Techlog

Conventional log analysis.

Workflow/Solutions Training.

Pre-release copy

The next generation of petrophysics

Techsia | Schlumberger

Table of Contents

Module 2 Python Script

About this Manual

Quanti is an ensemble of solutions for conventional log interpretation. It includes tools to help you with precomputations, creating flag curves, determining the standard petrophysical properties, and developing summaries.

Quanti uses our **Application Workflow Interface** (**AWI**), a generic tool that allows you to work in a multi-well and multi-zone environment, while controlling your parameters in an efficient manner. The ensemble of tools offered in the AWI will help you analyze data.

You will see how to program your own equations and applications using **Python™**. You also will see how to integrate them into your workflow.

Learning Objectives

After completing this training, you will know how to:

- create a workflow
- introduce zones
- manage your parameters
- report and save the results
- write scripts and integrate them to workflows
- use prepared scripts provided by Techsia.

What You Will Need

In this training you will need the following documents, hardware, and software:

- Operating system: Windows 2000 or later
- 256 MB of RAM available for **Techlog**
- **Techlog** installation file
- **Techlog** license.

Module 1 Quanti

The sample workflows are composed of several computational methods. In each method, you are introduced to new tools and concepts. Most of the processes, best practices and other instructions are applicable to all **Quanti** computational methods.

The dataset required for this training contains the following curves - caliper, gamma ray, neutron, density and resistivity. Before beginning the lessons, verify that these curves have been assigned a family and a unit.

Learning Objectives

After completing this module, you will know how to:

- use the **Quanti** interface
- use **Techlog** interpretation tools
- apply best practices when doing conventional log analysis using **Quanti**.

The Quanti Interface

The **Quanti** main window is shown in Figure 1. A description of its primary functions follows the figure.

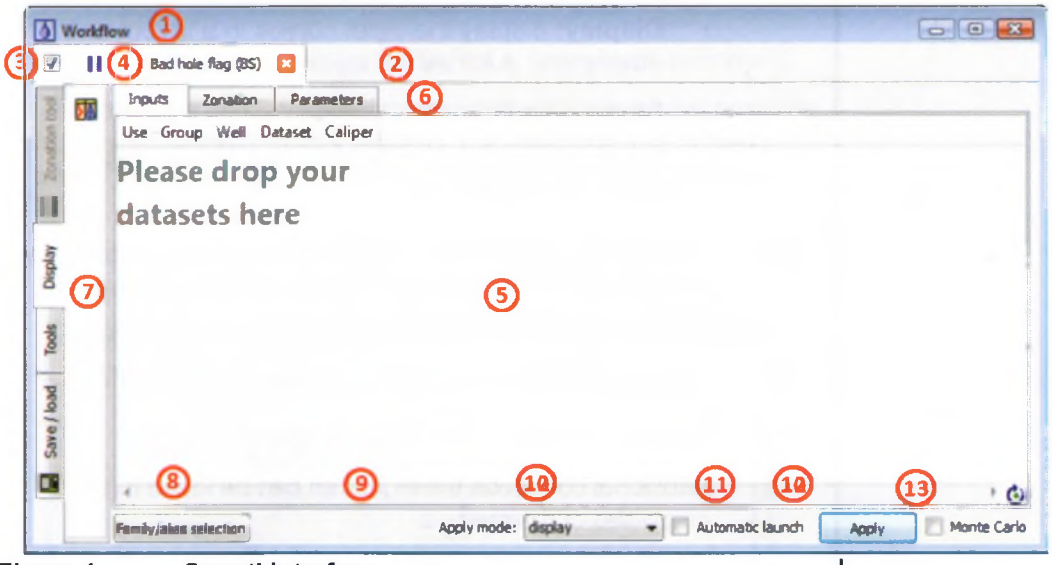


Figure 1 Quanti interface

1	Name of the workflow
2	Name of the method

3 Include/exclude a method for calculation. This is useful when launching **Cascade** mode in which the method is part of the workflow, but there is no calculation for it.

4 Pause the workflow. This is useful when launching **Cascade** mode. You may stopany method to verify the results before continuing. Click again to resume.

5 Table for input. Drag-and-drop the datasets here.

6 Horizontal tabs:

- **Input:** wells, datasets and curves
- **Zonation dataset:** define the zones for the calculation
- **Parameters:** additional equation parameters

7 Vertical tabs:

- **Display:** plots, parameters
- **Zonation tool:** additional tools to help in the definition of zones
- **Tools:** various tools, depending on the method
- **Save/Load:** workflow, parameters, reporting

8 Family/alias selection. Reopen the **Mnemonic selection** window to change Families and Aliases.

9 Apply mode :

- **Display:** display the input curves in a layout or in other plots, such as crossplots and histograms.
- **Save:** save the output curves
- **Save and Display:** a combination of both display and save modes

10 Activate the Automatic launch. Additional computation any time a modification is done in tables, including New Input, Modification of Parameters, and more.

11 Apply. Launch the calculation (keyboard shortcut **F7**).

12 Activate the Monte Carlo algorithm for the method.

Additional control on the algorithm can be found in the properties of the method.

13 Transpose the **Table** view. Reverse row and columns, which is useful when working with numerous parameters.

NOTE: Each tool, method, workflow, and plot has its own properties. Click on the desired object to view its properties.

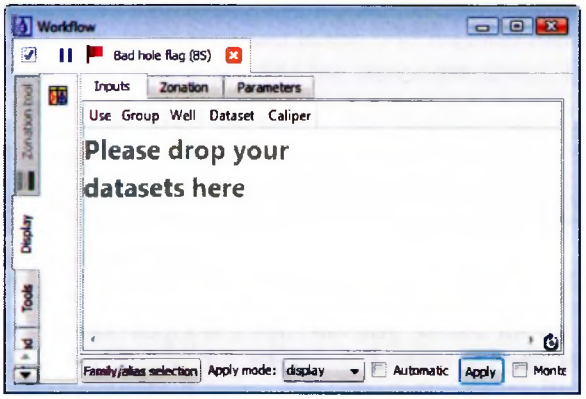
Lesson 1 Basic Concepts of the Interface

This lesson introduces you to basic concepts about using the **Quanti** interface and its interpretation tools.

Exercise 1 Applying Basic Concepts of the Interface

To get started with **Quanti** interpretation tools:

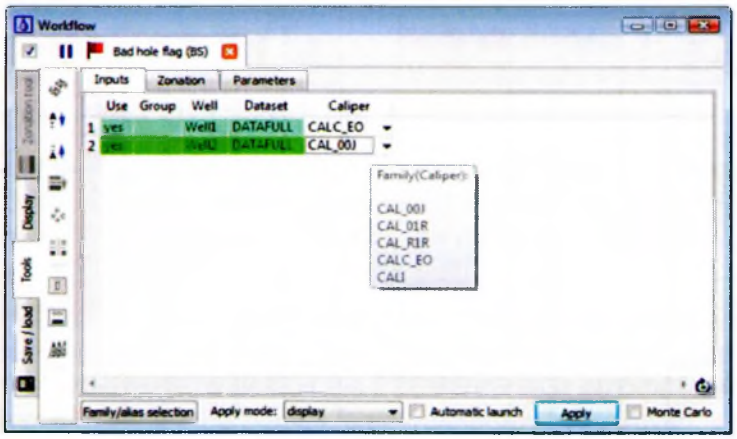
1. From the **Quanti** drop-down menu, select **Flag > Quality > Bad hole flag**.
2. In the **Family selection** window, select and insert the **Caliper** family to activate the module. The **Workflow** window opens.



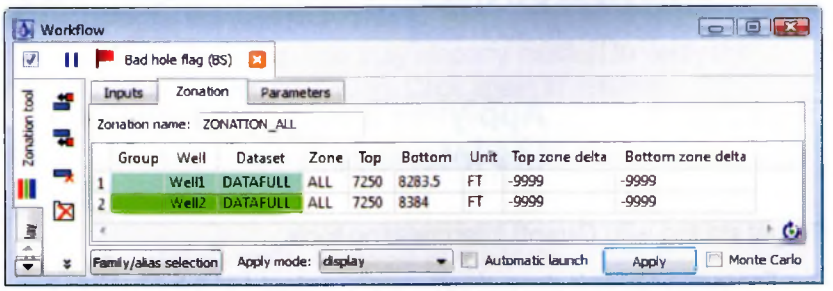
3. Drag-and-drop the dataset **DATAFULL** from Well1 and Well2.

   The interface displays one line per dataset, with all the possible curves to be used in this method. For example, the curves belonging to the **Caliper** family are arranged in alphabetical order.

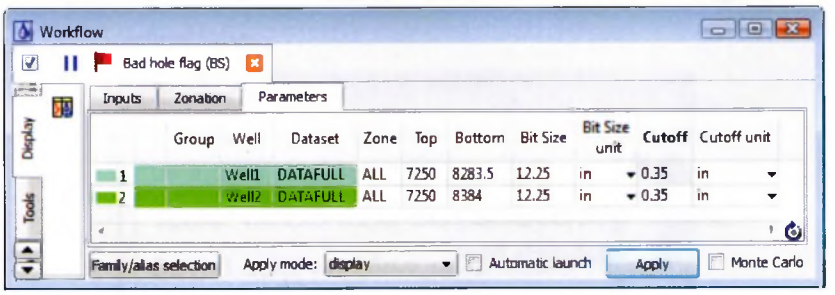
4. Mouse over a cell to view all its possible input variables and choose **CALI**.



5. Click the **Zonation** tab. Here, you define specific intervals to launch the calculations. By default, the calculations are performed from the top of the dataset to the bottom of the dataset.



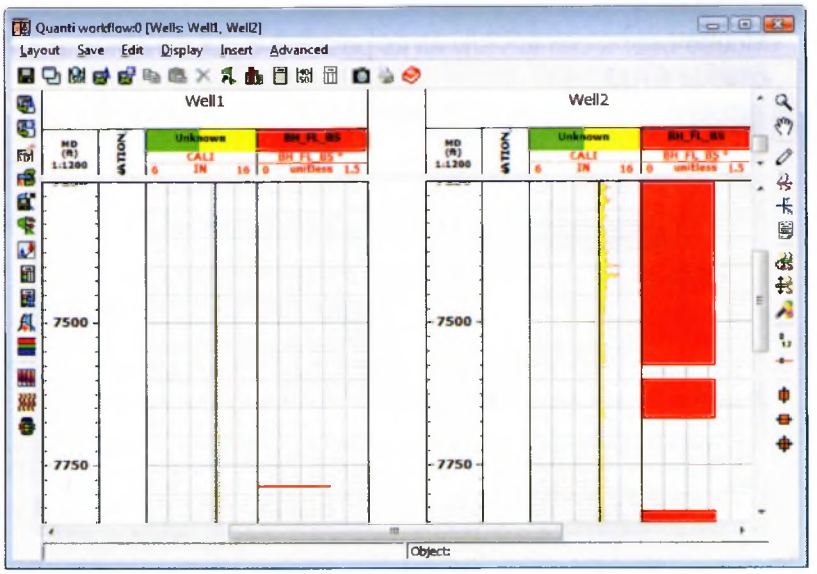
6. Define the parameters for each dataset by typing the correct values for bit size and cutoff into the table.



7. Click the **Display** tab to visualize the dataset in a layout.

   In the layout below, you can see the input curves for each dataset and the output curve **BH_FL_BS**. The output curves will be displayed only after you click **Apply** in either **display** mode or **save and display** mode.

   The output curve name has an asterisk (*), indicating the curve has not been saved in the database. In the figure below, you can see that a baseline was added in the caliper track.



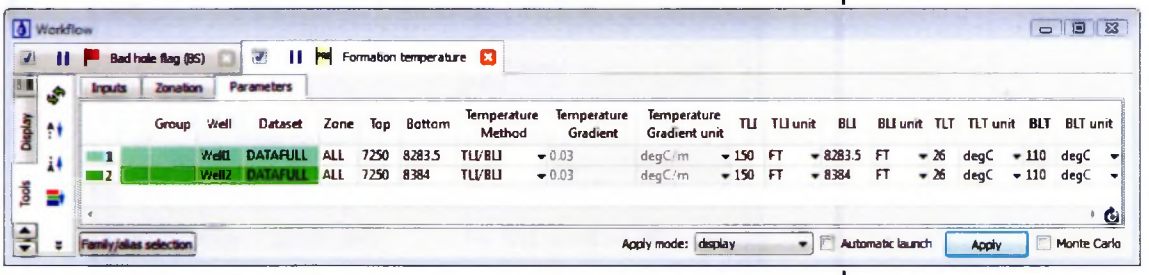
8. By default, the **Apply** Mode is set to **display**, meaning you visualize the result. You can save the result in by selecting either **Apply > save** or **Apply > save and display**.

9. Select **Apply > save and display** and click the **Apply** button. The output curve **BH_FL_BS** now is saved in the database.
10. Add another method:
    a. Select **Quanti > Precomputations > Formation Temperature**. The mandatory input for this method is a depth curve.
    b. Insert the **True Vertical Depth** family and click **Create**.

    Another method has been added to the workflow.

    Notice that the datasets used in the first method are already recognized and will be used in the second method. In addition, the zones are identical.

    In the **Parameters** tab, some parameters are grayed out and some are white. Depending on the method you choose, these parameters will become available or not. By default, the **Gradient** method is active.
11. Modify it to **TLI/BLI** and view the corresponding parameters.
12. To learn more about the method and its equations, click on the method to make it the active object and press the **F1** key. Refer to the Help file for an explanation of the workflow parameters:



13. Click **Apply**.

    Techlog does not distinction between unit systems. The input curve can be in *meters*, while the parameters can be in *feet*. Regardless of the unit, the units are converted on the fly when you click **Apply**.
14. Notice that the input and output variables of this method are now added to the layout. The output track looks empty, but this is because by default the FTEMP curve is automatically assigned to the *Formation temperature* family displayed between 20degC and 60 degC.

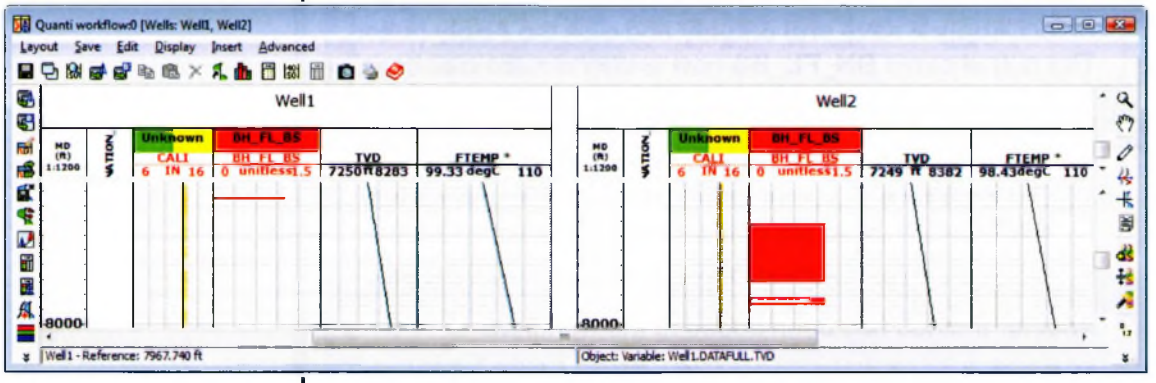
You can modify the display of the curve over the layout:

a. Select the curve over the layout.

b. In the **Properties** window, click the **Limits** tab.

c. Change the property *Horizontal axis, Type* from **Family** to **Variable**.

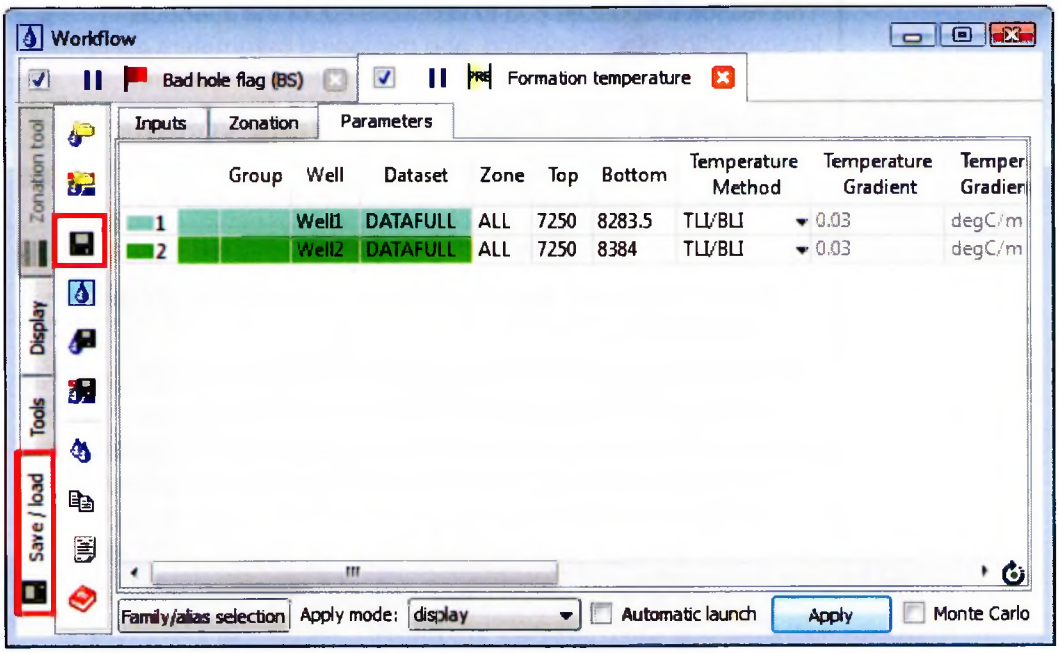
The figure shows the curve from minimum to maximum values.



15. In the **Properties** tab of the method under the **Output variables** tab, modify the default family, unit, and name of the output curve.

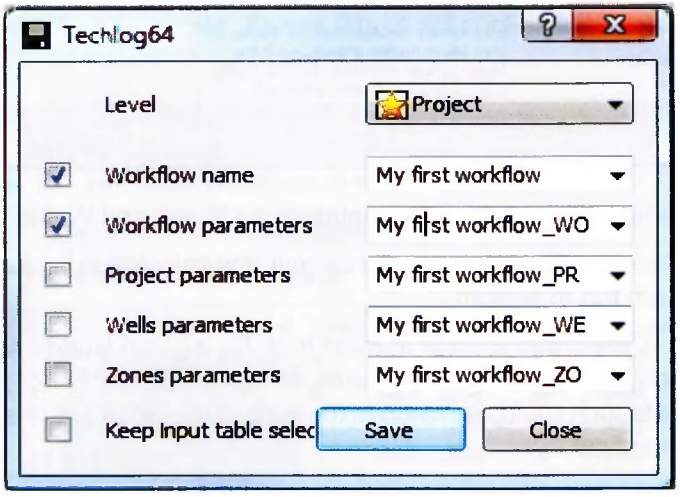
Output_1	FTEMP
Name	FTEMP
Use	yes
Description	
Family	Formation Temperature
Unit	degC
Palette	
Sensitivities	no
Monte Carlo distribution	yes

16. When the workflow is complete, save it with its associated parameters. Go to the **Save/Load** tab and click **Save workflow** .



17. A window opens in which you must define the name for the workflow and the level at which it will be saved: Project, User, or Company.

    After naming the workflow, the parameters file is named the same name, but with a suffix. This allows you to make a connection automatically between a workflow and its associated parameter file.



    Name the workflow and click **Save**. The workflow and the parameter file now are saved in the project and can be launched in another workspace or **Techlog** session.

