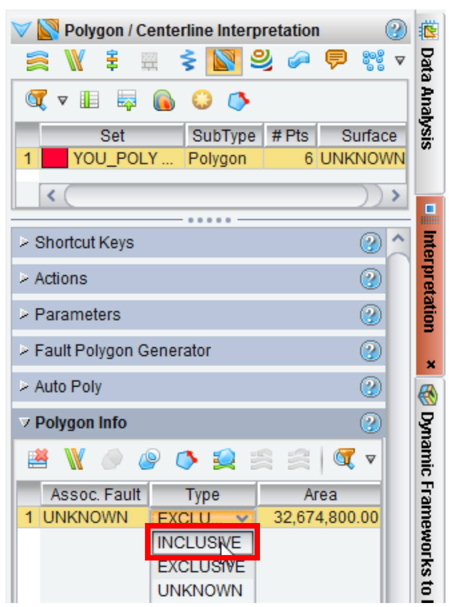
5. With your cursor on the newly created Polygon (**YOU_POLY**), **MB3 > Display Properties**. The **Display Properties** dialog opens.



6. In the *Display Properties* dialog, change the **Line** color to red and the **Fill** color to yellow and the Opacity: to 0.5. Make sure Fill option is toggled on so the polygon is color filled. Click **OK**.



7. In the *Polygon Info* panel of the *Polygon/Centerline Interpretation* task pane, set the polygon Type to **INCLUSIVE**.

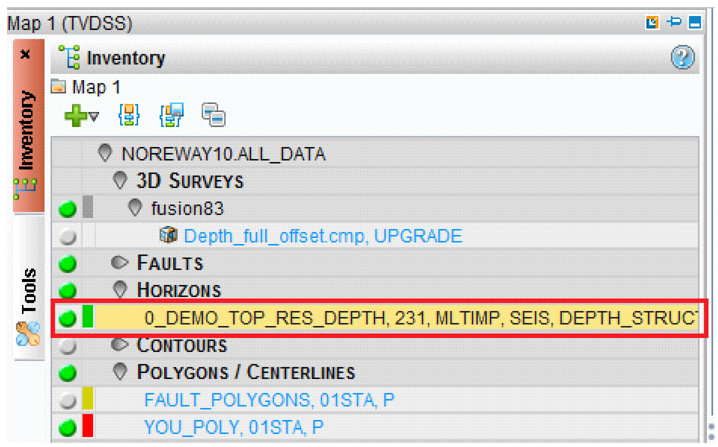


Note:

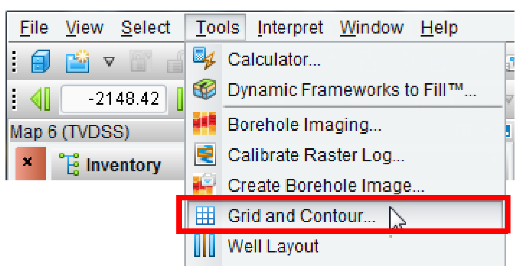
The polygon lost its color fill because an inclusive polygon does not show a fill. Thus, with a glance in *Map* view, you can tell which polygons are Inclusive and which are Exclusive.

Part 2: Gridding

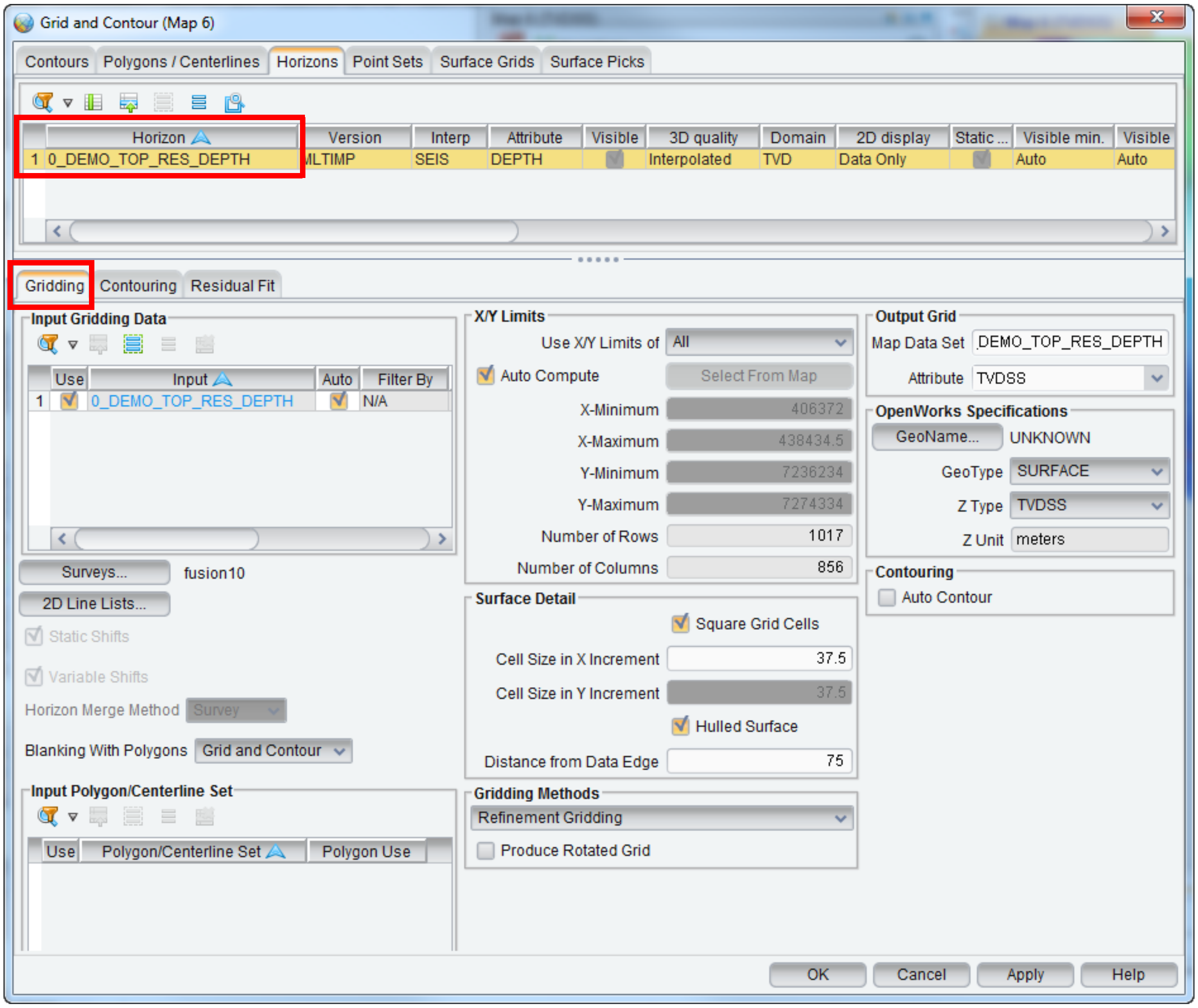
8. In the *Inventory* task pane of the *Map* view, toggle on the **0_DEMO_TOP_RES_DEPTH** horizon.



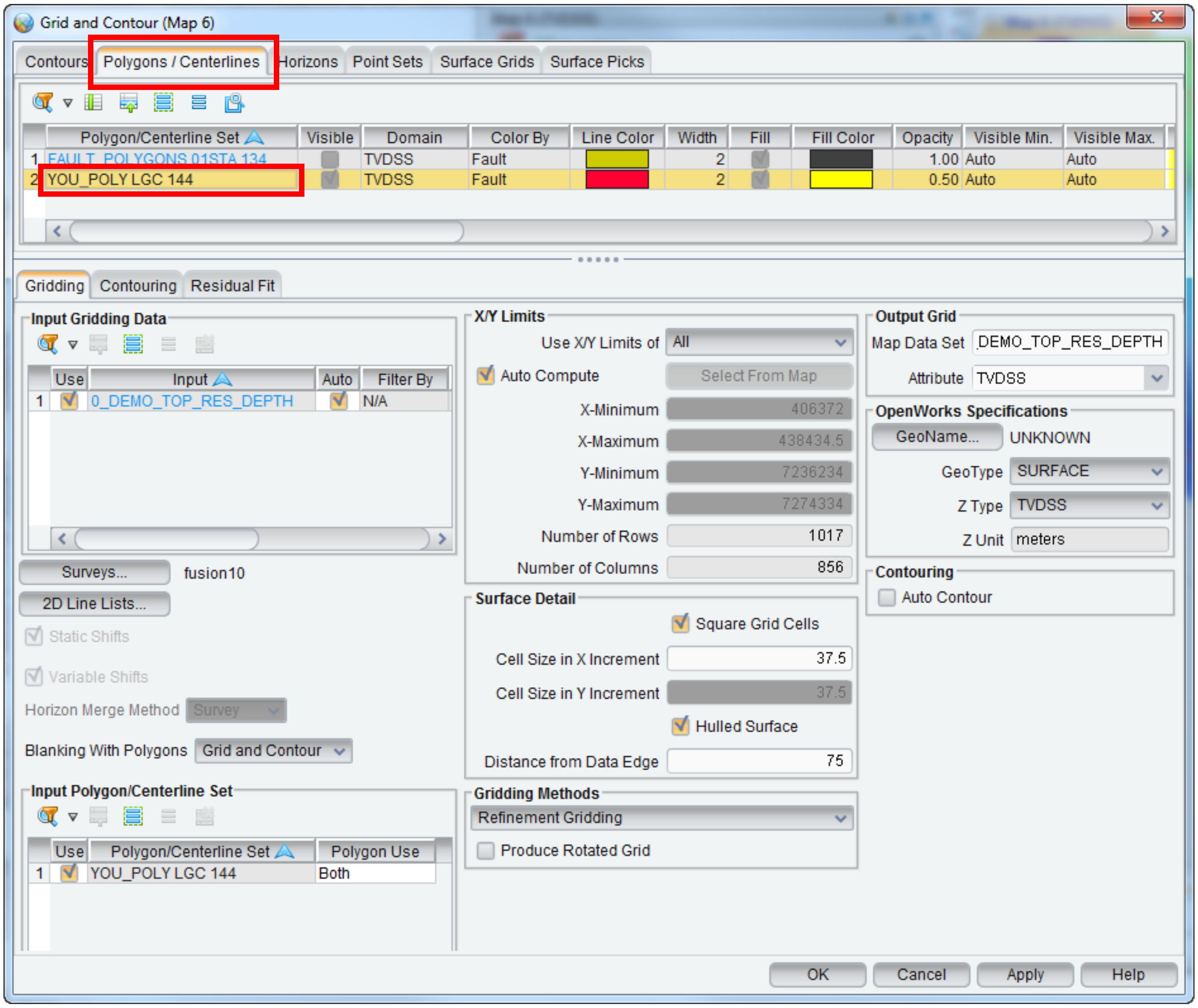
9. In the menu bar select **Tools > Grid and Contour** to display the *Grid and Contour* dialog.



10. In the *Grid and Contour* dialog, select the *Gridding* operation tab, then on the *Horizons* input data tab select Horizon **O_DEMO_TOP_RES_DEPTH**.



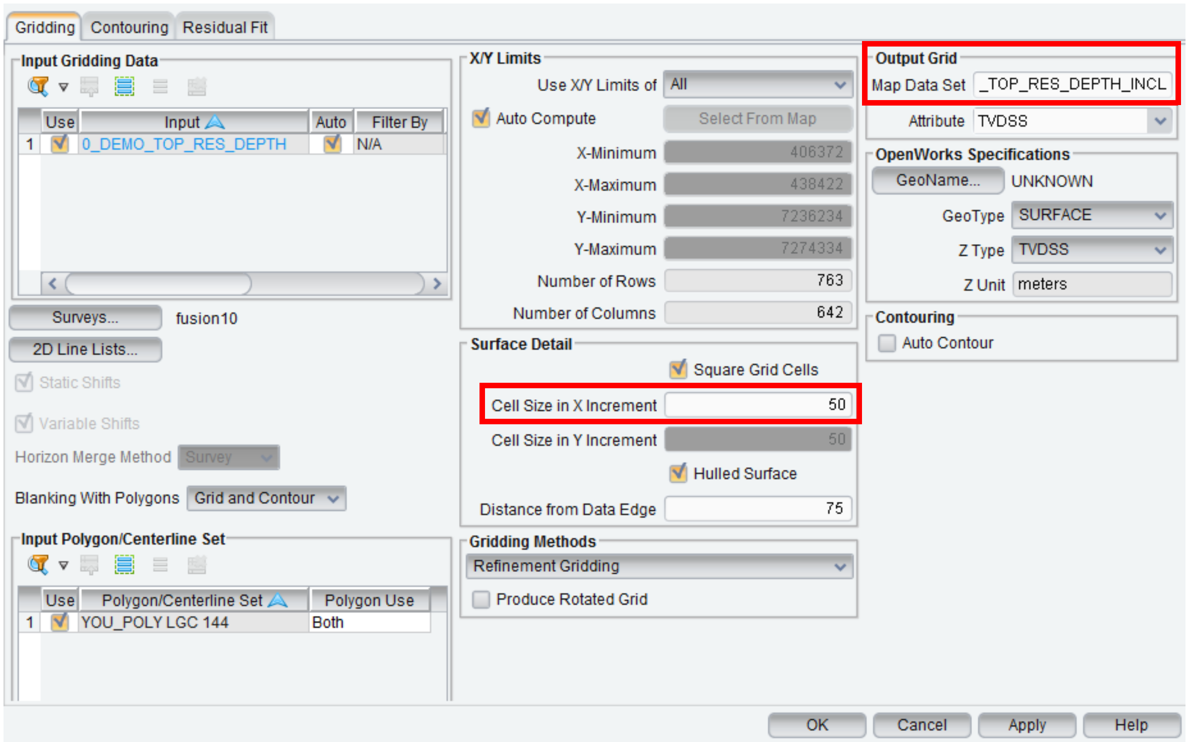
11. In the *Grid and Contour* dialog, click the *Polygons/Centerlines* input tab. Select **YOU_POLY**.



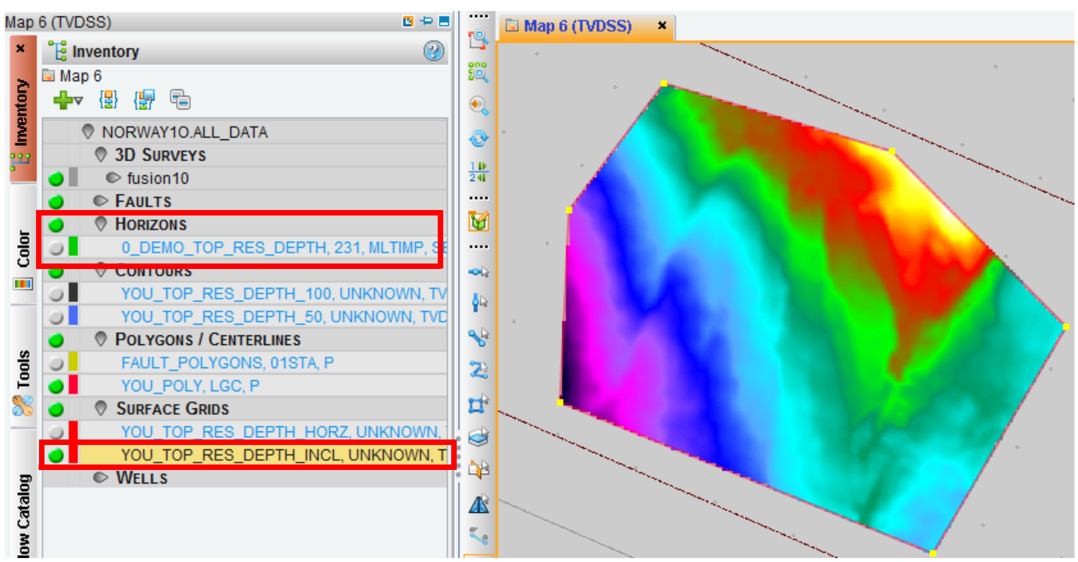
12. In the *Gridding* operation tab make following changes:

- In the *Output Grid* panel, on the Map Data Set: text field, enter **"YOU_ TOP_RES_DEPTH_INCL"**.
- In the *Surface Detail* panel, set the Cell Size in X Increment to **"50"**.

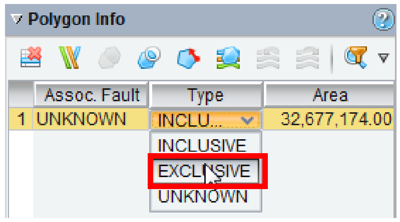
- Click **Apply**.



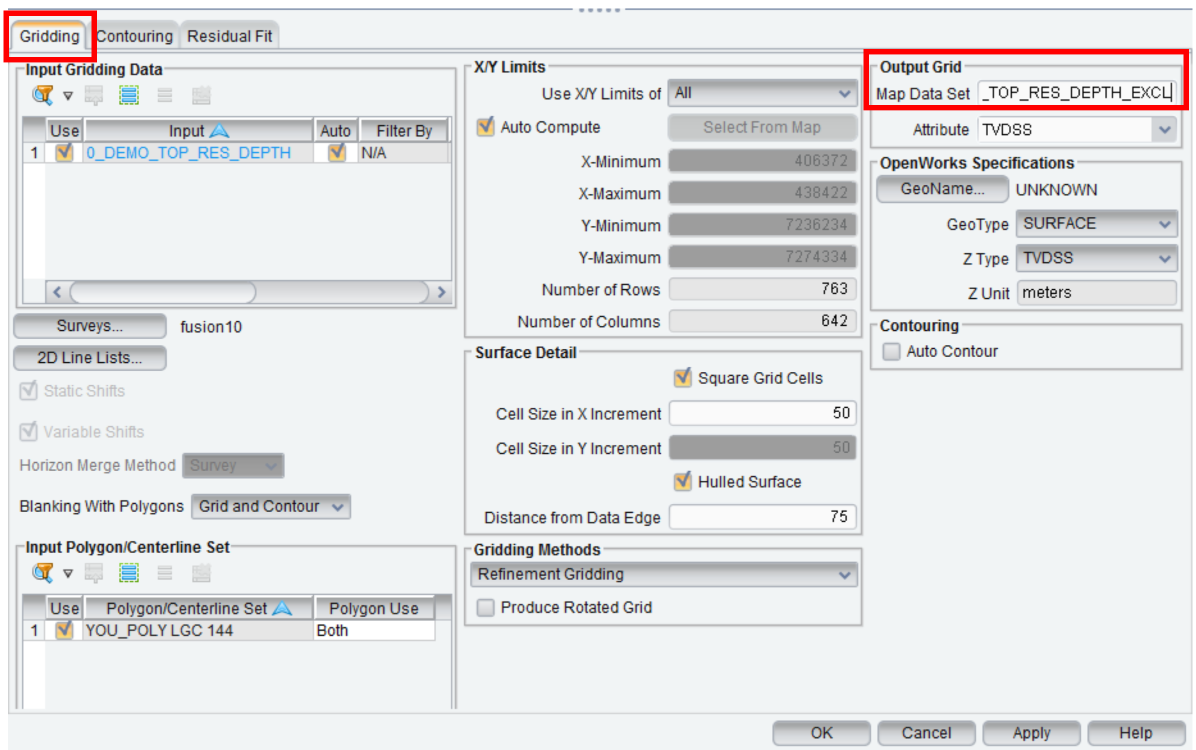
13. In *Map* view, toggle off all **horizons** and toggle on surface grid **YOU_TOP_RES_DEPTH_INCL**.



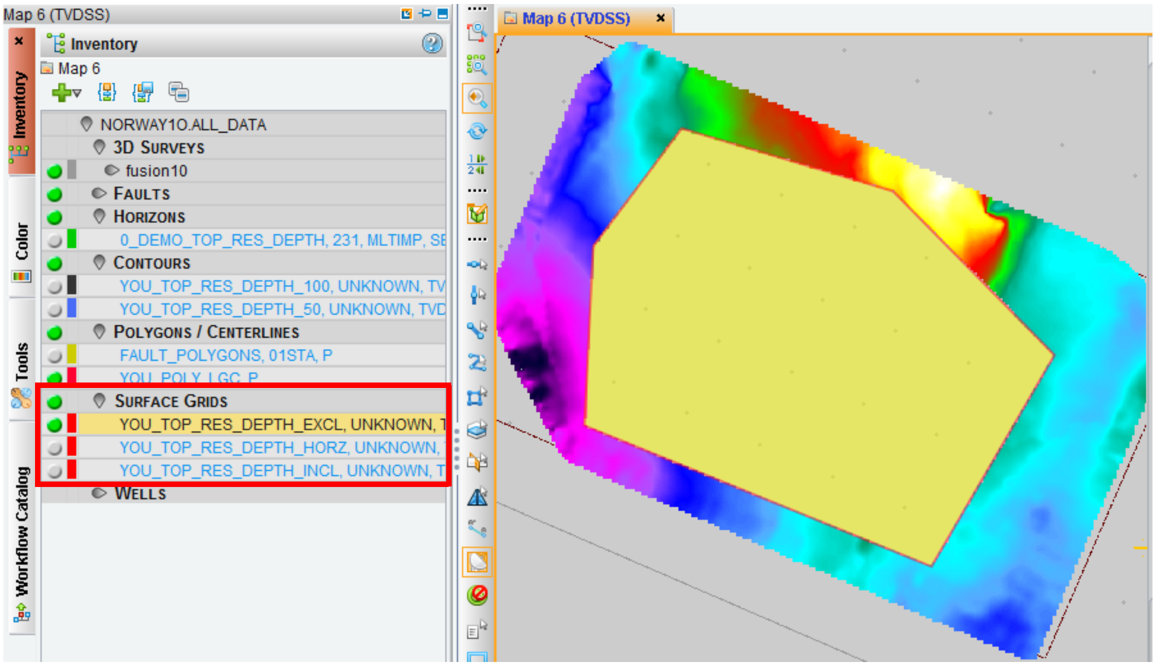
14. In the *Polygon Info* panel of the *Polygon/Centerline Interpretation* task pane, change the Type cell from INCLUSIVE to **EXCLUSIVE**.



15. In the *Gridding* operation tab of the *Grid and Contour* dialog, enter **"YOU_ TOP_RES_DEPTH_EXCL"** in the Map Data Set: text field. Click **Apply**.



16. In *Map* view, toggle on surface grid **YOU_TOP_RES_DEPTH_EXCL** and toggle off all other **surface grids**.



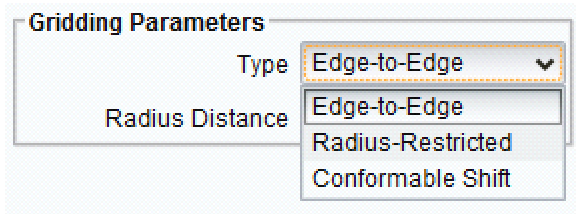
Using Residual Fit Operations

This section describes types of residual fit grids and contour maps. You may find that a surface grid does not match your surface pick locations. Perhaps your project includes new well information. In such cases, you may wish to correct the grid to match the picks. Use the Residual Fit operations in the *Grid and Contour* dialog to create corrected grids and contours by flexing or completely shifting the grid to match the exact XYZ locations of the surface picks.

This functionality is available under the *Residual Fit* operation tab. You must use a surface grid and either a point set or surface picks in the operation.

Types of Residual Fit Grids and Contour Maps

You can create three types of Residual Fit grids and contour maps. These types are listed in the Type pull-down menu in the *Gridding Parameters* panel of the *Residual Fit* tab.



- **Edge-to-Edge** — The surface is remolded within the grid perimeter to match the pick locations. The amount of correction gradually fades out at the grid edge, so the edges of the corrected and original grids match exactly. Edge-to-Edge corrected grids are especially well suited to an exploratory field with a sparse number of wells distributed over a wide area.

- **Radius-Restricted** — The correction zone is confined to a user-specified radius around each pick that reinterprets the surface. The corrected grid ties to the original grid along the borders of the combined radii. This method is well suited to reservoirs with dense well coverage. These are situations in which you want to flex the grid slightly to honor a small number of picks.

- **Conformable Shift** — The entire surface shifts to the pick level, flexing as necessary to fit the pick locations. At the edges of the corrected grid, the amount of correction is the average grid error. This method is useful when you want to create a corrected grid of a reservoir with conformable strata at a new depth. This corrected grid is approximately parallel to the input horizon.

Residual Fit Operation Basic Workflow

This section describes how to create different types of residual fit grids and contour maps. Follow these steps.

1. Display the following in a *Section* view that runs through any key wells:

   - The depth seismic volume
   - The depth surface grid
   - The well list

- Surface picks or point sets of interest

2. From the *Section* view menu bar, select **Tools > Grid and Contour**, to open the *Grid and Contour* dialog.

3. Click the ***Residual Fit*** operation tab of the *Grid and Contour* dialog.

4. Select the ***Surface Grids*** input data tab, to select a surface grid.

5. Click the ***Surface Picks*** or ***Point Sets*** tab to select the well pick or point set file you want to use.

6. In the *Residual Fit* operation tab, decide if you will use point set or pick data.

   - To use Surface Picks, toggle on **Use Pick Data**. The Input Pick Surface text field of the *Input* panel in the *Residual Fit* tab is populated.

   - To use Point Sets, toggle on **Use Point Set**. The Input Point Set text field of the *Input* panel in the *Residual Fit* tab is populated.

7. In the *Gridding Parameters* panel, select the type of gridding parameter you want from the Type: pull-down menu:

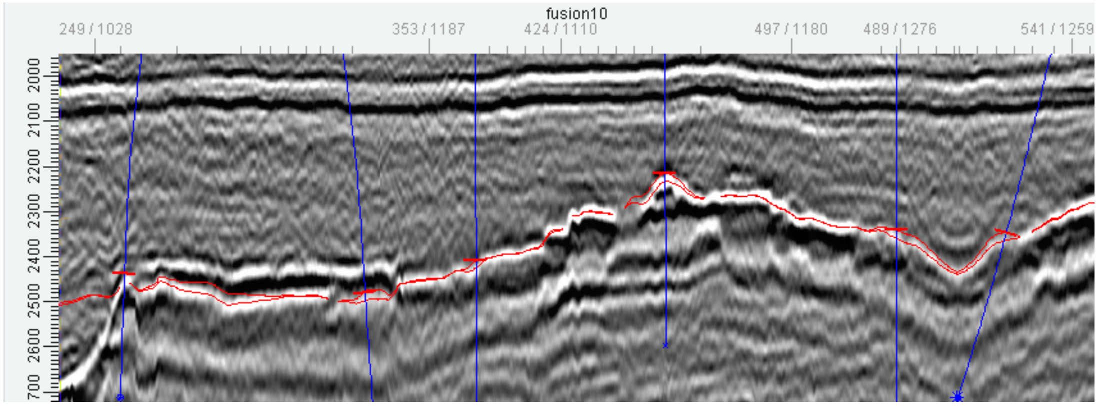
   - Edge-to-edge
   - Conformable shift
   - Radius-restricted

8. In the *Contouring* panel, toggle off **Auto Contour**.

9. Append the default **Output Grid** and default **Output Correction Grid** names with identifiers that indicate the gridding parameter used. For example, use **_E2E** for Edge-to-Edge Output Grid and **_E2Ecor** for the Edge-to-Edge Output Correction Grid.

10. Click **Apply**. When the processing is complete, the new surface grids are automatically added to the session. Repeat the last two steps to make residual fit grids using the Conformable Shift and Radius-Restricted gridding parameters.

11. In the *Inventory* task pane, toggle on the newly created **surface grids**.



The original grid and the three corrected grids now appear together in the section. Compare the corrected grids in the *Map* view.

Note:

If you want to compare all of the grids side by side, you can also create separate maps for each. Arrange the tiles (MB3 > Arrange Tiles) so you can see all the maps and compare the data.

Exercise A.6: Using the Residual Fit

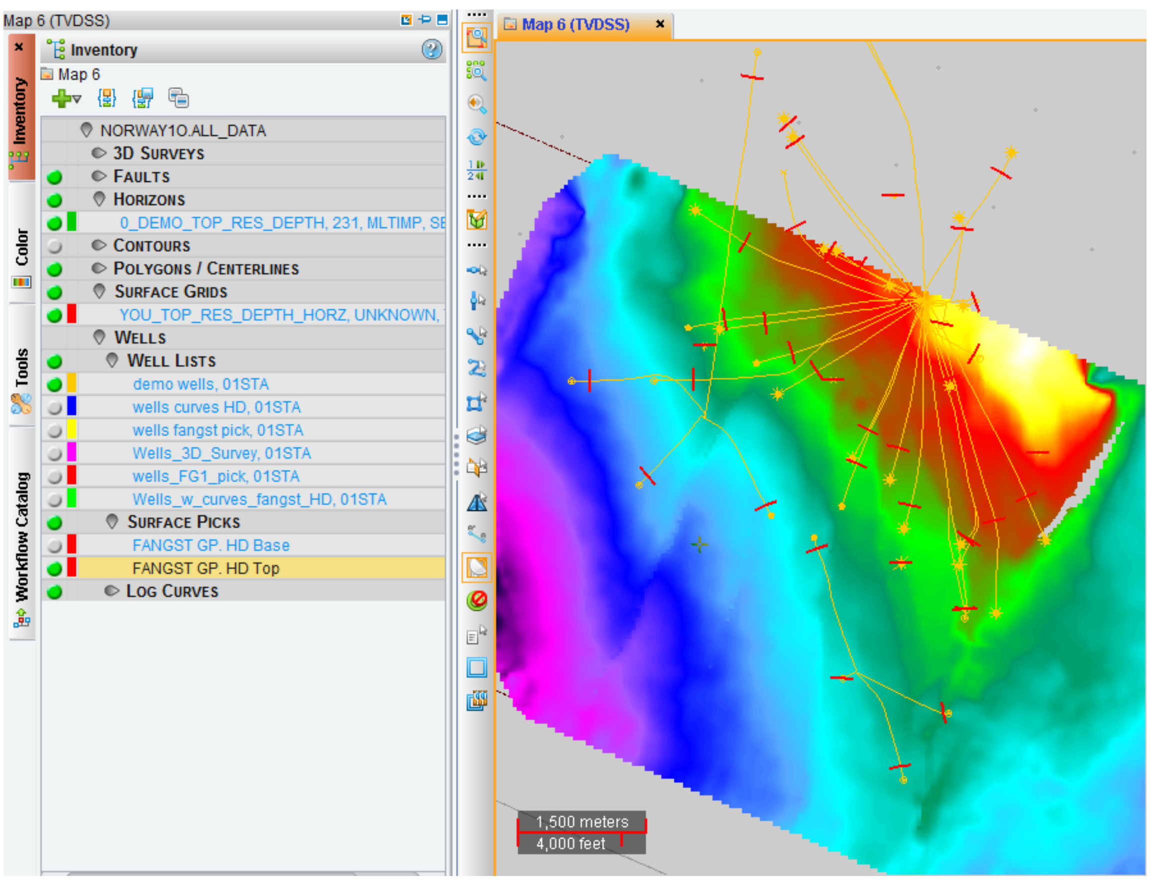
This exercise provides an opportunity to use the residual fit operation.

The *Grid and Contour* Residual Fit option makes it fast and easy to correct surface grids to match well pick information. Use three different algorithms to create surface grids, each with its own correction grid: edge-to-edge, radius-restricted, and conformable shift. You have already created the surface grid you need for this exercise.

