RETINA Simulation 1

User Manual

June. 2017

Table of Contents

List of Tables

List of Figures

1- Starting with RETINA Simulation 1

The startup window of the RETINA Simulation 1 (Figure 1) is displayed after running it.

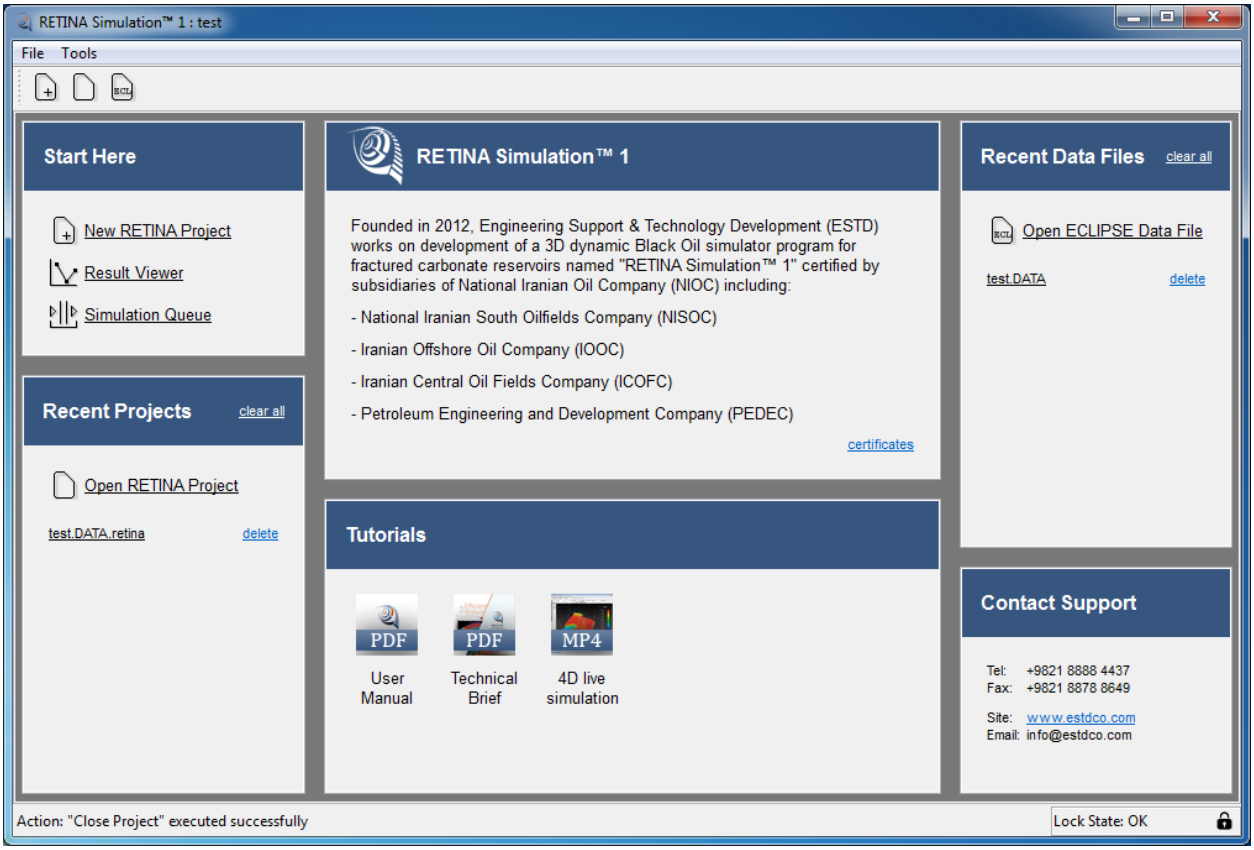


Figure 1 –Starting Window of the RETINA Simulation 1

In addition to a brief introduction about RETINA Simulation 1, certificates, contact support information and tutorials are available here. User can use one of the following options to start working with RETINA:

- Creating a New RETINA Project
- Opening Results
- Loading a Saved Project of the RETINA
- Opening an Eclipse (E100) Data File
- Simulating a Queue of Files

For accessing certificates of Engineering Support & Technology Development Co. (ESTD) information, links are available in “RETINA Simulation 1” and “Contact Support” sections, respectively.

Recently used projects, and opened Eclipse data files as a history list, are displayed in this page, also. User can remove history list either manually using "delete" option or remove all of them by "clear all" button. Clicking on each project or data files in recent items, will open the selected one. User can access to the file directory, by right clicking on each data file in history list.

Note: No need to open a model or project in case of using "Open Results", i.e. user can open any result without loading a model.

***Warning:** The RETINA will be close after 10 min if the status of lock is not "OK ". The 5 definitions of DEFCON_5 to DEFCON_1 exist for lock status, if you don't have proper lock the status will come down from DEFCON_5 to DEFCON_1; afterward, the program will be closed.*

The options of *open project*, *new project*, *open Eclipse project*, and *Exit* options are also available in the "File" tab (Figure 2). As displayed in Figure 3 the "Tools" tab includes the *open results* option. Keyboard shortcuts for each button are also displayed, for example the shortcut of "Ctrl+E" will lead user to open an Eclipse data file.

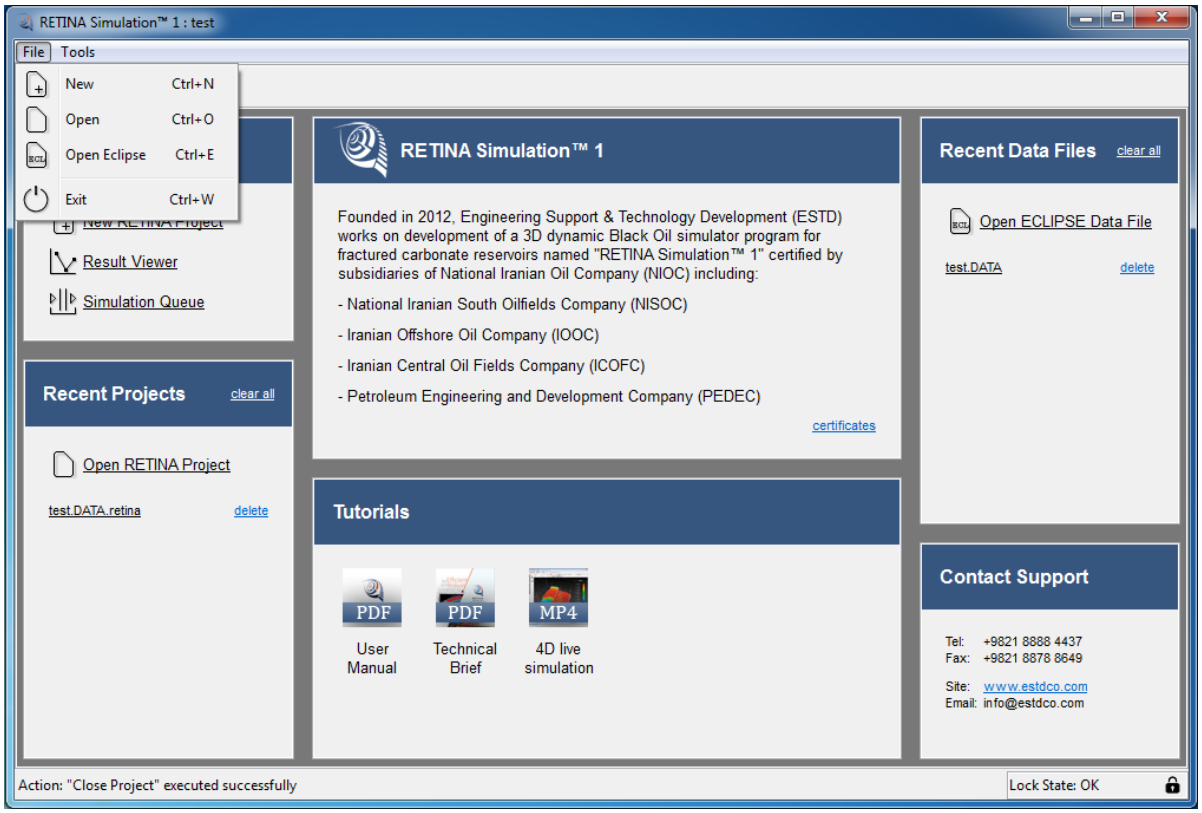


Figure 2 – Start Page File Tab

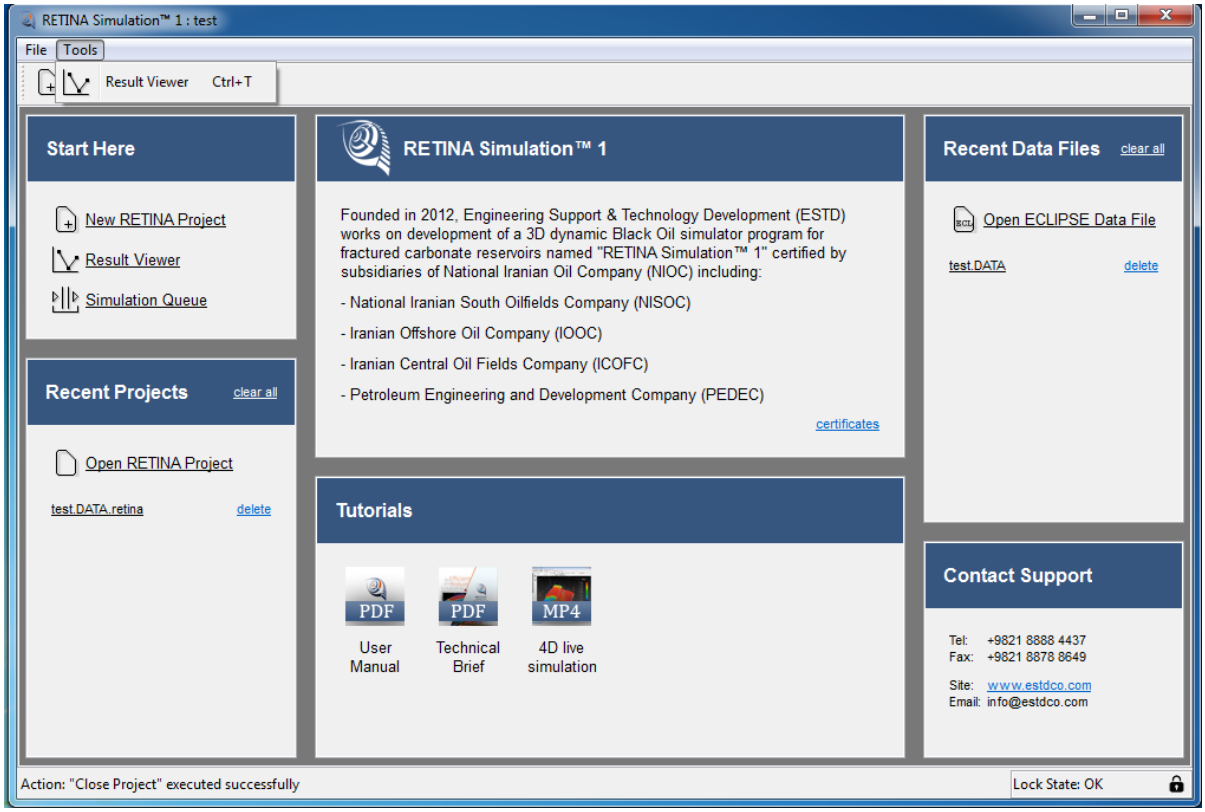


Figure 3 - Start Page Tools Tab

1-1- Create a New RETINA Project

Clicking  , the window of the Figure 4 which includes Project Name, Reservoir Name, Engineer Name, Supervisor Name, Validator Engineer Name, Start Date, Grid Type, and System Phase fields is displayed. User can either fill these fields or leaves them defaulted. “Grid Type” must be set to one of the “Single” or “Dual” cases. The single grid type describes a model with no fracture and the dual grid type is applied for the fractured models. User can select the system phase of the model, using the “system Phase” combo box. This combo box includes following five

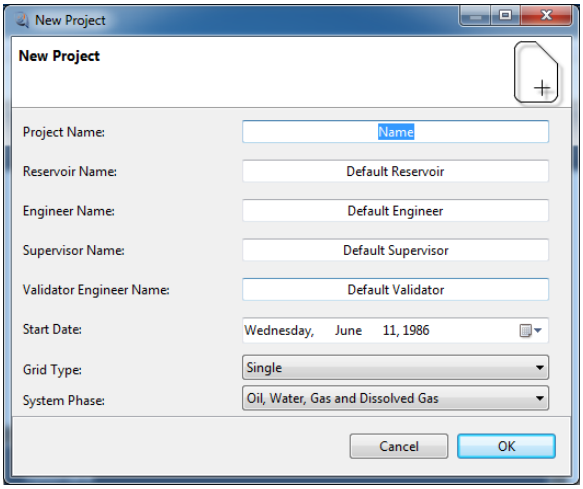


Figure 4 - Create a New Project

types of petroleum reservoir fluids: oil, gas, water, dissolved gas, and vaporized oil.

1-2- Open Results

User will be able to open multiple results of either RETINA or Eclipse for viewing or comparing, using the button of  ; afterward, the window of plot result viewer which has been displayed in Figure 5 is opened. User can import multiple cases, and select multiple vectors and objects. Also user can define multiple result sheets and select the numbers of chart in each sheet.

Before loading the result files, the "Table" and "Comparison" buttons are inactivated. User can import result files using  button or remove them by ; imported results can be renamed by clicking on them. There are three choices for importing result files in RETINA plot result viewer window:

- Import RETINA Result File (with an extension of *.rri)
- Import Eclipse Summary File (Eclipse unified or multiple binary result files with extension of *.SMSPEC)
- Import Eclipse RSM File (with an extension of *.RSM)

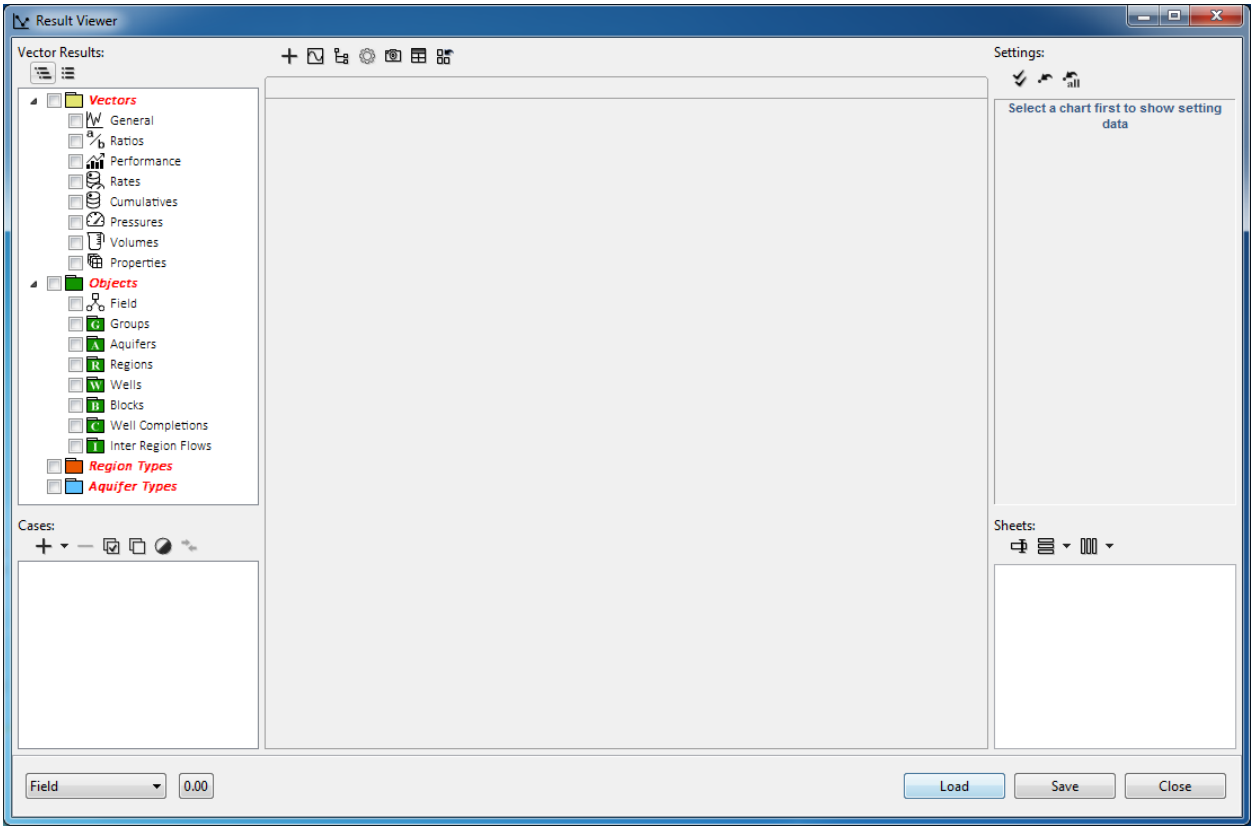


Figure 5 - Plot Result Viewer Window

User can also choose or change unit system or precision of data, using Figure 6 dropdown list and 0.00 option, in all windows of the RETINA. RETINA Result Viewer dialog will be described extensively, in Section 5.

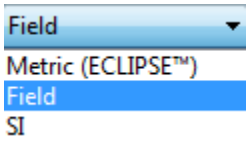


Figure 6 - Unit System

1-3- Open a Saved Project

Clicking on the button, user can load a RETINA saved project, using Figure 6 window. After selecting the path of the project, and a file with format of *.retina, the project is going to open.

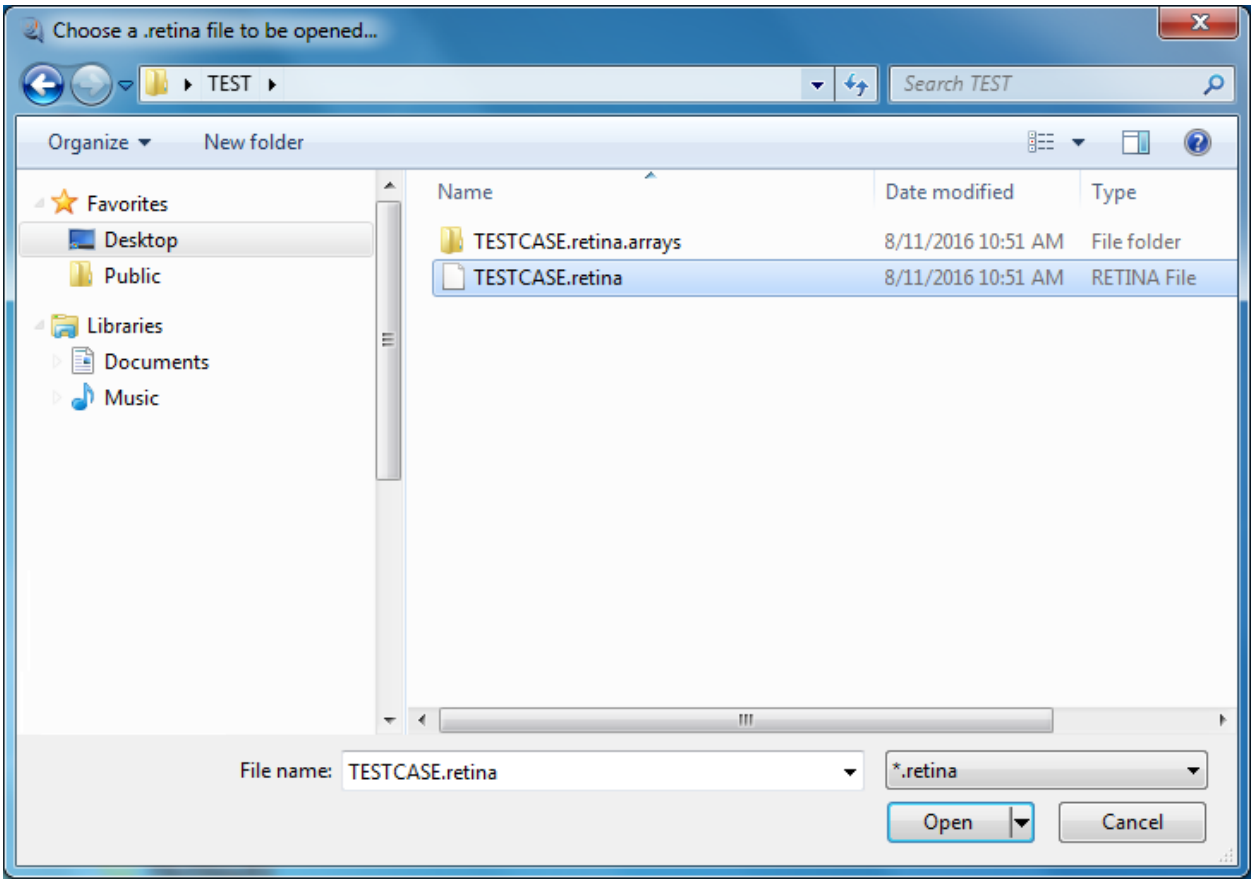


Figure 7 - Project Path Window

1-4- Open Eclipse Data File

The “Import Eclipse” window of Figure 8 is opened, using  button. It should be noted that, RETINA can open the Eclipse data files with *.DATA format.

In the “File Path” section the recently opened data file paths are shown. In this section the most recent item is displayed in the first row of file path table. For clearing all recent paths, you can use the “Clear History”. User can select 3D check-box for having three dimensional view of the reservoir model.

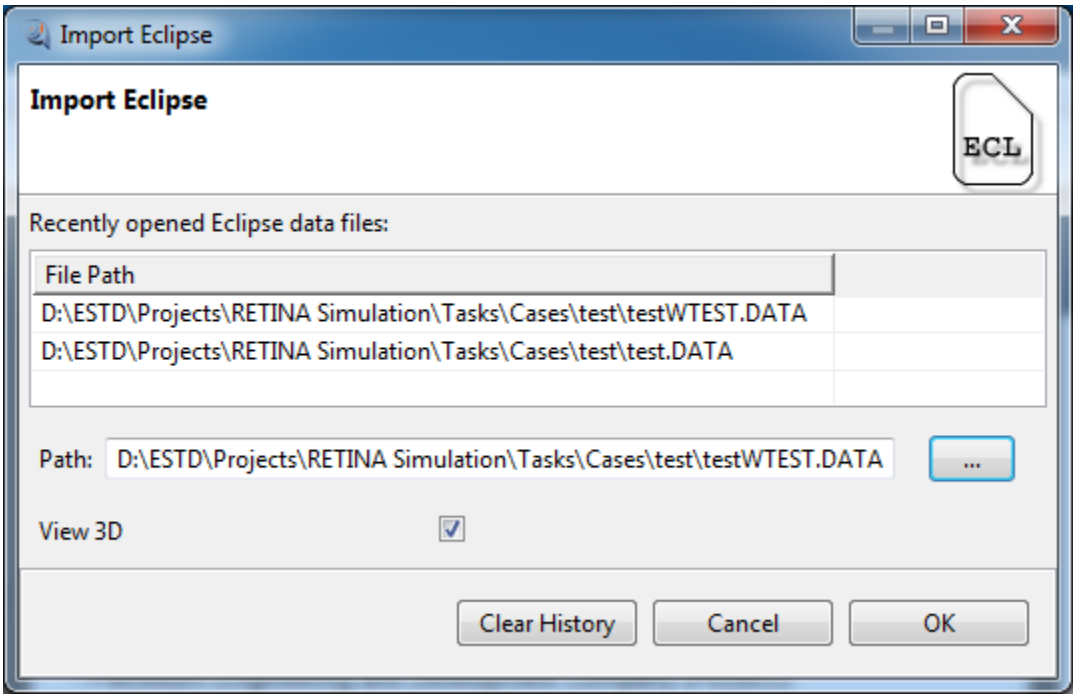


Figure 8 - Open an Eclipse Data File Window

Clicking the "OK" button and closing the window, "Message List" window is displayed. User can see three levels for the messages of window. The first level which is indicated by , is Error messages. The second and third levels of the messages are indicated by (warning messages) and (information messages), respectively (Figure 9).

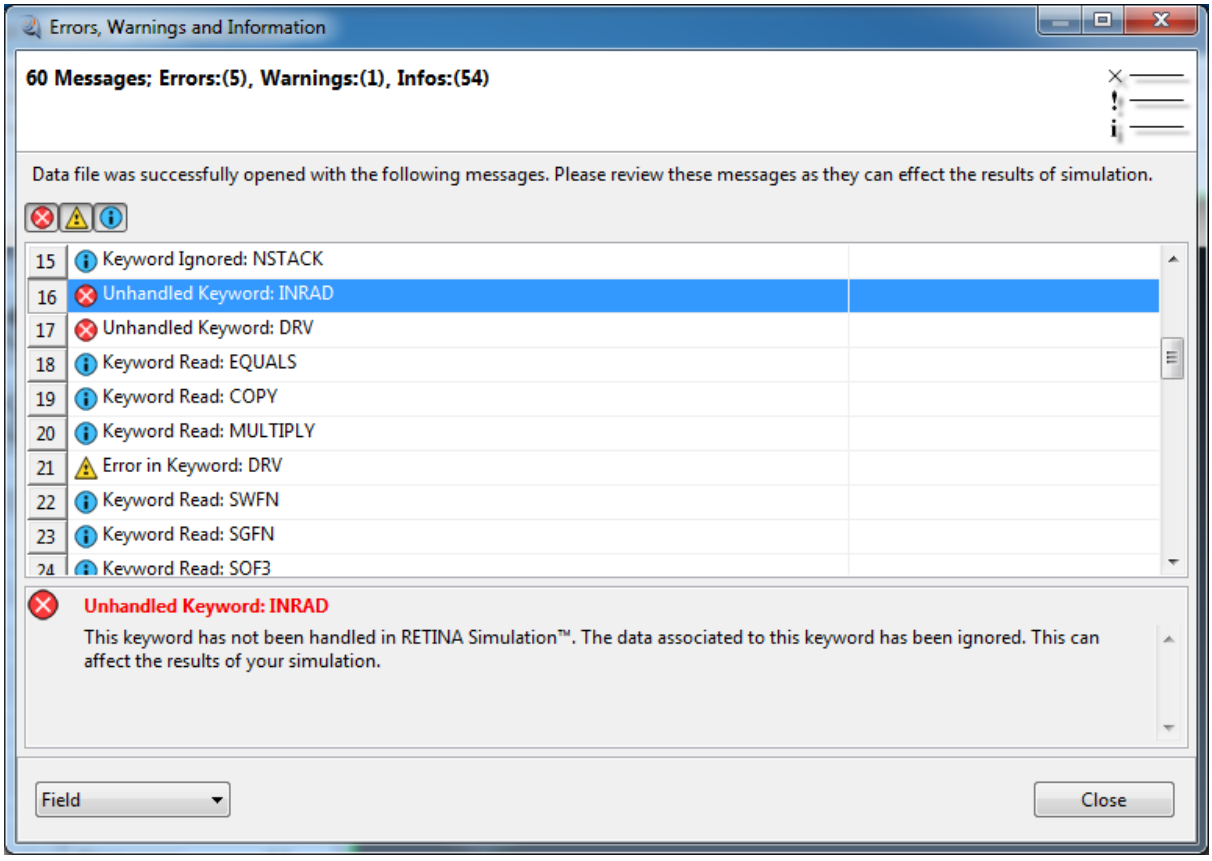


Figure 9- Message List Window

- Error messages are displayed when there are one or more keywords in the data file, which RETINA does not cover them. Simulating the data file without these keywords is impossible. User must resolve the displayed Errors by removing or modifying them. Note that for each keyword, the separate message is displayed where user can select each message to see its details.
- Warning messages are displayed when there are one or more keywords in the data file, which RETINA does not cover them, but simulating the data file without these keywords is possible. It should be noted that ignoring these types of keywords can affect the running behavior and results of the simulation. If user wants to have same results as Eclipse, these keywords must be removed from data file. Same as Error messages, the separate message is displayed for each keyword, where user can select each message to see its details.
- The information messages are displayed when keywords of the data file are covered by the RETINA. The information messages are categorized in two types: a) messages which are displayed for successful reading, and b) messages for ignoring some keywords which have not effect on results of the simulation. Same as Error and

warning messages, the separate message is displayed for each keyword, where user can select each message to see its details.

User can also filter the displayed messages for displaying just desired messages level (Figure 10).

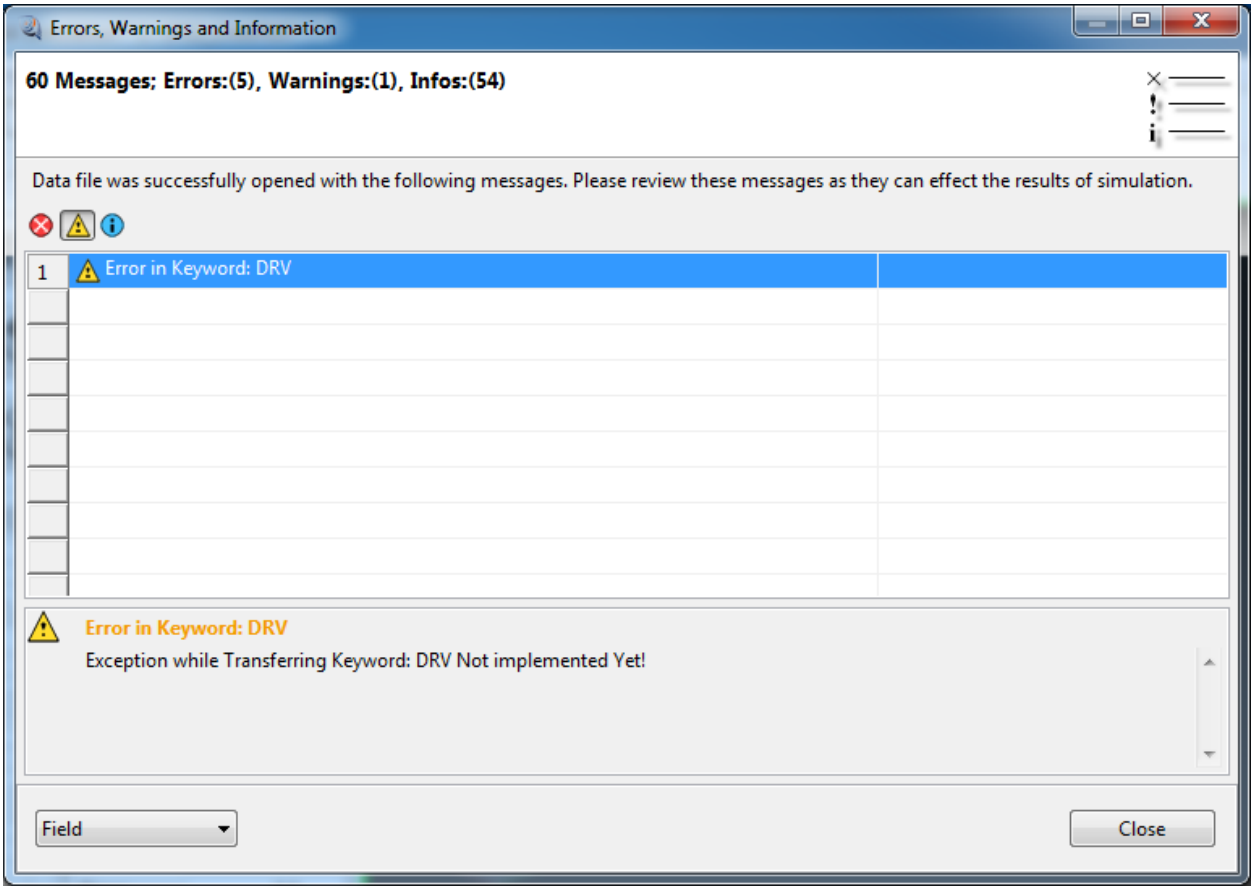


Figure 10 - Filtering of the Messages for warnings

2- Main Components of the RETINA View

The main view of the RETINA after loading or creating a project will be similar to Figure 11, which consists of the following components:

- Program Standard Toolbar
- 3D View Toolbar
- Data Toolbar
- Left Panel

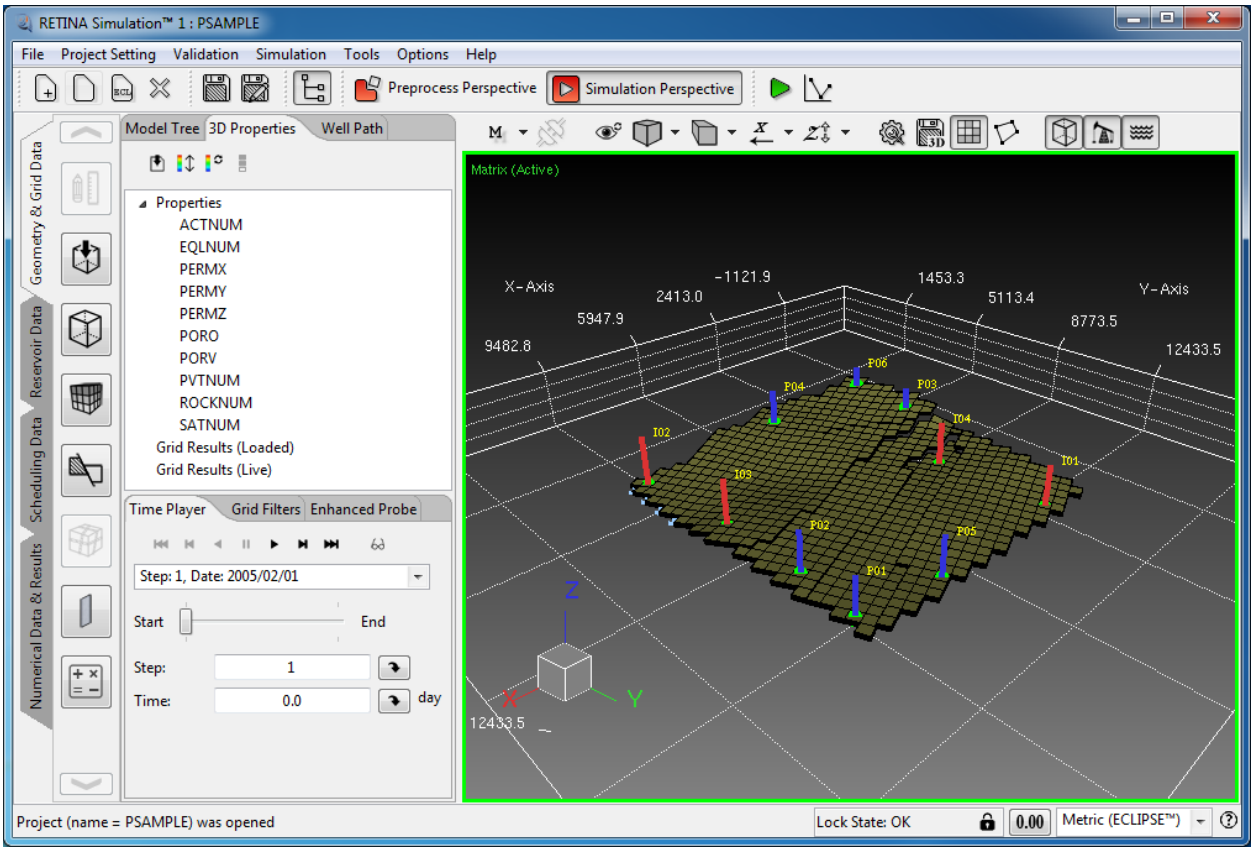


Figure 11 – RETINA Simulation 1 View

2-1- Program Standard Toolbar

Program standard toolbar has been placed in top of the main view and consist of File, Project Setting, Validation, Simulation, Tools, Options, and Help buttons. There are also Facilities for opening and creating project, closing, saving, left panel show/hide, perspectives, and result viewer options.

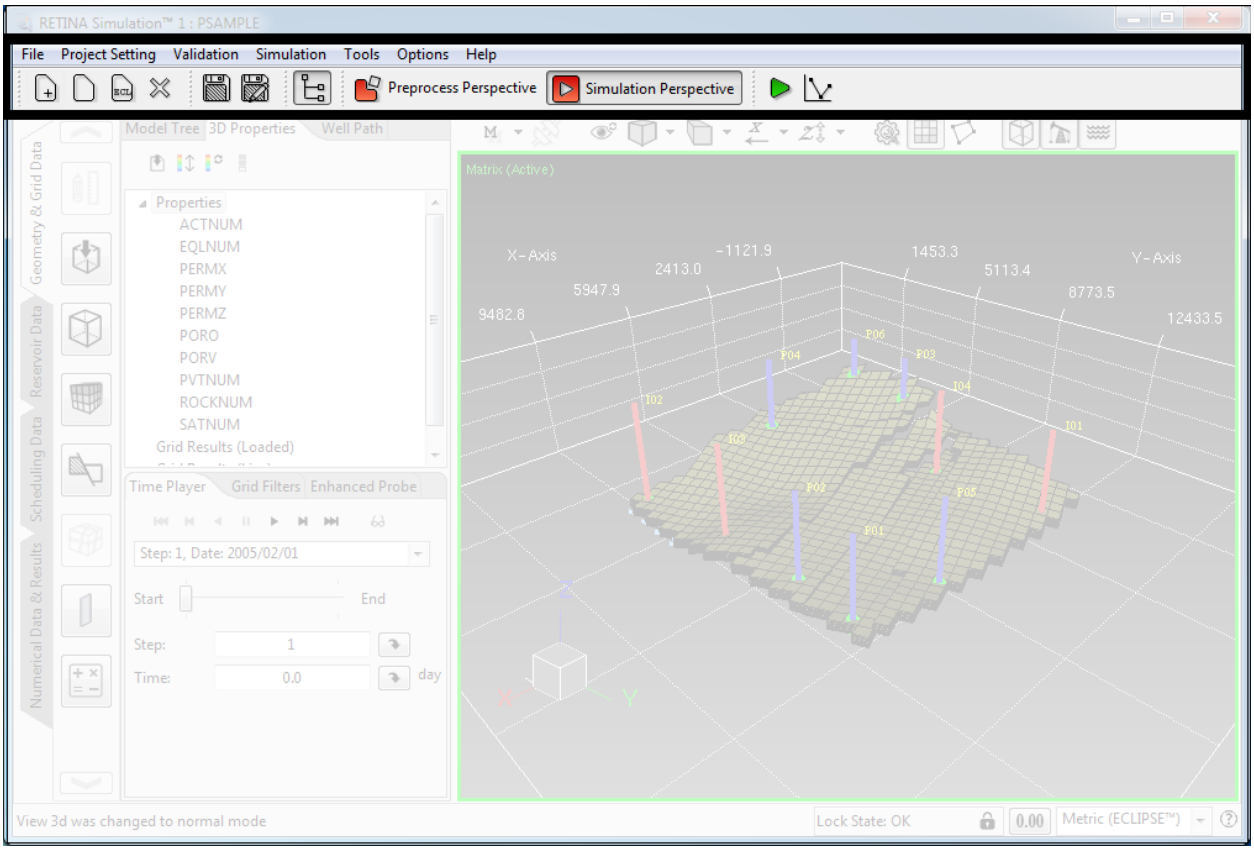


Figure 12 - Program Standard Toolbar

2-1-1- File

- Create a New Project
- Open a RETINA Saved Project
- Open an Eclipse Data File
- Close Project
- Save Current Project
- Save Project as a New One
- Exit

2-1-2- Project Setting

- Project Information
- Open Eclipse Import Messages

2-1-3- Validation

 Full Project Validation
 Grid Validation
 Function Validation
 Initialization Validation
 End-Point Scaling Validation
 Aquifers Validation
 Scheduling Validation
 Numerical Validation
 Output Validation

2-1-4- Simulation

 Start Simulation
 Simulation Configuration

2-1-5- Tools

 Open Plot Results Viewer

2-1-6- Options

 Input/output Path
 Validation
 Property Template

2-1-7- Help

 Help Document
 Technical Document
 Keyboard Shortcuts

2-1-8- Perspectives

 Preprocess Perspective
 Simulation Perspective

2-2- 3D View Toolbar

3D view toolbar has been placed below the program standard toolbar, which is shown in Figure 13.

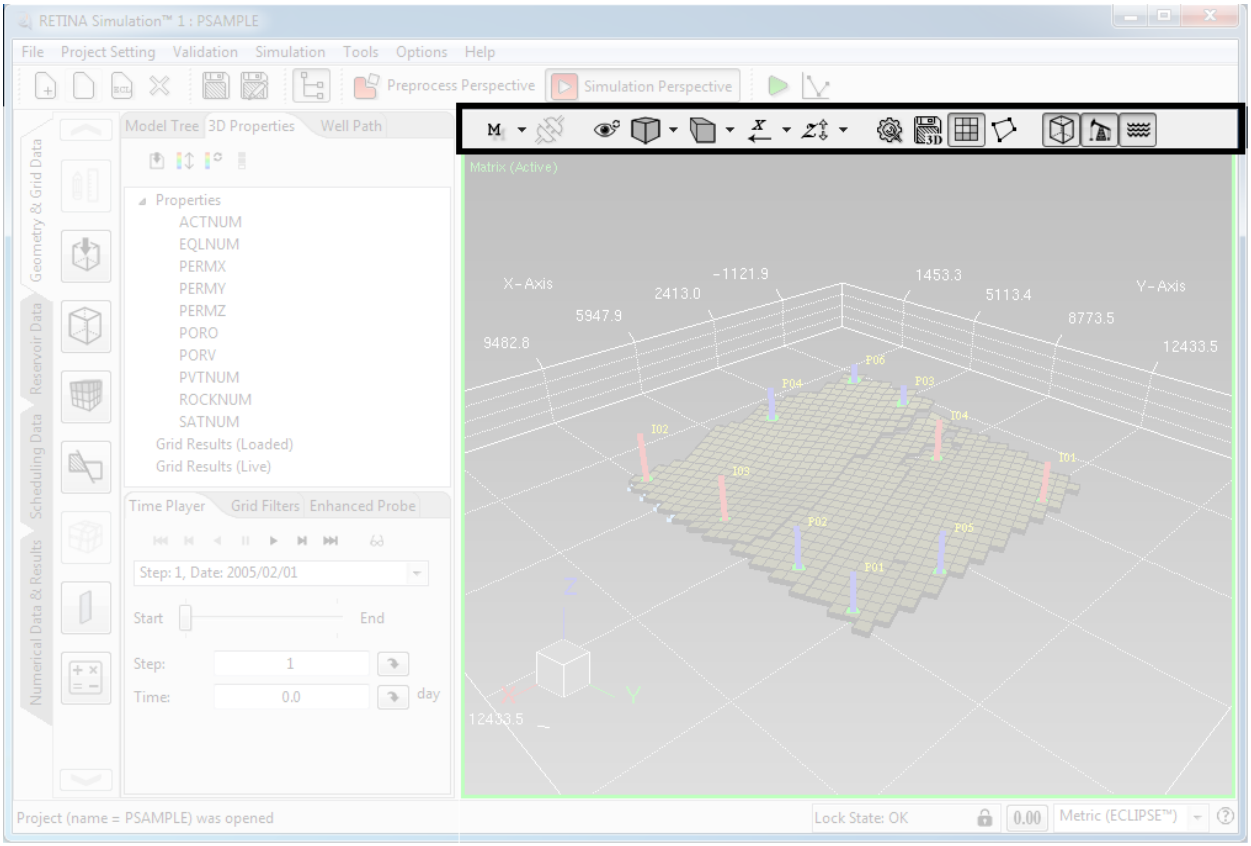


Figure 13 – 3D View Toolbar

Each buttons is introduced here:

- M — Dual View Option:
  - M — Show Matrix View
  - F — Show Fracture View
  - M/F — Show Both Matrix and Fracture View
- Sync Fracture and Matrix views
  - Un -Sync
  - Sync
- Reset View
- Render Actions consist of:

 Render Points

 Render Wireframe

 Render Solid

 Camera Mode Option:

 Perspective

 Orthographic

 Camera Direction Actions consist of:

 Camera direction on negative X-coordinate

 Camera direction on positive X-coordinate

 Camera direction on negative Y-coordinate

 Camera direction on positive Y-coordinate

 Camera direction on negative Z-coordinate

 Camera direction on positive Z-coordinate

 Z Scale Actions which is used for scaling the z-axis of a model in 3D view

 3D View Setting

 Save 3D View as .png File

 Show or hide inactive cells

 Create Polygons

 Show or hide model grids

 Show or hide model wells

 Show or hide aquifers

2-3- Data Toolbar

The Data Toolbar consist of following 4 main tabs, which will describe in this section:

- Geometry & Grid Data
- Reservoir Data
- Scheduling Data
- Numerical Data & Results

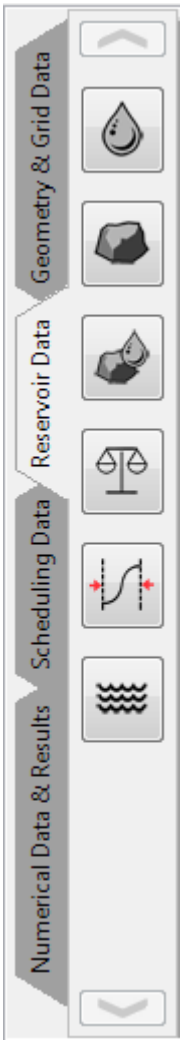


Figure 14 – Data Toolbar

2-3-1- Geometry & Grid Data

Geometry & Grid data tab is used for defining geometry and grid properties, which includes following buttons:

- Initializing Grids (is used just for new projects)
- Grid file Importer
- Grid data (like DX, DY, DZ, COORD and etc.)
- Grid Property (like porosity, permeability and etc.)
- Fault Manager
- Dual (fractured models) Options
- Face Manager
- Script Calculator

2-3-2- Reservoir Data

Reservoir data tab is used for defining reservoir data like rock and fluid properties, Aquifers, Regions, SCAL data and etc. This tab includes following 6 buttons:

- PVT Region Data
- Rock Region Data
- Saturation Region Data
- Equilibration Region Data
- End Point Scaling Region Data
- Aquifer Data

2-3-3- Scheduling Data

Well, group, drilling, rig and VFP definition and scheduling data are available in scheduling data tab using following buttons:

 Date Manger
 Well Manager
 Well Group Manager
 Group Recurrent Manager
 Vertical Flow Performance Manger
 Well Recurrent Manager
 Reservoir Recurrent Manager
 Drilling Manager
 Rig Manager

2-3-4- Numerical & Results Data

It includes two following buttons which are used for output selection before run, and some numerical parameters for the simulation like time steps, solver type and etc.

 Edit Numerical Parameters
 Manage Output Data

2-4- Left Panel

This panel has three main tabs of “Model tree”, “3D Properties”, and “Well Path”.

2-4-1- Model Tree

Initially, model tree includes a node of “Project” which is parent of the following nodes:

- Grid
- Functions
- Initialization
- EPS (End Point Scaling)
- Aquifer
- Scheduling
- Numerical

- Output Data

The “Grid” node has two child nodes of “Geometry” and “Properties”. The geometry node includes information about the geometry of the model (like ZCORD, COORD, DX, DY and etc.). The properties of model like permeability and porosity maps, Region Nums (like PVTNUM, SATNUM and etc.) and other grid properties which can be entered using “Grid Properties View”. Double clicking on each node of “geometry” or “properties” will lead user to related window.