18. Close the workflow and the layout, or open a new workspace.

Lesson 2 Shale Volume from Gamma Ray

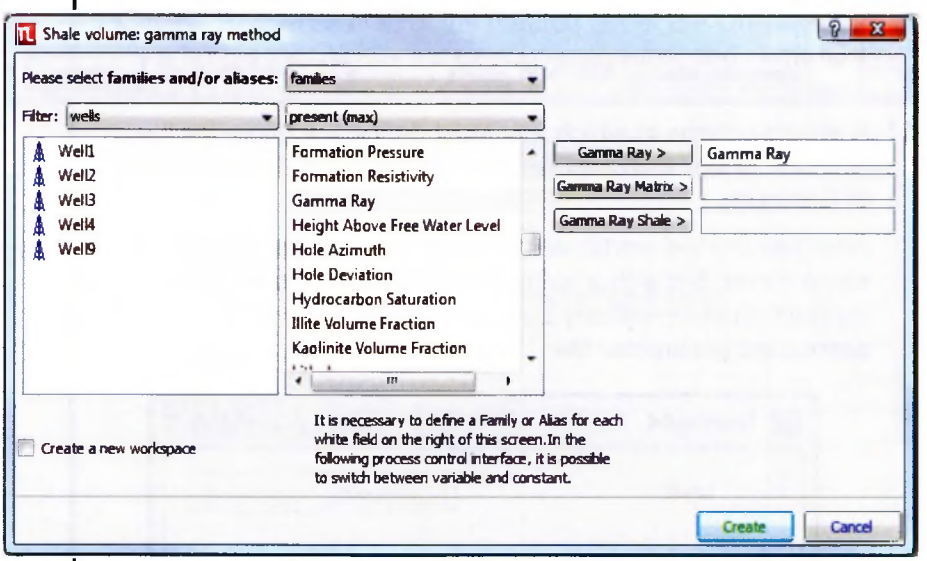
This lesson introduces you to concepts about the interface, interpretation plots, reporting, and managing parameters and zones.

Exercise 1 Calculating Shale Volume from Gamma Ray

To calculate shale volume from gamma ray:

1. From the **Quanti** drop-down menu, select **Shale volume > Gamma Ray**.

   A gamma ray curve is necessary to launch this method. In its absence, **GR_matrix** and **GR_shale** can be used as inputs; they will be inserted as continuous curves. If **GR_matrix** and **GR_shale** are not inserted as continuous curves, they will be applied as constants.



2. Drag-and-drop the **LQC** datasets for Well1 and Well2.
3. Click the **Zonation** tab. By default, the calculation is launched from top to bottom.

   It is important to bear in mind that, for a given workflow, you can use only one set of zones. In other words, it is not possible to launch multiple intervals to multiple methods in the same workflow.

   TIP: When a zone is inserted or removed for a method, it is disabled for all the methods within the workflow.

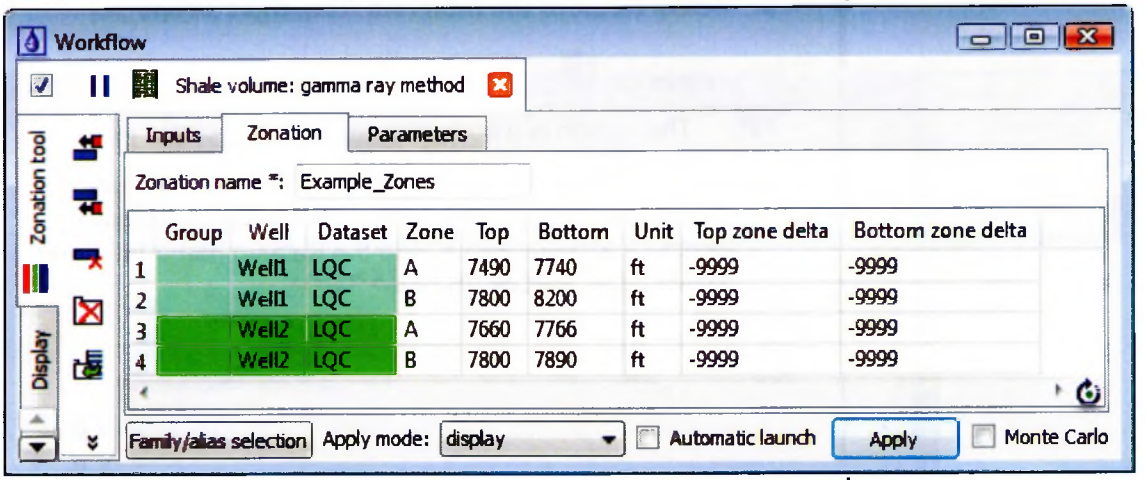
4. Create a zonation table with specific depth values using the tools that are available as icons on the left side of the **Workflow** window and on the contextual menu.

Icon	Shortcut	Action
	Ctrl + I, B	Add one line before the current position
	Ctrl + I, A	Add one line after the current position
	Shift + '	Delete the current row
	Ctrl + Del	Delete all

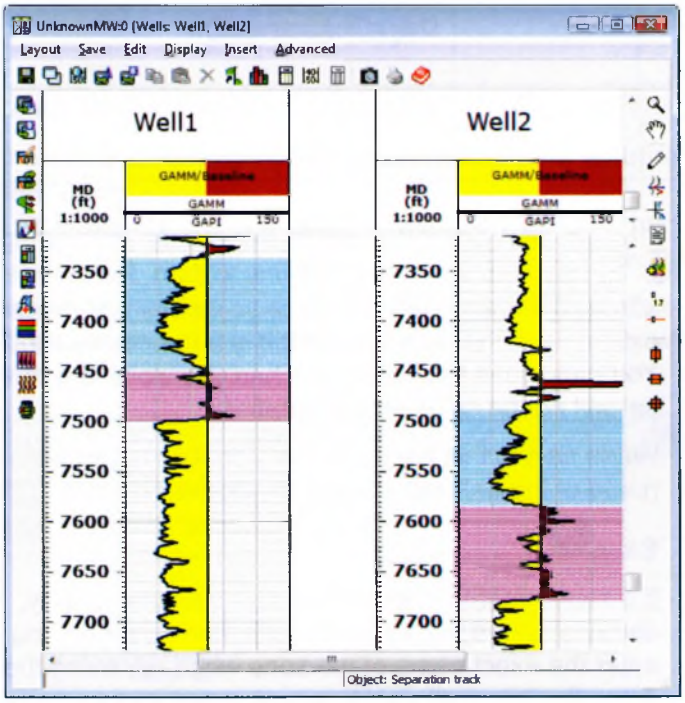
When creating the zonation table, it is important to note these issues:

- Manually creating a table of zones means that these zones exist only in the workflow, which is a temporary zone set. You must save the workflow when you close it, or the zone set will not be saved in the database.
- When saving this zonation, the *Zonation name* will be the name of the dataset. (Select the **Zonation** tab and click **Save**  .)
- If a zone repeats in several wells - lithostratigraphy, chronostratigraphy, hydraulic unit, and so forth - you must enter the exact name of the zone when retrieving the table. A spelling error will create a new zone.

The figure below shows two zones in two wells (4 lines) - Zone A and Zone B. When you save this new set of zones, the set name will be **Example_Zones**, and it will contain two zones per well.



5. Insert a zone using the **Interactive Selection** tool .
   a. Open a layout and insert zones using the **Interactive Selection** tool.
   b. Brush the intervals of interest; each color is defined as a different zone. The figure shows that four intervals are selected, with two intervals per well.



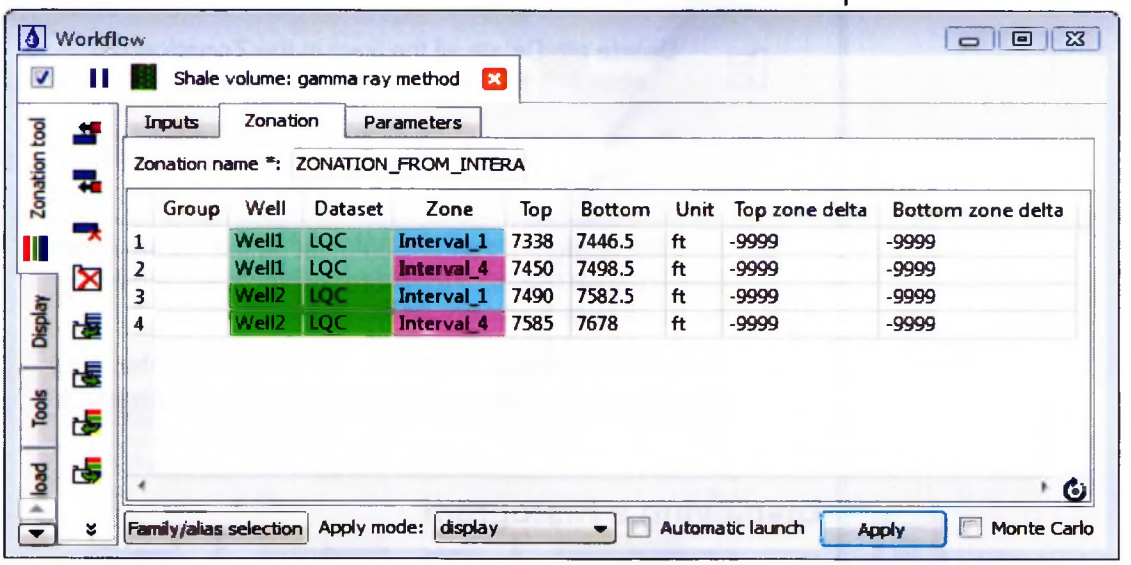
   c. After selecting the intervals over the layout, click the **Zonation** tab.

6. Insert an existing set of zones from the **Zonation** dock window:
   a. Select **Restore the intervals from the current selection** .

TIP: This option is also available on a right-click menu.

The zones populate the table with the corresponding depths and zone colors, and the dataset name becomes **ZONATION_FROM_INTERACTION**.

b. Save this set of temporary zones if you wish to use it later.



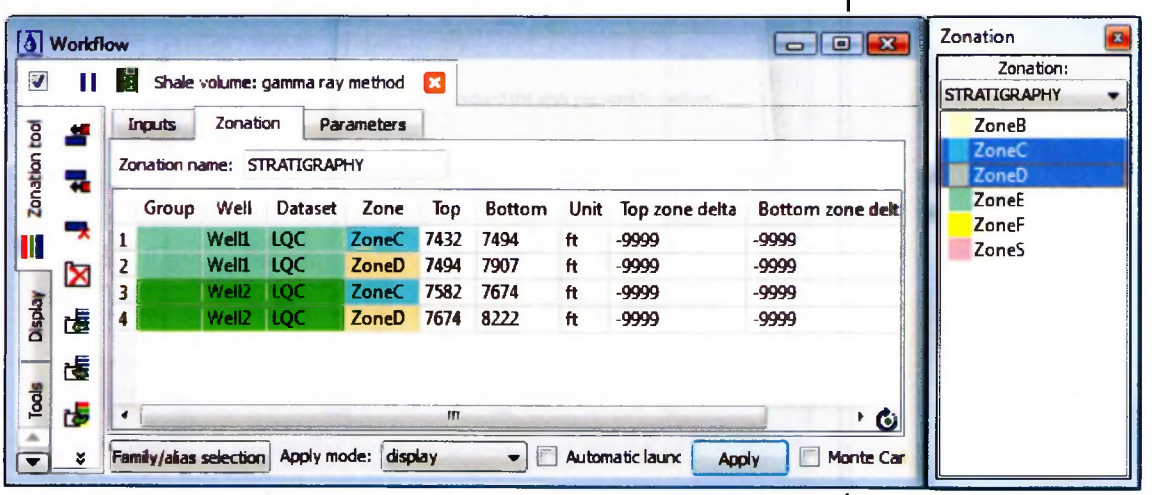
7. Insert the existing set of zones from the **Zonation** dock window.

   a. In the **Zonation** dock window, choose the zonation set **STRATIGRAPHY**.

   b. Inside the set, choose the zones to introduce. In this example, choose **ZoneC** and **ZoneD**.

   c. Insert the zones by selecting **Restore the intervals from the current zonation**  from the icon list, or right-click to display a menu.

      The top zone delta and bottom zone delta will launch the calculation on these intervals, even though they are outside the zones. It is useful to show a change in the nature of the data.



Two options are available as icons. They are located along the

left side of the **Workflow** window and on the right-click contextual menu.

Delete all: Delete all the lines in the **Zonation** tab.

Parameters copy: Copy the zonation table to the clipboard.

d. After declaring the zones, you must define parameters for each interval. Several tools are available for selecting these values.

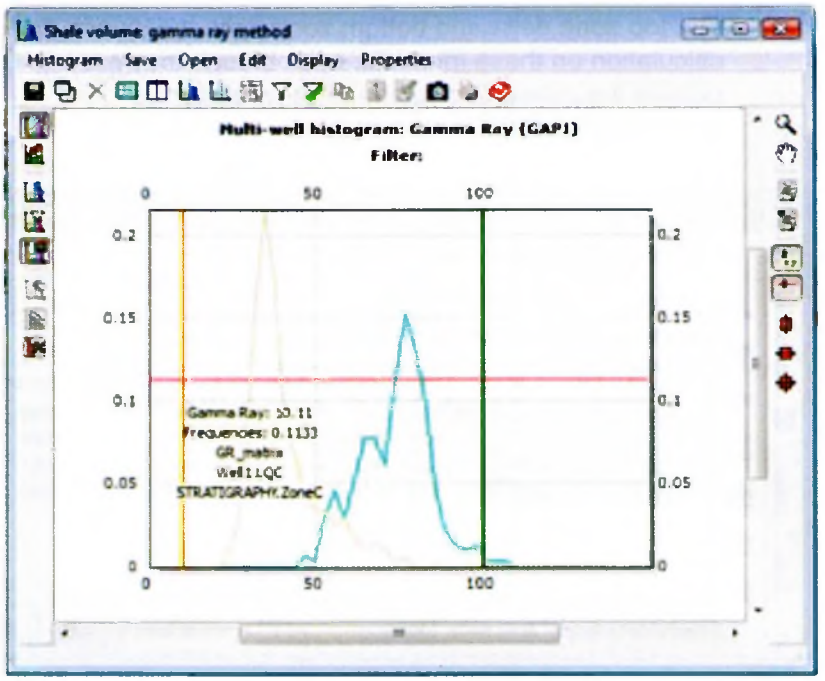
e. If you know the exact value for a given parameter, type the value in the corresponding cell on the **Parameters** tab.

Displaying a Histogram

To display a histogram:

1. Click the **Display** tab (vertical tab located on the left side of the **Quanti** window).
2. Click the **Histogram** icon.

   This histogram contains ONLY data from the corresponding wells and zones that are present in the **Quanti** window.
3. Each zone has two baselines over the histogram. Mouse over the baseline to visualize which baseline corresponds to which well and zone.

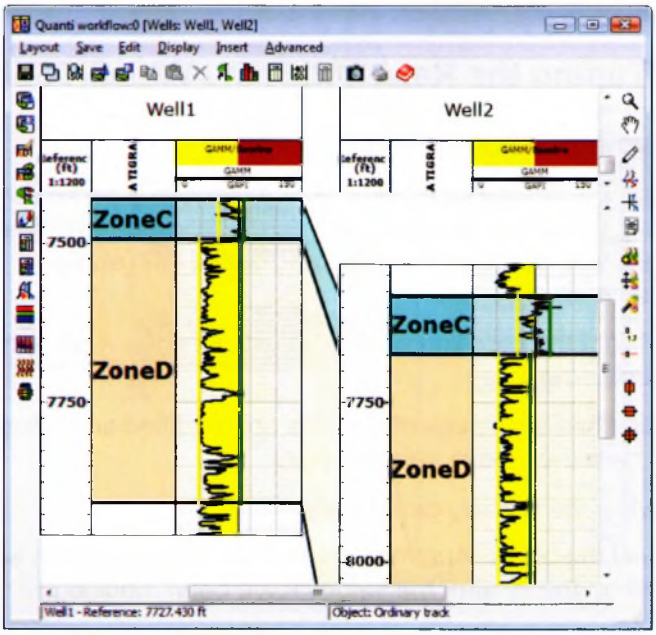


Displaying a Layout

To display a layout:

1. From the **Display** tab, click the **Layout** icon.
2. Click **Show current layout parameters** to display the layout with the zones and baselines for each zone.

NOTE: There is a full interactivity between the table and the plots. Modifying a value at one location automatically modifies the value in other locations - table, histogram, and layout.



Using Statistical Tools

For each zone, it is possible to choose the 5 or 95 percentile - the mean or the median value, respectively.

To use the **Statistical** tool:

1. Click the **Tools** tab.
2. Select the cells in which you wish to insert **5** or **95** percentile, and click the appropriate icon – **5%** or **95%** .

TIP: Icons without a small rig designate a single well, while icons with a rig represent multiple wells.

After changing the values, the baselines display over the plots.

3. Click **Apply** or press **F7** to view the output layout of the input curve and the output curve.

Additional Options

There are several more options that can help you manage parameters and zones.

Automatic launch: If you tick this option, modifying a parameter will result in a new calculation.

Fill-Up/Fill-Down: This option allows you to propagate a value throughout a table. The option is available on the right-click menu and as keyboard shortcut **Ctrl + D** or **Ctrl + U**.

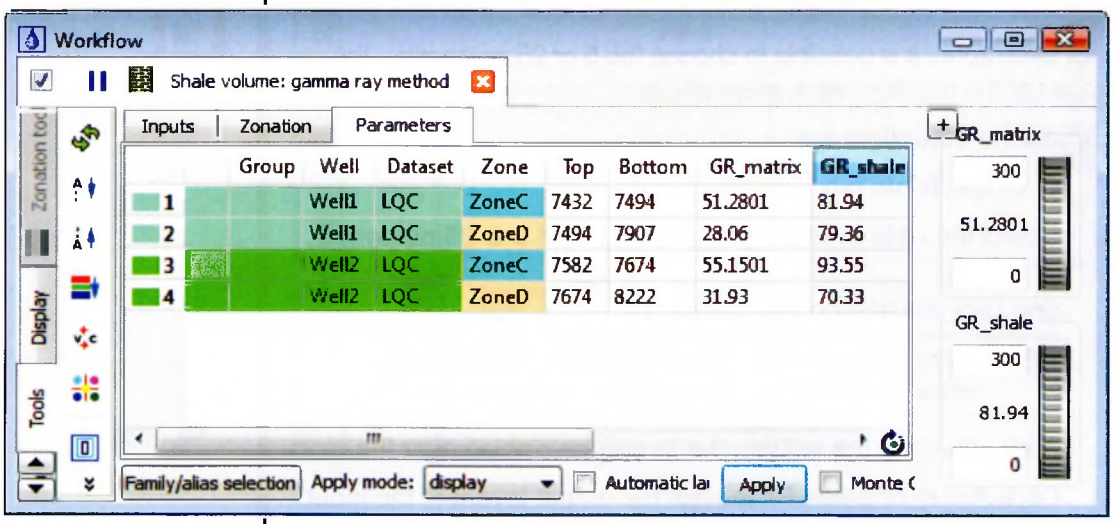
Extend to the zones: This option allows you to propagate an identical value from one zone to all wells in the same zone. The option is available on the right-click menu and as keyboard shortcut **Ctrl + Shift + Z**.

Fine-Tuning the Result with the Wheel Tool

To fine-tune the result:

1. In the **Parameters** tab, click on the name of a parameter and select the option **Add a wheel**. In the following example, a wheel was added to the **GR_matrix** and **GR_shale** parameters.
2. Highlight the cells you wish to adjust using the wheel (move it up and down).