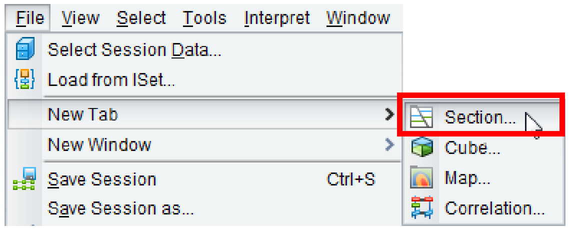
To use residual fit on your data, perform the following steps:

1. In the *Inventory* task pane of the *Map* view toggle on the following:

    - Horizon — **0_DEMO_TOP_RES_DEPTH**
    - Surface grid — **YOU_TOP_RES_DEPTH_HORZ**
    - Well list— **demo wells**
    - Surface picks — **FANGST GP.HD Top**

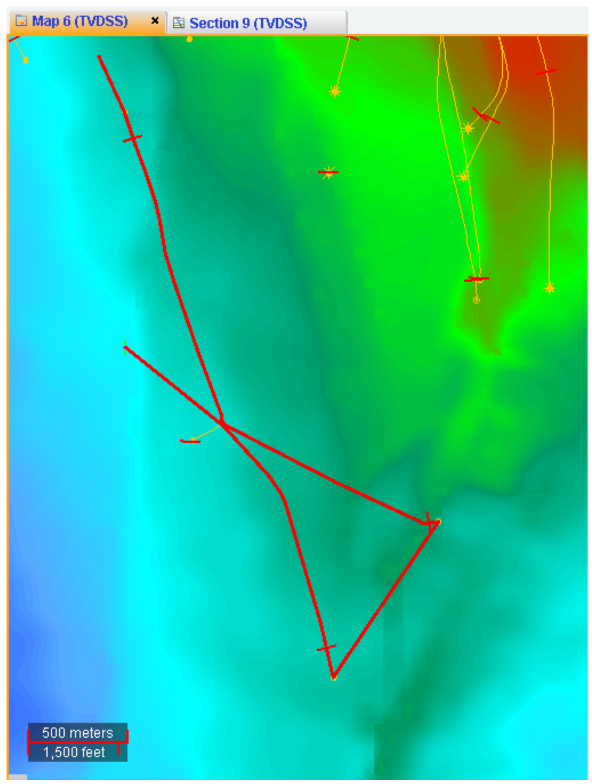


2. In the menu bar, select **File > New Tab > Section.**

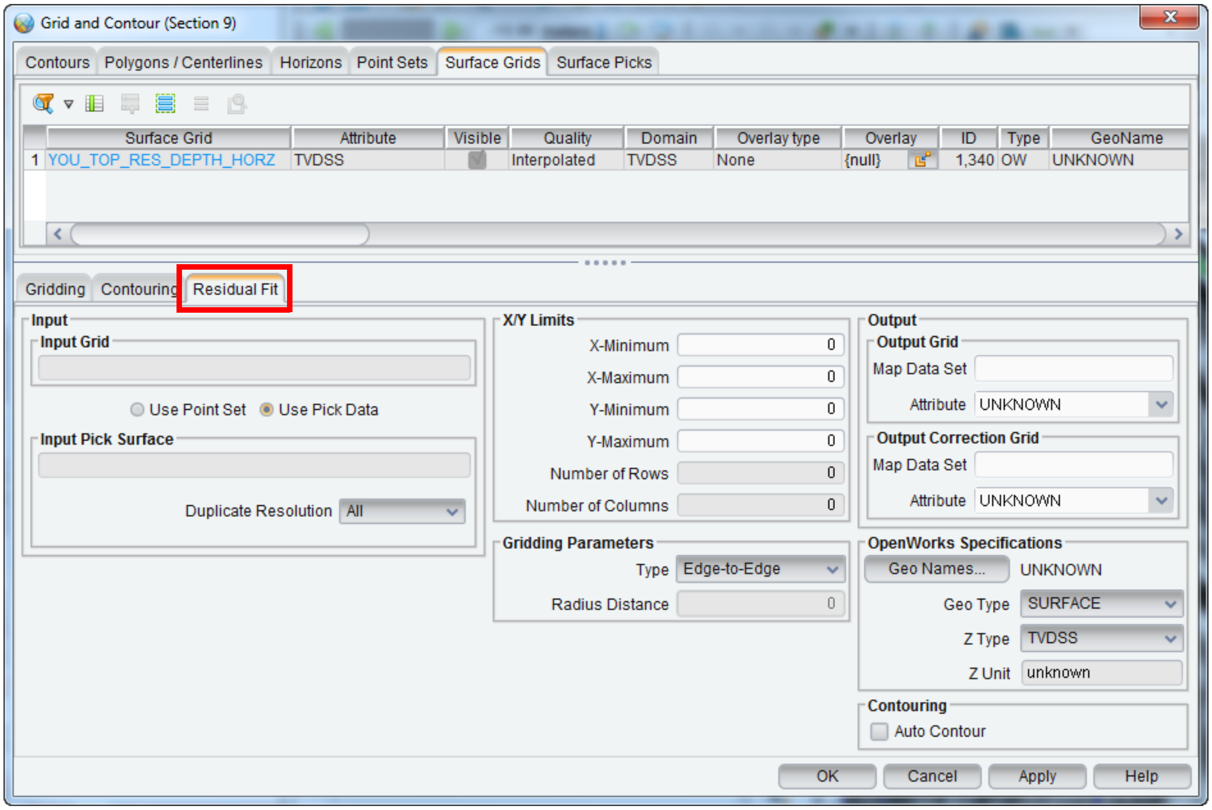


3. In the *Map* view vertical tool bar, click the **Select Point-to-Point** ( ) icon.

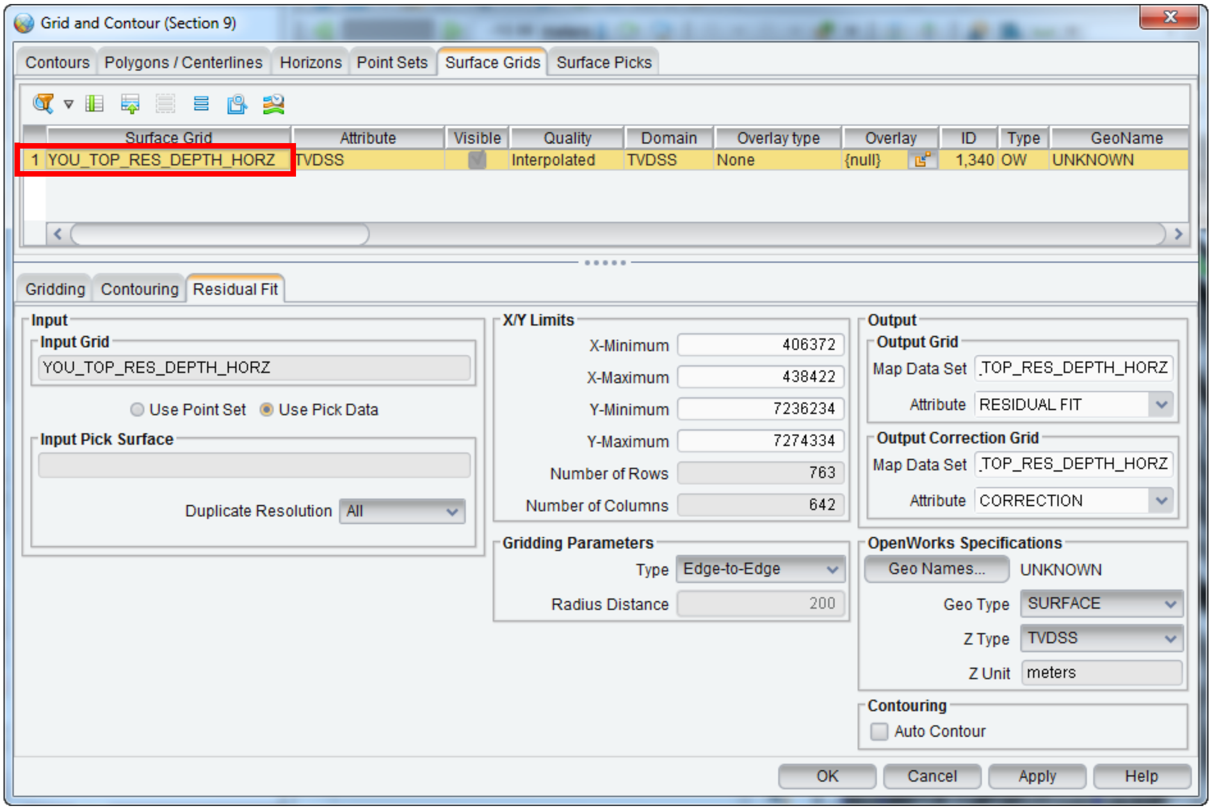
4. By means of **<Ctrl>** + **MB1**, select a section along the wellbore of wells 7-C-3 H, 7-C-4 H, and 7-C-1 H. **MB2** to finish and send to *Section* view.



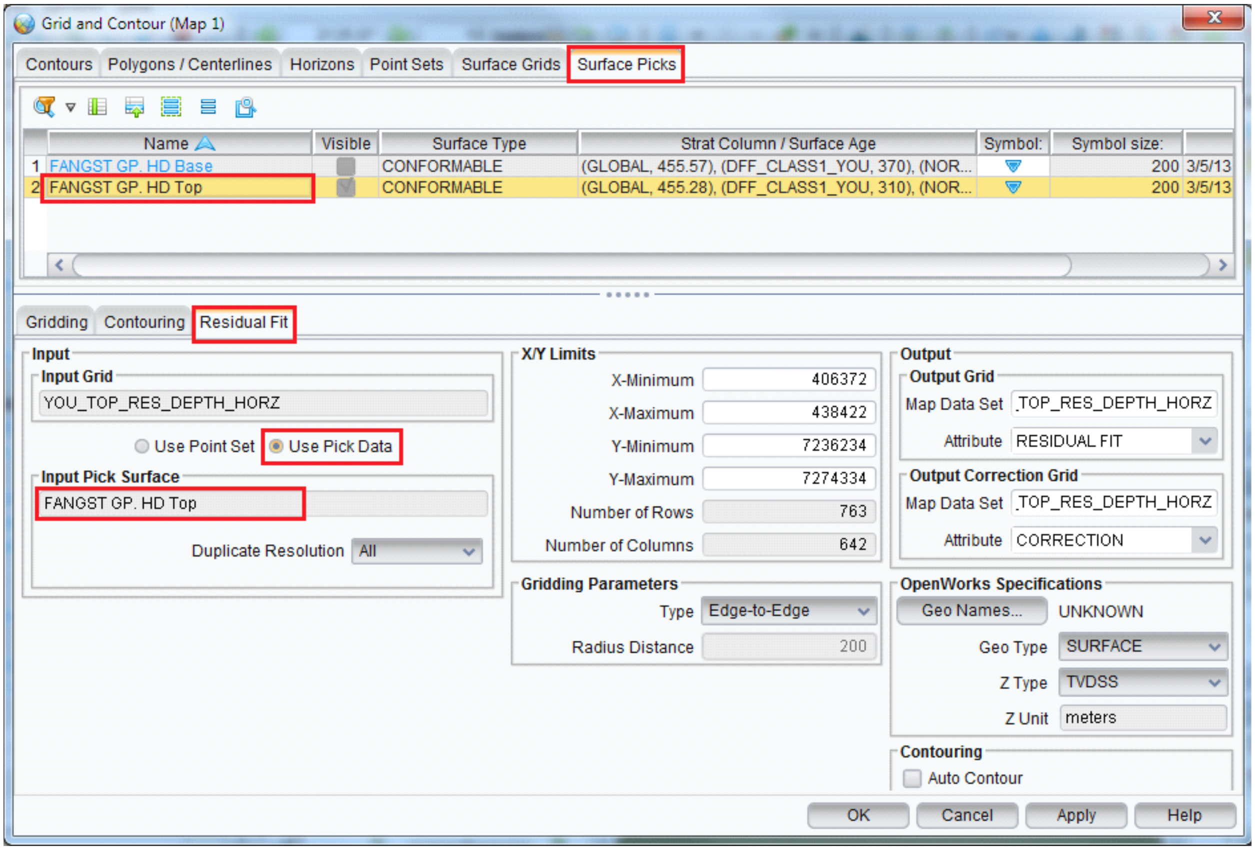
5. From the *DecisionSpace* main menu, select **Tools > Grid and Contour**. The *Grid and Contour* dialog opens. Click the ***Residual Fit*** operation tab.



6. Click the ***Surface Grid*** input data tab. Select Surface Grid **YOU_TOP_RES_DEPTH_HORZ**.



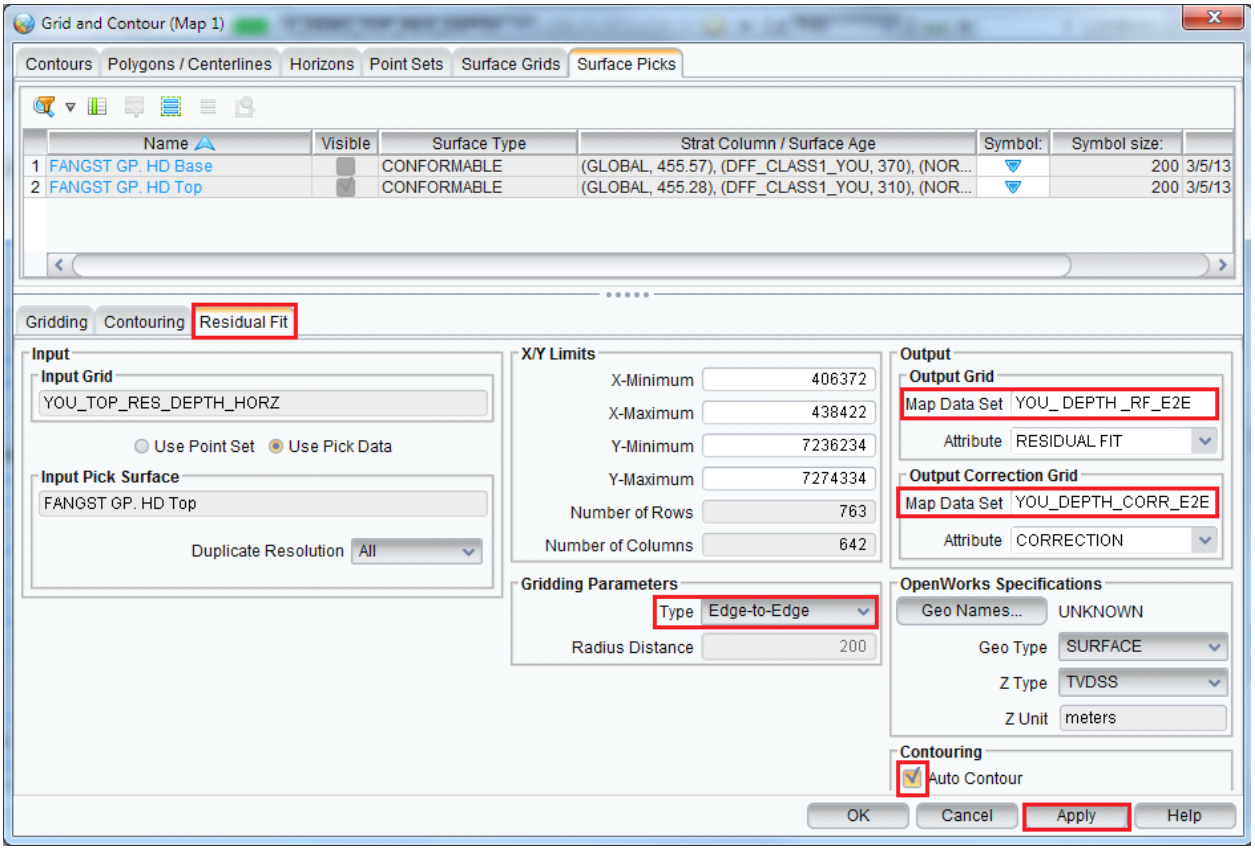
7. Click the *Surface Picks* tab and select **FANGST GP.HD Top**. In the *Input* panel of the *Residual Fit* operation tab, toggle on **Use Pick Data**.



8. In the *Residual Fit* operation tab, make the following changes.

- On the *Gridding Parameter* panel set Type: to **Edge-to-Edge.**
- On the *Contouring* panel, toggle on **Auto Contour**.
- In the *Output Grid* panel, enter **"YOU_ DEPTH _RF_E2E"** in the Map Data Set: text field.
- In the *Output Correction Grid* panel, enter **"YOU_DEPTH_CORR_E2E"** in Map Data Set text field.

- Click **OK**.



9. In the *Residual Fit* operation tab make following changes.

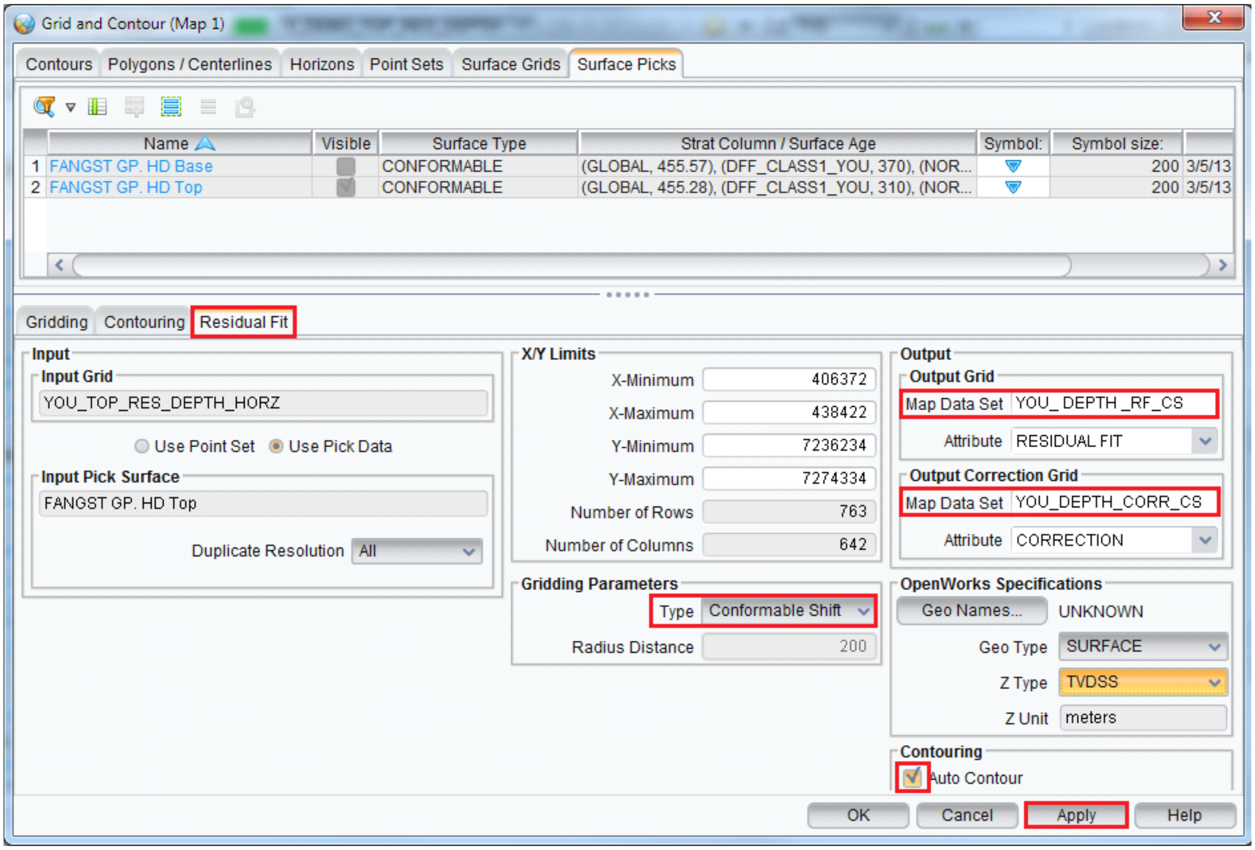
   - In the *Gridding Parameters* panel, set Type: to **Conformable Shift.**

   - In the *Contouring* panel, toggle on **Auto Contour**.

   - In the *Output Grid* panel, enter **"YOU_ DEPTH _RF_CS"** in the Map Data Set text field.

   - Under *Output Correction Grid* panel, enter **"YOU_DEPTH_CORR_CS"** in Map Data Set text field.

- Click **Apply**.



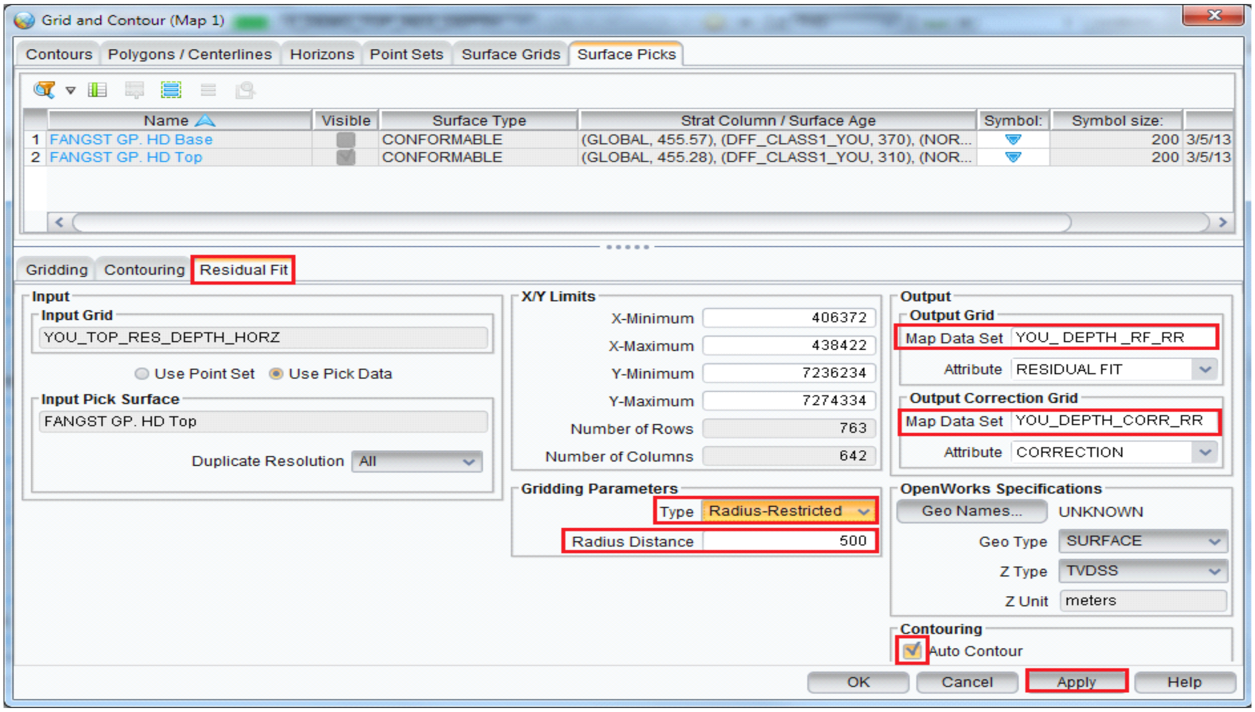
10. In the *Residual Fit* operation tab, make following changes.

- In the *Gridding Parameters* panel, set Type to **Radius Restricted** and enter **“500”** in the Radius Distance text field.

- In the *Contouring* panel, toggle on *Auto Contour*.

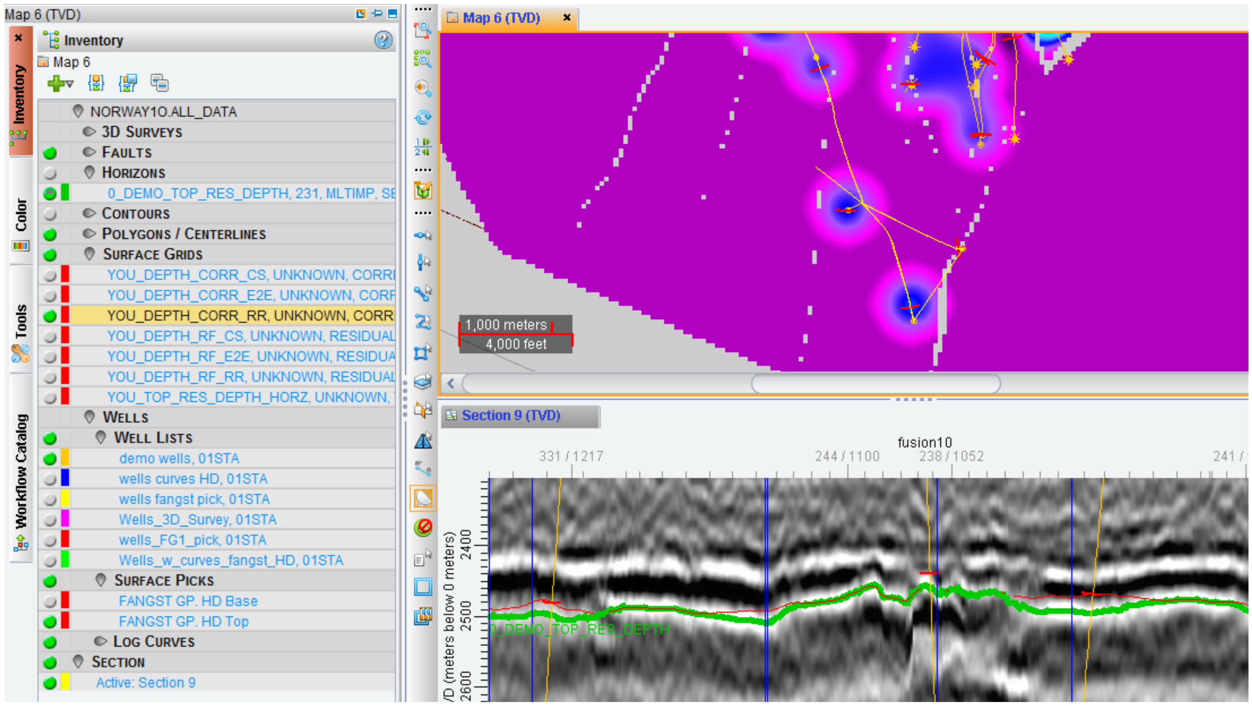
- In the *Output Grid* panel, enter **“YOU_ DEPTH _RF_RR”** in Map Data Set text field.

- In the *Output Correction Grid* panel, enter **"YOU_DEPTH_CORR_RR"** in the Map Data Set text field. Click **Apply**

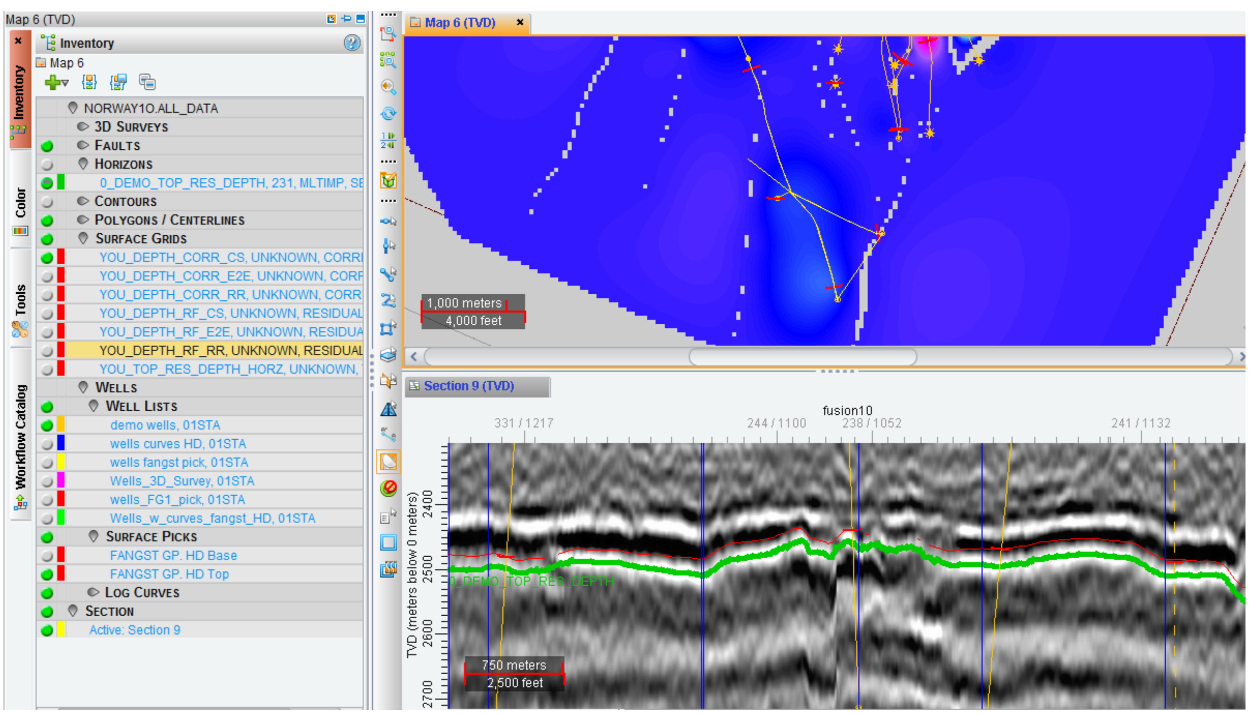


11. In the *Inventory* task panel of the *Section* view, toggle on Surface Grid **YOU_DEPTH_RF_RR**. In *Map* view, toggle of all horizons

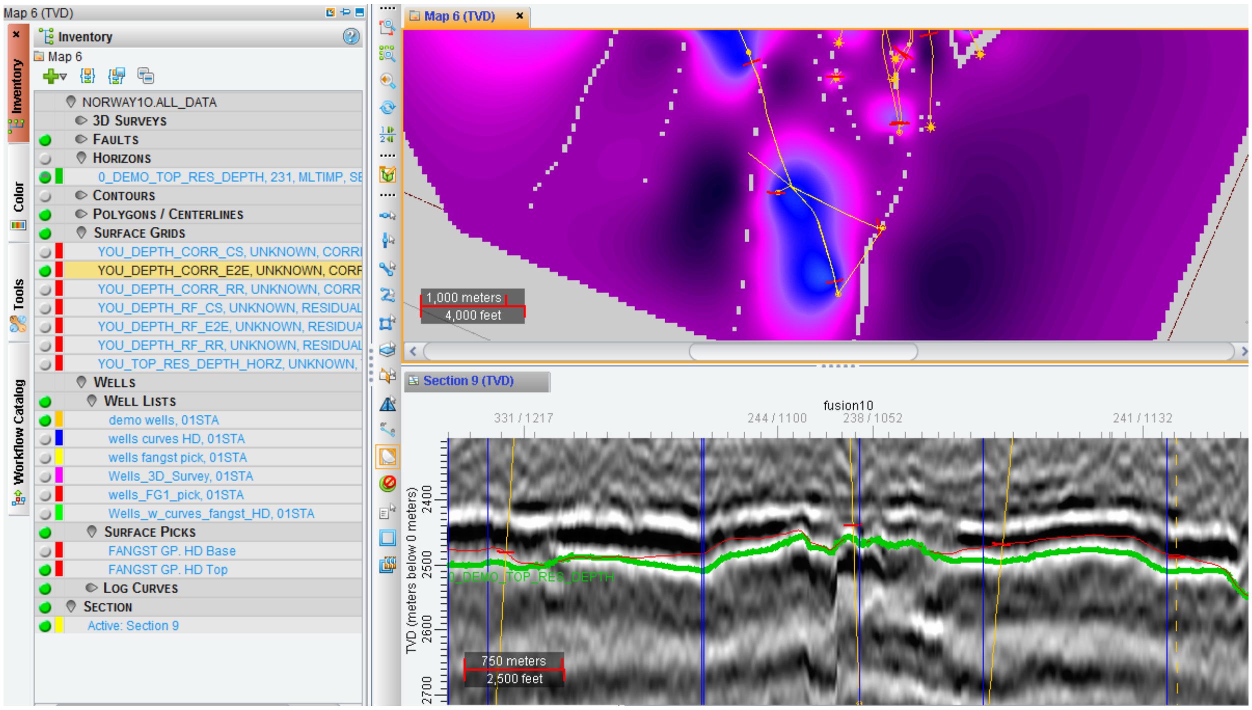
and toggle on surface grid **YOU_DEPTH_CORR_RR**. Compare these two views to see how the residual fit operation worked.



12. In the *Inventory* task pane of the *Section* view, toggle on surface grid **YOU_DEPTH_RF_CS**. In *Map* view, toggle off all horizons and toggle on surface grid **YOU_DEPTH_CORR_CS**.



13. In the *Inventory* task panes of the *Section* view, toggle on surface grid **YOU_DEPTH_RF_E2E.** In *Map* view, toggle on surface grid **YOU_DEPTH_CORR_E2E.**



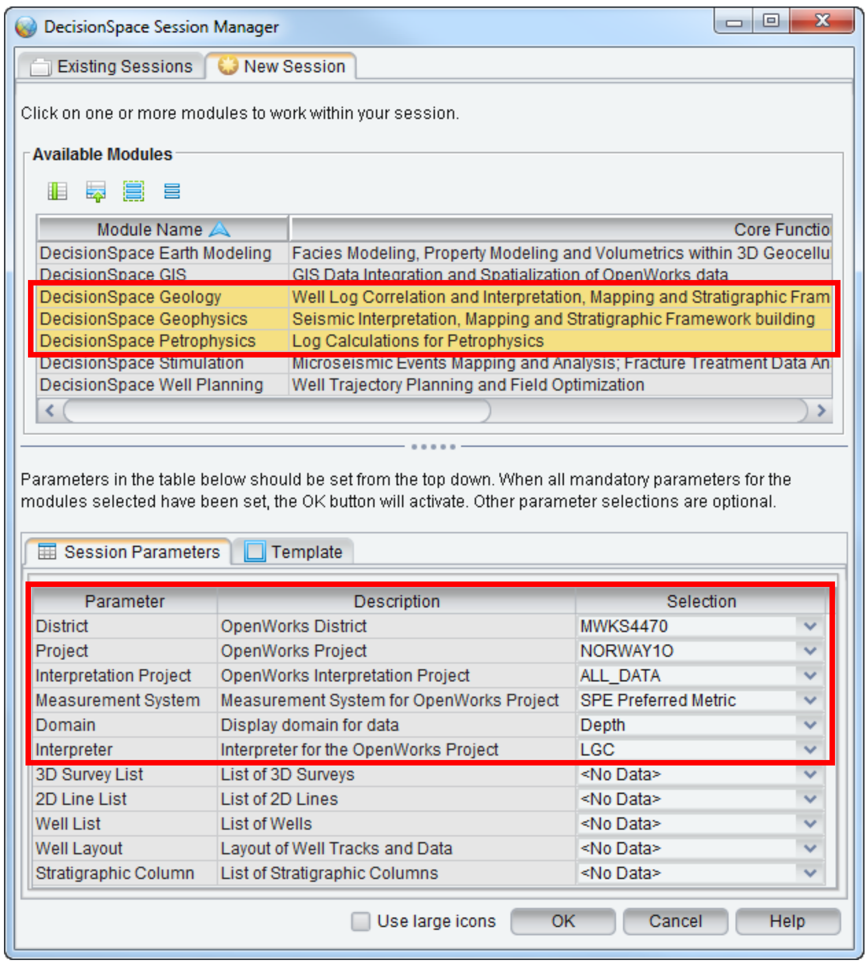
> **Note:**
>
> To change the color of your surface grids, MB3 > Display Properties. You must be in *Section* view to change the color of the surface grid line.

14. To close the session, select **File > Exit.**

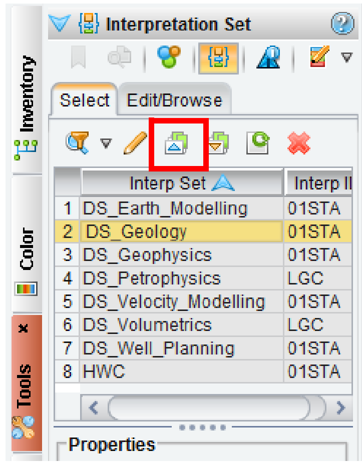
Appendix B
Wellbore Analyzer

The Wellbore Analyzer (WBA) is a tool for computing the intersections between wells or well plans with various data types to create a surface pick or a log curve. The WBA creates log curves or surface picks, depending on the data type the well or well plan intersects. Selecting surface grids or horizons in the data tree creates surface picks; selecting 3D seismic and 3D grids in the data tree creates log curves.

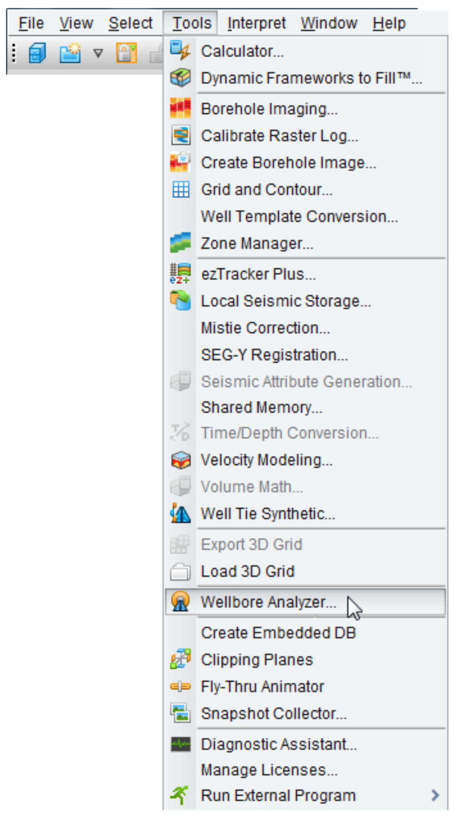
1. In the *DecisionSpace Session Manager* select the following modules and input the following session parameters, then click **OK**.



2. In the *Tools* task pane click the **Interpretation Set** ( ) icon. Select the **DS_Geology** Interp Set and click the **Load Data to Session** ( ) icon.



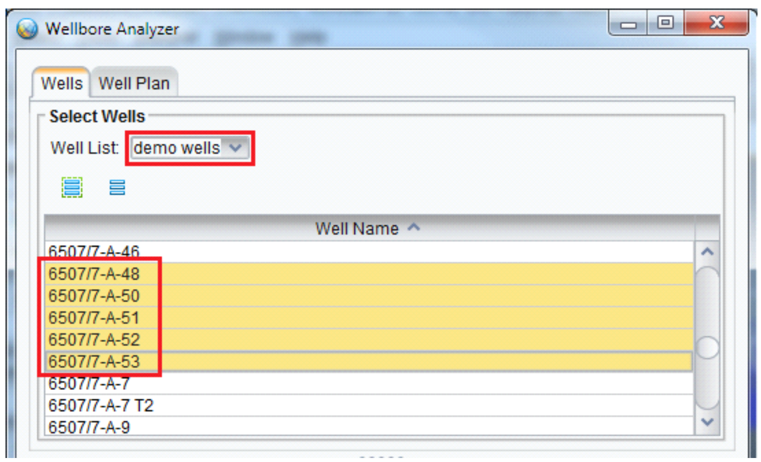
3. On the menu bar, click **Tools > Wellbore Analyzer**.



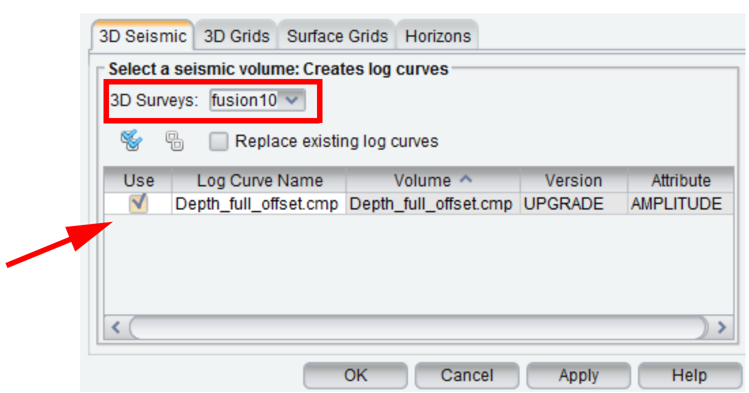
> **Note:**
>
> The *Wellbore Analyzer* dialog is divided into two parts. On the upper panel you can select the wells or well plans along which you desire to run the tool. On the bottom panel you can select the data type you want to use for your analysis. The *Wells* tab is highlighted by default, and in the Well List and in the pull-down menu the well lists available in the session are displayed.