The “Function” node includes “PVT”, “Rock”, and “Saturation” (SCAL) properties of the model which are available in “Reservoir Data” tab. User can open each node to see the defined reservoir functions. Double clicking on each node of the “PVT”, “Rock” or “Saturation” will lead user to the related window.

The equilibration data are displayed as child of the “Initialization” node. User can double click on the “Initialization” node to open the “Equilibration Data Management” window.

The end point scaling data are displayed as child of the “EPS” node. Like other nodes, user can double click on the “EPS” node to open the “End Point Scaling Data Management” window.

The “Aquifer” node includes defined aquifer data of the model which are available in “Aquifers Data” window; user can open this window by double clicking on the “Aquifer” node.

The “Scheduling” node includes “Wells”, “Groups”, “VFPs”, “Rigs”, and “Drilling Queues” nodes, which are available in “Scheduling Data” tab. User can open each node to see the defined reservoir functions. Double clicking on each node of the “Wells”, “Groups”, “VFPs”, “Rigs“ or “Drilling Queues” will lead user to the related window.

Double clicking the “Numerical” or “Output Data” nodes will lead user to related window in “Numerical Data & Result” tab.

If user validate project using “Validation” button of the program standard toolbar, the state of each node changes to error (⊗) or warning (⚠) in case of validation issues, otherwise it remains in validated state (✓). The validation full text messages for each node are available in “Validation Messages” tab below the “Model tree” tab (Figure 15a). If user wants to see all validation messages of current project, he/she will select “Project”

node from the model tree tab. In some cases user just wants to see a special node validation messages, in such cases he/she can click on the desired node (Figure 15b).

There is also a message filtering facility in "Validation Messages" tab.

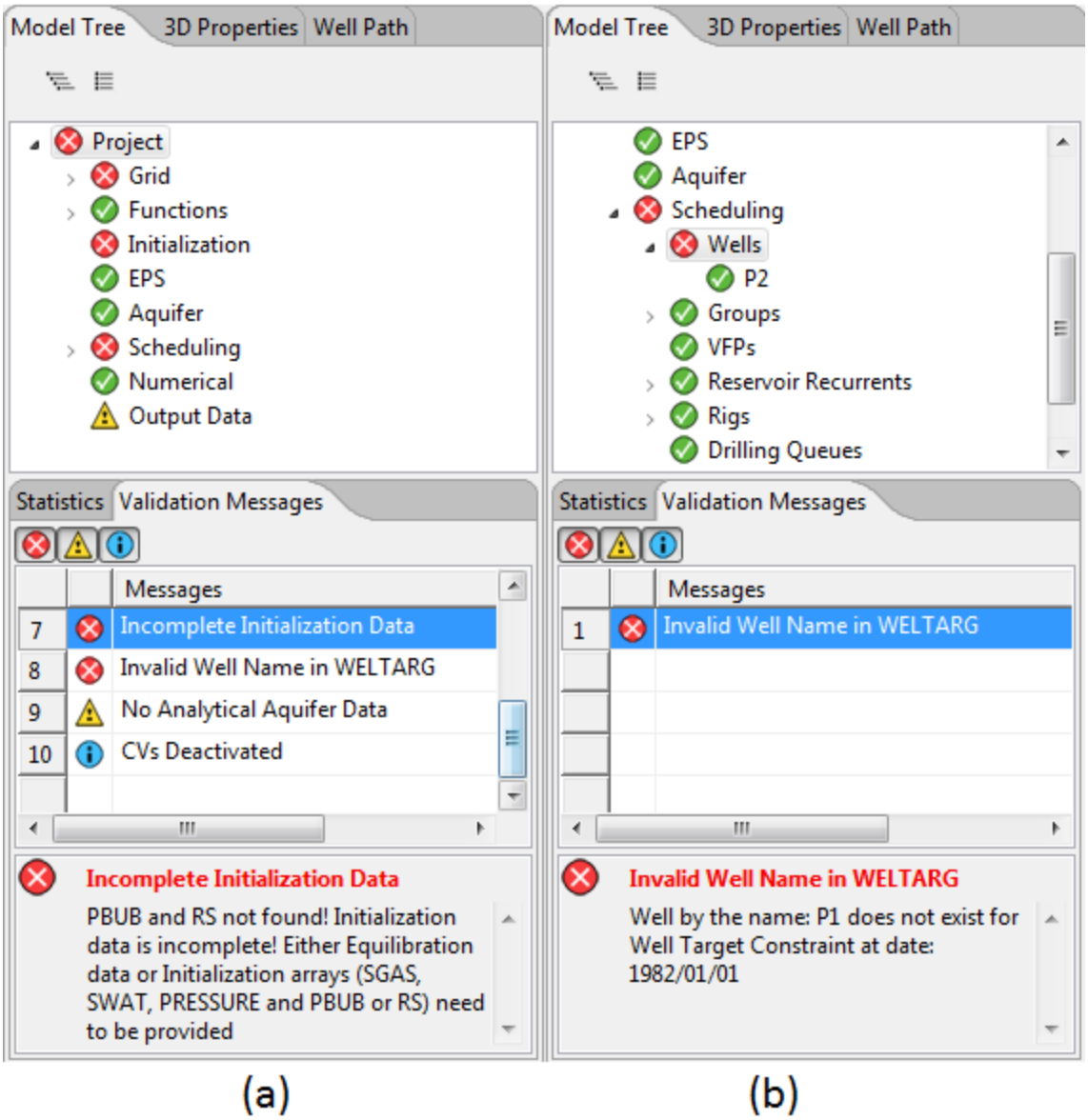


Figure 15- a) Model Tree All Validation Messages of the Sample Project, b) Model Tree Selected Node Validation Messages of the Sample Project

There is a "Statics" tab in addition to the validation messages tab for Model Tree. This tab displays specific description for each node. The "Project" node includes project name, validator, supervisor and engineer names, and finally project (simulation) start date. The "Grid" node includes type of grid geometry i.e. center-point, corner-point, or ZCORN

COORD, number of total, active, and inactive cells, minimum and maximum values for x,y,z, and the dimension of data file i.e. Nx, Ny, Nz; The "Function" node statistics tab displays the number of defined regions such as PVT, Rock and saturation; the type of the initialization is displayed in "Initialization" node statistics tab. number of End point scaling tables and type of end point scaling is displayed in "EPS" node statistics tab. the "Aquifer" node includes the number of aquifers, type of defined aquifers and number of aquifer tables. The number of groups, wells, VFP tables, and other scheduling data is listed in the "Scheduling" statistics tab. Solver options and Modeling options are displayed in "Numerical" node, and finally, the output data counts are displayed briefly in the statistics tab of the "Output Data" node. All of the aforementioned nodes have some child and each child has own statistics information in detail.

2-4-2- 3D Properties

In addition to legend options and grid results import facility, the 3D properties tab includes following nodes (Figure 16):

- Properties
- Grid Results (Loaded)
- Grid Results (Live)

Three tabs of "Time Player", "Grid Filters", and "Enhanced Probe" exist in lower section of the left panel for 3D properties tab.

The "Properties" node includes all grid properties which have been previously defined in "Geometry and Grid Data" tab. User can see the desired property map by clicking on it. For example Figure 16 shows the horizontal permeability of the sample reservoir in "Y" direction and Figure 17 shows the porosity map of the sample model.

Initially the legend data range of all properties which are displayed in 3D view is set to predefined template. User can adjust the legend data range to the displayed 3D property using (). For example, if this action is performed on Figure 17, the porosity map will be like Figure 18. The predefined template range of porosity is [0 - 0.3] (Figure 17), but after adjusting the legend the maximum and minimum values of the sample model is applied and figure 18 is displayed.

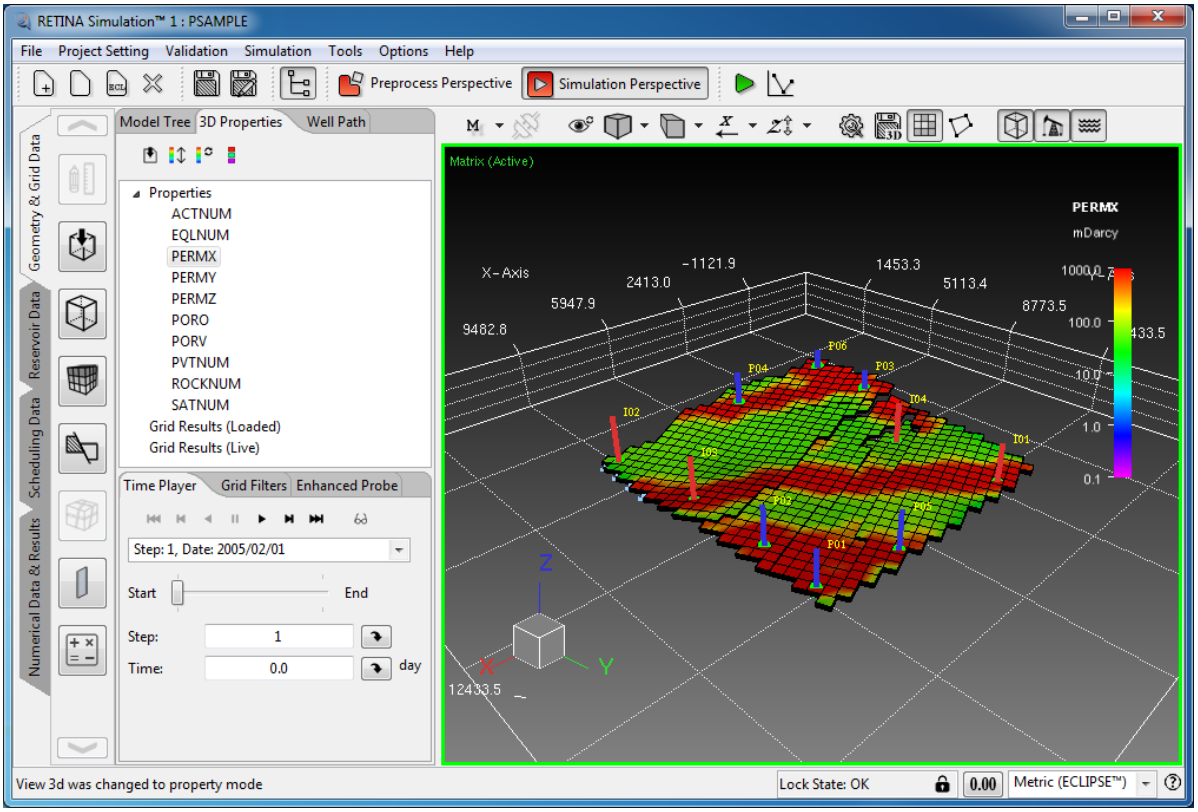


Figure 16- Y-Direction Horizontal Permeability Map of the Sample Model

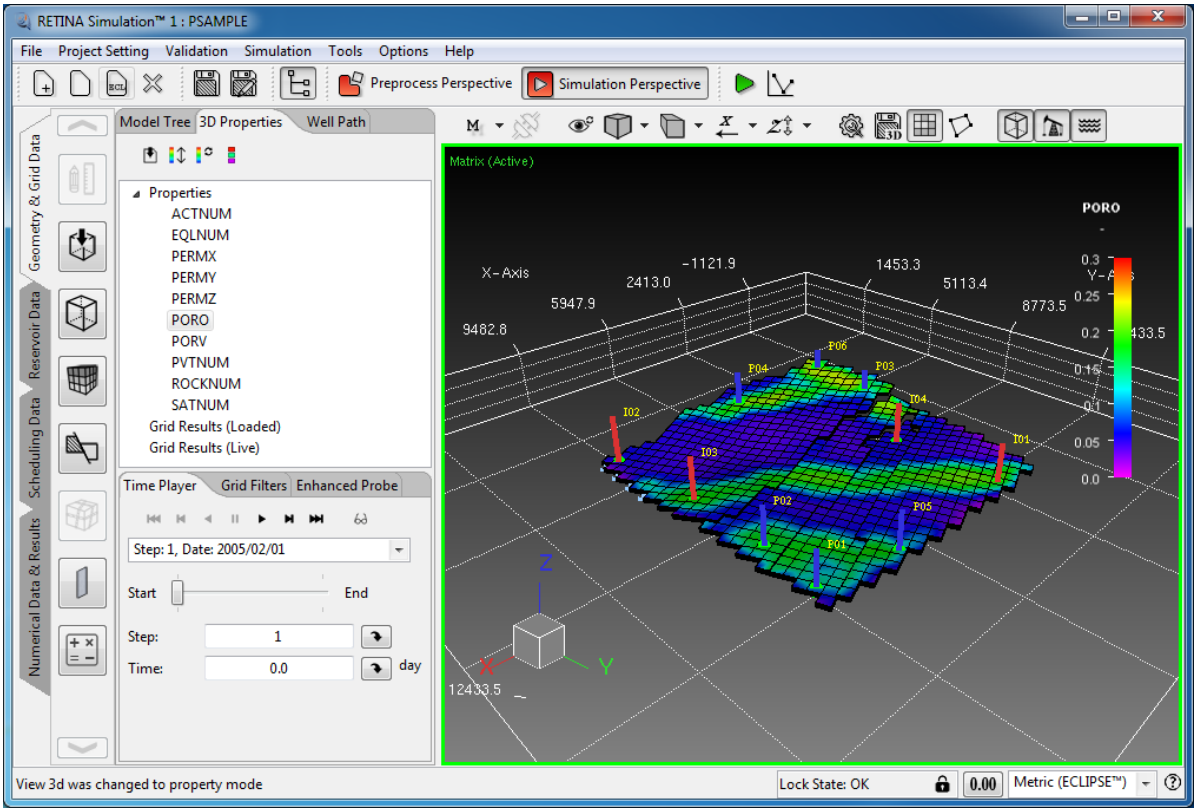


Figure 17- Porosity Map of the Sample Model

For resetting the legend configuration to its default, the button of ( ) is used. The displayed property value can be either continuous or discrete according to selected property; for example porosity is a continuous property whereas PVTNUM is a discreet one. User can change the displaying mode to discrete or continuous status using ( ).

"Import Grid Results" button ( ) is used for importing *.rri RETINA file format which is available during and after simulation in the project path in result folder. A list of grid properties is displayed in "Grid Results (Loaded)" node after importing result file. The property also is displayed in 3D view section (Figure 19). If imported file does not include any grid result, an information message is displayed.

Note: Grid results are generated only in case of selecting desired variables in "3D Visualization Variables" tab of "Output Data" window.

After importing result files, user can press *play* button of which is exists in "Time Player" tab to see the variation of selected variable values during time. For example Figure 20 shows the oil saturation distribution in reservoir in date of 2009/02/01. Note that the initial date of simulation is 2005/02/01 according to Figure 19. The following buttons exist in "Time Player" tab:

- ⏮ Going to the first step
- ⏭ Going to the previous step
- ◀ Playback
- ⏸ Pause
- ▶ Play
- ⏭ Going to the next step
- ⏭ Going to the last step
- Active live result loader (only during simulation)

User can see desired 3D variables lively during simulating using button.

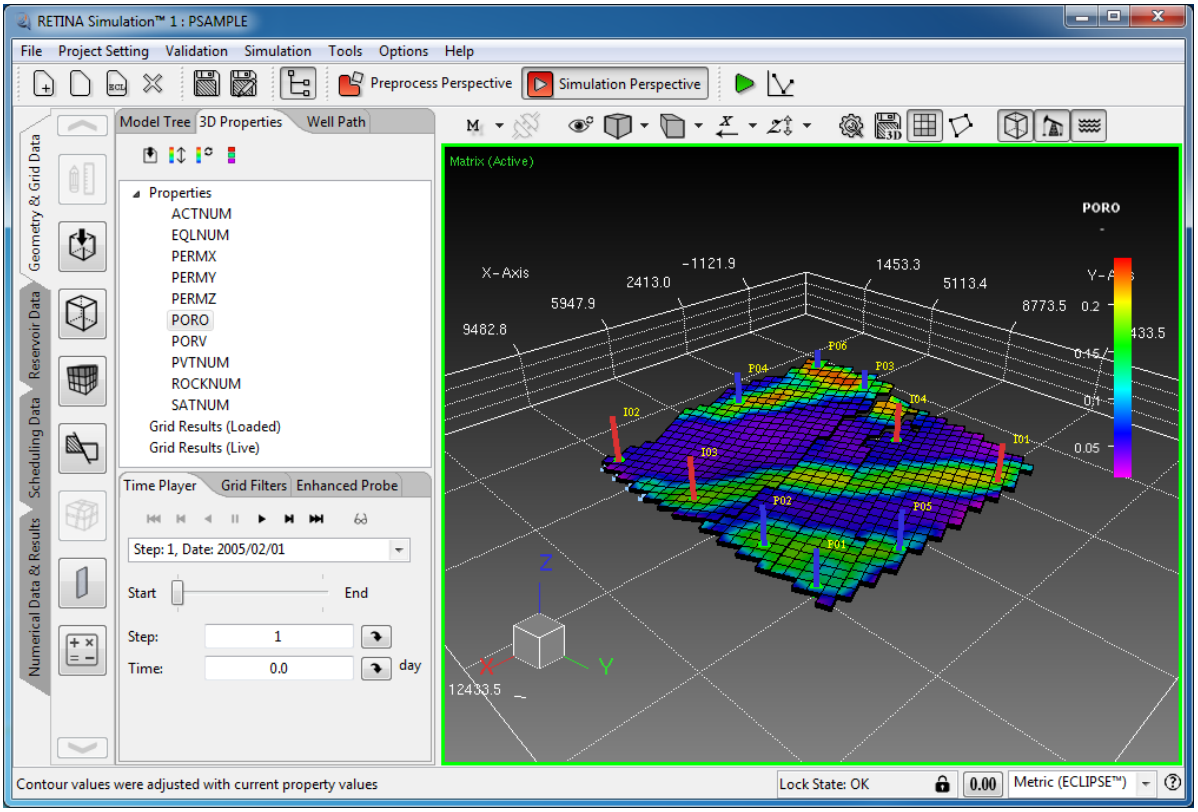


Figure 18- Adjusted Porosity Map of the Sample Model

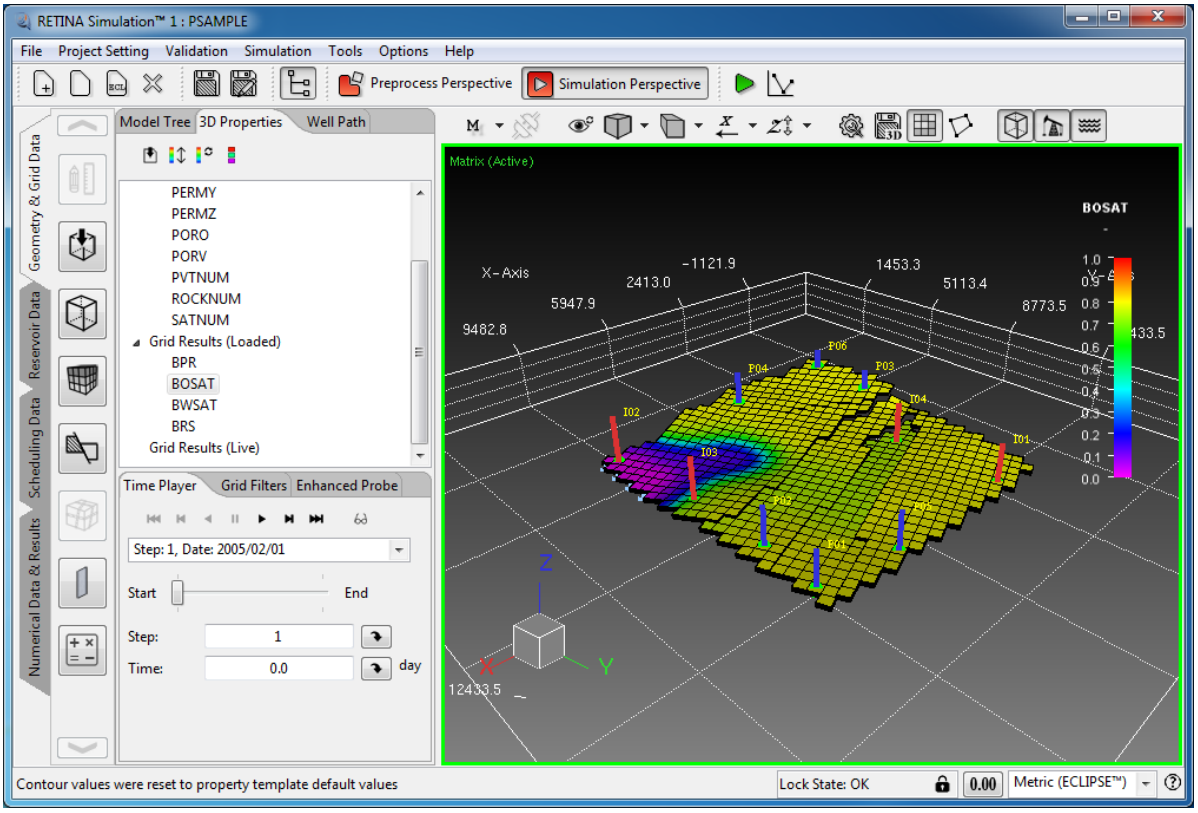


Figure 19- Loaded Grid Results for Oil Saturation Variable (Initial State)

"Grid Filter" tab is used to filtering grid 3D view to display desired section or layer of the model. This tab includes two separated sections of "IJK-Slice Player" and "Filters". The IJK-Slice Player section not only is used for displaying defaulted Slices in 3 main directions of I, J, and K but also is used for defining custom IJK slice. For example Figure 21 displays the 20th layer of the sample model in X direction.

The following buttons exist in "IJK-Slice Player" section of "Grid Filter" tab:

- ⏮ Going to the first layer
- ⏪ Going to the previous layer
- ◀ Playback layers
- ⏸ Pause
- ▶ Play layers
- ⏩ Going to the next layer
- ⏭ Going to the last step

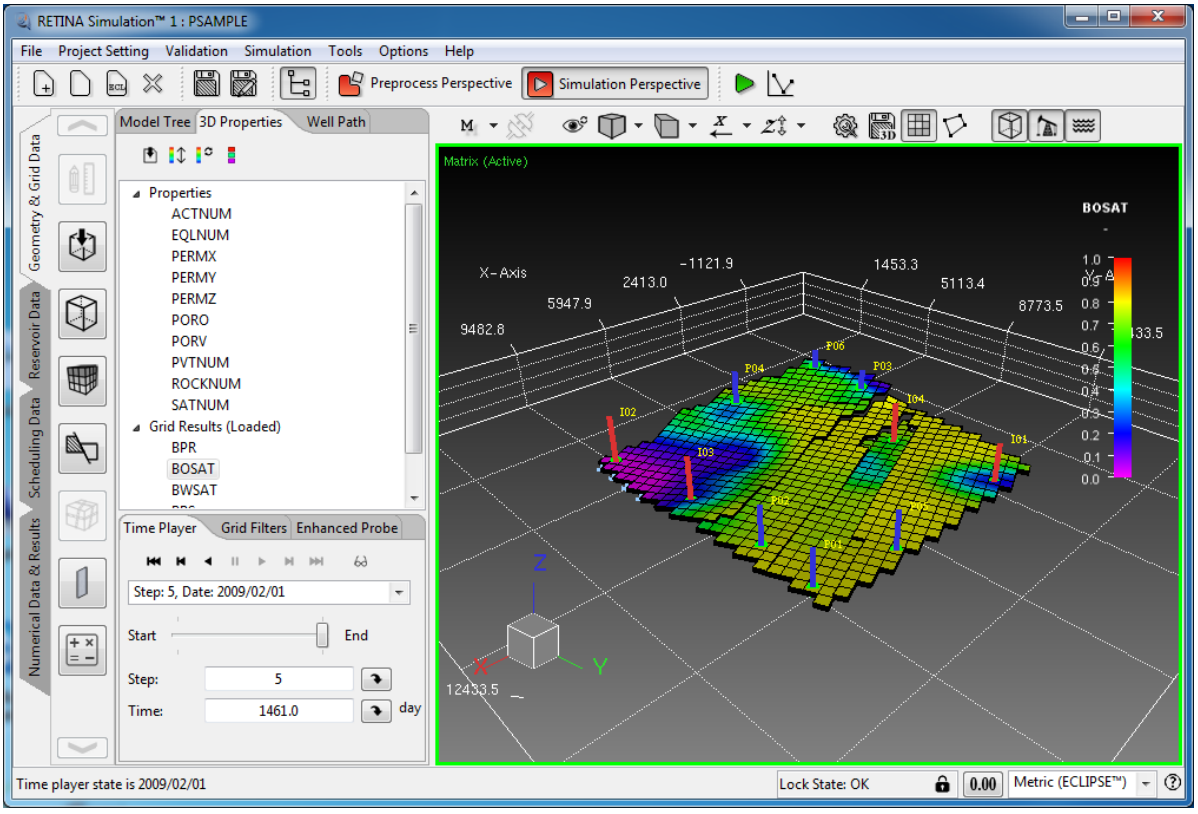


Figure 20- Loaded Grid Results for Oil Saturation Variable (After four years)

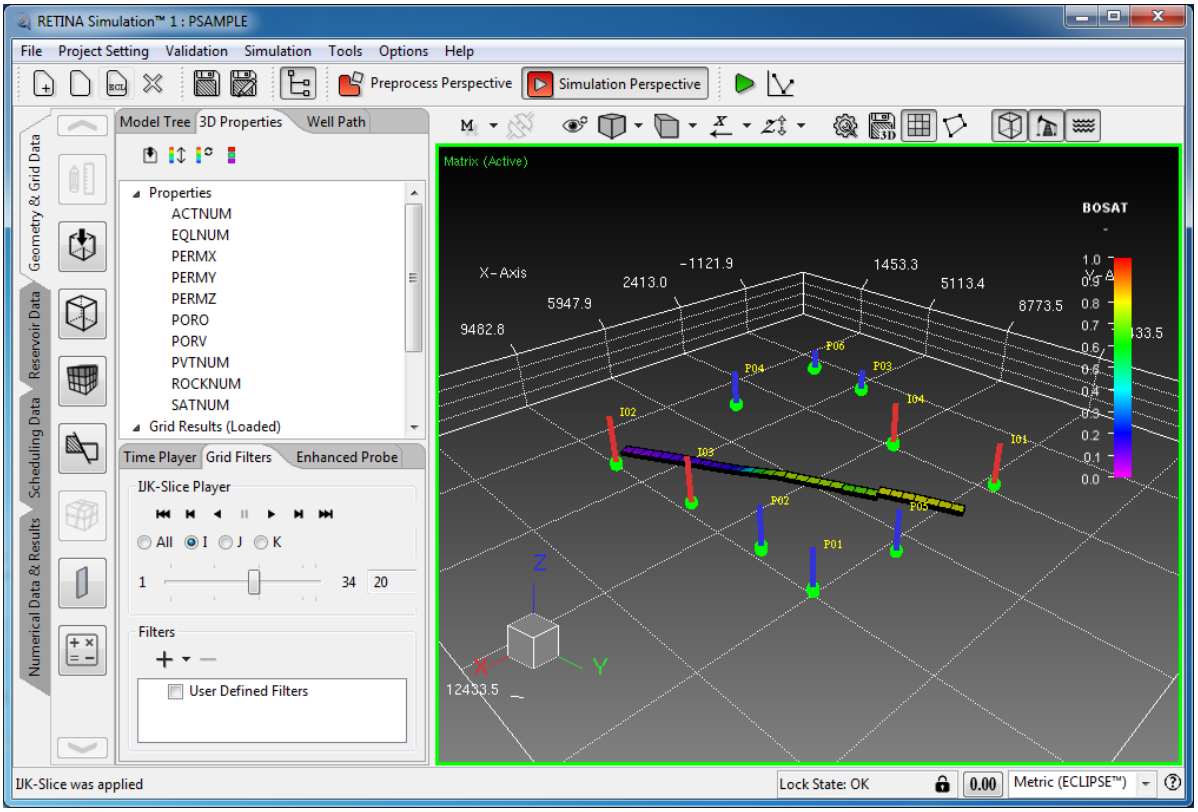


Figure 21- IJK-Slice in X-direction

It is possible to define five possible filters of box, geometry, polygon, property, and logical using "Filters section. Clicking on the plus sign in the filter section allows user to define desired filters. Each filter definition is explained here:

1. *Block Box Filter*: the dialog of the Figure 22 is displayed if user wants to add block box filter. Initially none of the boxes are checked which means no filter is added to 3D view. The name of the "Block_Box_##" is assigned to the defined block box filter, by default; this name is editable. For adding filter on each direction, user should just check the related box and enter the minimum and maximum index in the specified direction. For example the Figure 23

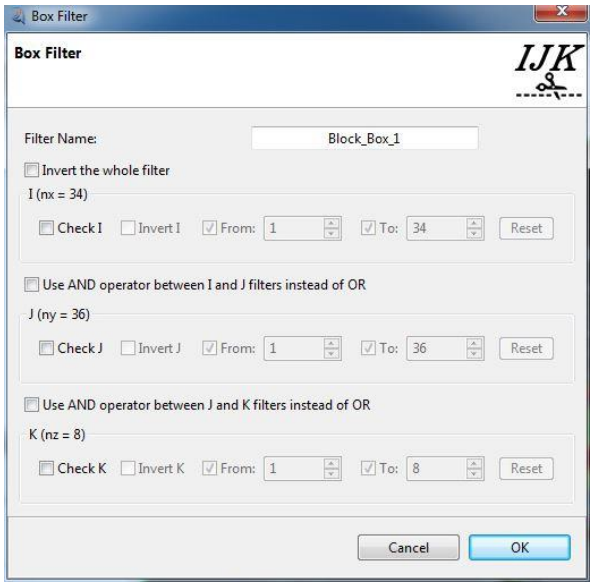


Figure 22 Adding Block Box Filter

displays the configurations for adding a filter index of 2 to 10 in the "I" direction. Clicking on the OK button, filter will be applied on 3D view (Figure 24). Note that the editing of the defined filter is allowed by double clicking on the added one in the Filters section. If user unchecks the "From" or "To" index values the 1 and maximum index of the model in desired direction is assumed for them, respectively.

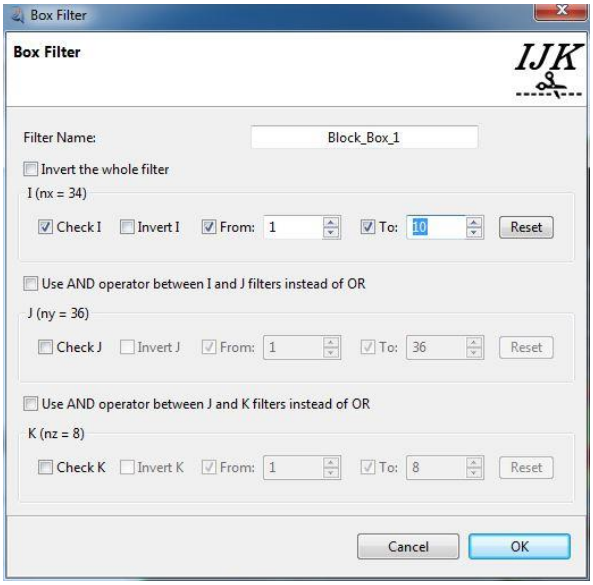


Figure 23 Adding Block Box Filter Index of 2 to 10 in the "I" Direction

User can define filters on two or three directions, for example the Figure 25 displays the filter configuration for I-index of 2 to 10 and J-index of 3 to 25.

In such cases the sum of both filters are added to the 3D view (Figure 26); if user checks the "AND operator" box (Figure 27) only grid cells which are exist in both filters are displayed such as Figure 28. For having invert filters the "invert" check box either for each direction or for whole filters must be checked. The Figure 29 displays the invert filter of the Figure 28 filters.

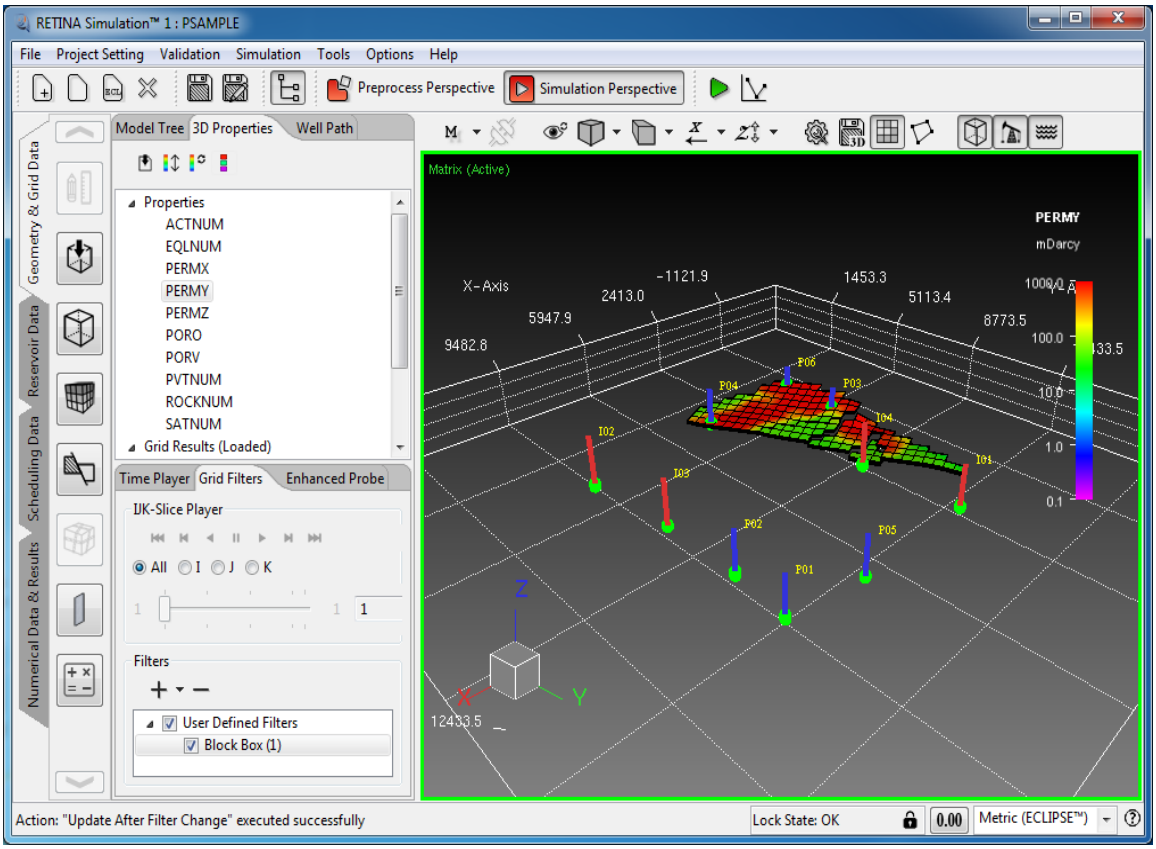


Figure 24 Applied Block Box Filter Index of 2 to 10 in the "I" Direction on 3D View

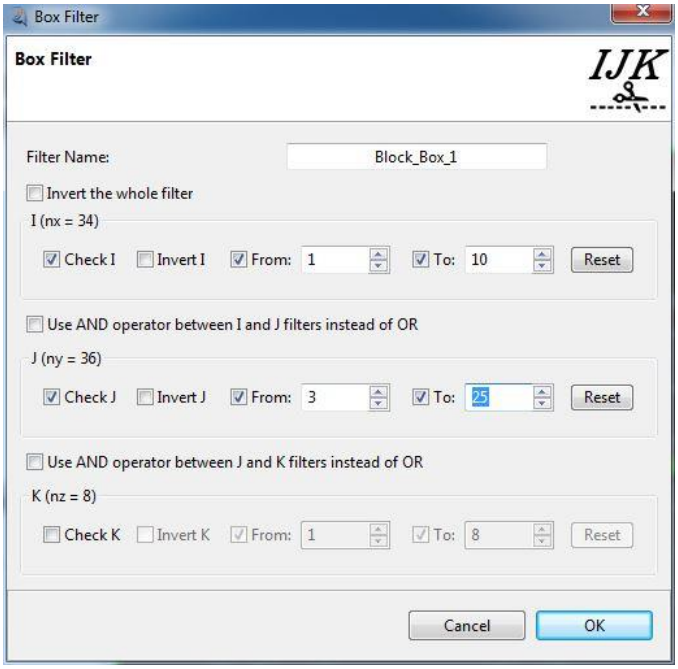


Figure 25 Adding Block Box Filter for I-index of 2 to 10 and J-index of 3 to 25

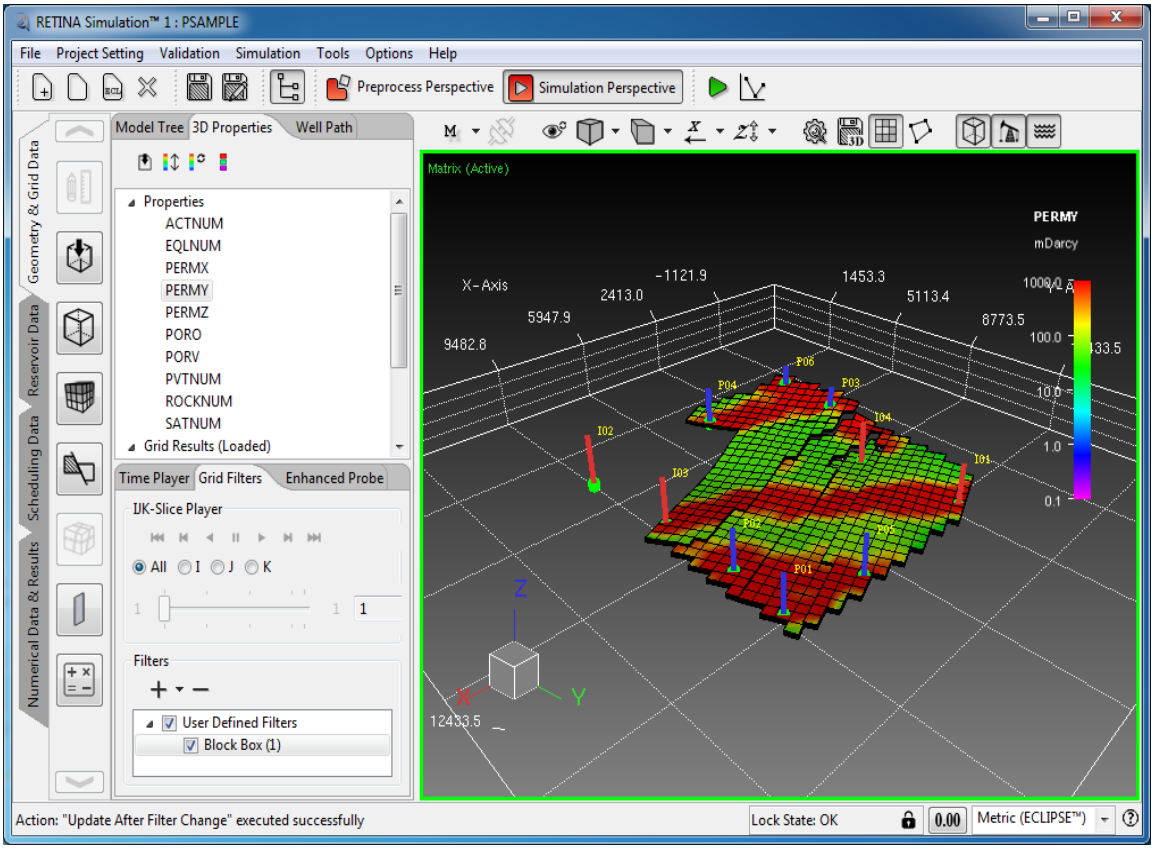


Figure 26 Applied Block Box Filter for I-index of 2 to 10 and J-index of 3 to 25 on 3D View

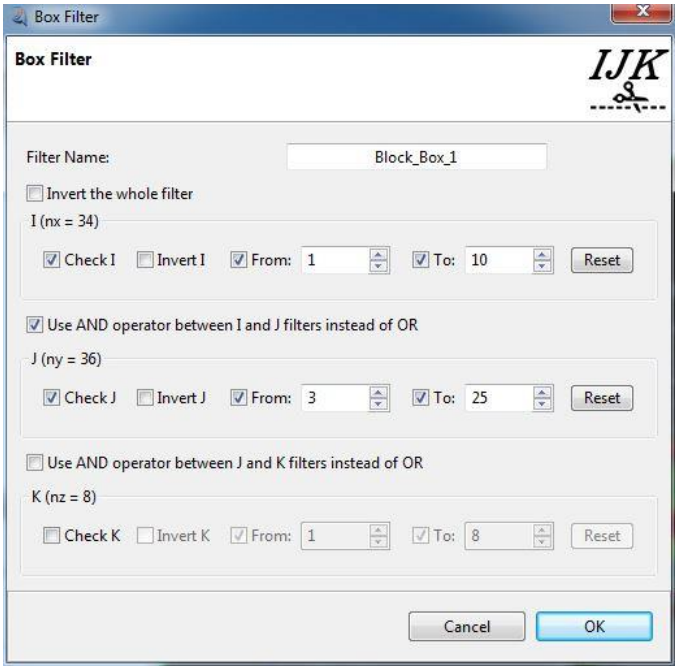


Figure 27 Adding Block Box Filters with AND Operator

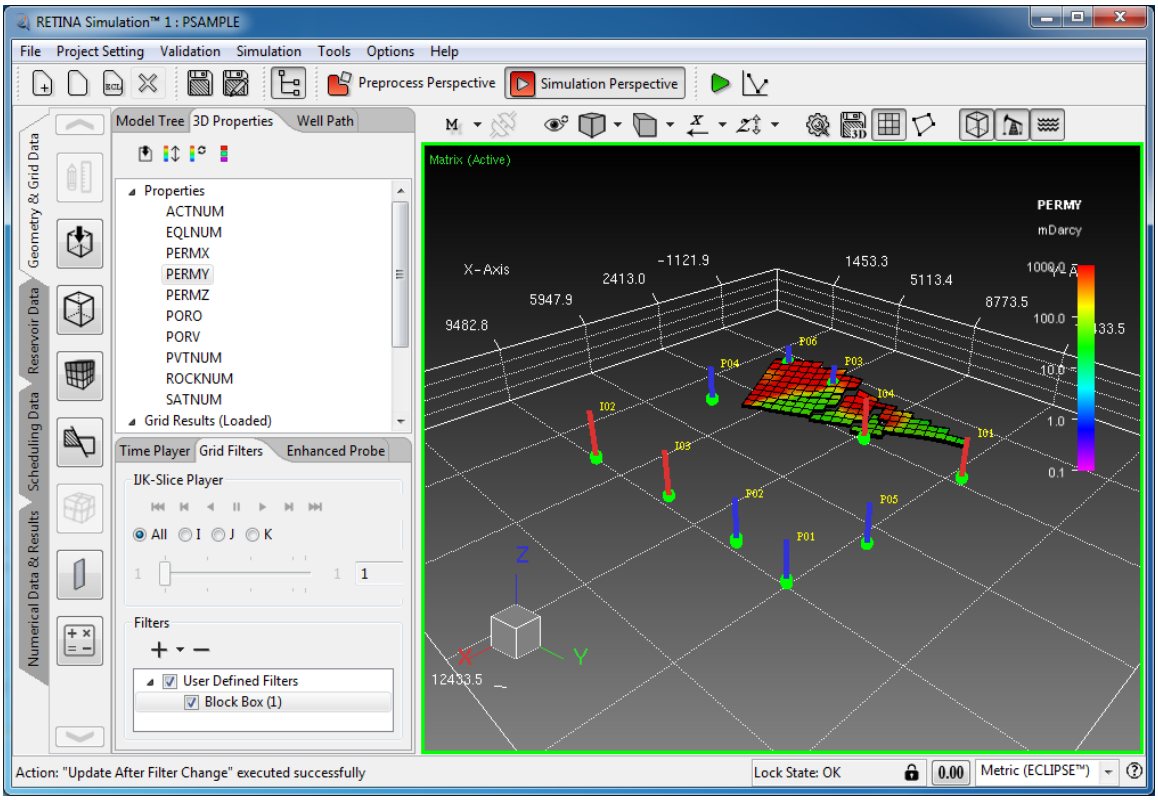


Figure 28 Applied Block Box Filters with AND Operator on 3D View

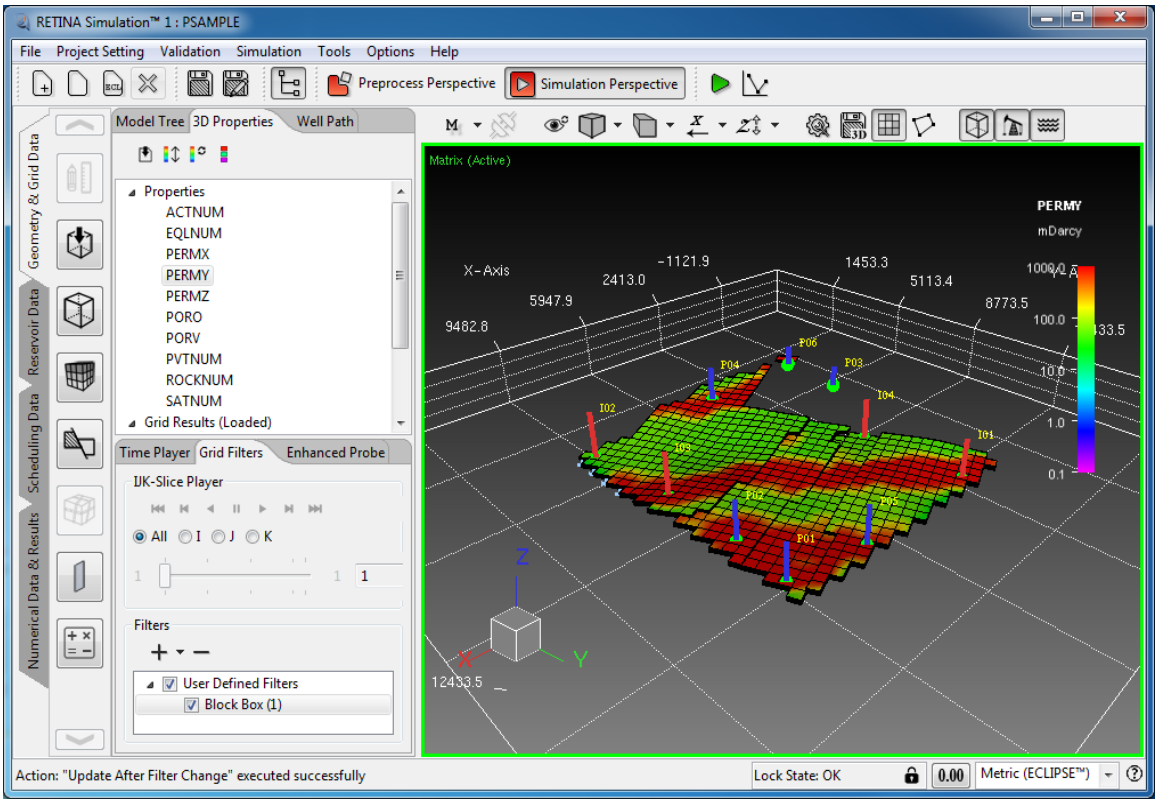


Figure 29 Applied Inver Block Box Filter

2. *Block Geometry Filter*: the dialog of the Figure 30 is displayed if user wants to add block geometry filter. Initially none of the boxes are checked which means no filter is added to 3D view. The name of the "Block_Geometry_##" is assigned to the defined block geometry filter, by default; this name is editable. For adding filter on each direction, user should just check the related box and enter the minimum and maximum coordinates in the specified direction. For example the Figure 31 displays the configurations for adding a filter from 0 to 5000 meter in the "X" direction. Clicking on the OK button, filter will be applied on 3D view (Figure 32). Note that the editing of the defined filter is allowed by double clicking on the added one in the Filters section. If user unchecks the "From" or "To" field values the minimum and maximum coordinate of the model in desired direction is assumed for them, respectively.