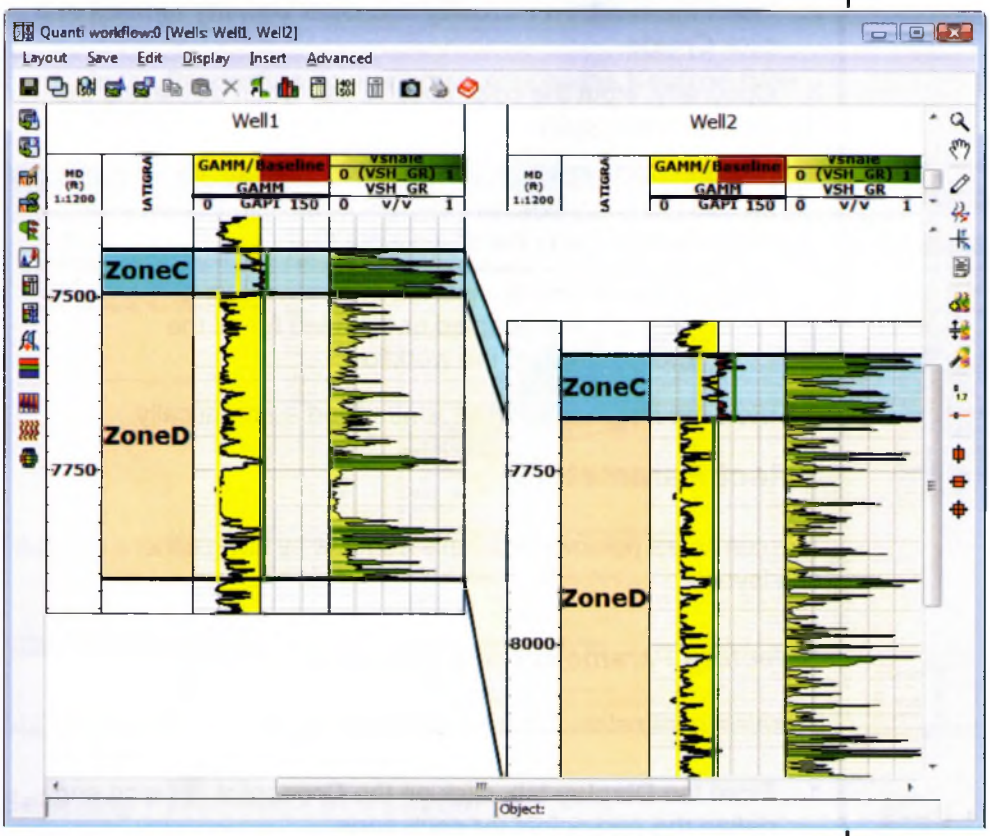
   Notice that the values of the cells are modified accordingly, and the baselines move over the plots.
3. To view the results, click **Apply**.
4. Select the option *Automatic launch* so the calculation will be done on the fly after the values have been modified.
5. Control the sensitivity of the wheel by modifying the minimum and maximum limits of the wheel.



6. Detach the wheel interface from the workflow by clicking .

7. When you are satisfied with the parameter settings and the display outputs, select either **Apply mode > Save** or **Apply Mode > Save and display**.

   The computed **VSH_GR** curve will be saved in the database and its name will appear in the **Project browser** under the selected dataset. The plot shows the input curve and the output curve for each well.



Lesson 3 Shale Volume from Neutron-Density

Techlog provides a way to compute a shale volume, but you will not convert all of them. This lesson describes how to compute a shale volume from neutron and density logs.

Exercise 1 Calculating Shale Volume from Neutron-Density

To calculate shale volume from neutron density:

1. From the drop-down menu in the **Quanti** window, select **Volume of Shale > Neutron-Density**.
2. Insert the **Neutron Porosity** and **Bulk Density** families and click **Create**.
3. Optionally, input the color code to use when displaying the data on a crossplot.

   IMPORTANT: For a given workflow, only one set of zones can be used. It is not possible to launch multiple intervals to multiple methods in the same workflow.

   TIP: When a zone is inserted or removed for a method, it is enabled or disabled for all the methods within the workflow.

4. View the input curves; the zones filled automatically.

Select Parameters

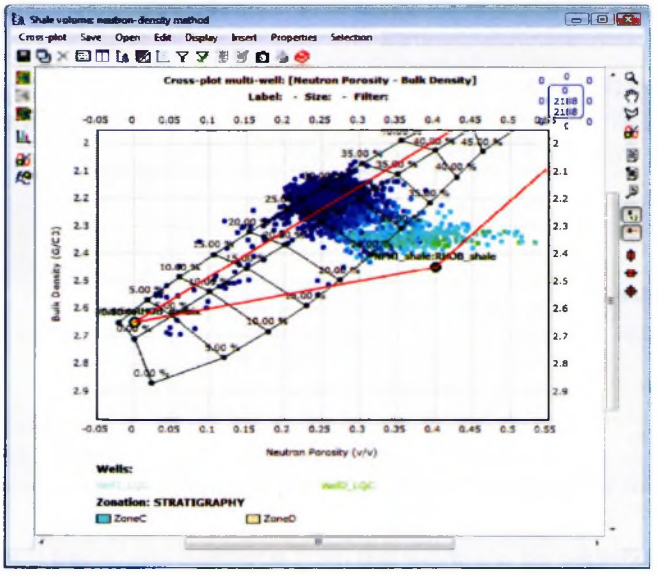
You can select parameters for this workflow by using either a crossplot or a layout.

Selecting Parameters in a Crossplot

To select parameters using a crossplot:

1. From the **Display** tab, click on the **Crossplot** icon and define the end-points for each zone.

   TIP: Any modification on the plot also changes the parameters table.



2. The default chart corresponds to the **NPHI** tool.

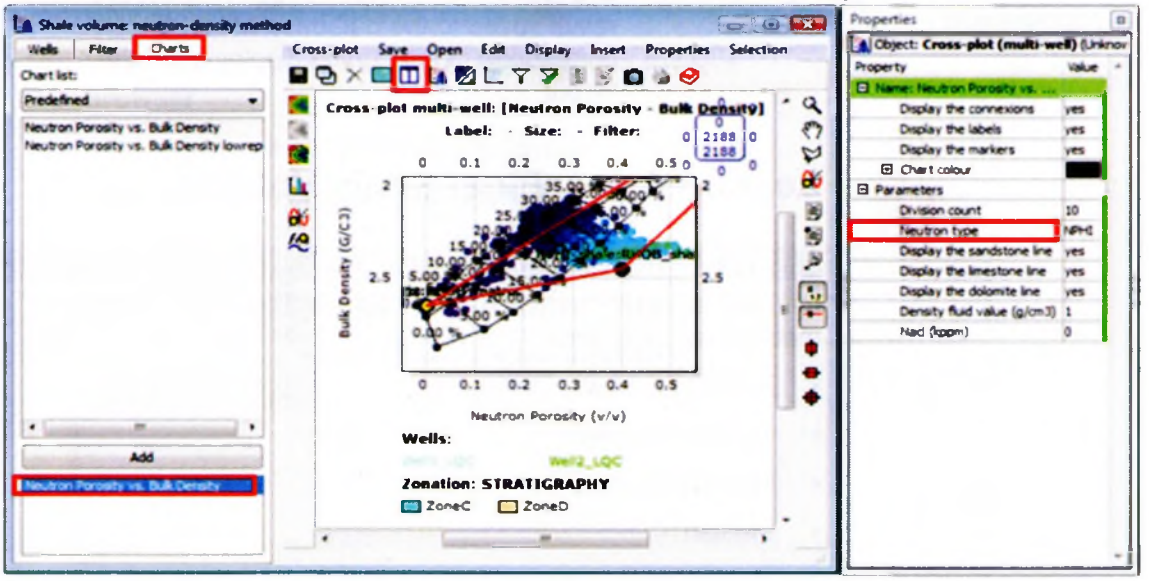
   To modify the chart according to the neutron type:

   a. On the plot, under the **Display** tab, select **Display variable side box** or press **Ctrl + G**.

   b. In the **Variable** side panel, click the **Charts** tab.

   c. Click on the chart name **Neutron Porosity vs. Bulk Density** to display the chart properties.

   d. In the **Properties** dock window, choose the **Neutron type** property.



Selecting Parameters in a Layout

To select parameters using a layout:

1. Click **Apply** to display input curves (neutron, density) and the new output curve (**VSH_ND**) on a single plot.
2. When input curves are displayed, click **Show current layout parameters** on the **Display** tab.

   The baselines on the plot control the shale points (green lines) and the matrix points (yellow lines).
3. Choose parameters for each zone and click **Apply**. Do not save the resulting curve.
4. In the properties of the method, click the **Output variables** tab.

   Two output curves are listed, but only one is enabled by default. More information about additional curves can be found in Help by clicking **Help** or pressing **F1**.

Each method has its own **Apply Mode**:

- Shale volume from gamma ray was set to **Apply Mode > Save and display**.
- By default, shale volume from neutron-density is set to **Apply Mode > Display**.

  Remember that in the header of the layout, curves with an asterisk (*) next to their names are not saved in the database. These curves are used in *Display* mode only.

WARNING: If you launch computations for the first time in *Display* mode, then change a parameter and launch computation in *Save* mode, the result saved to the database will NOT correspond to the curves displayed on the layout.

Lesson 4 Final Shale Volume

This method calculates the volume of shale by combining different methods for computing the volume of shale. These computation methods are proposed:

- Arithmetic mean
- Geometric mean
- Harmonic mean
- Median
- Minimum
- Maximum
- First present
- Sum

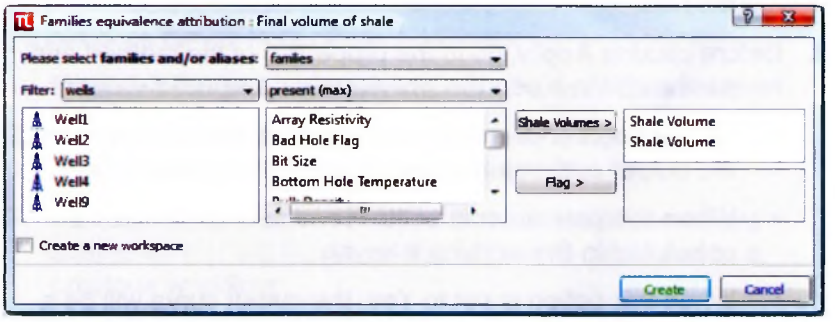
Exercise 1 Calculating Final Shale Volume

To calculate final shale volume:

1. From the drop-down menu in the **Quanti** window, select **Shale volume > Final**.
2. In the **Family/Alias Assignment** window, insert the family name several times, one time for each curve you wish to use.

   In this example, you will calculate a final **Vshale** curve from two curves - **VSH_GR** and **VSH_ND**.
3. Optionally, you can add a **Flag** (qualitative) curve; additional parameters are available if you need to use a flag curve.

4. After choosing the families, click **Create**.



5. In the **Input** tab, mouse over one of the cells and view the possible options. The available inputs are
   - **VSH_GR**: as it is saved in the database and **Project browser**
   - **VSH_GR***: as it appears over the layout
   - **VSH_ND***: as it appears over the layout

Even if you do not save the curve in the database, you can use it in additional methods if it was calculated previously in the workflow.

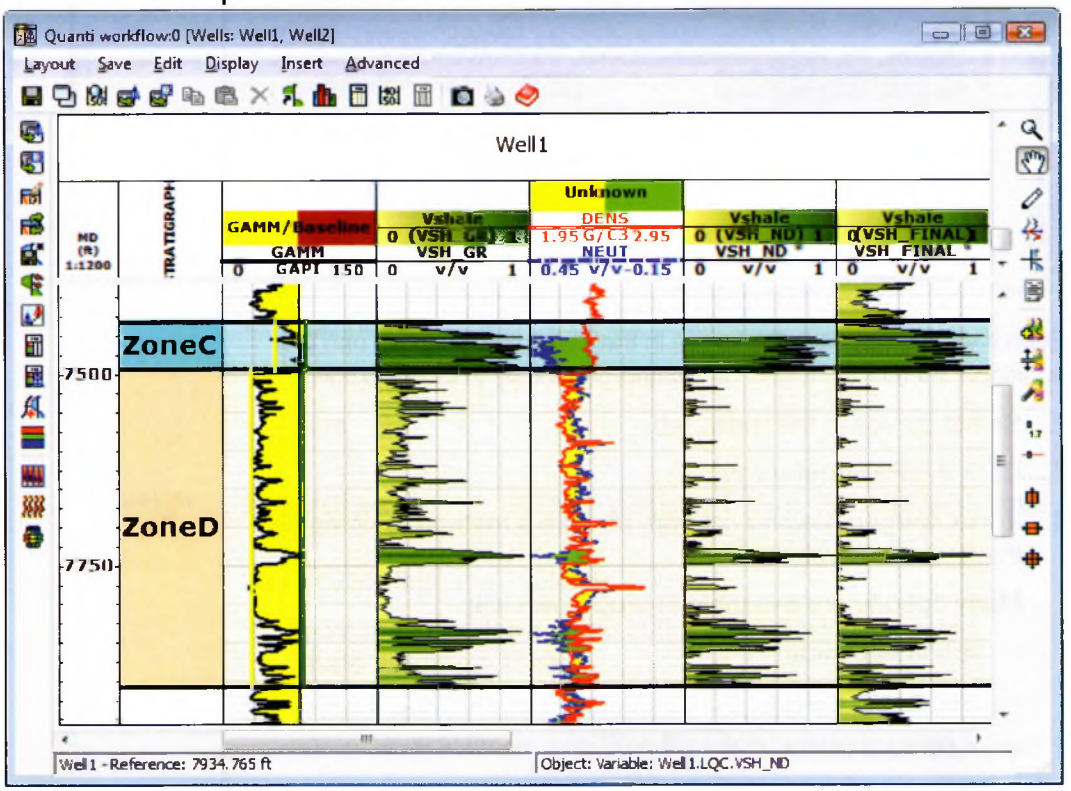
6. Select curves in the **Input** tab.
7. Click the **Parameters** tab.
8. In the *Merged Method* column, choose a method to compute the final shale volume.

   Many options are available in drop-down list:
   - arithmetic mean
   - geometric mean
   - harmonic mean
   - median
   - minimum
   - maximum
   - first present
   - product and sum

For more information about each option, refer to the Help file of the method.

9. Before clicking **Apply**, go to the properties of the method and review the option **Keep the previously computed values**:
   - This option works only if a curve with a name identical to the output curve already exists in the **Project browser**.
   - When this parameter is set to **No**, a new curve will be calculated in the working intervals.
   - When this option is set to **Yes**, the output curve will be a mixture of two curves:
     - In the zones of calculation, the curve will be calculated using the parameters present in the workflow.
     - Outside these intervals, the curve from the database will be displayed.

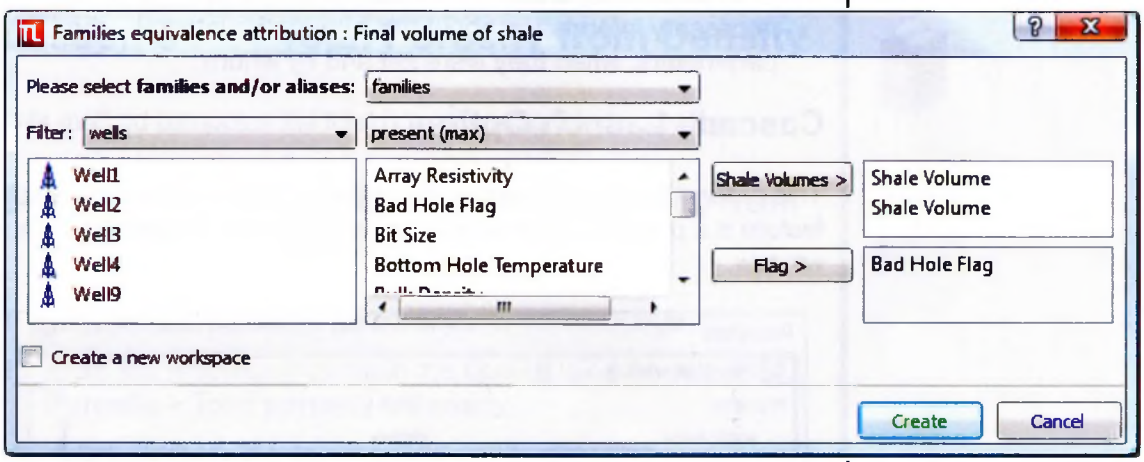
   In the **LQC** datasets, a **VSH_FINAL** curve already exists. When you click **Apply**, an output curve is created outside ZoneC and ZoneD.



Introducing Curves from Different Datasets

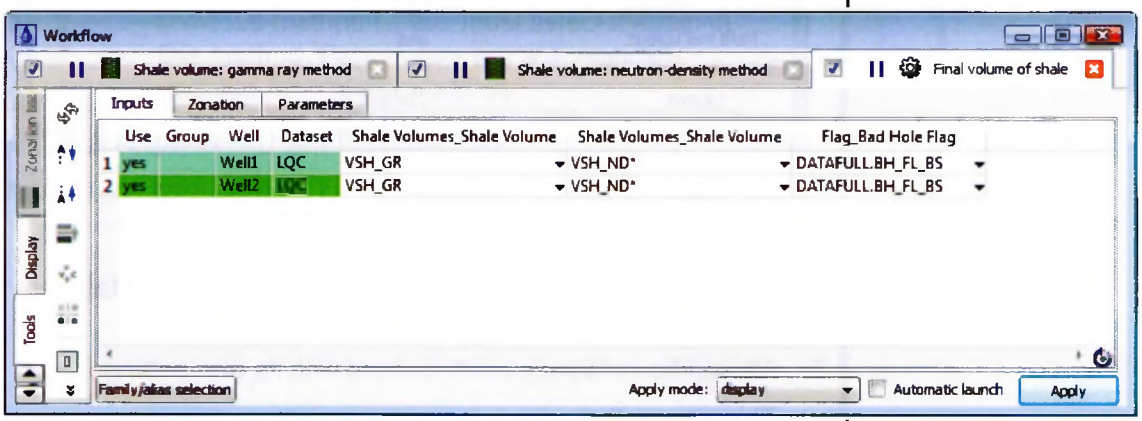
To introduce curves from different datasets:

1. From the **Project browser**, click on the word **Datasets**.
2. In the **Properties** dock window, under the **Options** tab, toggle *Resampling, Quick resampling* to **Yes**.
3. Mouse over any cell and notice that additional curves are available. You will introduce the flag curve calculated in a previous workflow.
4. Return to the workflow interface and click **Family/Alias selection**  .
5. Add the **Bad hole flag** as a flag curve and click **Create**.



6. An additional column has been added to the input table.

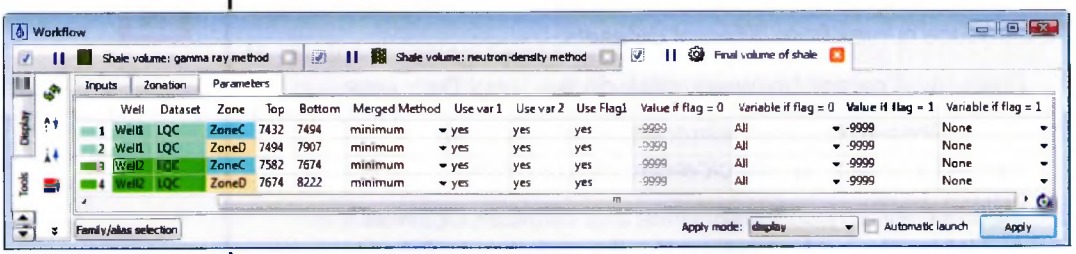
   Select the variable **BH_FL_BS** from **DATAFULL**.