4. In the *Wellbore Analyzer* dialog, select Well List **demo wells**. In the Well Name table, select the following wells:

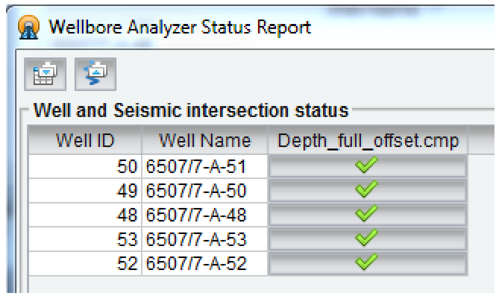
   6507/7-A-48
   6507/7-A-50
   6507/7-A-51
   6507/7-A-52
   6507/7-A-53



5. In the *3D Seismic* data type tab, select 3D Surveys: **fusion10.** Click the **checkbox** associated with Curve Name Depth_full_offset.cmp. Click **Apply**.



6. In the *Wellbore Analyzer Status Report*, check on the **progress** of the tool run.



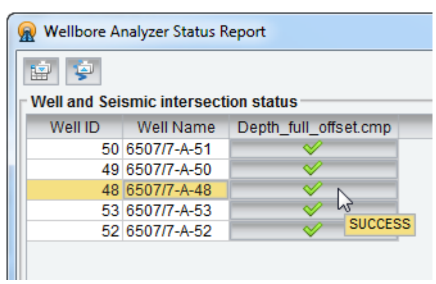
Note:

The *Wellbore Analyzer Status Report* indicates whether each well or well plan crosses the selected data and whether the intersection was successful. If the well intersected the selected data, a green check mark ( ) is displayed. If the well did not intersect the selected data, a No Intersection icon ( ) is displayed.

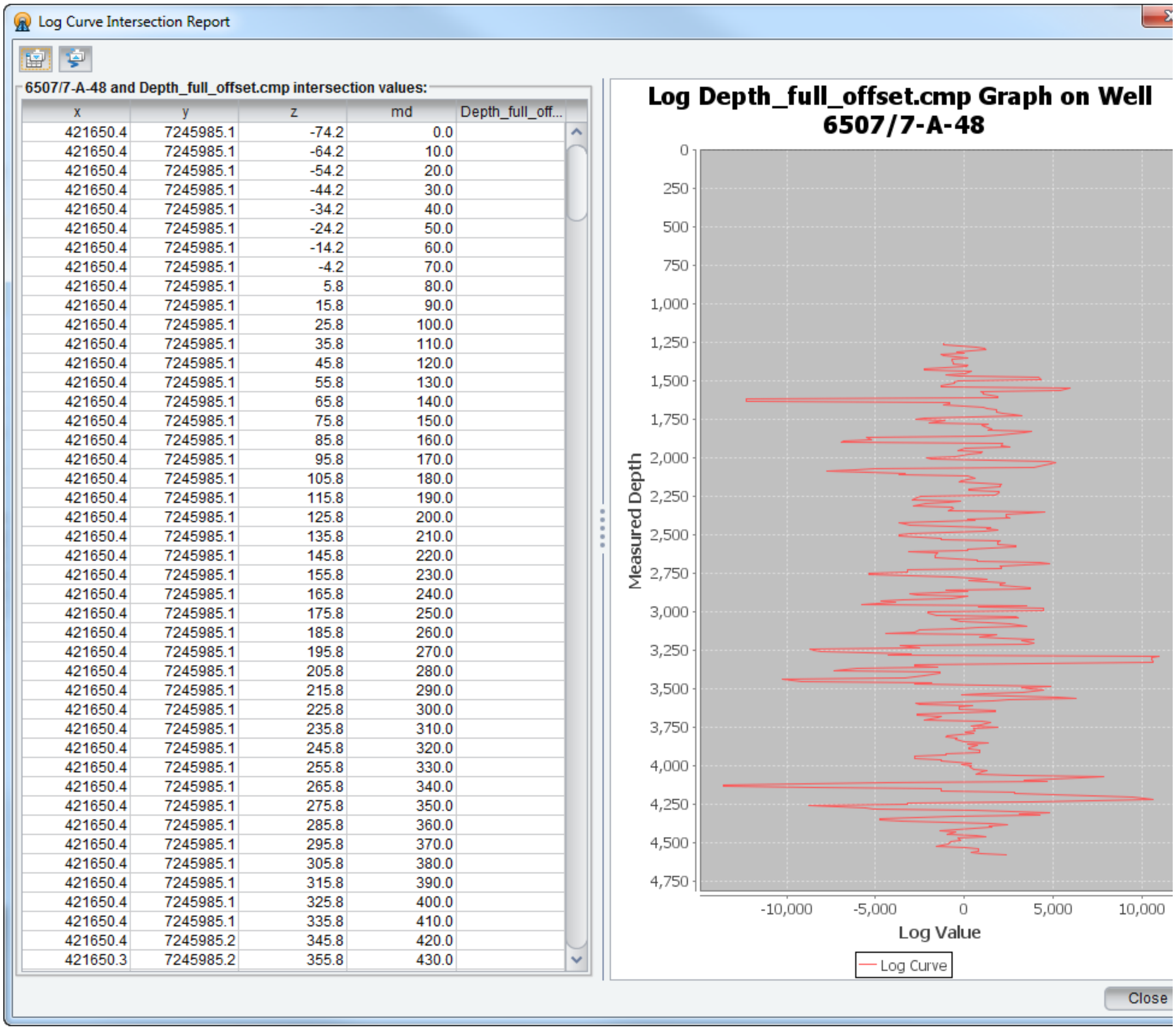
Note:

These values can be exported using the **Export all CSV** ( ) or **Export all LAS** ( ) icons.

7. Click the **Depth_full_offset.cmp** ( ) button associated with well 6507/7-A-48.

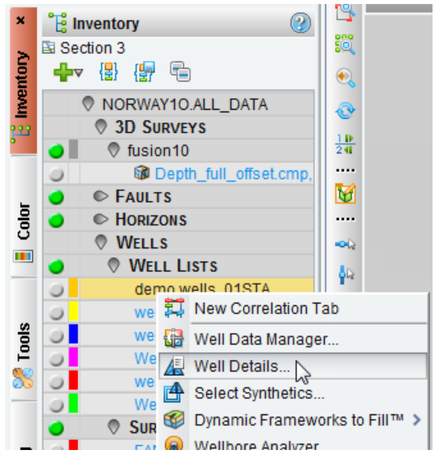


8. In the *Log Curve Intersection Report*, the X,Y,Z and MD values are displayed in a table along with a graphical preview of the log curve. Click **Close**.



9. In the *Wellbore Analyzer Status Report*, click **Close**.

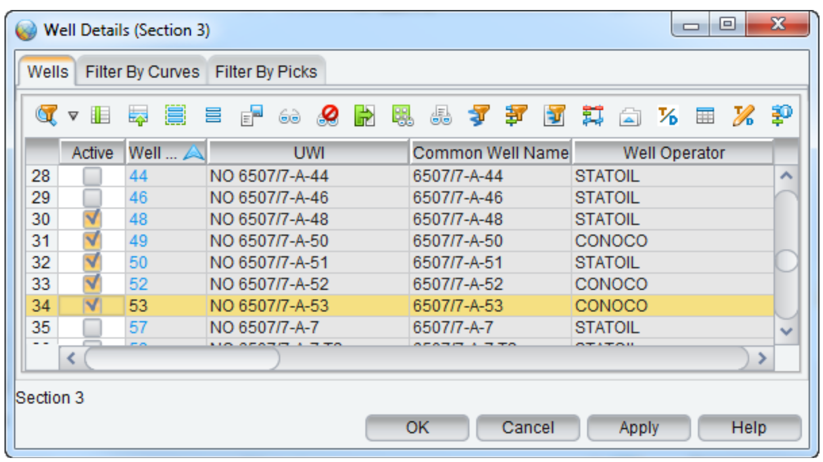
10. In the *Inventory* task pane of the *Section* view, put your cursor on **demo wells** and **MB3 > Well Details.**



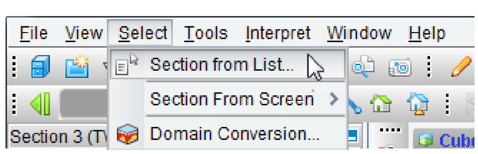
11. In the Active column of the *Well Details* dialog, uncheck **all wells** except the following:

6507/7-A-48
6507/7-A-50
6507/7-A-51
6507/7-A-52
6507/7-A-53

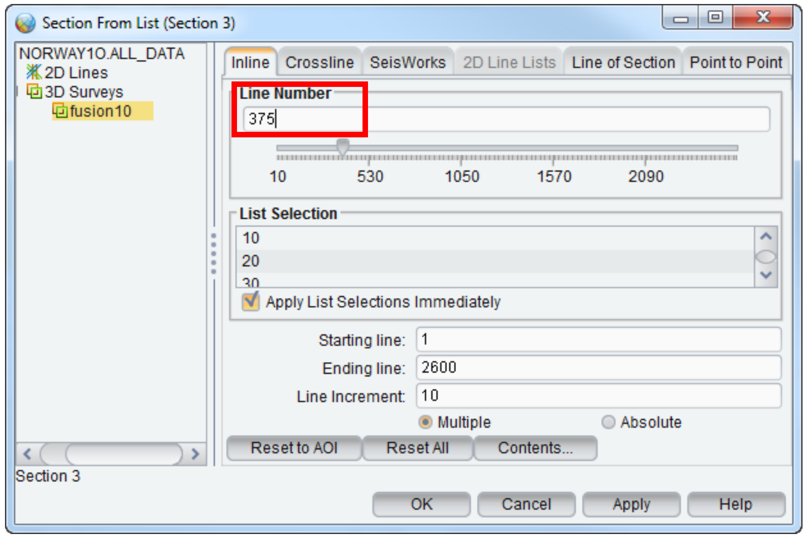
12. In the *Well Details* dialog, click **OK**.



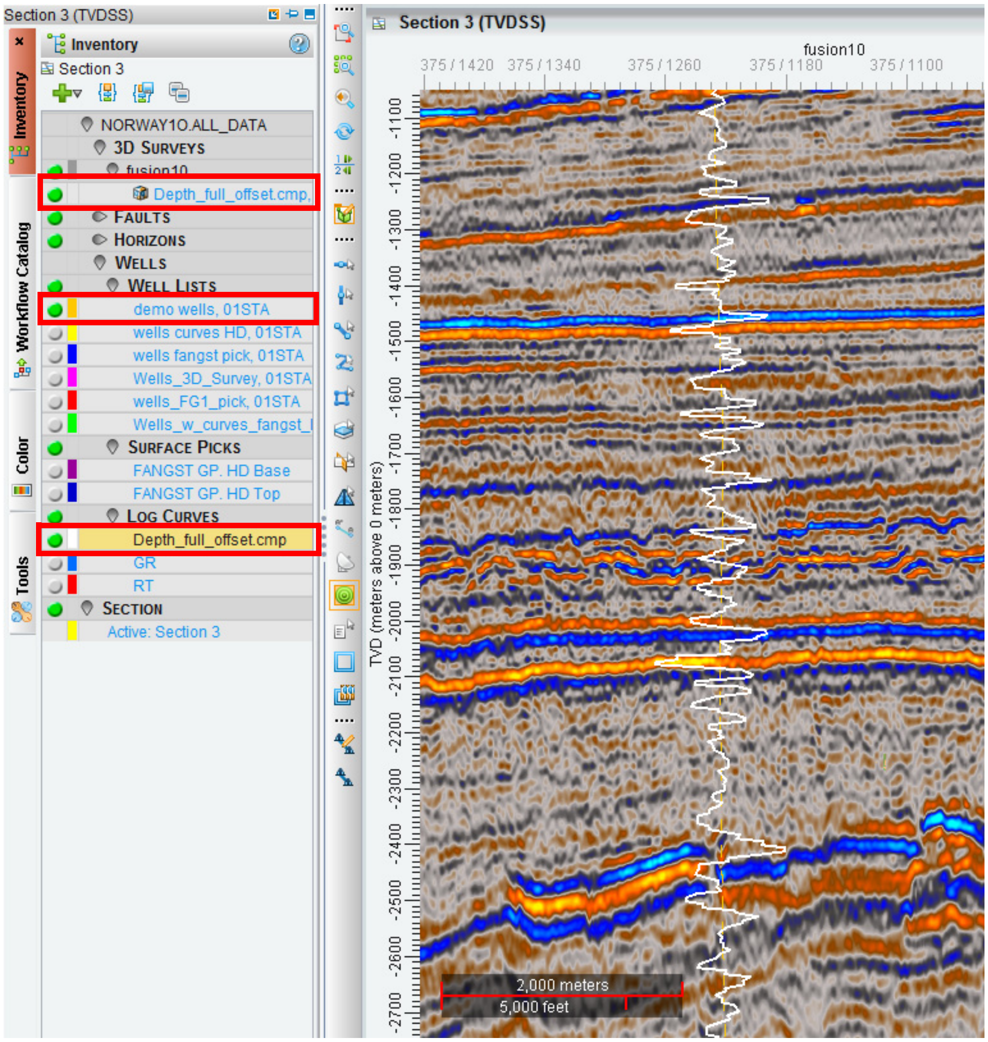
13. In *Section* view, choose **Select > Section from List**.



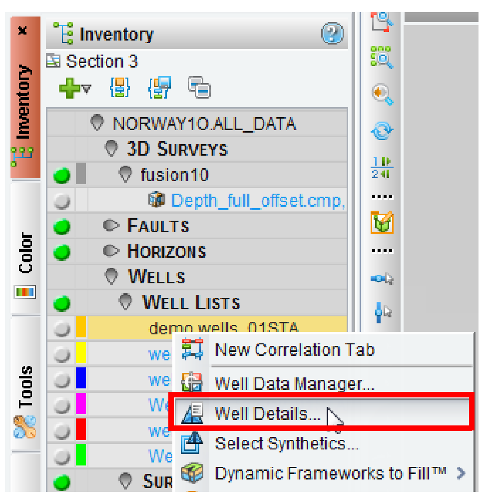
14. In the *Inline* tab of the *Selection from List* dialog, enter **“375”** in the Line Number text field. Click **OK**.



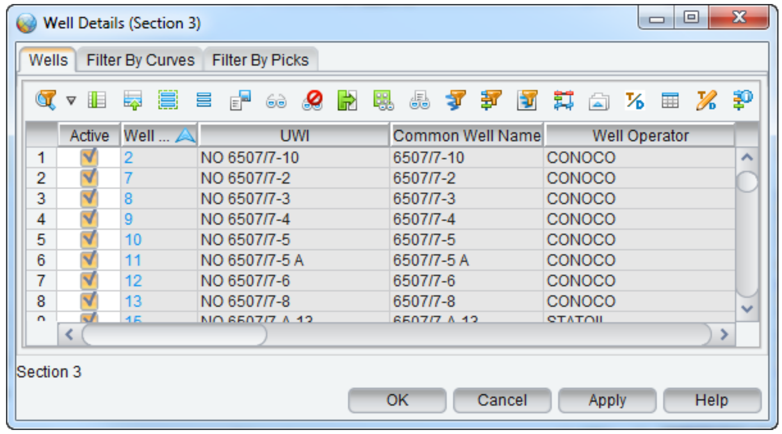
15. In the *Inventory,* toggle on seismic volume **Depth_full_offset.cmp,** well list **demo wells,** and log curve **Depth_full_offset.cmp**.



16. In the *Inventory* task pane, put your cursor on **demo wells** and **MB3 > Well Details.**



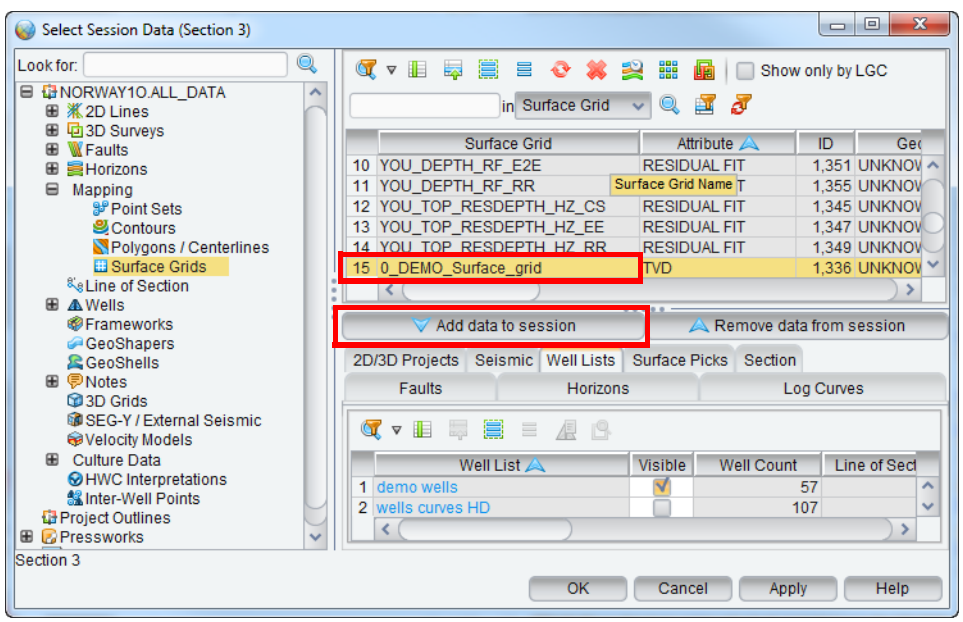
17. In the *Well Details* dialog, toggle on **all wells** in the Active column. Click **OK**.



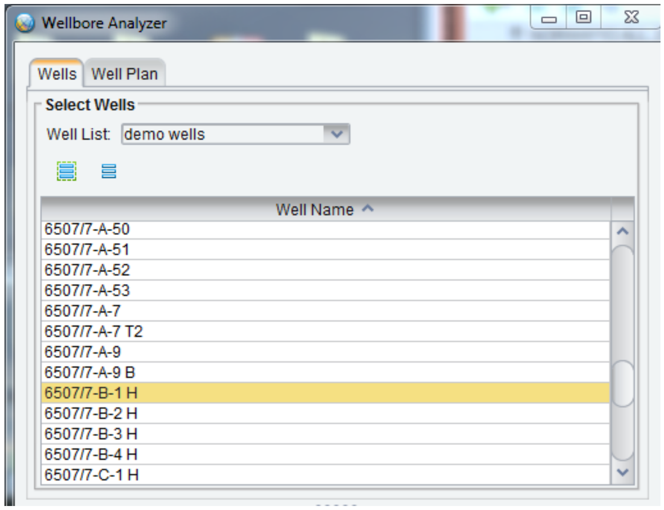
Creating Surface Picks with Surface Grids

You will now use WBA to digitize surface picks on new wells that intersection the existing surface grid.

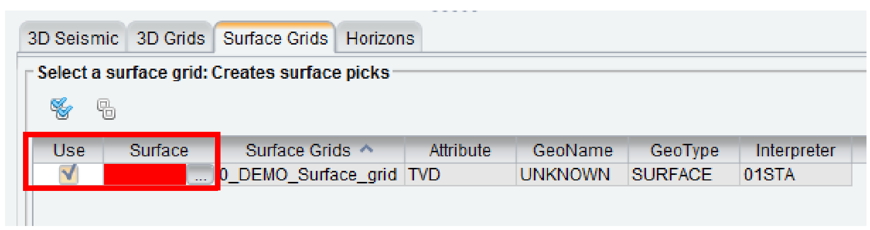
18. In the menu bar, click the **Select Session Data** ( ) icon. In the *Select Session Data* dialog, select surface grid **0_DEMO_Surface_grid**, then click the **Add data to session** button. Click **OK.**



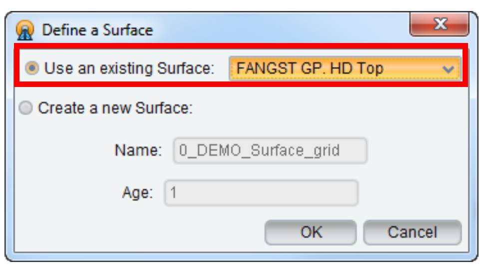
19. In the *Wellbore Analyzer* dialog, **<Ctrl>** + **MB1** on wells **6507/7-A-35** and **6507/7-B-1 H**.



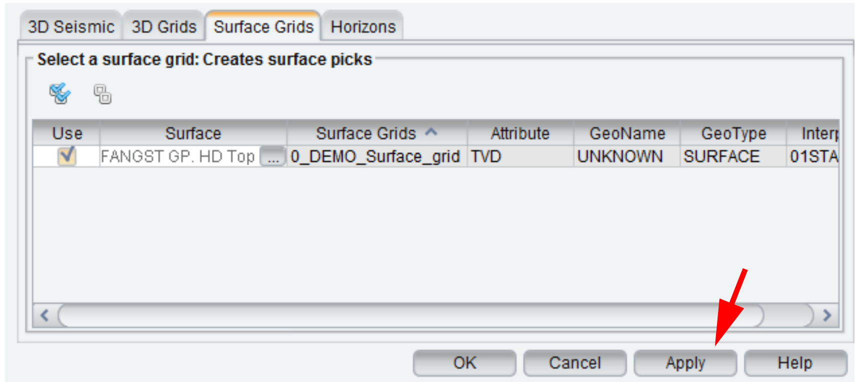
20. In the *Surface Grids* data tab, toggle on the **Use cell** associated with Surface Grids 0_DEMO_Surface_Grid. Click the **Surface** column.



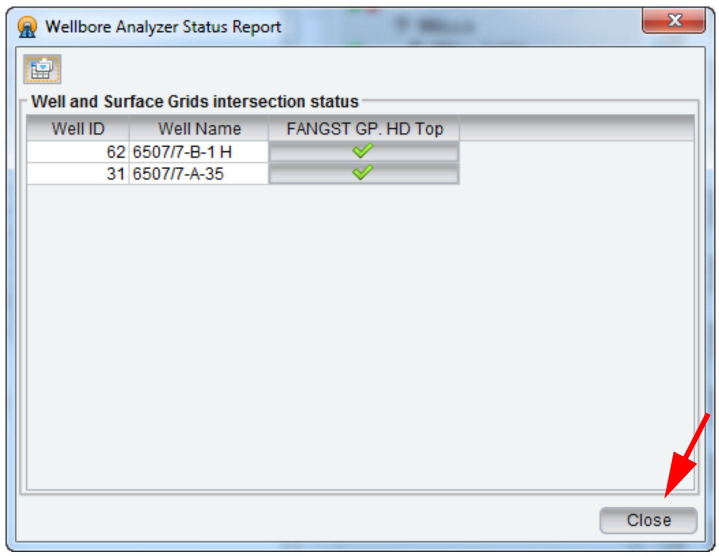
21. In the *Define a Surface* dialog, toggle on **Use an existing Surface:** pull-down menu, select **FANGST GP.HD Top**. Click **OK**.



22. In the *Wellbore Analyzer* dialog, click **Apply**.



23. In the *Wellbore Analyzer Status Report*, click **Close**.



Wells 7-B-1 H and 7-A-35 now have a FANGST GP.HD Top surface pick that can be seen in this session. These surface picks have been saved to the OpenWorks database.

24. Select **File > Exit** to end the session.

Appendix C
Horizontal Well Correlation

Horizontal Well Correlation (HWC) is a tool for geosteering and interpreting highly deviated or horizontal wells. The tool works by correlating the logging-while-drilling (LWD) log curves and the measuring-while-drilling (MWD) log curves of the drilling well to predicted log curves that are generated for the drilling and offset wells. The DecisionSpace Geosciences software automatically generates predicted log curves for drilling wells in True Stratigraphic Thickness (TST) and enables geoscientists to correlate with wells in *Horizontal Correlation* view as drilling progresses.

The Topics Covered in This Appendix

In this appendix you will learn to:

- Set up a DSG session for geosteering with HWC.
- Create a predicted curve in the HWC task pane and a Framework reference surface.
- Learn how to use inter-well points in the HWC workflow to update structures.
- Stretch and squeeze a predicted curve for interpretation.
- Geosteer the well using HWC.
- Update the Framework structure while drilling the well (real-time).
- Use target line drawing functionality to edit the well trajectory and generate a Geosteering report.

What is Horizontal Well Correlation?

DecisionSpace Geosciences Horizontal Well Correlation allows geoscientists to correlate the logged information from a nearby offset well with an active drilling well. You can use the correlations to predict structure and update the structural model during drilling. This interpretation uses the correlation workflow to keep the drill bit within the target zone of the reservoir.

The tool uniquely generates predicted log curves based on the stratigraphic position of the well and the log curves of the drilling or offset well. By correlating the predicted log curve to the MWD or LWD curves, you can determine stratigraphically where the well has penetrated.

These correlations can be used to update the structural model (Framework), and revise well plans based on a continuously updated structural model. As you correlate, you can interpret inter-well points that act as an additional data source for the structural model. The software determines the position of each inter-well point, adds them to the surface, and interactively updates the structural framework.

Data Requirements for HWC

The following data is required for HWC:

- One or two surface grids in depth. These surfaces can be created in the Grid and Contour tool or in the *Frameworks to Fill Workspace*. Note that automatic updates will only be possible using a Dynamic Framework to Fill.

- A well list containing key wells (minimum: the drilling and offset wells).

- An input log curve and any other log curves you find useful from the offset wells.

- Surface picks from the offset wells.

- Surface picks for the drilling well, once the drilling well crosses the reference surface.

In addition to above data requirements, it is desirable to have depth seismic and at least one interpreted seismic line available in the session.

Working With HWC

TST Type Log Generation

Horizontal Well Correlation is performed by correlating the real-time MWD log curves and the LWD log curves of the drilling well to Type logs generated for the drilling and offset wells. In many cases, the closest and most accurate Type log is created from the drilling well itself. These correlations require that the drilling well has landed and is drilling up dip. During the initial transition of the well from vertical to horizontal and when drilling down dip into unpenetrated strata, correlations must be performed with the Type log of an offset well.

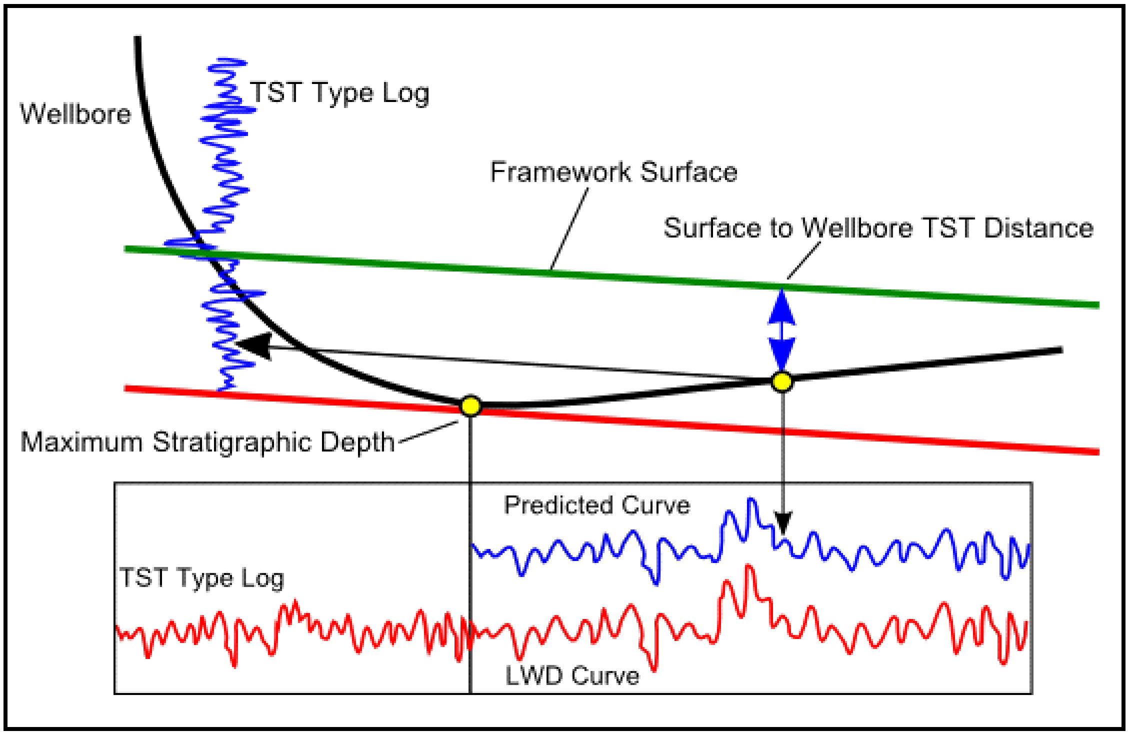
The Type logs for drilling and offset wells are generated in TST (relative to the Framework reference surface). To calculate TST, the wellbore to surface True Vertical Depth (TVD) and surface dip values are derived at each point in the well. TST is calculated using the relationship:

$$\mathrm{TST} = \mathrm{TVD} \times \cos(\mathrm{dip})$$

In the wellbore heel region (transition region of wellbore vertical to horizontal), the maximum TST value determines the Maximum Stratigraphic Depth (MSD) position of the well. TST Type logs are generated from log curves in the wellbore heel region (down to the MSD) by projecting LWD log curves into TST depth. Offset well TST Type logs are also generated using the described procedure. When offset wells are used with a drilling well, the Type log of the offset well is automatically depth-shifted according to the reference surface.

Predicted Curves

Horizontal Correlation view displays the LWD log curves of the drilling well with Predicted Log Curves. Predicted Log Curves are the expected log curve response based on the TST Type log and the wellbore’s stratigraphic position. The default horizontal scale for the *Horizontal Correlation* view is True Horizontal Distance (THD), which is the horizontal distance the wellbore travels from the surface location. Predicted Log Curves are calculated based on the surface to wellbore TST distance and the drilling or offset well TST Type log values. The TST distance determines the relative position on the Type log and the corresponding Predicted Curve value at each position on the wellbore.



In the wellbore heel to MSD region, the Predicted Curve and LWD curves from the Type log of the drilling well are identical. HWC requires an offset well Type log during the initial transition of the well from vertical to horizontal or when drilling down dip into un-penetrated strata. Additionally, the Predicted Curve will be null if the drilling well TST depth is greater than the well or offset well Type log. The Predicted Curve will also be null if the Framework Surface is absent due to an unconformity, fault, or the drilling well extending beyond the AOI of the Framework surface.

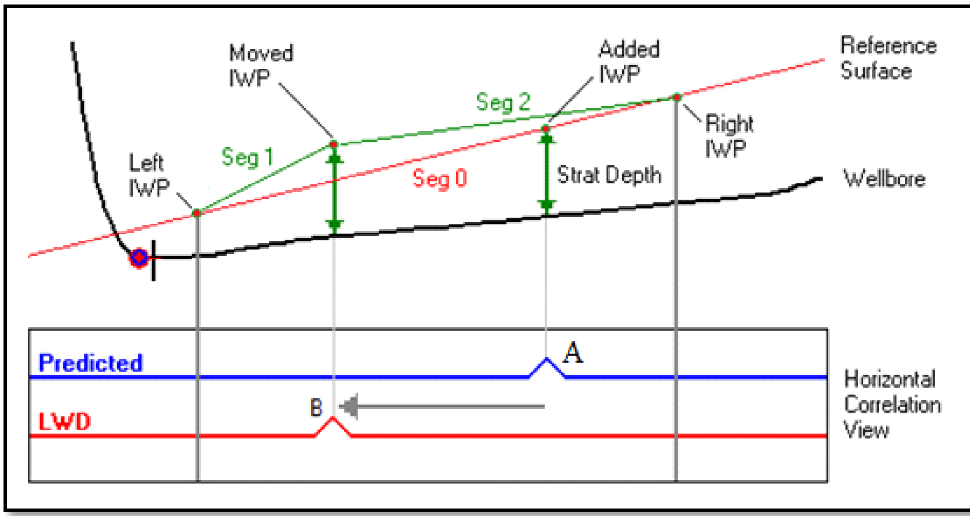
Horizontal Well Correlation

In HWC interpretation mode, the Predicted curve is correlated to the real-time MWD and LWD log curves. The interpretation mode displays a track outline with Anchor lines tied to the reference surface picks and inter-well points. Anchor lines are designed to stretch and squeeze the Predicted Curve to match the LWD curve. Additionally, these log correlations reposition inter-well points horizontally and vertically in such a way that the wellbore stratigraphic depth matches the correlation.

The following figure exemplifies the Predicted-MWD-LWD correlation process.

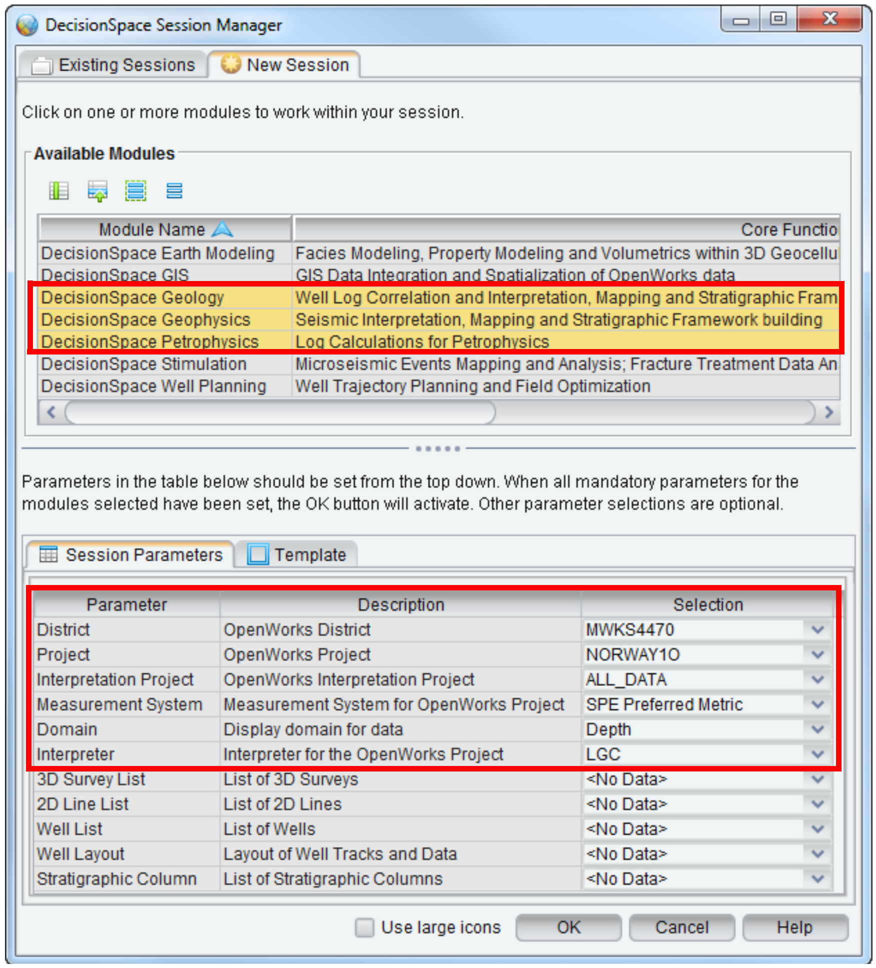
- Add an IWP on the peak or trough of the Predicted log curve, where you wish to correlate to the LWD curve.
- Drag the IWP anchor line to match the IWP log curve.
- To aid in the correlation, an interactive Predicted curve displays between the bounding left and right anchor lines.