   User can define filters on two or three directions, for example the Figure 33 displays the filter configuration for 0 to 5000 m in "X" direction and 3000 to 10000 m in "Y" direction.

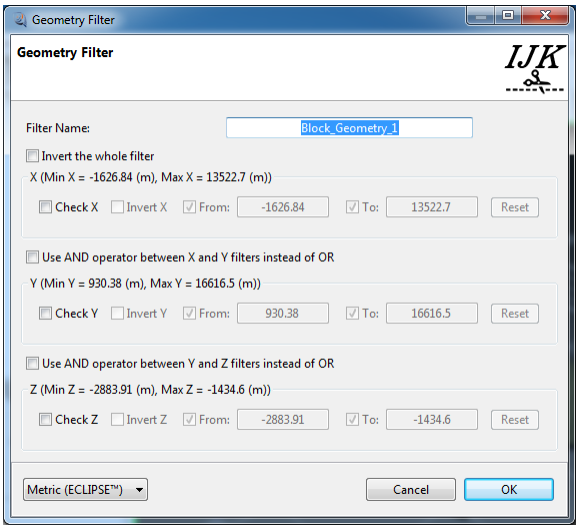


Figure 30 Adding Block Geometry Filter

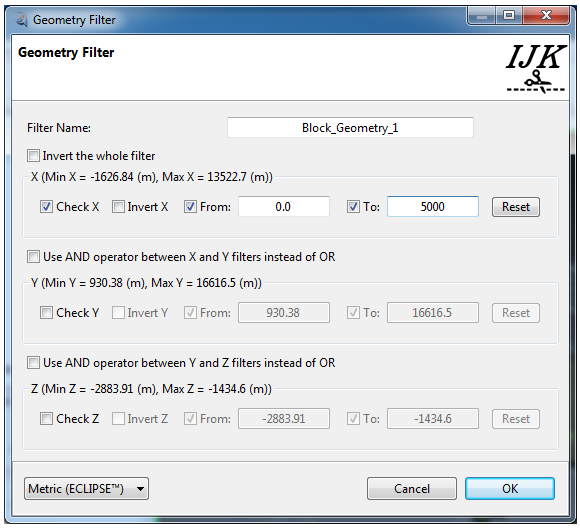


Figure 31 Adding Block Geometry Filter from 0 to 5000m in "X" Direction

In such cases the sum of both filters are added to the 3D view (Figure 34); if user checks the "AND operator" check box (Figure 35) only grid cells which are exist in both filters are displayed such as Figure 36. For having invert filters the "invert" check box either for each direction or for whole filters must be checked. The Figure 37 displays the invert filter of the Figure 36 filters. There is the unit system combo box in this dialog, in which the user can select desired unit system including Field, Metric, and SI.

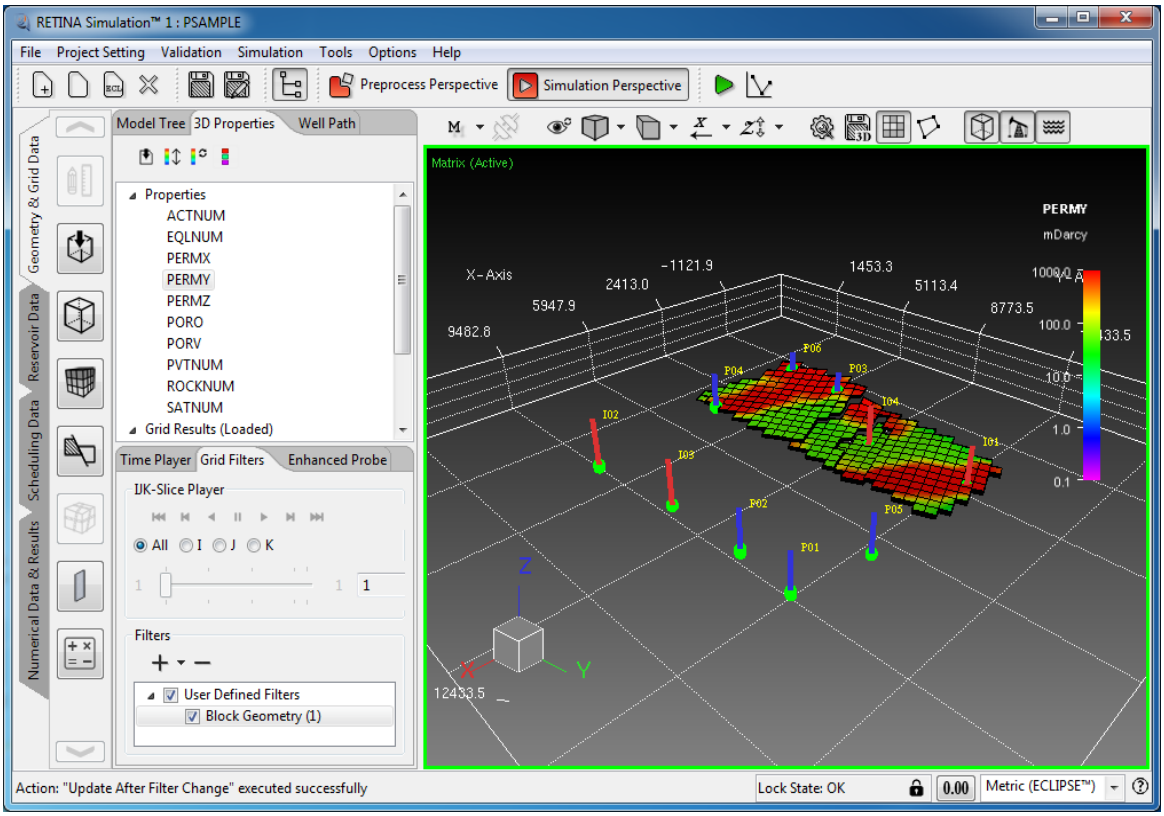


Figure 32 Applied Block Geometry Filter from 0 to 5000m in "X" Direction on 3D View

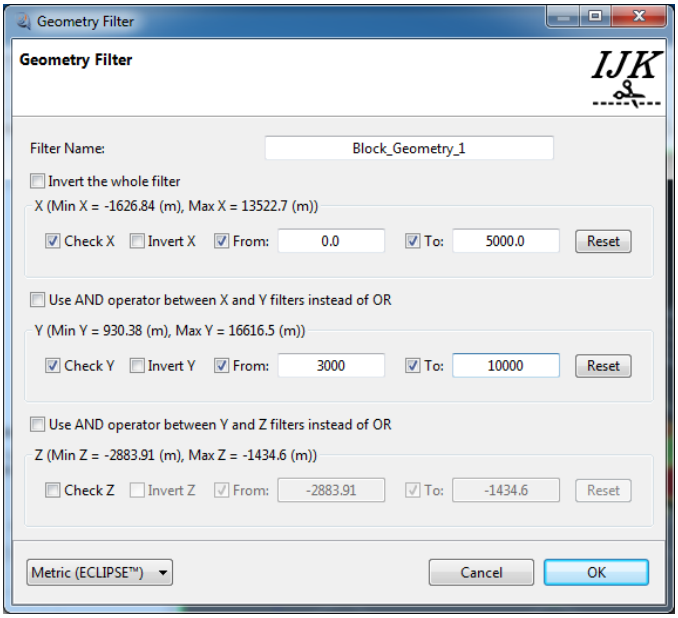


Figure 33 Adding Block Geometry Filter from 0 to 5000m in "X" Direction and from 3000 to 10000m in "Y" Direction

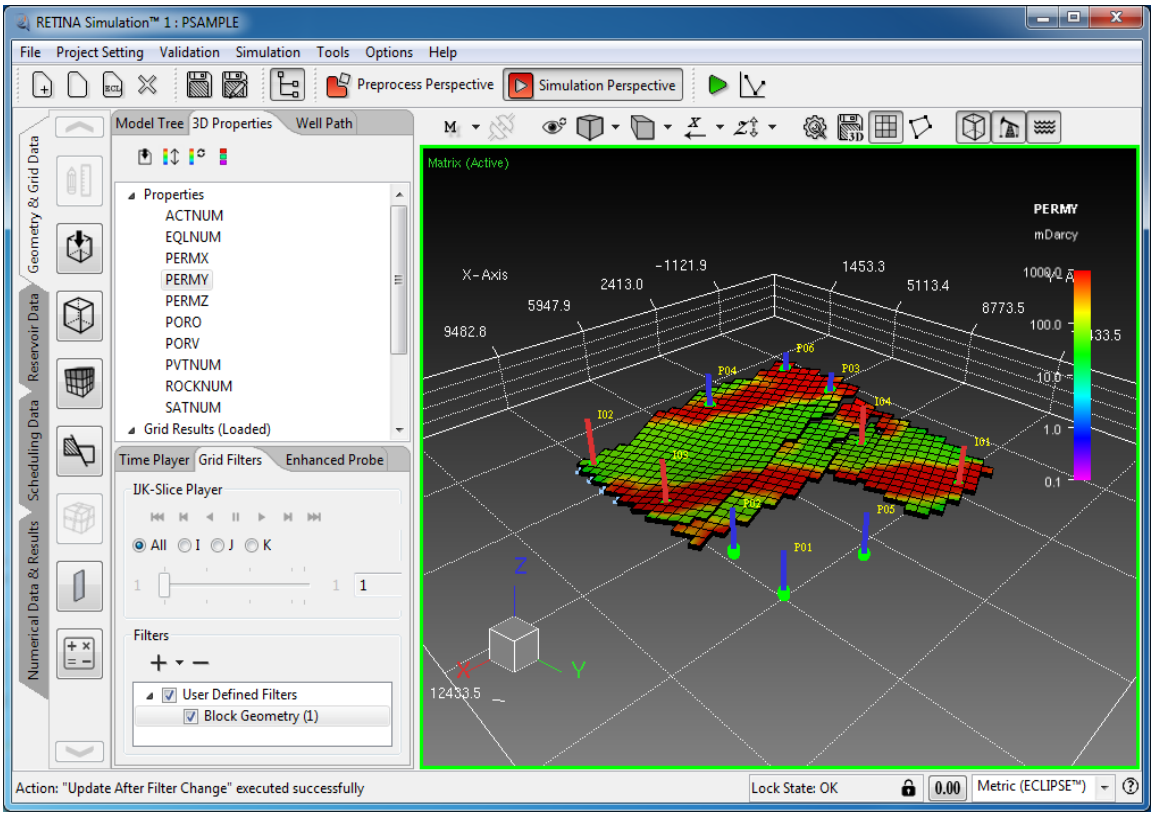


Figure 34 Applied Block Geometry Filter from 0 to 5000m in "X" Direction and from 3000 to 10000m in "Y" Direction on 3D View

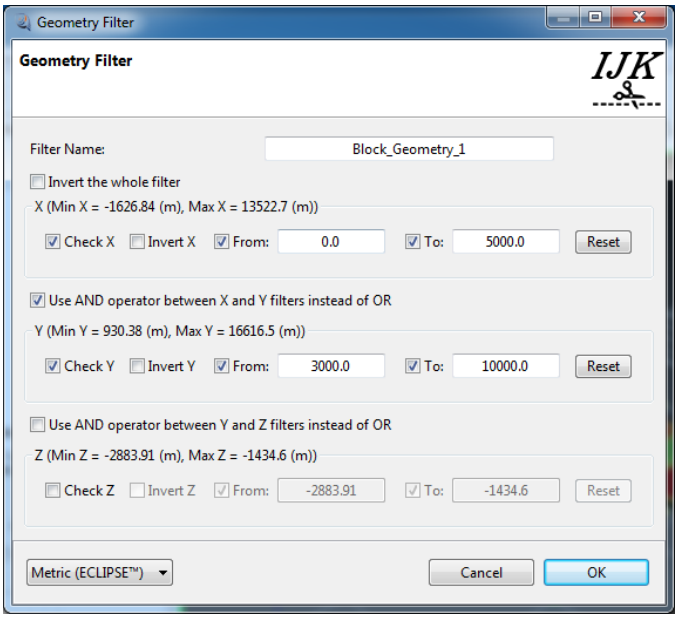


Figure 35 Adding Block Geometry Filters with AND Operator

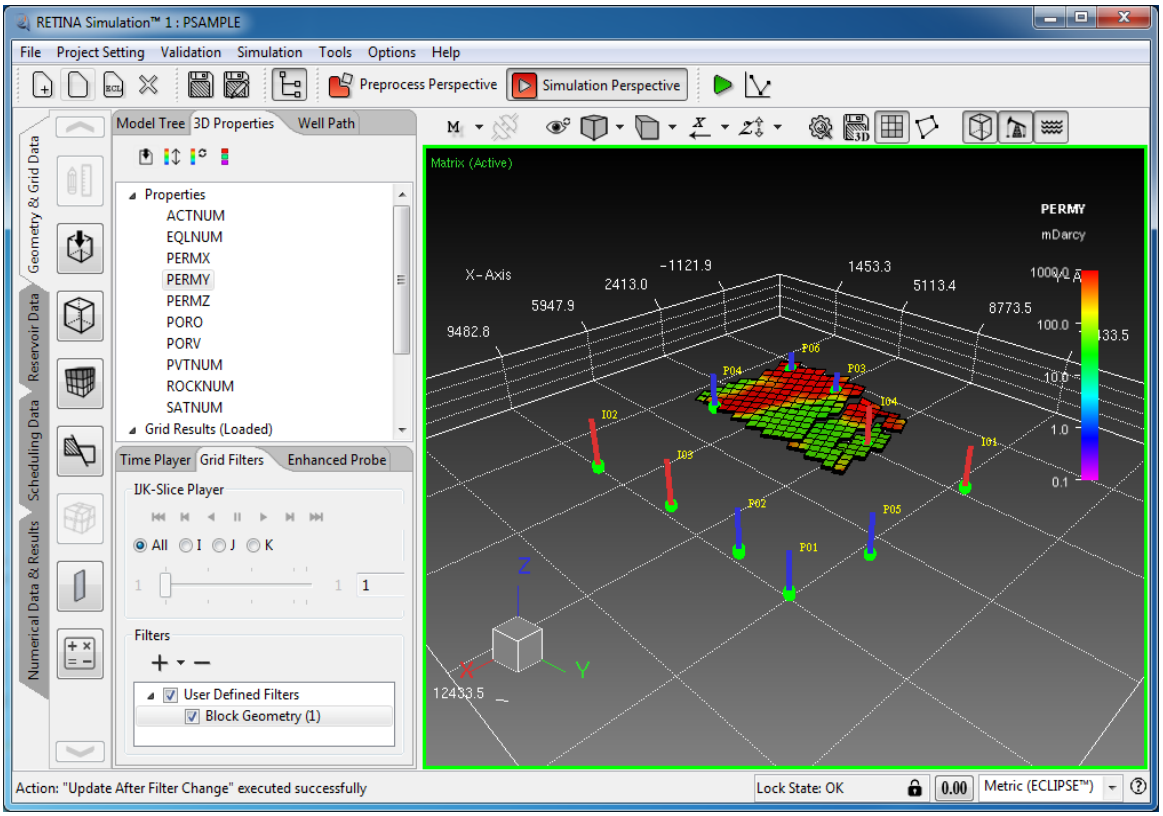


Figure 36 Applied Block Geometry Filters with AND Operator on 3D View

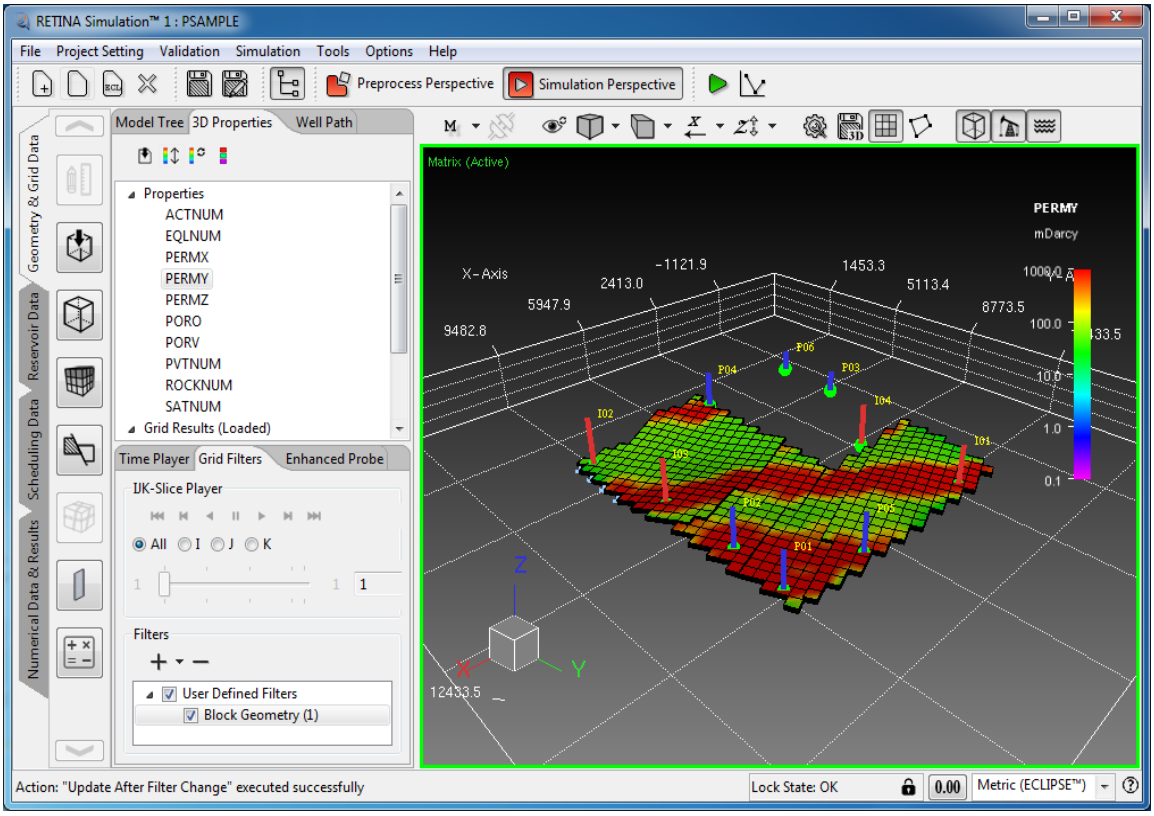


Figure 37 Applied Inver Block Geometry Filter

3. *Block Polygon Filter*: for creating a polygon filter user must be pick the polygon icon "  " from the 3D view toolbar (Figure 38a), and draw polygons by clicking (Figure 38b). User can draw multiple polygons (Figure 38c). Double clicking in each position will close the polygon. After creating all desired polygons, user can make the polygon filter (Figure 38d) using "Add Polygon Filter" which is available in "Filter" section. The name of the "Block_Polygon_##" is assigned to the defined block polygon filter, by default; this name is editable by double clicking on the created filter. If user wants to exit polygon drawing status the "Esc" key must be used. Figure 38 displays the polygon drawing on 3D view.

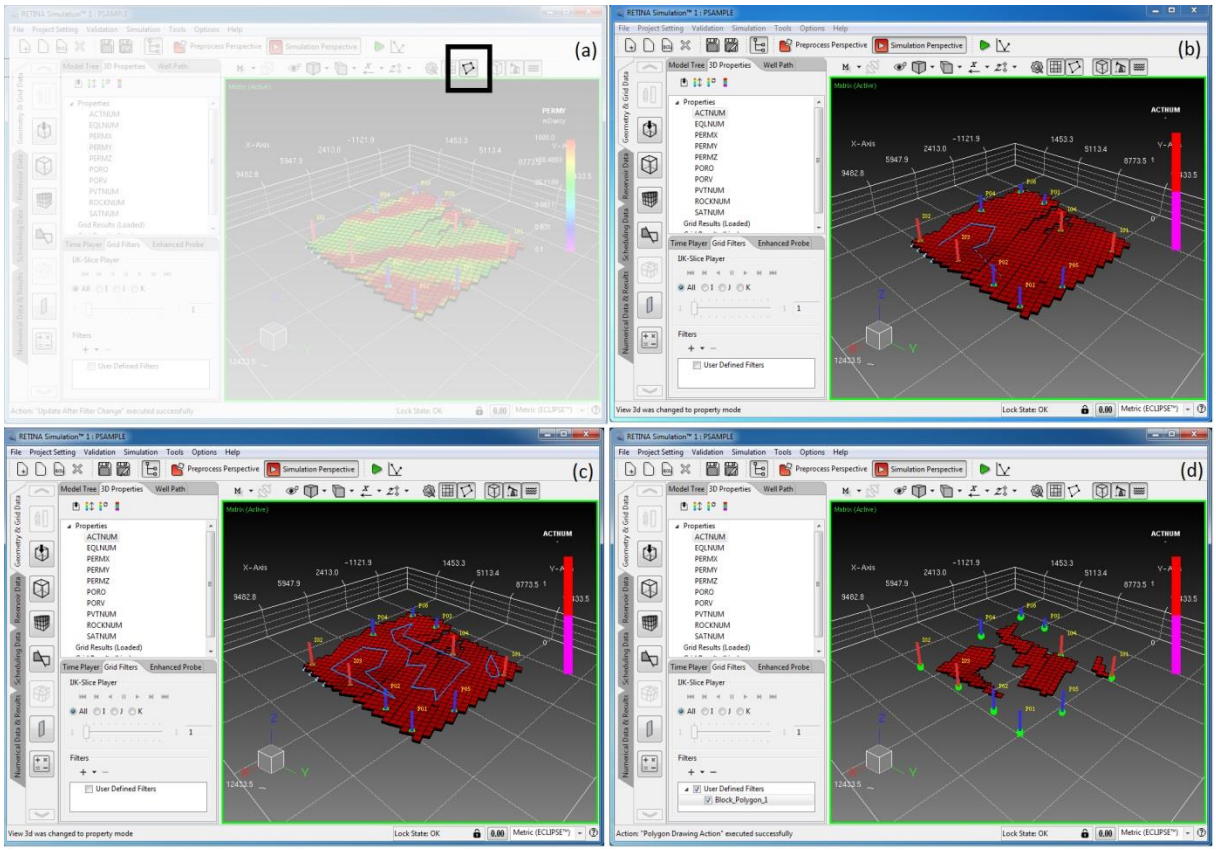


Figure 38 Creating Polygon Filter

Block Property Filter: the dialog of the Figure 39 is displayed if user wants to add block property filter. Initially none of the properties are selected which means no filter is added to 3D view. The name of the "Block_Property_##" is assigned to the defined block property filter, by default; this name is editable. For adding filter on each property, user must select it using "Property to Filter" combo box; if the property is schedulable user should select the desired date using "Property Date" combo box. For adding filter on each property, user should just check the related box and enter the minimum and maximum values for the property. For example the Figure 40 displays the configurations for adding a

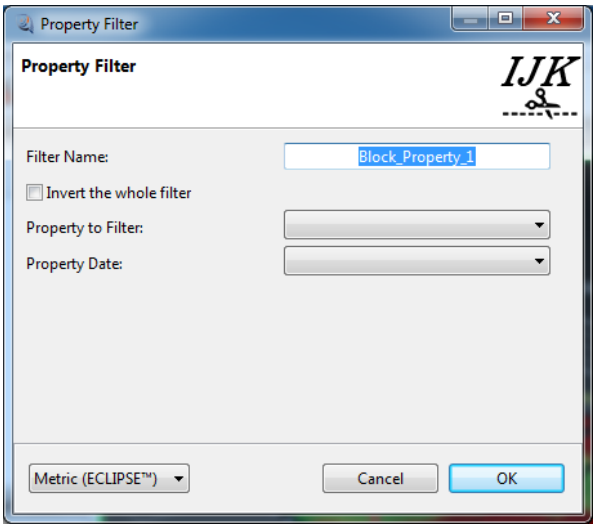


Figure 39 Adding Block Property Filter

filter of permeability between 2000 and 4905 md. Clicking on the OK button, filter will be applied on 3D view (Figure 41). Note that the editing of the defined filter is allowed by double clicking on the added one in the Filters section. If user unchecks the minimum or maximum limits of the filter, the minimum and maximum values of the property in the model are assumed for them, respectively. The invert filter check box is available also, for this type of filtering.

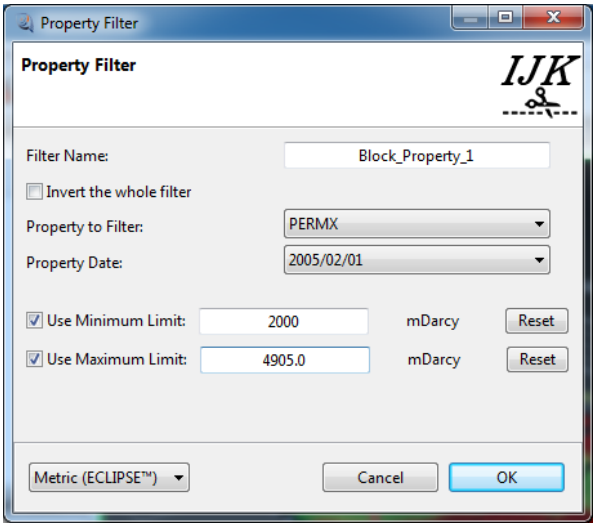


Figure 40 Adding Block Property Filter on X-Permeability

4.

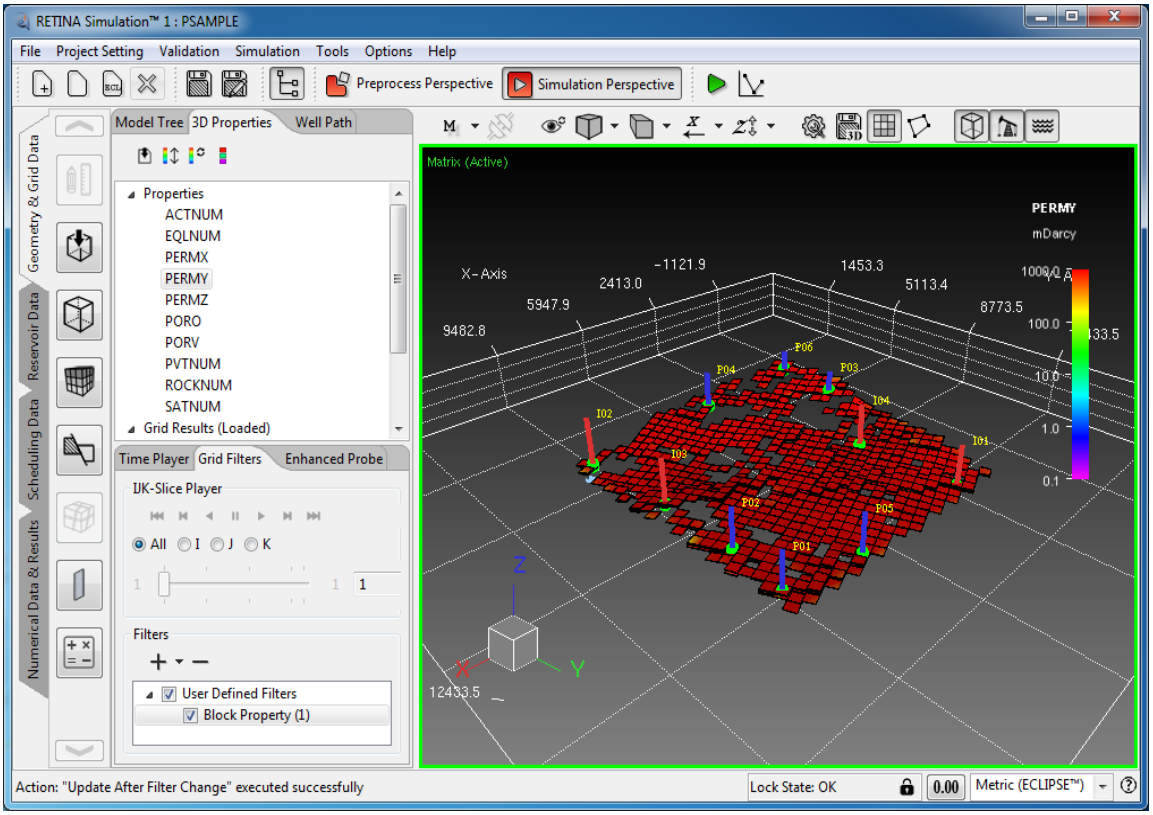


Figure 41 Applied Permeability Filter on 3D View

5. *Block Logical Filter*: This type of filter is used for performing logical operations of "AND", "OR", or "NOT". For performing logical filter at least two filters are required. The dialog of the Figure 42 is displayed if user wants to add block logical filter.

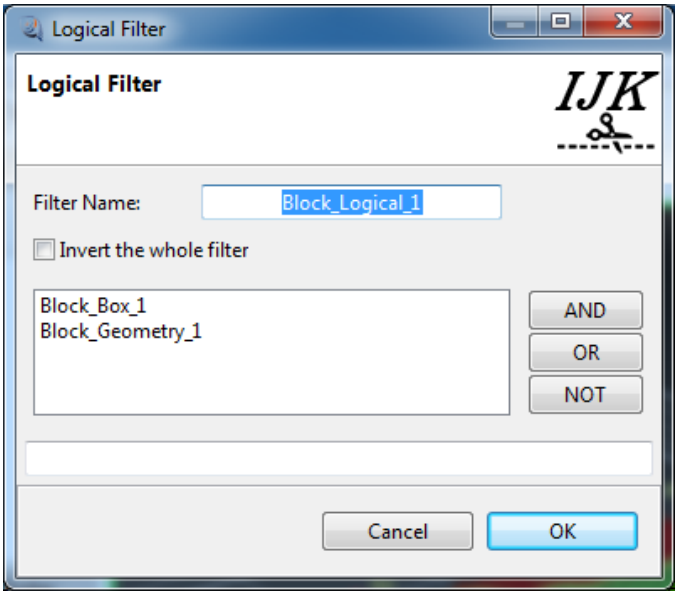


Figure 42 Adding Logical Filter

At first user must select a filter, a logical operation and then another filter. For example Figure 43 displays the Block_Box_1 or Block_Geometry_1 Filters i.e. both of them is displayed in 3D view. The name of the "Block_logical_##" is assigned to the defined block property filter, by default; this name is editable. Note that since the "NOT" operation is not a binary operation so it must be used before desired filter which you want to exclude it. Figure 44 displays the using "NOT" logical operation. The multiple and selective remove options are available for created filters.

Note: The sign of "OR" in script is ||, the sign of "NOT" and "AND" is "!" and "&&" respectively.

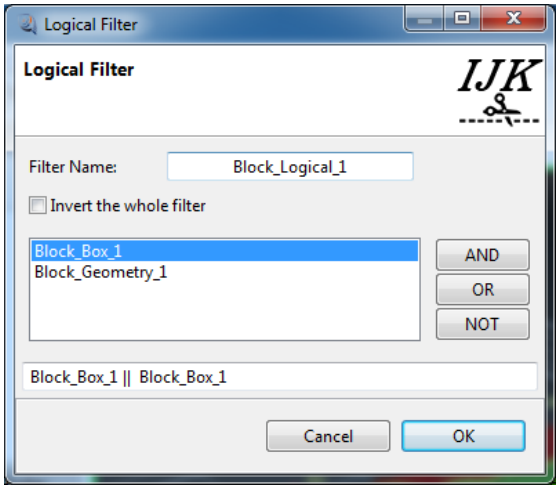


Figure 43 Adding Logical Filter with "OR" Operation

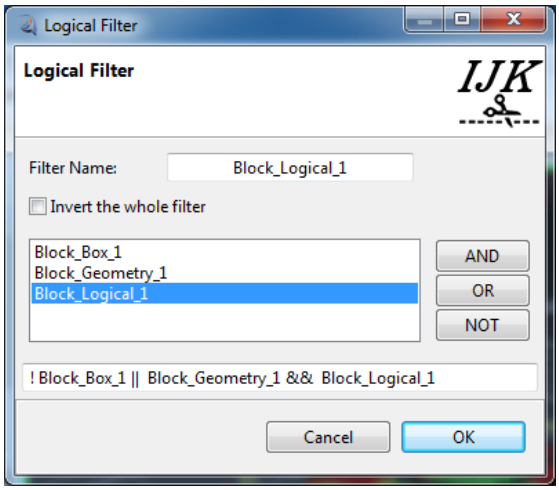


Figure 44 Using "NOT" Logical Operation

The "Enhanced Probe" tab is used to show the block information such as IJK index, global index and grid properties. In this tab user can insert node Id and press apply to get the related information. Also, user can select a nod from the 3D view of the model, by right clicking on specific node.

If user selects a node from 3D view by click, a pink sphere will appear on the selected node to indicate the selected grid and related information is displayed in status bar messages. Figure 45 shows the "Enhanced Probe" tab, status bar and the selected node on 3D view.

Note: The left panel hide or show actions can be performed using  *button.*

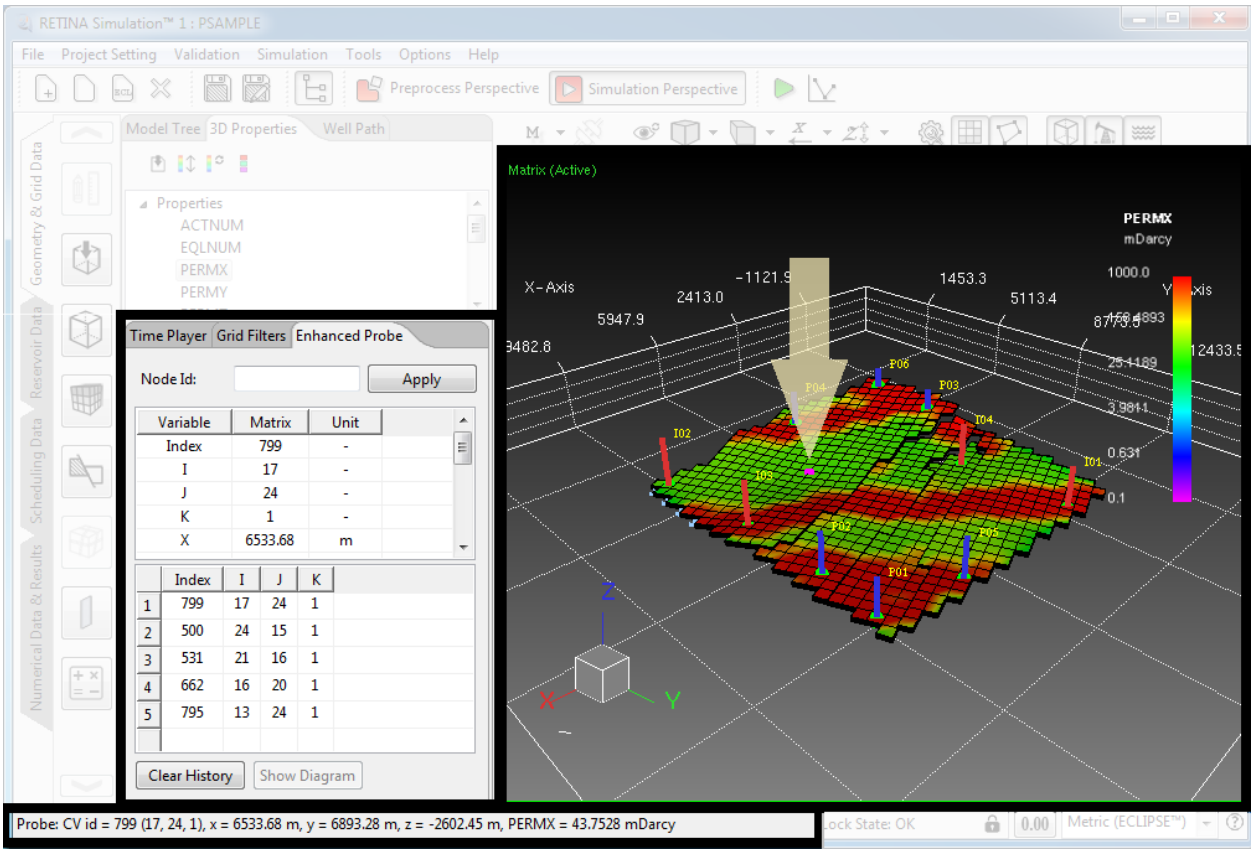


Figure 45- Enhanced Probe Tab

2-4-3- Well Path

There are three import options in this tab:

1. Import Well Path
2. Import Well Event
3. Import Well Observed Data

Figure 46 displays the well path tab in the left panel, user can import desired data using .

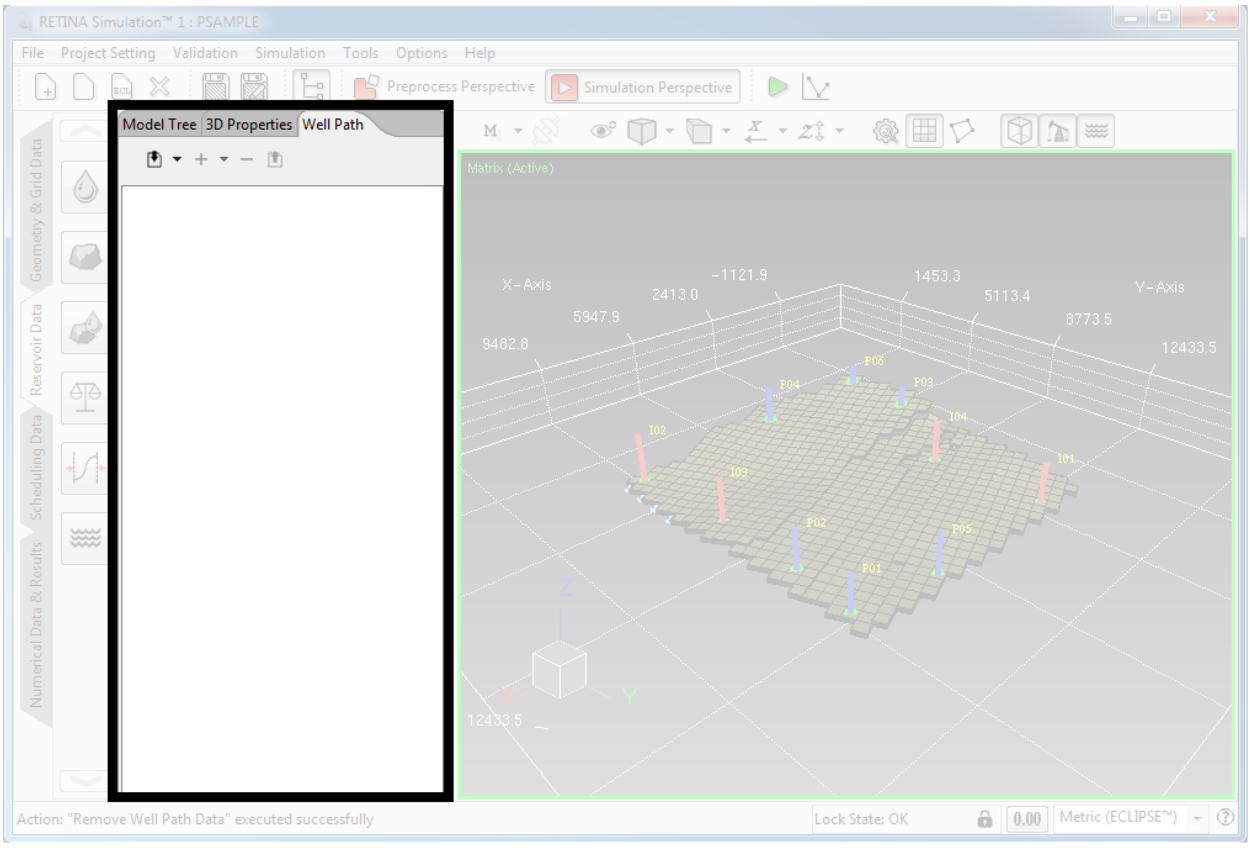


Figure 46 Well Path Importing Facility

1- Import Well Path:

Import of well path data is the first step in the well import process. The deviation file can be imported well by well, for several wells together or for multiple wells in one file or even for multiple separated files. If multiple wells are included in a single file there are a series of options for determining how the settings for different wells are distinguished. The deviation file should be in ASCII format with the data organized in columns.

There are eight sections in the import well path dialog including: File Path, Hint, Format, Column and Unit, Reference depth, Vertical Direction, Options, and Preview for the first 200 lines of the imported data.

In the file path section the path for the imported deviation file is displayed. There is a hint section in this dialog which informs user that the "%" symbol in a well name donates a parent/branch relationship. For example a well named P1%B1 indicates that it is a branch from well P1 and the name P1%B1%B2 indicates a branch from well P1%B1 well.

The survey format and calculation method can be set in this section. Data can be of several survey formats:

- Measured depth, inclination, azimuth (MD_INC_AZIM)
  - *Angles can be Deg or Rad, azimuth (in degrees) should be between -360 and 360, inclination (in degrees) between 0 and 90, 0 being vertical, 90 horizontal.*
- X-offset, Y-offset, True vertical depth (DX_DY_TVD)
- X, Y, True vertical depth (X,Y,TVD)
- X, Y, Z (X_Y_Z)
- Measured depth, X-offset, Y-offset, True vertical depth (MD_DX_DY_TVD)
- Measured depth, X, Y, True vertical depth (MD_X_Y_TVD)
- Measured depth, X, Y, Z (MD_X_Y_Z)

There are three methods of "Linear", "Minimum Curvature", and "Automatic", to calculate the well trajectory using the imported data. If one of the surveys formats which include measured depth is selected, the method is not selectable.

Note that only three columns of the data are enough for calculating well path. User can assign units and number of columns in this section. The reference coordinate and Kelly bushing depth must be entered in "Reference depth" section; if user leaves all the reference fields blank a warning message is displayed. The vertical directions of TVD or Z related to the selected survey format can be set upward or downward in the "Vertical Direction" section. There another section in this dialog titles "Options" which includes "Column and Well Name" and "Comments and Prefix" sub-sections. User can set the columns as number or string by entering N or S for each column, respectively; the well name option which indicates where the well name has been entered in the deviation file is available for user. In the sub-section of "Comments and Prefix" the comment sign, well name line prefix, and missing value sign fields are available for user to editing, also. User can see the first 200 lines data preview in the "Import Well Path" dialog. Figure 47 displays the Import Well Path dialog for a typical deviation file. If user press "OK for all" the setting are applied to all of the imported wells. The imported wells are displayed in well path tab and user can check their check boxes for displaying the in the 3D view. Figure 48 displays the imported well path in 3D view. The multiple selection and remove options are available in this list.

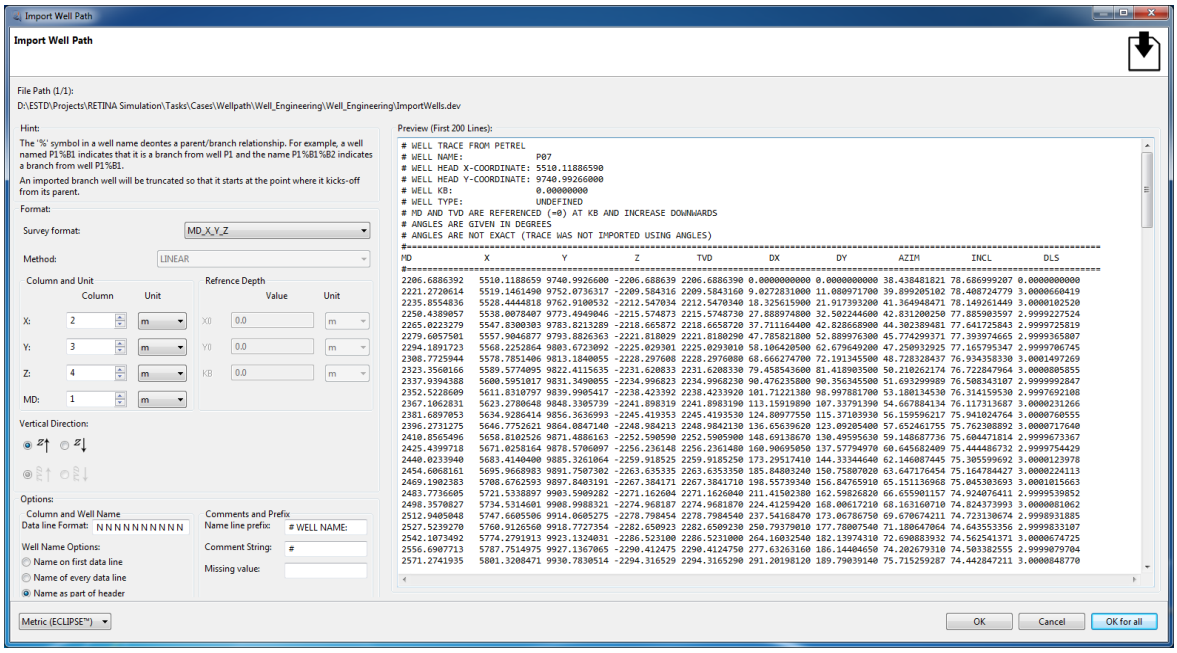


Figure 47 Import Well Path Dialog

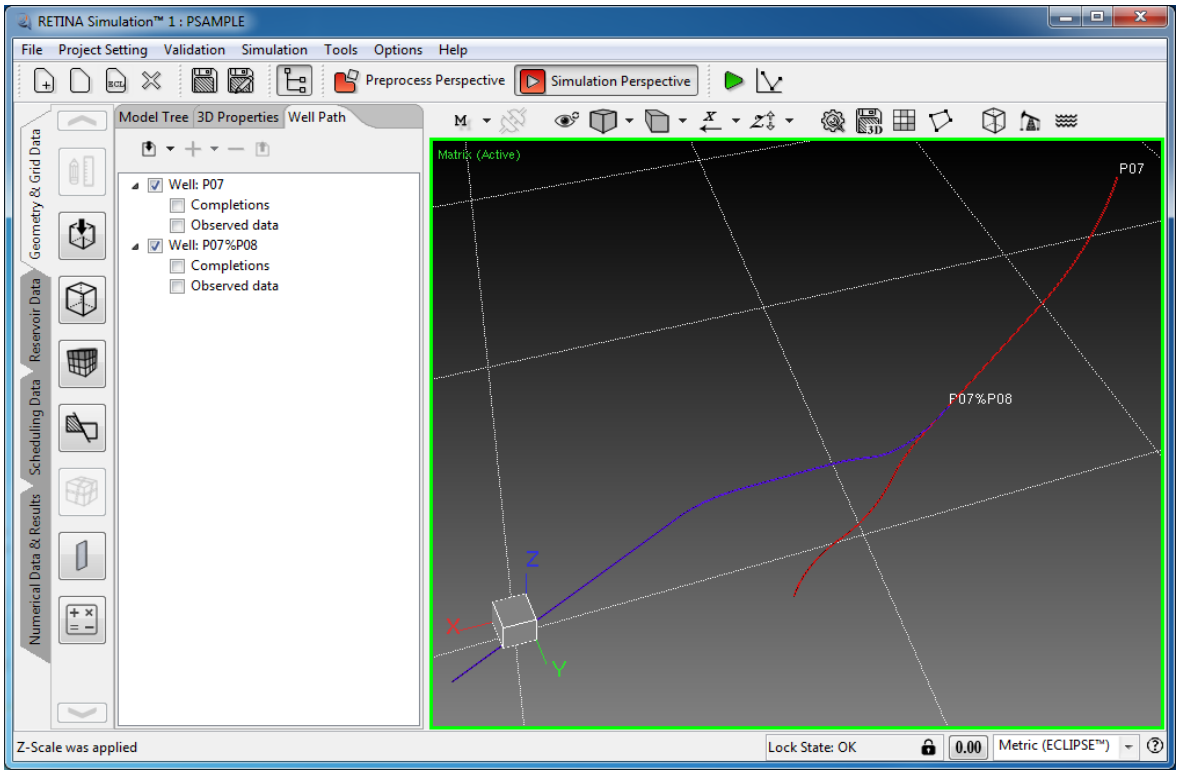


Figure 48 Visualizing the Imported Well Paths

2- Import Event File:

The completion events are stored in the completions node under each well node. Choose the import event file option from well path tab import icon; RETINA can import well event data (ASCII) (*.ev) file. Events listed in Table 1 have been supported in current version of the RETINA Simulation™.