7. Additional columns have been added to the **Parameters** tab. You must specify what you wish done in a column if the flag value is **0** or **1**.

   In this instance of the BH_FL_BS variable, flag = 0 means that hole conditions are good, while flag = 1 means hole conditions

are not good. The parameter table should look like the figure.

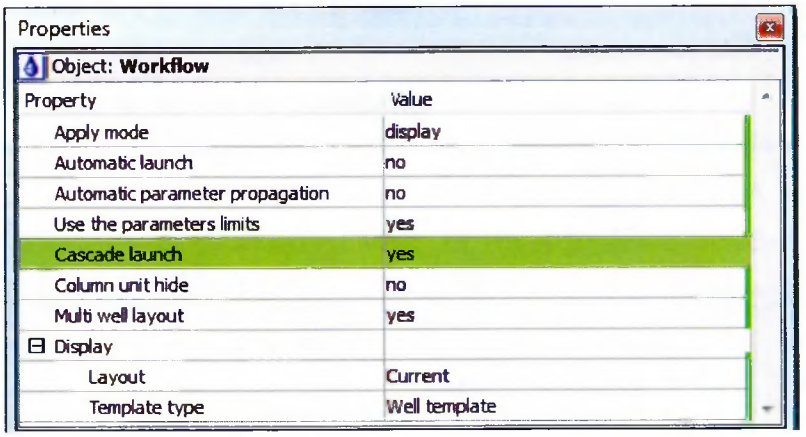


8. Select **Apply Mode > Save and display** and save a new **VSH_FINAL** variable.
9. In the **Project browser**, highlight the newly created **VSH_FINAL**.

   In its properties, under the **History** tab, you can see all the necessary information about this curve - the intervals, the parameters, when they were set and by whom.

Cascade Launch Option

This option allows you to launch the entire workflow in one click. This feature is a property of the workflow available in the **Properties** dock window.



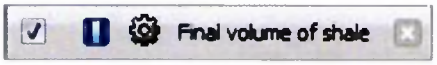
To use **Cascade Launch** mode:

1. Close all interpretation plots.
2. Select the first method **Shale Volume from Gamma Ray** and click **Apply**.
3. All workflows launch, each method with its own settings.

   This means that:

   - **VSH_GR** is saved into the database
   - **VSH_ND** is not saved (It is for display only.)
   - **VSH_FINAL** is the minimum among these curves, taking into account the **BH_FL_BS** curve.

4. Close the interpretation plot and click **Pause** for the *Final Shale volume*.



5. Choose **Shale Volume from Gamma Ray** and click **Apply**. This time, the workflow launches until encountering the pause, which allows you to examine the results.
6. When you are satisfied, click **Pause** again, and the workflow launch will be completed.
7. Clear the tick mark next to the method name, and the method is grayed out. This method will not be used in the calculations.

Lesson 5 Total Porosity from Density

This method computes the total porosity using the density log.

Exercise 1 Calculating Total Porosity from Density

To calculate total porosity from density:

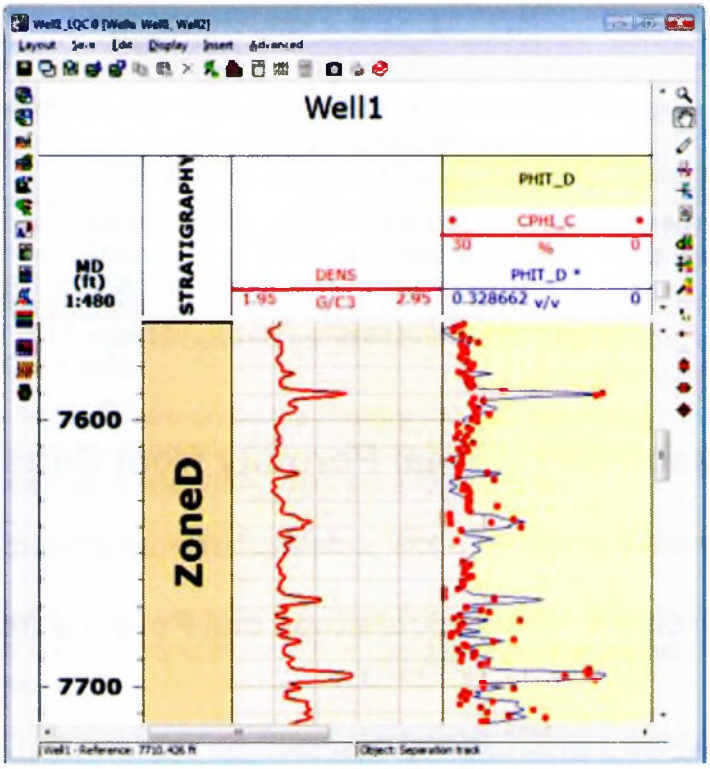
1. From the drop-down menu in the **Quanti** window, select **Porosity > Total porosity > Density**.

   In this method, the density curve (white space) is mandatory, while and all other inputs (gray space) are optional. Without the density curve, the calculation could not be launched,

   NOTE: Even though the density curve was selected in a previous method (), you must select it again in this method. Parameters are automatically updated for a given workflow.

2. Click **Apply** and view the output curve.
3. Add the core porosity curve within the dataset **CPHI_C**, and overlay it with the **PHIT_D** curve.

Use the wheels and automatic launch options to fit the two curves.The result is shown in the figure on the following page.



4. When you are satisfied with the results, review your parameter table.
5. Return to , *Shale Volume from Neutron-Density* and review the values of the **RHOB_matrix** and **RHOB_fluid** parameters.The initial values that were used to calculate the shale volume were modified; hence, the parameters do not reflect the final result.

   Click **Apply** again, and the output shale volume curve will be different.
6. Because the name of the parameter was identical in both methods (same physical magnitude), when you modify a parameter in one location, the value is modified in other locations within the workflow. To avoid this behavior, be sure to give different names to parameters.

   Over the interface, click on a parameter name and from the drop-down menu. select **Rename parameter**.
7. Type a new name for the parameter. Notice that changing a parameter value does not modify another parameter.
8. For all porosity methods that include density, neutron and sonic curves, you can perform an on-the-fly hydrocarbon correction. However, this correction is activated only if you introduce a resistivity curve.

To add a resistivity curve:

1. Click **Family/Alias selection** and add a resistivity mnemonic (Family: deep resistivity).
2. After the resistivity variable is added, a new tab with several parameters becomes active. These parameters are used during the hydrocarbon correction.
3. The correction is an iterative process that solves the value of porosity, taking into account the **Tools Response** equation.
   a. Compute porosity.
   b. Compute water saturation (via Archie's law).
   c. With a correction factor, correct the input curve for the presence of hydrocarbon. The salinity of the mud filtrate and the density of the hydrocarbons are taken into account.
   d. Compute a new porosity value and compare it to the previous one. If the difference between the two porosity values is infinitesimally small, the iteration stops.
   e. Verify the parameters.
   f. Select **Apply Mode > Save and Display** and click **Apply**.
   g. Compute a total porosity.
   h. Use total porosity and the shale volume information, compute an effective porosity curve.

Lesson 6 Saturation using Archie's Method

This method computes the formation water saturation using the Archie equation.

Exercise 1 Calculating Saturation using Archie's Method

To calculate saturation using Archie's Method:

1. From the drop-down menu in the **Quanti** window, select **Saturation > Archie**.
2. Select **Resistivity** and **Porosity** mnemonics and click **Create**.
3. Select the input curves.
4. Click the **Parameter** tab and choose parameters.

Selecting Parameters

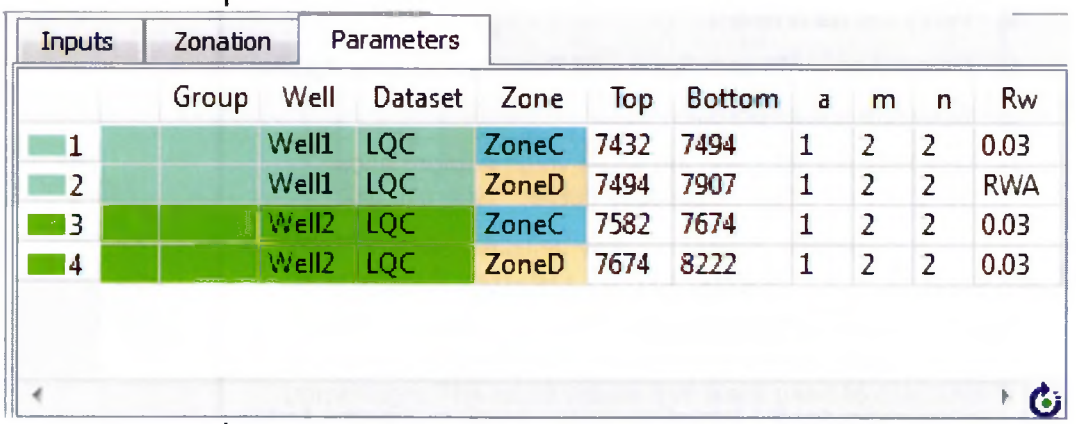
There are several ways in which you can choose parameters.

Using a Continuous Curve for a Specific Zone

If you have a formation water curve, and you wish to use it for a given zone:

1. Highlight the cell you wish to modify and click **Switch between a variable or a constant parameter** . This opens a window with all the curves in the same dataset.
2. Choose the curve you wish to use instead of a constant for a specific zone.

   In this example, Well1 uses a constant value for ZoneC (0.03) and a curve for ZoneD (RWA).



	Group	Well	Dataset	Zone	Top	Bottom	a	m	n	Rw
1		Well1	LQC	ZoneC	7432	7494	1	2	2	0.03
2		Well1	LQC	ZoneD	7494	7907	1	2	2	RWA
3		Well2	LQC	ZoneC	7582	7674	1	2	2	0.03
4		Well2	LQC	ZoneD	7674	8222	1	2	2	0.03

3. Remove the RWA curve before continuing.

Using a Pickett Plot

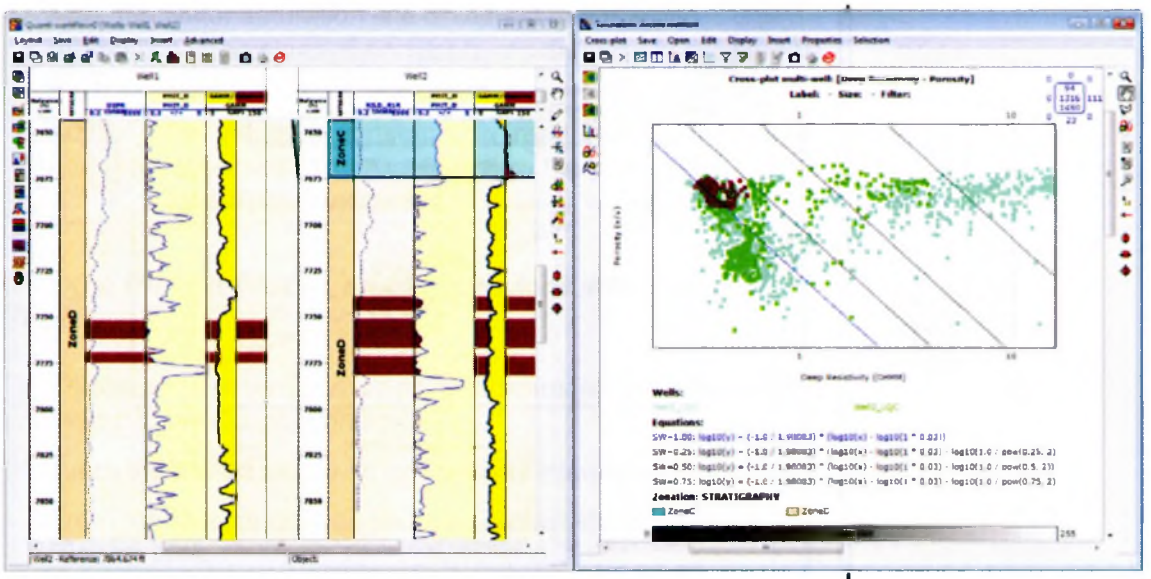
To use a Pickett plot:

1. From the **Display** tab, launch a layout and a Pickett plot.

   Use the **Interactive Selection** tool over the layout and choose a water zone. The depth points you select over the layout are highlighted over the Pickett plot as well.

   TIP: You may wish to add a gamma ray curve as well to this layout.

   In the example below, the **Interactive Selection** tool (in brown) has selected specific intervals. The data points present in these depth intervals are selected over the crossplot (also in brown).



2. Visualize only the selected data points by selecting **Display selection only** from the **Display** drop-down menu.
3. Insert regressions from the **Insert** menu by selecting **Current selection** to create a regression that fits all the selected points.

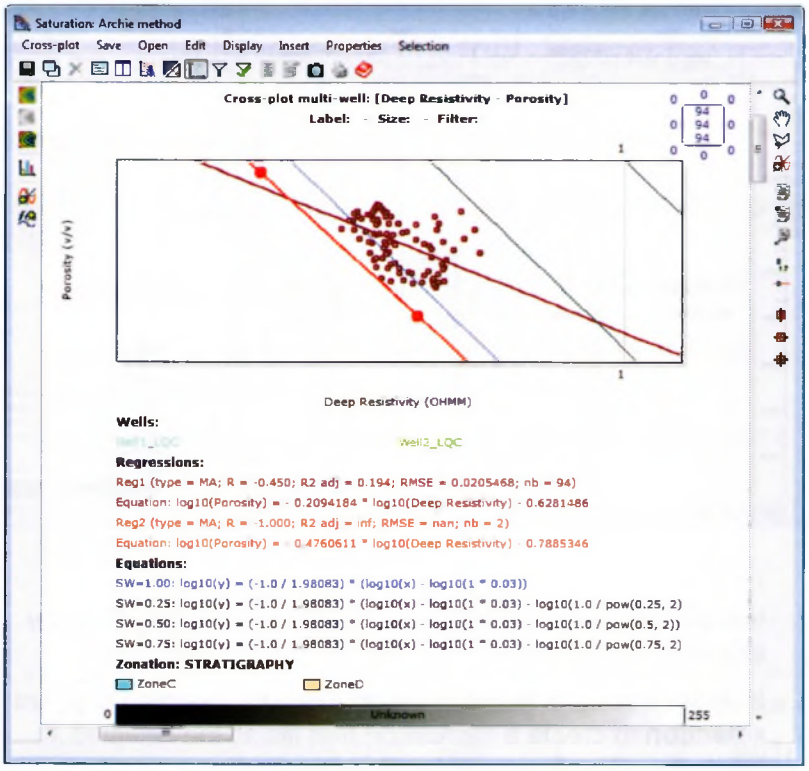
   OR

   Click **Custom regression**  to create a regression according to user-defined points.
4. In the example below, the crossplot displays over only the data that were selected from both wells.

   The colors are meaningful:

   - The blue regression line is the one that will be used to calculate the Archie parameters: a, m, n, and RW.
   - The brown regression line is the one that was inserted on the current selection.
   - The red regression line is a user-defined regression. In this example, the two red points were inserted by a user.

   Refer to the figure at the top of the following page.



In the properties of this plot, a unique tab named **Pickett plot** displays with the equation's default parameters. It is possible to link the Sw=1 line (blue regression) to any of the selected regressions (brown or red) by setting the option to **Yes**.

By default, the Sw=1 line clips to the first regression. You can toggle between regressions (Reg1, Reg2). After setting the option to **Yes**, parameter values update automatically over the plot and over the parameters table.

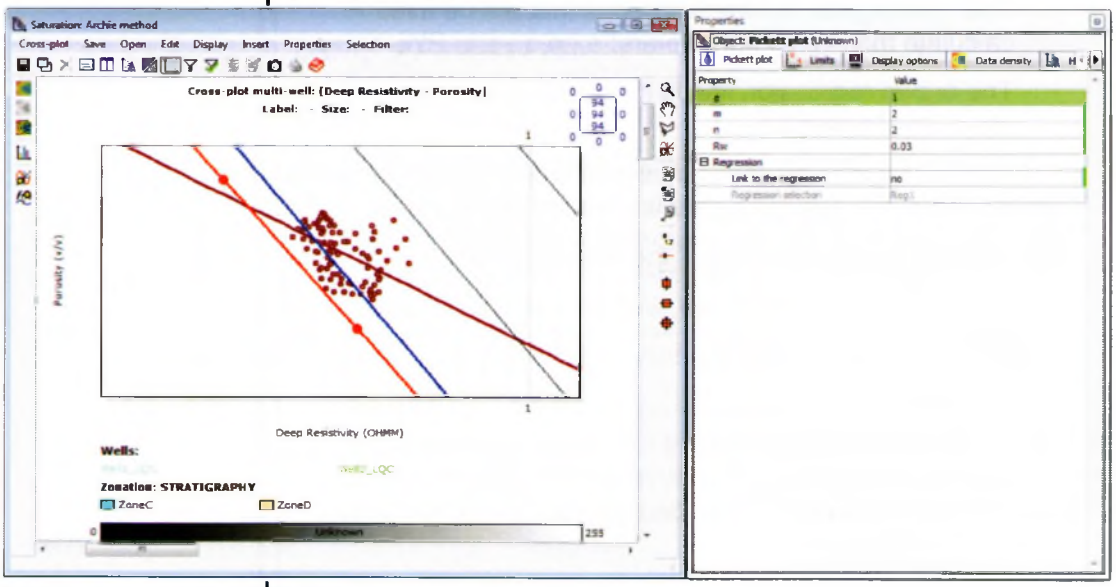


Figure 2 Pickett plot

After the water saturation line is clipped to a specific regression, the values of constants are modified in the **Quanti** table.

NOTE: The values of constants are read from the regression. You should review the parameters to verify their accuracy.

Using a Pickett Plot as an Independent Interpretation Tool

The Pickett plot can be launched as an independent interpretation tool not related to the **Quanti** workflow.

To launch the Pickett plot as an independent interpretation tool:

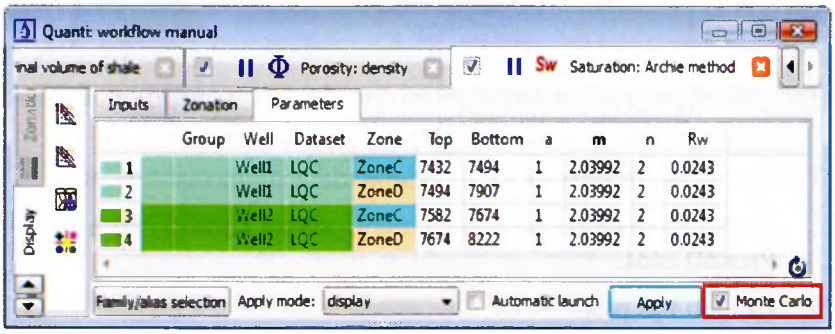
1. From the **Quanti** drop-down menu, select **Quanti plots > Pickett plot**.
2. Launch the plot.
3. Choosing the input mnemonics. You may use a saved parameters file with values of a,m,n and Rw).
4. If a parameter file does not exist, click **Cancel**. By default, not all possible output variables are calculated.
5. View all the possible output curves and their automatic assignments (name, family, unit, and palette).
   a. Go to the properties of the method and click the **Output variables** tab.
   b. In this tab, define a prefix and/or suffix to the names of all curves. This is useful when comparing several versions of the curve.
   c. Group the curves. This is useful when you have several output curves.