In the example below, the peak on the Predicted curve (A) is correlated with the LWD peak (B). Initially add an IWP at the Predicted-curve peak (A) using MB1. Drag the IWP anchor line (A) to the LWD peak (B). In this correlation, the stratigraphic depth at position (A) is moved to position (B). The original reference surface structure shown by the red line (Seg 0) transforms into the correlated surface structure shown by green line segments (Seg 1 and Seg 2).

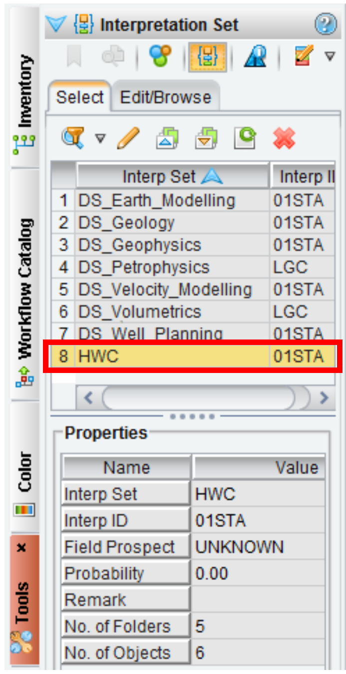


Exercise C.1: Create a Framework for HWC Workflow

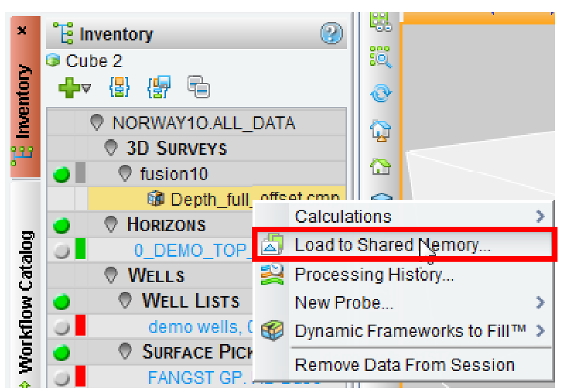
1. On the *DecisionSpace Session Manager*, click the *New Session* tab. On the *New Session* tab, select the modules highlighted below, and enter parameters as shown in the *Session Parameters* tab, below.



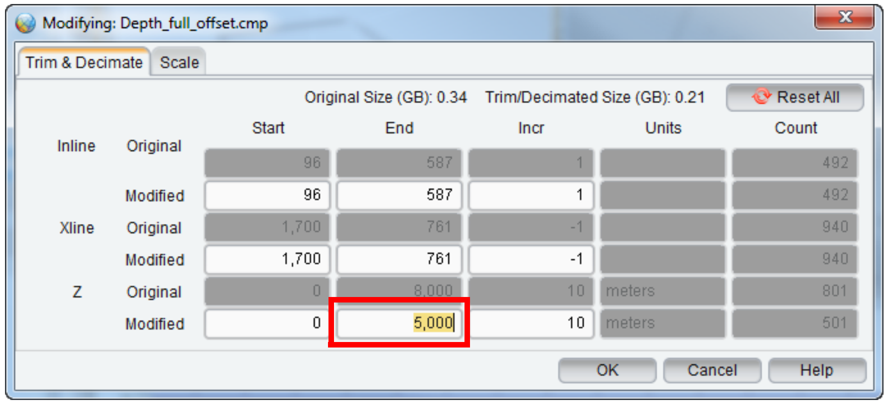
2. Select the **Interpretation Set** ( ) icon on the *Tools* task pane. In the *Select* tab, choose Interp Set **HWC**, then click the **Load to Data to Session** ( ) icon.



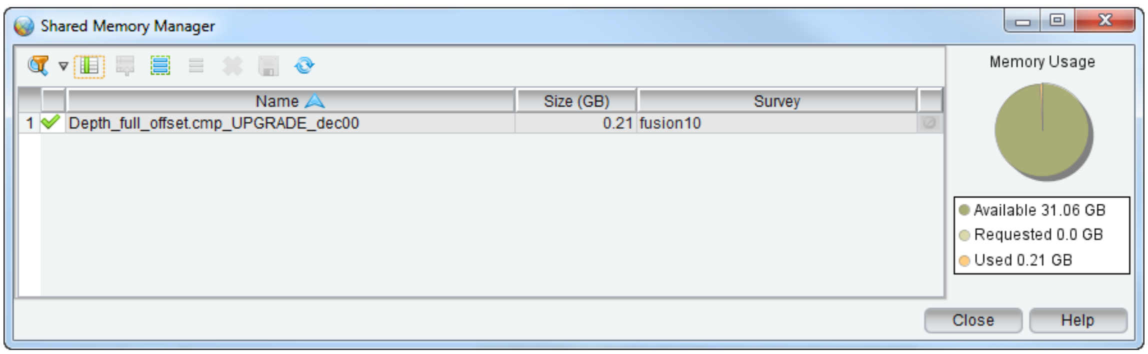
3. In the *Inventory,* put your cursor on seismic volume **Depth_full_offset.com** and **MB3 > Load to Shared Memory.**



4. In the *Modifying* dialog, in the Z Modified row, enter **"5,000"** in the End cell. Click **OK.**

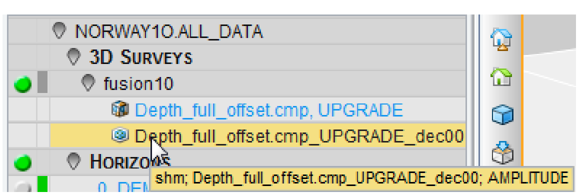


5. The *Shared Memory Manager* dialog appears, displaying progress. When the process is complete, click **Close.**

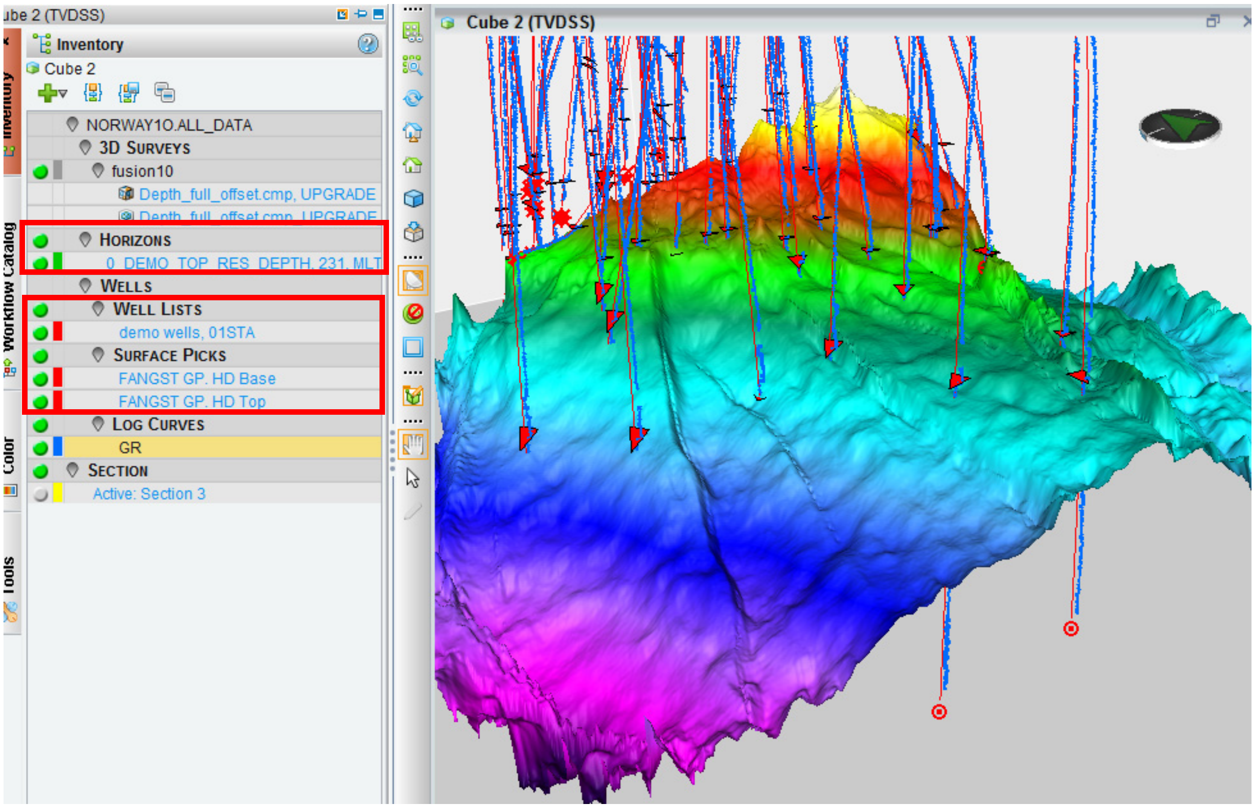


Note:

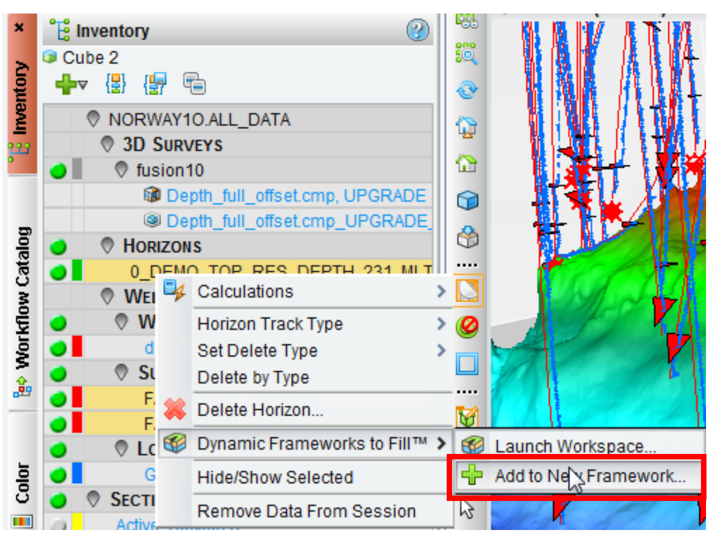
In the *Inventory* task pane, two seismic volumes are listed with similar names. You can tell the difference between seismic volumes from the database and those in Shared Memory. A seismic volume from the database is associated with this icon (). A Shared Memory volume is associated with this icon (), with "_dec" appended to its name, and "shm;" at the beginning of its information pop-up.



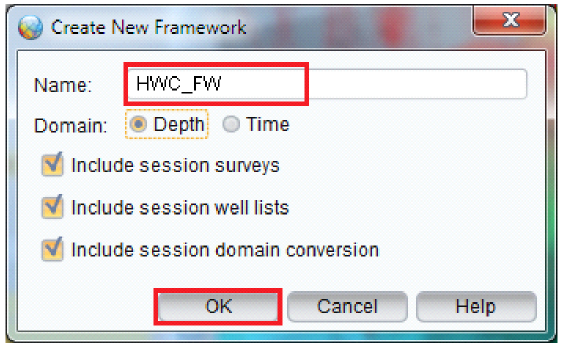
6. In the *Inventory* task pane of the *Cube* view, toggle on **0_DEMO_TOP_RES_DEPTH,** well list **demo wells,** surface picks **FANGST GP.HD Top** and **FANGST GP.HD Base.**



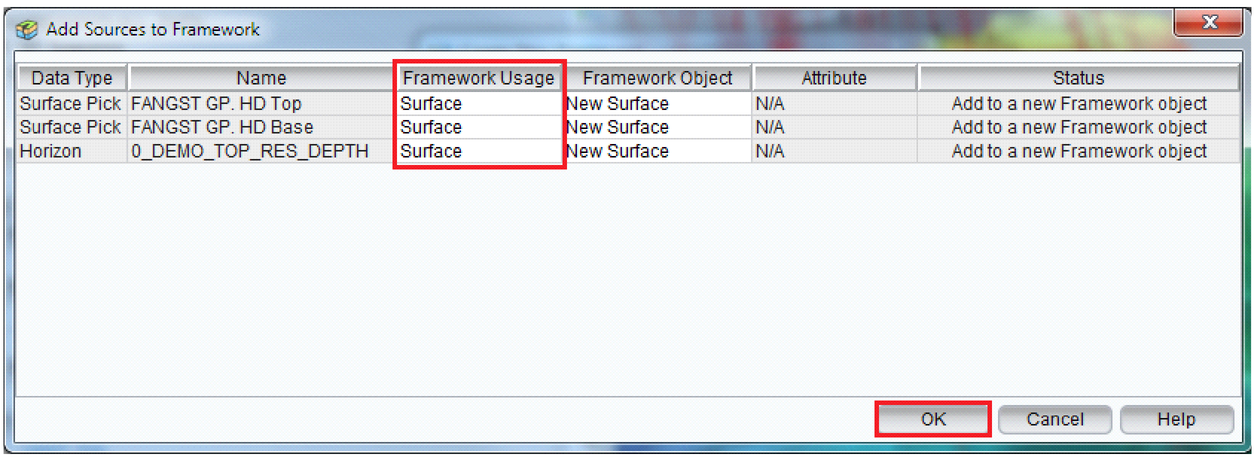
7. In the *Inventory*, select horizon **0_DEMO_TOP_RES_DEPTH** and surface picks **FANGST GP. HD Top** and **FANGST GP. HD Base.** Then, with your cursor on the horizon or one of the surface picks, **MB3 > Dynamic Framework to Fill > Add to New Framework.**



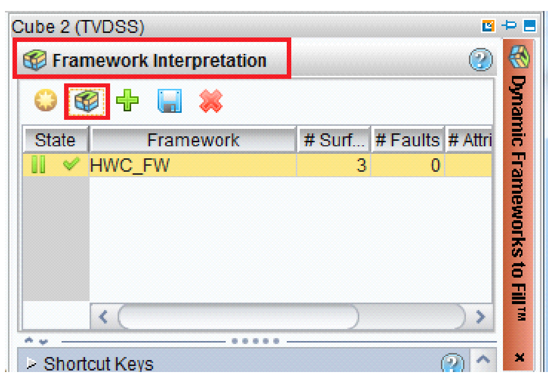
8. In the *Create New Framework* dialog enter **HWC_FW** in the Name: field and click **OK.**



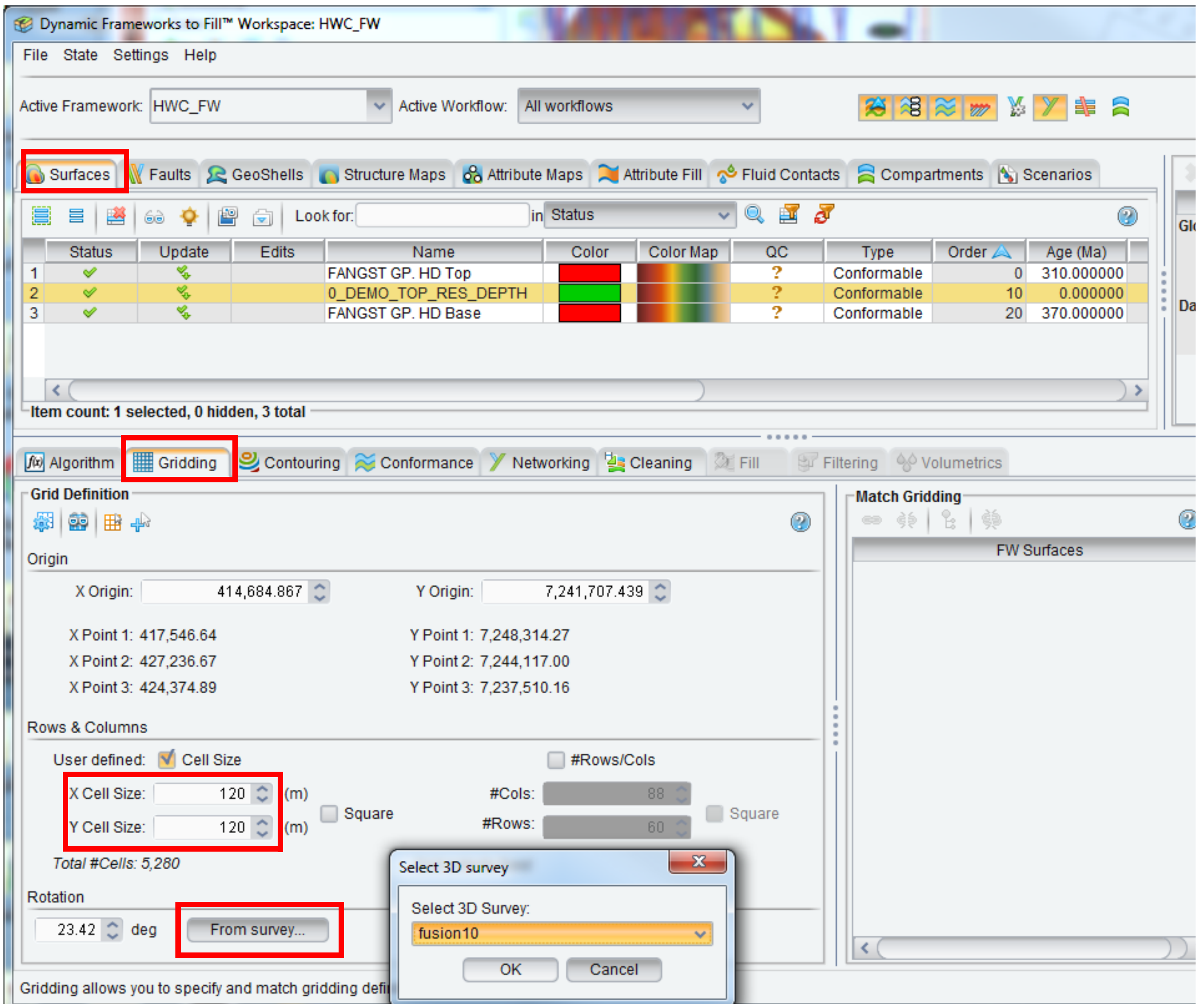
9. In the *Add Sources to Framework* dialog, click **OK.**



10. In the *Dynamic Frameworks to Fill* task pane, click the **Launch Framework Workspace Window** ( ) icon.

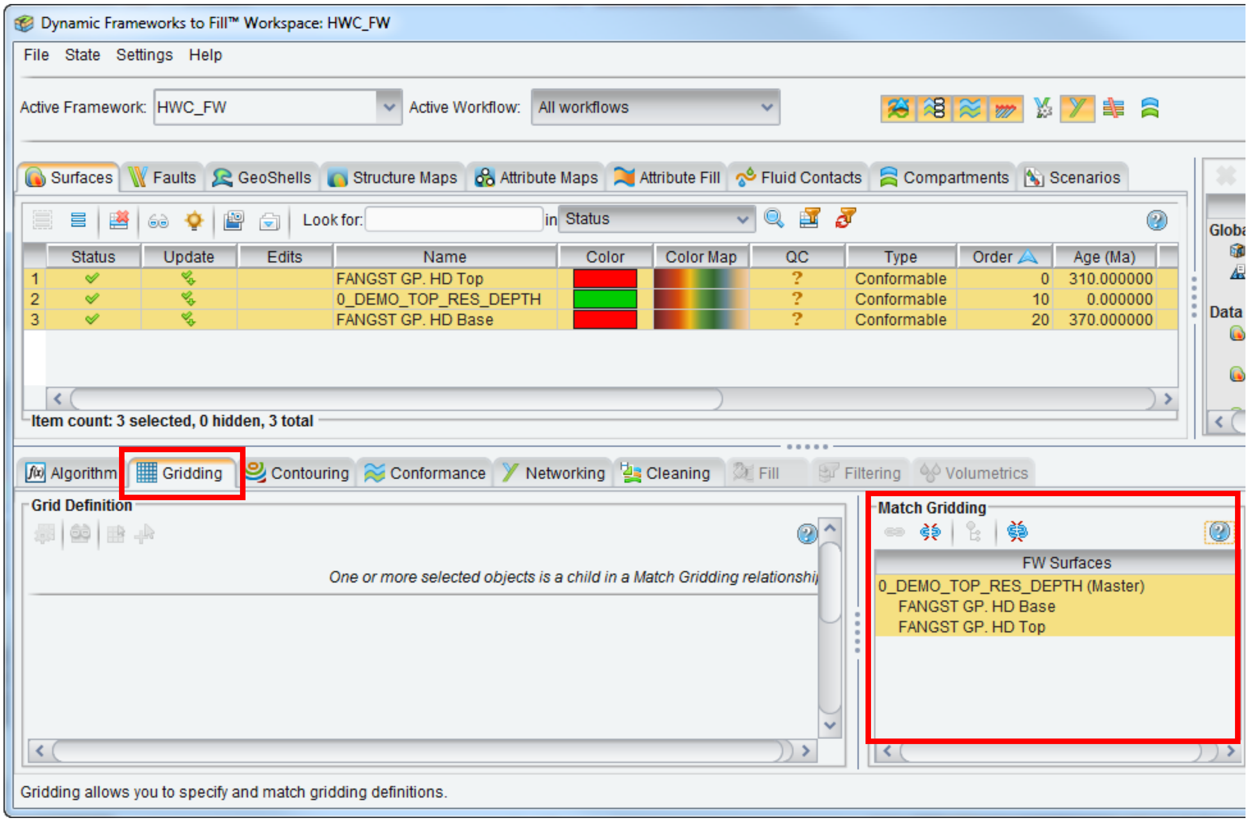


11. In the *Dynamic Framework to Fill Workspace*, select the ***Surfaces*** object tab and select **0_DEMO_TOP_RES_DEPTH**. In the *Gridding* action tab, enter **“120”** in the X Cell Size: field, and enter **“120”** in the Y Cell Size: field. Under Rotation, click the **From a Survey...** button. In the *Select 3D Survey* dialog, Select 3D Survey: **fusion10** and click **OK**.



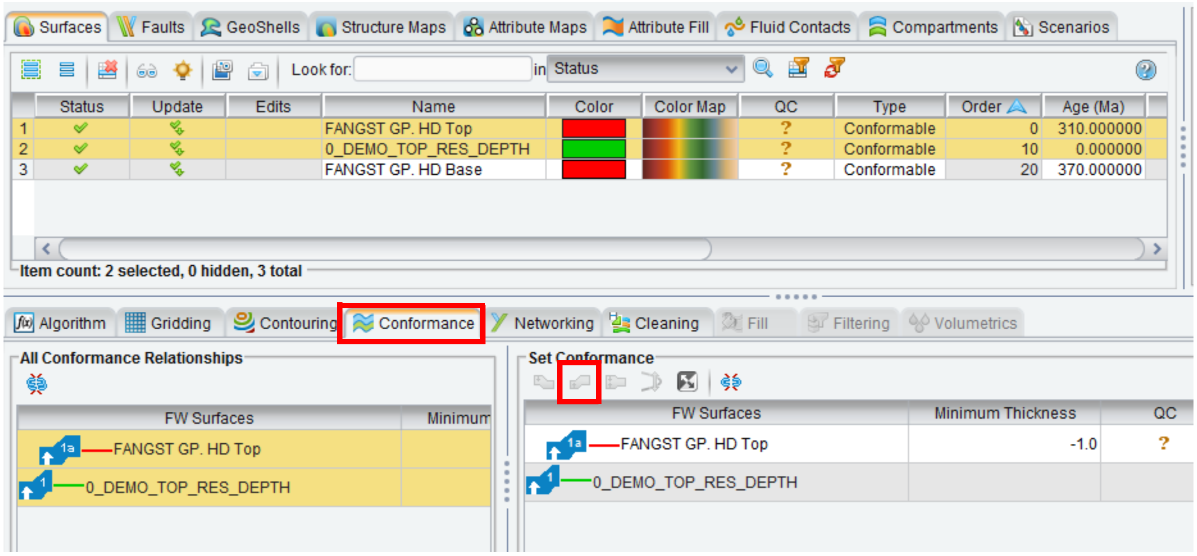
12. In the *Surfaces* object tab, click the **Select All** ( ) icon.

13. In the *Match Gridding* panel of the *Gridding* action tab, click the **Link** ( ) icon to set the **0_DEMO_TO_RES_DEPTH** as master grid.

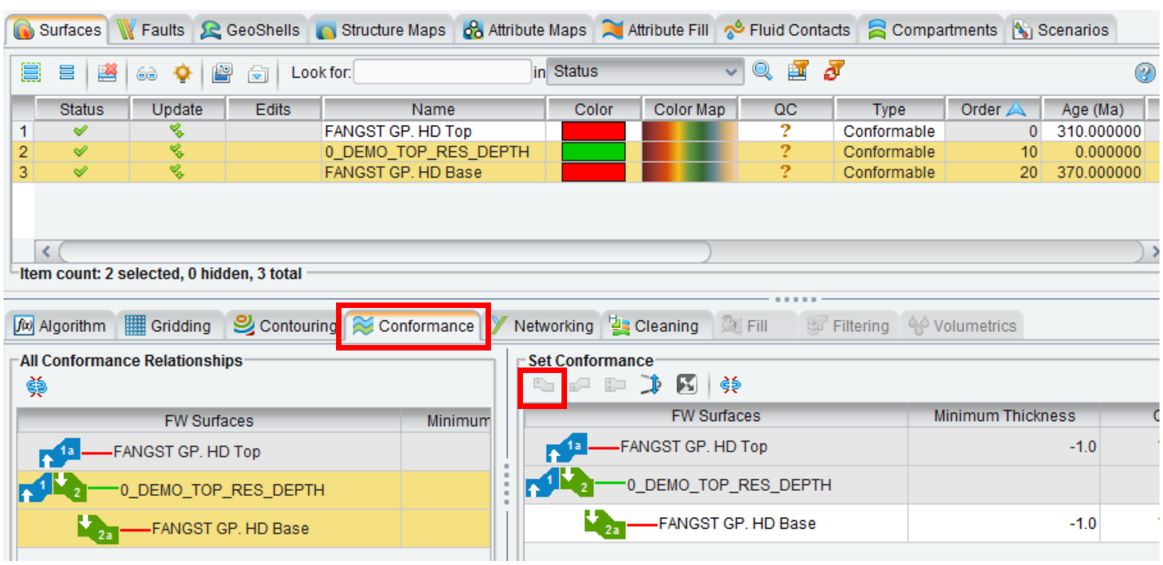


14. In the *Surfaces* object tab of the *Dynamic Framework to Fill Workspace,* select **FANGST GP. HD Top** and **0_DEMO_TOP_RES_DEPTH.** Click the *Conformance* action

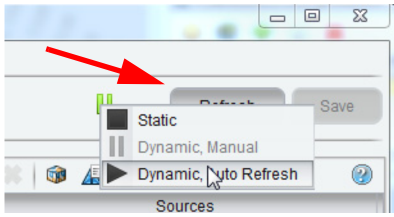
tab. In the *Set Conformance* panel, click the **Conformance Surface bottom up** () icon.



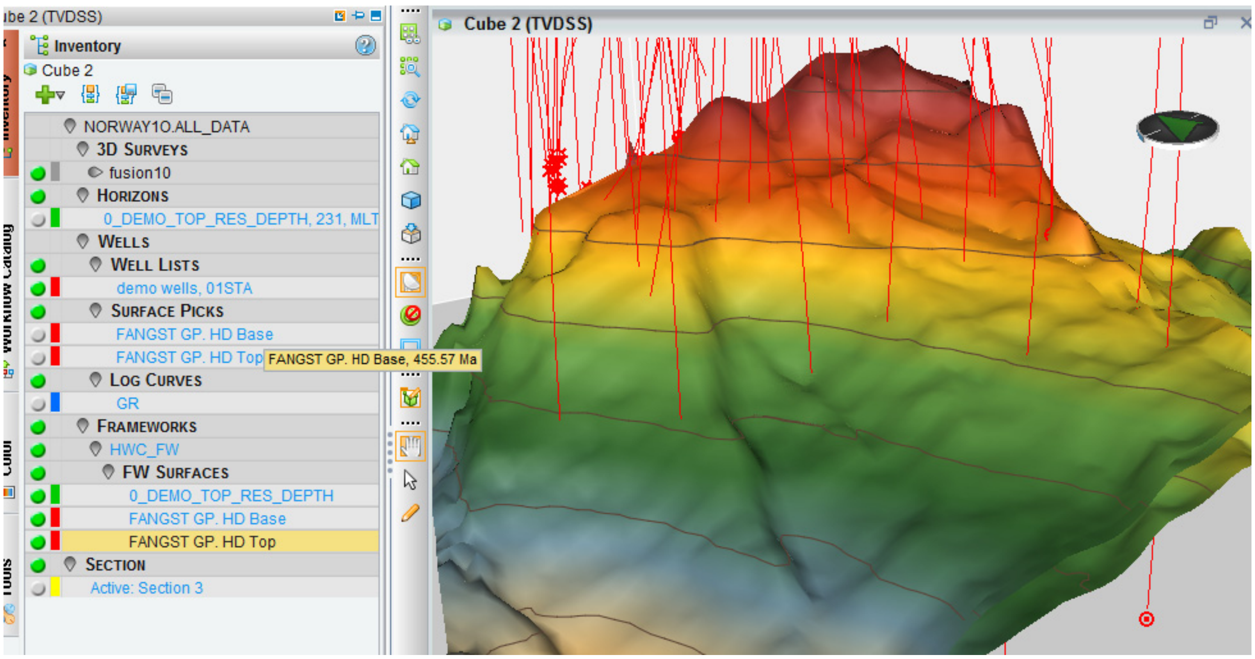
15. In the *Surfaces* object tab, select **0_DEMO_TOP_RES_DEPTH** and **FANGST GP. HD Base,** then in the *Set Conformance* panel of the *Conformance* action tab, click the **Conformance Surface top down** icon ().



16. In the top right corner, click the **Refresh** button. Then change the state to **Dynamic, Auto Refresh**. Click the **Save** button. In the *Save Framework* dialog, click **OK**.



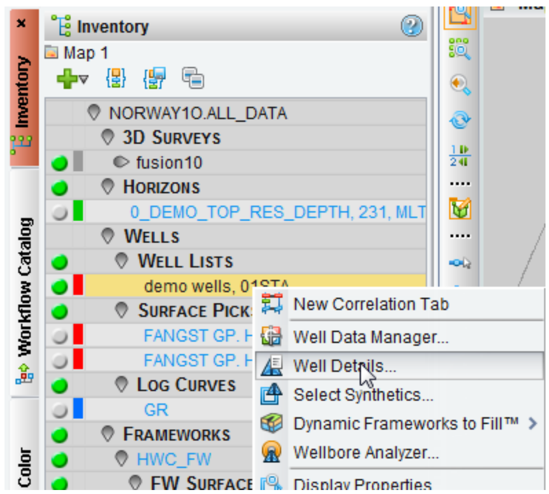
17. In *Cube* view, toggle off **all items** except Wells List demo well the three FW surfaces.



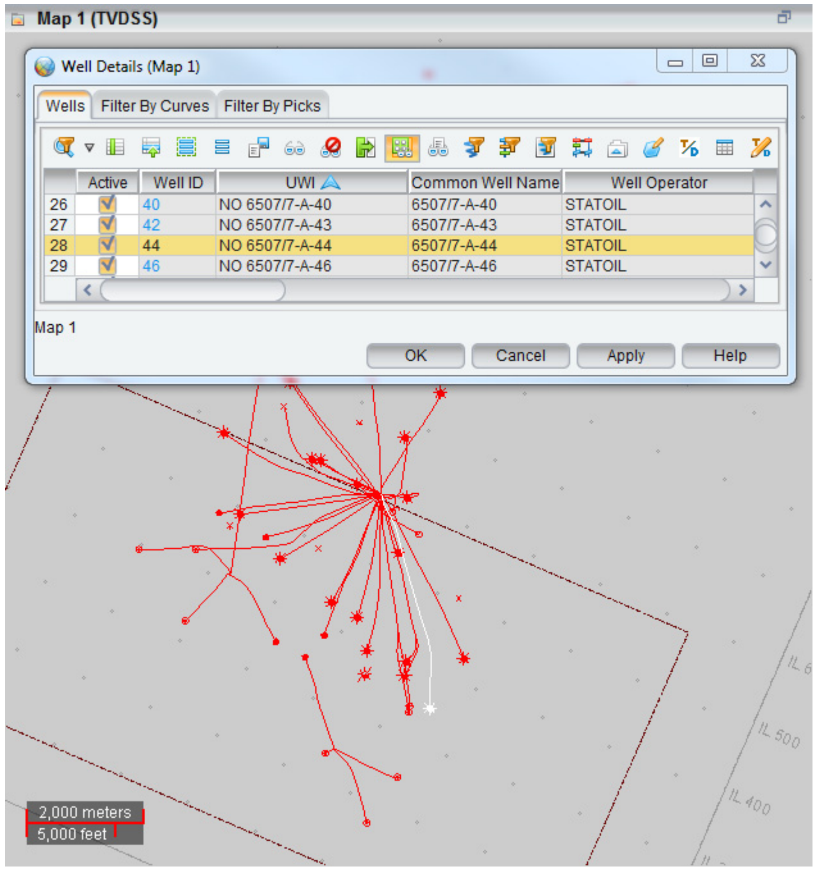
Exercise C.2: Launch and Configure the HWC Workspace

In this exercise you will use two wells: well 6507/7-A-36 and well 6705/7-A-44. Well 6507/7-A-36 is the offset well and well 6705/7-A-44 is the geosteering (active) well for log correlation and structural framework updates. The well is already drilled; however, for this demonstration we will assume that it is actively being drilled.

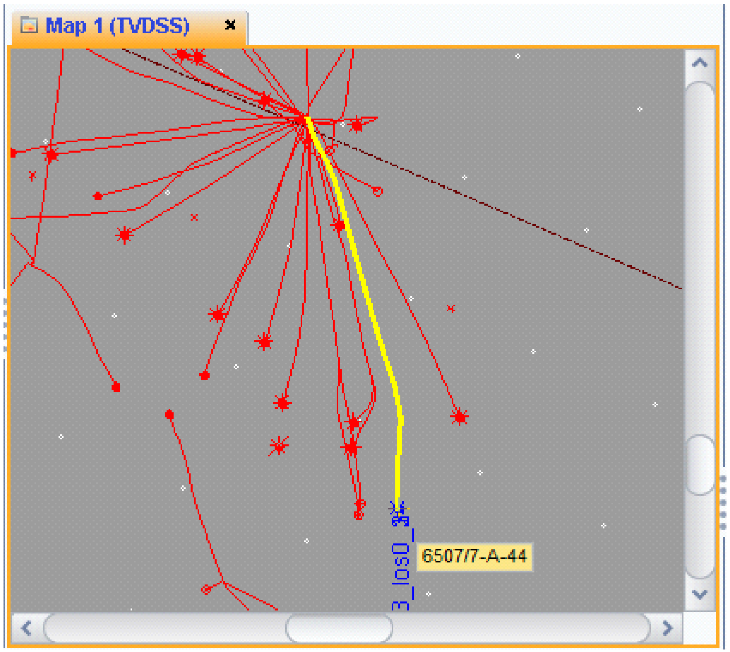
1. In *Map* view, toggle on Wells List **demo wells**. With your cursor on well list, **MB3 > demo wells > Wells Details**.



2. In the *Well Details* dialog, select well **6507/7-A-44** (the well to be geosteered) and click the **Highlight selected wells on the map (** **)** icon**,** to ensure that the correct well has been identified.