Table 1 Event and Completion Names

Event name in File	Completion name
Perforation	Perforation
Barefoot	Perforation
Acidize	Stimulation
Stimulate	Stimulation
Frac	Stimulation
Rework	Perforation
Plug	Plug
Squeeze	Squeeze
Welltest	Well test
Welltest-PI	Well test
Welltest-PM	Well test

The format of the events in the event file should be as following:

- *Perforation, barefoot and rework:*
  The WELLNAME keyword is followed by the well. The first column of data is the date at which the events were performed. The following information is the event type, the top Mdepth, the base Mdepth, the wellbore diameter, the skin, optional lump connection, and optional relative permeability scaling.
- *Acidize, stimulate, and frac:*
  These events modify the damaged skin used in the connection factor calculation over a specified measured depth. WELLNAME is present afterward Date, top and bottom Mdepth and Skin should be entered.
- *Plug and squeeze (cement squeeze):*
  These both act to close connections.

- *Squeeze*: closing connections over a specified range. The squeeze has two items following the squeeze name namely top and bottom measured depth.
- Plug: closing all connections below a given measured depth. The plug has only one data item following the plug name, namely top Mdepth.

- *Welltest:*

  Welltest pacifies the Kh product over the entire well. Following the Welltest name is the Kh value. DATE EVENT Kh skin start-Mdepth stop-Mdepth is after that.
- *Welltest-PI*

  DATE, EVENT, pi-liquid, and pi-gas must be entered respectively, in this event.
- *Welltest-PM*

  DATE, EVENT, pi-multiplier, start-Mdepth, and stop-Mdepth must be entered respectively in this event.

The Import Well Event dialog (Figure 49) is displayed after selecting .ev file using well path tab import facility.

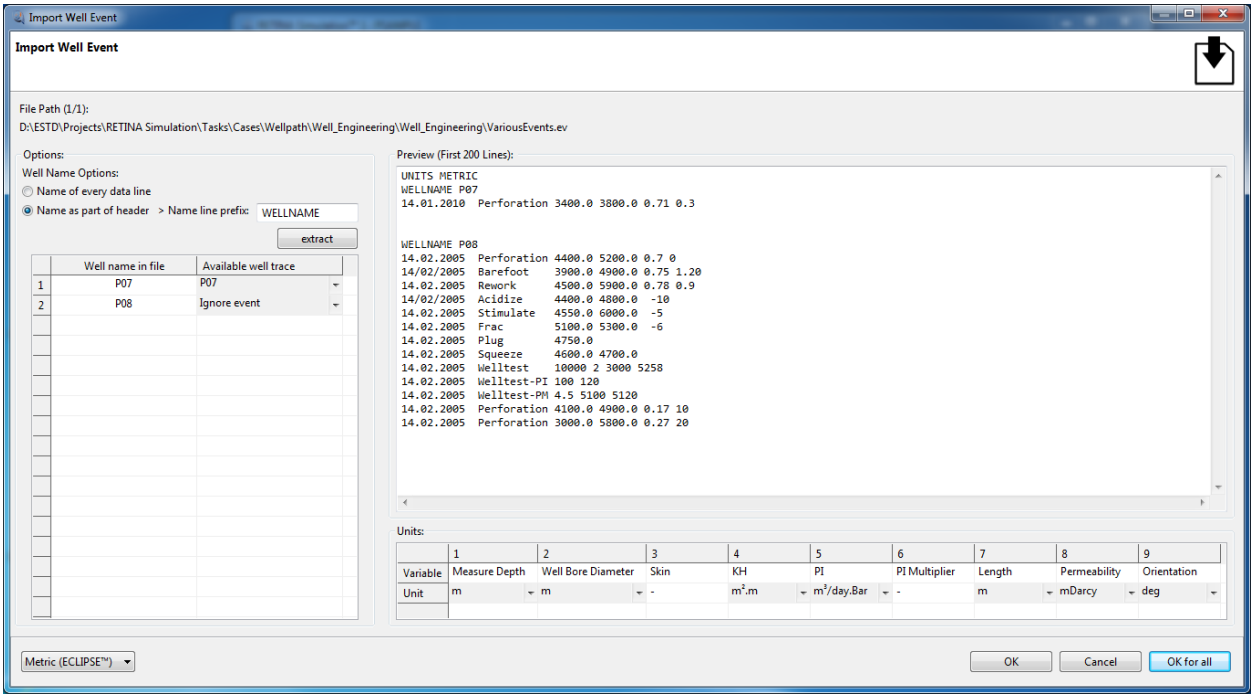


Figure 49 Import Well Event Dialog

The import well event dialog consists of four main sections including File Path, Options, Units, and Preview.

In the file path section the path for the imported event file is displayed. User can specify that the well name is repeated every line or as part of header. If user chooses “Name as part of header” the name line prefix must be entered. There is a table of imported wells; user can ignore or map the available wells for importing. There is a section titled “Unit” in this dialog in which user can set the desired unit for each variable independently. The first 200 lines of the imported file are displayed in Preview section. If user press “OK for all” the setting are applied to all of the imported wells. The imported events are displayed in well path tab and user can check their check boxes for displaying the in the 3D view. Figure 50 displays the imported well path in 3D view. The multiple selection and remove options are available in this list. User can define desired event using “+” button.

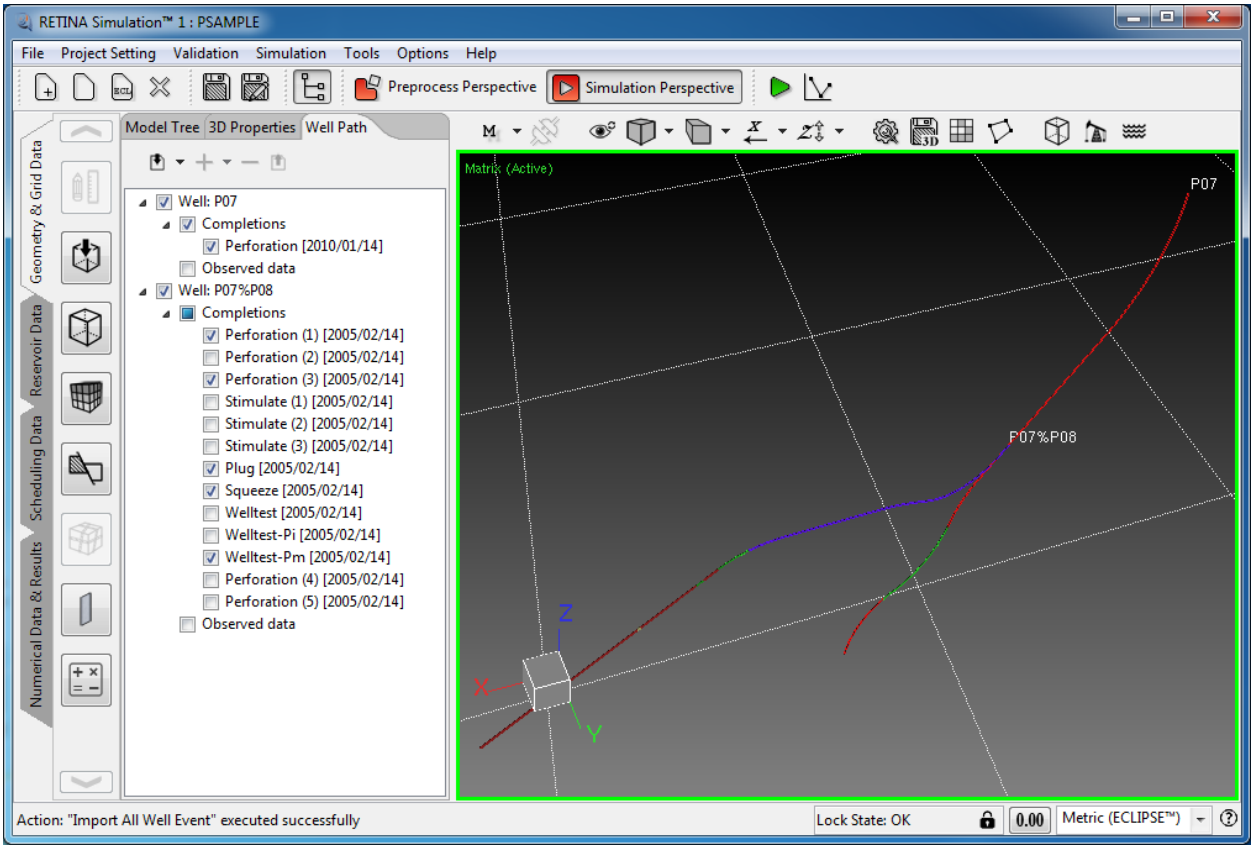


Figure 50 Visualizing the Imported Well Events

3- Import Observed Data File:

The Observed data is stored in the observed data node under each well node. Choose the import observed data option from well path tab; RETINA can import well event data (ASCII) (*.vol) file.

The import observed data dialog consists of four main sections including File Path, Hint, Options, and Preview as displayed in Figure 51.

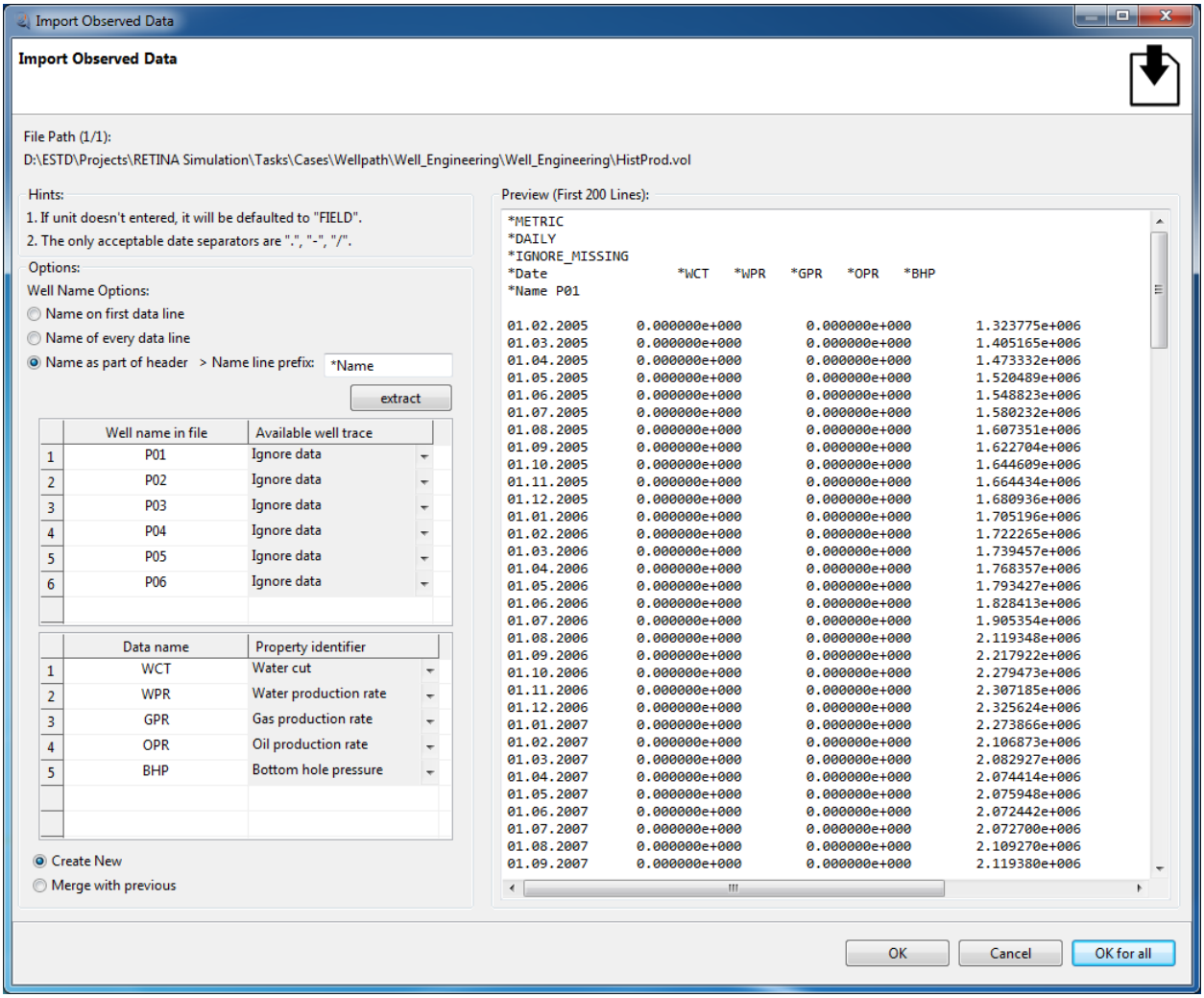


Figure 51 Import Observed Data Dialog

In the file path section the path for the imported .vol file is displayed. User can specify that the well name is repeated every line or as part of header. If user chooses “Name as part of header” the name line prefix must be entered. There is a table of imported wells and a separate table of imported historical data; user can ignore or map the available wells and also property identifiers.

If user press “OK for all” the setting are applied to all of the imported wells. The imported observed data are displayed in well path tab. The multiple selection and remove options are available in this list.

4- Convert Imported Data to RETINA Simulation™:

User can export the imported data to RETINA using . The Figure 52's dialog, is displayed in case of export data.
If the well path has some intersection with the model grid (Figure 53) the well can be created in the RETINA well manager dialog with related keywords. The exported wells and related keywords are displayed in the Figure 54.

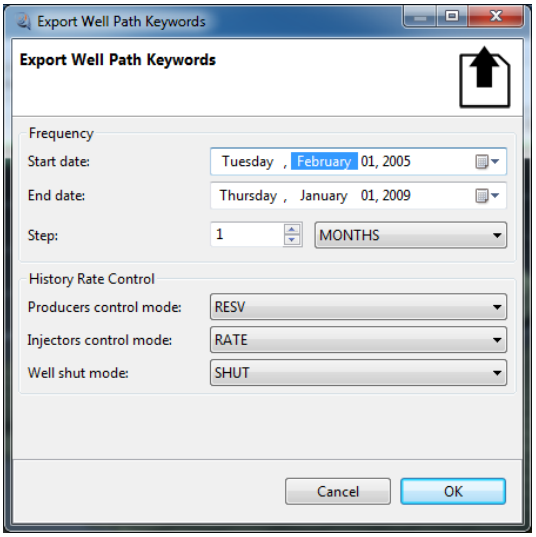


Figure 52 Export Well Path Keywords Dialog

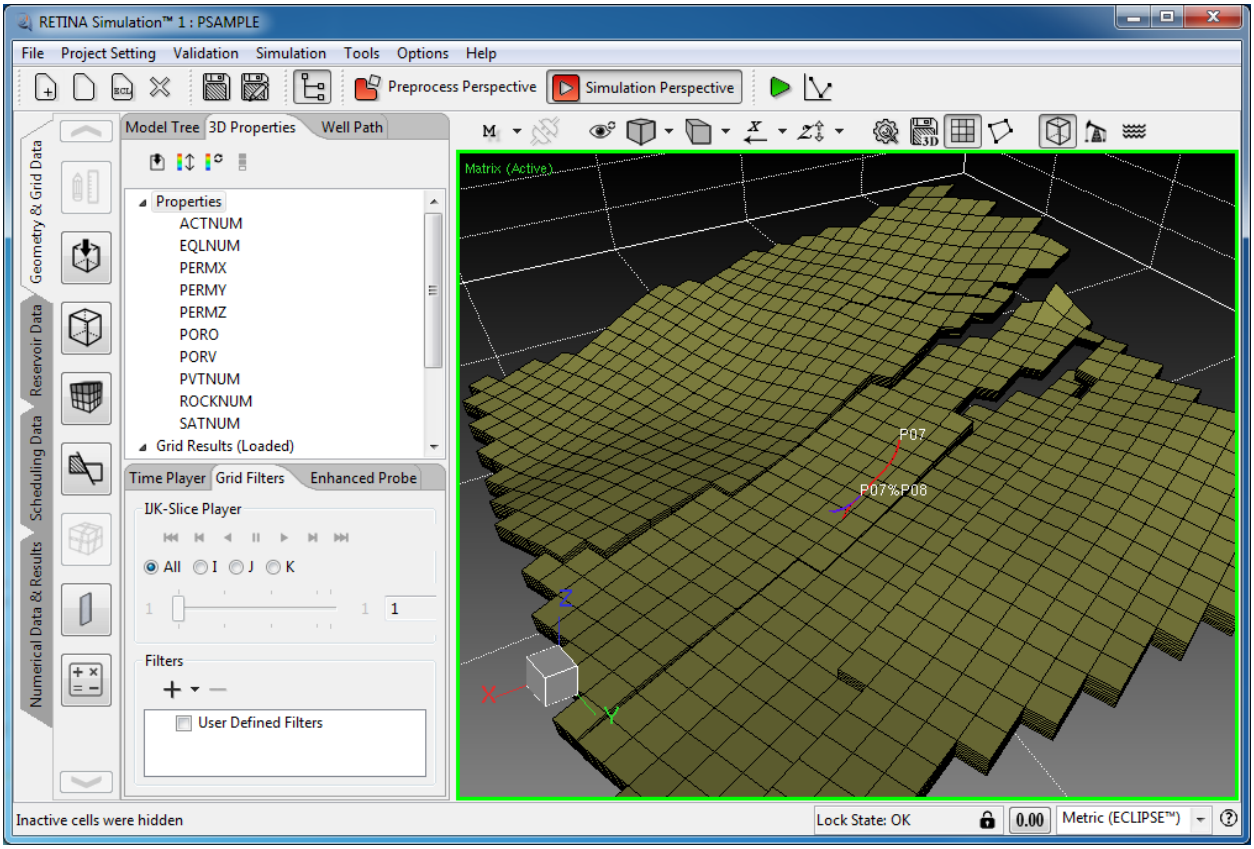


Figure 53 Intersecting Model Grids and Imported Well Path

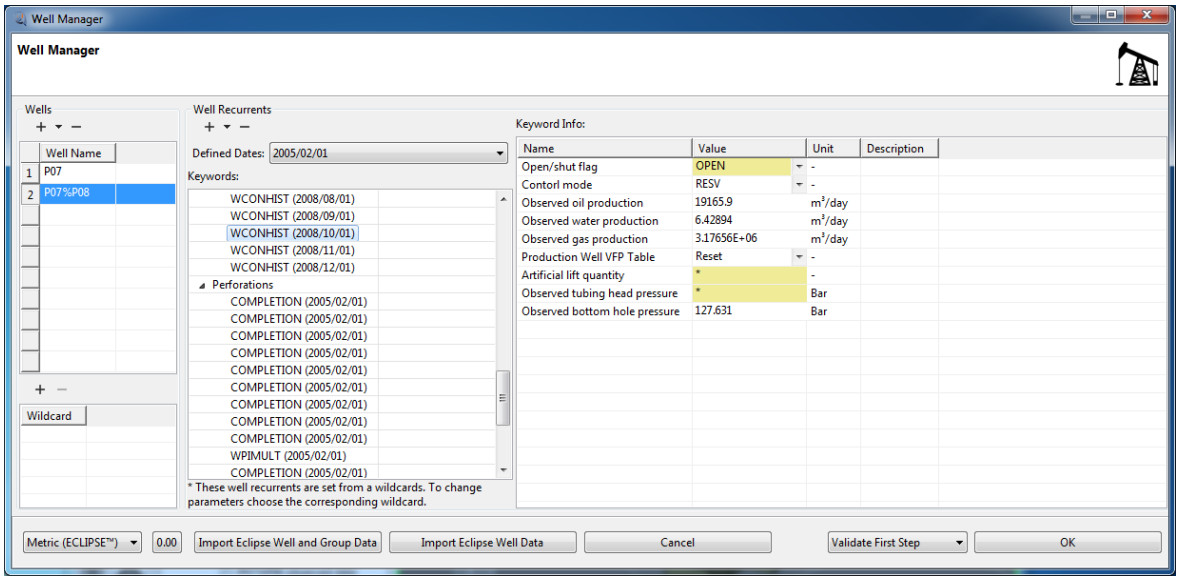
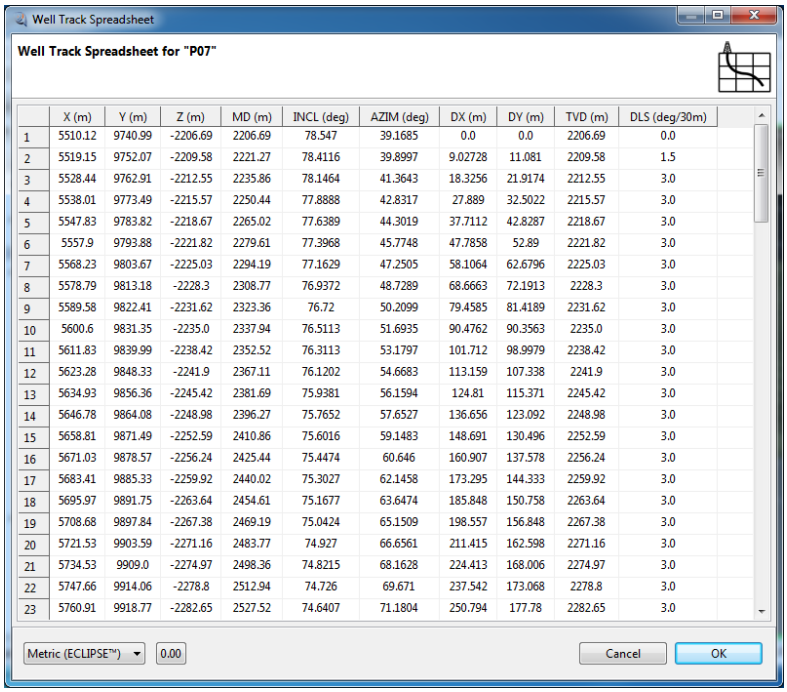


Figure 54 Exported Well Path Keywords

Each component of the well path tab has its own data which user can access and edit the desired data by double clicking on the node. For example double clicking on imported well name in well path tab will open the Figure 55's dialog.



	X (m)	Y (m)	Z (m)	MD (m)	INCL (deg)	AZIM (deg)	DX (m)	DY (m)	TVD (m)	DLS (deg/30m)
1	5510.12	9740.99	-2206.69	2206.69	78.547	39.1685	0.0	0.0	2206.69	0.0
2	5519.15	9752.07	-2209.58	2221.27	78.4116	39.8997	9.02728	11.081	2209.58	1.5
3	5528.44	9762.91	-2212.55	2235.86	78.1464	41.3643	18.3256	21.9174	2212.55	3.0
4	5538.01	9773.49	-2215.57	2250.44	77.8888	42.8317	27.889	32.5022	2215.57	3.0
5	5547.83	9783.82	-2218.67	2265.02	77.6389	44.3019	37.7112	42.8287	2218.67	3.0
6	5557.9	9793.88	-2221.82	2279.61	77.3968	45.7748	47.7858	52.89	2221.82	3.0
7	5568.23	9803.67	-2225.03	2294.19	77.1629	47.2505	58.1064	62.6796	2225.03	3.0
8	5578.79	9813.18	-2228.3	2308.77	76.9372	48.7289	68.6663	72.1913	2228.3	3.0
9	5589.58	9822.41	-2231.62	2323.36	76.72	50.2099	79.4585	81.4189	2231.62	3.0
10	5600.6	9831.35	-2235.0	2337.94	76.5113	51.6935	90.4762	90.3563	2235.0	3.0
11	5611.83	9839.99	-2238.42	2352.52	76.3113	53.1797	101.712	98.9979	2238.42	3.0
12	5623.28	9848.33	-2241.9	2367.11	76.1202	54.6683	113.159	107.338	2241.9	3.0
13	5634.93	9856.36	-2245.42	2381.69	75.9381	56.1594	124.81	115.371	2245.42	3.0
14	5646.78	9864.08	-2248.98	2396.27	75.7652	57.6527	136.656	123.092	2248.98	3.0
15	5658.81	9871.49	-2252.59	2410.86	75.6016	59.1483	148.691	130.496	2252.59	3.0
16	5671.03	9878.57	-2256.24	2425.44	75.4474	60.646	160.907	137.578	2256.24	3.0
17	5683.41	9885.33	-2259.92	2440.02	75.3027	62.1458	173.295	144.333	2259.92	3.0
18	5695.97	9891.75	-2263.64	2454.61	75.1677	63.6474	185.848	150.758	2263.64	3.0
19	5708.68	9897.84	-2267.38	2469.19	75.0424	65.1509	198.557	156.848	2267.38	3.0
20	5721.53	9903.59	-2271.16	2483.77	74.927	66.6561	211.415	162.598	2271.16	3.0
21	5734.53	9909.0	-2274.97	2498.36	74.8215	68.1628	224.413	168.006	2274.97	3.0
22	5747.66	9914.06	-2278.8	2512.94	74.726	69.671	237.542	173.068	2278.8	3.0
23	5760.91	9918.77	-2282.65	2527.52	74.6407	71.1804	250.794	177.78	2282.65	3.0

Figure 55 Well Tack Spread Sheet

3- How to Use Data Toolbar

After creating a new project (refer to section 1-1) and defining desired unit system, user should use data toolbar to define grid geometry, reservoir, wells and output data.

3-1- Using Geometry & Grid Data Tab

This tab which includes seven buttons (refer to section 2-3-1) is used for defining the reservoir dimensions (number of grids in each directions of X, Y, Z), grid data, grid properties, Fault Face or connections and fracture (dual) options.

3-1-1- Define Grid Data

Define Grid Data window (Figure 56) is opened, using the button of initialize simple grid . In this dialog the number of grids is defined only for center point grid data.
Nx, Ny, Nz indicates number of grids in X, Y, Z directions, respectively.

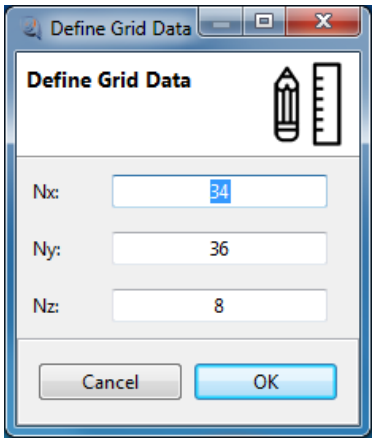


Figure 56- Define Grid Data

3-1-2- Grid Data Tab

Pressing OK in the "Define Grid Data" will lead user to center point grid data window which is shown in Figure 57. The DX, DY, DZ and TOPS values are required for this type of geometry. DX, DY and DZ are distance of the two neighbor blocks in X, Y and Z directions, respectively.

A) Center Point Geometry Type

As mentioned before, the DX, DY, DZ and TOPS values are required for this type of geometry. In this type of the geometry, user can define dip for the model, using "Define Dip" button. Define Dip window has been shown Figure 58 in which the axis and shift in Z direction (delta Z) must be entered. For example assume you want to define dip for a model with dimension of 3*3*2 with DX=DY=DZ=1 (Figure 59a), if you define dip in X direction with 0.5 m shift in Z direction Figure 59b will be displayed.

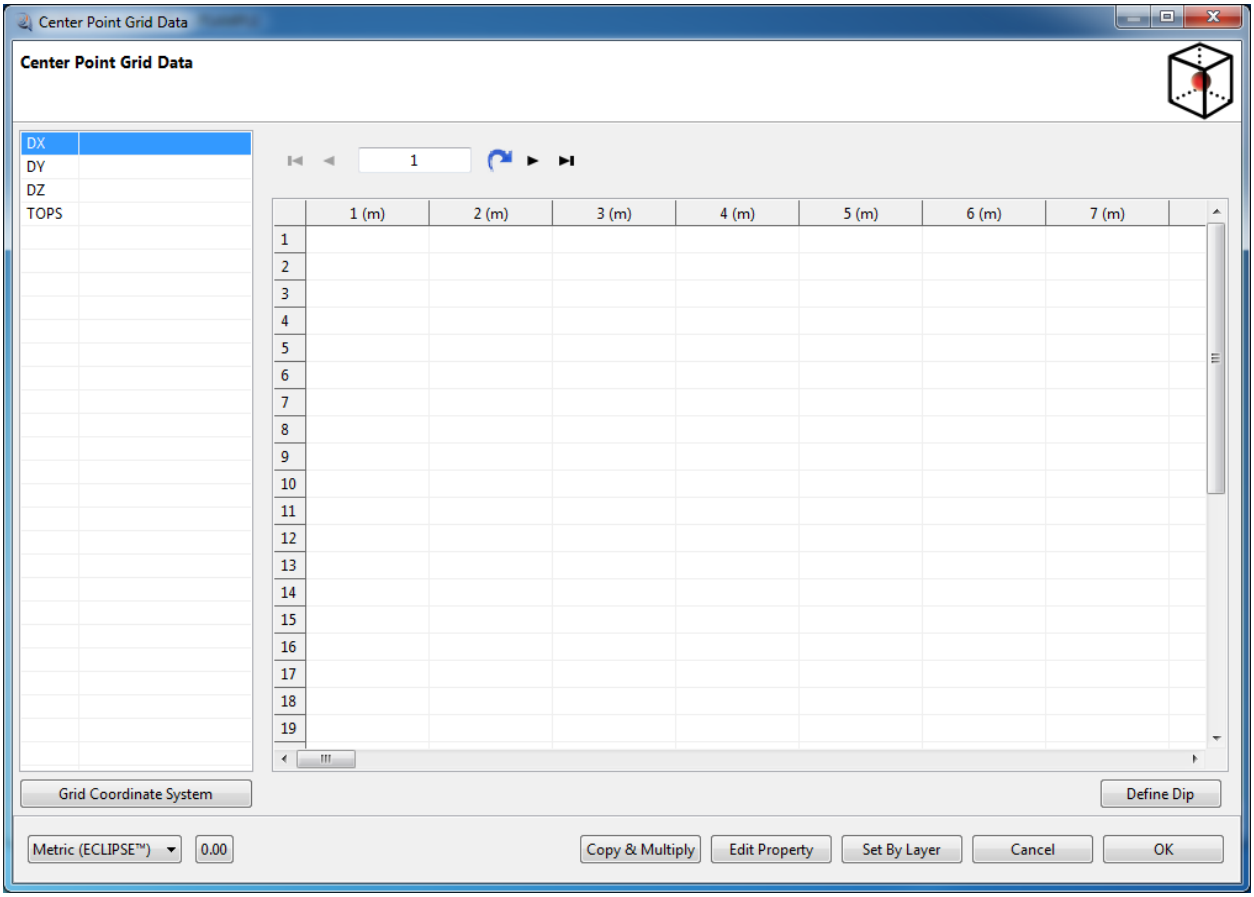


Figure 57- Center Point Grid Data Window

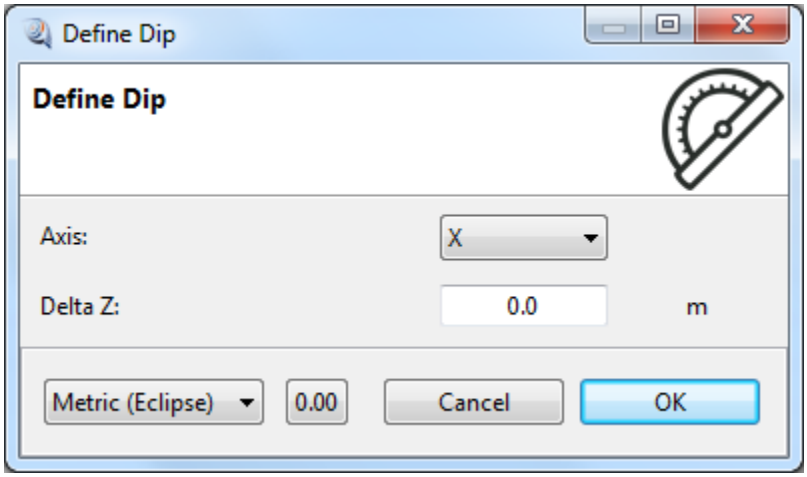


Figure 58 - Define Dip

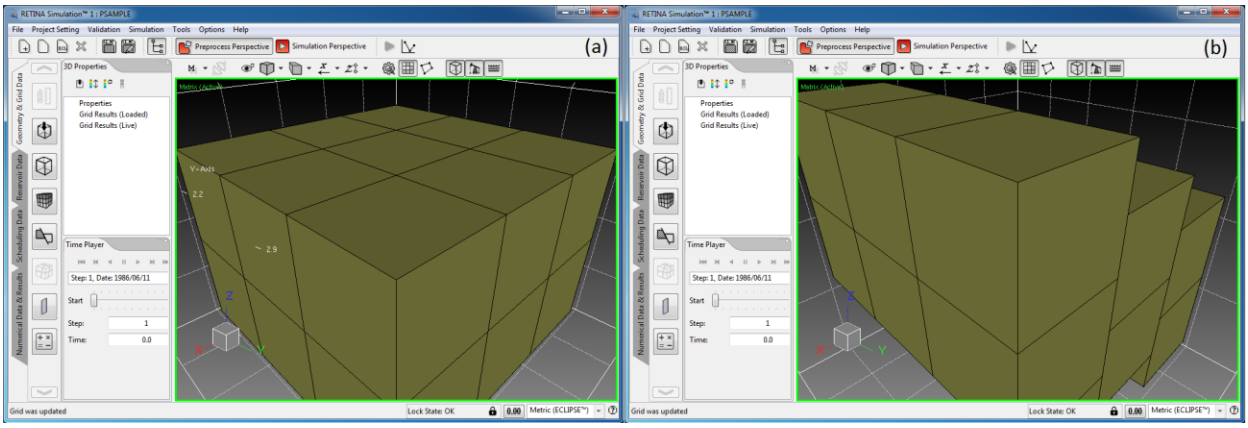


Figure 59- a) A model without Dip, b) Dip in X direction with 0.5 m shift in Z direction

There is a square field in Figure 57, in which user can enter layer number (K=1, 2…) to see or edit the related values of the data. Also user can use arrows to go to the first layer, previous layer, next layer last layer and jump to a specific layer. Jumping will be done after entering the layer number and press the button of (↷). There are following buttons in the "Center Point Grid Data" window:

- Define Grid Coordinate System
- Copy & Multiply
- Edit Property
- Set By Layer
- OK
- Cancel

Define Grid Coordinate System:

The local coordinate system is defined using this button. It includes origin coordinate (X, Y, Z), input type, and axis vectors (Figure 60). The input type can be one of the following items:

- X axis & Y axis
- X axis & Z axis
- Y axis & Z axis
- X axis, Y axis & Z axis

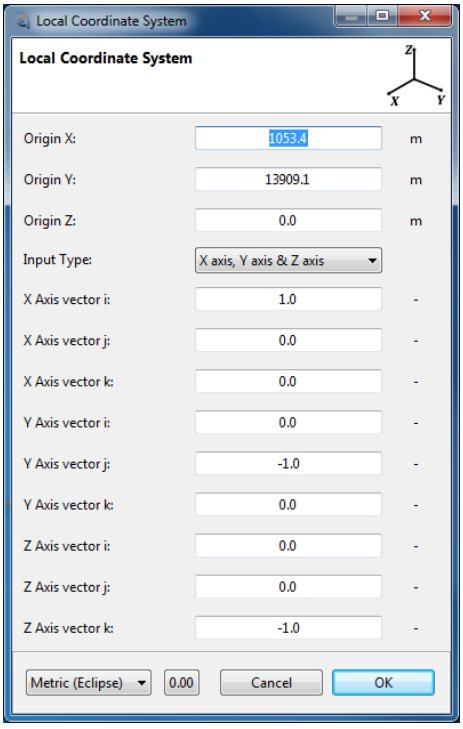


Figure 60- Local Coordinate System

Copy and Multiply:

The "Copy and Multiply" window has been displayed in Figure 61. User can perform copy, multiply or add offset to the selected property, defining the ijk range of the grids in "Box" section. Currently the $Y = aX + b$ formula can be used, Where Y and X are destination and source grid properties, respectively. These properties can be chosen from "Equation" section of the window. If you want to copy a property values to other one you just insert 1 for multiplier "a". If you want to add an offset to the selected property, fill Offset (b) field.

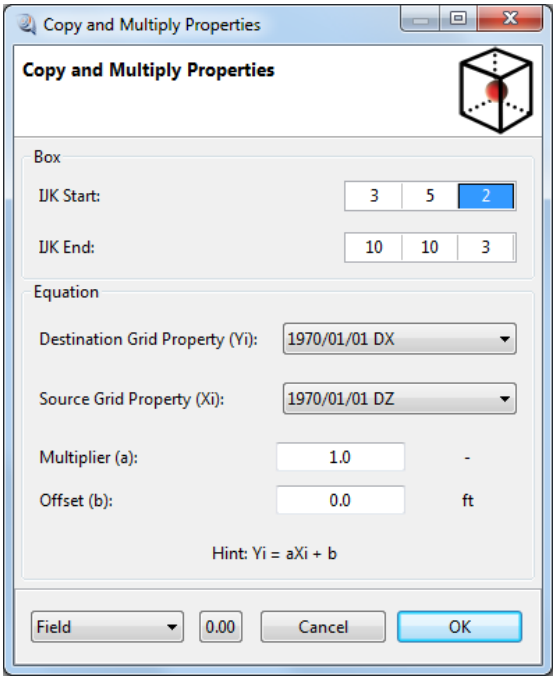


Figure 61- Copy and Multiply Window

Edit Property:

Edit property window consist of four separated sections of, Layer, Region, Grid Data, and Value.

Layer: The layer range is specified (for example 1 to 3).

Region: The region is specified using "Structured Region" button. There are three definition options for defining desired region:

- Define: Structured Box (IJK)
- Define: List of Nod Indices
- Define: Range of Indices

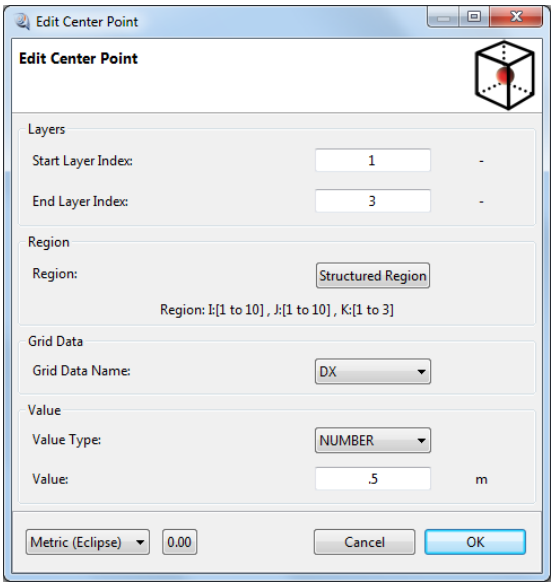


Figure 62- Edit Center Point Grid Data Window

Structured Box (IJK): in this case, the region is defined using grid indices. The box boundary indices of I, J, K are inserted in this window (Figure 63).

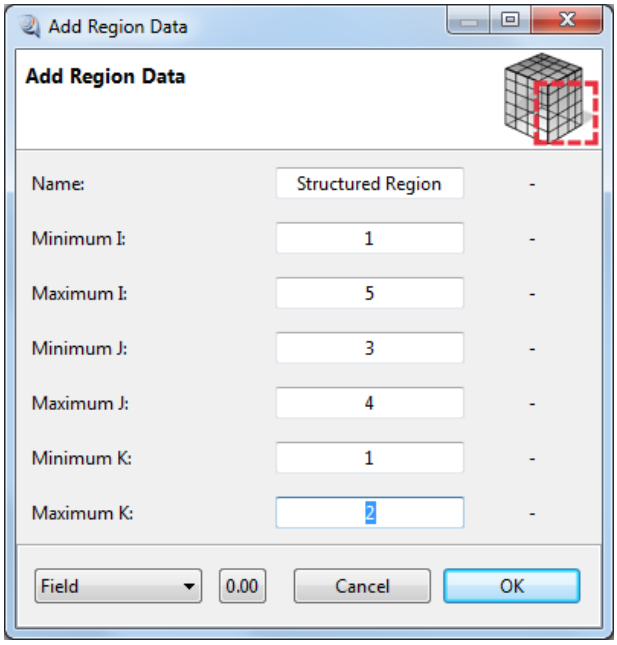


Figure 63-Define Region Using Structured Box Option

List of Node Indices: in this case, the global indices of the desired list of grids are entered. Note that each index must separate with the next and previous one using space character (Figure 64).

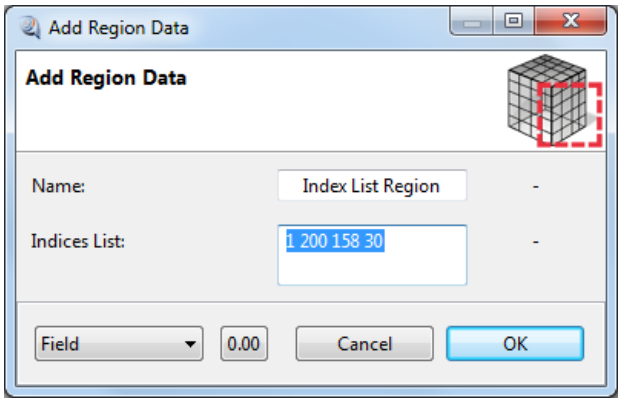


Figure 64- Define Region Using List of Node Indices

Range of Indices: The range of grid global indices and region name should be entered using this option (Figure 65).

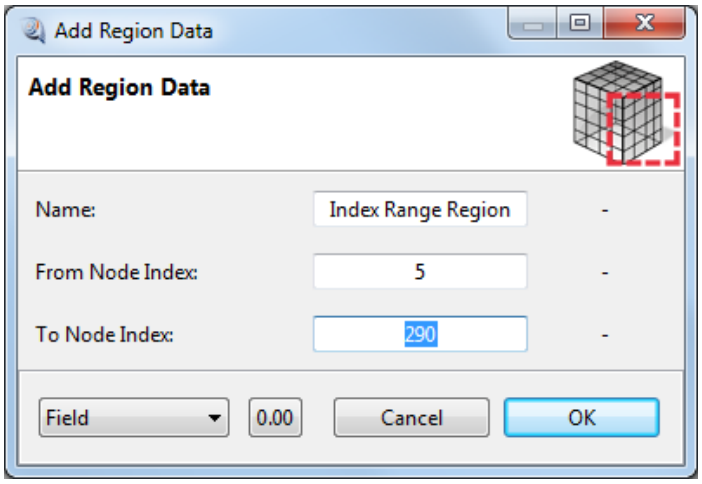


Figure 65- Define Region Using Range of Indices

Grid Data: Grid data which is one of the DX, DY, DZ, or TOPS, is selected from exist list.

Value: The desired value of the specified grid data is entered here.

Set By Layer:

This button will lead user to the set layer value window of Figure 66 window. User can enter values for each layer in "value" column. After choosing unit system and grid property name. One of the DX, DY, DZ, or TOPS can be selected from "Grid Property Name" list.

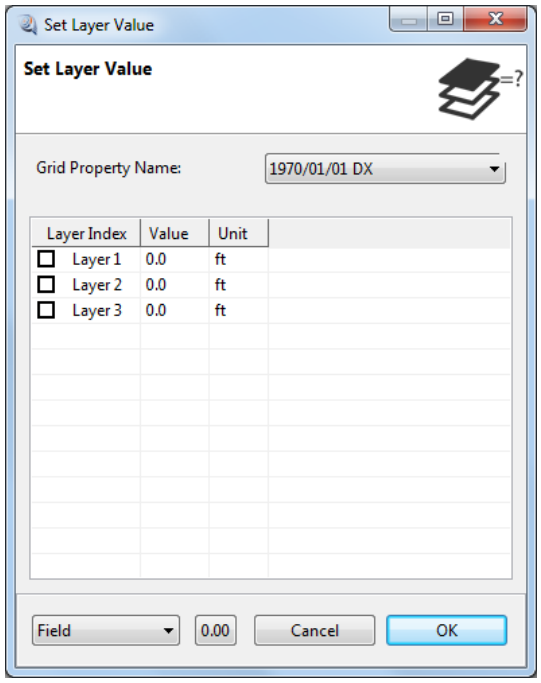


Figure 66- Set Grid Data By Layer

String Conversion Option:

The Button of  in Figure 66 and all other RETINA windows is used to specify precision and formatting flag of the local window displayed numbers (Figure 67). Currently, the following formats for the numbers are available:

- Scientific Notation (e)
- Scientific Notation (E)
- Floating Point Notation
- Automatic Notation (E)
- Automatic Notation (e)
- Integer

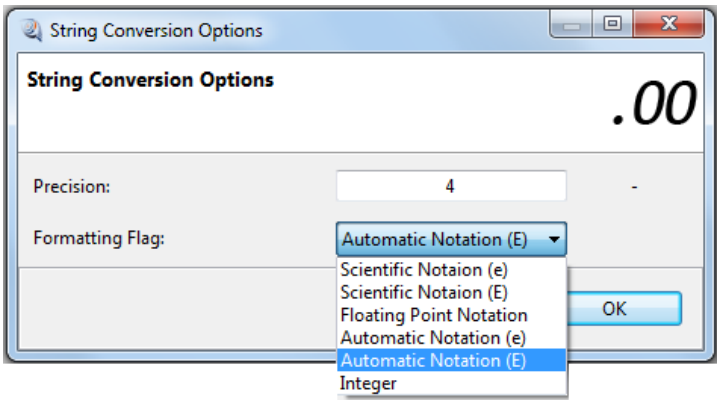


Figure 67- String Conversion Option

B) ZCORN, COORD Geometry Type

This type of geometry is available, only when user imports a data file or a grid file (using grid importer which will be explained in the next section) that includes ZCORN (Corner point Zs) and COORD (coordinate lines) data. In this case the "Grid Data" window will be like Figure 58. Selecting "Coordinate Lines (COORD)" or "Corner Point Zs" by clicking on each one will show the Figure 69 or Figure 70, respectively.