Lesson 7 The Monte Carlo Algorithm

This algorithm can be launched separately on each method.

To launch the Monte Carlo algorithm:

1. Place a tick mark in the box next to *Monte Carlo* in the lower right corner.



2. Click **Apply** and the algorithm launches with all its default values.
3. Visualize and modify these values by going to the properties of the method under the **Monte Carlo** tab.

   When activated, the algorithm displays additional output curves for each output variable: mean value and standard deviation.

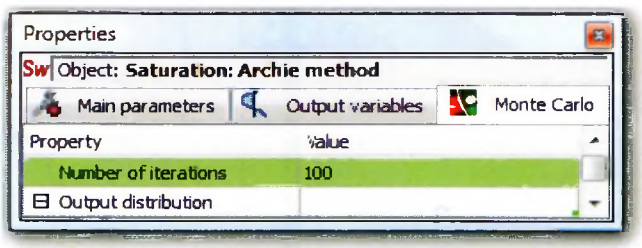
   Additional output curves also are available in the **Properties** dock window: median, maximum, minimum, percentile minimum (with a default of 15%) and percentile maximum (with a default of 85%).

   A histogram with the distribution of the results and a sensitivity analysis, in the form of a tornado plot, are available.

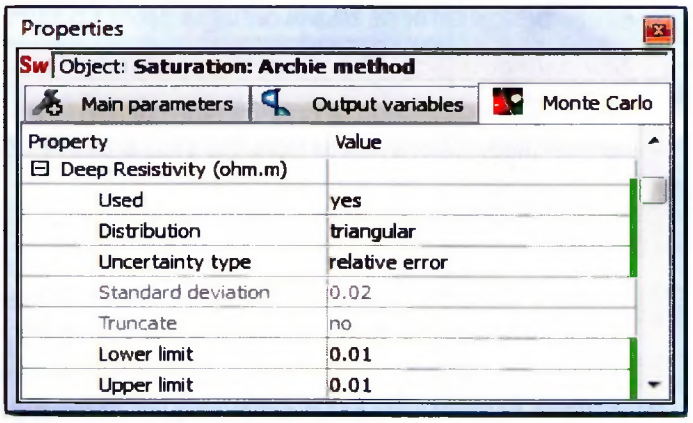
Running a Monte Carlo Analysis

To perform a Monte Carlo analysis:

1. Select the *Monte Carlo* option.
2. Press **F4** in the **Properties** window and click the **Monte Carlo** tab.

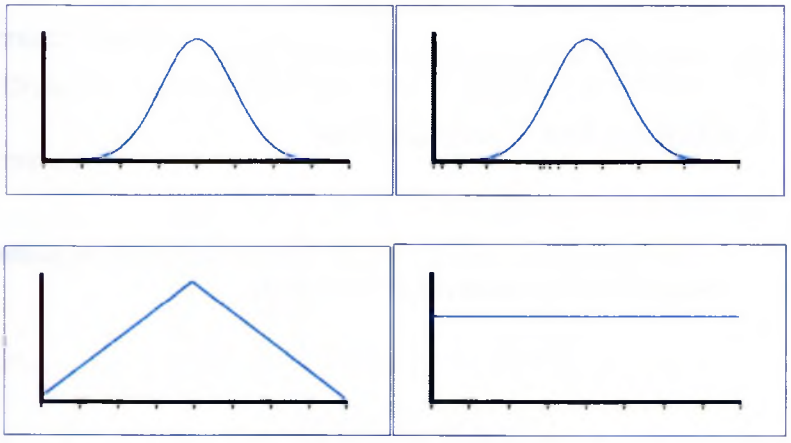


3. In the *Number of iterations* field, specify the number of times the calculation will be performed. By default, all input curves and parameters are used in the calculation.
4. Assign and verify the accuracy of the properties listed in the **Properties** window.



Monte Carlo properties include the following options:

- *Used:* Is this input going to be used in the Monte Carlo calculation? Select yes or no; the default is **yes**.
- *Distribution:* Choose among several modes of distribution. An example of each option is shown in the figures that follow.
  - *Normal* (upper left): Gaussian distribution over a linear scale
  - *Logarithmic normal* (upper right): Gaussian distribution on a logarithmic scale
  - *Triangle* (lower left): Triangular distribution
  - *Uniform distribution* (lower right)



- Uncertainty type
  - *Absolute error*: The magnitude of the difference between the exact value and the approximation.

- *Relative error*: The absolute error divided by the magnitude of the exact value (between 0 and 1).

For example, from a given value of 50:

- An *absolute* error of 10 will result in 40 or 60.
- A *relative* error of 10 will result in 45 or 55 (10%).

- *Standard deviation:* The value of one standard deviation

  This property is available only for Normal and Logarithmic normal distributions. It controls the shape of the bell.

- *Truncate:* Allows you to limit the data distribution to specific values.
- *Lower limit* and *Upper limit*: The maximum and minimum range between the exact value (original data point) and its limits.

Refer to the help files for practical examples of each property.

Activating the Histogram

To activate the histogram:

1. In the **Properties** window on the **Monte Carlo** tab, activate the property **Output distribution > Compute**.
2. Enter a value in the *Minimum* and *Maximum* properties to display the histogram.

   If no value is entered (-9999), each histogram displays with the minimum and maximum values of the curve.
3. Activate the **Compute** option to create a histogram for each output curve.
4. In the **Output variables** tab, you can de-activate the histogram display for each curve.

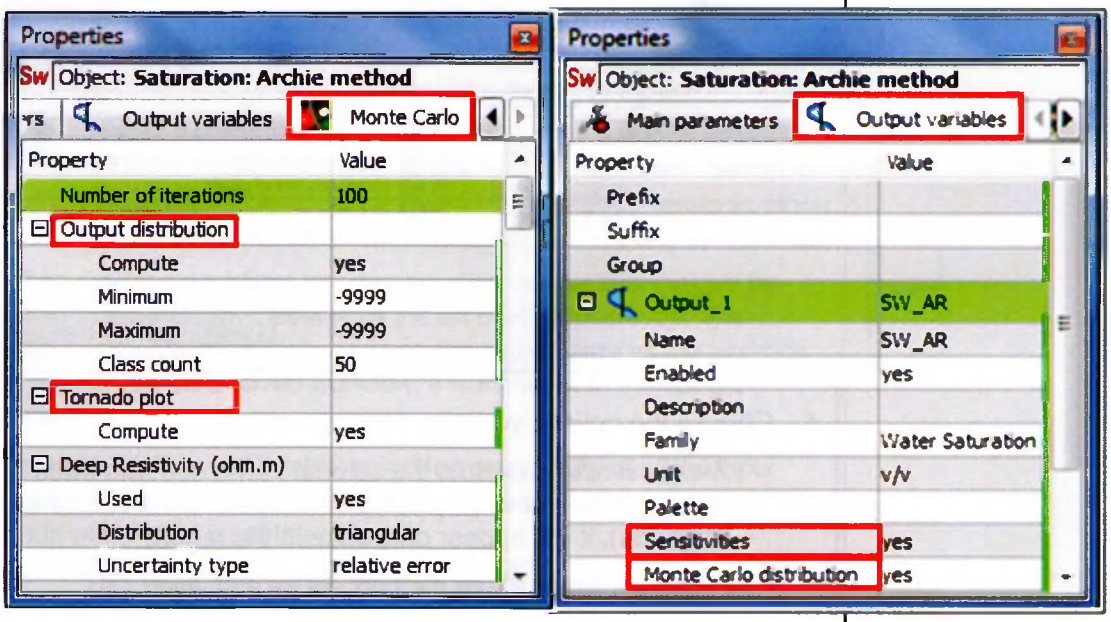
The *Class count* property is the number of bins.

Activating the Tornado Plot

To activate the Tornado plot:

1. In the **Properties** window on the **Monte Carlo** tab, activate the property **Tornado plot > Compute**.

2. Activate the *Sensitivities* property to choose a curve to be used on the **Output variables** tab. Each curve will have its own Tornado plot.



Lesson 8 Reports

Reporting from **Quanti** provides information about how the output curves were calculated, which input curves were used, the depth intervals, and the values of the parameters for each interval. It is a tabular display of the following tabs: **Input**, **Zonation**, **Parameters**.

If a method includes additional tabs in the workflow, such as **HC correction** for a porosity method, this information is also included.

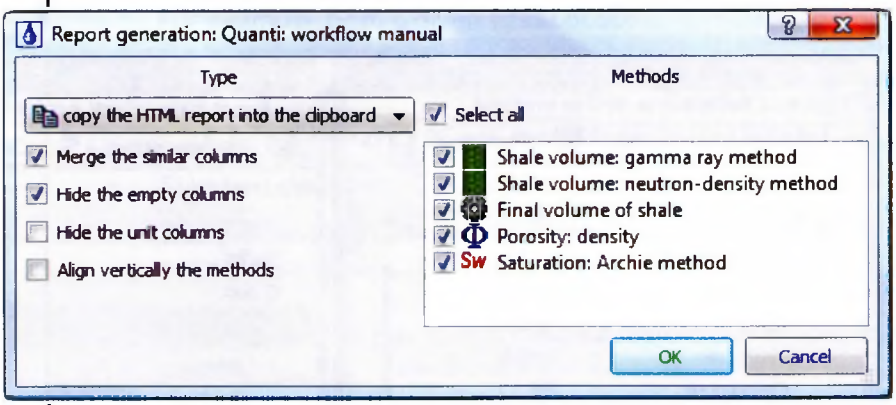
Creating a Report

To create a report:

1. Go to the **Save/load** tab and click **Generate a workflow report** .

2. In the window that opens, choose the methods you wish to include in the report. By default, all methods within the workflow are included.

3. Choose a format for the report from the drop-down menu: HTML, CSV, PDF, or well property.



4. Choose the options you wish to apply and click **OK**.
   - *Merge similar columns:* If a column name appears twice in the workflow (for example, the same parameter in different methods), it will appear only once in the report.

   The input curves for each method will be displayed.
   - *Hide the empty columns:* A column with only missing values (-9999) will not be displayed in the report.
   - *Hide the unit columns:* This hides the parameter units in the generated report.
   - *Align vertically the methods:* Each method will be reported separately in the same document.
5. If the output format is a CSV or PDF, specify the location where the file should be saved.
6. If only a specific table is needed, click **Parameters copy** to copy the table to the clipboard.

Lesson 9 Workflow Templates and Parameters

This lesson explains the distinction in **TechLog** between the *workflow* and the *parameters* used in the workflow.

A *workflow* contains a list of wells and zone methods and their order of appearance in the workflow. Also it includes all the features you defined in the properties of all the methods, such as the names of output curves, **Monte Carlo**, **Automatic Launch Cascade** mode, and more.

Parameters are values in a table of constants used within the workflow. You can save both the workflow and the parameters using the **Save/Load** tab, but you have two options when saving a workflow:

Save the workflow:

Save the workflow as a template

NOTE: The second option will not save the list of wells.

When saving a workflow, you can save parameter files that will be linked to the workflow, as well as the list of all input curves. The associated parameter files are given the same name as the workflow; when you launch a saved workflow, it launches with the correct parameter files.

The option **Keep input table selection** saves a list of all the input curves. Even if the curve does not exist in the database, the workflow loads with the name of the curve.

TIP: This is an alternative solution for **Cascade** mode.

Methods for Saving Parameters

There are several methods for saving parameters.

Zonation Dataset Method

The quickest way to save parameters is to click **Save the parameters in the zonation dataset**  . When using this option, the parameters table is saved in the zonation dataset for each well in the workflow. The names of new variables will be identical to the names of the parameters.

To load parameters into another workflow, click **Load the parameters from the zonation dataset**  .

NOTE: Because the parameters are saved in the zonation dataset, they are attached to this set of zones.

When loading parameters from a zonation dataset, the application compares the names of the curves with the names of the parameters. If your zonation dataset contains a curve with the correct parameter values, verify that the names match.

For example, in the **Shale volume from gamma ray** method, the names of parameters are **GR_matrix** and **GR_shale**. If your curve names are **GR_ma** and **GR_sh**, they will not load.

To load the curves, rename the curves or change the name of the parameter within the workflow. The first option is a global solution, while the second option is a specific solution for a workflow.

Parameter Manager Method

Another method is to define, save, and load parameters of various types using the **Parameter manager** interface. **Techlog** distinguishes between several types of parameters:

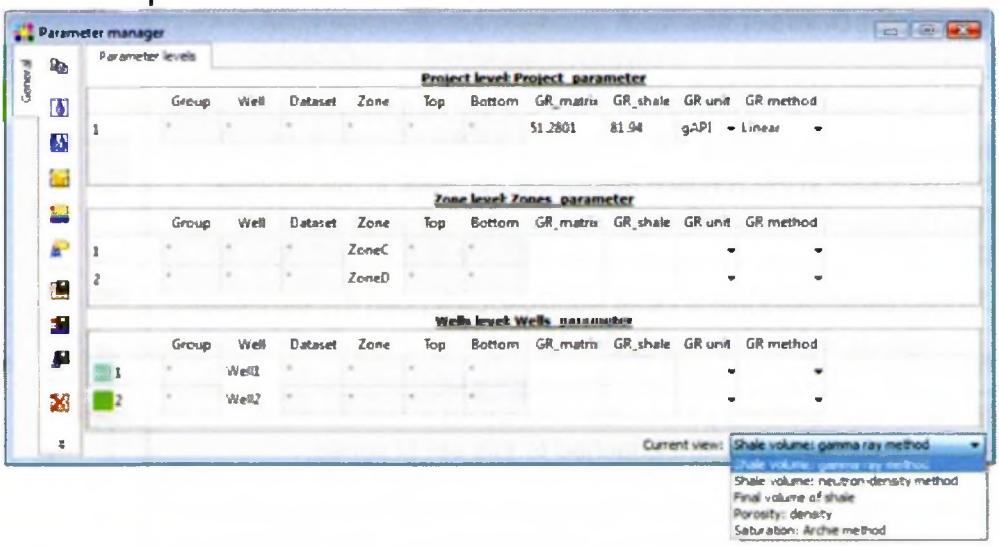
- *Workflow parameters* – Unless it is defined otherwise, a parameter is used as a workflow parameter.
- *Project parameters* – An identical value for the entire project, such as all wells in all zones.
- *Well parameters* – A unique value for the well throughout all zones.
- *Zone parameters* – An identical value to the zone throughout all wells.

You can view workflow parameters within the workflow for each method, while other parameter types can be viewed using the **Parameter manager** interface.

Launching the Parameter Manager Interface

To launch the **Parameter Manager** interface:

1. Go to the **Tools** tab and click . A window like the one shown below opens.



The window is divided to three tables: Project level, Zone level, and Wells level. In the previous example, you used two wells - Well1 and Well2 - with two zones - ZoneC and ZoneD. The workflow in the example comprises all the methods that were used within the workflow.

The **Parameter manager** displays only one method at a time, but it is linked to the **Quanti** interface. This means that if you change a method within **Quanti**, the **Parameter interface** changes to display the current method.

2. If a parameter should be a special parameter and not a workflow parameter, you must declare it accordingly. When you save the parameters, each parameter type is saved in a different file.

Defining Parameters

You can define parameters in any of three ways.

To define parameters:

1. Click on a parameter.
2. From the **Tools** tab, click one of the following:

Define as a project parameter

Define as a zone parameter

Define as a well parameter

The **Parameter manager** is updated automatically based on your choice.

3. Highlight the value you wish to define as a parameter.
4. Right-click and select **Define as project/zone/well parameter**.
5. In the **Parameter manager**, go to the appropriate method and fill in the value for the parameter.

Saving Parameters

There are two ways you can save parameters: with the workflow and using the **Parameter manager**.

Saving Parameters with the Workflow

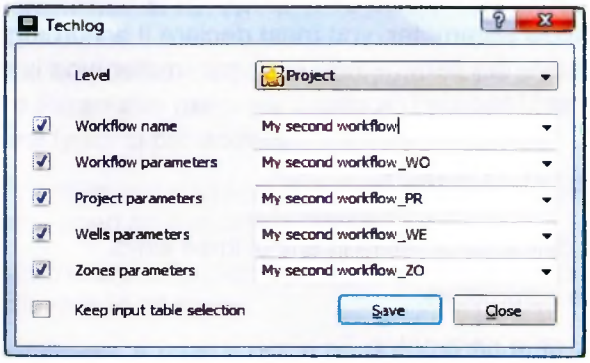
When you save a workflow, you can also save the associated parameters.

To save parameters with the workflow:

1. Click the **Save/Load** tab.
2. Save the workflow. The **Techlog** window opens.

TIP: Naming is very important! Remember that when you name a workflow, the associated parameter files are given the same name.

When you launch a saved workflow, the workflow launches with the correct parameters if the parameter files have the same name as the workflow. Refer to the figure that follows.



Parameters and the workflow are saved in separate files.

- The *Workflow* file contains the methods, their order, and their input families.
- The *Project parameters* file contains parameters defined as project parameters.
- The *Wells parameters* file contains parameters defined as wells parameters.
- The *Zones parameters* file contains parameters defined as zones parameters.
- The *Workflow parameter* file contains all the other parameters.

You can save files in the *User* folder, the *Company* folder or the *Project* folder. If you save them in the *Project* folder, they display in the **Project browser**.

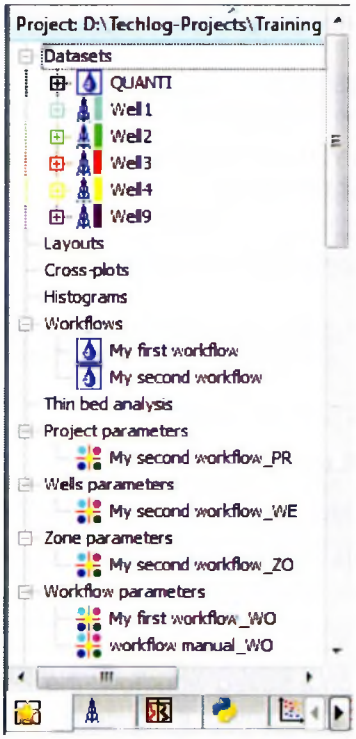


Figure 3 **Parameters and workflow in the Project browser**

Saving Parameters using the Parameter Manager

After defining the parameters, you populate the values in the **Parameter manager**. You can save parameters in three ways using the **Parameter manager**:


Save the project parameters


Save the wells parameters


Save the zone parameters

Recall that *workflow* parameters are saved directly on the **Save/Load** tab.

Loading Parameters from a Parameter File

Parameter files are loaded into a workflow in two steps: load the file to the **Parameter manager** and then propagate parameters to the workflow.

To load parameters in a workflow:

1. Take one of these actions:


Load a project parameter file


Load a zone parameter file


Load a wells parameter file

2. After all parameters are loaded to the **Parameter manager**, click  to propagate them to the workflow.
3. Propagate the parameters one at a time.

Removing Parameters

To remove some types of parameters, choose one of the following actions:


Remove project parameters


Remove zone parameters


Remove Wells parameters

Hierarchy of Parameter Files

When simultaneously propagating several parameter files, the order of priority is Well, Zone, and Project.

- First, propagate project parameters.
- Second, propagate zone parameters, overwriting project parameters if duplicates exist.
- Third, propagate well parameters.

For example, the **RHOB_fluid** parameter is defined once as a *zone* parameter 1.05 [g/cm3] and again as a *well* parameter 1.08 [g/cm3].