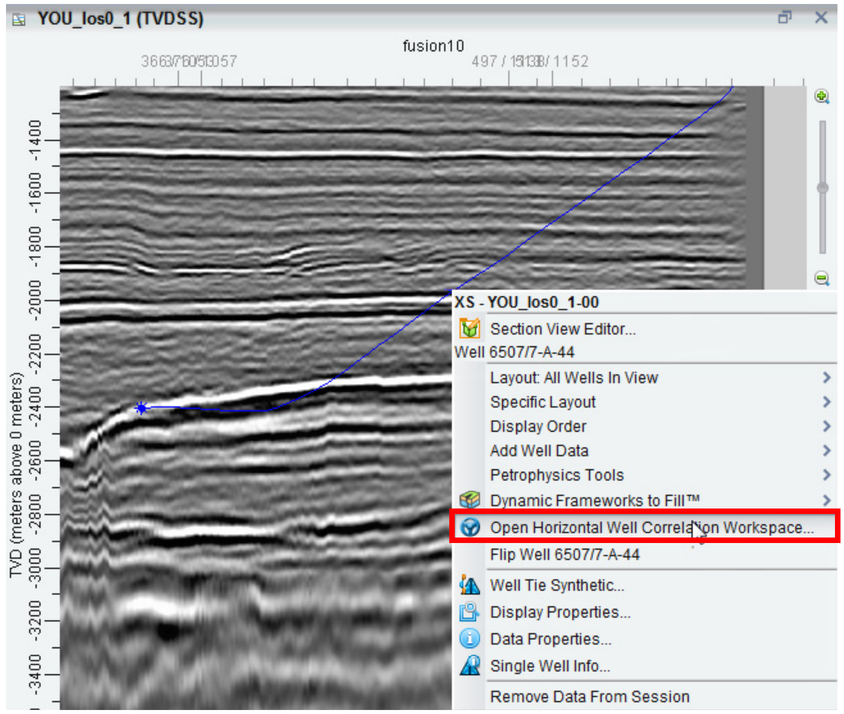
3. On the Vertical tool bar, click the **Create LOS** () icon; and **MB1** to select the highlighted well. Then, **MB2** to broadcast the well path selection to the *Section* view.



4. In the *LOS and Well Projection Settings* dialog, click **Close**.

5. In the *Well Details* dialog, click **OK** to close the window and remove the well highlight.

6. In *Section* view, put your cursor on **6507/7-A-44)** and **MB3 > Open Horizontal Well Correlation Workspace**.



> **Note:**
>
> *Horizontal Well Correlation* workspace can be launched from any view (*Cube*, *Map, Section,* or *Well Correlation* views).

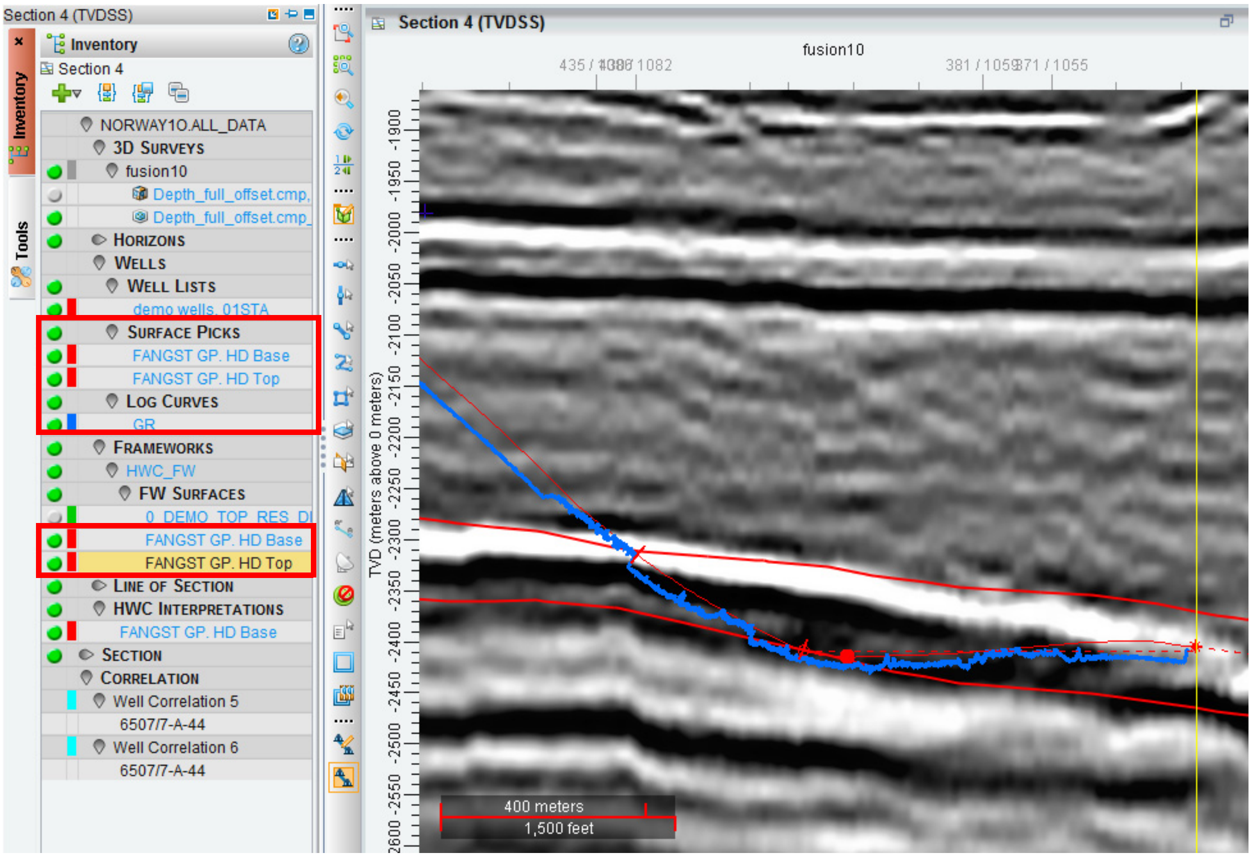
The *Horizontal Well Correlation* workspace is initialized in a separate DecisionSpace Geosciences window, with three view tabs:

- Vertical well correlation in TST
- *Section* view in TVDSS
- *Correlation* view in True Horizontal Distance (THD)

7. In the *Inventory* task pane of the *Section* view, toggle on the following:

- Well list **demo wells, 01STA**
- Log curve **GR**

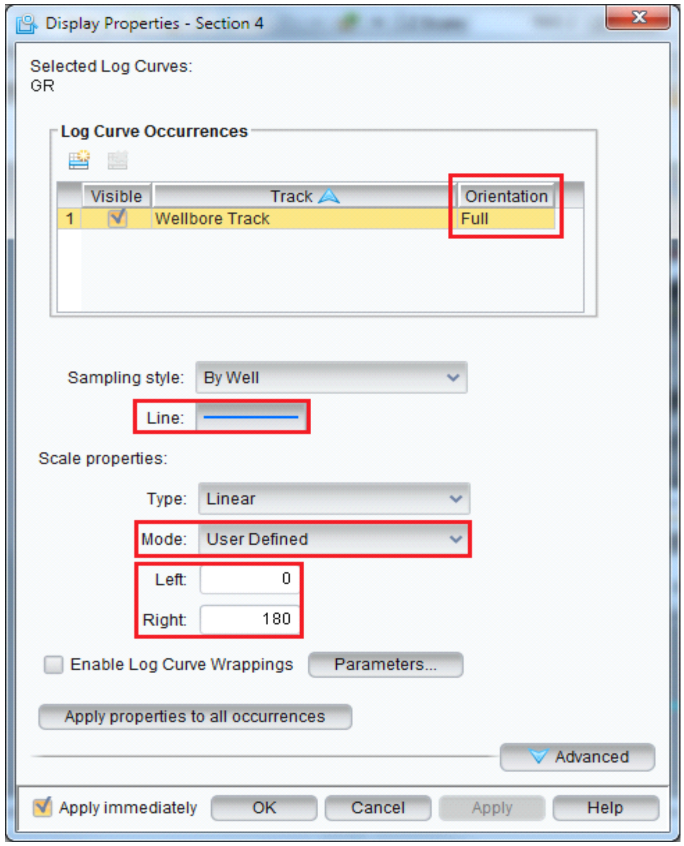
- Surface picks **FANGST GP.HD Top** and **FANGST GP.HD Base**
- Framework surfaces **FANGST GP.HD Top** and **FANGST GP.HD Base**



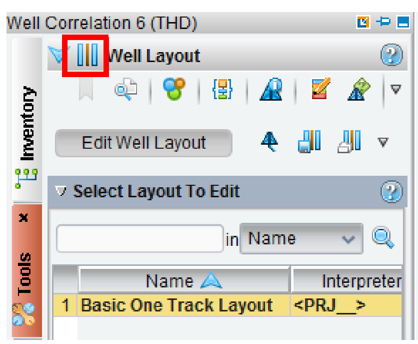
> **Note:**
>
> It may be helpful to change the display of your seismic volume to a different color map, to see the other data you turn on in the view. The manual uses System>Grayscale.

8. In the *Inventory,* with your cursor on log curve GR, **MB3 > Display Properties.**
   - In the *Log Curve Occurrences* panel, select **Full** in the Orientation column for the Wellbore Track line.
   - Click the button associated with Line: and select **Line Width 2**. Click **OK**.

- On the Mode: pull-down menu, select **User Defined.**
- In the Left: field, enter **"0"** and in the Right: field, enter **"180"**. Click **OK**.

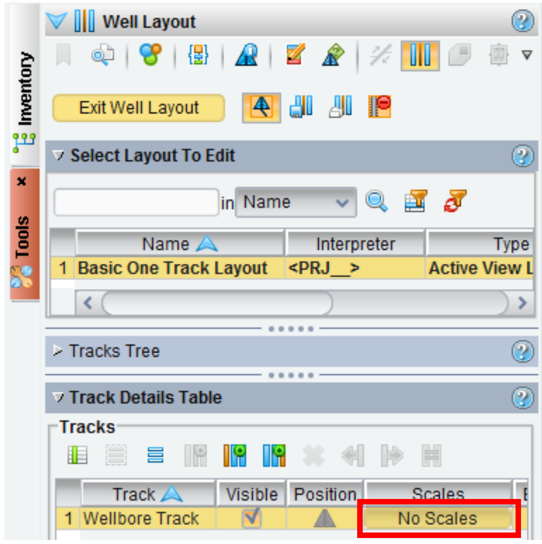


9. In the *Tools* task pane of the *Well Correlation* (THD) view, click the *Tools* task pane. Click the **Well Layout** ( ) icon.



10. In the *Well Details Table* of the *Well Layout* panel, click the **No Scale** button associated with Wellbore Track to add a scale and

lattice to the well layout. This will automatically put you in Edit Well Layout mode and Single Well Layout mode.



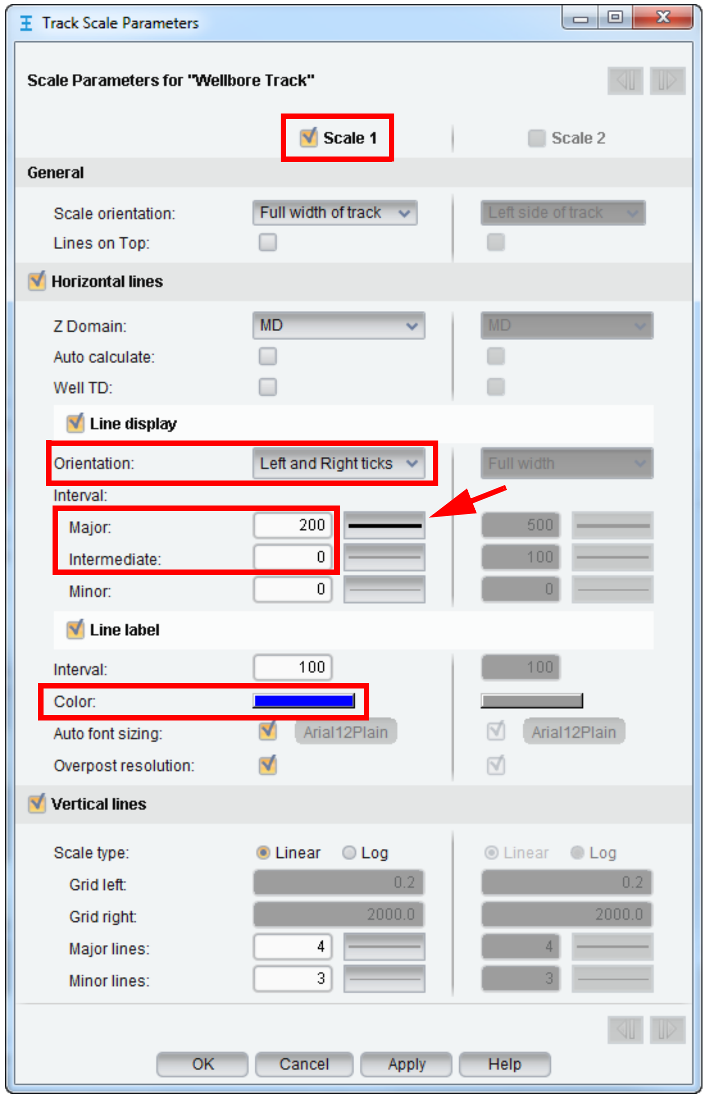
11. In the *Track Scale Parameters* dialog, toggle on **Scale 1**, and change the following:

   - On the Line Display Orientation: pull-down menu, select **Left and Right ticks**.

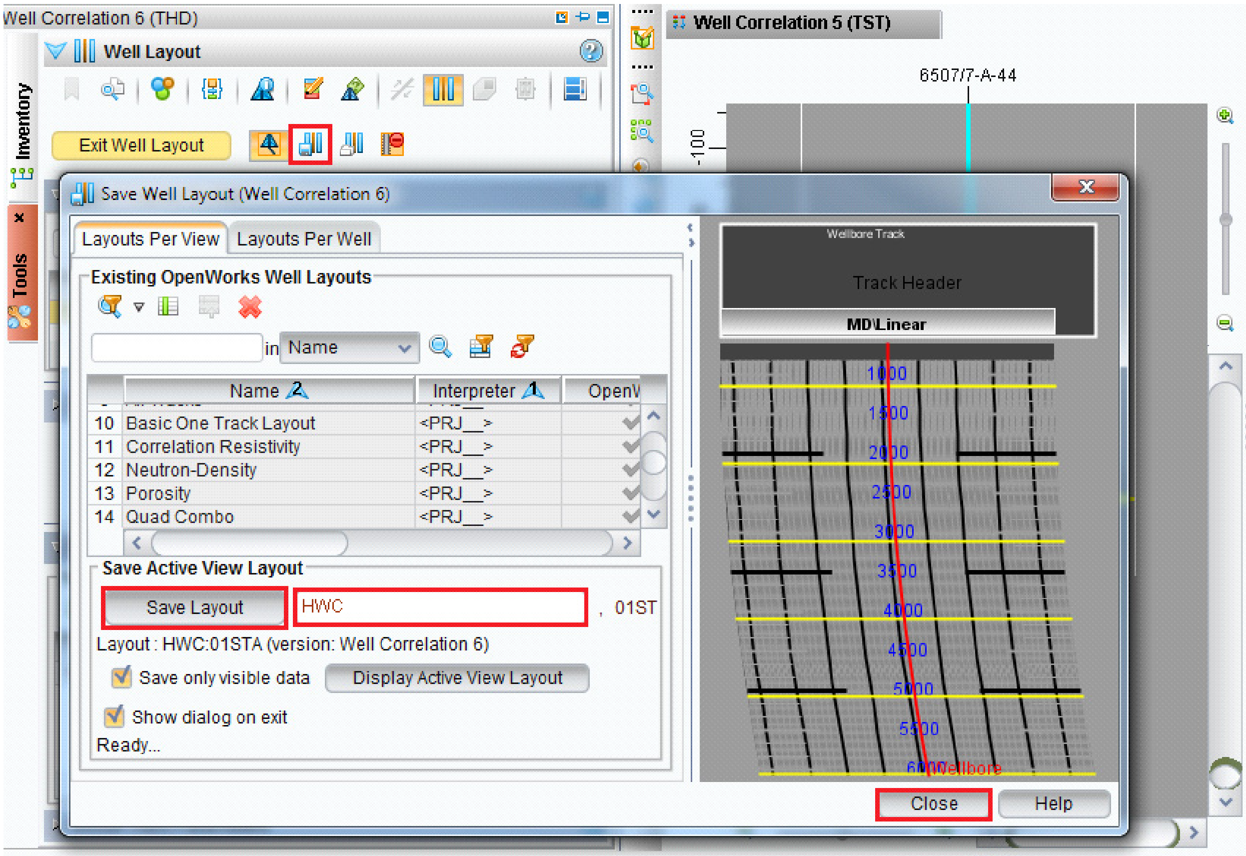
   - In the Interval Major: field enter "**200**", in the Interval Intermediate: field enter **"0".** Set line color to **Black.**

   - In the Line label section, set the color to **Blue**.

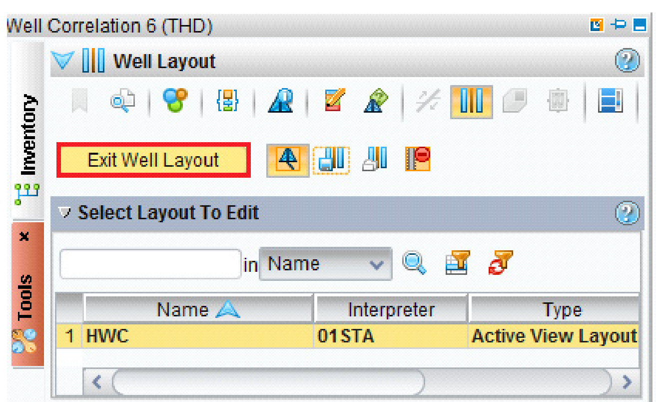
- Click **OK**.


Track Scale Parameters
Scale Parameters for "Wellbore Track"
Scale 1
Scale 2
General
Scale orientation:
Full width of track
Left side of track
Lines on Top:
Horizontal lines
Z Domain:
MD
MD
Auto calculate:
Well TD:
Line display
Orientation:
Left and Right ticks
Full width
Interval:
Major:
200
500
Intermediate:
0
100
Minor:
0
0
Line label
Interval:
100
100
Color:
Auto font sizing:
Arial12Plain
Arial12Plain
Overpost resolution:
Vertical lines
Scale type:
Linear
Log
Linear
Log
Grid left:
0.2
0.2
Grid right:
2000.0
2000.0
Major lines:
4
4
Minor lines:
3
3
OK
Cancel
Apply
Help

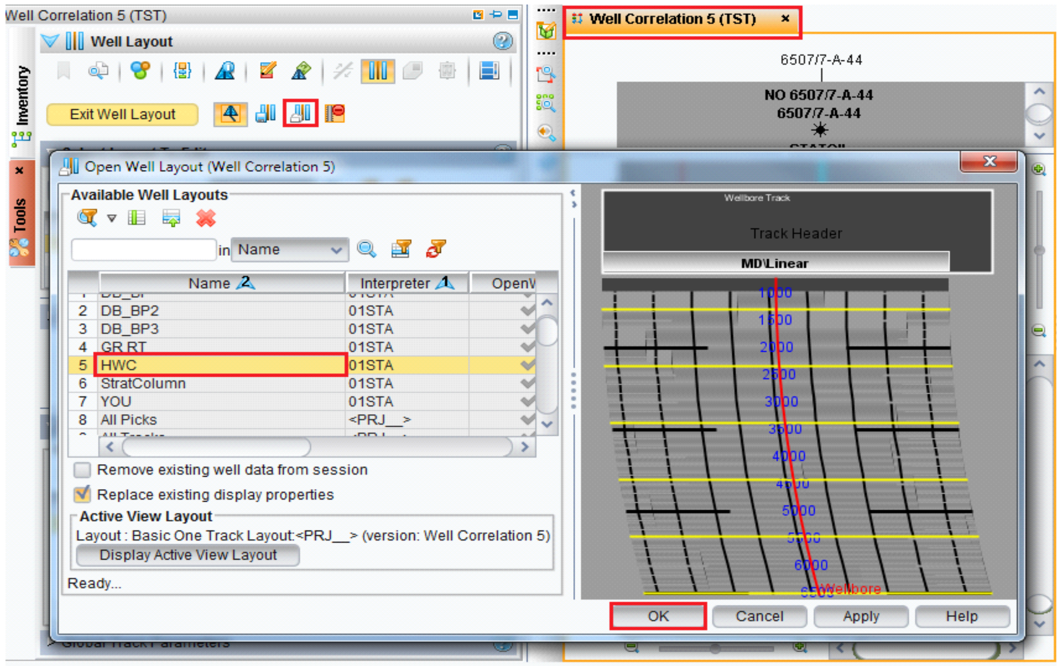
12. In *Well Layout* task pane, click the **Save Well Layout** ( ) icon. Enter **“HWC”** in the Save Layout field and click the **Save Layout** button. Click **Close.**



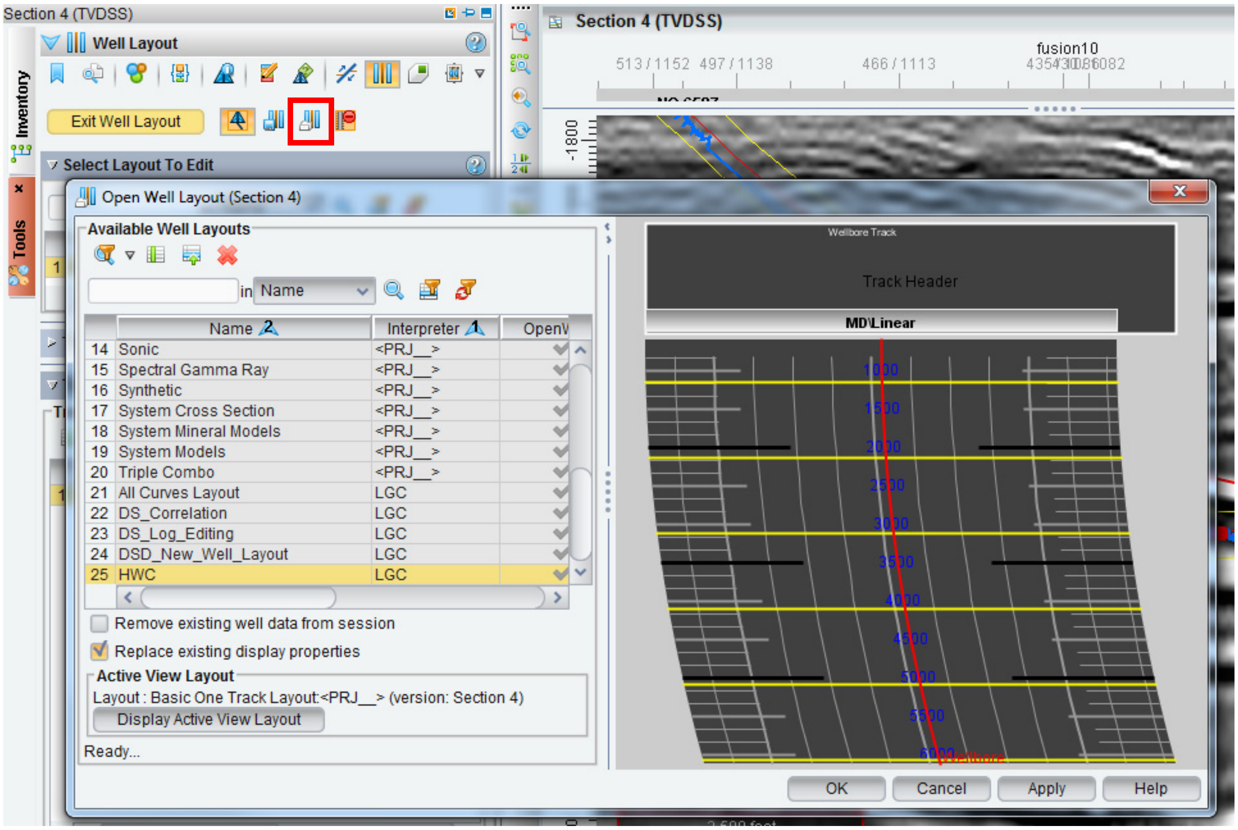
13. In the *Well Layout* task pane, click **Exit Well Layout.**



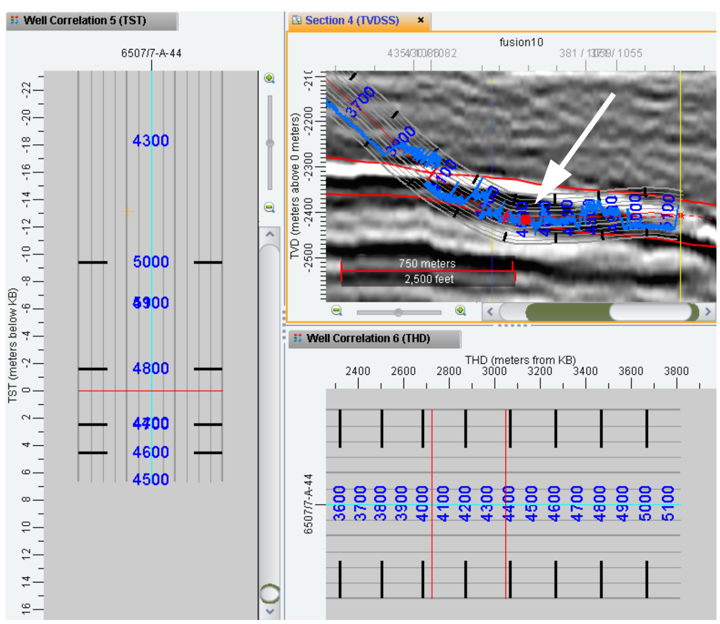
14. In the *Well Layout* task pane of the *Vertical Well Correlation* view, click the **Open Well Layout** ( ) icon and select the saved well layout **HWC.** Click **OK**. Click **Exit Well Layout.**



15. In the *Well Layout* task pane of the *Horizontal Well Correlation* view, click the **Open Well Layout** icon, and in the *Open Well* dialog, select **HWC**. Click **OK**. Click **Exit Well Layout**.

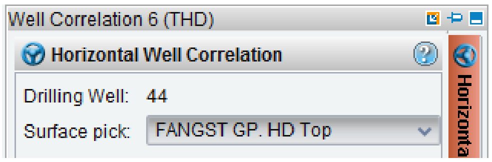


The Maximum Stratigraphic Distance (MSD) for the well is indicated by a solid red circle in *Section* view.

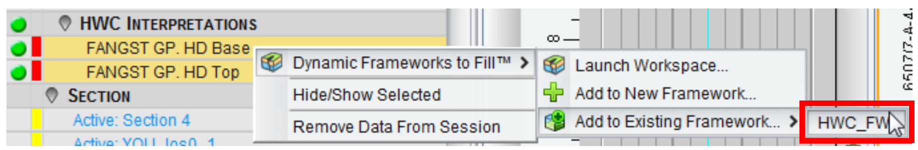


Exercise C.3: Create a Predicted Curve from an Offset Well

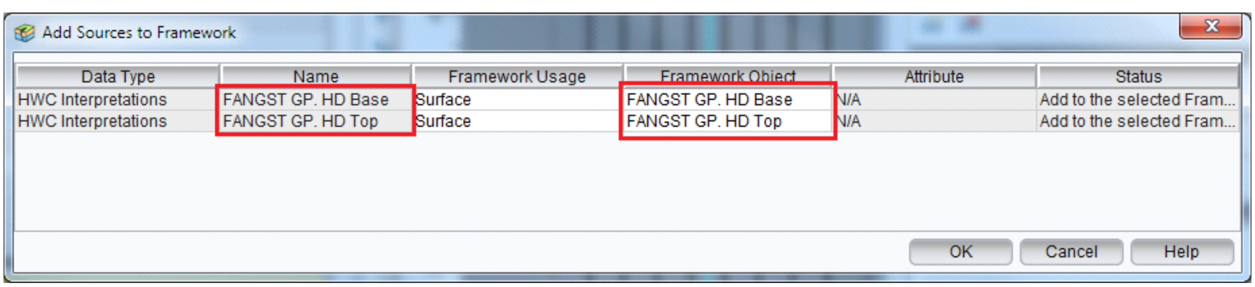
1. In *Horizontal Well Correlation* task pane, with the *Horizontal Well Correlation* view active, select **FANGST GP. HD Top** on the Surface pick: pull-down menu.



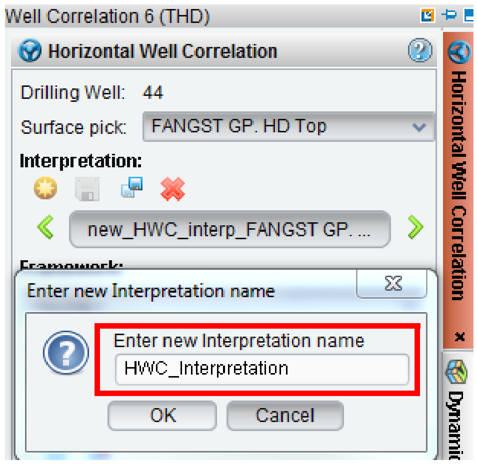
2. On the *Inventory* task pane, select the HWC Interpretations **FANGST GP. HD Top** and **FANGST GP. HD Base,** then **MB3 > Dynamic Framework to Fill > Add to Existing Framework > HWC_FW.**



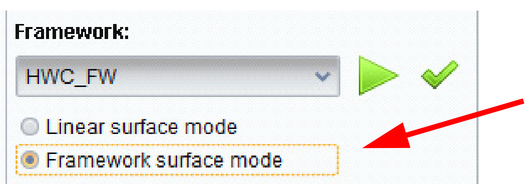
3. In the *Add Sources to Framework* dialog set the Framework Object column to emulate the Name column, as shown below. Click **OK**. This will associate the HWC Interpretation with the main FW Surfaces. Therefore, any change you make to the MWC Interpretations will be reflected in the surfaces.



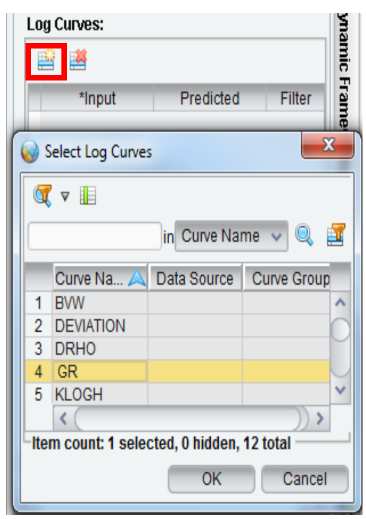
4. In the *Horizontal Well Correlation* task pane, click the **Create New Interpretation** ( ) icon. The *Enter new Interpretation name* dialog opens. In the Enter new Interpretation name field, enter "**HWC_Interpretation**" and click **OK.**



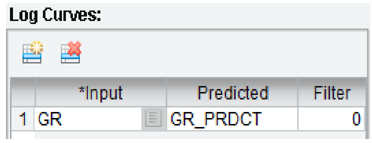
5. In the Framework: section of the *Horizontal Well Correlation* task pane, select **HWC_FW** from the pull-down menu and toggle on **Framework surface mode.**



6. In the Log Curves: section, click the **Add new Log Curve Row** **(** **)** icon. The *Select Log Curve* dialog opens. Select Curve Name: **GR** and click **OK.**



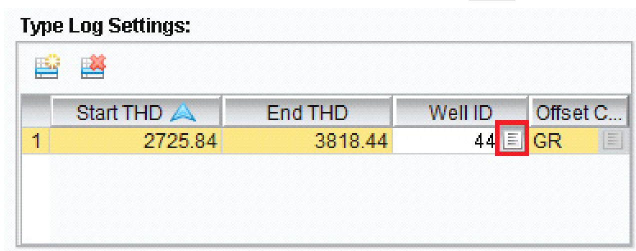
A Predicted curve has now been created with the default name GR_PRDCT.



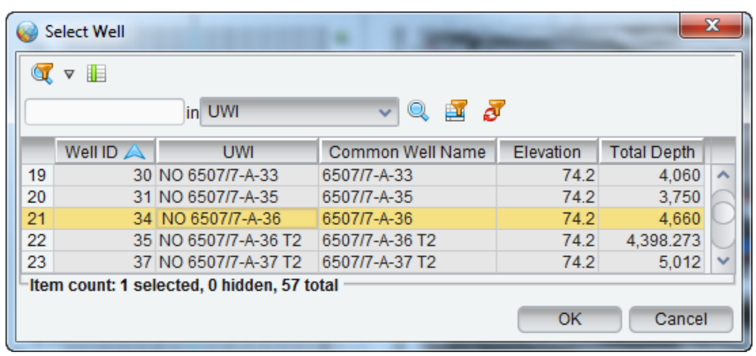
Note:

Entering an integer greater than 0 in the Filter column will create a filtered input curve (GR) filtered over 10 samples.

7. In the *Type Log Settings* panel, click the **well ID** box (to activate it) and click the **Select well offset** ( ) icon.



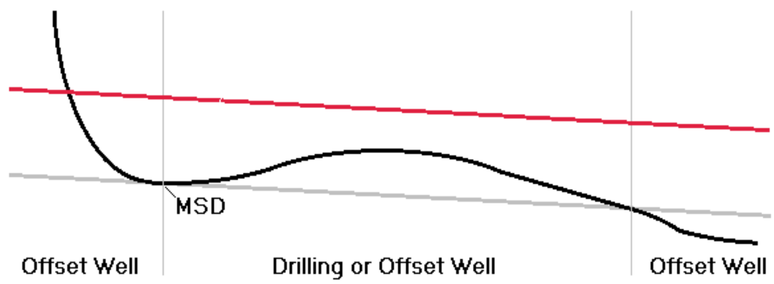
8. The *Select Well* dialog opens. Select the offset **well 6507/7-A-36**. Click **OK**.



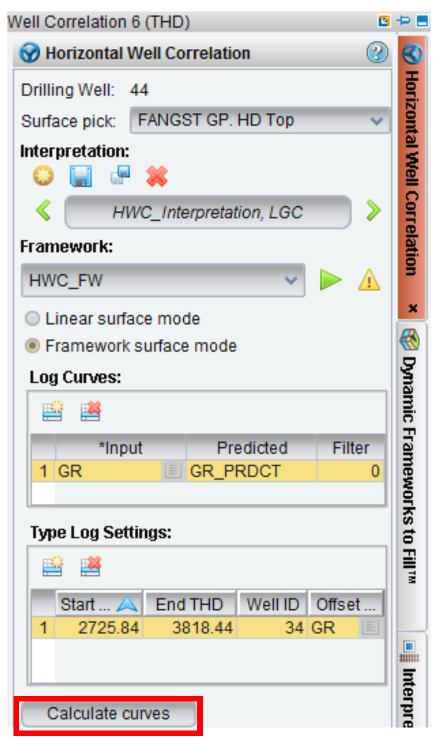
These settings will create the GR_PRDCT curve from the TST Type log of the Offset Well over the entire length of the horizontal well. The Start True Horizontal Distance (THD) is the THD value at the FANGST GP. HD Top surface pick.The End THD is the THD value at the total depth of the drilling well.