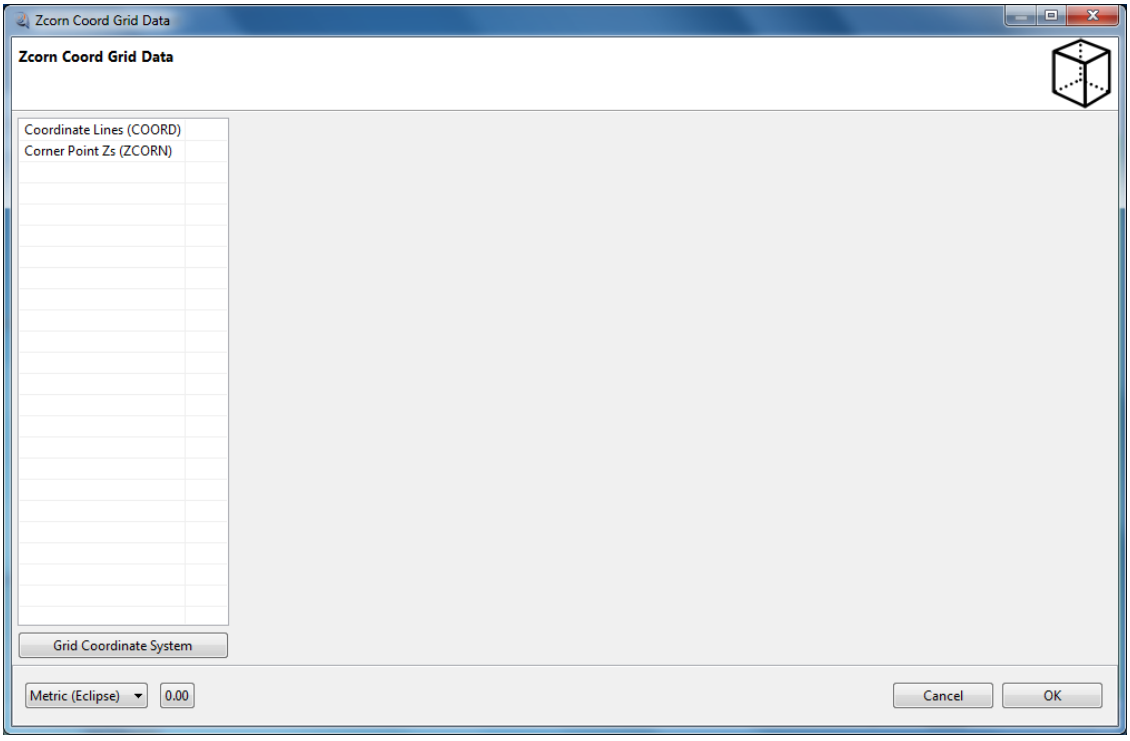
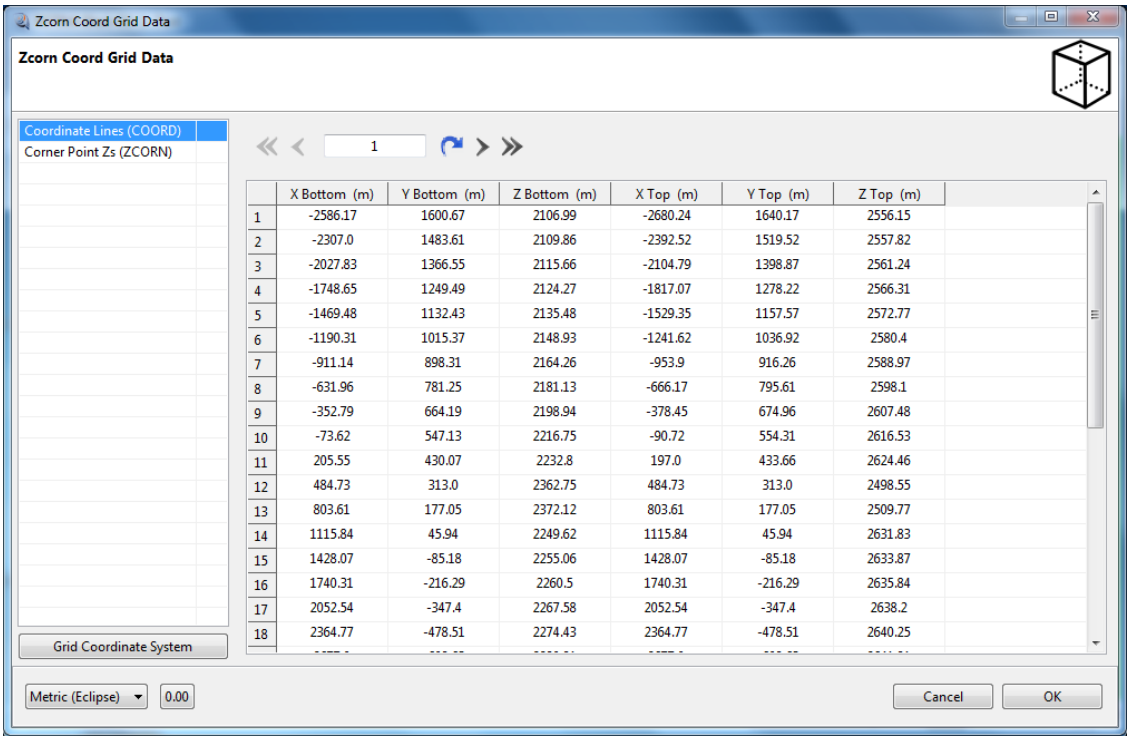
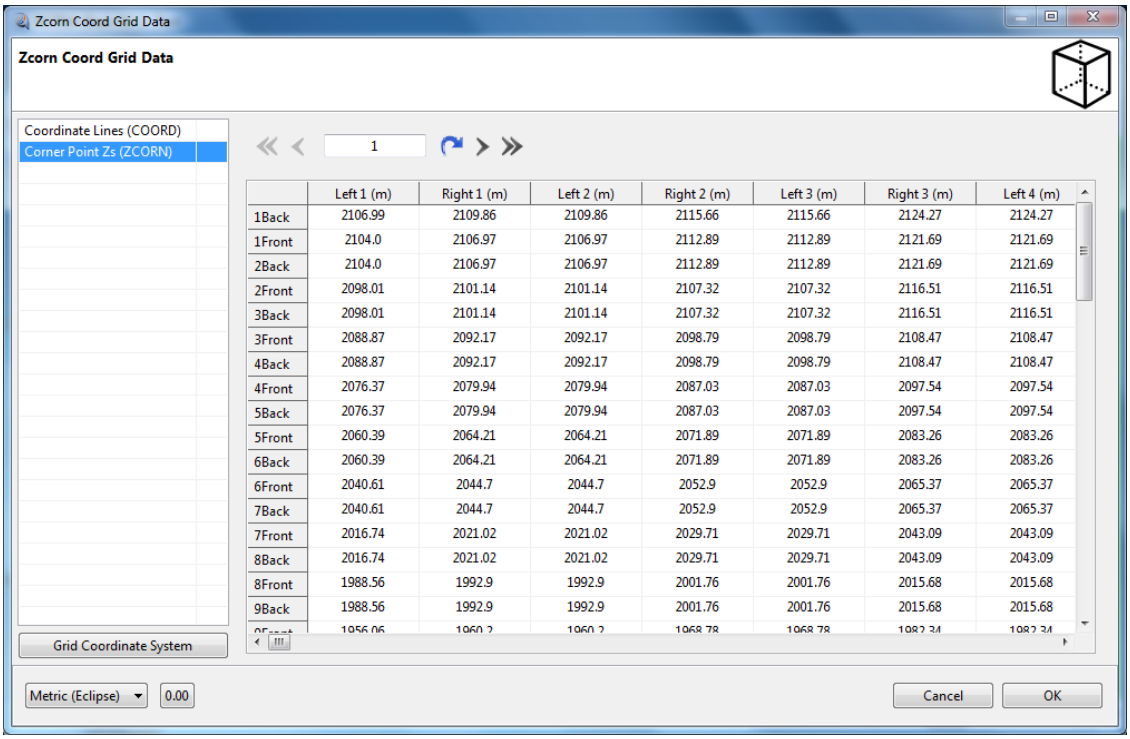


Figure 68- ZCORN COORD Geometry Type



	X Bottom (m)	Y Bottom (m)	Z Bottom (m)	X Top (m)	Y Top (m)	Z Top (m)
1	-2586.17	1600.67	2106.99	-2680.24	1640.17	2556.15
2	-2307.0	1483.61	2109.86	-2392.52	1519.52	2557.82
3	-2027.83	1366.55	2115.66	-2104.79	1398.87	2561.24
4	-1748.65	1249.49	2124.27	-1817.07	1278.22	2566.31
5	-1469.48	1132.43	2135.48	-1529.35	1157.57	2572.77
6	-1190.31	1015.37	2148.93	-1241.62	1036.92	2580.4
7	-911.14	898.31	2164.26	-953.9	916.26	2588.97
8	-631.96	781.25	2181.13	-666.17	795.61	2598.1
9	-352.79	664.19	2198.94	-378.45	674.96	2607.48
10	-73.62	547.13	2216.75	-90.72	554.31	2616.53
11	205.55	430.07	2232.8	197.0	433.66	2624.46
12	484.73	313.0	2362.75	484.73	313.0	2498.55
13	803.61	177.05	2372.12	803.61	177.05	2509.77
14	1115.84	45.94	2249.62	1115.84	45.94	2631.83
15	1428.07	-85.18	2255.06	1428.07	-85.18	2633.87
16	1740.31	-216.29	2260.5	1740.31	-216.29	2635.84
17	2052.54	-347.4	2267.58	2052.54	-347.4	2638.2
18	2364.77	-478.51	2274.43	2364.77	-478.51	2640.25

Figure 69- Coordinate Line Window



	Left 1 (m)	Right 1 (m)	Left 2 (m)	Right 2 (m)	Left 3 (m)	Right 3 (m)	Left 4 (m)
1Back	2106.99	2109.86	2109.86	2115.66	2115.66	2124.27	2124.27
1Front	2104.0	2106.97	2106.97	2112.89	2112.89	2121.69	2121.69
2Back	2104.0	2106.97	2106.97	2112.89	2112.89	2121.69	2121.69
2Front	2098.01	2101.14	2101.14	2107.32	2107.32	2116.51	2116.51
3Back	2098.01	2101.14	2101.14	2107.32	2107.32	2116.51	2116.51
3Front	2088.87	2092.17	2092.17	2098.79	2098.79	2108.47	2108.47
4Back	2088.87	2092.17	2092.17	2098.79	2098.79	2108.47	2108.47
4Front	2076.37	2079.94	2079.94	2087.03	2087.03	2097.54	2097.54
5Back	2076.37	2079.94	2079.94	2087.03	2087.03	2097.54	2097.54
5Front	2060.39	2064.21	2064.21	2071.89	2071.89	2083.26	2083.26
6Back	2060.39	2064.21	2064.21	2071.89	2071.89	2083.26	2083.26
6Front	2040.61	2044.7	2044.7	2052.9	2052.9	2065.37	2065.37
7Back	2040.61	2044.7	2044.7	2052.9	2052.9	2065.37	2065.37
7Front	2016.74	2021.02	2021.02	2029.71	2029.71	2043.09	2043.09
8Back	2016.74	2021.02	2021.02	2029.71	2029.71	2043.09	2043.09
8Front	1988.56	1992.9	1992.9	2001.76	2001.76	2015.68	2015.68
9Back	1988.56	1992.9	1992.9	2001.76	2001.76	2015.68	2015.68

Figure 70 - Corner Point Zs Window

Always there is numerous data of COORD and ZCORN exists for defining reservoir geometry, so inserting them one by one is impossible. This is the reason for designing grid importer.

Jumping to desired layer or going to next, Previous, first and last layer procedure is similar to center point grid data window.

C) Corner Point Geometry Type

This type of geometry is available, only when user imports a data file or a grid file (using grid importer which will be explained in the next section). Each grid is specified using 8 corners coordinate. The corner point grid data window has been displayed in Figure 71.

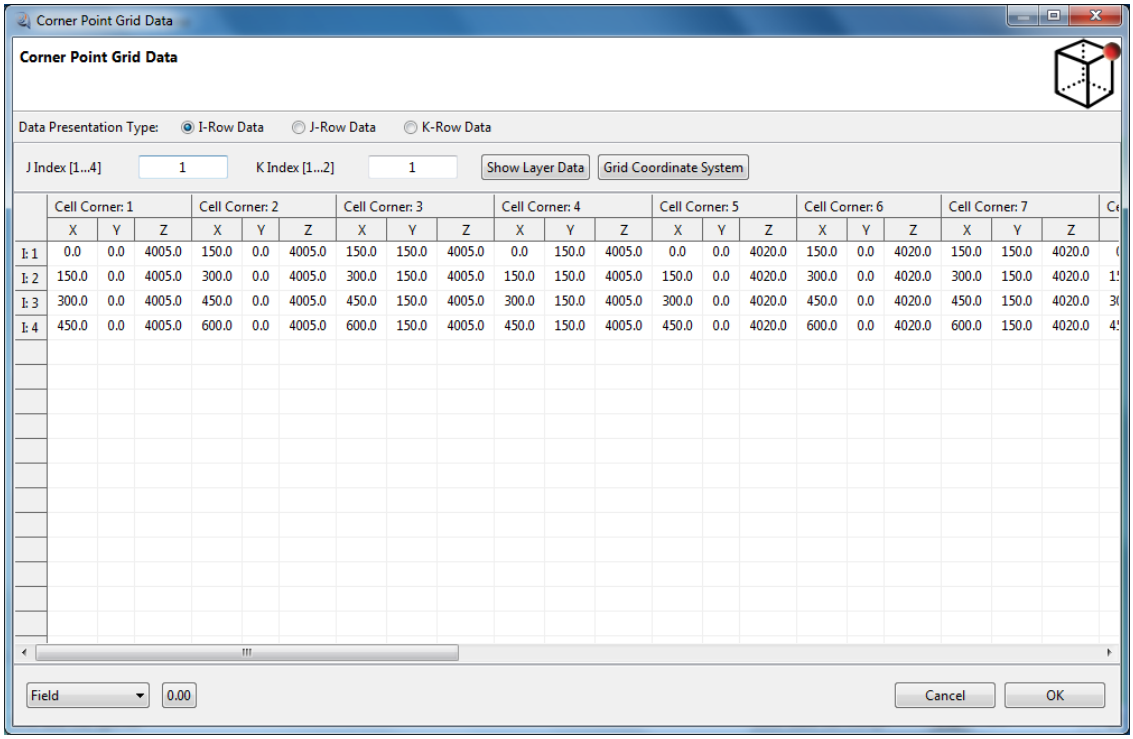


Figure 71- Corner Point Geometry Type

User can select data presentation types of I, J or K. for example if user select “I-Raw data” and then enter “J Index” and “K Index” of 2 and 1, respectively, layer data is displayed just in case of pressing “Show Layer Data” button (Figure 72).

There is already a “Grid Coordinate System” button which was explained previously in center point grid data section.

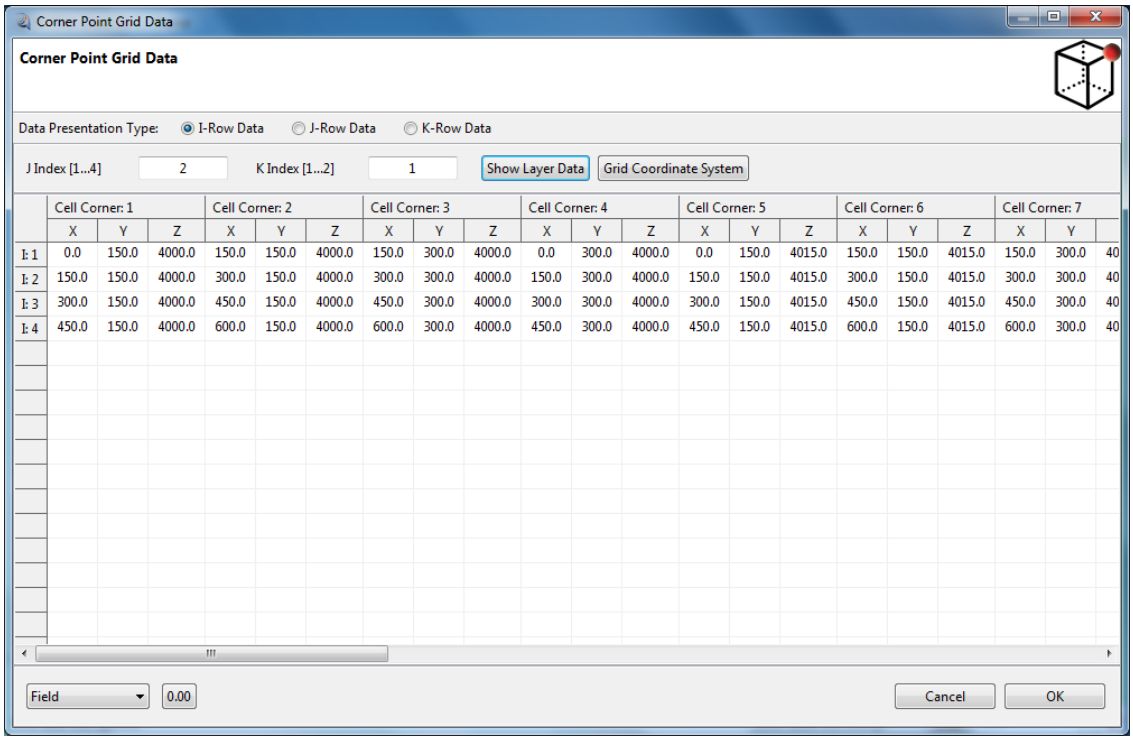


Figure 72- "I-Raw data" for "J Index" and "K Index" of 2 and 1, respectively

3-1-3- Grid Importer

This window is used to import various grid files for creating model geometry. The grid importer window which has been displayed in Figure 73 includes following components:

- A field for grid File Path
- Grid File Type (one of the GDFILE, RESCUE or GRDECL files)
- Grid Nx, Ny, Nz (number of grids in each direction)
- Grid File Unit
- Nz is Doubled check box

The "GDFILE" grid file type supports the files with format of *.GRID, *.EGRID, *.FEGRID, and *.FGRID. The "RESCUE" file supports *.BIN and *.TXT and GRDECL can use for ZCORN/COORD or DX/DY/DZ/TOPS keyword families. In case of importing grid files user can check the "Nz is Doubled" check box if the imported file has doubled Nz.

Note: Nx, Ny, and Nz are needed only for GRDECL families of keywords.

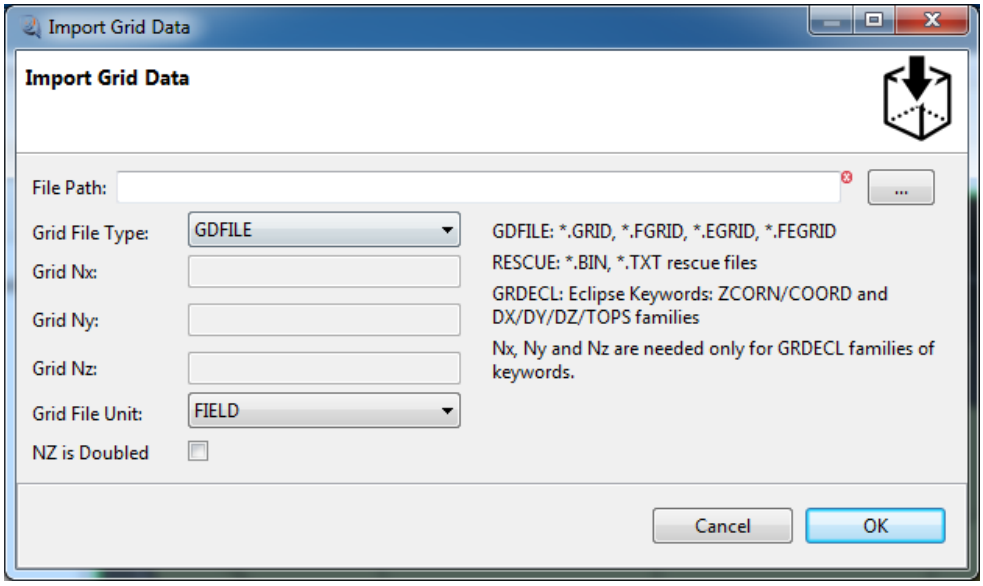


Figure 73 Import Grid Data Dialog

3-1-4- Grid Property

Grid properties window is used to define and edit grid properties. Figure 74 shows the typical "Grid Properties" window.

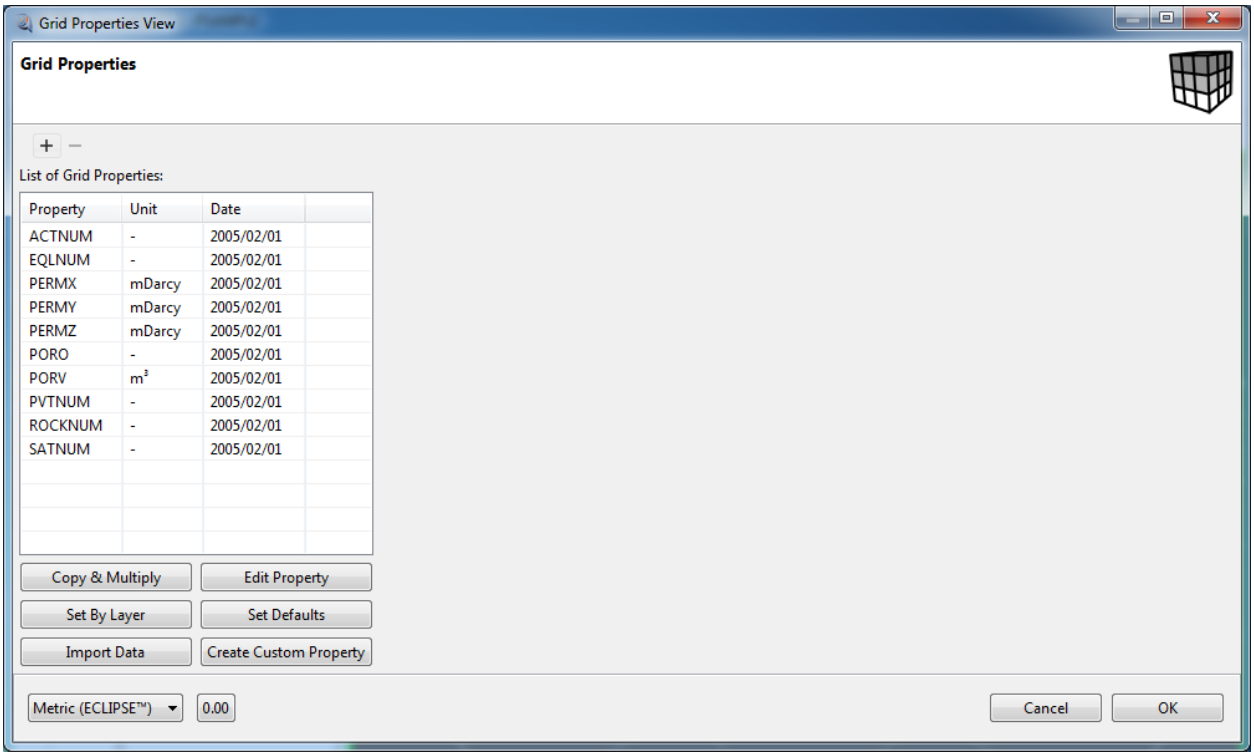


Figure 74- Grid Properties Window

Supported Grid Properties:

The following list of grid array properties has been supported in RETINA:

1- *ACTNUM (Active grid block identification):*

The value of 0 Indicates that the corresponding grid block is inactive, and 1 Indicates that the corresponding grid block is active. It should be noted that ACTNUM is one of a number of ways of identifying inactive grid blocks; any method that results in zero pore volume will cause a grid block to be treated as inactive. Any grid blocks that do not have an ACTNUM value entered default to active (1). Other possibilities include:

- Setting the porosity, or net to gross thickness ratio, to zero.
- Setting a minimum threshold pore volume; any grid blocks having a pore volume smaller than the minimum pore volume will be treated as inactive.

2- *DPNUM (Identifies extent of dual porosity region):*

It may be used for a dual porosity, single permeability run to specify regions within the reservoir to be treated as single porosity only. Only the values of 0 or 1 are allowed for each grid block in the first matrix layers, to indicate whether the grid block should be modeled as single porosity (0) or dual porosity (1). Any grid blocks that do not have a DPNUM value entered default to dual porosity. Note that this grid block property are only required for the matrix cells; the fracture cells are made inactive and ignored. Within single porosity regions, flow is between matrix blocks. Wells may only connect with the matrix blocks in single porosity regions.

3- *DZMTRXV (The vertical dimension of a block of matrix material):*

If the dual porosity gravity imbibition option is being used (GRAVDR, GRAVDRM,), this grid property should be used to specify the vertical dimension of a typical block of matrix material. The strength of the gravity imbibition effect is directly proportional to this. The effect is usually very small. However, in the absence of capillary pressure, there is no other mechanism to equalize the fluid levels in matrix and corresponding fracture cells. Any positive real number is allowed for this grid property. Note that this grid block property are only required for the matrix cells; so the matrix cell values are copied into the fracture grids.

4- *ENDNUM (End point scaling versus depth region numbers):*

Any integer for every grid block in this property specifies the end point scaling versus depth table region to which it belongs. The region number should not be less than 1. The region number specifies which end point scaling versus depth

table should be used to calculate the saturation table end points for each grid block. The end point scaling option should be activated in the End Point Scaling dialog.

5- *EQLNUM (Equilibration region numbers):*

Any integer for every grid block in this property specifies the equilibration region to which it belongs. The region number should not be less than 1. All blocks with the same equilibration region number must also have the same PVT region number (PVTNUM).

6- *FIPNUM (Fluid in place region numbers):*

Any integer for every grid block in this property specifies the fluid in place region to which it belongs. The region numbers should not be less than 1. This property defines the standard set of fluid in place regions. Note that the pore volume will include the pore volume of any aquifers associated with the region. Aquifer cells with no explicit region number are assigned to region 1.

7- *FLUXNUM (Identifies extent of each flux region):*

Any integer for every grid block in this property specifies the flux region number to which it belongs. If a zero FLUXNUM region is copied in to a REGIONS array then the destination array will be set to 1. Any cells not explicitly assigned a FLUXNUM value will be defaulted to a value of 1.

8- *IMBNUM (Imbibition saturation function region numbers):*

Any integer for every grid block in this property specifies the saturation function region number to which it belongs. This property is used in runs in which the hysteresis option is being used. It specifies which saturation table is to be used for each cell for imbibition processes. The usual saturation table numbers specified using the SATNUM property is used for drainage processes and equilibration. The region number should not be less than 1. The saturation function region number specifies which set of saturation functions should be used to calculate relative permeabilities and capillary pressures in each grid block when hysteresis is being used.

9- *IWORK (Integer work array names):*

Work arrays of double-precision or integer type may be specified, as required by the desired operation.

10- *KRG, KRGR (Scaled end-point gas relative permeabilities):*

The KRG and KRGR allow scaling of the relative permeability of gas within each grid block. KRG scales the Kr at the maximum gas saturation (typically at connate

water), and KRGR scales the relative permeability at residual oil saturation (or critical water in a gas-water run). The property can only be used when the end point scaling option is active. If the KRGR keyword has not been used, the KRG keyword has the effect of scaling the relative permeability value calculated from the appropriate saturation table after the scaled saturation end-points have been accounted for. The relative permeability is taken to be the value at either the maximum saturation of the saturation table or at SGU if this has been specified. This is usually equal to 1-Swco . Note that, none of the KRG keywords should be used if gas is not present in the model. Any real number can be inserted for the KRG for each grid block.

11-*KRO, KRORG, KRORW (Scaled end point oil relative permeabilities):*

The KRO, KRORW, and KRORG properties scale the relative permeability of oil within each grid block. KRO scales the Kr at the maximum oil saturation (typically at connate water), KRORW scales the relative permeability at the critical water saturation, and KRORG scales the relative permeability at the critical gas saturation. The property can only be used when the end-point scaling option is active. If the KRORW and KRORG properties have not been used, the KRO has the effect of scaling the relative permeability value calculated from the appropriate saturation table after the scaled saturation endpoints have been accounted for. The relative permeability is taken to be the value at either the maximum saturation of the saturation table or at SWL if this has been specified.

In three-phase cases using the STONE II Model for three-phase relative permeability, the oil relative permeability is a function of the water and gas relative permeabilities, and hence the oil relative permeability is modified by use of the KRW and KRG keywords to scale the water and gas relative permeabilities.

12-*KRW, KRWR (Scaled end point water relative permeabilities):*

The KRW and KRWR scale the relative permeability of water within each grid block. KRW scales the Kr at the maximum water saturation (typically at a water saturation of 1.0), and KRWR scales the relative permeability at residual oil saturation (or residual gas in a gas- water run). The keyword can only be used when the end-point scaling option is active. If the KRWR has not been used, the KRW has the effect of scaling the relative permeability value calculated from the appropriate saturation table after the scaled saturation end-points have been accounted for. The relative permeability is taken to be the value at either the maximum saturation of the saturation table or at SWU if this has been specified.

13-LX (X direction matrix block sizes for the viscous displacement option):

This property defines the representative matrix block size in the X direction. Any positive real number for every grid block is allowed for this property.

14-LY (Y direction matrix block sizes for the viscous displacement option):

This property defines the representative matrix block size in the Y direction. Any positive real number for every grid block is allowed for this property.

15-LZ (Z direction matrix block sizes for the viscous displacement option):

This property defines the representative matrix block size in the Z direction. Any positive real number for every grid block is allowed for this property.

16-MINPVV (Sets minimum pore volumes cells must have to be active):

This property is used to declare a threshold pore volume that a cell must exceed or it will made inactive. Any positive real number for every cell specifying its threshold pore volume in the current units, are allowed. This causes any cell with a pore volume less than the input value associated with that cell to become inactive. An inactive cell does not contribute to the total pore volume of the system, and is treated by default as a barrier. In order for non-neighbor connections to be established between adjoining cells across an inactive cell, pinchouts must be enabled using the keyword PINCH or PINCHOUT, and non-neighbor connections must be enabled.

Cells which have been set inactive via the ACTNUM property remain so even if their pore volume exceeds the threshold; using MINPVV only affects active cells. Any minimum pore volumes that are not specified is set to 0.

Note: Cells which have been deactivated by the ACTNUM keyword will remain inactive even if the cell volume is greater than the minimum set by MINPVV. Cells which have been left active by the ACTNUM keyword will be made inactive if the cell volume is less than the minimum set by MINPVV.

Note: Using the PORV to reset the pore volumes to greater than the minimum value will reactivate an inactive cell regardless of how it was deactivated, whether by ACTNUM or by MINPVV.

17-MULTNUM (Defines regions for applying inter-region transmissibility multipliers):

MULTNUM provides an alternative region definition for applying inter-region multipliers, instead of applying them to flux regions defined by FLUXNUM. The

MULTNUM property can be used to work with transmissibility multipliers using the MULTREGT, or with the MULTREGP (which sets a pore volume multiplier for a specific region). Any integer is allowed for MULTNUM for each grid cell, specifying the region number to which it belongs. Any cells not explicitly assigned a value of MULTNUM will be taken to be in region 1.

Note: 0 is a valid value for MULTNUM similarly to FLUXNUM.

18- MULTPV (Pore volume multipliers):

Any non-negative real number for every grid block can be assigned to the MULTPV property. The values specified act as multipliers on the pore volumes. Any MULTPV values that are not specified, default to 1.0.

19- NTG (Net to gross thickness ratio):

Any non-negative real number for every grid block can be assigned to the NTG property. The values specified are used to convert from gross to net thicknesses, and act as multipliers of grid block pore volumes and transmissibility values in the X and Y directions, and also on DZ for the calculation of well connection transmissibility factors. Any NTG values which are not specified default to 1.0.

Grid blocks whose pore volume is zero are assumed as inactive. Since the computing time and storage space requirements of a run depend primarily on the number of active grid blocks, it is important that inactive blocks should be unambiguously identified. This is best done by setting either PORO or NTG to zero in inactive blocks. It is also possible to set pore volumes directly, or to use the ACTNUM property.

20- OILAPI (Initial oil API values, for API tracking option):

Any real number for every grid block, specifying the initial oil API gravity can be assigned to this property.

21- OPERNUM (Defines regions for performing operations on arrays):

The OPERNUM property provides a region definition for performing arithmetic operations on property arrays. Any integer for each grid cell, specifying the region number to which it belongs, can be set to the OPERNUM. Any cells not explicitly assigned a value of OPERNUM are taken to be in region 0, which implies that no operation is allowed for this particular cell.

Note: For use with the Dual Porosity option, only the matrix cells or the fracture cells need to be input. The cell values in the other porosity are copied automatically.

22- PBUB (Initial bubble point pressure):

This is an alternative to the RS for defining the initial dissolved gas distribution in runs with enumerated initial conditions. When using explicit initialization, you must ensure that the initial solution is stable and physically reasonable.

23-PCG (Scaled maximum gas capillary pressures):

PCG allows the maximum gas-oil capillary pressure to be scaled on a grid block by grid block basis. It can only be used when the end-point scaling option is active, otherwise it is ignored. If the PCG is omitted, the maximum gas Pc value defaults to the value used in the appropriate gas saturation function table.

The PCG should not be used in combination with the J-function option. In this case, the J-function calculation always defines the gas capillary pressure scaling and the input PCG array is ignored.

24-PCW (Scaled maximum water capillary pressures):

PCW allows the maximum water-oil (or water-gas) capillary pressure to be scaled on a grid block by grid block basis. It can only be used when the end-point scaling option is active, otherwise it is ignored. If the PCW is omitted, the maximum water Pc value defaults to the value used in the appropriate water saturation function table.

The PCW should not be used in combination with the J-function option. In this case, the J-function calculation always defines the water capillary pressure scaling and the input PCW array is ignored.

25-PDEW (Initial dew point pressure):

This is an alternative to the RV for defining the vaporized oil distribution in runs with enumerated initial conditions. When using explicit initialization, you must ensure that the initial solution is stable and physically reasonable.

26-PERMX (Specifies X-permeability values):

PERMX specifies the permeability values in the X-direction. Permeability values must be non-negative. Every PERMX value in the top plane (K = 1) must be specified. Values in lower planes (K > 1) that are not specified default to the value in the plane above.

Note: This is not the permeability between a cell and its neighbor. The transmissibility between a cell and its neighbor involves the permeability values for both cells.

27-PERMY (Specifies Y-permeability values):

PERMY specifies the permeability values in the Y-direction. Permeability values must be non-negative. Every PERMY value in the top plane (K = 1) must be

specified. Values in lower planes (K > 1) that are not specified default to the value in the plane above.

Note: This is not the permeability between a cell and its neighbor. The transmissibility between a cell and its neighbor involves the permeability values for both cells.

28- PERMZ (Specifies Z-permeability values):

PERMZ specifies the permeability values in the Z-direction. Permeability values must be non-negative. Every PERMZ value in the top plane (K = 1) must be specified. Values in lower planes (K > 1) that are not specified default to the value in the plane above.

Note: This is not the permeability between a cell and its neighbor. The transmissibility between a cell and its neighbor involves the permeability values for both cells.

29- PORO (Specifies the grid block porosity values):

Any non-negative real number for every grid block can be assigned to the PORO, specifying the fractional porosity value for each cell. Grid blocks whose pore volume is zero are treated by the program as inactive. Since the computing time and storage space requirements of a run depend primarily on the number of active grid blocks, it is important that inactive blocks should be unambiguously identified. This is best done by setting either PORO or NTG to zero in inactive blocks. It is also possible to set pore volumes directly (array PORV), or to use the ACTNUM property.

30- PORV (Specifies the grid block pore volumes):

PORV can be assigned by any non-negative real number for every grid block in the specifying the grid block pore volumes. Any pore volumes that are not altered remain at the values calculated by the program from the grid data. Grid blocks whose pore volume is zero are treated by the program as inactive. Since the computing time and storage space requirements of a run depend primarily on the number of active grid blocks, it is important that inactive blocks should be unambiguously identified. This can be done either by using the PORV to overwrite pore volumes directly, or by setting PORO or NTG to zero. The ACTNUM can also be used to identify inactive grid blocks.

31- PRESSURE (Initial pressures):

Any real number for every grid block specifying the initial pressure can be assigned to the PRESSURE. This property must be used with other properties (such as SWAT or SGAS) sufficient to define the initial state explicitly for the run.

32-PVTNUM (PVT region numbers):

Any integer for every grid block specifying the PVT region to which it belongs can be assigned to the PVTNUM. The region number should not be less than 1. All grid blocks within a particular equilibration region must have the same PVT region number. The PVT region number specifies which set of PVT tables (input using DENSITY, PVDG, PVDO, PVTG, PVTO, PVCO, PVTW and ROCK) should be used to calculate PVT properties of fluids in each grid block.

33-ROCKNUM (Rock compaction table region numbers):

Any integer for every grid block, specifying the rock compaction table region to which it belongs, can be set to the ROCKNUM. The region number should not be less than 1. The region number specifies which rock compaction table should be used to calculate the porosity and permeability multipliers for each grid block. The Rock Compaction option must be enabled in Rock region data dialog.

If the ROCKTSIG table is used to specify pressure tables of the sigma multiplier for matrix-fracture coupling in dual porosity runs, only the ROCKNUM numbers of the matrix cells are used to obtain the pressure tables. The ROCKNUM numbers of the fracture cells are ignored.

34-RS (Initial solution gas-oil ratios):

Any real number for every grid block, specifying the initial solution gas-oil ratio can be set to the RS. This is an alternative to the PBUB for defining the initial dissolved gas distribution in runs with enumerated initial conditions

35-RV (Initial vapor oil-gas ratios):

Any real number for every grid block, specifying the vapor oil-gas ratio can be set to the RV. This is an alternative to the PDEW for defining initial vaporized oil distribution in runs with enumerated initial conditions

36-SATNUM (Saturation function region numbers):

Any integer for every grid block, specifying the saturation function region to which it belongs, can be set to the SATNUM. The region number should not be less than 1. The saturation function region number specifies which set of saturation functions (input using SGFN, SOF3 and etc.) should be used to calculate relative permeabilities and capillary pressures in each grid block.

37-SGAS (Initial gas saturations):

SGAS can include any real number for every grid block specifying the initial gas saturation. This may be used to specify the initial solution explicitly, as an alternative to equilibration using EQUIL. It is expected if the PRESSURE is present, indicating explicit initialization.

38-SGCR (Scaled critical gas saturations):

The SGCR and specifies the critical gas saturation (that is the largest gas saturation for which the gas relative permeability is zero) within each grid cell. The free gas flow across each grid face is calculated from a transformed gas relative permeability curve obtained by linearly scaling the tabulated relative permeability data between the new critical gas saturation defined using an SGCR, and the new maximum gas saturation (SGU). The SGCR also permits scaling of the relative permeability table used in computing the flow of free gas between grid cells and well connections.

39-SGL (Scaled connate gas saturations):

The SGL specifies the connate gas saturation (that is the smallest gas saturation in a gas saturation function table) within each grid cell. The gas-oil capillary pressure for gas flow across each grid face is calculated from a transformed gas-oil capillary pressure curve obtained by linearly scaling the tabulated capillary pressure data between the new connate gas saturation defined using an SGL, and the new maximum gas saturation (SGU). The SGL also permits scaling of the gas-oil capillary pressure table used in computing the initial gas saturations.

40-SGLPC (Scaled connate gas saturations, for Pc curves only):

The SGLPC scales the connate gas end point of the gas-oil capillary pressure curves without scaling the corresponding oil relative permeability curves. The SGLPC should not be used unless gas is present in the model. The endpoint scaling option must be enabled for this property. The SGLPC values can be any real number for each grid block, specifying the connate gas saturation within the grid block. The saturation should be in the range 0.0 to 1.0 inclusive. If SGLPC is not specified, the gas-oil capillary pressure scaling is performed using the connate gas saturation specified with the SGL. If in turn SGL, are omitted, then the connate gas saturation for a gas-oil capillary pressure calculation defaults to the value used in the appropriate gas saturation function table. If the hysteresis model is active then the SGLPC data will scale the drainage curve.

41-SGU (Scaled saturation table maximum gas saturations):

SGU specifies the maximum gas saturation (that is the largest gas saturation in a gas saturation function table) within each grid cell. The scaled maximum gas saturation is used to determine the scaled forms for the gas-oil capillary pressure curves and the gas relative permeability curves. The scaled form of the capillary pressure curves is obtained by linearly scaling the tabulated capillary pressure data between the new connate gas saturation and the new maximum gas saturation defined using an SGU. A similar linear transformation is used to produce the scaled gas relative permeability curves using the new critical gas saturation and the new maximum gas saturation defined using an SGU. The scaled capillary pressure and relative permeability curves are then used to compute the free gas flow out of each grid cell face, the free gas flow between grid cells and well connections and in the equilibration algorithm.

42-SGUPC (Scaled maximum gas saturations, for Pc curves only):

The SGUPC scales the maximum gas end point of the gas-oil capillary pressure curves without scaling the corresponding oil relative permeability curves. The SGUPC should not be used unless gas is present in the model. The endpoint scaling option must be enabled for this property. The SGUPC values can be any real number for each grid block, specifying the maximum gas saturation within the grid block. The saturation should be in the range 0.0 to 1.0 inclusive. If SGUPC is not specified, the gas-oil capillary pressure scaling is performed using the maximum gas saturation specified with the SGU. If in turn SGU, are omitted, then the maximum gas saturation for a gas-oil capillary pressure calculation defaults to the value used in the appropriate gas saturation function table. If the hysteresis model is active then the SGUPC data will scale the drainage curve.

43-SIGMAGDV (Matrix-fracture coupling for oil-gas gravity drainage):

The SIGMAGDV can be used to specify an alternative matrix-fracture coupling for matrix cells in which the production mechanism is gravity drainage due to the presence of gas in the fractures. Any positive real number for every grid block can be assign to the SIGMAGDV. Any values input in the fracture grids will be ignored. The SIGMAGDV property is only available in three-phase dual porosity runs.

44-SIGMAV (Dual porosity matrix-fracture coupling):

If the dual porosity option is enabled, this property can be used to specify a multiplier to be applied in the construction of the matrix-fracture coupling transmissibilities.

45-SOGCR (Scaled critical oil-in-gas saturations):

The SOGCR specifies the critical oil-in-gas saturation (that is the largest oil saturation for which the oil relative permeability is zero in an oil-gas-connate water system) within each grid cell. The oil flow across each grid face in an oil-gas connate water system is calculated from a transformed oil-gas relative permeability curve obtained by linearly scaling the tabulated oil-gas relative permeability data between the new critical oil-in-gas saturation defined using an SOGCR, and the new maximum oil saturation. The SOGCR also permits scaling of the oil-gas relative permeability table used in computing the flow of oil between grid blocks and well connections. The SOGCR should not be used if either oil or gas is absent from the model. The end point scaling must be enabled in the end point scaling dialog.

46-SOWCR (Scaled critical oil-in-water saturations):

The SOWCR specifies the critical oil-in-water saturation (that is the largest oil saturation for which the oil relative permeability is zero in an oil-water system) within each grid cell. The oil flow across each grid face in an oil-water system is calculated from a transformed oil-water relative permeability curve obtained by linearly scaling the tabulated oil-water relative permeability data between the new critical oil-in-water saturation defined using SOWCR, and the new maximum oil saturation. The SOWCR also permits scaling of the oil-water relative permeability table used in computing the flow of oil between grid cells and well connections. The SOWCR should not be used if either oil or water is absent from the model. The end point scaling must be enabled in the end point scaling dialog.

47-SWAT (Initial water saturations):

This property may be used to specify the initial solution explicitly, as an alternative to equilibration using EQUIL. It will be expected if the PRESSURE property is present, indicating explicit initialization, and if a water phase is present. When using explicit initialization, you must ensure that the initial solution is stable and physically reasonable.

48-SWATINIT (Initial water saturations for capillary pressure scaling):

This property allows you to input a water distribution and to scale the water-oil capillary pressure curves such that this water distribution is honored in the equilibrated initial solution. Note that any water saturation value that cannot be honored because it is below the water contact is reset to the maximum water saturation in the subsequent equilibration calculation. The scaled maximum water capillary pressures (PPCW) can be output to the print file. This can be used to

check for un-physically high maximum capillary pressures, which can be limited by using the PPCWMAX. The SWATINIT property may only be used if the end point scaling option is active.

Note: If a grid block is given a water saturation less than 1.0 below the oil water contact (where Pc=0), then the SWATINIT saturation is not honored. If a cell is given saturation corresponding to a zero capillary pressure (typically 1.0) above the contact, then the Pc curve cannot be scaled to honor the saturation, hence the Pc curve is left unscaled.

Note: If the SWATINIT saturation is less than the connate water saturation for a cell, the initial water saturation will be reset to the connate saturation and the capillary pressure will be scaled according to this value.

Note: If the PCW property has been used, then this is ignored in regions where SWATINIT is set.

Note: If the Leverette J-function calculation for the water-oil capillary pressure has been activated through the JFunc combo box in the saturation region data dialog, then J-function calculation is ignored in regions where SWATINIT is set.

Note: The SWATINIT saturation is not honored if the capillary pressure curve is a constant. The input capillary pressure curve needs to be monotonically decreasing with increasing water saturation.

49-SWCR (Scaled critical water saturations):

The SWCR specifies the critical water saturation (that is the largest water saturation for which the water relative permeability is zero) within each grid cell. The water flow across each grid face is calculated from transformed water relative permeability curve obtained by linearly scaling the tabulated relative permeability data between the new critical water saturation defined using SWCR, and the new maximum water saturation (SWU). The SWCR property also permits scaling of the relative permeability table used in computing the flow of water between grid cells and well connections. The SWCR should be used if water exists in the model. The end point scaling must be enabled in the end point scaling dialog.