If the parameter files are loaded and propagated *one-by-one* and parameters are deleted in the **Parameter manager** between each propagation, the final value is the one from the file that that was propagated last.

If *both* files are present in the **Parameter manager** during the propagation, the final value will be the one from the *wells* parameter file.

Lesson 10 Summaries

On the basis of user-defined cutoffs, the summaries algorithm determines the reservoir intervals, their type, quality, and thickness.

Open the **Summaries** window by selecting **Summaries** from the drop-down menu in the **Quanti** window. The **Family and Alias** panel opens.

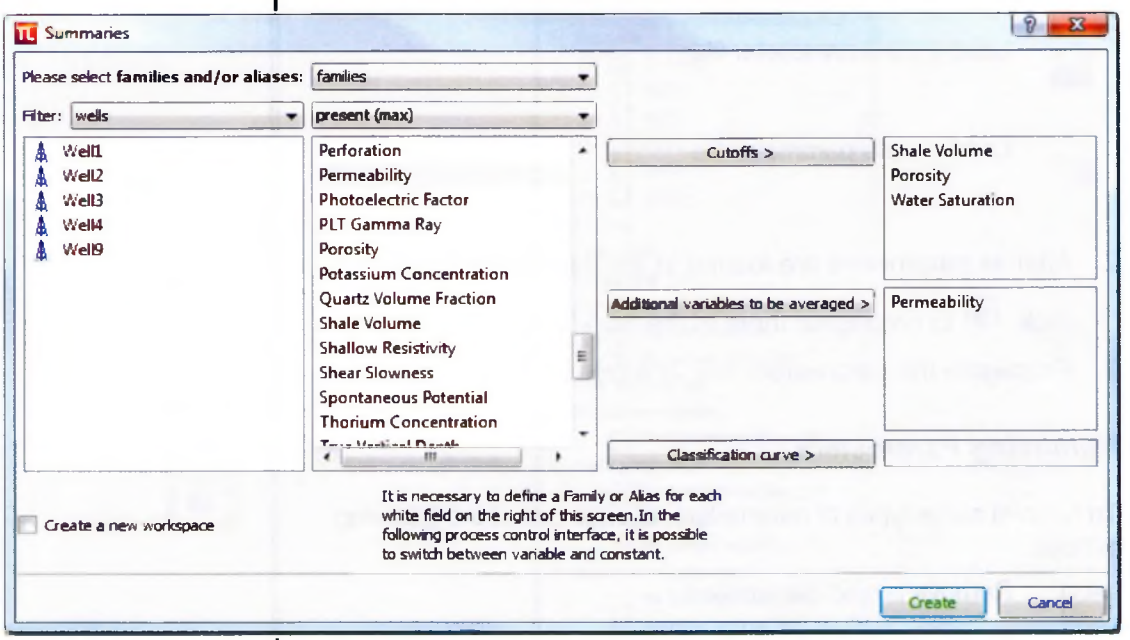


Figure 4 Summaries window

All the inputs are optional, but remember these interactions and results:

- *Cutoffs list:* These are curves to be used as cutoff for the module. If several curves carry the same Family/Alias, such as two porosity curves, insert the Family/Alias twice.
- *Additional curves to be averaged*: For each interval, a mean value is calculated for these curves. You can define what type of mean to calculate: arithmetic, geometric, harmonic, and others.

  No weighting is done in this computation. If you need only an average per zone, this is the place to put your data.
- *Classification curve*: If a classification curve is *not* entered, the algorithm computes an average value per zone.

  If a classification curve *is* entered, the algorithm calculates the results per zone and per group. For example, calculating an average for one zone with a water leg and an oil leg results in separate average values for each leg.

Tips and Tricks

When inserting variables to the **Cutoff** tab, an automatic detection of curves is activated.

- The first family name containing the word *Volume* in its name is assigned as a shale volume curve.
- The first family name containing the word *Porosity* in its name is assigned as a porosity curve.
- The first family name containing the word *Saturation* in the name is assigned as a water saturation curve.
- If the family contains the word *Core* in its name, depth weighting is disabled.
- If the family name contains the word *Hydrocarbon*, the property **Hydrocarbon saturation** is automatically toggled to **yes**.

Click **Create** to add **Summaries** to **My second workflow** window.

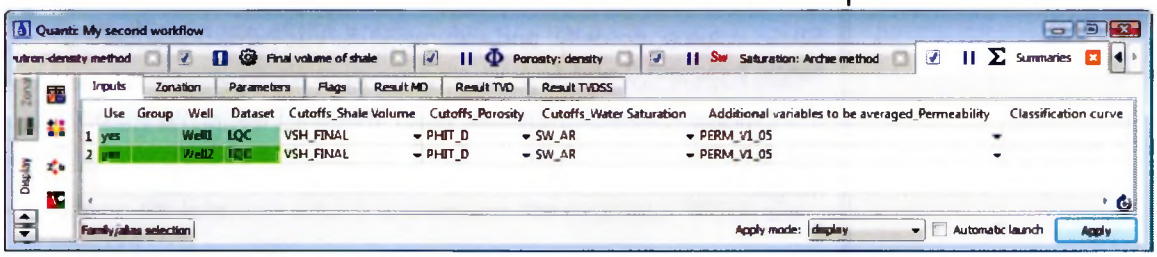


Figure 5 Options in the Quanti window

The **Quanti** window contains additional tabs and curves: **Flag**, **Parameters**, and **Results**.

Flag Tab

In this tab, you determine which cutoff curve will be used to define the output flags.

By default, **ROCK**, **RES** and **PAY** are associated with the shale volume, porosity and saturation curves, respectively.

- *ROCK*: Defines the shale-free depth intervals. By default, only the shale volume cutoff is active.
- *RES*: Defines the possible reservoir intervals within the ROCK intervals. By default, the shale volume and the porosity cutoffs are active.
- *PAY*: Defines the hydrocarbon impregnated reservoir intervals. By default, the shale volume, porosity, and water saturation are taken into account.

You can modify all the information in the table by double clicking on any cell. You also can add or remove a flag using the right-click context menu.

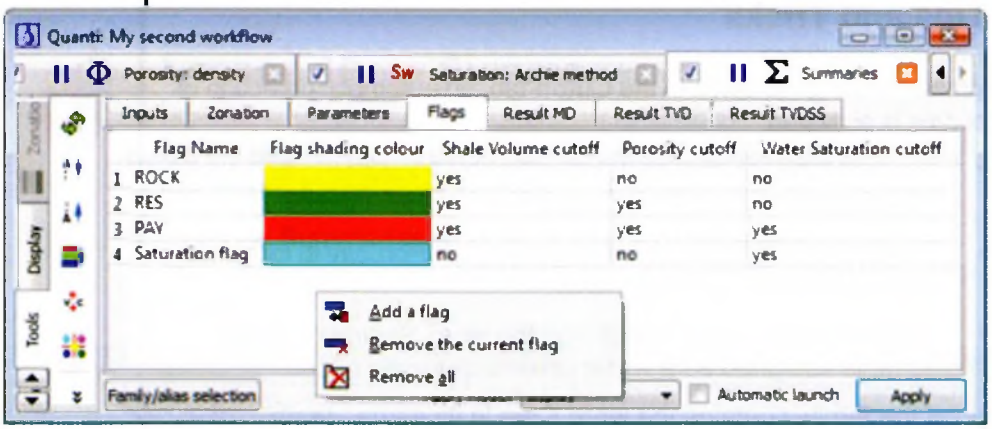


Figure 6 Adding or removing a flag

Parameters Tab

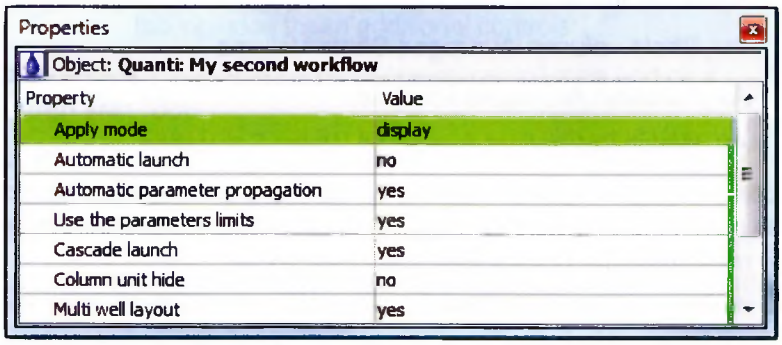
In this tab, you must establish the cutoffs for each curve. By default, each curve has a predefined range.

Shale volume, porosity, and saturation curves have a predefined range between 0 and 1.

If additional curves/families are used, the limits for the range of their display are extracted from the family database, but you can modify the range.

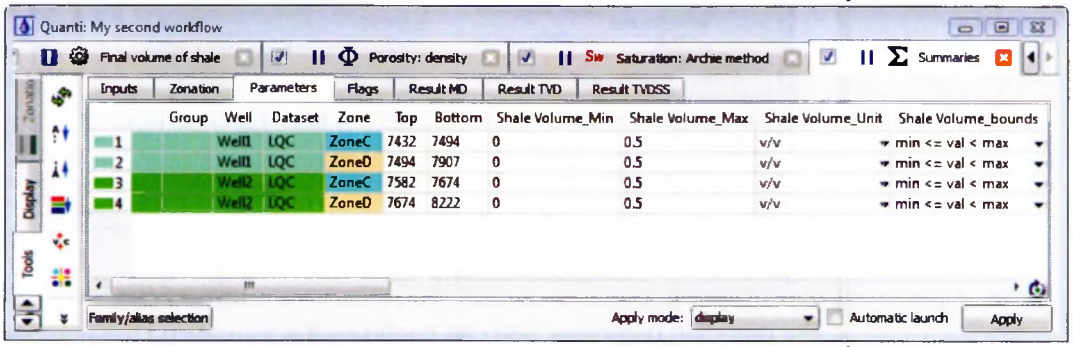
To apply a cutoff value that is outside the predefined limits:

1. Click on the **Workflow** window.
2. In the properties of the workflow, set the property *Use the parameters limits* to **No**. This applies to any parameter within the workflow.



3. You can modify the limitations in the predefined drop-down list.

   By default, the range is between the minimum and maximum cutoff values and includes the actual minimum value: minimum cutoff <= range of values < maximum cutoff



4. After defining the parameters, click **Apply**.

Results Tab

The **Results** tab summarizes the following measurements:

- *Gross*:Thickness of the zone.
- *Net:* Thickness of the interval defined by the cutoff values and the different flags that were used in the analysis.
- *Not Net:* Gross - net.
- *Unknown*: Intervals where null values (for example -9999) are present.
- *Net/Gross*: The ratio of net-to-gross.
- *BVW*: bulk volume of water. Equation:
  BVW = net * average porosity * average water saturation

- *POR-TH*: Porosity thickness. Equation: POR-TH = net * average porosity
- *HCPOR-TH*: hydrocarbon * Porosity thickness for each selected zone.
- *Equation*: HCPOR-TH = net * average porosity * average Hydrocarbon saturation
- *Av_Shale volume*: Averaged Shale Volume
- *Av_Porosity*: Averaged Porosity
- *Av_Water Saturation*: Averaged Water Saturation

These average values are calculated using the arithmetic method. If you need another type of average, you can define it in the properties of the method on the **Computation and report** tab.

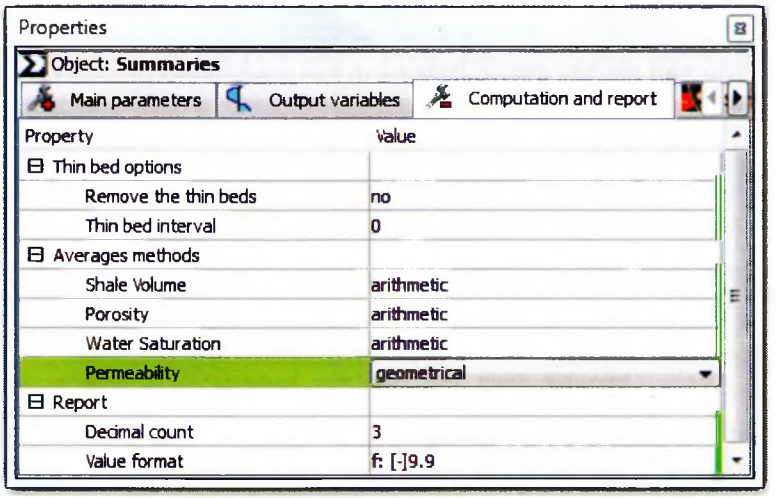


Figure 7 Computation and report tab in Properties

For reporting purposes, you can configure the format and the number of significant figures for the results.

If the **Index** dataset contains additional reference curves, the summaries are calculated for these curves as well. You can control this calculation in the properties of the method. You also can control the unit for calculation and the reference for the layout.

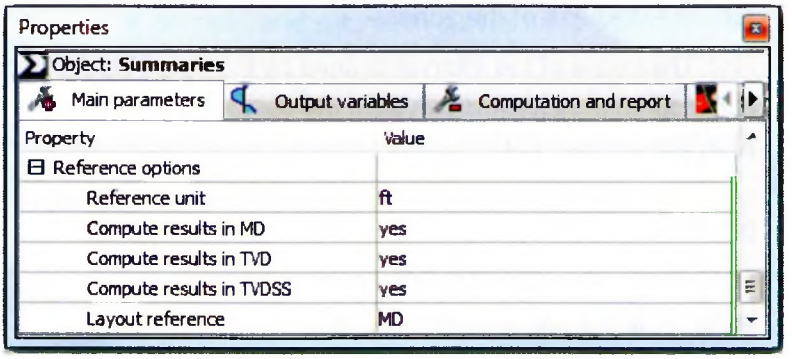


Figure 8 Main parameters tab in Properties

The results are displayed *per zone*. If you wish to see the results table *per bed*, click **Change the display zones/beds type** on the **Display** tab.

If the table contains too much information, you can filter it as needed by clicking on the name of the parameter tables.

The **Results** tab includes these additional controls:

Hide columns: Click on a name of one of the columns and click **Column selection**.

Sort the table according to a certain value: Click on the name of a column. In the window that displays, choose the options to sort: ALL, 10 first vales, and more.

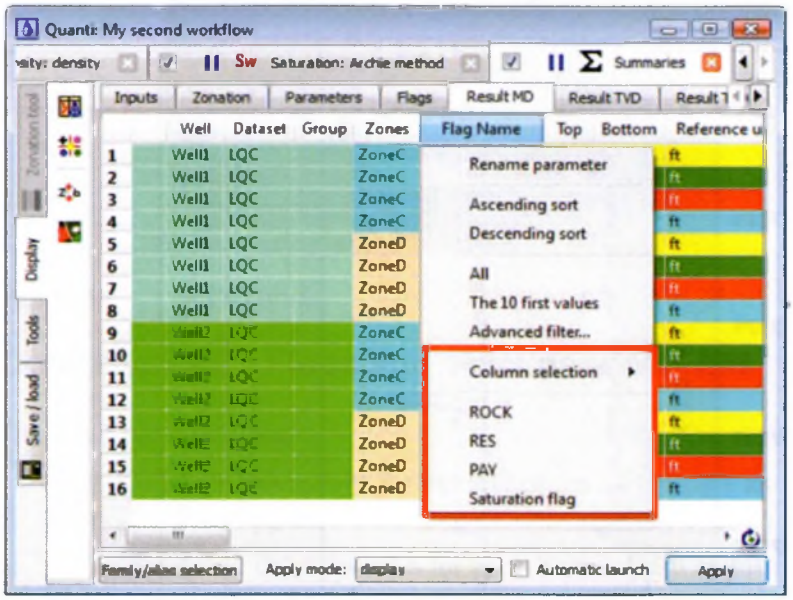


Figure 9 Column selection option

The following example uses **Flag Name**, hence the option to filter the table on the different flags. In this instance, the filter is set on the **PAY** results. In addition, use the **Column selection** option to choose the well name, dataset name, zone, flag, and average values per zone.

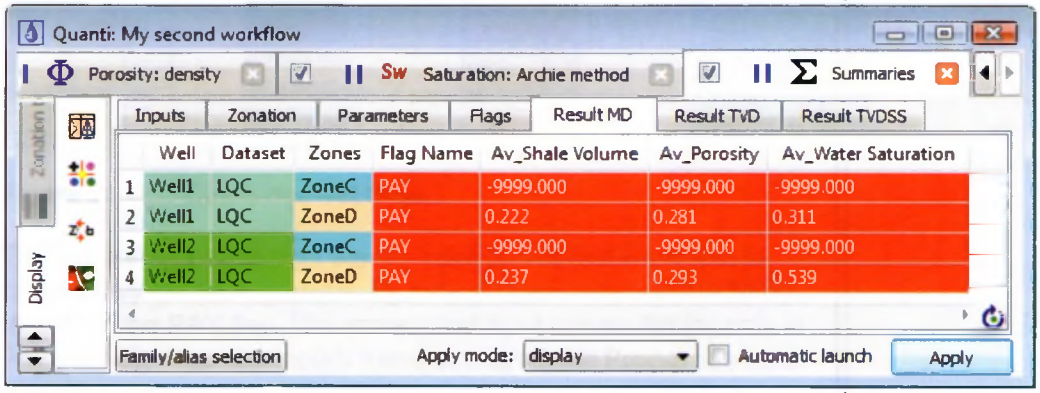


Figure 10 Filtering on PAY results

Figure 11 shows a layout with the result curves.

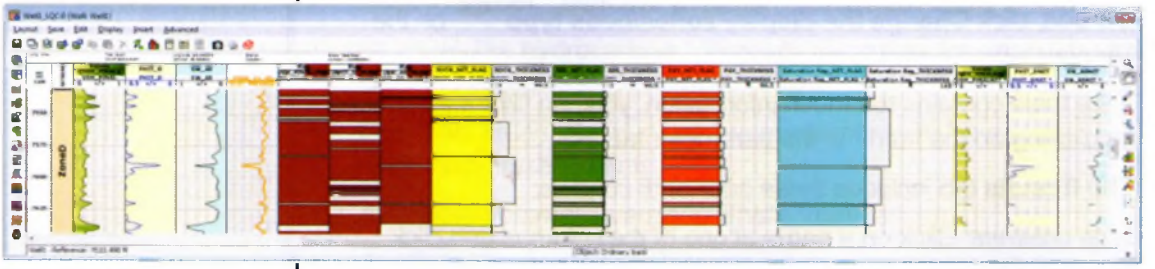


Figure 11 Curves layout

Over the layout, the default output curves are (from left to right) flag curves, combined net flags and thickness results, and input curves containing only NET intervals.

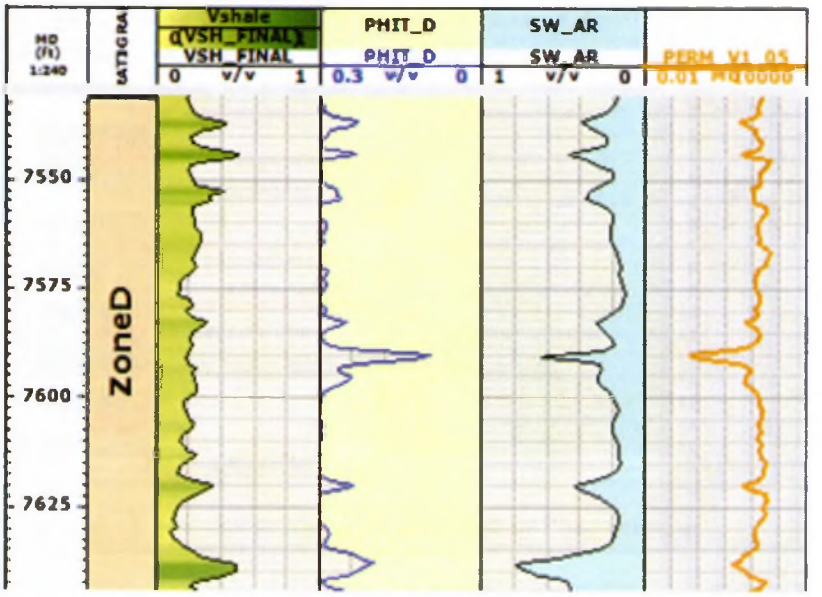


Figure 12 Ensemble of the input curves

Flag Curves

Each input curve used as a cutoff also will have a corresponding flag: *_FLAG*. The flag shows the intervals in which the value of the input is within the range defined in the **Parameters** tab.