Note:

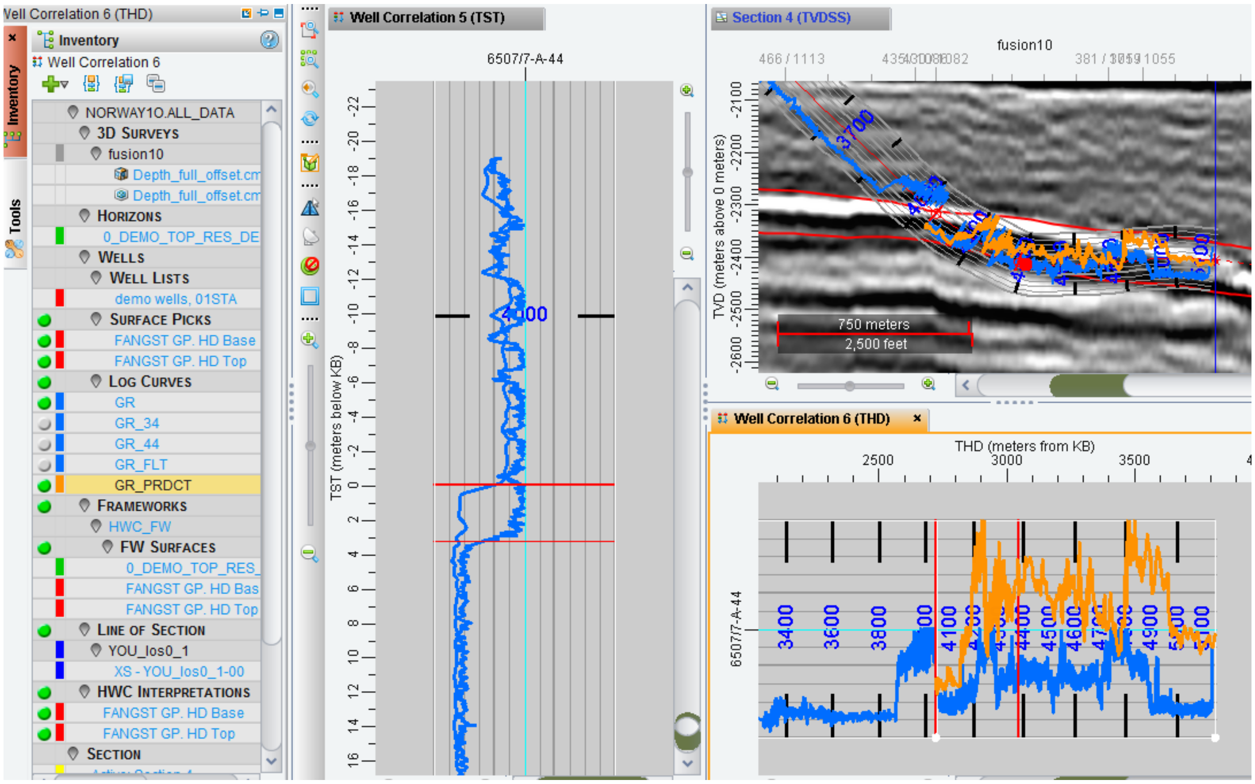
HWC correlation requires the use of an offset well (or offset wells) in the wellbore heel region and wellbore sections deeper than Maximum Stratigraphic Depth.



9. Click the **Calculate Curves** button to generate the predicted curve (GR_PRDCT).



You have now created a predicted curve from the input curve with the FANGST Top reference surface.



Exercise C.4: HWC Interpretation and Edit Options

The objective of horizontal well correlation is to achieve a match between the predicted and input curves. You can do it using one or all of the following actions:

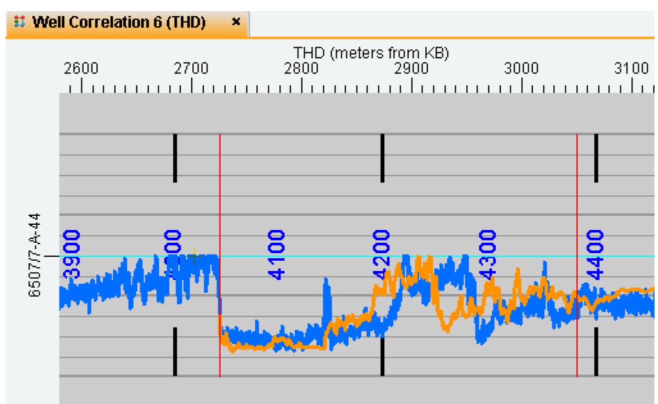
Horizontal Drag: Add an anchor to a predicted curve position and drag horizontally. MB1 to add an anchor at a correlation point on the predicted curve. Drag the anchor horizontally, so the predicted curve signature most closely matches the signature of the input log.

Vertical Drag: Add an anchor to an input curve position and drag vertically. MB1 to add an anchor at a correlation point on the input curve. <Shift> + MB1 to drag the anchor vertically, so the predicted curve signature most closely matches the signature of the input log.

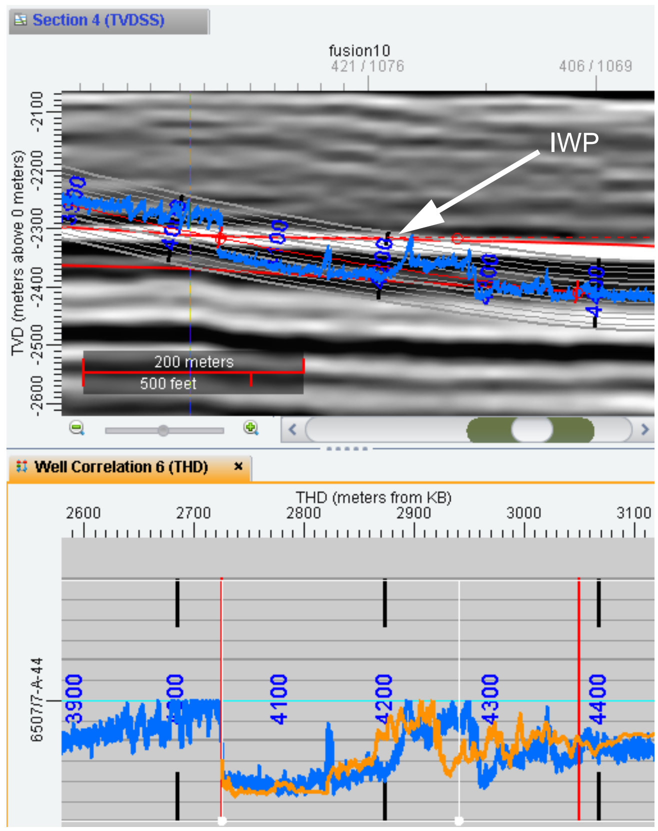
Add Anchors: Add anchors to lock down correlation positions. At a good correlation of the predicted and input curves, MB1 to add an anchor. These anchors are used to lock down the correlation and bound the stretch and squeeze portions of the predicted curve.

Remove Anchors: Remove an anchor and associated IWP by MB2.

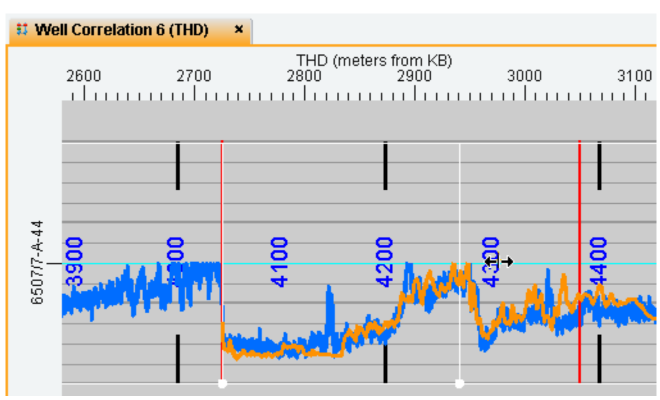
1. In the Horizontal *Well Correlation* view, zoom in to the **well** between THD 2600 m and 3100 m.

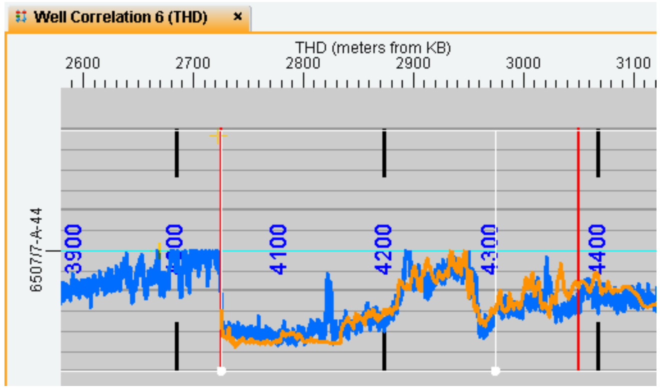


2. In the Horizontal *Well Correlation* view, click in the **Interpretation Mode** and **MB1** at approximately **2940 m** THD to add an anchor line at the trough of the blue GR_FLT curve. A corresponding IWP is created automatically on the framework surface (visible in the *Section* view). This will be used to correlate with the input curves.



3. In the Horizontal *Well Correlation* view, when you move your cursor over the Anchor line the cursor pointer will change to a right and left headed arrow. Click-and-drag the blue trough towards the **right** until it matches with the orange trough at **2970** m.

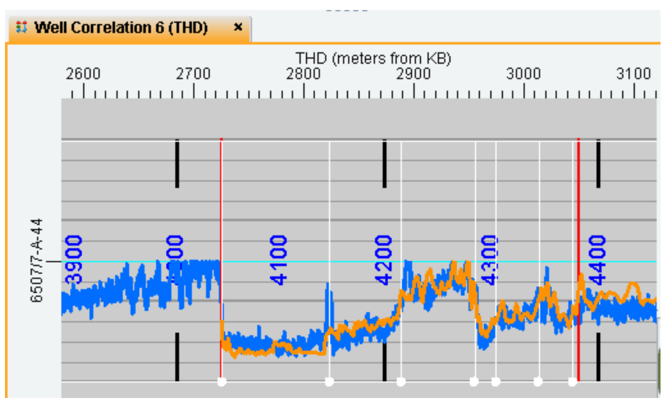




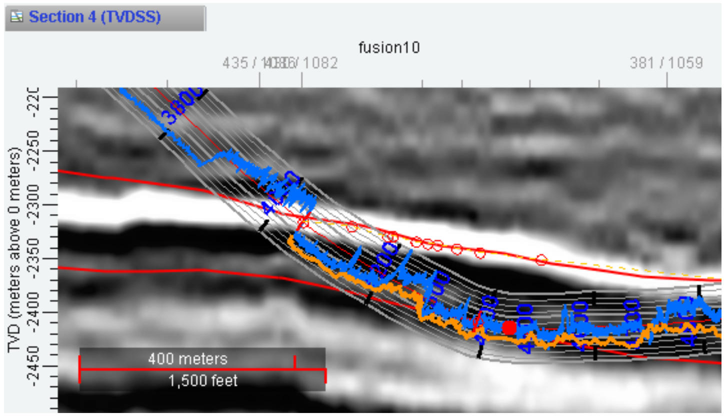
4. Create more **anchor lines** or **inter-well points** to correlate the curves along the well. On the section view, observe how the anchor points modify the reference surface (as dotted line).

The initial anchor line correlation will be iterative and require multiple minor adjustments. After the framework surface in the wellbore heel region is interpreted, the Type log of the drilling well stabilizes and anchor line correlations are non-iterative.

The correlation should look like the following image, wherein a near perfect match is achieved between the predicted and actual curve.



Because the Framework is set to automatic refresh, the framework surface is respecting the IWPs.

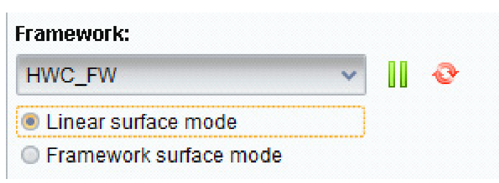


Anchor Line at the TD of the Drilling Well

The rightmost anchor line at the TD of the drilling well does not originally have an IWP. You can click to add an IWP at TD. The last anchor is tied to the wells TD and can be adjusted vertically, using <Shift> + MB1, but cannot be moved horizontally. Note the changes on the framework surface near the TD of the wellbore and compare with the original framework surface.

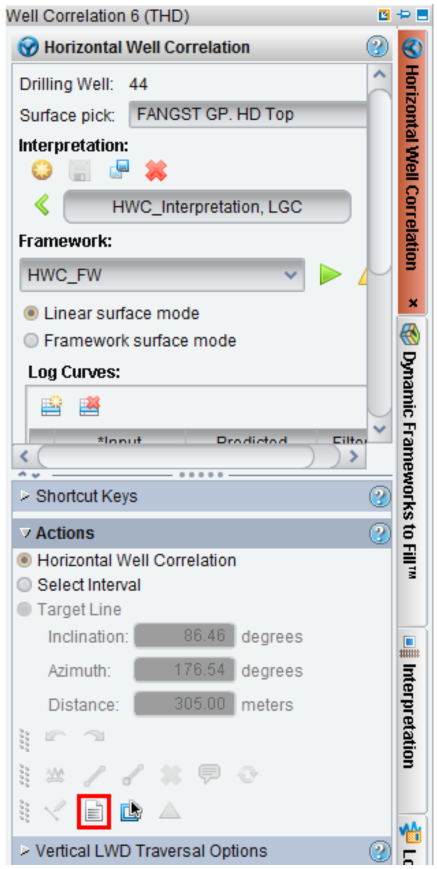
Linear Surface Mode

The Linear Surface Mode is a flat surface drawn between Surface Picks (observation 1 and greater inter-well).

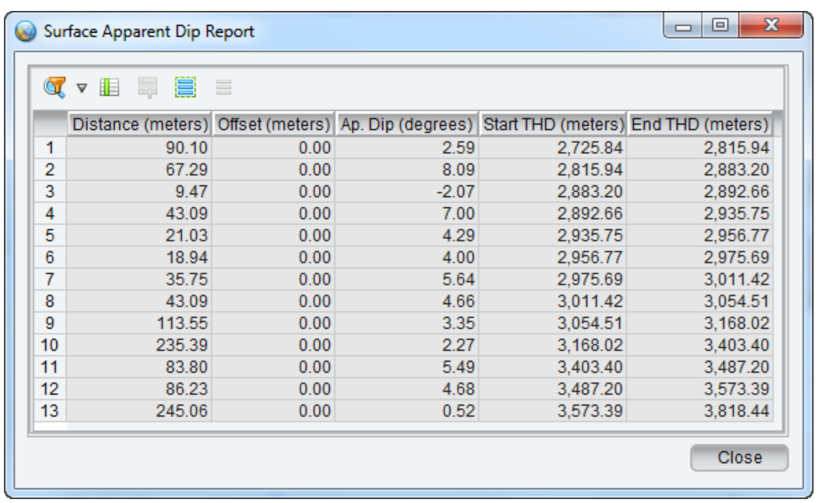


Toggling between linear surface and the predicted curves of Framework surfaces allows you to see differences between straight line and curved framework surfaces. This is an effective way to see where anchors need to be added or removed from the HWC interpretation. IWPs added in Linear Surface mode are added at the linear surface TVDSS location. IWPs added in Framework Surface mode are added on the framework surface TVDSS location.

1. In the Horizontal *Well Correlation* view navigate to the ***Horizontal Well Correlation*** task pane, expand the ***Actions*** panel. Click the **Surface Apparent Dip Report** ( ) icon.

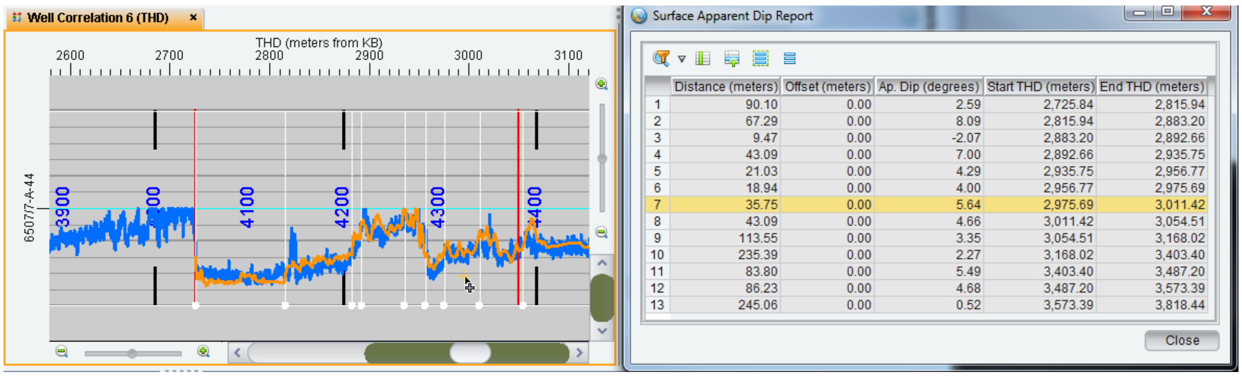


A *Surface Apparent Dip Report* of the interpreted framework surface will appear, displaying the apparent dip between IWPs.

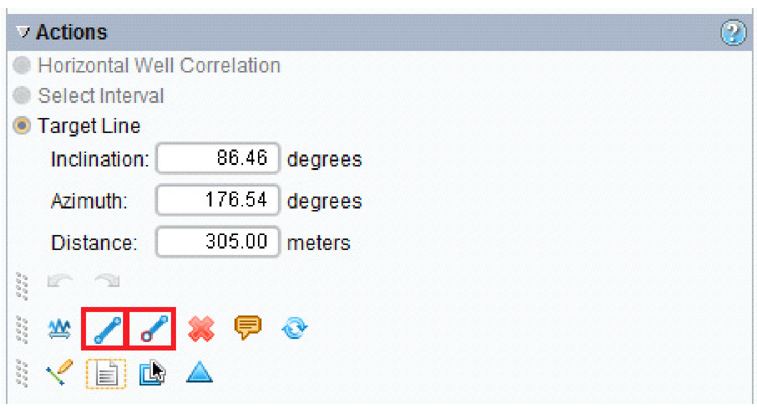


	Distance (meters)	Offset (meters)	Ap. Dip (degrees)	Start THD (meters)	End THD (meters)
1	90.10	0.00	2.59	2,725.84	2,815.94
2	67.29	0.00	8.09	2,815.94	2,883.20
3	9.47	0.00	-2.07	2,883.20	2,892.66
4	43.09	0.00	7.00	2,892.66	2,935.75
5	21.03	0.00	4.29	2,935.75	2,956.77
6	18.94	0.00	4.00	2,956.77	2,975.69
7	35.75	0.00	5.64	2,975.69	3,011.42
8	43.09	0.00	4.66	3,011.42	3,054.51
9	113.55	0.00	3.35	3,054.51	3,168.02
10	235.39	0.00	2.27	3,168.02	3,403.40
11	83.80	0.00	5.49	3,403.40	3,487.20
12	86.23	0.00	4.68	3,487.20	3,573.39
13	245.06	0.00	0.52	3,573.39	3,818.44

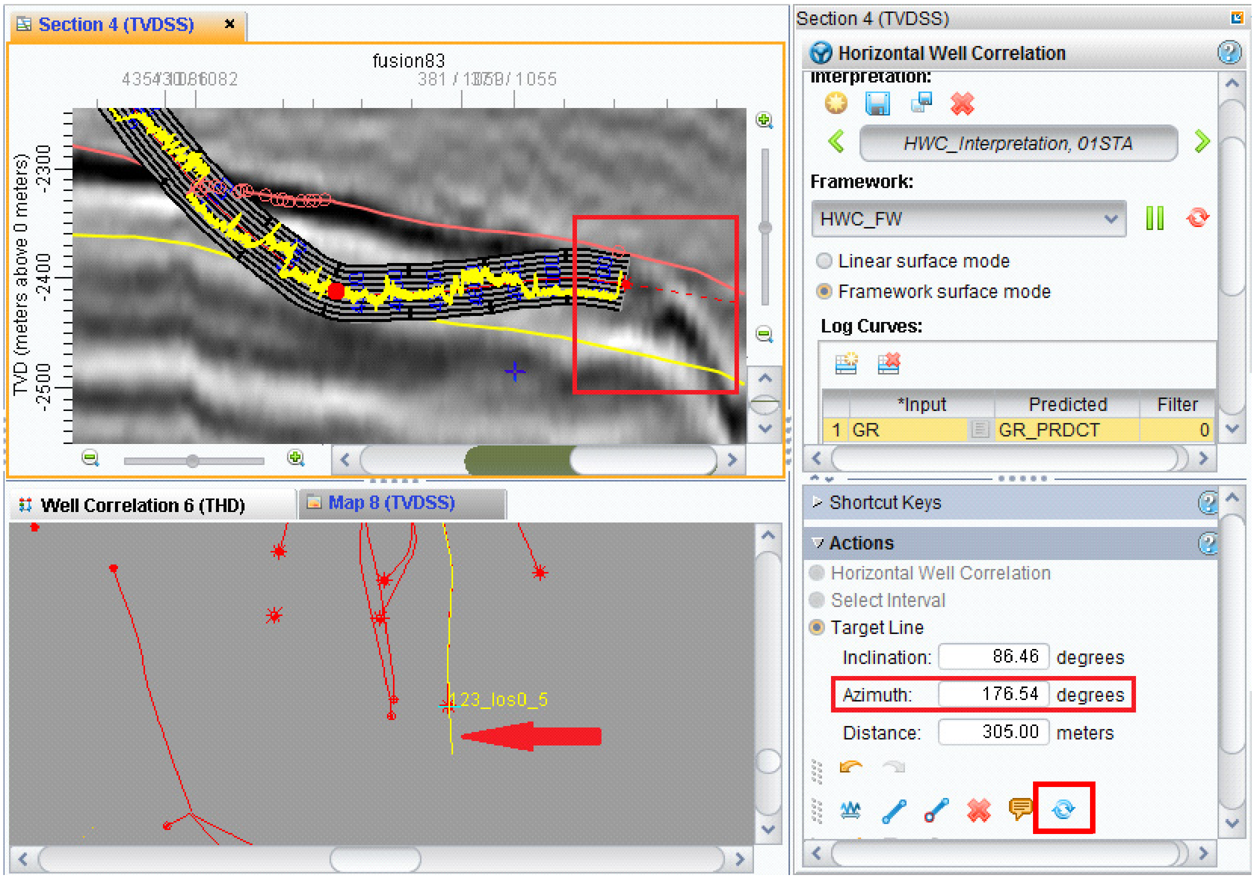
2. In the Horizontal *Well Correlation (THD)* view, put your cursor between two anchor points. The corresponding apparent dip for that section in the framework surface will be highlighted in the report. If you add a new IWP between any two existing IWPs, the surface apparent dip report will automatically update.



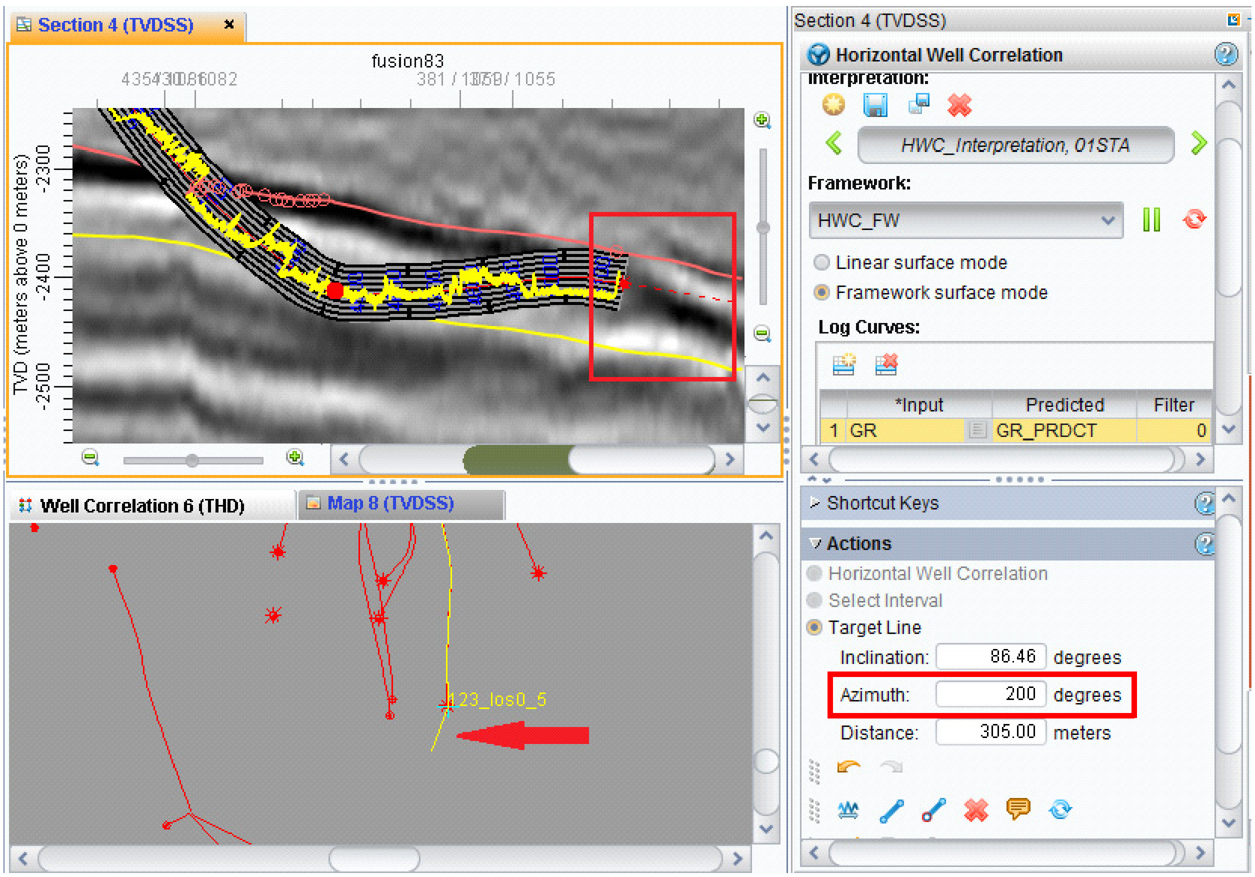
3. In the *Horizontal Well Correlation* task pane, with the *Section* view active, expand the *Actions* panel. The Target Line option enables you to draw wellbore target lines using two modes: TD target line or two points target line.



4. The displayed Azimuth is the azimuth of the well, derived from the bottom section of the wellbore.



5. On the *Actions* panel, enter “**200**” in the Azimuth field and press <**Enter**>. The azimuth of the extended line of section from wellbore TD is shown in the *Map* and HWC *Section* view.

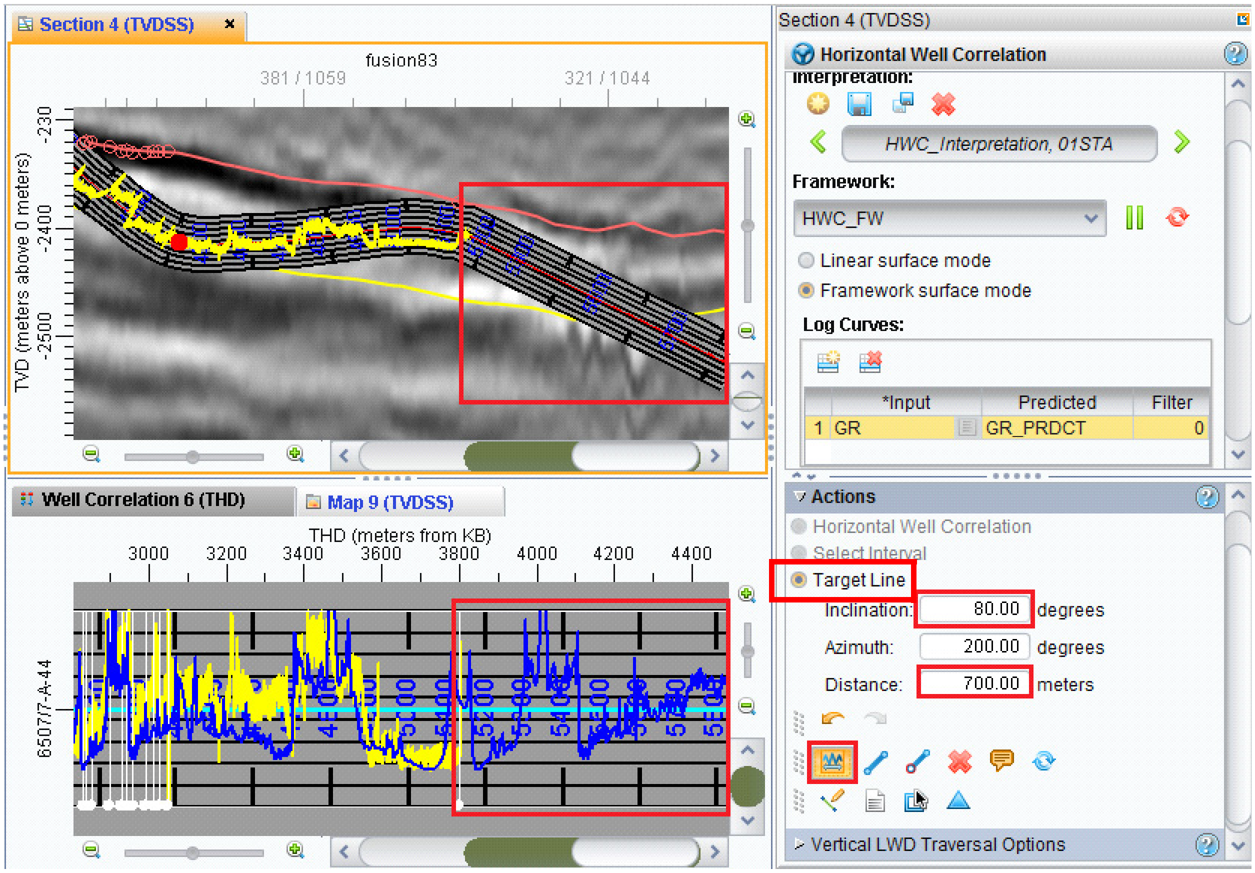


6. To revert to the actual azimuth of the wellbore, click the **Refresh** button.

The look-ahead distance is the THD distance displayed in *Section* view, downhole from the TD of the drilling well. The default value is 305 meters. A value entered in this box supersedes the default value. The *Section* and *Map* views are updated when you enter a value and press <Enter> or exit the text box.

7. In *Section* view, click the *Horizontal Well Correlation* task pane. In the *Actions* panel, make sure Target Line is toggled **on.** In the Inclination field, enter **“80”** and in the Distance field, enter **“700”**. Notice the display area of the section has been extended 700 m from the last survey point in the well. In the *Actions* panel, click the **Target Line Extrapolation** ( ) icon.

In the *Section* view and *Well Correlation* (THD) view, the Predicted curve is extrapolated 700 m ahead of the current depth along the 80 degree inclination.

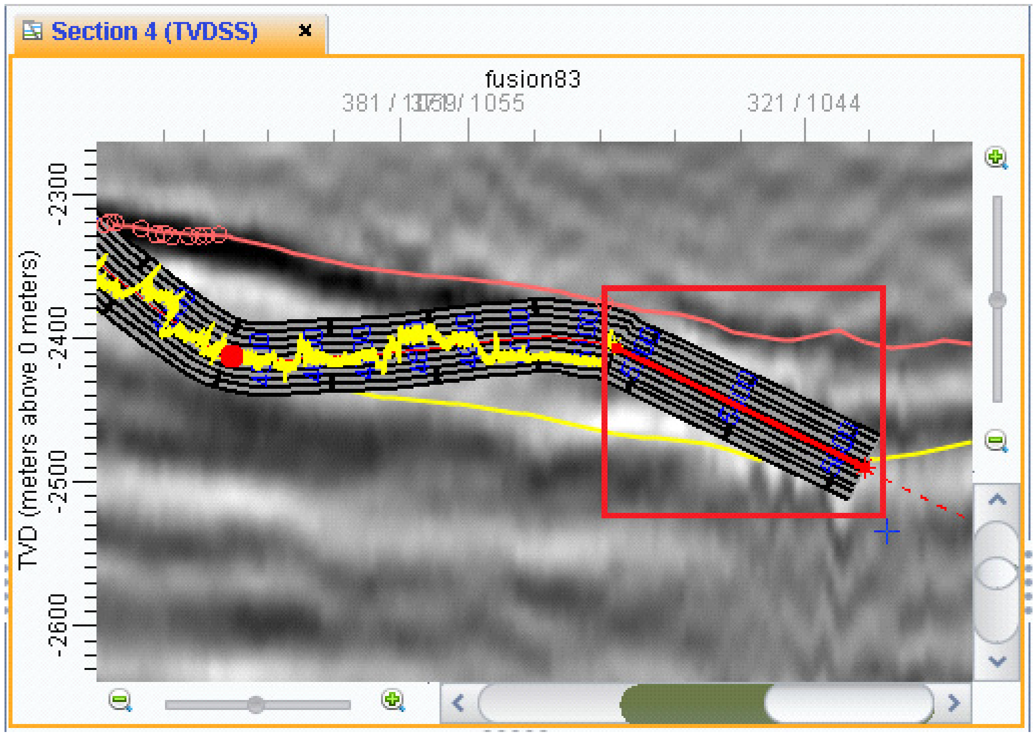


Note:

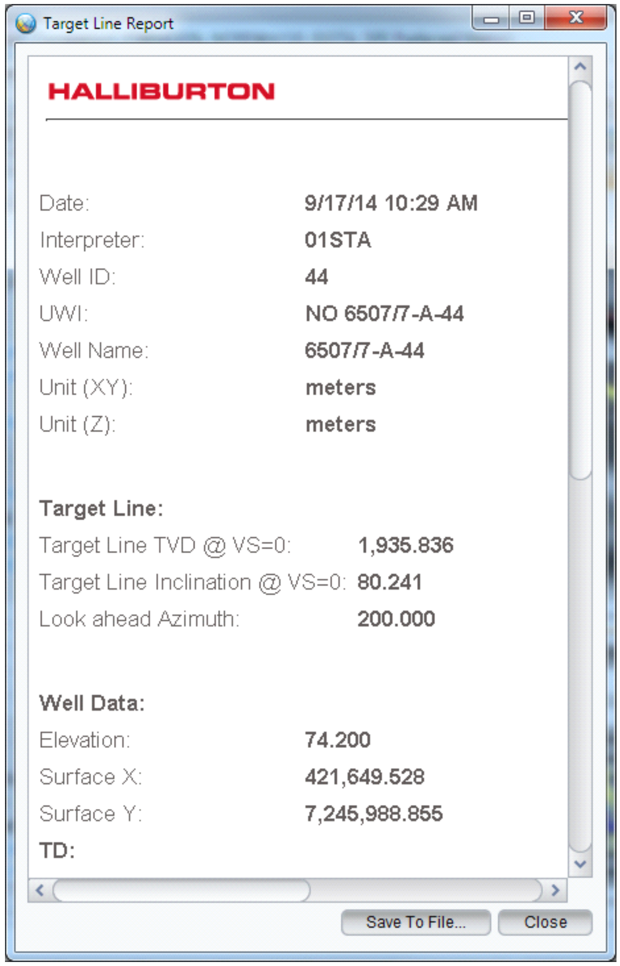
You might find that the extrapolated predicted curve is straight and does not show any signature. In that case, activate the *Well Correlation* (THD) view, and in the *Actions* menu of the *Horizontal Well Correlation* tab toggle on Framework surface instead of Linear surface. The software will extrapolate the predicted curve with reference to Framework surface (instead of linear surface).

8. Toggle off the **Target line extrapolation** () icon to remove the extrapolated curve along the specified inclination.