50-SWL (Scaled connate water saturations):

SWL specifies the connate water saturation (that is the smallest water saturation in a water saturation function table) within each grid cell. The oil-water (or gas water) capillary pressure used in computing the water flow across each grid face is calculated from a transformed water capillary pressure curve obtained by linearly scaling the tabulated capillary pressure data between the new connate water

saturation defined using SWL, and the new maximum water saturation (SWU). The SWL also permits scaling of the water capillary pressure table used in computing the initial water saturation distribution. The SWL should be used if water is present in the model.

51-SWLPC (Scaled connate water saturations, for Pc curves only):

The SWLPC scales the connate water end point of the oil-water capillary pressure curves without scaling the corresponding oil relative permeability curves. The SWLPC should not be used unless water is present in the model. The end point scaling must be enabled in the end point scaling dialog. The saturation should be in the range 0.0 to 1.0 inclusive. If SWLPC is not specified, the oil-water (or gas water) capillary pressure scaling is performed using the connate water saturation specified with the SWL. If in turn SWL is omitted, then the connate water saturation for an oil-water (or gas-water) capillary pressure calculation defaults to the value used in the appropriate water saturation function table. If the hysteresis model is active then the SWLPC data will scale the drainage curve.

52-TRANX (X-direction transmissibility values):

TRANX specifies the transmissibility values explicitly, replacing those calculated by the RETINA. The TRANX applies to transmissibilities in the X-direction. Any non-negative real number for every grid block can be assigned to the TRANX. The values specified overwrite the X-direction transmissibilities calculated by RETINA for the +X face of each grid block. Thus, a value specified for block (I, J, K) is the transmissibility between blocks (I, J, K) and (I+1, J, K).

Note: TRANX does not affect non-neighbor connections due to faults.

53-TRANY (Y-direction transmissibility values):

TRANY specifies the transmissibility values explicitly, replacing those calculated by the RETINA. The TRANY applies to transmissibilities in the Y-direction. Any non-negative real number for every grid block can be assigned to the TRANY. The values specified overwrite the Y-direction transmissibilities calculated by RETINA for the +Y face of each grid block. Thus, a value specified for block (I, J, K) is the transmissibility between blocks (I, J, K) and (I, J+1, K).

Note: TRANY does not affect non-neighbor connections due to faults.

54-TRANZ (Z-direction transmissibility values):

TRANZ specifies the transmissibility values explicitly, replacing those calculated by the RETINA. The TRANZ applies to transmissibilities in the Z-direction. Any non-negative real number for every grid block can be assigned to the TRANZ. The

values specified overwrite the Z-direction transmissibilities calculated by RETINA for the +Z face of each grid block. Thus, a value specified for block (I, J, K) is the transmissibility between blocks (I, J, K) and (I, J, K+1).

Note: TRANZ does not affect non-neighbor connections.

55-*WORK (work array names):*

Work arrays of double-precision type may be specified, as required by the desired operation.

56-MULTX (Transmissibility multipliers in X-direction):

The values specified act as multipliers on the transmissibilities calculated by the program for the +X face of each grid block or specified directly using TRANX. Thus, a value of MULTX specified for block (I, J, K) multiplies the transmissibility between blocks (I, J, K) and (I+1, J, K). Any non-neighbor connections generated due to faults have transmissibilities that reflect the MULTX values. Non-neighbor connections entered explicitly using the NNC keyword are not affected.

57-MULTX- (Transmissibility multipliers in negative X-direction):

The values specified act as multipliers on the transmissibilities calculated by the program for the -X face of each grid block or specified directly using TRANX. Thus, a value of MULTX- specified for block (I, J, K) multiplies the transmissibility between blocks (I, J, K) and (I-1, J, K). Any non-neighbor connections generated due to faults have transmissibilities that reflect the MULTX- values. Non-neighbor connections entered explicitly using the NNC keyword are not affected.

58-MULTY (Transmissibility multipliers in Y-direction):

The values specified act as multipliers on the transmissibilities calculated by the program for the +Y face of each grid block or specified directly using TRANY. Thus, a value of MULTY specified for block (I, J, K) multiplies the transmissibility between blocks (I, J, K) and (I, J+1, K). Any non-neighbor connections generated due to faults have transmissibilities that reflect the MULTY values. Non-neighbor connections entered explicitly using the NNC keyword are not affected.

59-MULTY- (Transmissibility multipliers in negative Y-direction):

The values specified act as multipliers on the transmissibilities calculated by the program for the -Y face of each grid block or specified directly using TRANY. Thus, a value of MULTY- specified for block (I, J, K) multiplies the transmissibility between blocks (I, J, K) and (I, J-1, K). Any non-neighbor connections generated due to faults have transmissibilities that reflect the MULTY- values. Non-neighbor connections entered explicitly using the NNC keyword are not affected.

60-MULTZ (Transmissibility multipliers in Z-direction):

The values specified act as multipliers on the transmissibilities calculated by the program for the +Z face of each grid block or specified directly using TRANZ. Thus, a value of MULTZ specified for block (I, J, K) multiplies the transmissibility between blocks (I, J, K) and (I, J, K+1). Any non-neighbor connections generated due to faults have transmissibilities that reflect the MULTZ values. Non-neighbor connections entered explicitly using the NNC keyword are not affected.

61-MULTZ- (Transmissibility multipliers in negative Z-direction):

The values specified act as multipliers on the transmissibilities calculated by the program for the -Z face of each grid block or specified directly using TRANZ. Thus, a value of MULTZ- specified for block (I, J, K) multiplies the transmissibility between blocks (I, J, K) and (I, J, K-1). Any non-neighbor connections generated due to faults have transmissibilities that reflect the MULTZ- values. Non-neighbor connections entered explicitly using the NNC keyword are not affected.

User can add or remove an existed property using **+** and **-** buttons, respectively. New property can be selected (checking box) and added to the property list from "Add Properties" window (Figure 75). You can also assign a default value for each selected property in this window. In case of dual models the values for matrix and fracture grids are entered in separate columns. Pressing "OK" button will add the selected properties to grid property list.

There are Edit Property, Copy and Multiply, Set By Layer, Set defaults, and Import data buttons in "Grid Properties" window. The first three buttons act like what we introduced in "Grid Data" window, so here just "Set Defaults", "Import Data", and "Create Custom Property" buttons are introduced.

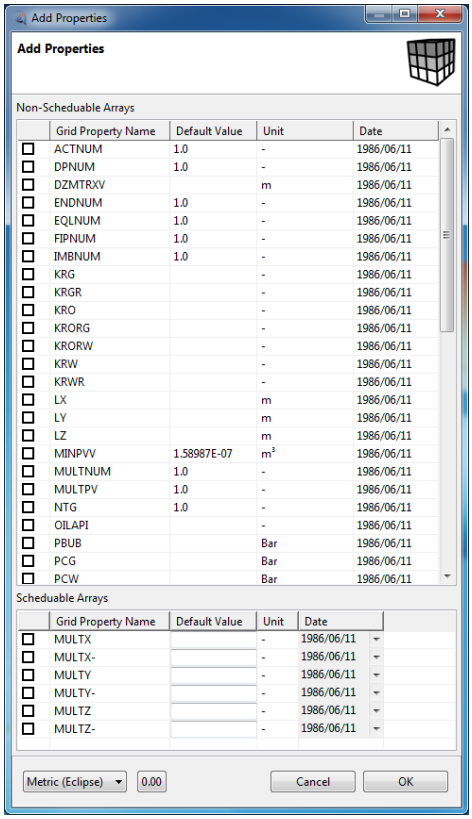


Figure 75- Add Properties Window

Set Defaults:

In addition to add properties action, assigning of the default values to grid properties facility is also available in "Set Default" value button.

Import Data:

Always there are only maps of the grid properties are available, so for importing these maps a facility of "Import Data" has been provided to import required grid properties. The Import Data window has been shown in Figure 76. Note to the file and project units in case of importing data.

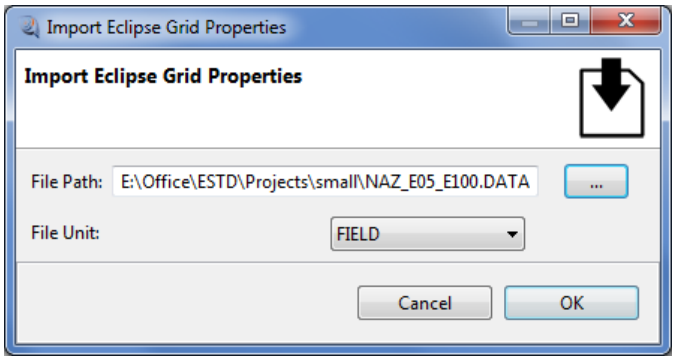


Figure 76- Import Grid Properties Window

Create Custom Property:

User can define own custom grid property, using "Custom Grid Property" dialog (Figure 77). The property name, template and value in the selected unit must be entered in this dialog. After defining the custom grid property, it is added to the grid property dialog.

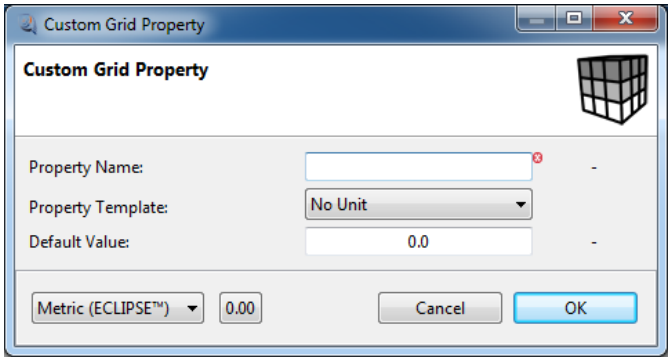


Figure 77 Create Custom Property Dialog

3-1-5- Fault Manager

The "Fault Manager" is used for defining fault and its segments. It leads user to the Figure 78 which has two columns of "fault name" and its "segments".

For adding new fault the **+** button of "fault name" column is used (for removing exist faults **-** is used). Multiple selection and remove options are available using Ctrl+Click. After adding a fault using **+** from first column, we can add segments in the second column in same way. In the segment column, the face direction and fault zone must be specified (Figure 79).

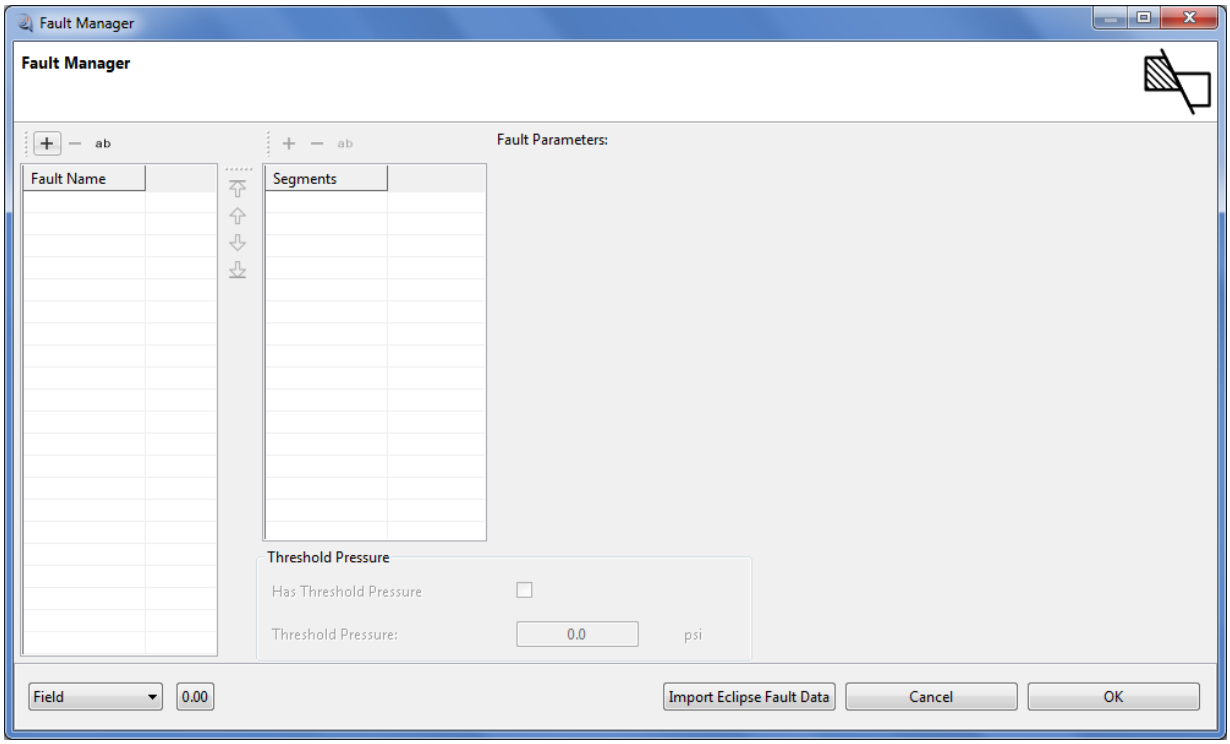


Figure 78- Define Fault

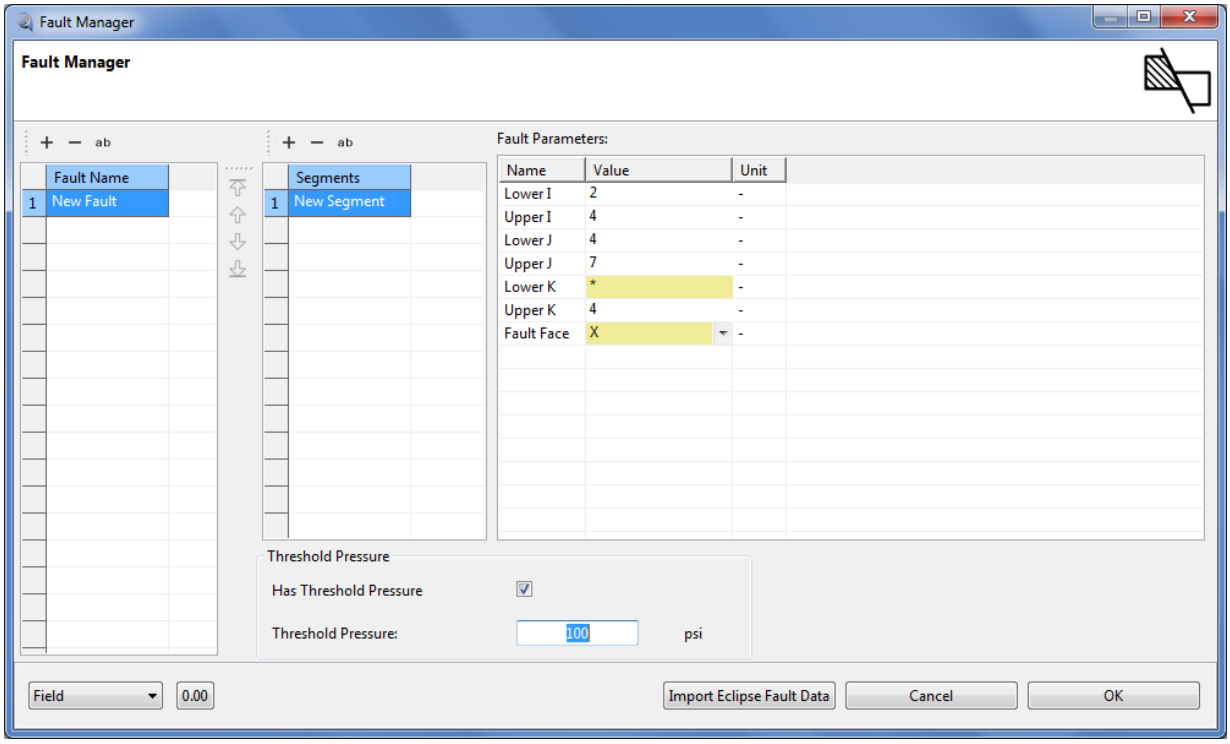


Figure 79- Fault Properties

It should be noted that in all of the RETINA windows, the yellow fields indicate default values for that field. There is a button of ab in the Fault Manager window, which is used to auto naming of the defined Faults. Assume you want to auto name 4 faults which you defined before, clicking on ab will change the names of faults to "Fault 1" to Fault 4". If you want to rename a specific Fault you can double click on it and type the name.

The navigating arrows are used to move a selected fault up or down.

If your created model includes threshold pressure, the related check box of "Has Threshold Pressure" must be checked; the value of this pressure also must be entered. If you load an Eclipse data file in which threshold pressure exists, this checkbox will be checked automatically.

3-1-6- Dual Option

This button is activated only in case of dual (fractured) models. In this window following options are available for user (Figure 80).

- Fracture model: which can be one of Dual Porosity or Dual Permeability
- Shape Factor Formulation: it can be selected one of the following formulation for shape factor:
    - Eclipse formulation
    - Warren and Root (1963)
    - Kazemi et al. (1976)
    - Gilman and Kazemi (1983)
    - Coats (1989)
    - Quandalle and Sabathier (1989)
    - Lim and Aziz (1995)
    - None
- Gravity Drainage and Imbibition: it can be selected one of the following formulation in:
    - GRAVDR (Sonier et al. (1986))
    - GRAVDRM (Quandalle and Sabathier (1989))
    - None
- Gravity Drainage Re-infiltration: it is activated if GRAVDRM is selected for gravity drainage and imbibition.

- L to Sigma Calculations: all L to Sigma calculation values is required just in case of Eclipse formulation for shape factor.
- Dual Porosity Permeability Multiplier: it can be checked if needed in model.

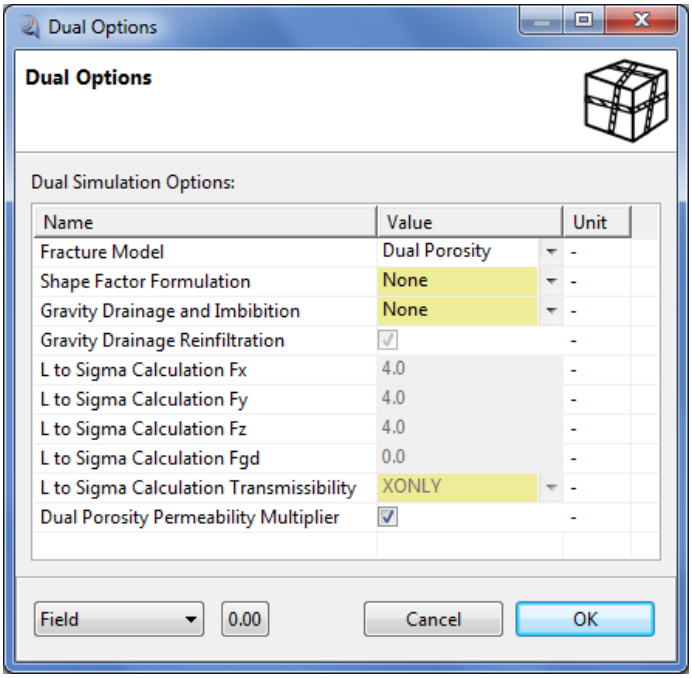


Figure 80- Dual Options

3-1-7- Face Manager

The face manager dialog shows properties of the faces between two grid block. The face id, Cv id1 (first grid block id), Cv id2 (second grid block id), I1,J1,K1 (I,J,K indices of the first grid block), I2,J2,K2 (I,J,K indices of the second grid block), cross section area between two block, NNC Type (non-neighbor connection type), normal vector of face, Permeability direction, Transmissibility and rock region values of the face is displayed in face manager dialog. Face manager has been displayed in Figure 81.

User can use the provided facilities (in the right side of the dialog) to hide some columns, make filters and find the desired parameter. For example user can uncheck the columns which are not necessary for his/her specific purpose (Figure 82).

Adding and removing face filters can be applied using “Filters” expandable item by + and -, respectively. User can find matched values for each parameter by typing the value and selecting the parameter in the “Find” expandable field. The Figure 83 displays the use of Find and Filter.

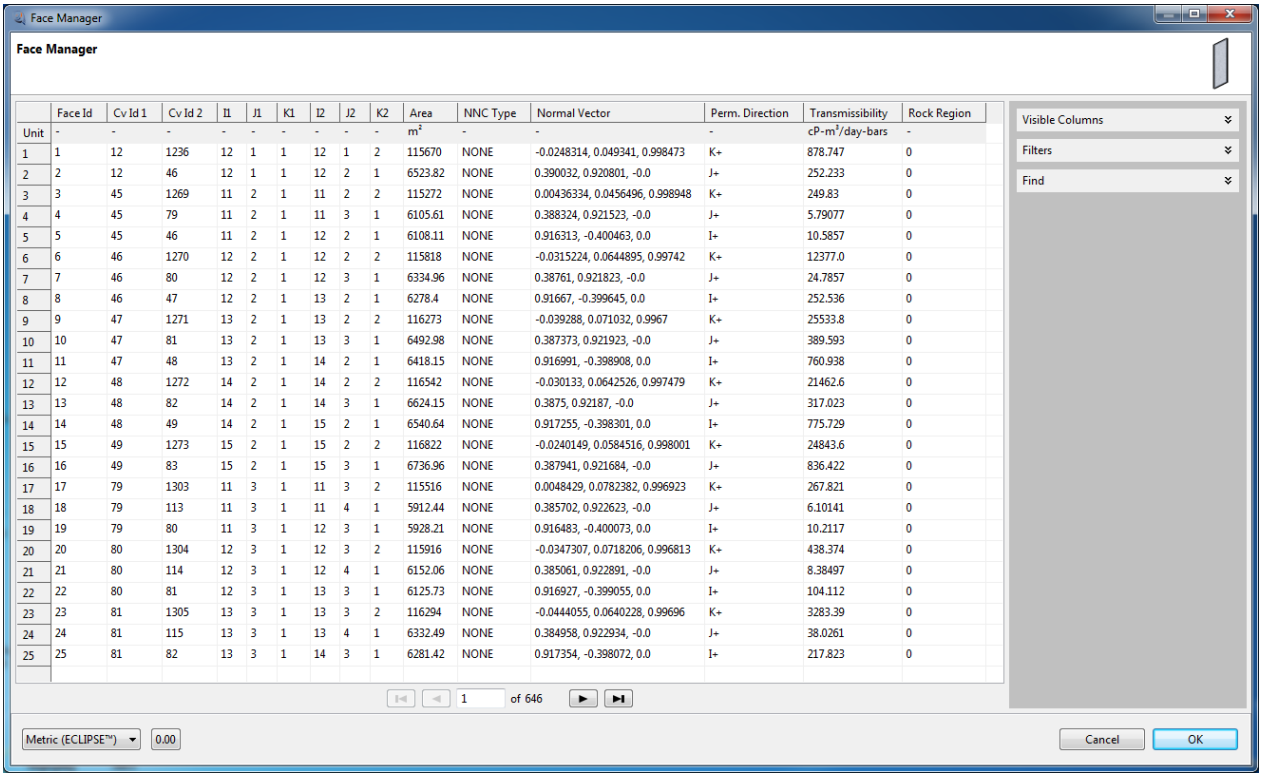


Figure 81 Face Manager Dialog

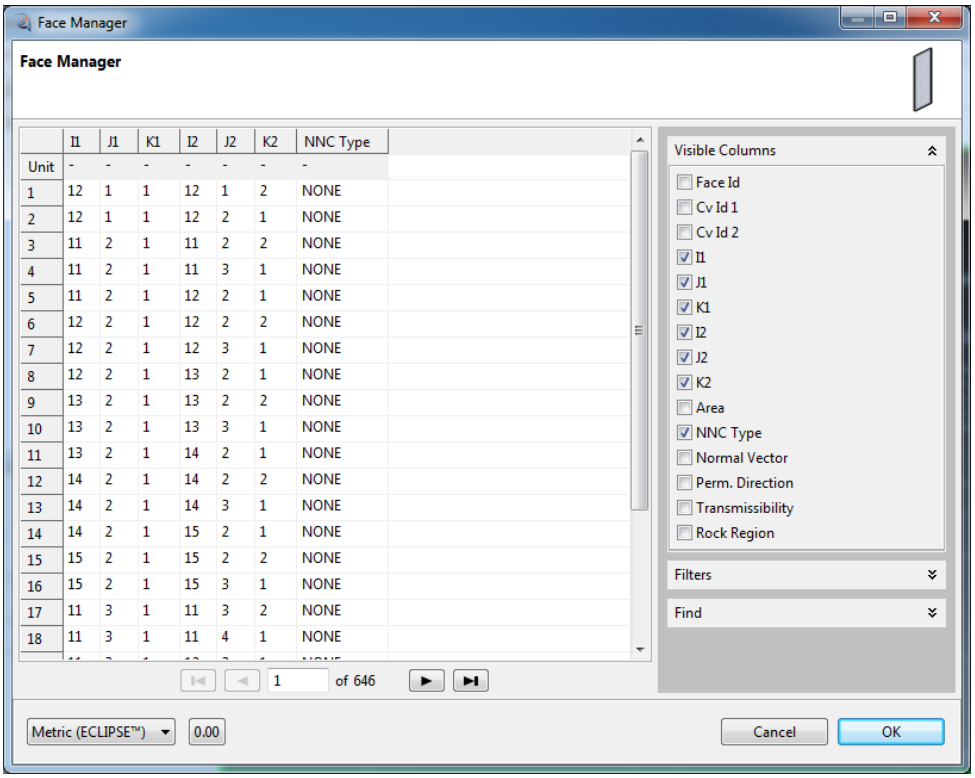


Figure 82 Editing Visibility of Columns in Face Manger

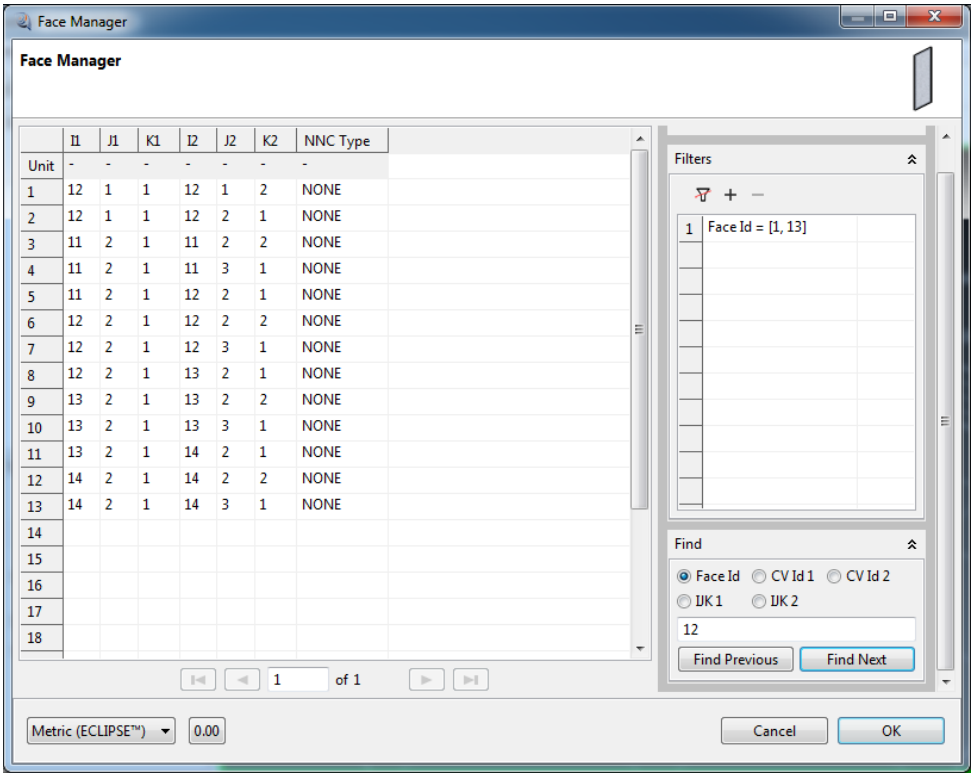


Figure 83 Find and Filter Action on Face Manager

3-1-8- Script Calculator

Property calculator window is used for defining a formula for editing values of some grid or face properties. Face calculator and property calculator dialogs have been displayed in Figure 84 and 85, respectively. Only transmissibility values allowed editing in the face calculator; since all action in the face calculator and property calculator are same we explain property calculator; it is obvious that all explanation are applicable in face calculator, also.

If the project perspective is on the “Simulation Perspective” both face calculator and property calculator (Prop. Calculator) radio buttons are active and user can select each of them; in this perspective all the grid properties except PORV, are read-only. Note that custom property is editable for both perspectives. If the project perspective is on the “Preprocess Perspective” the face calculator will be disabled and PORV will be removed form grid properties; the PORV will be calculated again and displayed in the grid property list if perspective is switched to the Simulation.

For the starting with the script calculator user should type the script name which is optional; in case of blank script name, the first line of script is set to that. In the second step the execution unit must be selected; the project unit is select for this filed, by default.

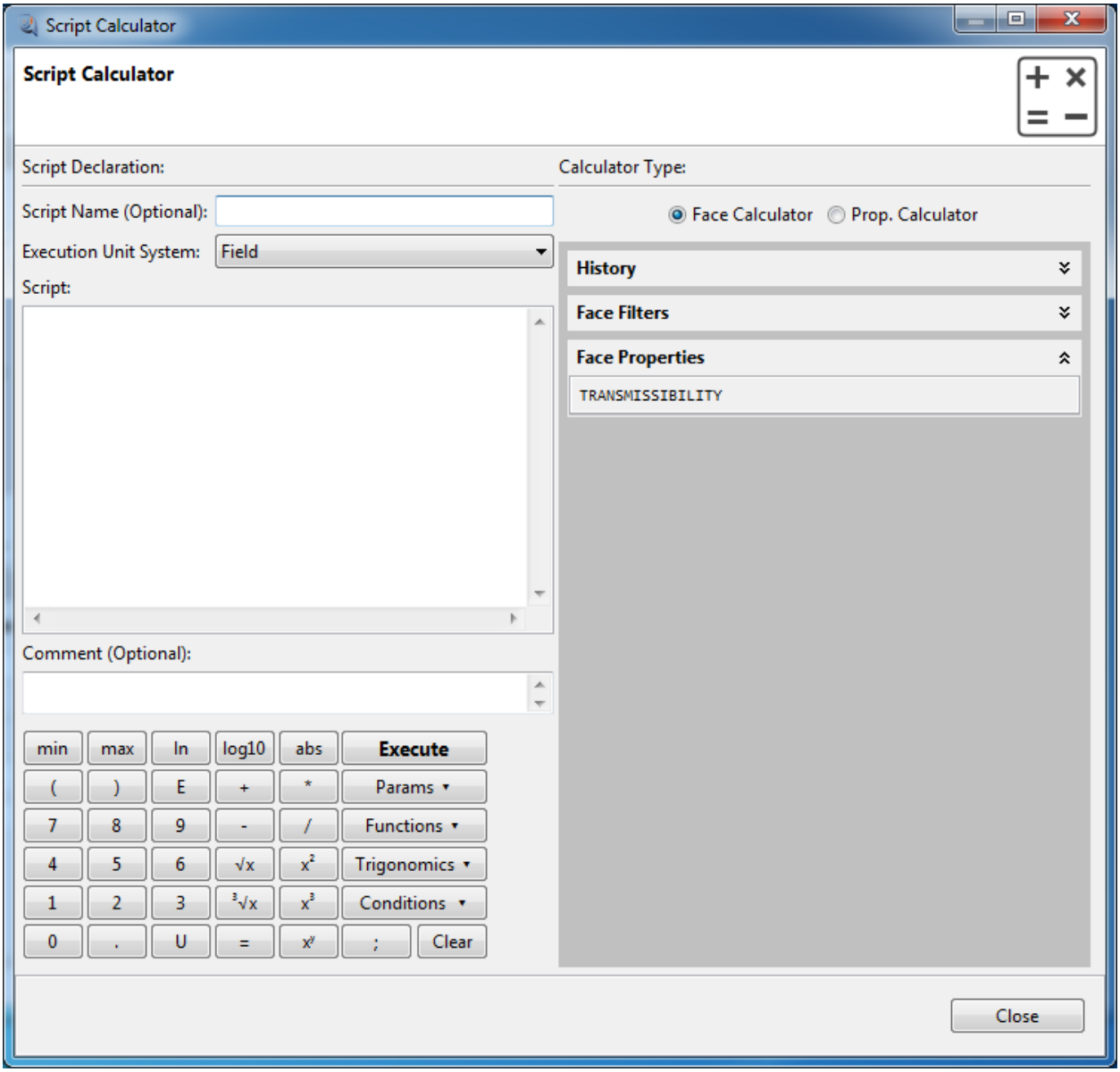


Figure 84- Face Calculator Dialog

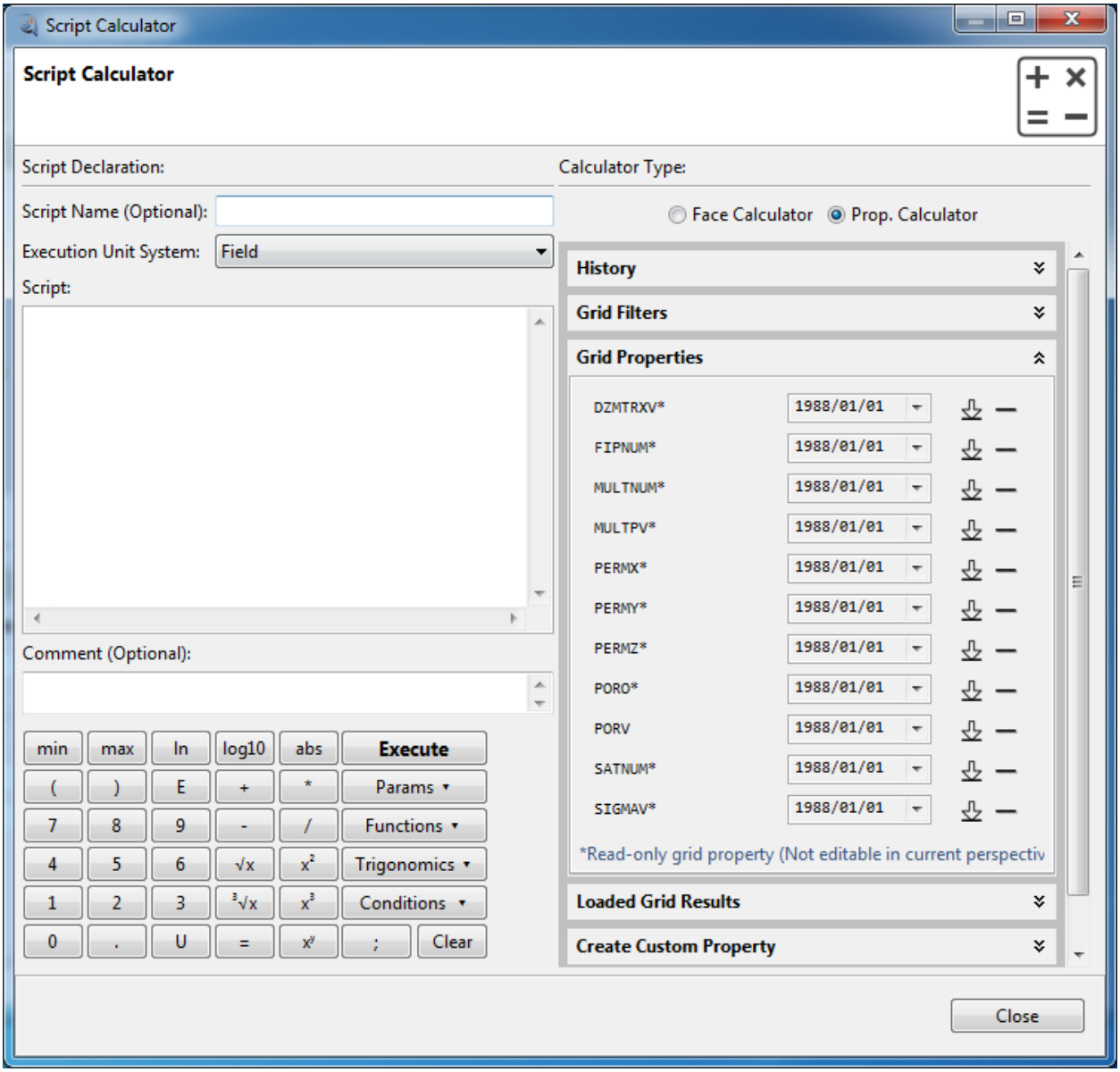


Figure 85 Property Calculator Dialog

In the third step, user can start the writing script in java format, and assign some formula and values to the properties in the property calculator (or transmissibility values in face calculator). User can type code or use the provided buttons, params, Functions, Trigonomics, and Conditions for entering the code to the script. There is also a comment field which user can fill it optionally. After writing code the "Execute" button will apply the code to the model and changes are not irreversible, after execution.

Note: do not forget entering ";" at the end of script statements

The “Params” dropdown button includes I, J, and K indices parameters.

The “Functions” dropdown button includes following functions:

1- If
2- Volume
3- Index
4- Dimension
5- Coordinate
6- Result
7- Property
8- DistanceToWell
9- WellConnection
10- AllWellConnection

The “Trigonomics” dropdown button includes following trigonomic functions:

1- sin
2- cos
3- tan
4- cot
5- asin
6- acos
7- atan
8- acot
9- sinh
10- cosh
11- tanh
12- coth
13- asinh
14- acosh
15- atanh
16- acoth
17- sec
18- csc
19- asec
20- acsc
21- sech
22- csch
23- asech
24- acsch

The “Conditions” dropdown button includes following functions:

1- if
2- else
3- elseif
4- < (less than)
5- <= (less or equal than)
6- == (equal to)
7- >= (greater or equal than)
8- > (greater than)
9- != (not equal)
10- for
11- while
12- do while
13- and
14- or

After executing some scripts you can access to the defined scripts in the “History” section in the right side of the script calculator dialog.

User can apply script to some filters using “Grid Filters” section which is displayed in the Figure 86.

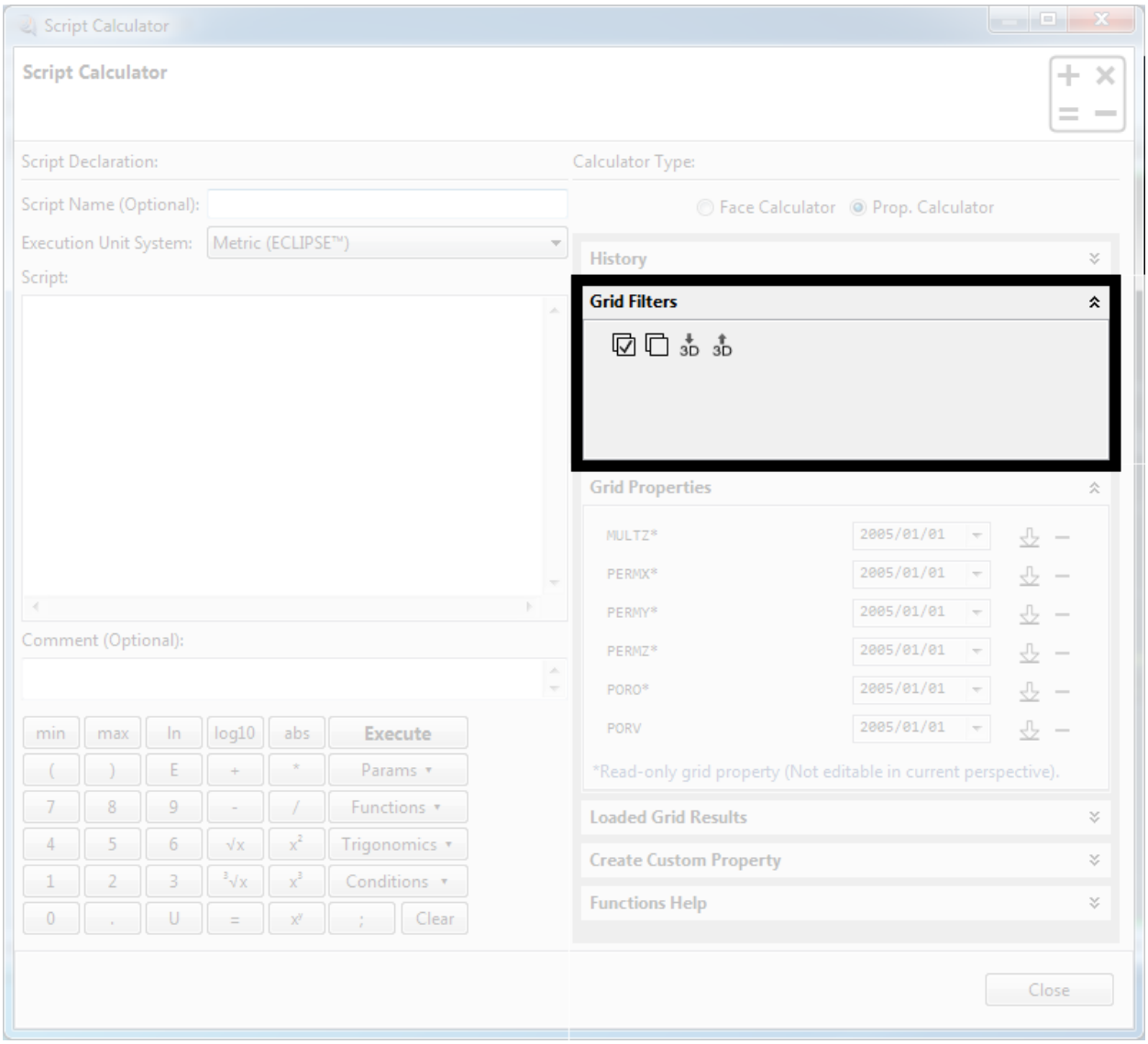


Figure 86 Grid Filters section of Script Calculator

If user defined some filters using left panel of the RETINA main page, the filters will be observable in the Grid Filters section of the script calculator (Figure 87). User can select all, deselect all, or select some desired filters; calculation will be applied on selected filters. In case of no filter, calculations will be applied on all grids. User can apply filters to view or get them from view using two distinct buttons in the Grid Filters section.

There is section of grid properties in the script calculator in which all defined schedulable and non-schedulable grid properties of the model are displayed. These properties are

editable only in pre-process perspective, except PORV. The '*' sign indicates the read-only properties. There is a date combo box in front of each grid property which indicates the define date of each grid property. User can remove each property using "-", or put it into the script using inert button.

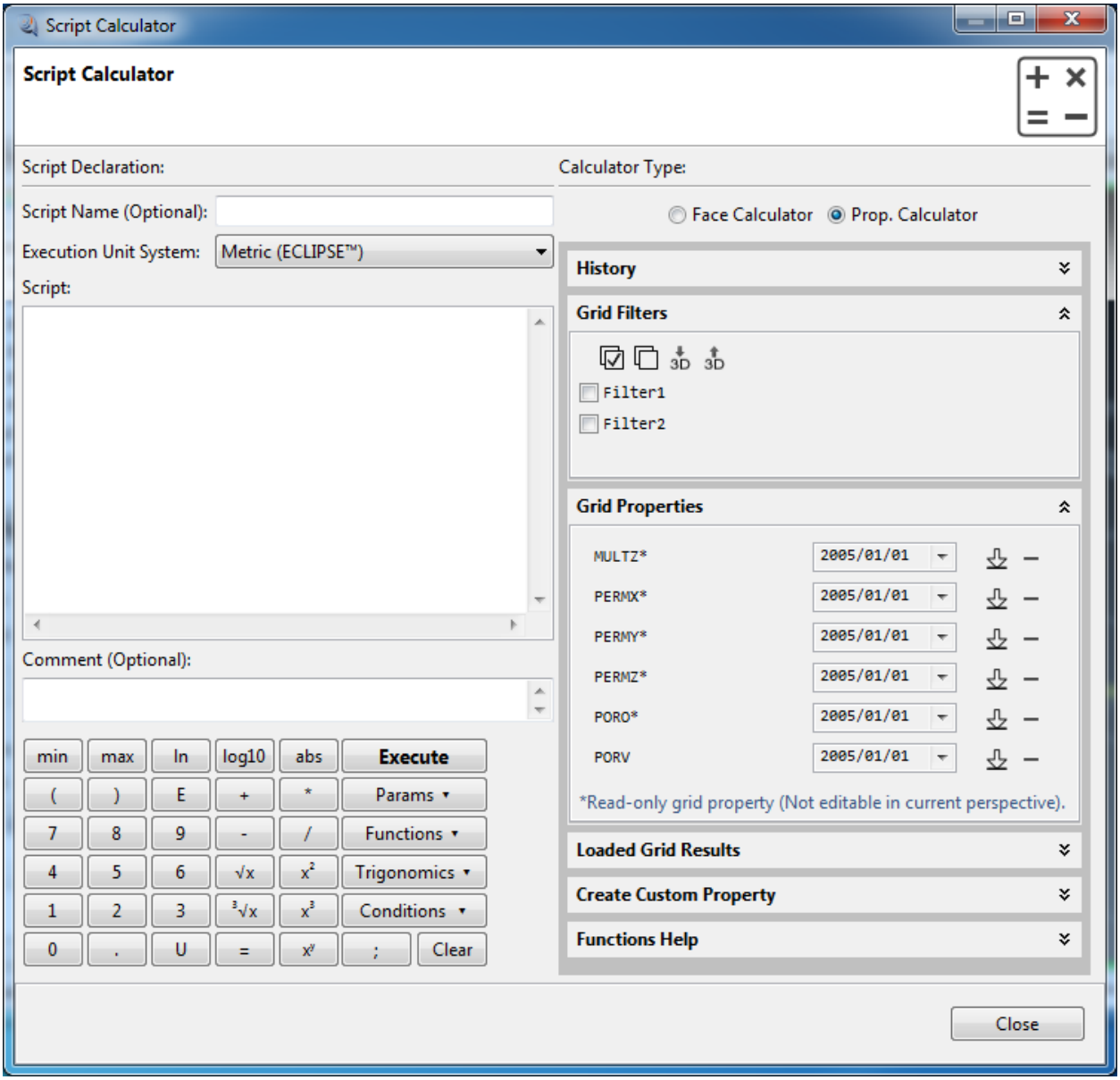


Figure 87 3D View Filters in Script Calculator

If user requests some grid results (the request grid result action will be explained in the "Numerical and Result" data tab) the results can be observable in the "Loaded Grid Results". The loading action of the grid result must be done using "tab" Left panel of the RERINA main page (Figure 88) before performing any action on them.