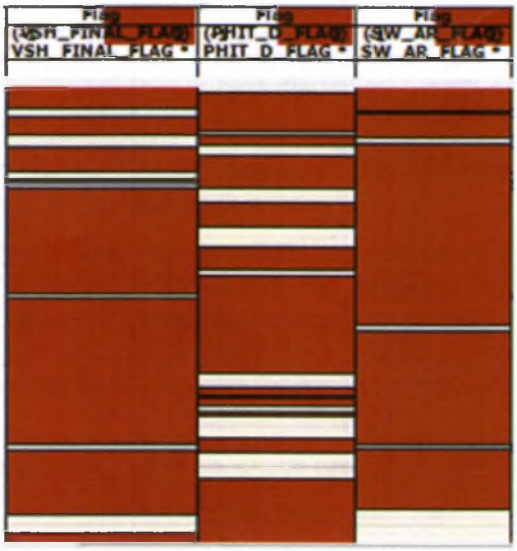


Figure 13 Flag curves

Combined NET Flags and Thickness Results

There are two NET flags:

_NET_FLAG: The results for the any of the flag curves as defined in the **Flag** tab.

_THICKNESS: The thickness for each bed defined by the associated ***_NET_FLAG** variable.

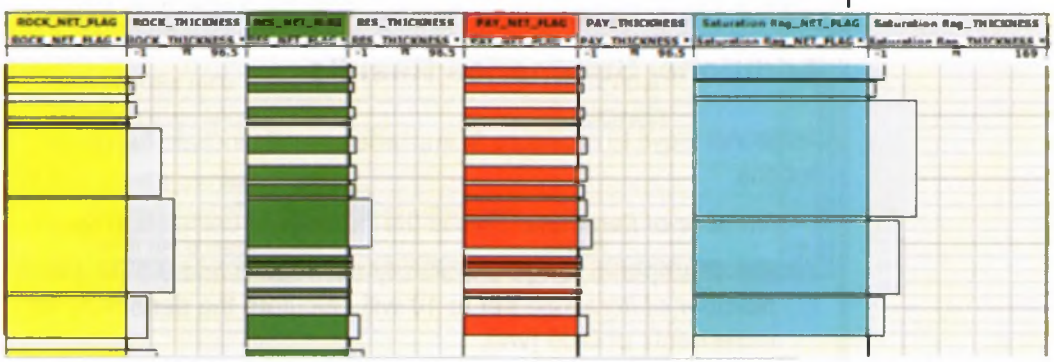


Figure 14 Combined Net flags

The last curves on the right are the input curves that were split to contain only the NET intervals. By default, these intervals are being read from the **PAY** flag. This means that input curves display only in Pay intervals. You can modify these settings in the **Properties** dock window.

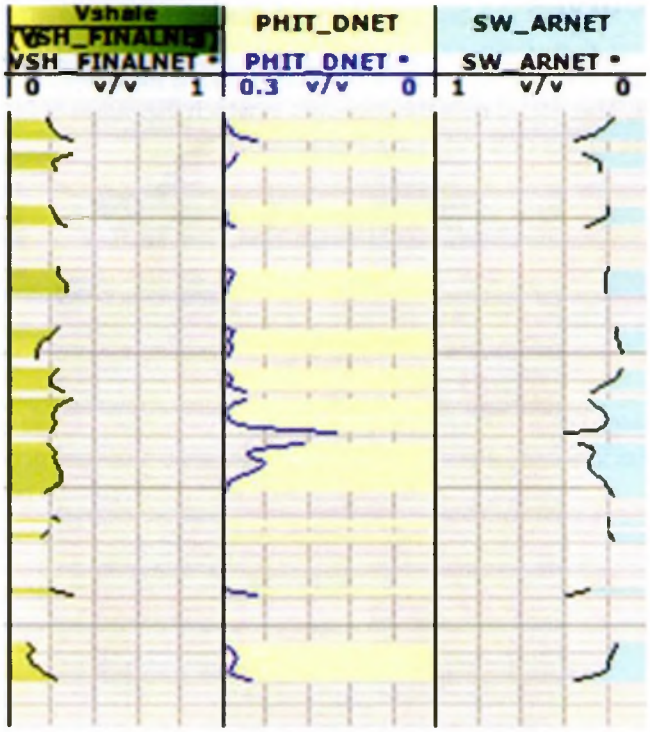


Figure 15 Curves with NET intervals

Sensitivity Analysis

To help with the analysis and selecting parameters, you can launch a sensitivity analysis.

From the **Display** tab, click **Display the sensitivities** . The analyses are done separately for each zone, and the results display in matrix crossplots. The controlling parameters for the analysis are found in the properties of the method, under the **Sensitivity** tab.

Additional Summaries Results

Additional ways to configure **Summaries** results include these options:

- *Number of iterations:* This is the number of calculation steps.

  For example, a range of porosity between 0 and 0.5 [v/v] with *Number of iterations* set to **10** will calculate the sensitivity for increments of 0.05 [v/v].

- *Reference used:* Because the summaries algorithm can be launched using several references simultaneously (MD, TVD, TVDSS), it is important to specify which reference to use for the sensitivity analysis. By default, the measured depth reference is used.

- The analysis can be launched separately for each input curve by deactivating all other curves.

If several cutoffs are activated, the results are *not* independent. This means that the sensitivity of **PAY** depends on the results of the **RES**. In turn, **RES** depends on the results of **ROCK**.

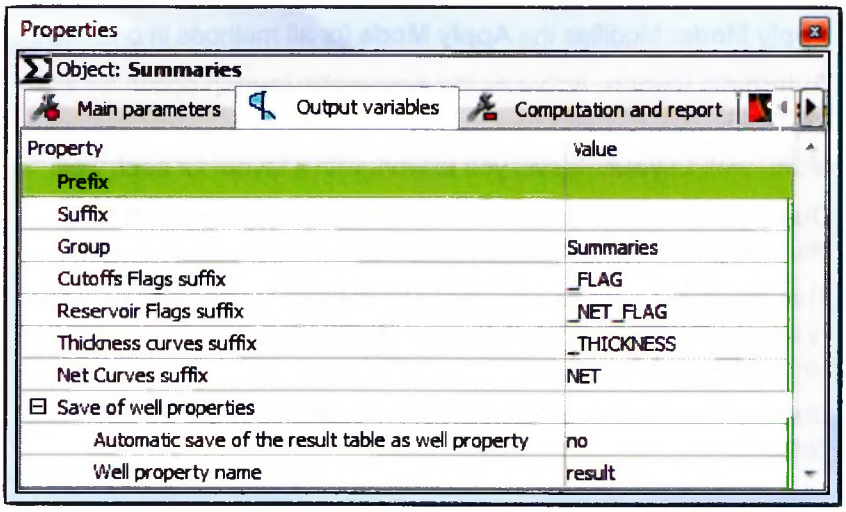
Saving the Summaries Results

1. Select **Apply Mode > Save and display**.
2. Click **Apply**. The output curves displayed over the layout are saved in the **Project browser**.
3. Optionally, you can modify the naming conventions of these variables in the **Output** dock window.
4. You automatically can save the results table (**ResultMD**, **ResultTVD**, **ResultTVDSS**) as a well property. This is useful because a table is always associated with a well when moving wells between projects. Because it is a property, it can be inserted in the header for reporting proposes.

   To save the results table as a well property, click  .

   As with other methods that use the **Save/Load** tab, you can save parameters (from the **Parameters** tab) and the results tables, together with the zones to which they refer.

   The figure on the facing page shos the summaries results.



Lesson 11 Additional Tips

Some additional options offer ways to accelerate a process or task.

Options for Specific Events

These are some of the options for working on specific events.

Reload data from the Project browser after opening the workflow: If a workflow is already open when you create a new curve, the new curve will not display as an optional input in the workflow. To add this curve to other input curves, use the right-click contextual menu.

Refresh (keyboard shortcut **F5**): Reload all new curves with the same Family/Alias.

Refresh and select the first choice (keyboard shortcut **F6**): Reload all new curves and choose the first Alias (*Alias* mode) or the first family in alphabetical order (*Family* mode).

Duplicate the workflow: Duplicate the current workflow in a new workspace. This is useful for creating 'what if' scenarios. Click the **Save/Load** button to save the duplicate workflow.

Global Options

Several options are global and can be found in the properties of the workflow. These options affect ALL methods in the workflow.

Apply Mode: Modifies the **Apply Mode** fpr all methods in one click.

Automatic launch: Activates the **Automatic launch** option for all methods.

Multi- well Layout: Allows you to work with a layout for each well.

Display/Layout: Allows you to choose a preferred template for displaying results. By default, this option is set to *current*.

This option generates a new layout for the first method and continues to display the input and output curves for the other methods over the same layout.

Display/Template type: Launch the selected template for each dataset.

Summary

In this module, you learned about:

- using the **Quanti** interface
- using **Techlog** interpretation tools
- applying best practices when doing conventional log analysis using **Quanti**.

Module 2 Python Script

Python is an open source programming language imbedded within **Techlog**. Although the **Techlog** software interface library and existing modules available from the Internet provide tools that are compatible with **Techlog**, **Python** scripts give you additional resources. The scripts run fast, and there is no need to compile them. In **Techlog**, a few lines of code can help you accomplish complex tasks. Some of the tasks you can do with **Python** scripts in **Techlog** are write your own equations, perform tasks on the database, and create new modules.

Learning Objectives

After completing this training, you will be able to:

- create and launch scripts using the **Python editor**
- understand the basics of programming with the language
- add your own program to a **Quanti** workflow.

Lesson 1 Python Editor

There are five main areas in the **Python editor** window. Refer to the alphabetical designations in Lesson 16 on the following page.

A Toolbar and shortcuts of the actions available in the **Python editor**

B Script description and comments about each script

C **Script Parameter Editor** where inputs, outputs and parameters of the script are defined

D Coding area where the script is written and edited

E Output window displays error messages occurring during the running of a script, or printing commands within the script.

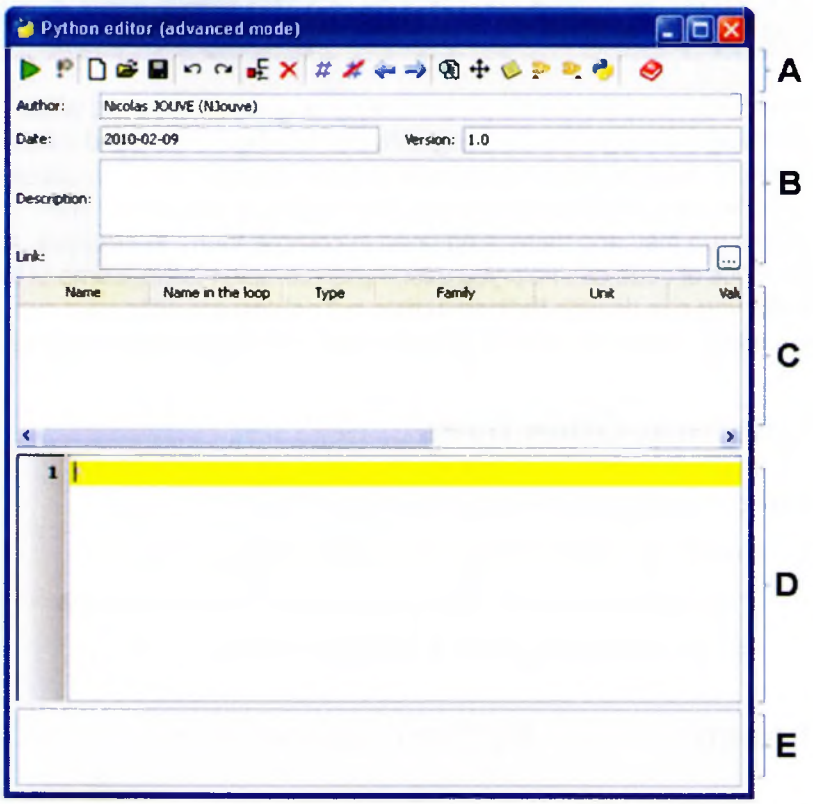


Figure 16 **Python editor main window**

Lesson 2 Scripts

Generate and run a script using the **Python editor**. This script computes the shale volume from the Gamma Ray curve. Lesson 17 shows a section of script in the coding area of the **Python editor**.

```
1  LOOP:
2
3      if gamm >= GR_MA and gamm <= GR_VSH:
4          vsh_py = (gamm-GR_MA)/(GR_VSH-GR_MA)
5
6      if gamm < GR_MA:
7          vsh_py = 0
8
9      if gamm > GR_VSH:
10         vsh_py = 1
11
12 print "end"
13
```

Figure 17 **Script in the Python editor coding area**

Notice that some characters are displayed in blue, pink, red and green.

- Blue indicates reserved words defined in Python.
- Pink indicates Operators.
- Red indicates Numbers.
- Green indicates comments.

There are four important format and syntax rules:

- Indentation defines a block of commands attached to one function.
- A colon ":" is used at the end of the first line of a block of command, for example:

```
LOOP:
if…:
else…:
```

  and so forth.
- The LOOP keyword handles all the input and output for the user. Behind the LOOP, the following operations are performed automatically:
  - Creation of a loop 'for' on the first variable.
  - The content of the variable is transferred into the *Name in the loop* at each iteration of the loop.
  - At the end of the process, the all output curves are saved.
- If a parameter is defined as a variable, three names should be defined:
  - **Name:** Refers to the variable
  - **Name in the loop:** Refers to the value of the variable at one specific reference increment.
  - **Value:** Refers to the name of the variable in the database.

On the top right corner of the **Python** script editor, the name of the well and the dataset are displayed.

Exercise 1 Generating a Script

To generate a script:

1. Select **Python > Python editor**.
2. In the project browser, drag-and-drop the variable **GAMM (Well1>LQC)** in the **Script Parameter Editor**.
3. Click Add a new parameter  or press **Crtl + N**.
4. Define the new parameter with these descriptors:
   - Type: Number

NOTE: Some fields become grayed out when you set this option.

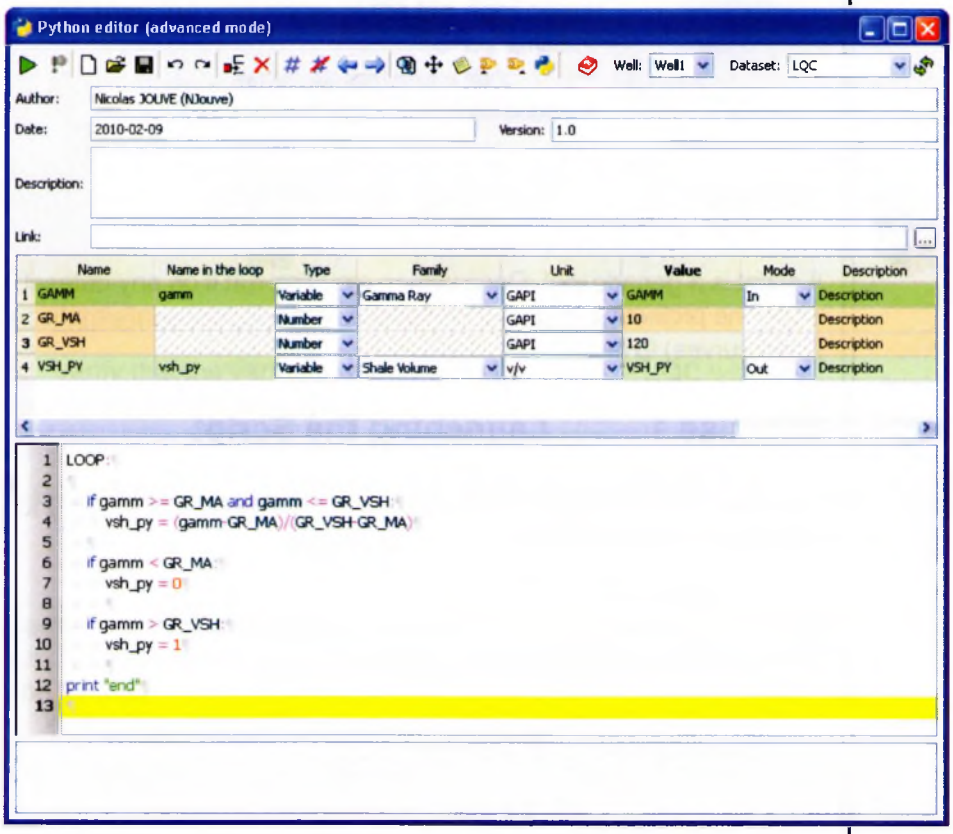
- Name: GR_MA
- Unit: GAPI
- Value: 10

5. Add a second parameter and define it with these descriptors:
   - Type: Number
   - Name: GR_SH
   - Unit: GAPI
   - Value: 120
6. Add a third parameter and define it as follows:
   - Name: VSH
   - Name in the loop: vsh
   - Family: Shale Volume
   - Unit: v/v
   - Value: VSH_PY
   - Mode: Out
7. In the **Coding Area**, type the following code:

```
1  LOOP:
2
3    if gamm >= GR_MA and gamm <= GR_SH:
4      vsh = (gamm - GR_MA)/(GR_SH - GR_MA)
5
6    if gamm < GR_MA:
7      vsh = 0
8
9    if gamm > GR_SH:
10     vsh = 1
11
12 print "end"
13
```

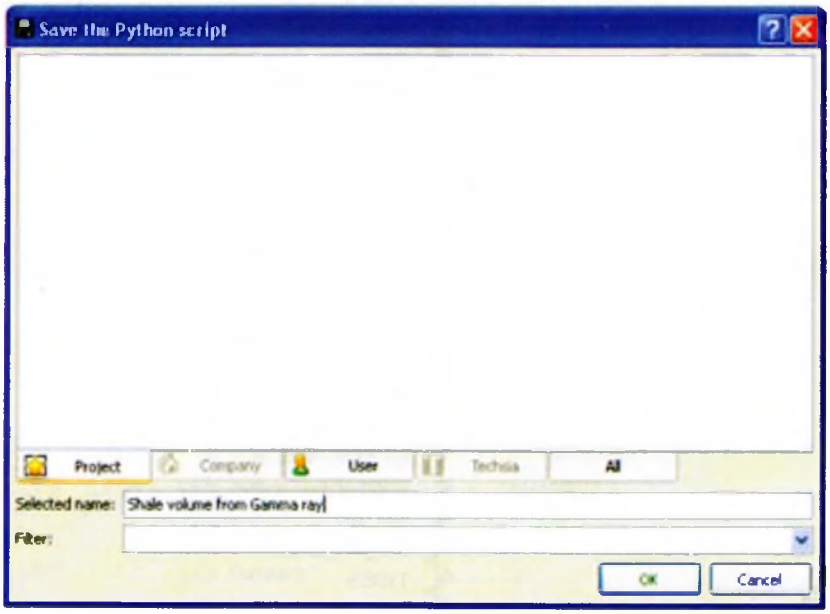
NOTE: Per the syntax rules, the **LOOP** command creates a 'for' statement on the first variable (GAMM).