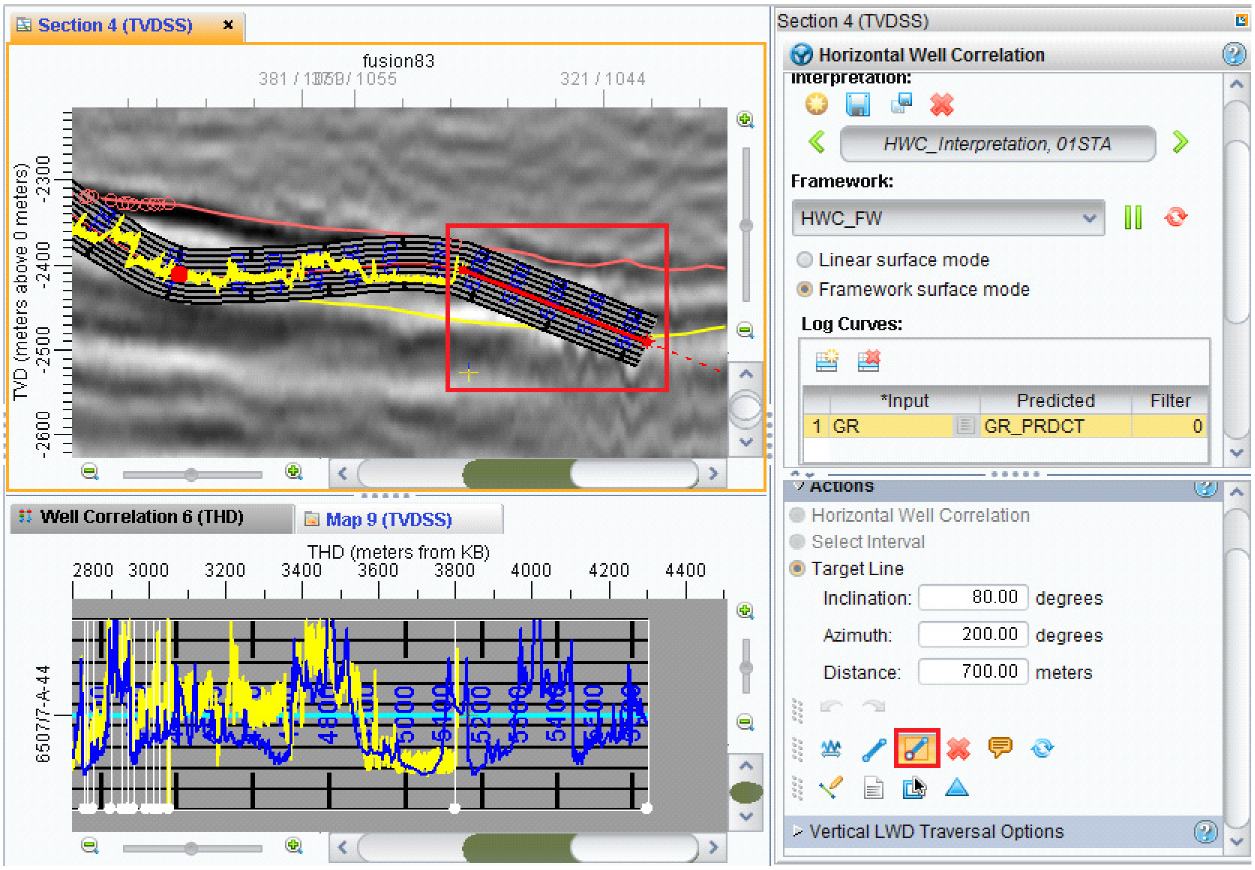
9. In the *Actions* panel, click the **Two Point Target Line** (icon) icon and draw a two-point target line in the *Section* view as shown below. Placing the cursor over either endpoint changes it to a four-way cursor and allows you to move the endpoint.



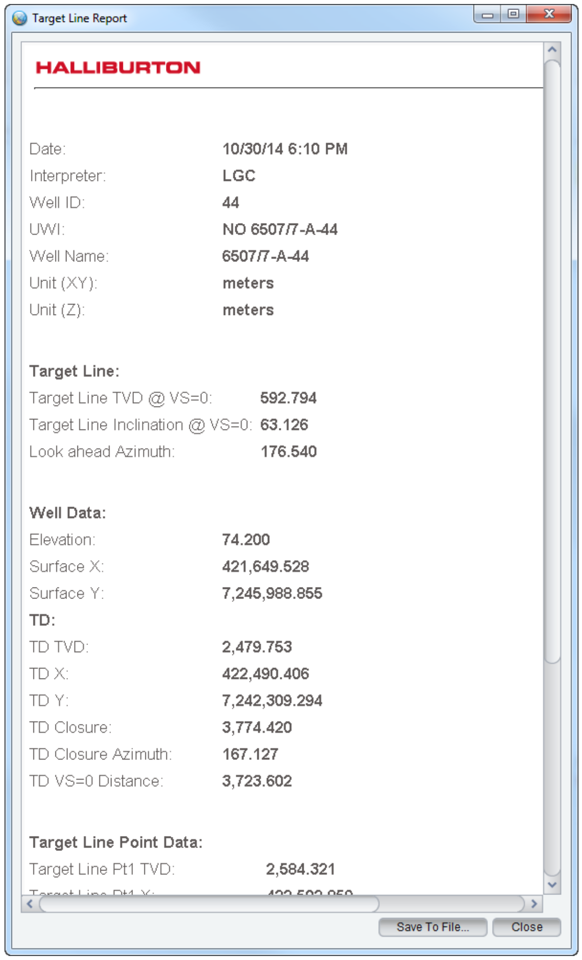
10. To view the report, click the **Target line report** ( ) icon. The *Target Line Report* calculates the Target Line TVD at VS = 0 and Target Line Inclination at VS = 0 values, and TD information. Click **Close**.



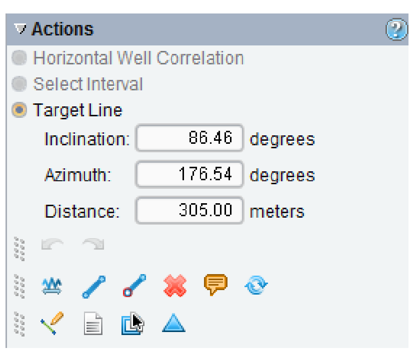
11. Click the **TD target line** ( ) icon to draw a target line starting at the TD of the well. When you release the mouse button, the predicted curve is extrapolated along the target line in any Azimuth on the section.



12. Click the **Target Line Report** ( ) icon to view the report. The *Target Line Report* calculates the Target Line TVD at VS = 0 and Target Line Inclination at VS = 0 values and the Target Line Point Data.



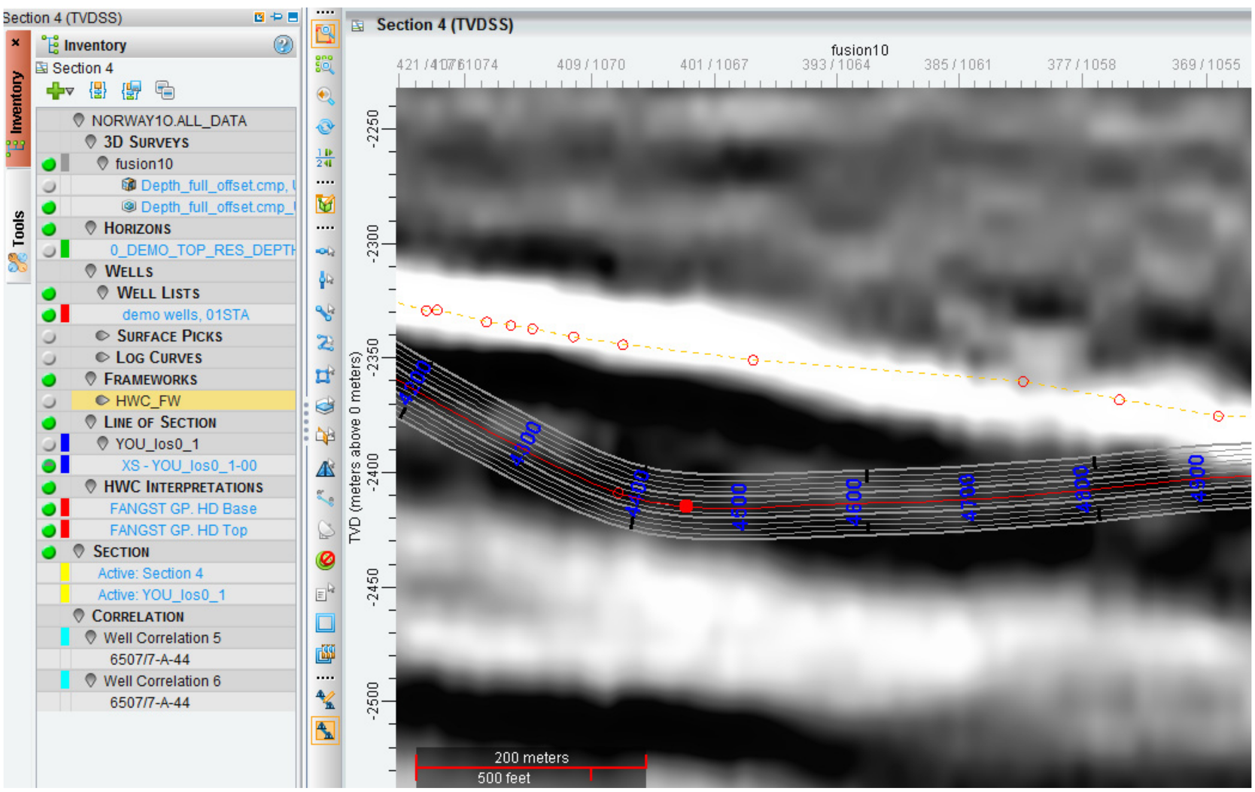
13. To clear the target line, click the **Clear and Delete Target Line** ( ) icon.



Exercise C.5: Displaying and Generating an Apparent Dip Calculation

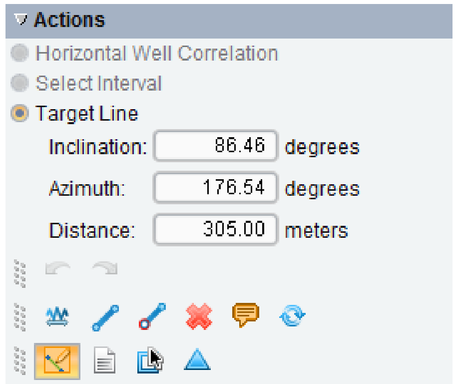
The *Apparent Dip* tool functions like the *Two Points Target Line* tool, in that there are no position restrictions below the well TD. The tool draws a green line anywhere in the *Section* view (within the LOS) and displays the apparent dip calculation.

1. In the *Inventory* task pane of the *Section* view toggle off the **Framework, log curves,** and **Surface Picks.** Only well 7-A-44 will then be displayed in your *Section* view**.**

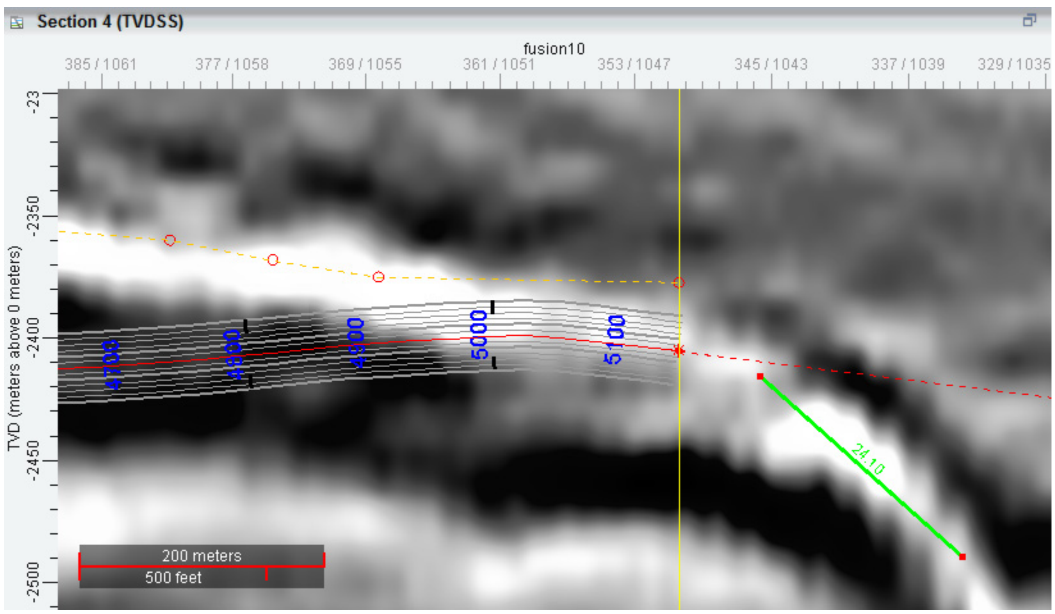


2. In the *Actions* panel of *Horizontal Well Correlation* task pane, click the **Apparent Dip Line (**  **)** icon.

> **Note:**
>
> You can also put your cursor anywhere in a *Section* view and MB3 > Apparent Dip Line.



3. Click the **position** where you want to begin your dip measurement, then click again where you want to end the measurement.



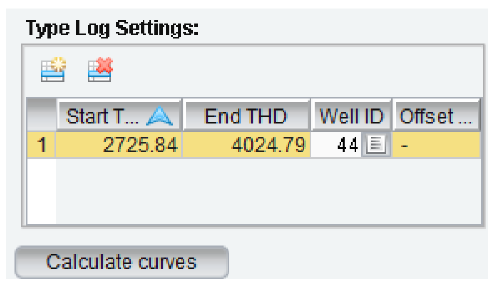
The dip angle will appear above the line you created. You can alter the line by clicking-and-dragging either **end node**, or by **<Shift> + click-and-drag** to move the entire line.

Optional Workflow 1: Create a Predicted Curve from a Drilling Well

When the drilled well has reached the MSD, you can switch the well to generate the predicted curves from the drilling well. At this point, the log from the vertical portion of the drilling well is the closest information available to generate a predicted curve and will probably be the most accurate to use, as long as the well continues its up-dip path.

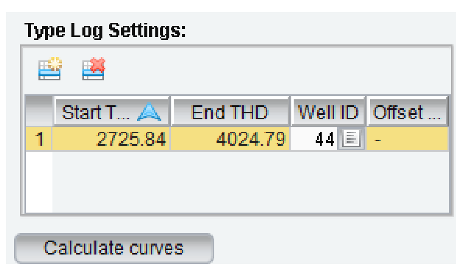
1. In the *Type Log Settings* panel of the *Horizontal Well Correlation* task pane, click the **Well ID** checkbox, then the **Select offset well** ( ) icon. In the *Select Well* dialog, select **6507/7-A-44**, then click **OK**.

2. Click Calculate **Curves**.



The full process will mimic that outlined in Exercise C.3. This method only varies by the well selected from the *Select Offset Well* dialog.

3. Click **Calculate Curves**.

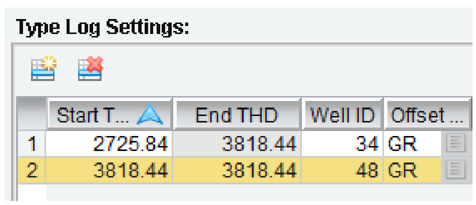


Optional Workflow 2: Create Predicted Curve from Multiple wells

During the correlation process, different parts of the drilling well may be adjacent to multiple offset wells. In such scenarios, you can make a composite predicted log curve by choosing depth ranges from several offset wells.

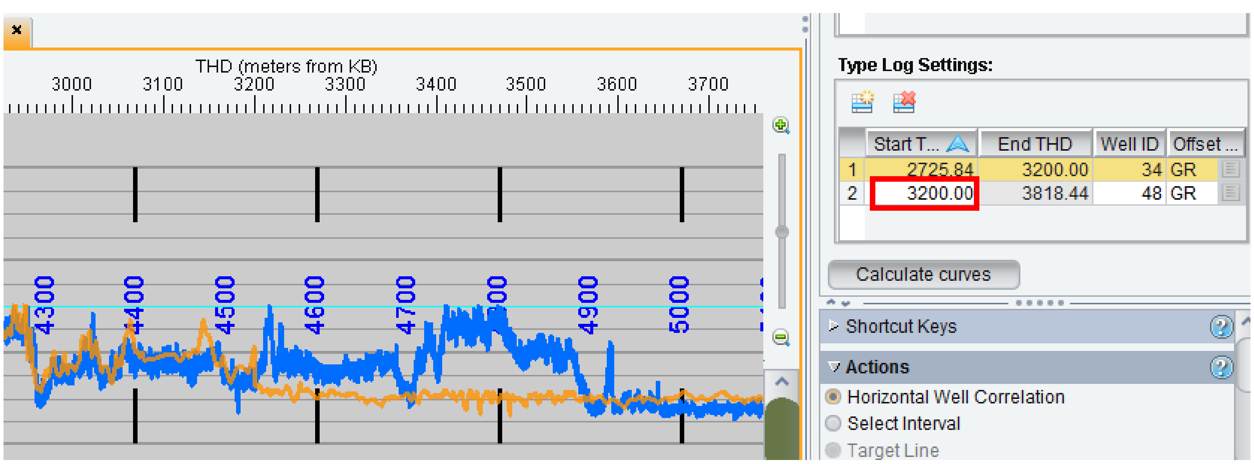
1. In the *Type Log Settings* panel of the *Horizontal Well Correlation* task pane, click the **Add New Type Log Settings** ( ) icon to insert a new row that will be used to define another well to use during correlation.

2. On the second row, click the Well ID **cell**, then **Select offset well** ( ) icon. In the *Select Well* dialog, select **6507/7-A-48**, then click **OK**.



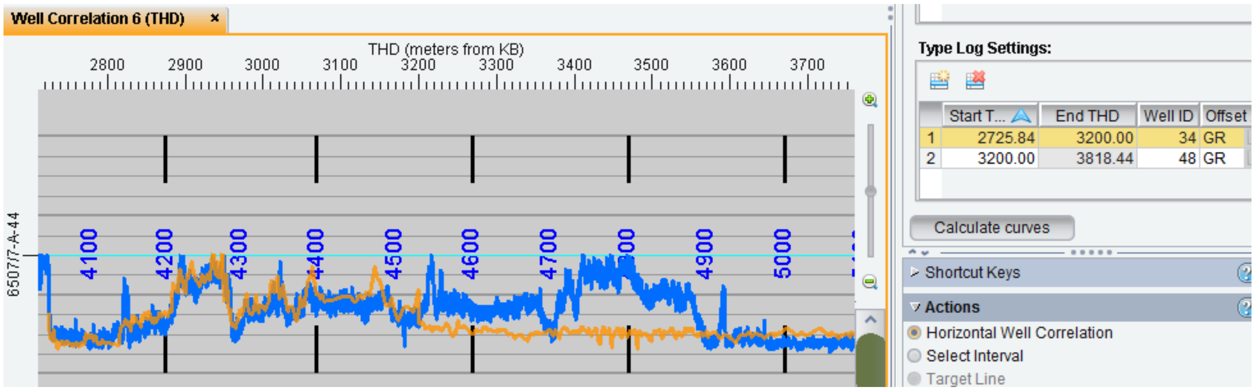
The box has a start THD and an end THD. This is the THD for which the well will create a predicted log. The horizontal *Well Correlation* tab is annotated on THD, so you get the values from this window.

3. In the second row of the Type Log Settings panel, enter **“3200”** in the Start THD column.



Note that the first row is automatically adjusted.

4. Click **Calculate Curves**.



Appendix D
Dipmeter Analysis

The Dipmeter Analysis tool provides a simple but powerful visual and statistical evaluation of wellbore dipmeter data in depth *Section* views. The tool includes:

- Visual display of the data orientations in a stereonet view
- Calculation of the best-fit fold axis and average plane orientations
- Estimate of the horizon geometry that honors the selected dip data

Dipmeter analysis can be applied in any depth *Section* view that has one or more wellbore directional surveys displayed with dipmeter data. Although the dipmeter data shown in a *Section* view represents two-dimensional projected apparent dips, Dipmeter Analysis uses the three-dimensional dip and dip azimuth values for its calculations.

How Dipmeter Analysis Works

When three or more dipmeter points have been selected in the *Section* view, the Show Stereonet option becomes active in the *Dipmeter Analysis* dialog. Toggling this option on creates the *Stereonet Analysis* reminder, which contains two tab panes: *Stereonet View* and *Statistics View*.

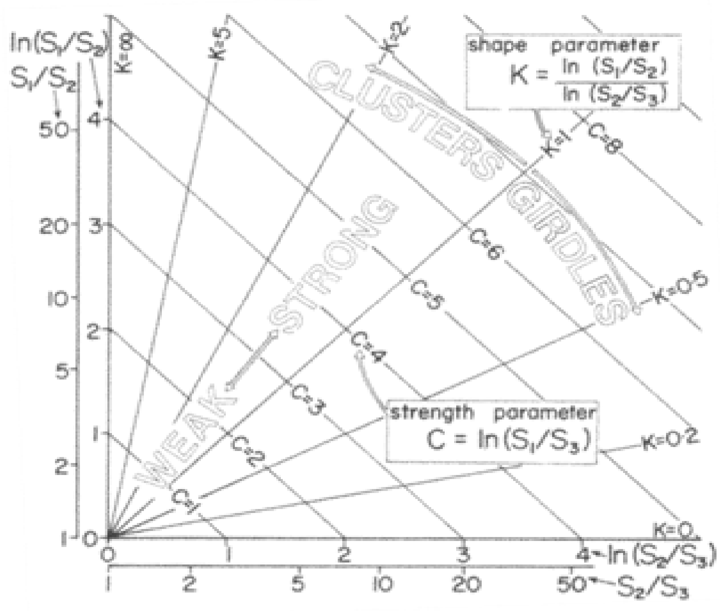
The *Statistics View* tab presents the results of an eigenanalysis of the orientation tensor for the selected data. The following are presented:

- The best-fit plane (average orientation of the dips)
- The best-fit fold axis
- The eigendata results (principal eigenvalues and their orientations)

The *Stereonet View* presents an equal-area, lower-hemisphere stereonet display of the poles to the dipmeter orientations, the best-fit plane great circle, and the eigenvectors. The eigenvectors include the fold axis (represented by a triangle, the minimum principal eigenvector) and the best-fit plane (represented by a square, the maximum principal eigenvector). A typical set of results can be evaluated by the following steps:

- Examine the Uniformity Statistic (Su) to confirm the data are non-random. (A 95% confidence level is appropriate for small sample sizes.) If the Su value is less than the test value, the data cannot be distinguished from a random set of dips.

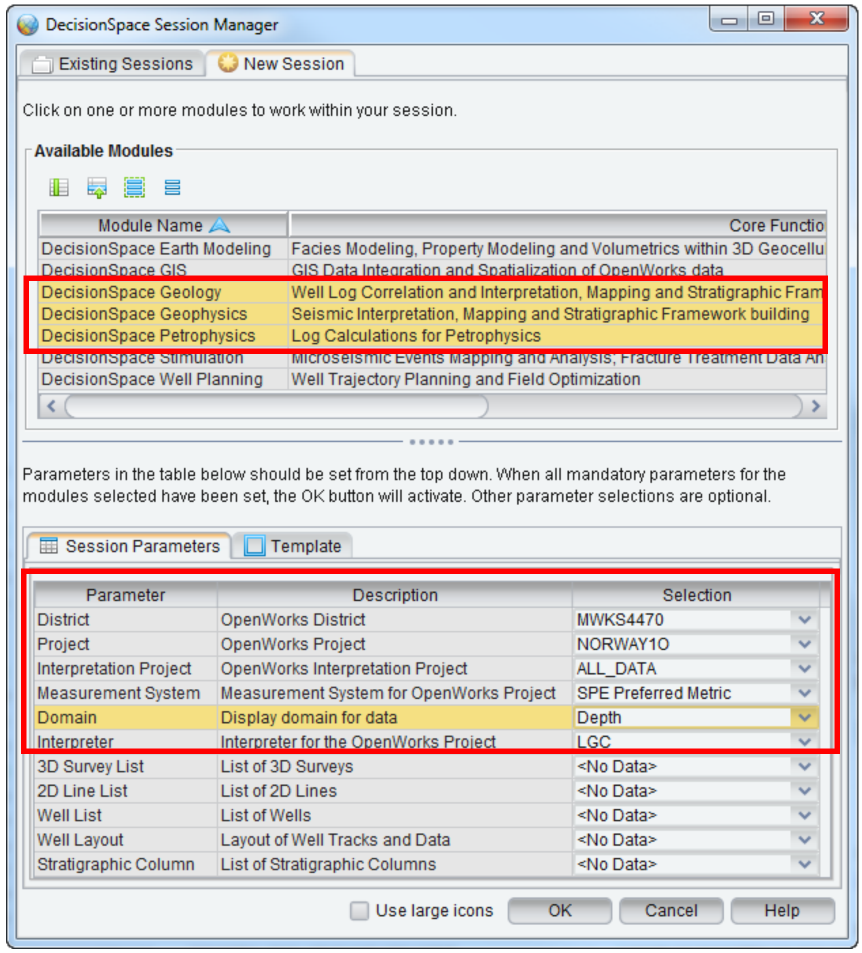
- If the data are non-random, their shape (K) and strength (C) parameters can be evaluated for degree of cluster or girdle preferred orientation. A girdle (K<1) may indicate sufficient sampling to adequately define the best-fit fold axis orientation, whereas a cluster (K>1) may suggest sampling limited to the limb of a fold — or higher confidence in the best-fit plane. The K and C values are simple ratios of the orientation tensor eigenvalues and can be compared to the graph below for a visual check on the degree of confidence.



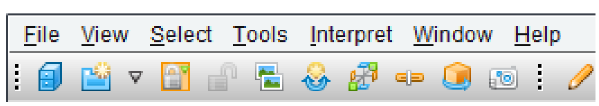
- Regardless of the orientation tensor results, the *Stereonet View* should always be inspected for evidence of multi-modal data (such as dips from different folds separated by a fault), which cannot be successfully characterized without further analysis.

Using the Dipmeter Analysis Tool

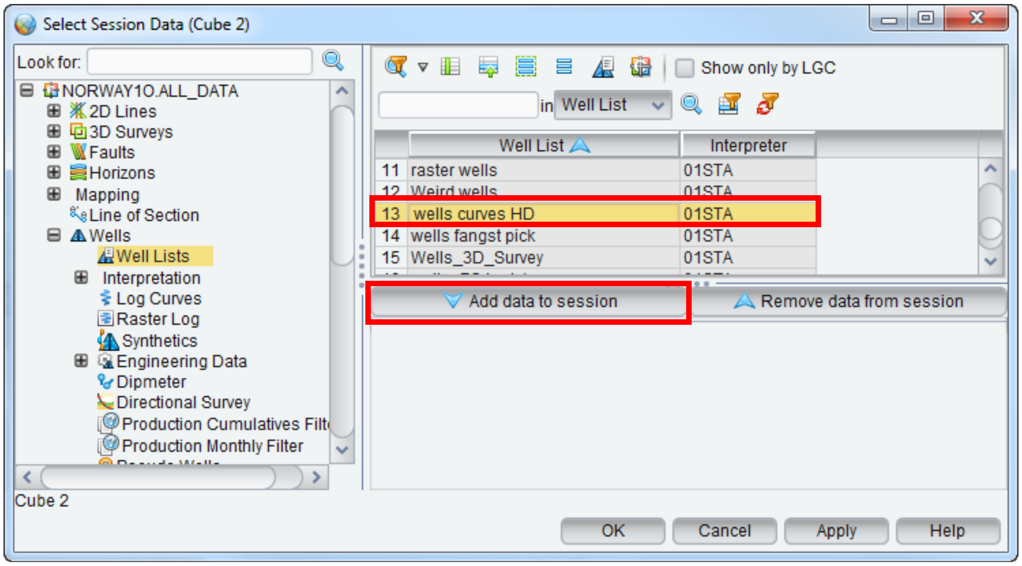
1. In the *DecisionSpace Session Manager*, select the following modules and session parameters, then click **OK**.



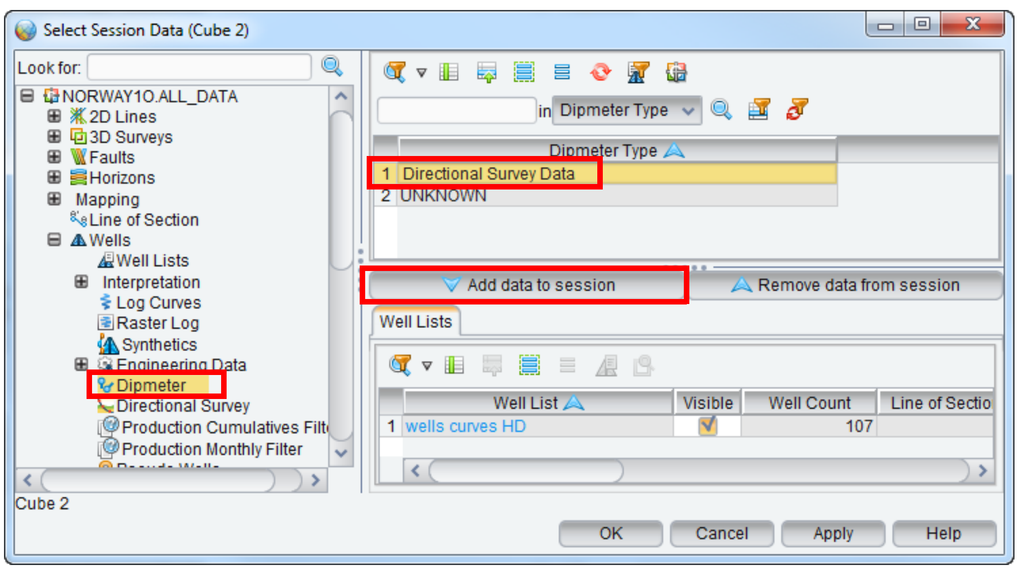
2. In the menu toolbar, click the **Select Session Data** ( ) icon.



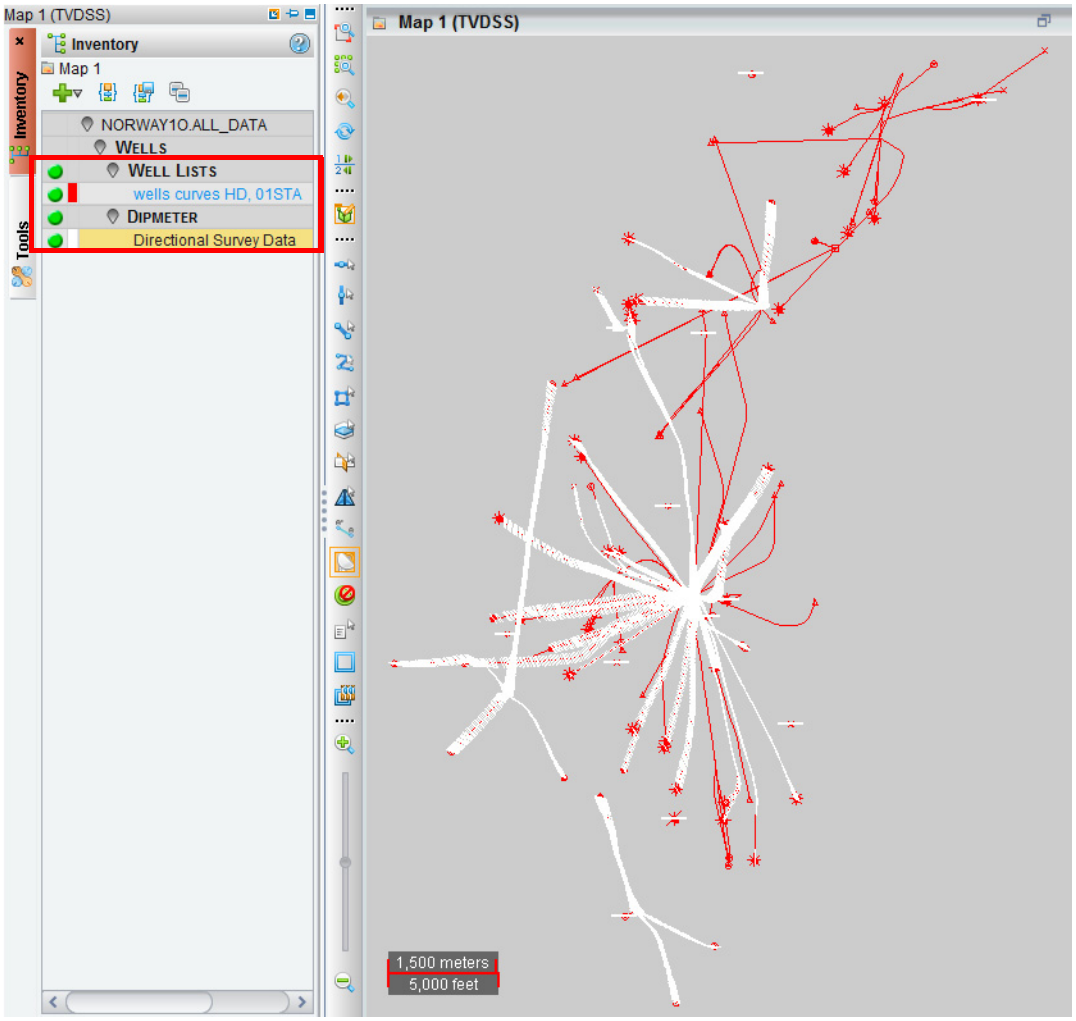
3. In the Well List column of the *Select Session Data* dialog, select **wells curves HD** and click the **Add data to session** button.



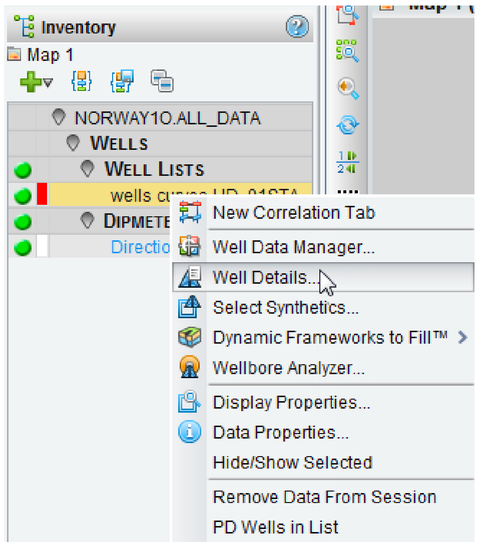
4. In the *Engineering Data* task pane of the *Select Session Data* dialog, select **Dipmeter.** Then, in the *Dipmeter Type* panel, select **Directional Survey Data**. Click the **Add data to session** button, then click **OK.**



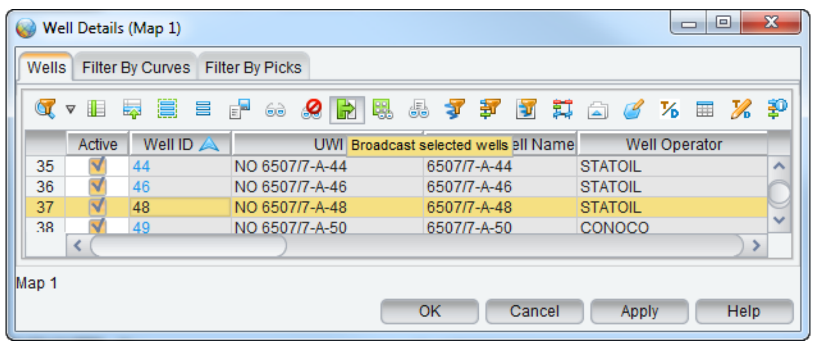
5. In the *Inventory* task pane of the *Map* view, toggle on Well Lists **wells curves HD** and Dipmeter **Directional Survey Data**.