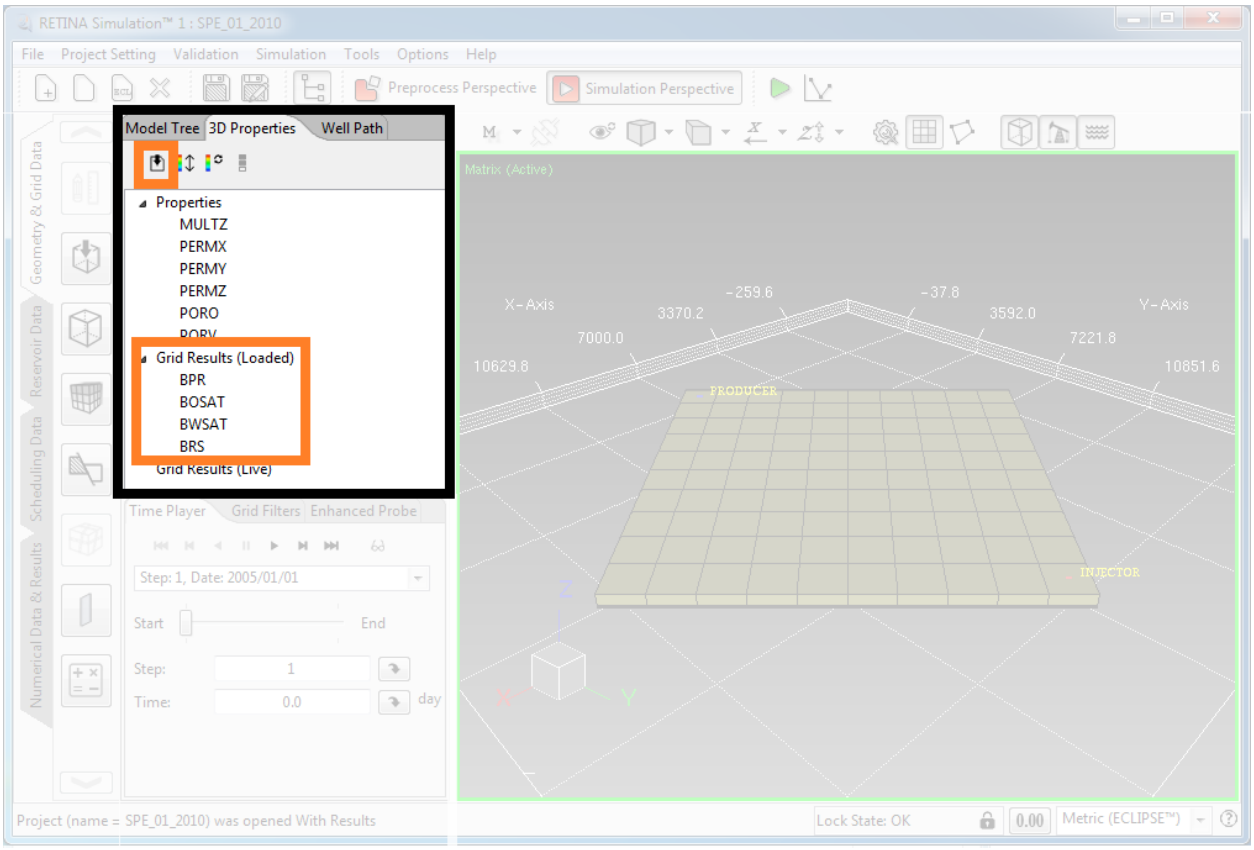


Figure 88 Loading Grid Results

After loading the grid results using Figure 88 instructions, the imported results are displayed in the script calculator (Figure 89). There is a date combo box in front of each loaded grid property which indicates the existing date of each grid property. User can remove each property using "-", or put it into the script using inert button.

The facility of "create Custom Property" is available to user. User can define own custom grid property. The property name, template and value must be entered in the "Create Custom Property" section. After defining the custom grid property, it is added to both the grid property dialog, and script calculator.

Finally there is a "Functions Help" section in the property calculator, in which user can see the functions templates, in a quick view.

Using of "functions" or "conditions" in script calculator might be relatively complicated for the beginners; so here minimal examples for each function and conditions are presented. You can create more complicated scripts using this feature.

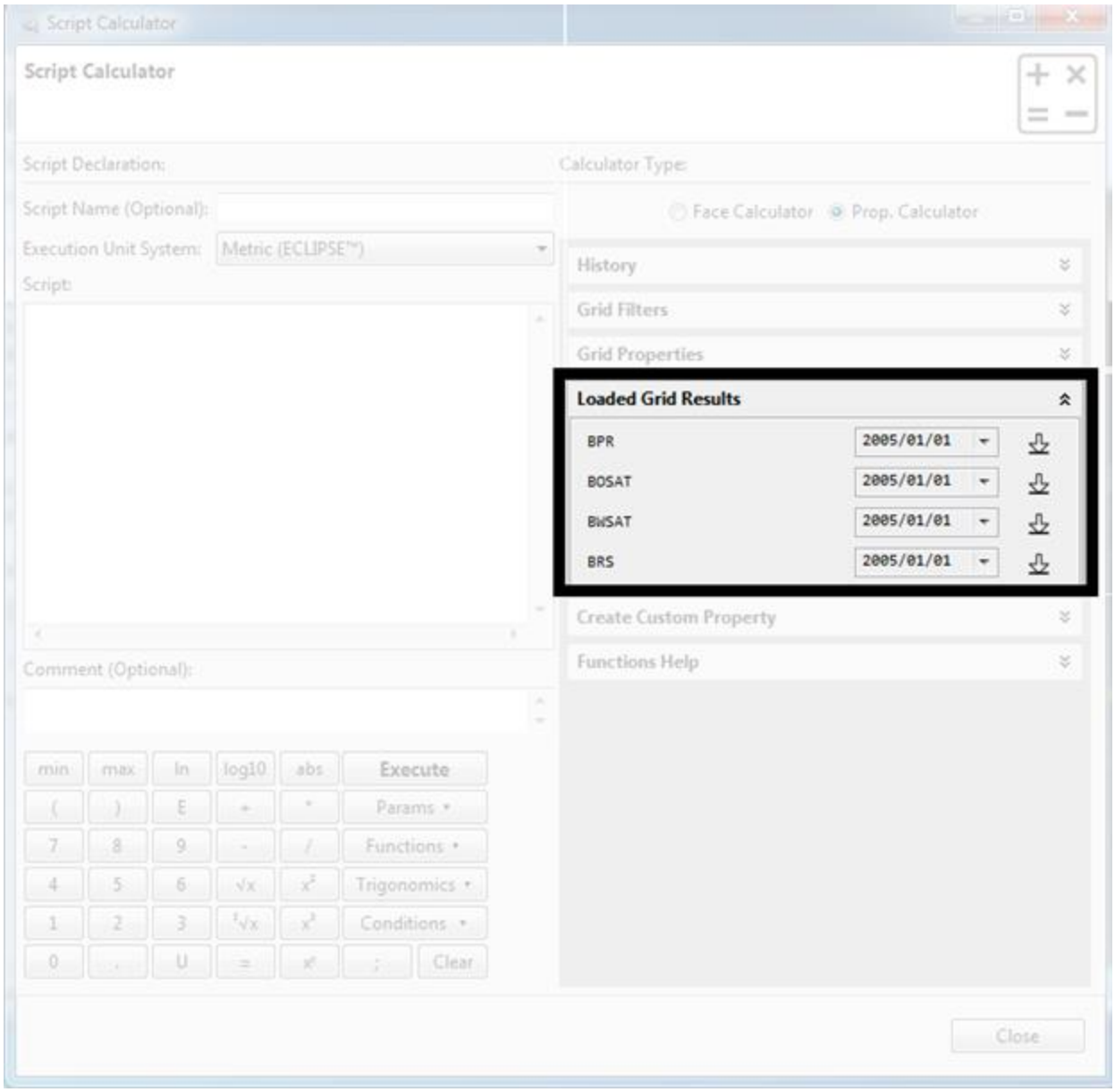


Figure 89 Loaded Grid Results in Script Calculator

Function Examples:

- *Using* "If" *Function:*

It has three inputs; the first input parameter is a logical expression, the second one is the value if the condition is true, and the third input parameter is the value if the condition is false.

Example:

```
If(PERMX>200,5 ,0 );
```

If the value of x-permeability for any grid block is greater than 200, it will be replaced with value of 5, else it will be replaced with 0.

- *Using* "Volume" *Function:*

It has no input; it calculates the geometric volume of each grid block.

Example 1:

```
PERMX=Volume(;
```

The volume of each grid block is calculated and assigned to x-permeability of each grid.

Example 2:

```
A=Volume();
```

The volume of each grid block is calculated and assigned to the user defined property of "A" for each grid.

Example 3:

```
A= If(Volume>5000, 1000 ,500);
```

The A property value will be 1000 for all grid blocks with grid geometric volume greater than 5000; simultaneously, the value of 500 is assigned for the other grid blocks.

- *Using* "Index" *Function:*

It has one input of the I, J, or K index as string in the "I", "J", or "K" format; it calculates the index of each grid block in desired direction. The "I", "J", and "K" parameters can be inserted either using "Params" dropdown list or by typing in the script as mentioned format.

Example 1:

```
PERMX= Index("J");
```

The index of "J" for each grid block is assigned to its x-permeability value.

Example 2:

```
PERMY=If(Index("J")==5, 2.5 ,1.5 );
```

The y-permeability value will be 2.5 for all grid blocks with J-index=5, and the value of 1.5 is assigned for other grid blocks.

- *Using* "Dimension" *Function:*

It has one input of the I, J, or K index as string in the "I", "J", or "K" format; it calculates the dimension of each grid block in desired direction. The "I", "J", and "K" parameters can be inserted either using "Params" dropdown list or by typing in the script as mentioned format.

Example 1:

```
PERMX=Dimension("J");
```

The Dimension of grid block in "J" direction for each grid block is assigned to its x-permeability value.

Example 2:

```
PERMY=If(Dimension("J")>=50, 25 ,15 );
```

The y-permeability value will be 25 for all grid blocks with J direction dimension greater than or equal to the 50, and the value of 15 is assigned for other grid blocks.

- *Using* **"Coordinate"** *Function:*

It has one input of the I, J, or K index as string in the "I", "J", or "K" format; it calculates the coordinate of each grid block in desired direction. The "I", "J", and "K" parameters can be inserted either using "Params" dropdown list or by typing in the script as mentioned format.

Example 1:

```
PERMX=Coordinate ("J");
```

The coordinate of grid block in "J" direction is assigned to each grid block x-permeability value.

Example 2:

```
PERMY=If(Coordinate("J")<12500, 2000 ,1500 );
```

The y-permeability value will be 2000 for all grid blocks with J direction coordinate less than 12500, and the value of 1500 is assigned for other grid blocks.

- *Using* **"Results"** *Function:*

It has two inputs; the first one the name of the desired result grid property as "Name" and the second one is the date in which the result is requested as "YY/MM/DD". If user uses this function the inputs must be inserted by typing; there is another and more user friendlier way to use this function: after importing grid results using import item of the RETINA main page's left panel, a list of "Loaded Grid Results" are displayed in the related section in script calculator; user can insert the result function with its inputs just choosing the date (from date combo box in front of each result) and push the "put into the script" icon; in this way the "Result" function with both input parameters will be inserted automatically to the script.

Example 1:

```
PERMX=Result("BOSAT", "2007/02/01");
```

The result values of block oil saturation at the date 2007/02/01 are assigned

to each grid block x-permeability value.

Example 2:

```
PERMY=If(Result("BOSAT", "2005/07/01")>0.5,10 ,5);
```

The y-permeability value will be 10 for all grid blocks with result block oil saturation at date 2007/07/01 greater than 0.5, and the value of 5 is assigned for other grid blocks.

- *Using* **"Property"** *Function:*

It has two inputs; the first one the name of the desired grid property as `"Name"` and the second one is the date in which the property is requested as `"YY/MM/DD"`. If user uses this function the inputs must be inserted by typing; there is another and more user friendlier way to use this function: if you have schedulable grid properties, user can insert the "Property" function with its inputs just choosing the date (from date combo box in front of each grid) and push the "put into the script" icon; in this way the "Property" function with both input parameters will be inserted automatically to the script. In case of using grid properties at start date, the "property" function does not need to use; only putting desired property in the script is enough.

Example 1:

```
PERMX= Property("MULTX", "2006/02/01");
```

The property values of the transmissibility in x direction at the date 2006/02/01 are assigned to each grid block x-permeability value.

Example 2:

```
PERMY=If(Property("MULTX", "2006/02/01")>0.5,10 ,5);
```

The y-permeability value will be 10 for all grid blocks with transmissibility multiplier value of x direction at date 2006/02/01 greater than 0.5, and the value of 5 are assigned for other grid blocks.

- *Using* **"DistanceToWell"** *Function:*

It has only one input of the well name as `"WellName"` string format. It will give the distance of each grid block to the desired well.

Example 1:

```
PERMX=DistanceToWell("P01");
```

The grid block distance to the well P01 value is assigned to its permeability value in x direction for each grid block.

Example 2:

```
PERMY=If(DistanceToWell("P01")>DistanceToWell("P02"),PERMX,Property("MULTX",
"2006/02/01"));
```

The x permeability value will be assigned to the y permeability value for each grid which its distance to well P01 is greater than its distance to well P02; otherwise the property of transmissibility multiplier in x direction at date 2006/02/01 is assigned to the y permeability value.

- *Using* **"WellConnection"** *Function:*

It has only one input of the well name as `"WellName"` string format. This function will give the connection of the desired well.

Example:

```
A=WellConnection("P01");
```

A includes an array in which the well P01's connections are specified. After using this function you can see the connections on the 3D view.

Using "AllWellConnection" *Function:*

It does not input. This function will give the all connections of the defined wells.

Example:

```
A=AllWellConnection();
```

A includes an array in which the all well's connections are specified. After using this function you can see the connections on the 3D view.

3-2- Using Reservoir Data Tab

Reservoir Data tab is used to define PVT, Rock, Saturation, Equilibration, End Point Scaling, and Aquifer data.

3-2-1- PVT Region Data

The "PVT Region Data" button is used to define PVT regions and API tracking Option. Figure 90 displays the PVT Region Data dialog. The model system phase is displayed as a comment in this dialog. There are two tables in left side of this dialog; 1- PVT region and 2- Table list.

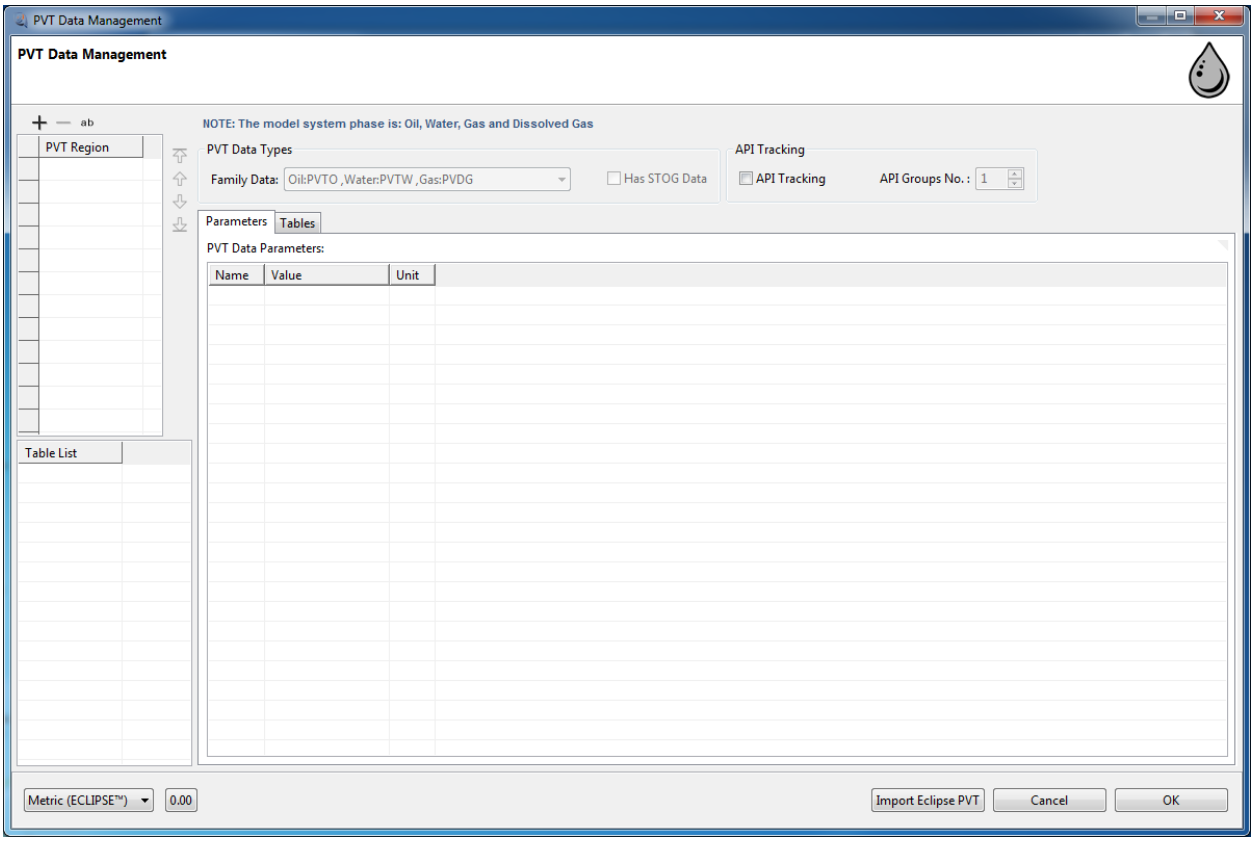


Figure 90 PVT Region Data Dialog

For adding or removing some PVT regions, the **+** and **–** buttons are used, respectively. Add, remove and auto naming is same as section 3-2-6. After adding desired region, window of Figure 91 is displayed, in which all the fields are active. The components of the PVT window will describe in next sections. User can move defined PVT regions using up or down arrows. Same as Fault manager window, there is a button of ab in the PVT data management window, which is used to auto naming of the defined regions. Assume you

want to auto name 3 regions which are defined before, clicking on ab will change the names to “PVT 1” to PVT 3”. If you want to rename a specific PVT region you can double click on it and type the name. In case of API Tracking, after activating it by check the related box, the number of it groups is inserted in “API Group No”.

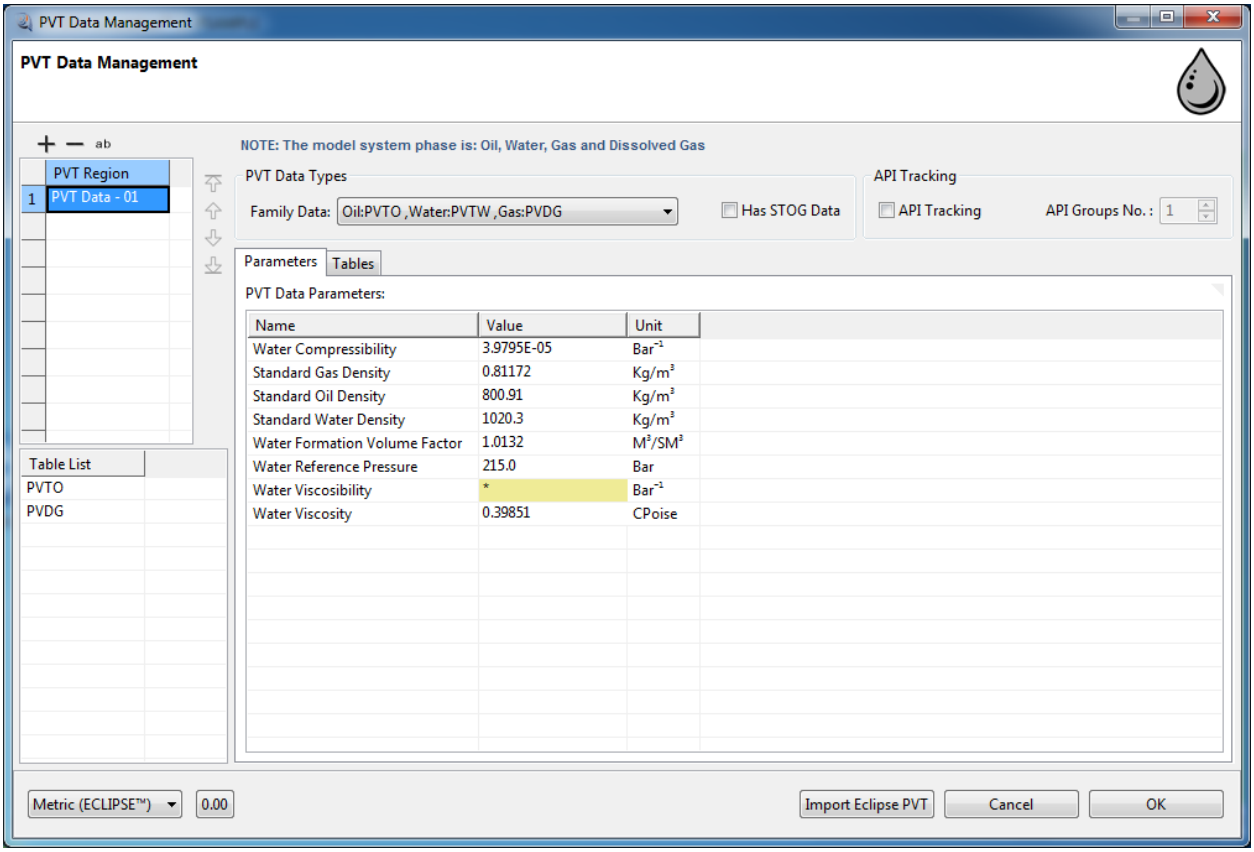


Figure 91- PVT Data Management Dialog after Adding PVT Region Data

Sometimes user wants to copy a PVT Region data to some others. In this case right clicking on the source PVT region and selecting copy or paste can be useful. Also an “Import Eclipse PVT” facility has been provided for PVT data importing in a user friendly way.

PVT Data Types:

In these section types of PVT data for oil, water and gas are selected. If there are some data of oil/gas surface tension the “Has STOG Data” should be checked. There are eight families of the PVT data in this section, which user can select each of the based on the model system phase.

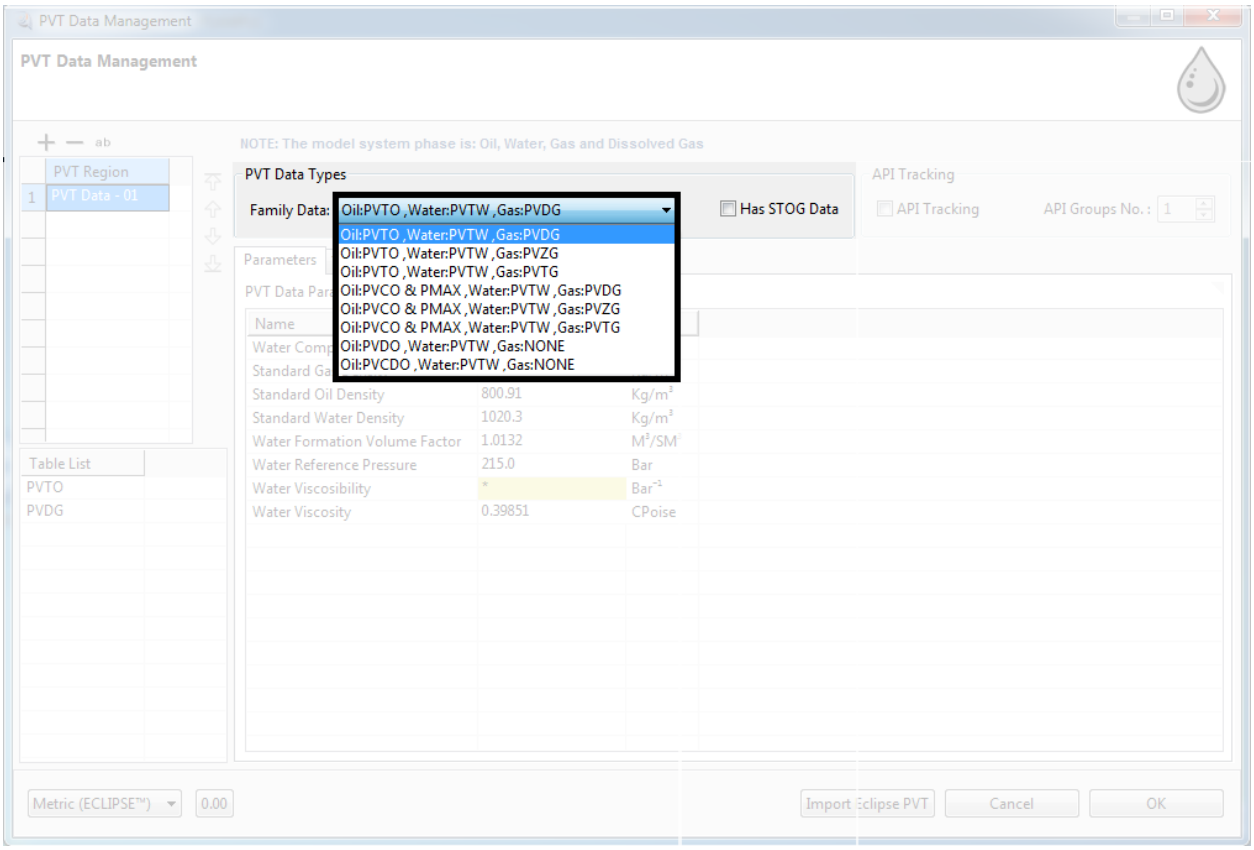


Figure 92- PVT Data Types

Family 1: Oil: PVTO, Water: PVTW, Gas: PVDG:

Only two tables of PVTO and PVDG are necessary for this PVT family data; Note that the PVTW data are not entered in table; this data will be entered in parameters tab.

PVTO Table:

The PVTO table is used to define the PVT properties of live oil (with dissolved gas), in which PVT data for a particular dissolved gas-oil ratio (Rs) are given. Within a PVTO table, records should be arranged in order of increasing Rs (first column). The second column of the PVTO table includes bubble point pressure (Pb) values for oil with dissolved gas-oil ratio given by column 1. The third and fourth columns of the table include oil formation volume factor (Bo), and oil viscosity (μo) for saturated oil at bubble point pressure. Some tables (optionally all) contain additional data which defines the properties of under-saturated oil at the specified value of Rs. This extra data must be specified for the highest Rs in each table. The additional data takes the form of 3 columns that continue from

column 2, 3 and 4 above. In such cases the oil phase pressure (Po) values must increase down the column (starting from Pb), the oil formation volume factor (Bo) corresponding to Rs and Po must decrease down the column, and finally the oil viscosity corresponding to Rs and Po should be increasing down the column.

There must be the same number of entries in each column. If no additional data is supplied for a particular value of Rs, it locates the next higher value of Rs for which under saturated PVT data is available, and takes a scaled copy of the Bo and viscosity curves supplied there. Figure 93 displays a typical PVTO table and related cures.

Note: the data for under saturated oil must be provided for the highest Rs in the PVTO table.

Note: RETINA linearly interpolates the 1/Bo and 1/(Bo.μo) between data points, rather than the values themselves.

Note: In case of using the "API Tracking" option the set of tables must be entered in increasing order of surface density (or decreasing order of API gravity).

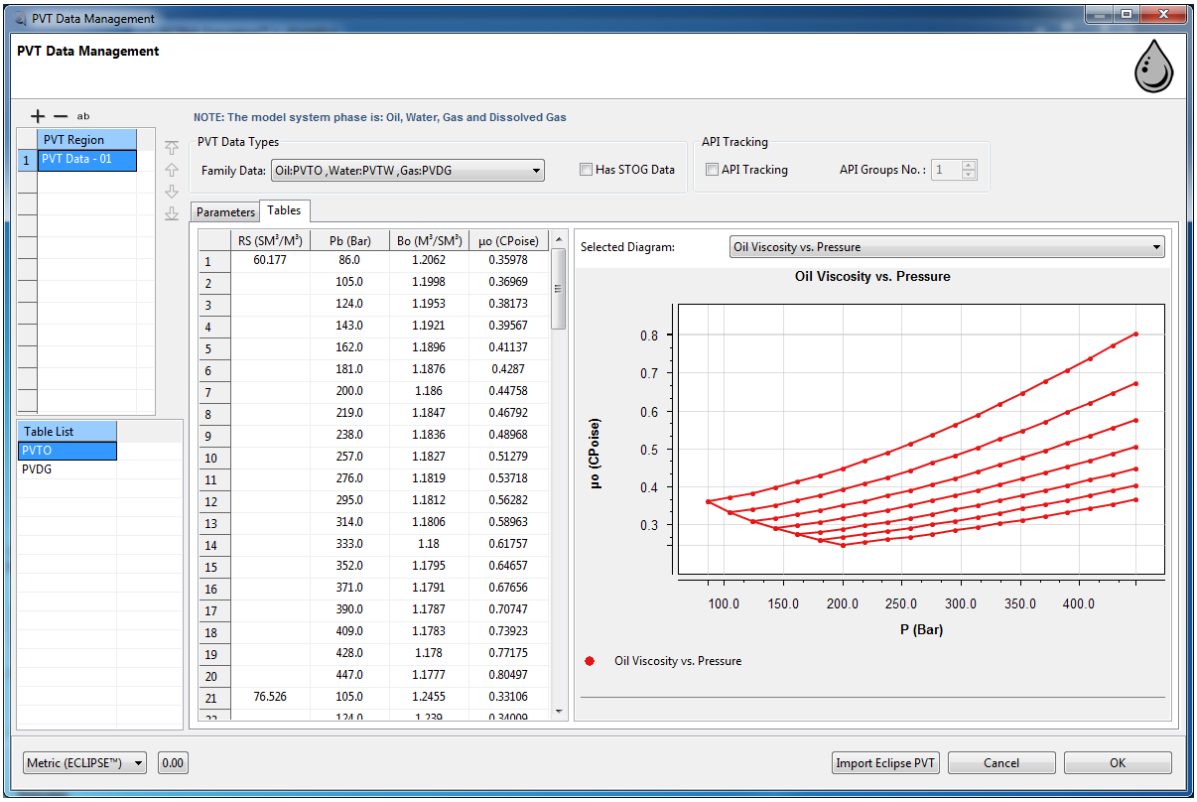


Figure 93 A Typical PVTO Table and Curves

PVDG Table:

The PVDG table is used to define the PVT properties of dry gas (no vaporized oil). Each table consists of three columns of data; the first one is the gas phase pressure which its values should increase monotonically down the column; the second column includes the corresponding gas formation volume factor which its values should decrease down the column; and finally the third column includes the corresponding gas viscosity; the values of the gas viscosity should be level or increasing down the column. There must be the same number of entries in each column of a given table. Figure 94 displays a typical PVDG table and corresponding curve.

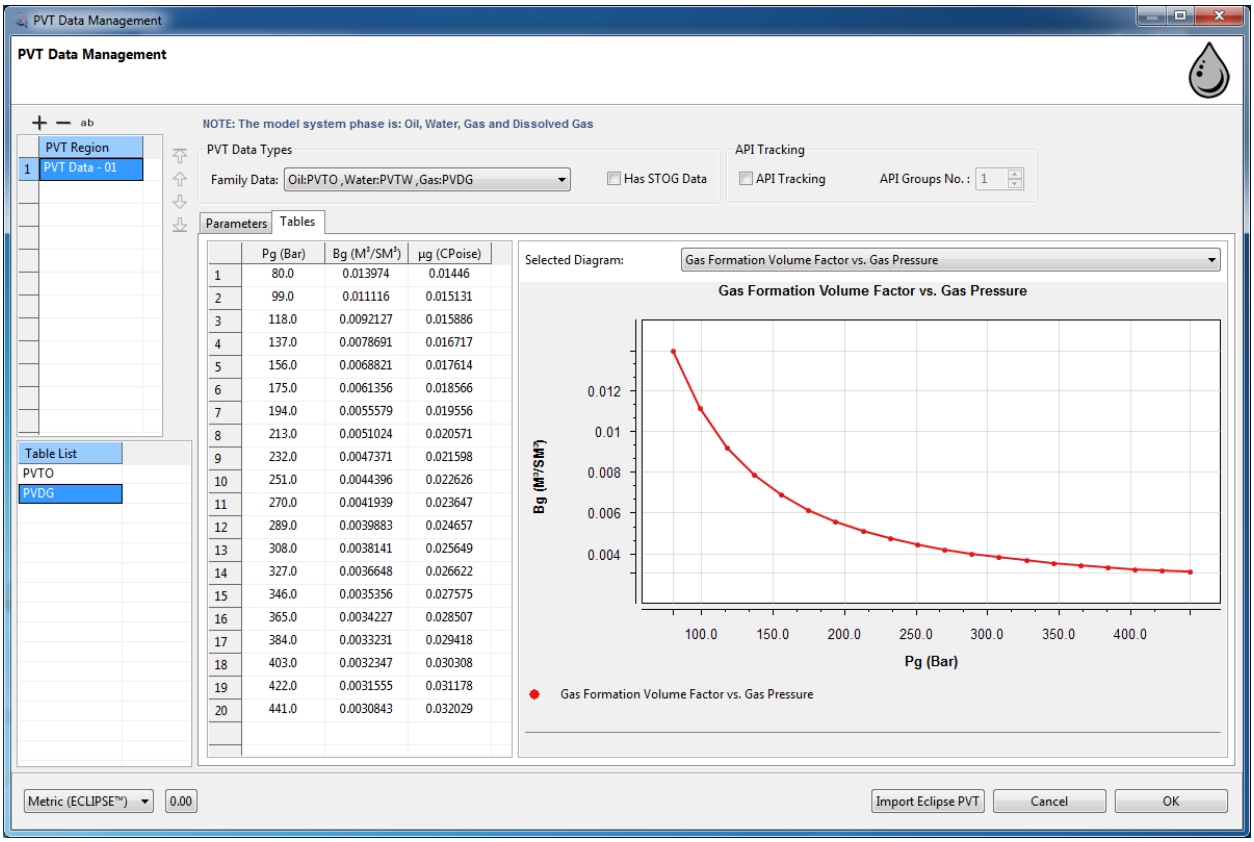


Figure 94 A Typical PVDG Table and Curve

PVTW Data:

The PVTW includes water PVT functions including: The water compressibility, standard gas density, standard oil density, standard water density, water formation volume factor, water reference pressure, water viscosity, and viscosibility. The Figure 95 displays the PVT Data parameters which includes PVTW data.

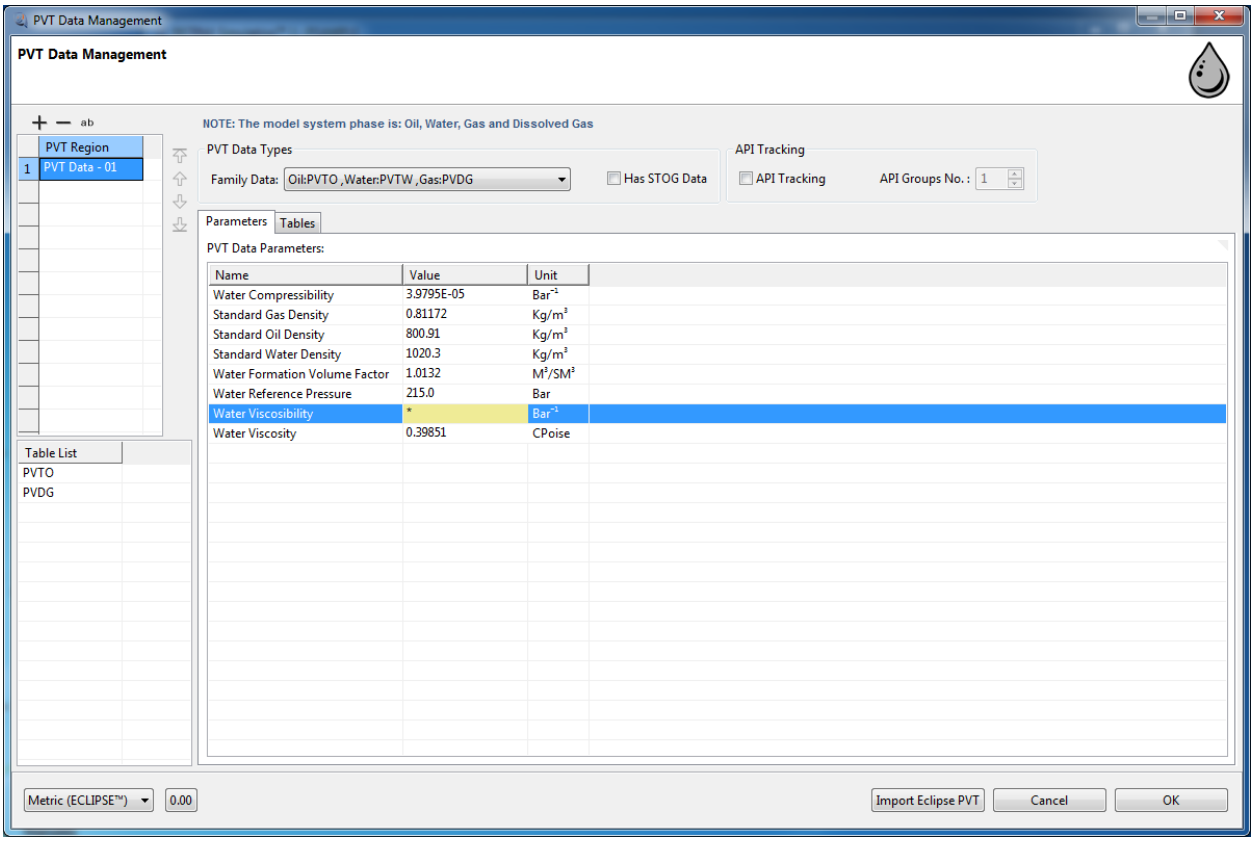


Figure 95 A Typical Water PVT Data

Family 2: Oil: PVTO, Water: PVTW, Gas: PVZG:

Only two tables of PVTO and PVZG are necessary for this PVT family data; Note that the PVTW data are not entered in table; this data will be entered in parameters tab.

PVTO Table:

The PVTO table is used to define the PVT properties of live oil (with dissolved gas), in which PVT data for a particular dissolved gas-oil ratio (Rs) are given. Within a PVTO table, records should be arranged in order of increasing Rs (first column). The second column of the PVTO table includes bubble point pressure (Pb) values for oil with dissolved gas-oil ratio given by column 1. The third and fourth columns of the table include oil formation volume factor (Bo), and oil viscosity (μo) for saturated oil at bubble point pressure. Some tables (optionally all) contain additional data which defines the properties of under-saturated oil at the specified value of Rs. This extra data must be specified for the highest Rs in each table. The additional data takes the form of 3 columns that continue from column 2, 3 and 4 above. In such cases the oil phase pressure (Po) values must increase down the column (starting from Pb), the oil formation volume factor (Bo) corresponding

to Rs and Po must decrease down the column, and finally the oil viscosity corresponding to Rs and Po should be increasing down the column.

There must be the same number of entries in each column. If no additional data is supplied for a particular value of Rs, it locates the next higher value of Rs for which under saturated PVT data is available, and takes a scaled copy of the Bo and viscosity curves supplied there. Figure 93 displays a typical PVTO table and related cures.

Note: the data for under saturated oil must be provided for the highest Rs in the PVTO table.

Note: RETINA linearly interpolates the 1/Bo and 1/(Bo.μo) between data points, rather than the values themselves.

Note: In case of using the "API Tracking" option the set of tables must be entered in increasing order of surface density (or decreasing order of API gravity).

PVTW Data:

The PVTW includes water PVT functions including: The water compressibility, standard gas density, standard oil density, standard water density, water formation volume factor, water reference pressure, water viscosity, and viscosibility. The Figure 95 displays the PVT Data parameters which includes PVTW data.

PVZG Table:

The PVZG table is used to define the PVT properties of dry gas (using Z-factors). Each table consists of three columns of data; the reference temperature should be defined in parameters tab for this table. The three columns of the PVZG table include the gas phase pressure, the corresponding compressibility (Z) factor, and the corresponding gas viscosity, respectively. The gas phase pressure values should increase monotonically down the column, whereas the gas viscosity values should be level or increasing down the column. There must be the same number of entries in each column of a given table. The Figure 96 displays a typical PVZG table and curve.

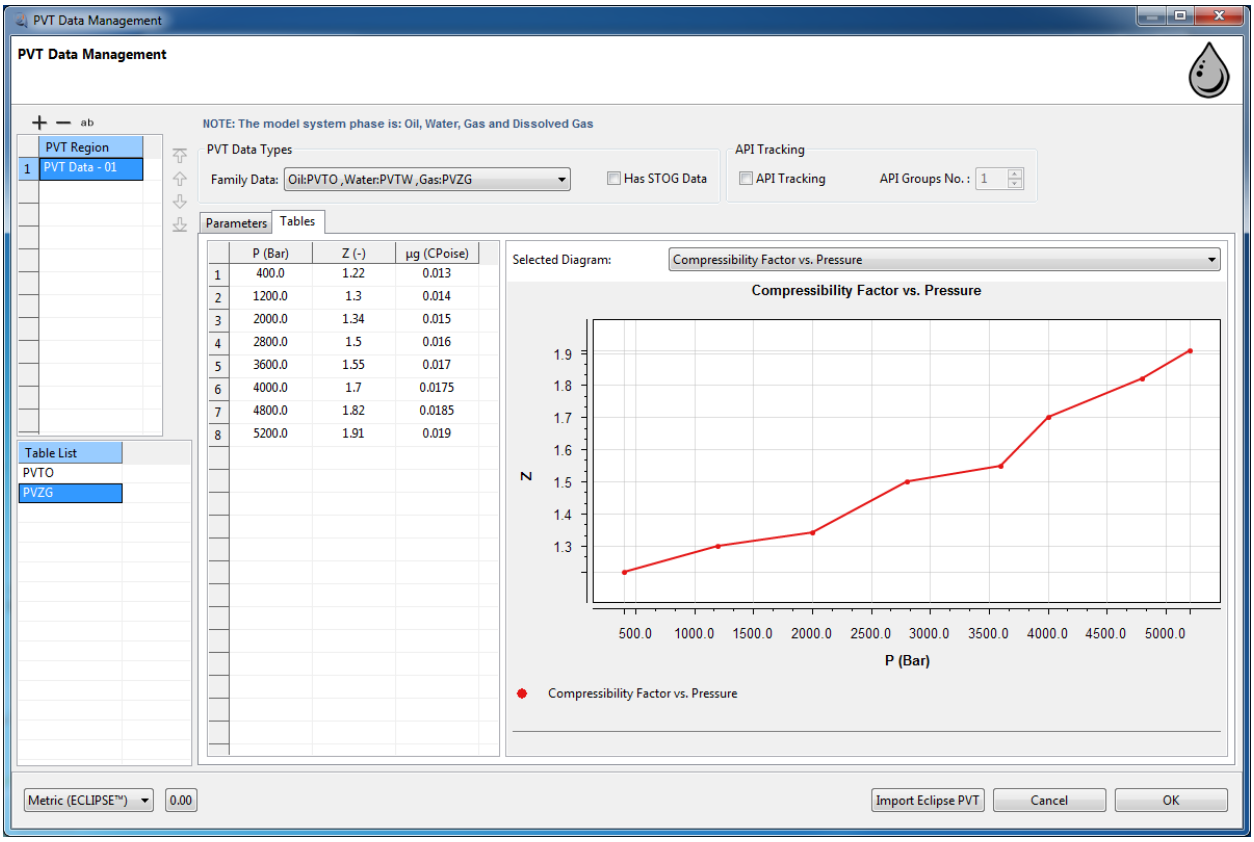


Figure 96 A Typical PVZG Table and Curve

Note: The Z- factor is related to the formation volume factor, reference temperature and pressure.

Family 3: Oil: PVTO, Water: PVTW, Gas: PVTG:

Only two tables of PVTO and PVTG are necessary for this PVT family data; Note that the PVTW data are not entered in table; this data will be entered in parameters tab.

PVTO Table:

The PVTO table is used to define the PVT properties of live oil (with dissolved gas), in which PVT data for a particular dissolved gas-oil ratio (Rs) are given. Within a PVTO table, records should be arranged in order of increasing Rs (first column). The second column of the PVTO table includes bubble point pressure (Pb) values for oil with dissolved gas-oil ratio given by column 1. The third and fourth columns of the table include oil formation volume factor (Bo), and oil viscosity (μo) for saturated oil at bubble point pressure. Some tables (optionally all) contain additional data which defines the properties of under-saturated oil at the specified value of Rs. This extra data must be specified for the highest

Rs in each table. The additional data takes the form of 3 columns that continue from column 2, 3 and 4 above. In such cases the oil phase pressure (Po) values must increase down the column (starting from Pb), the oil formation volume factor (Bo) corresponding to Rs and Po must decrease down the column, and finally the oil viscosity corresponding to Rs and Po should be increasing down the column.

There must be the same number of entries in each column. If no additional data is supplied for a particular value of Rs, it locates the next higher value of Rs for which under saturated PVT data is available, and takes a scaled copy of the Bo and viscosity curves supplied there. Figure 93 displays a typical PVTO table and related cures.

Note: the data for under saturated oil must be provided for the highest Rs in the PVTO table.

Note: RETINA linearly interpolates the 1/Bo and 1/(Bo.μo) between data points, rather than the values themselves.

Note: In case of using the "API Tracking" option the set of tables must be entered in increasing order of surface density (or decreasing order of API gravity).

PVTW Data:

The PVTW includes water PVT functions including: The water compressibility, standard gas density, standard oil density, standard water density, water formation volume factor, water reference pressure, water viscosity, and viscosibility. The Figure 95 displays the PVT Data parameters which includes PVTW data.

PVTG Table:

The PVTG table is used to define the PVT properties of wet gas (with vaporized oil). Each table gives PVT data for a particular gas phase pressure (Pg). There are four columns in the PVTG table; the first column includes the gas phase pressure in which the values should be increasing down the column. The second column includes vaporized oil-gas ratio (Rv) for saturated gas at pressure Pg; the third and fourth items are the gas formation volume factor for saturated gas, and the gas viscosity for saturated gas at Pg, respectively.

Always PVTG tables contain additional data that defines the properties of under saturated gas at the specified value of gas phase pressure. This data must be specified for

the highest Pg in each table. The additional data takes the form of three columns which continue from items 2, 3 and 4 above. In such cases, the values of Rv must decrease down the column (starting from the value for saturated gas); There must be the same number of entries in each column. If no additional data is supplied for a particular value of Pg, it locates the next higher value of Pg for which under saturated PVT data is available, and takes a scaled copy of the formation volume factor and viscosity curves supplied there. The typical PVTG table and curves have been displayed in the Figure 97.

Note: the data for under saturated gas must be provided for the highest Pg in the PVTG table.