8. Confirm that your **Python editor** looks like the screenshot below.

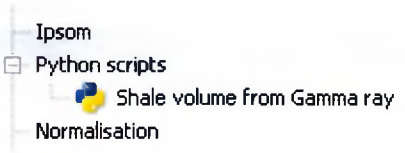


9. Click **Save the script** , or press **Crtl + S** to save the script.

10. Enter *Shale volume from Gamma ray* as the name for the script, and click **OK**.



11. Verify that the script is present in the **Project browser**.



Lesson 3 Launch the Script

A script is a generic tool. Once written, you can run it on any dataset within the project as long as it contains all the necessary information (input curves) to launch it.

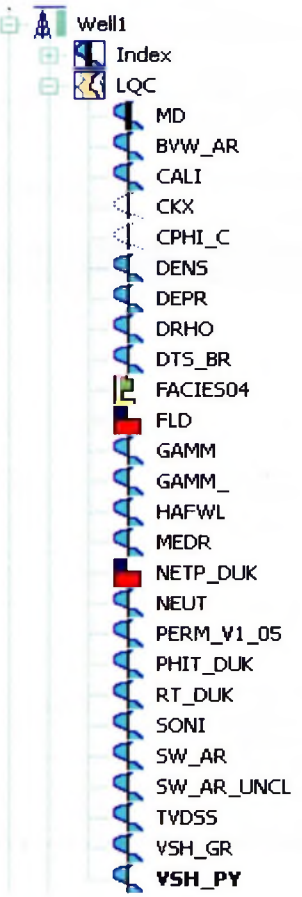
Exercise 1 Launching the Script

This exercise shows you how to launch the first script and to apply it with data from a second well.

To launch and apply a script:

1. Click  or press **F7** to launch the script.

   Notice that the output window displays 'end' and that a new curve is created in the **Well1 > LQC**.



2. Verify that the family and the unit defined for this curve are **Shale Volume** and **v/v**, respectively.
3. Change the well name in the top right corner.

4. Launch the script.
5. Verify that the variable is created in the **Well2 > LQC** with the correct family and unit.

Lesson 4 Open an Existing Python Script

Besides creating new scripts, you can open and use existing **Python** scripts. You may access scripts at all levels: User, Project, Company, and Techsia.

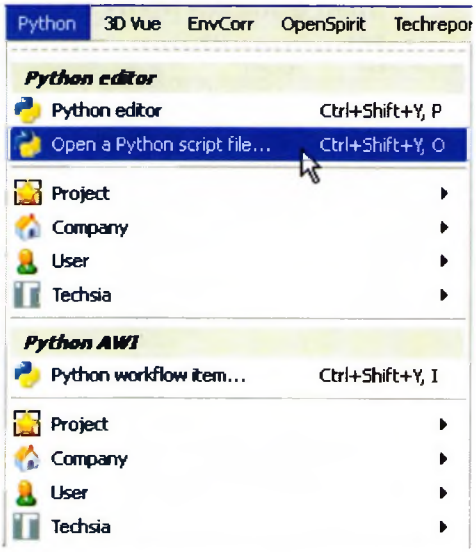
Opening a Script Saved in the Project Level

To open an existing **Python** script, double-click on it in the **Project browser**. The **Python** script displays in a new **Python editor**.

Opening a Script Saved in Other Levels

To configure access to all available scripts saved in all levels (User, Project, Company, and **Techlog**):

1. Select **Python > Open a Python script file**.



A window opens in which you can select a **Python** script. Tabs along the bottom of the window allow you to display scripts available in each level.

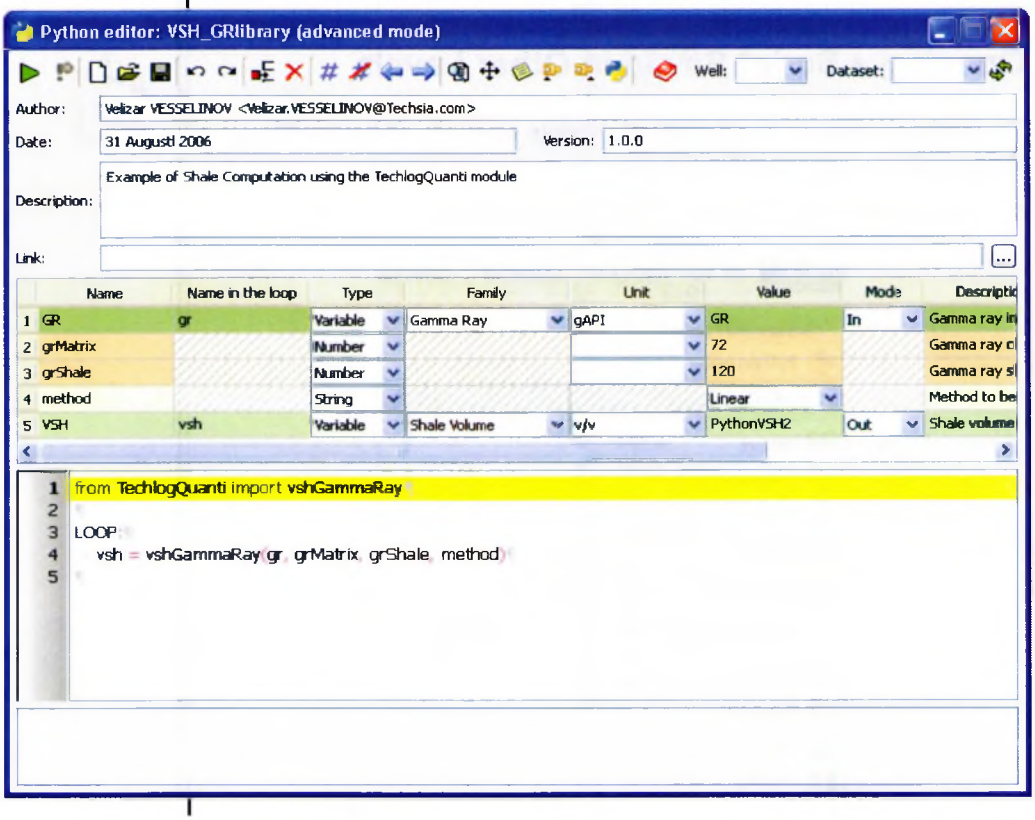
2. Select the script and click **OK**.

Exercise 1 Launching a Script Saved in the Techsia Level

This exercise shows you how to launch a script saved in the **Techsia** level. These scripts are provided with the **Techlog** installation.

To launch the script:

1. Select **Python > Techsia > Quanti > Shale volume > Gamma Ray (library)**.
2. Confirm that your **Python editor** looks like the screenshot below.



3. Drag-and-drop the variable **GAMM** in **Well1 > LQC** over the parameter **GR**. Note that *Value* becomes **GAMM**, and the well and dataset are defined.

   Notice that this script uses a previously defined library. The list of the available libraries is described in the Help file (**F1**).

4. Launch the script.
5. Verify that the curve **PythonVSH2** is correctly created in the **Well1 > LQC**.

Lesson 5 Python Scripts in Workflows

You can use a **Python** script as a stand-alone method, or as part of a workflow. These are the limiting conditions to add a script to a workflow:

- The script has to contain input and output variables.
- The **LOOP** command has to be used at the beginning of the main block of commands (beginning of the script).

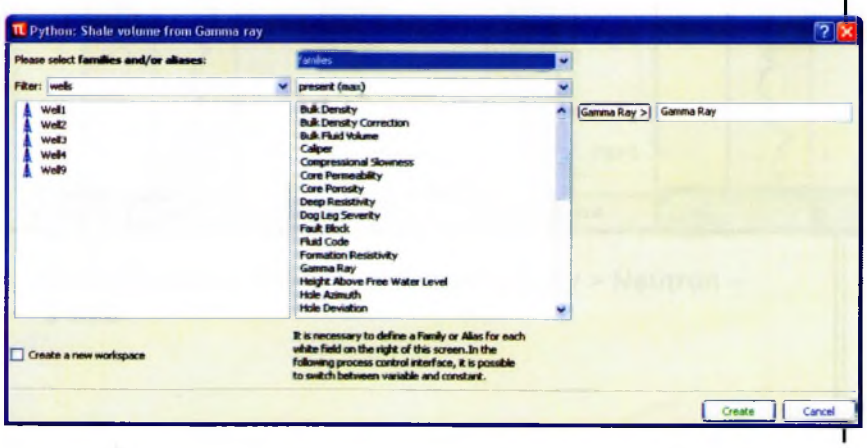
After you add a script to a workflow, it displays in the **Application Workflow Interface** (AWI). The AWI allows you to:

- work in multiple wells
- work with zones
- define parameters per zones
- plot the results curves in **LogView**.
- insert the script into an existing workflow.

Using a Python Script in a Quanti Workflow

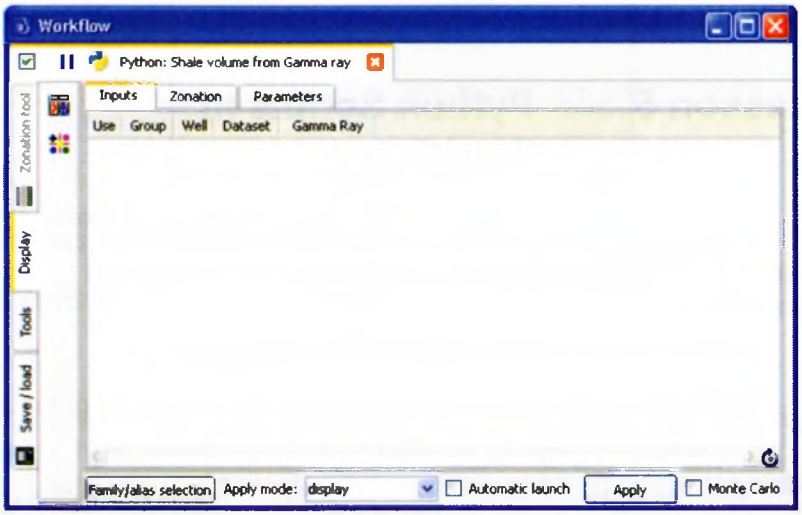
To open a **Python** script in the Application Workflow Interface (AWI):

1. Select **Python > Python workflow item**.
2. Choose the **Python** script and click **OK**. The **Families/ Aliases** selection window opens.



NOTE: For any parameter in the original script that has a variable defined as input, you have to assign a **Family** or **Alias**. This screenshot shows a Gamma Ray curve is being used to launch the script.

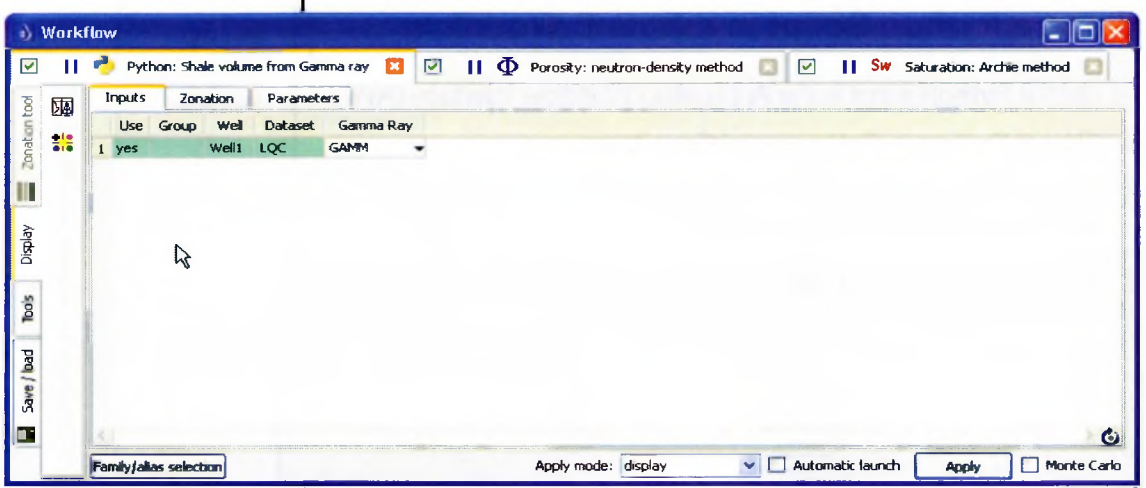
3. Choose the correct **Families** or **Aliases** and click **Create**.
4. Drag-and-drop your datasets into the script display area.



A workflow displays with the name of the **Python** script as the method name.

TIP: You can use all options in the Application Workflow Interface with this method.

If you choose a second **Python** script from the **Python** menu or launch a **Quanti** method, it will be added to the current workflow.

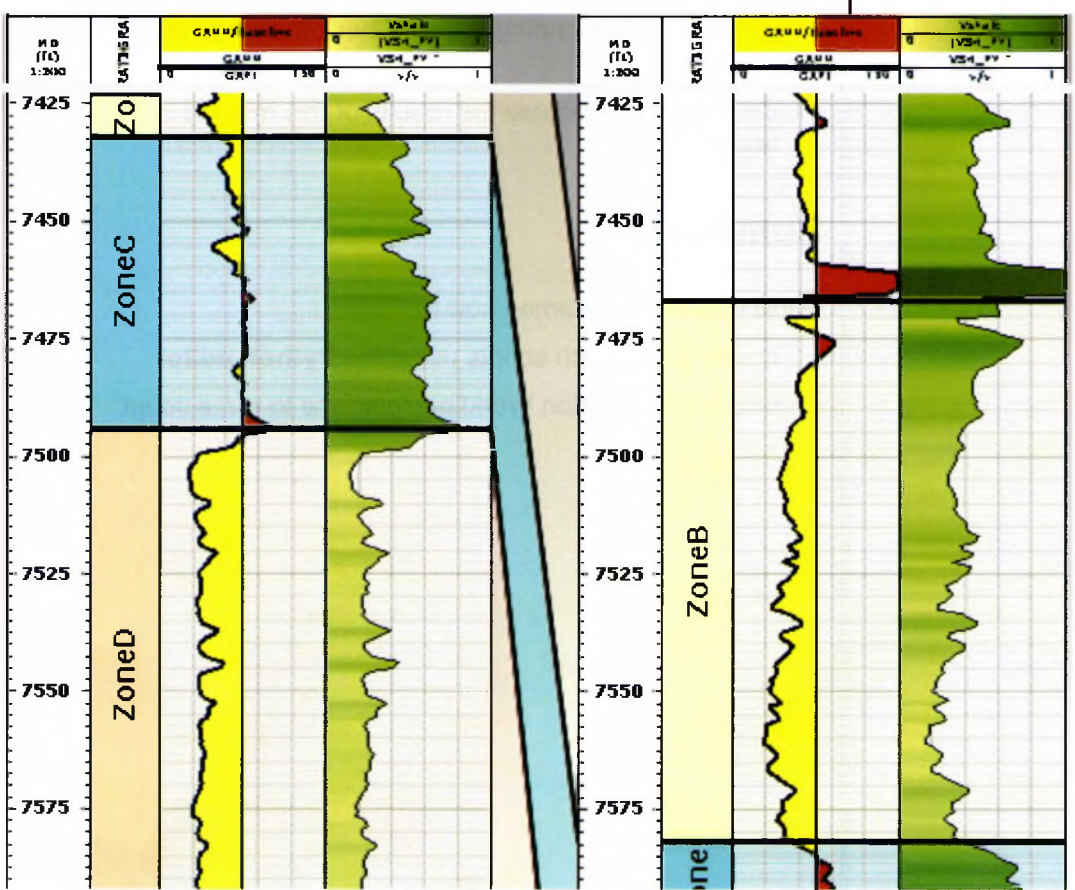


Exercise 1 Launching a Script in a Quanti Workflow

This exercise shows you how to launch a script in a **Quanti** workflow.

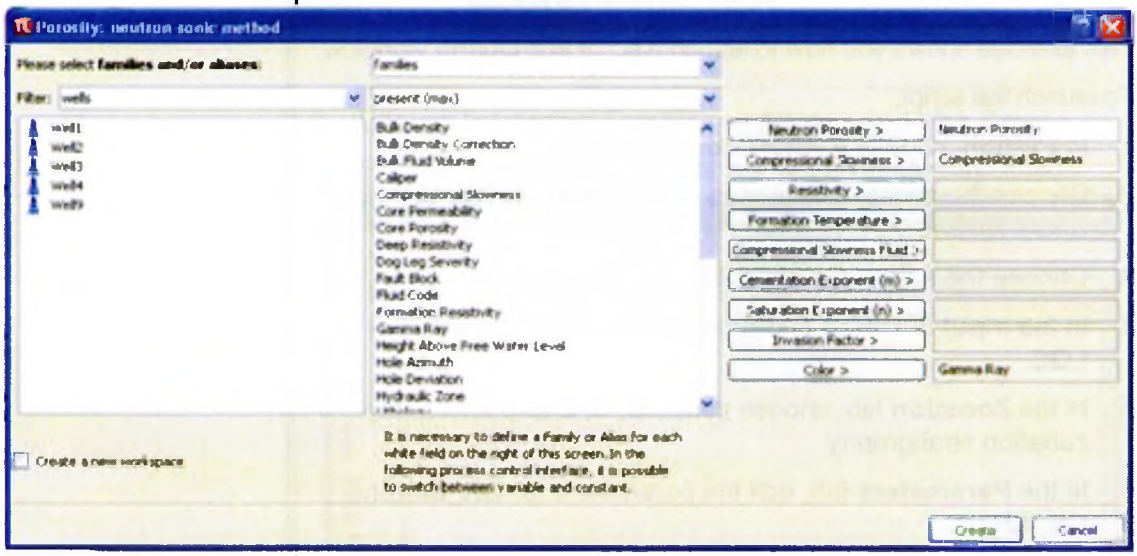
To launch the script:

1. In **Python**, choose **Python workflow item...**.
2. Choose the script that you saved in *Lesson 2, Exercise 1, Shale volume from Gamma Ray*. Click **OK**.
3. Choose the Family **Gamma Ray** and click **Create**.
4. In the **Input** tab, drag-and-drop **Well1 > LQC** and **Well2 > LQC**.
5. In the **Zonation** tab, choose zones **B**, **C**, **D** and **E** from the zonation stratigraphy.
6. In the **Parameters** tab, edit the parameters for **GR_MA** and **GR_SH**.
7. Click **Apply** to display the results in a layout.



8. Select **Quanti > Porosity > Total Porosity > Neutron – Sonic**.

9. Choose the **Families** Neutron Porosity, Compressional Slowness and Gamma Ray. Click **Create**.



10. Notice that this method was added to the current workflow with the corresponding zones. Check the parameters for this method.
11. Click **Apply** to display the results of this method in the same layout.

Summary

In this module, you learned how to:

- create and launch scripts using the **Python editor**
- use the Application Workflow Interface to run a script.

Pre-release copy

Techlog Interactive Suite.

Conventional log analysis.

Workflow/Solutions Training.

Quanti is the ensemble of solutions for deterministic formation evaluation, and includes tools to help you perform precomputations, create flag curves, determine the standard petrophysical properties and calculate summaries.

Quanti uses our Application Workflow Interface (AWI), a generic tool that allows you to work in a multi-well and multi-zone environment, and control your parameters in an efficient manner.

In addition, you will be shown how to program your own equations and applications using Python™. You will also learn how to integrate the scripts that you create into your workflow.