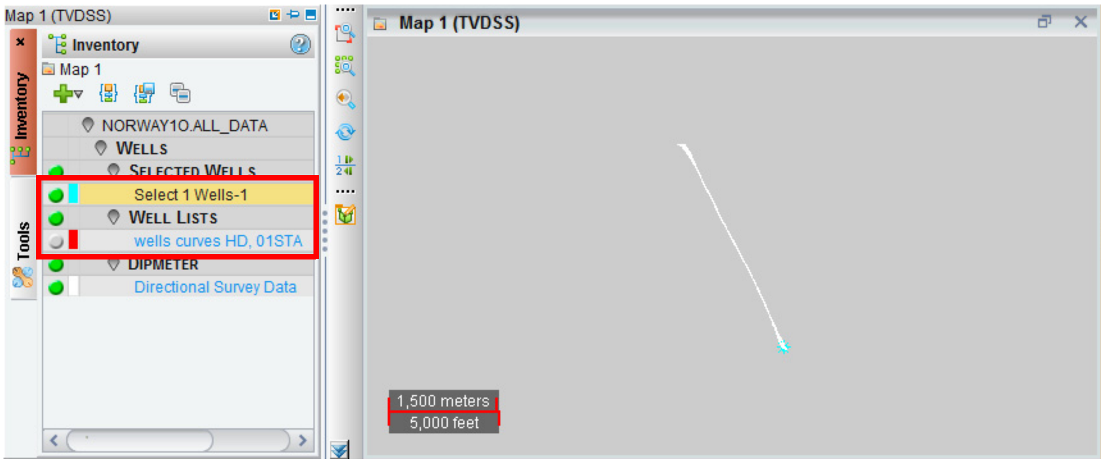
6. In the *Inventory* task pane, put your cursor on Well Lists **wells curves HD, 01STA** and **MB3 > Well Details** to open the *Well Details* dialog.



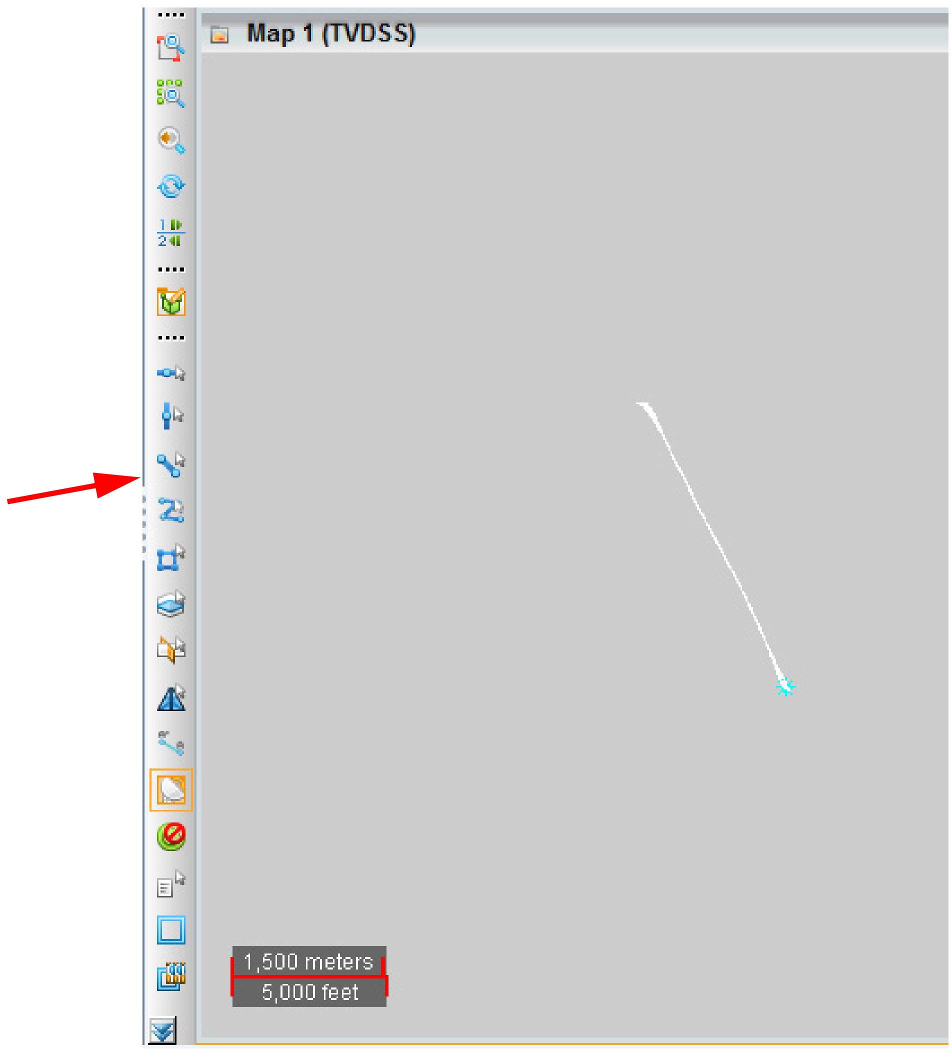
7. In the *Well Details* dialog, select **Well ID 48,** then click the **Broadcast selected wells** ( ) icon. Click **Cancel** to close the *Well Details* dialog.



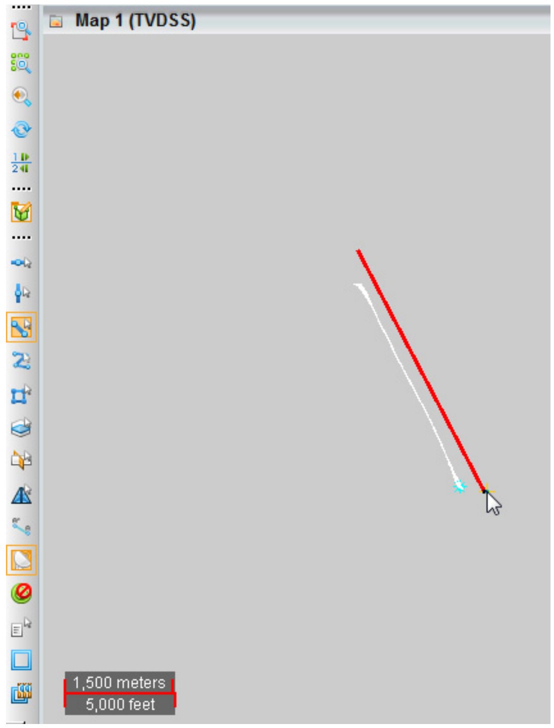
8. In the *Inventory* task pane, toggle off the Well Lists **wells curves HD, 01STA.** Toggle on **Select 1 Wells-1.**



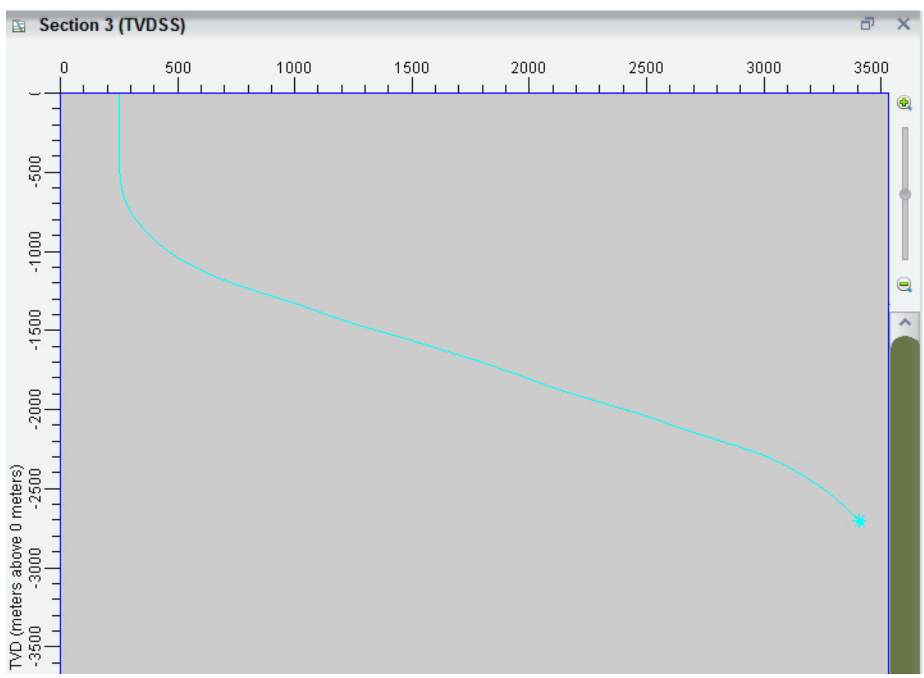
9. On the vertical tool bar, click the **Select Point to Point** () icon.



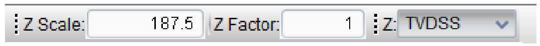
10. In *Map* view, **MB1** north of the well head, then **MB1** east of **TD,** and **MB2** to broadcast to *Section* view.



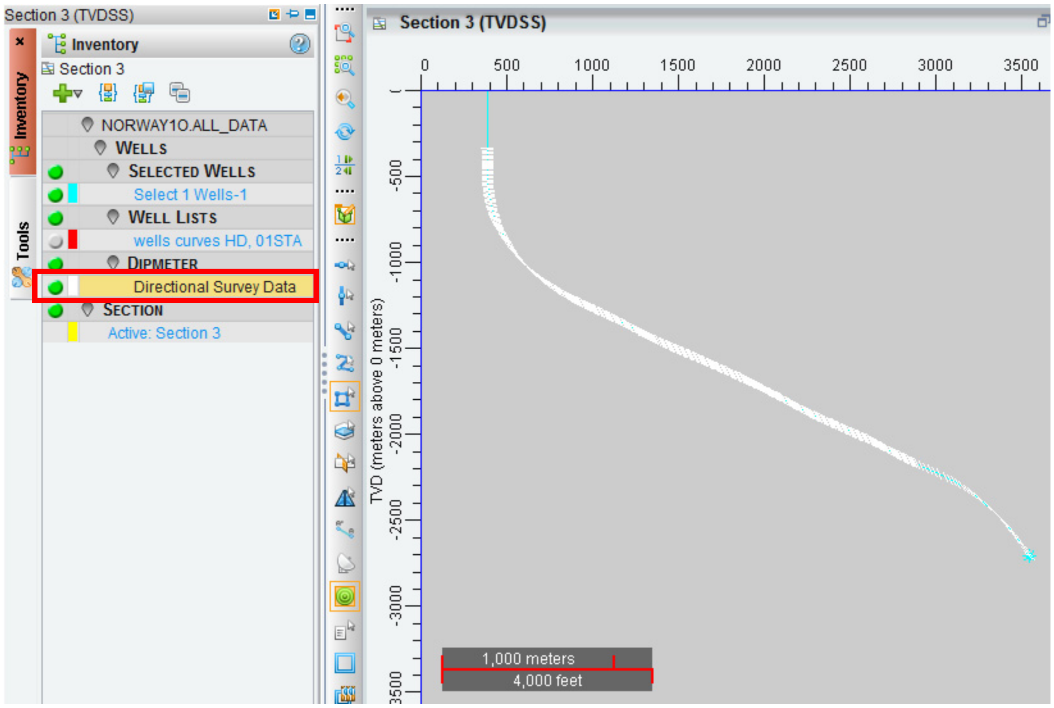
11. Double-click the *Section* view to maximize it.



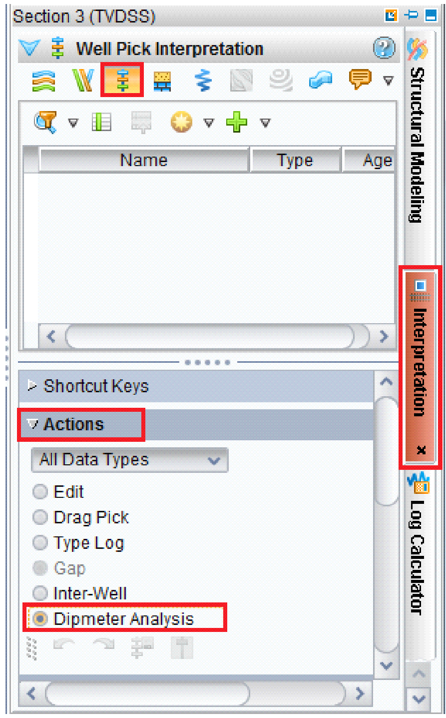
12. On the *DecisionSpace* dialog tool bar, set the Z Factor to **1**.



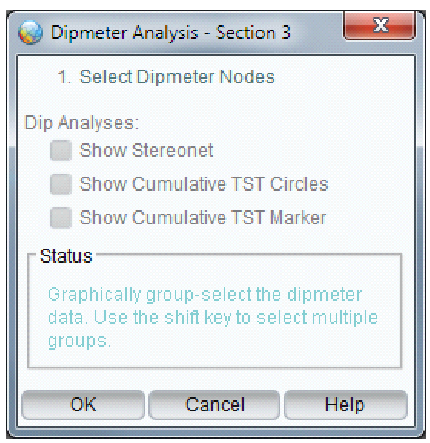
13. In the *Inventory* task pane of the *Section* view, toggle on **Directional Survey Data**.



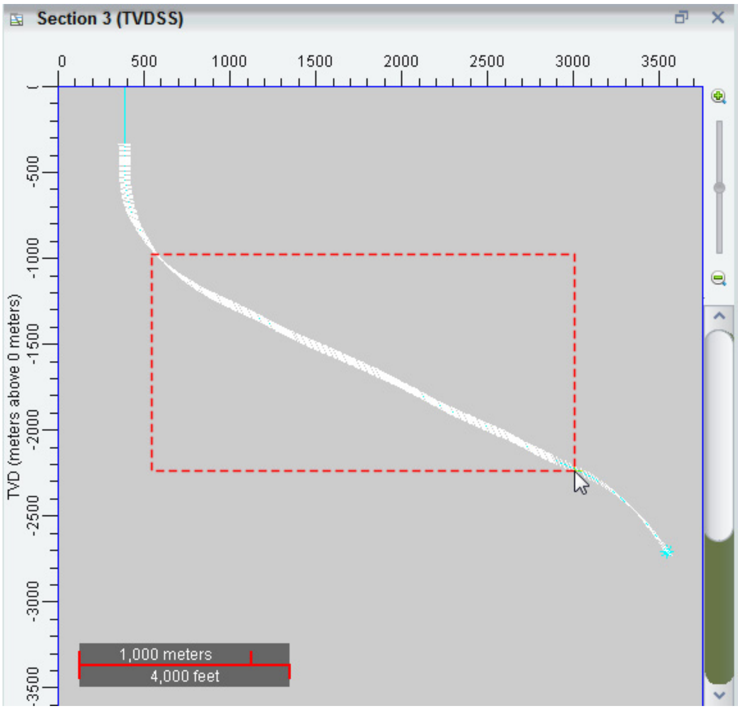
14. In *Section* view, select the ***Interpretation*** task pane tab and click the **Well Pick Interpretation** ( ) icon. On the *Actions* panel, toggle on **Dipmeter Analysis.**



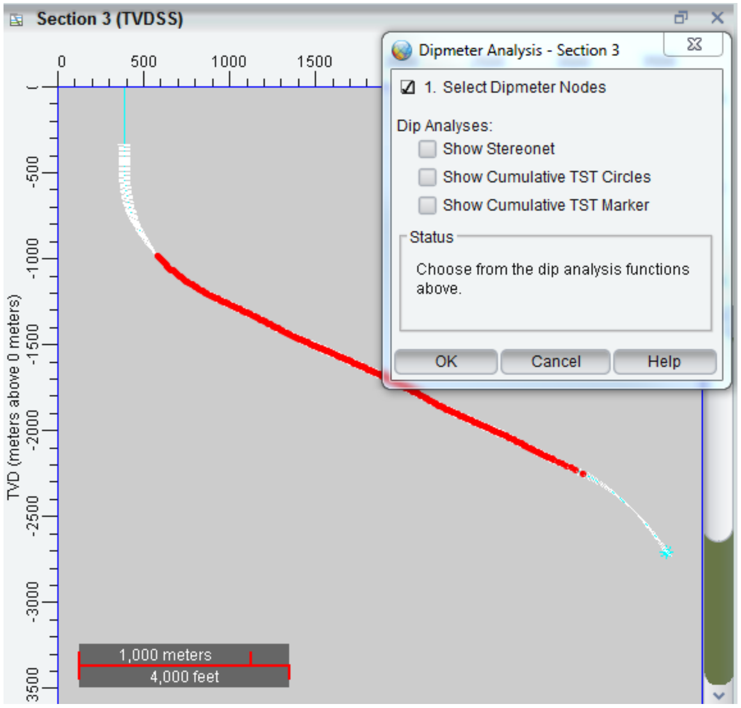
The *Dipmeter Analysis* dialog is displayed.



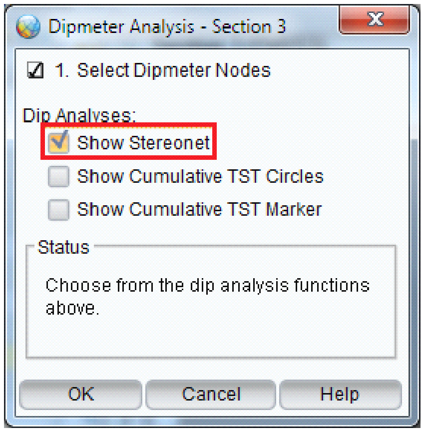
15. In *Section* view, click a location at about TVD-1000 meters and drag to about TVD-2200 meters along the wellbore to create a bounding box to select dipmeter nodes.



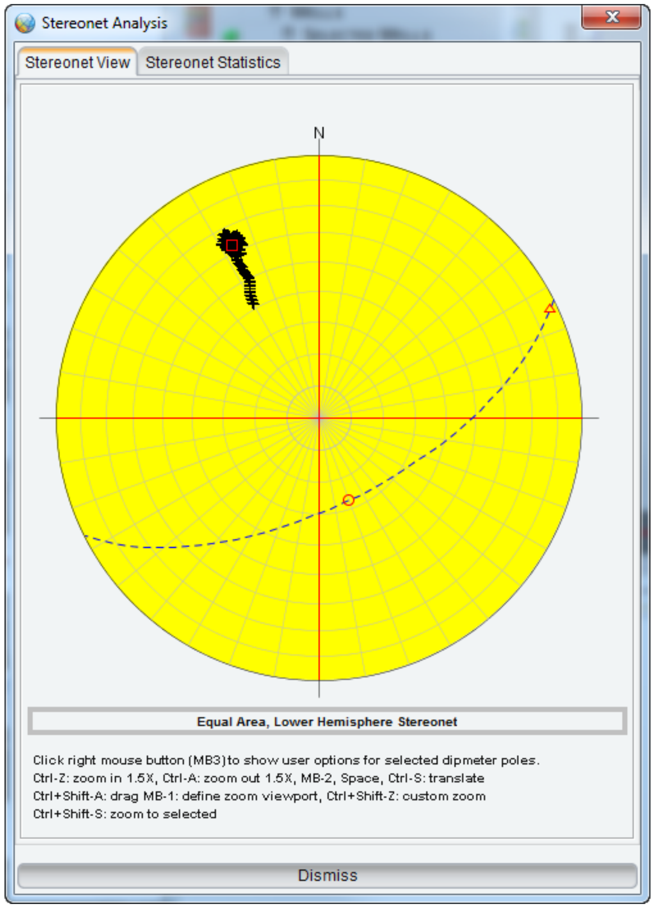
As you completed the bounding box the dipmeter nodes are highlighted.



16. In the *Dipmeter Analysis* dialog toggle on **Show Stereonet**.



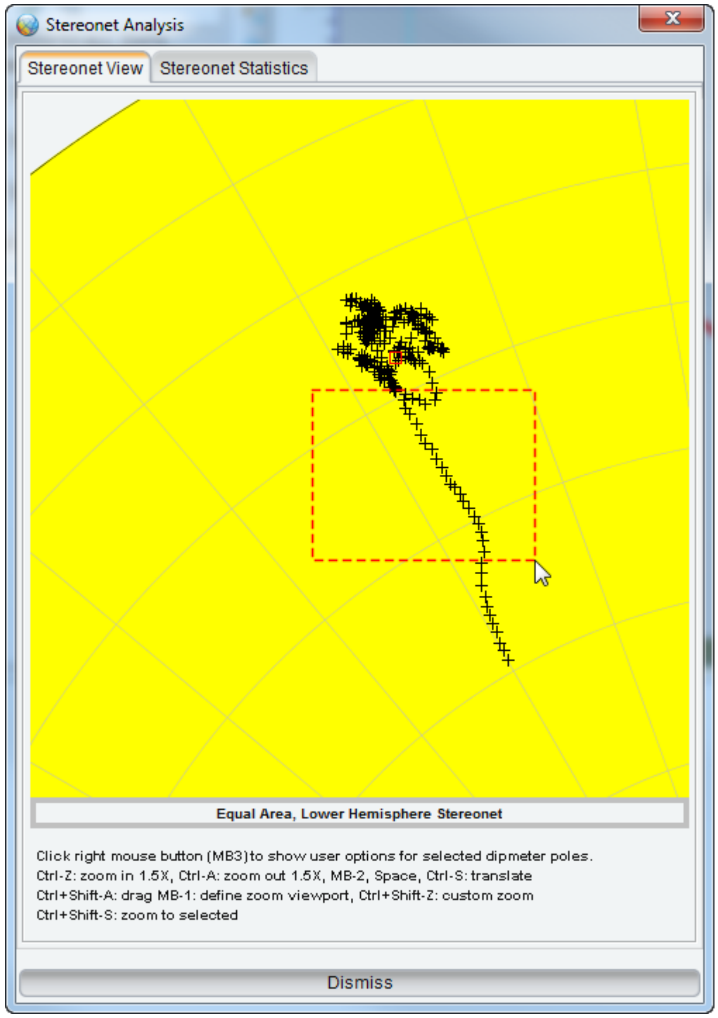
> **Note:**
>
> An equal-area, lower hemisphere stereonet display appears, showing poles of the individual dips (black crosses), best-fit plane (red rectangle and blue great circle), and best-fit fold axis (red triangle).

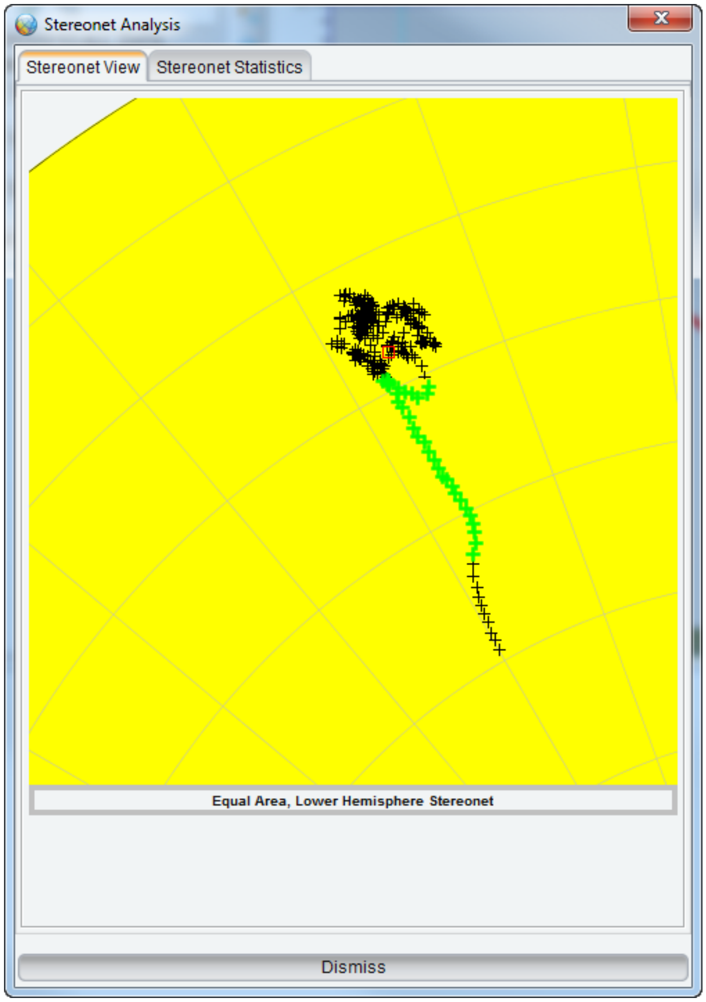


Stereonet View tab is controlled by these options:

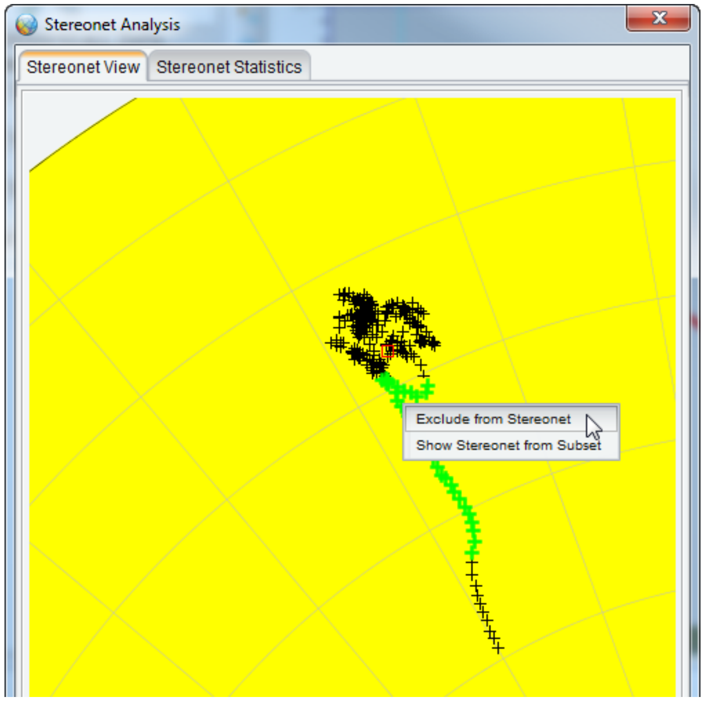
- **MB2 rollerball** — zoom in/out.
- **Click-and-drag**— select dipmeter poles.
- **MB3** — show user options for selected dipmeter poles (exclude from Stereonet or Show Stereonet from Subset).
- **<Ctrl> + Z** — zoom in by 1.5x current zoom.
- **<Ctrl> + A** — zoom out by 1.5x current zoom.
- **MB2, Spacebar,** or **<Ctrl> + S** — pan mode.
- **<Ctrl> + Shift + A** — zoom in/out to View All.

17. In the *Stereonet Analysis* dialog, click-and-drag to select a few **dipmeter poles**.

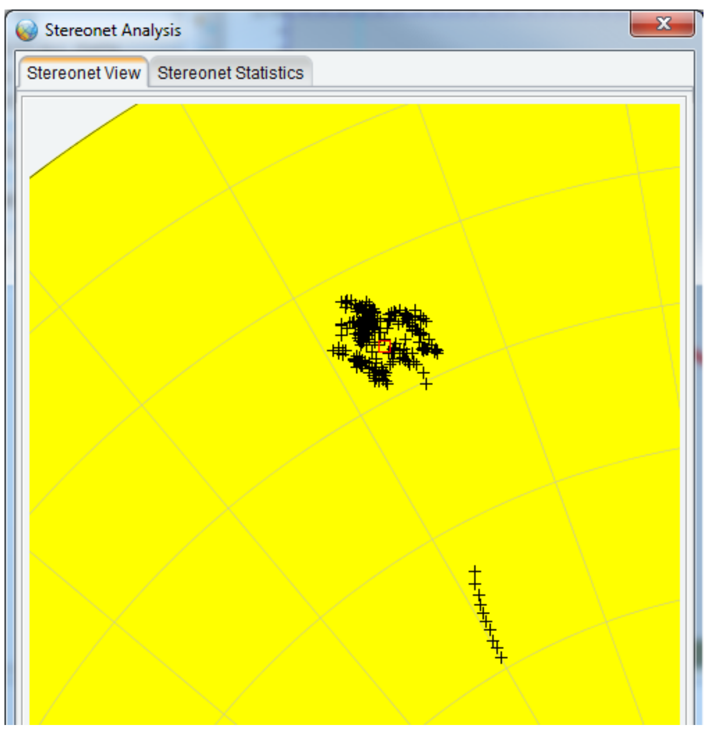



Stereonet Analysis
Stereonet View
Stereonet Statistics
Equal Area, Lower Hemisphere Stereonet
Dismiss

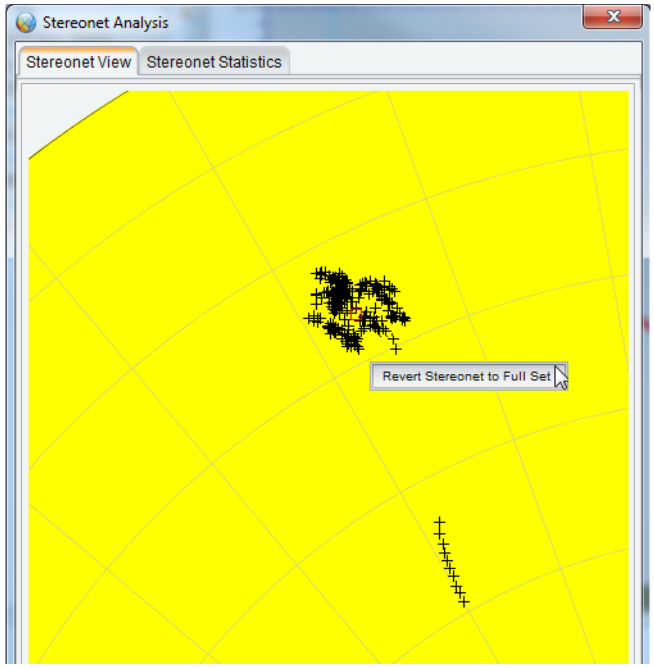
18. Put your cursor over the selected (green) dipmeter poles and **MB3 > Exclude from Stereonet**.



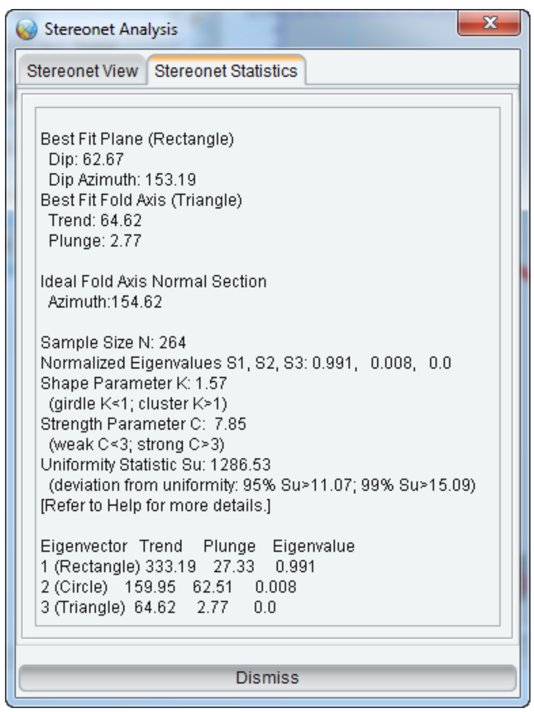
The stereonet no longer includes the previously selected dipmeter poles.



19. In the *Stereonet View* tab, **MB3 > Revert Stereonet to Full Set**.

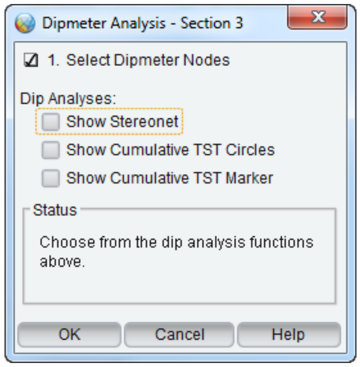


20. In the *Stereonet Analysis* dialog, click the ***Stereonet Statistics*** tab.

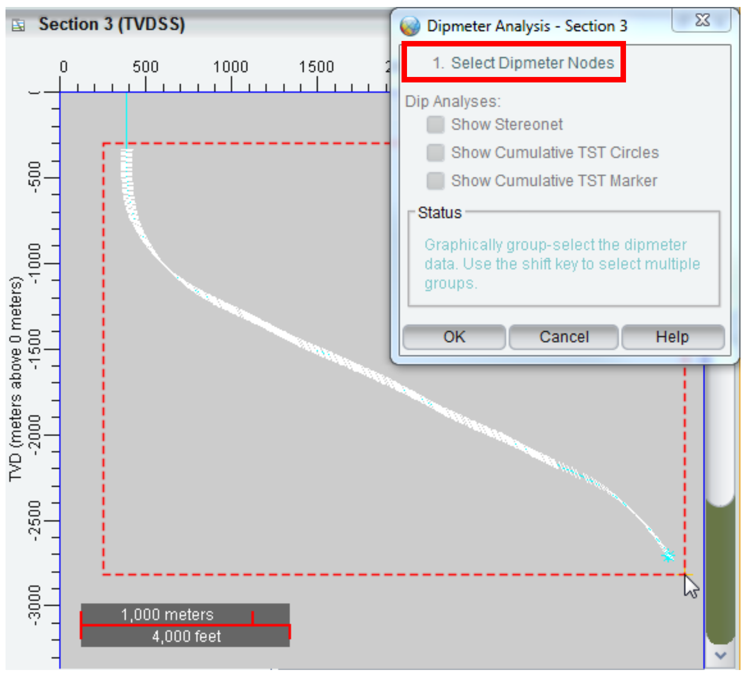


The *Stereonet Statistics* tab shows detailed information on the data, including eigenvector values and an ideal value for fold-axis normal section. The latter represents the ideal azimuth with which to draw your arbitrary section line to view the dip data.

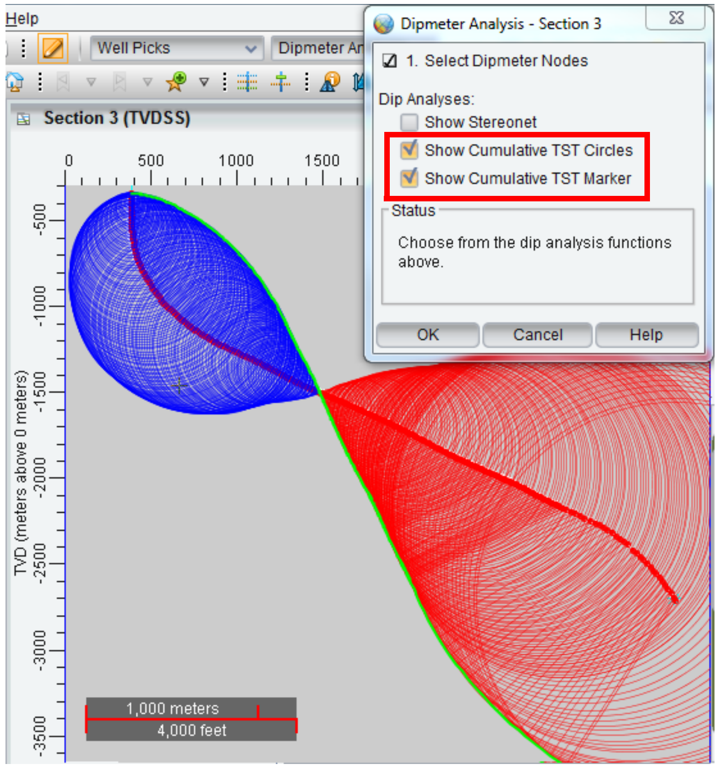
In the *Stereonet Analysis* dialog, click the **Dismiss** button. In the *Dipmeter Analysis* dialog, Show Stereonet has been toggled off with the closing of the dialog.



21. In the *Dipmeter Analysis* dialog, toggle off **1. Select Dipmeter Nodes**. In *Section* view, **click-and-drag** to select the entire well.



22. In the *Dipmeter Analysis* dialog, toggle on **Show Cumulative TST Circles** and **Show Cumulative TST Marker**.



Note:

The tool shows TST circles centered around the selected dipmeter points with radii representing the cumulative true thickness to a horizon that intersects the wellbore at the highest (minimum measured depth) selected dip node. The tool also shows a green TST Marker line connecting the TST circle radii tangent points. This line represents the best-fit horizon honoring the dipmeter data relative to the highest selected dip node.

23. In the *Dipmeter Analysis* dialog, click **Cancel** to exit.