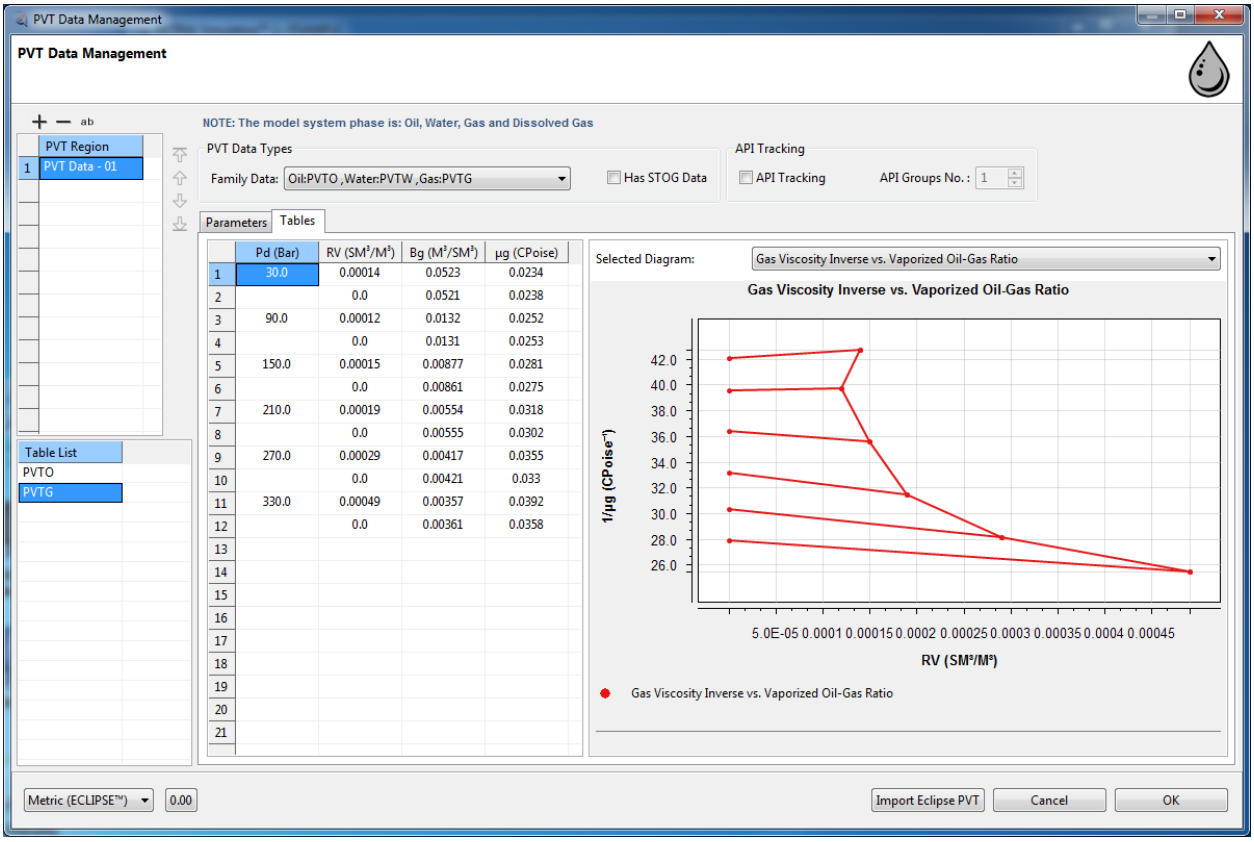


Figure 97 A Typical PVTG Table and Curves

Family 4: Oil: PVCO & PMAX, Water: PVTW, Gas: PVDG:

Only two tables of PVCO and PVDG are necessary for this PVT family data; Note that the PVTW data are not entered in table; this data will be entered in parameters tab. the maximum pressure is available in the parameter tab for this type of data.

PVCO Table:

The PVCO table is used to define PVT properties of live oil in compressibility form (with dissolved gas). This data family offers a simpler method of entering tables of live oil PVT properties than with the PVTO. The difference between the two families is that the PVCO assumes that under saturated oil with a particular Rs value has a compressibility that is independent of the pressure, and that the viscosity of under saturated oil has a pressure-independent derivative. Thus there is no need to provide tables of under saturated oil formation volume factor and viscosity versus pressure. Each table consists of six columns.

- First column: The bubble point pressure (Pb) for oil with the dissolved gas-oil ratio given by second column. In each table, the pressure values should increase monotonically down the column.
- Second column: The dissolved gas-oil ratio (Rs) of saturated oil with bubble point given by first column. The values should increase monotonically down the column.
- Third column: The formation volume factor of saturated oil at Pb.
- Fourth column: The viscosity of saturated oil at Pb.
- Fifth column: The compressibility of under saturated oil with a dissolved gas-oil ratio given by the second column.
- Sixth column: The "viscosibility", or viscosity compressibility, of under saturated oil with a dissolved gas-oil ratio given by the second column.

If the PVCO including family is used, the maximum pressure (PMAX) expected in the simulation must also be entered. This information is used for two purposes. Firstly, the under saturated parts of the oil PVT tables are automatically filled out in the form of the PVTO table, up to the pressure entered in PMAX. Secondly, RETINA carries out a check to see if compressibility values are changing too rapidly with so that the curves of under saturated oil formation volume factor for two different values of cross each other. A warning is printed if they cross at a pressure below the maximum pressure entered in PMAX.

Note: RETINA linearly interpolates the 1/Bo and 1/(Bo.μo) between data points, rather than the values themselves.

Note: When using the API Tracking option, the set of tables must be entered in increasing order of surface density (or decreasing order of API gravity).

PVTW Data:

The PVTW includes water PVT functions including: The water compressibility, standard gas density, standard oil density, standard water density, water formation volume factor, water reference pressure, water viscosity, and viscosibility. The Figure 95 displays the PVT Data parameters which includes PVTW data.

PVDG Table:

The PVDG table is used to define the PVT properties of dry gas (no vaporized oil). Each table consists of three columns of data; the first one is the gas phase pressure which its values should increase monotonically down the column; the second column includes the corresponding gas formation volume factor which its values should decrease down the column; and finally the third column includes the corresponding gas viscosity; the values of the gas viscosity should be level or increasing down the column. There must be the same number of entries in each column of a given table. Figure 94 displays a typical PVDG table and corresponding curve.

Family 5: Oil: PVCO & PMAX, Water: PVTW, Gas: PVZG:

Only two tables of PVCO and PVZG are necessary for this PVT family data; Note that the PVTW data are not entered in table; this data will be entered in parameters tab.

PVCO Table:

The PVCO table is used to define PVT properties of live oil in compressibility form (with dissolved gas). This data family offers a simpler method of entering tables of live oil PVT properties than with the PVTO. The difference between the two families is that the PVCO assumes that under saturated oil with a particular Rs value has a compressibility that is independent of the pressure, and that the viscosity of under saturated oil has a pressure-independent derivative. Thus there is no need to provide tables of under saturated oil formation volume factor and viscosity versus pressure. Each table consists of six columns.

- First column: The bubble point pressure (Pb) for oil with the dissolved gas-oil ratio given by second column. In each table, the pressure values should increase monotonically down the column.
- Second column: The dissolved gas-oil ratio (Rs) of saturated oil with bubble point given by first column. The values should increase monotonically down the column.
- Third column: The formation volume factor of saturated oil at Pb.
- Fourth column: The viscosity of saturated oil at Pb.

- Fifth column: The compressibility of under saturated oil with a dissolved gas-oil ratio given by the second column.
- Sixth column: The "viscosibility", or viscosity compressibility, of under saturated oil with a dissolved gas-oil ratio given by the second column.

If the PVCO including family is used, the maximum pressure (PMAX) expected in the simulation must also be entered. This information is used for two purposes. Firstly, the under saturated parts of the oil PVT tables are automatically filled out in the form of the PVTO table, up to the pressure entered in PMAX. Secondly, RETINA carries out a check to see if compressibility values are changing too rapidly with so that the curves of under saturated oil formation volume factor for two different values of cross each other. A warning is printed if they cross at a pressure below the maximum pressure entered in PMAX.

Note: RETINA linearly interpolates the 1/Bo and 1/(Bo.μo) between data points, rather than the values themselves.

Note: When using the API Tracking option, the set of tables must be entered in increasing order of surface density (or decreasing order of API gravity).

PVTW Data:

The PVTW includes water PVT functions including: The water compressibility, standard gas density, standard oil density, standard water density, water formation volume factor, water reference pressure, water viscosity, and viscosibility. The Figure 95 displays the PVT Data parameters which includes PVTW data.

PVZG Table:

The PVZG table is used to define the PVT properties of dry gas (using Z-factors). Each table consists of three columns of data; the reference temperature should be defined in parameters tab for this table. The three columns of the PVZG table include the gas phase pressure, the corresponding compressibility (Z) factor, and the corresponding gas viscosity, respectively. The gas phase pressure values should increase monotonically down the column, whereas the gas viscosity values should be level or increasing down the column. There must be the same number of entries in each column of a given table. The Figure 96 displays a typical PVZG table and curve.

Family 6: Oil: PVCO & PMAX, Water: PVTW, Gas: PVTG:

Only two tables of PVCO and PVTG are necessary for this PVT family data; Note that the PVTW data are not entered in table; this data will be entered in parameters tab.

PVCO Table:

The PVCO table is used to define PVT properties of live oil in compressibility form (with dissolved gas). This data family offers a simpler method of entering tables of live oil PVT properties than with the PVTO. The difference between the two families is that the PVCO assumes that under saturated oil with a particular Rs value has a compressibility that is independent of the pressure, and that the viscosity of under saturated oil has a pressure-independent derivative. Thus there is no need to provide tables of under saturated oil formation volume factor and viscosity versus pressure. Each table consists of six columns.

- First column: The bubble point pressure (Pb) for oil with the dissolved gas-oil ratio given by second column. In each table, the pressure values should increase monotonically down the column.
- Second column: The dissolved gas-oil ratio (Rs) of saturated oil with bubble point given by first column. The values should increase monotonically down the column.
- Third column: The formation volume factor of saturated oil at Pb.
- Fourth column: The viscosity of saturated oil at Pb.
- Fifth column: The compressibility of under saturated oil with a dissolved gas-oil ratio given by the second column.
- Sixth column: The “viscosibility”, or viscosity compressibility, of under saturated oil with a dissolved gas-oil ratio given by the second column.

If the PVCO including family is used, the maximum pressure (PMAX) expected in the simulation must also be entered. This information is used for two purposes. Firstly, the under saturated parts of the oil PVT tables are automatically filled out in the form of the PVTO table, up to the pressure entered in PMAX. Secondly, RETINA carries out a check to see if compressibility values are changing too rapidly with so that the curves of under saturated oil formation volume factor for two different values of cross each other. A warning is printed if they cross at a pressure below the maximum pressure entered in PMAX.

Note: RETINA linearly interpolates the 1/Bo and 1/(Bo.μo) between data points, rather than the values themselves.

Note: When using the API Tracking option, the set of tables must be entered in increasing order of surface density (or decreasing order of API gravity).

PVTW Data:

The PVTW includes water PVT functions including: The water compressibility, standard gas density, standard oil density, standard water density, water formation volume factor, water reference pressure, water viscosity, and viscosibility. The Figure 95 displays the PVT Data parameters which includes PVTW data.

PVTG Table:

The PVTG table is used to define the PVT properties of wet gas (with vaporized oil). Each table gives PVT data for a particular gas phase pressure (Pg). There are four columns in the PVTG table; the first column includes the gas phase pressure in which the values should be increasing down the column. The second column includes vaporized oil-gas ratio (Rv) for saturated gas at pressure Pg; the third and fourth items are the gas formation volume factor for saturated gas, and the gas viscosity for saturated gas at Pg, respectively.

Always PVTG tables contain additional data that defines the properties of under saturated gas at the specified value of gas phase pressure. This data must be specified for the highest Pg in each table. The additional data takes the form of three columns which continue from items 2, 3 and 4 above. In such cases, the values of Rv must decrease down the column (starting from the value for saturated gas); There must be the same number of entries in each column. If no additional data is supplied for a particular value of Pg, it locates the next higher value of Pg for which under saturated PVT data is available, and takes a scaled copy of the formation volume factor and viscosity curves supplied there. The typical PVTG table and curves have been displayed in the Figure 97.

Note: the data for under saturated gas must be provided for the highest Pg in the PVTG table.

Family 7: Oil: PVDO, Water: PVTW, Gas: NONE:

Only two tables of PVDO table is necessary for this PVT family data; Note that the PVTW data are not entered in table; this family is for defining dead oil and no gas PVT is necessary for this data family.

PVDO Table:

The PVDO table is used to define PVT properties of dead oil (no dissolved gas). Each table consists of three columns. The first column is the oil phase pressure; the oil pressure values should increase monotonically down the column. The second column includes the corresponding oil formation volume factor in which, the values should decrease down the column. Third column is the corresponding oil viscosity. The oil viscosity values should be level or increasing down the column. There must be the same number of entries in each column of a given table. Figure 98 displays a typical PVDO table and curve. Two parameters of bubble point pressure and constant dissolved gas concentration must be entered in parameters tab.

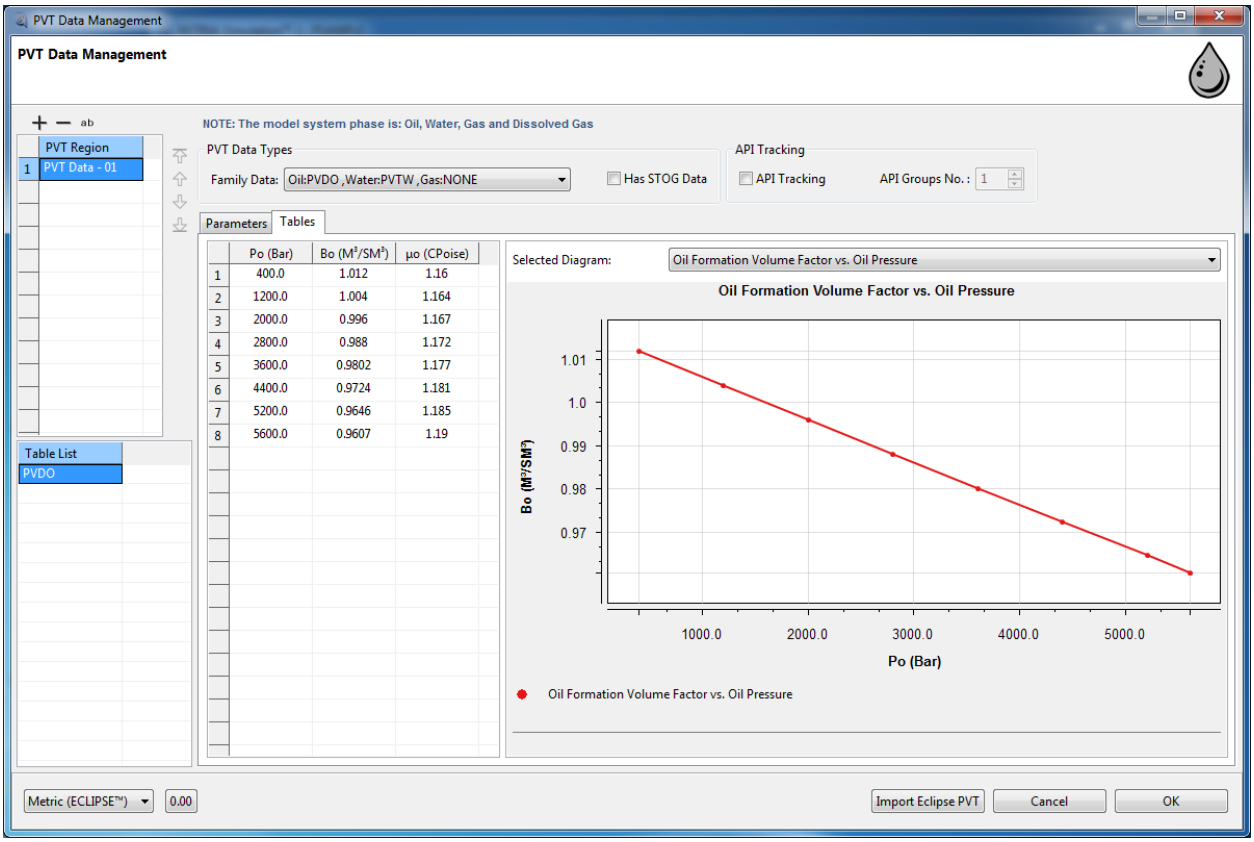


Figure 98 A Typical PVDO Table and Curves

PVTW Data:

The PVTW includes water PVT functions including: The water compressibility, standard gas density, standard oil density, standard water density, water formation volume factor, water reference pressure, water viscosity, and viscosibility. The Figure 95 displays the PVT Data parameters which includes PVTW data.

Family 8: Oil: PVCDO, Water: PVTW, Gas: NONE:

PVCDO Data:

There is no table in this family of the PVT data. All the values should be entered in parameters tab. the PVCDO data is used to define the dead oil PVT properties (with constant compressibility). It includes following parameters: oil formation volume factor, oil compressibility, oil viscosity, oil viscosibility, oil reference pressure, bubble point pressure, and dissolved gas concentration.

PVTW Data:

The PVTW includes water PVT functions including: The water compressibility, standard gas density, standard oil density, standard water density, water formation volume factor, water reference pressure, water viscosity, and viscosibility. The Figure 95 displays the PVT Data parameters which includes PVTW data.

PVT Plot Setting:

User can use right click on the plot to see setting (Figure 99). Setting includes save, fit data, plot property. The zoom action is performed using mouse wheel. Clicking on “Properties” leads user to the “Chart Properties” window. In this window, user can edit title, description and related fonts, axis setting like axis minimum and maximum values, scales, marker types, colors and etc.

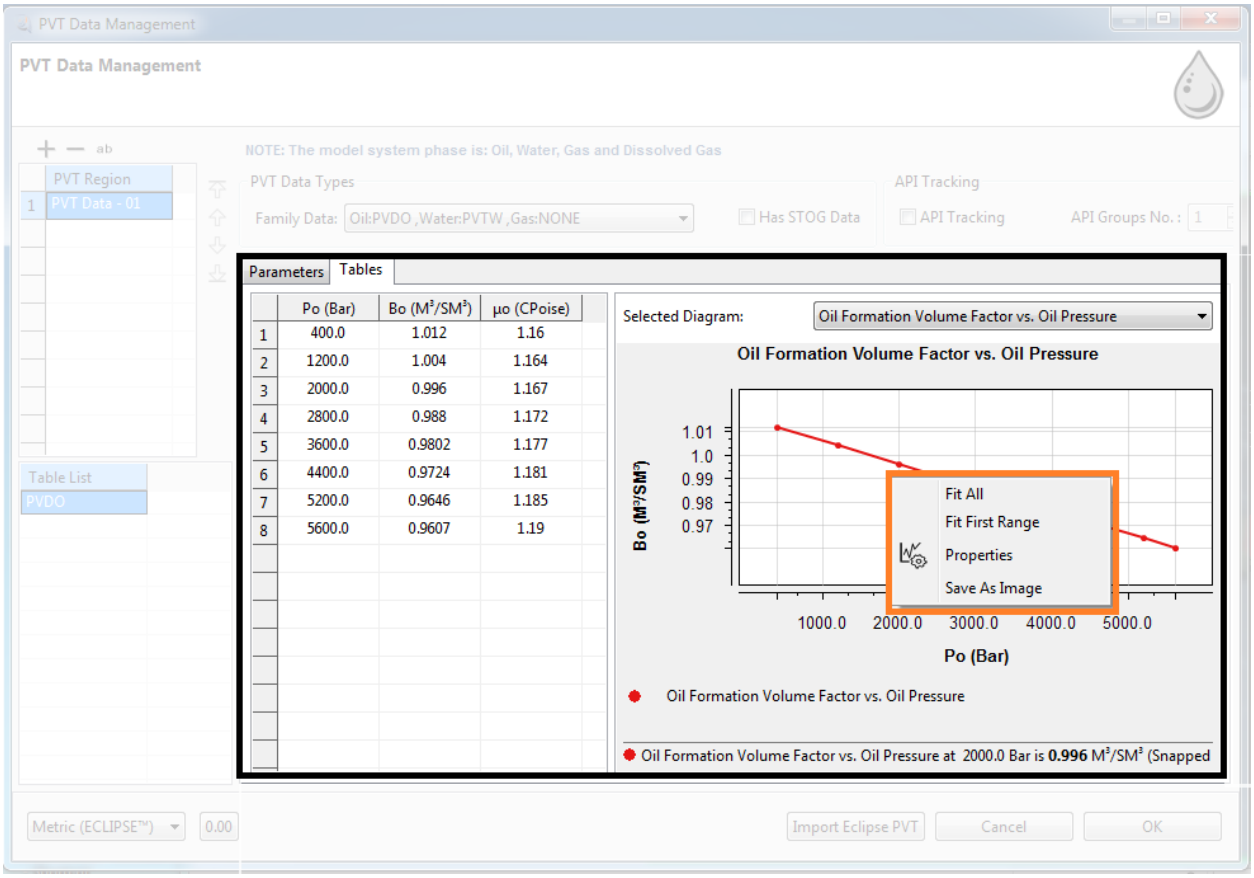


Figure 99 Plots Setting

3-2-2- Rock Region Data

The rock region data can only define in this section. Clicking “Rock Region Data” button, the window of Figure 100 is opened. The naming, sorting and add or remove actions procedure is same as section 3-2-1. The Rock Region dialog includes Rock Parameters and Rock table. Using Rock Option tab one of the PVTNUM, SATNUM or ROCKNUM have to be selected.

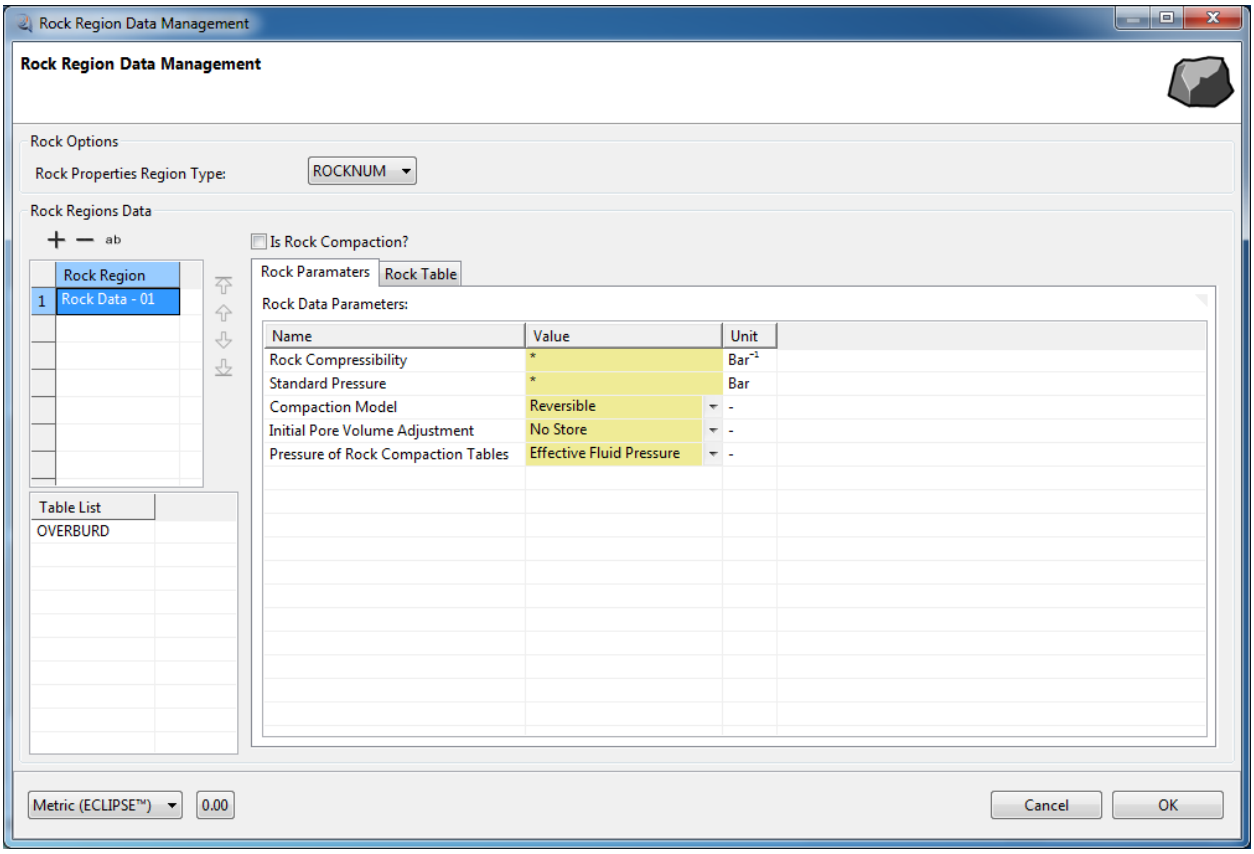


Figure 100- Rock Region Data Management

If the "Is Rock compaction?" does not checked only the OVERBURD table will be available in the table list and following data in the "Rock Parameters" tab: Rock compressibility, standard pressure, compaction model, initial pore volume adjustment, and pressure of rock compaction tables. There are two methods of reversible and irreversible for defining compaction model. The "Store" and "No Store" options are available for the initial pore volume adjustment. The pressure of rock compaction table either can be effective fluid pressure or effective rock stress. When the "Is Rock compaction?" is checked, the rock compressibility and standard pressure fields are removed and two new tables of ROCKTAB and ROCTSIG are added to the table list.

Note: the ROCKTSIG are only available for dual (fractured) models

Rock Properties Region Type:

If SATNUM or ROCKNUM are not found in the defined REGIONs, default PVTNUM values will be used.

Compaction Model:

There are two models for rock compaction: "Reversible" in which the Rock compaction is fully reversible with increasing pressure, and "Irreversible", in which the Rock compaction is irreversible; the pore space does not re-inflate when the pressure increases

Initial Pore Volume Adjustment:

The "Store" option copies the initial equilibrated pressure at the start of the run into the overburden array. This has the effect of referencing the pore volume (as a function of pressure) to the initial pressure rather than the reference pressure, that is, the pressure with a pore volume multiplier of 1.0 in the ROCKTAB table. Thus the input pore volume for a cell is defined as the pore volume at initial conditions rather than the pore volume at the reference pressure.

Note: the OVERBURD table should not be used with the "Store" option as the input OVERBURD data will be overwritten and ignored.

Pressure of Rock Compaction Tables:

If the overburden pressure supplied in the OVERBURD table is greater than the fluid pressure (which it often will be), the effective fluid pressure (Peff fl= Pfluid- Overburden) will be negative. In this case the rock compaction tables will have to be entered with negative pressure values. The tables can be input in a more intuitive manner by selecting the "Effective Rock Stress" option before the tables are entered. In this case the pore volume and transmissibility multipliers will be tabulated against the effective overburden pressure, Peff ov = Overburden - Pfluid, which will then be positive. This item should be left defaulted if the OVERBURD table is not used. The default value of the "pressure of rock compaction tables" is effective fluid pressure.

The three tables of ROCKTAB, ROCKTSIG, and OVERBURD are explained in this section.

1- *ROCKTAB (Rock compaction data) Table:*

   Each table consists of five columns of data. Apart from the last or first rows, values in the second and third columns may be defaulted, and such values are linearly interpolated. All values in the fourth and fifth columns can be defaulted, and if this is done the values from the third column are used.

   The first column includes the pressure values which should increase monotonically down the column, unless the "Effective Rock Stress" option has been used in the

"Pressure of Rock Compaction Tables", in such case the values should decrease monotonically down the column. The second column is the corresponding pore volume multiplier; these values should be level or increasing down the column. Third column contains the corresponding transmissibility multiplier in x direction, which its values should be level or increasing down the column. The y and z direction transmissibility multipliers are placed in column 4 and 5, respectively. There must be the same number of entries in each column of a given table. Figure 101 displays the ROCKTAB table and corresponding plot.

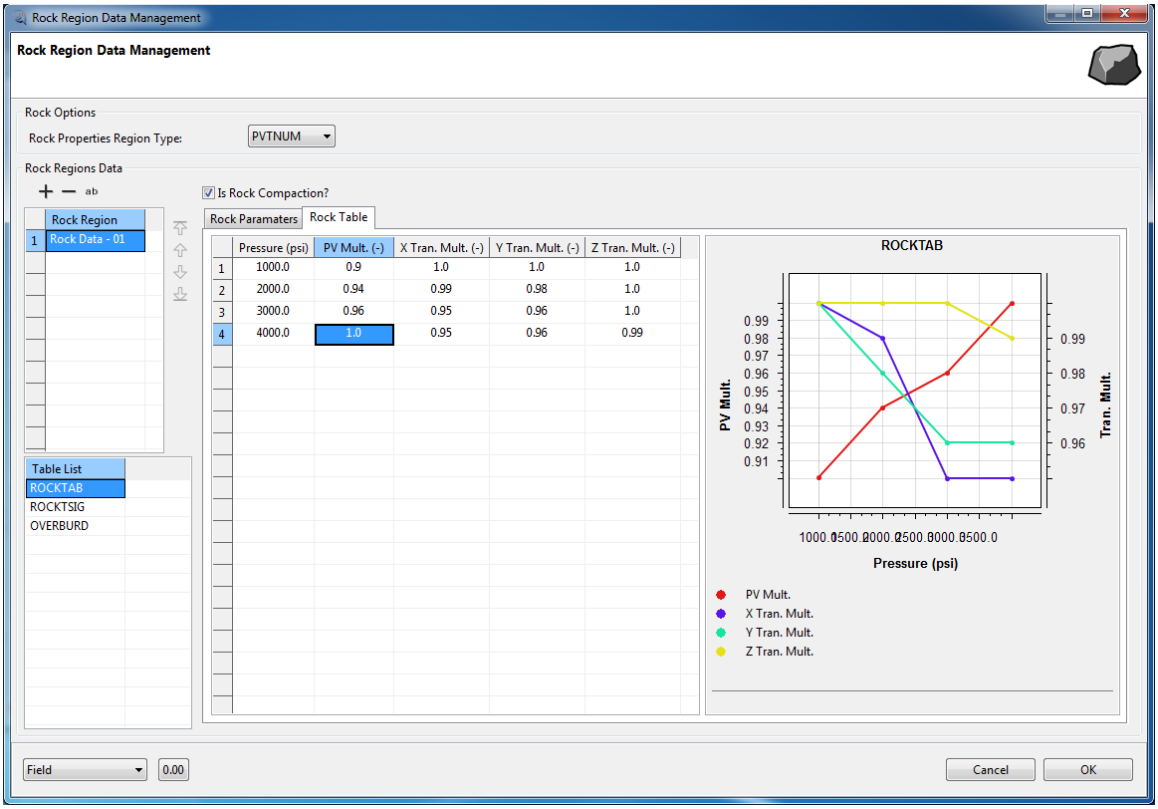


Figure 101 Rock Compaction Data Table and Plot

2- *OVERBURD (Rock overburden pressure) Table:*

This table consists two columns of the depth and the corresponding overburden pressure. The depth values should decrease monotonically down the column by default, since the positive z direction in RETINA is upward. The over burden values should increase monotonically down the column.

There must be the same number of entries in each column of a given table. The OVERBURD table can be used if the Rock Compaction option is selected. The overburden pressure at the grid block depth is subtracted from the pore pressure

to look up the pore volume and transmissibility multiplier values. If the keyword is not present then the overburden pressure is taken to be zero. if the OVERBURD is specified, the ROCKTAB table tabulate the compaction multipliers against the effective fluid pressure instead. If the overburden pressure is greater than the fluid pressure, as it usually will be, then the effective fluid pressure (or stress) will be negative. This leads to non-intuitive compaction tables with negative effective pressures in the first column. Figure 102 displays the typical rock overburden pressure table and plot.

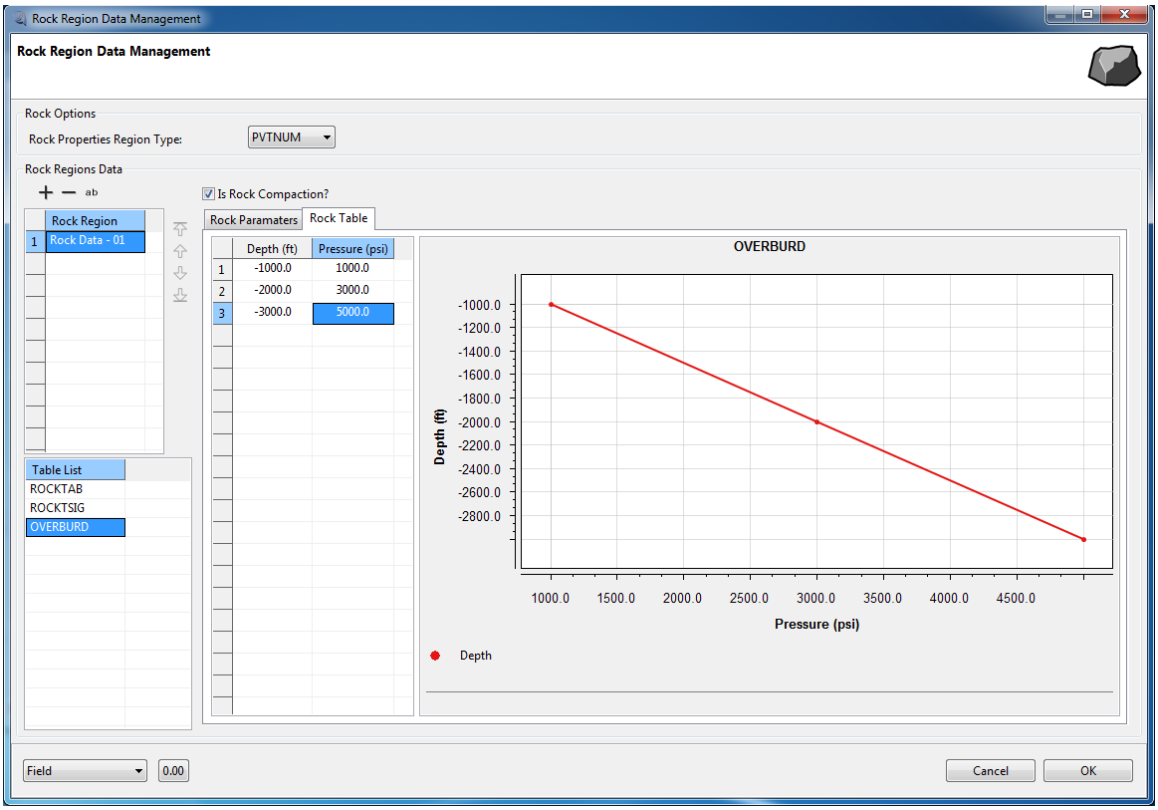


Figure 102 Rock Overburden Pressure Table and Plot

3- *ROCKTSIG (Rock compaction data tables in dual porosity runs) Table:*

This table consists two columns of the pressure and the sigma multiplier, and only is available in dual models. The pressure values should increase monotonically down the column, unless the "rock effective stress" option has been used, in which case the values should decrease monotonically down the column. The sigma values should be level or increasing down the column. Each row must be complete, containing an entry in every column of the table. The Figure 103 displays the rock ROCKTSIG table and corresponding plot.

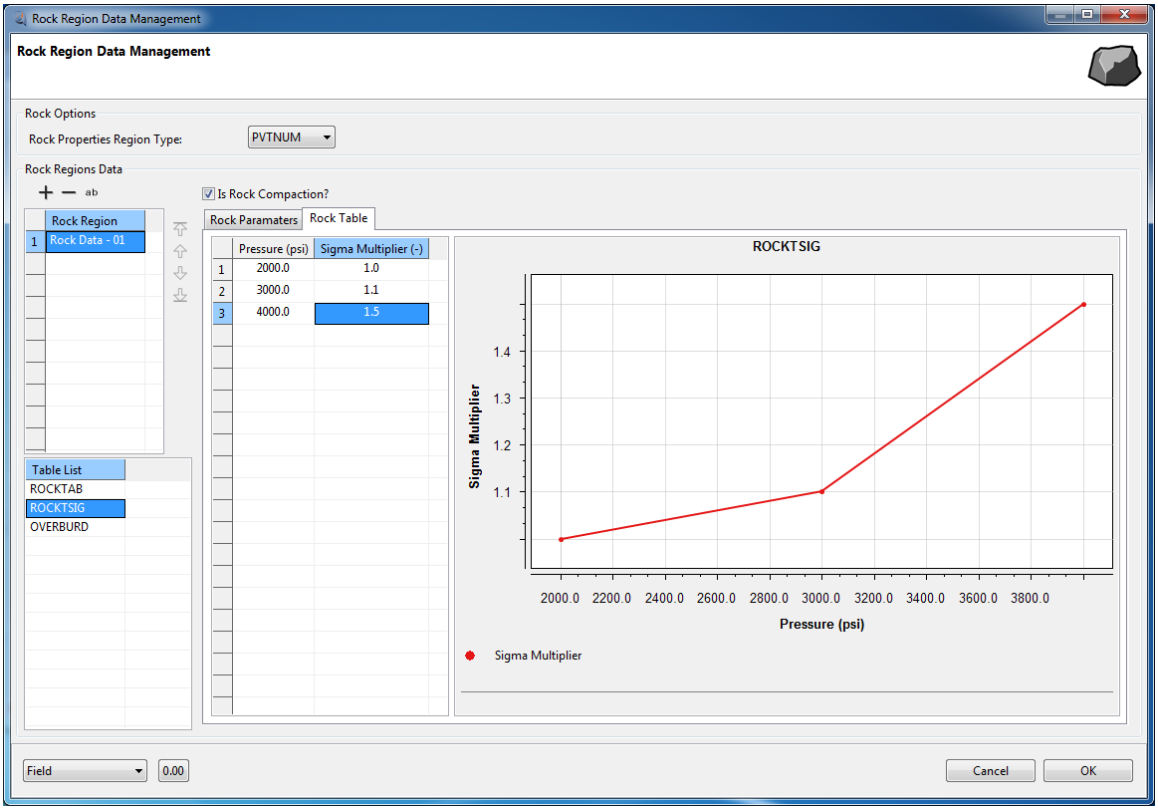


Figure 103 Rock Compaction Data in Dual Porosity Table and Plot

Rock Plot Setting:

User can use right click on the plot to see setting. Setting includes save, fit data, plot property. The zoom action is performed using mouse wheel. Clicking on “Properties” leads user to the “Chart Properties” window. In this window, user can edit title, description and related fonts, axis setting like axis minimum and maximum values, scales, marker types, colors and etc.

3-2-3- Saturation Region Data

In these section types of Saturation data for oil, water and gas are selected. There are six families of the saturation data in this section, which user can select each of the based on the model system phase (Figure 104). For defining saturation region or editing the existed one, the “Saturation Region Data” button must be clicked. This dialog includes “Saturation Region” and “Hysteresis” tabs. There are two tables of “saturation regions” and “table list” in the “Saturation Regions” tab. The upper one is for defining of the

desired regions and another displays saturation tables. All add, remove, edit and naming procedure are same as section 3-2-1.

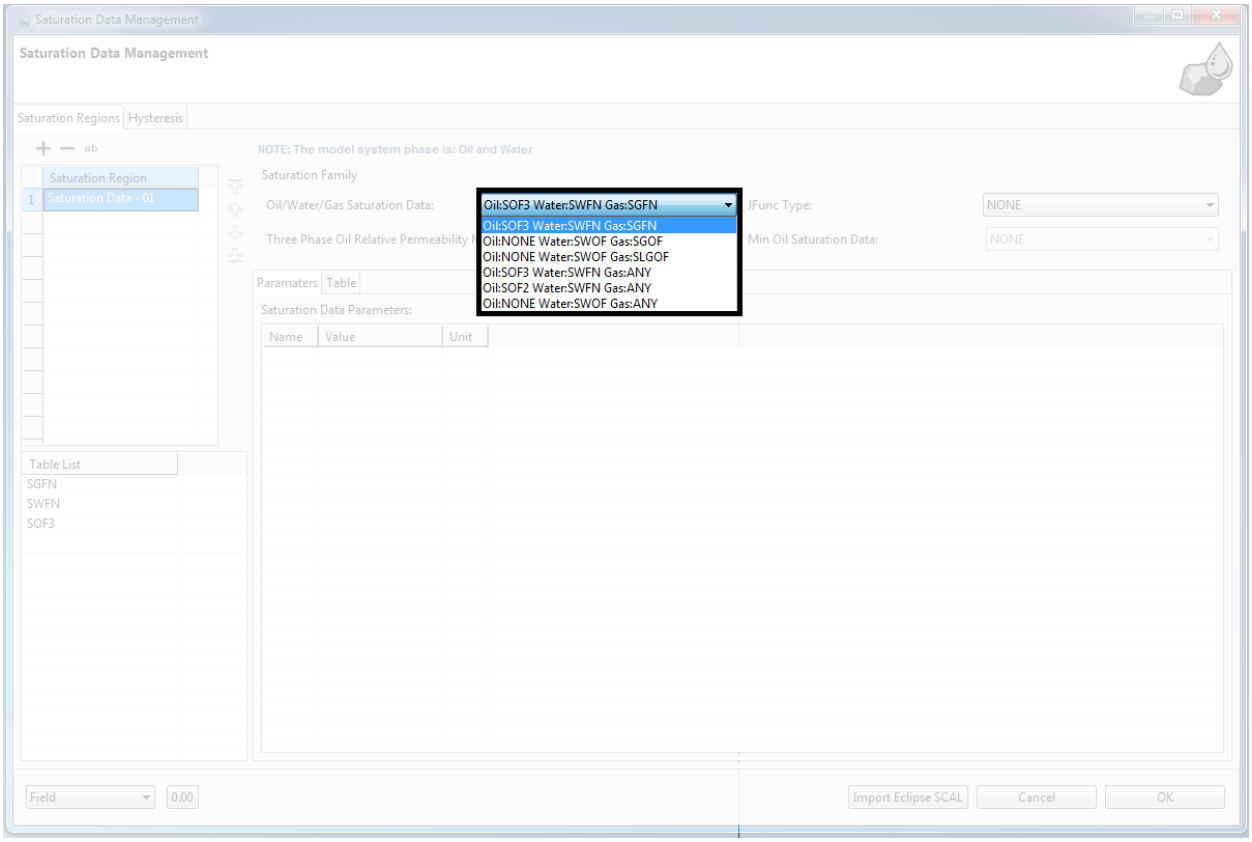


Figure 104- Saturation Data Management

Family 1: Oil: SOF3, Water: SWFN, Gas: SGFN

SOF3 Table:

There are three column for each SOF3 (Oil saturation functions (three-phase)) table. The first column includes the oil saturation. Its values should be between 0 and 1 and should increase monotonically down the column. The maximum oil saturation should be equal to 1-Swco, where Swco is the connate water saturation. The second column includes corresponding oil relative permeability for regions where only oil and water are present. Its values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be 0. The third column consists the corresponding oil relative permeability for regions where only oil, gas and connate water are present. Values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be 0.

Default values (represented by 1*, or empty) may be inserted as required in columns 2 and 3. When the table is read in, defaults are replaced by values computed by linear interpolation. Each table must contain at least two rows. Each row must be complete, containing an entry in every column of the table. The maximum values in columns 2 and 3 both represent the oil relative permeability at maximum oil saturation and should therefore have the same value. Figure 105 displays the SOF3 table and plot.

Note: SOF3 should be used only in three phase cases.

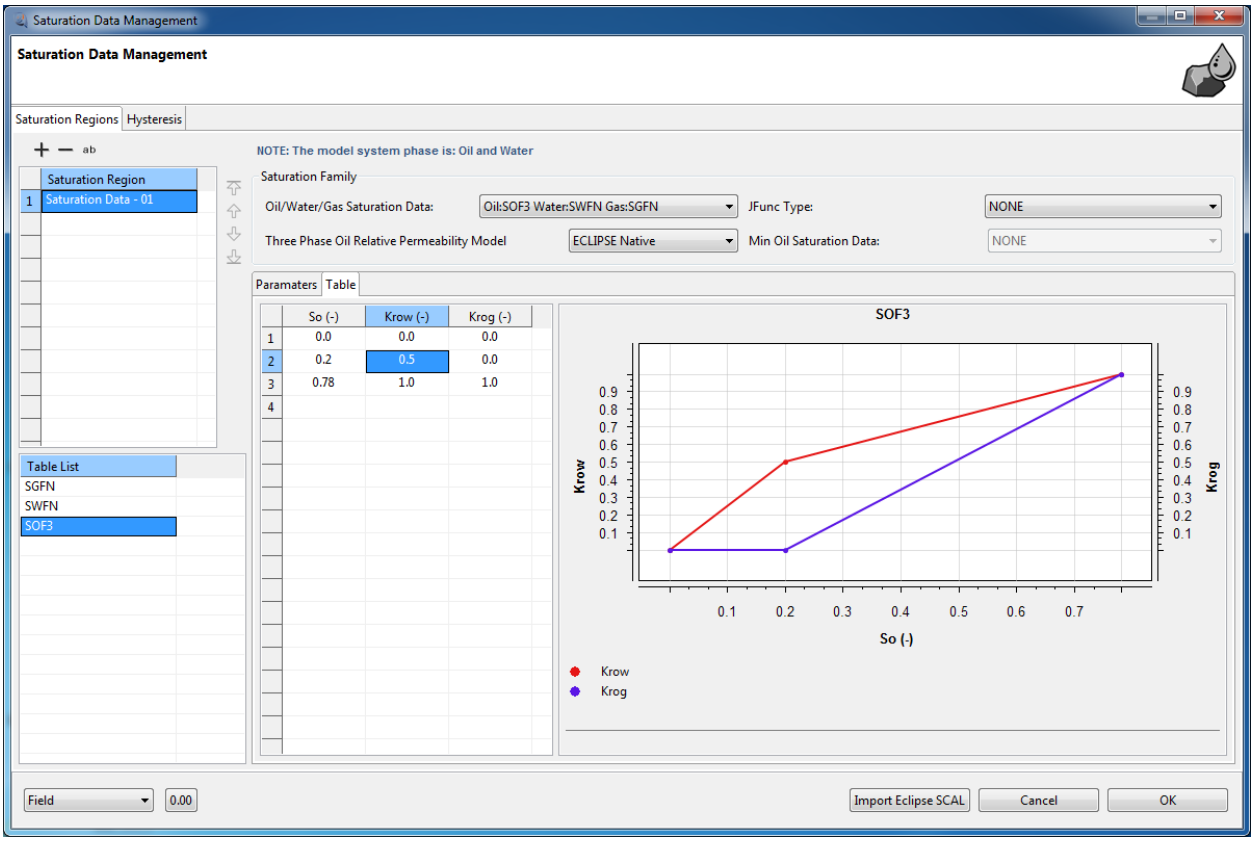


Figure 105 SOF3 Saturation Table and Plots

SWFN Table:

Each SWFN (Water saturation functions) table consists of three columns of data. The first column includes the water saturation. Its values should be between 0 and 1 and should increase monotonically down the column. The second column consists the corresponding water relative permeability. For this column the values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be 0. The third column includes the corresponding water-oil capillary pressure. Values should be level or decreasing down the column in this column of the data. If SWATINIT property

is used, then the water-oil capillary pressures must be monotonically decreasing. In gas-water systems (that is with no active oil phase), the water-gas capillary pressure should be entered here.

Default values (represented by 1*, or empty) may be inserted as required in columns 2 and 3. When the table is read in, defaults are replaced by values computed by linear interpolation. Each table must contain at least two rows. Each row must be complete, containing an entry in every column of the table. The Figure 106 shows the SWFN table and plots.

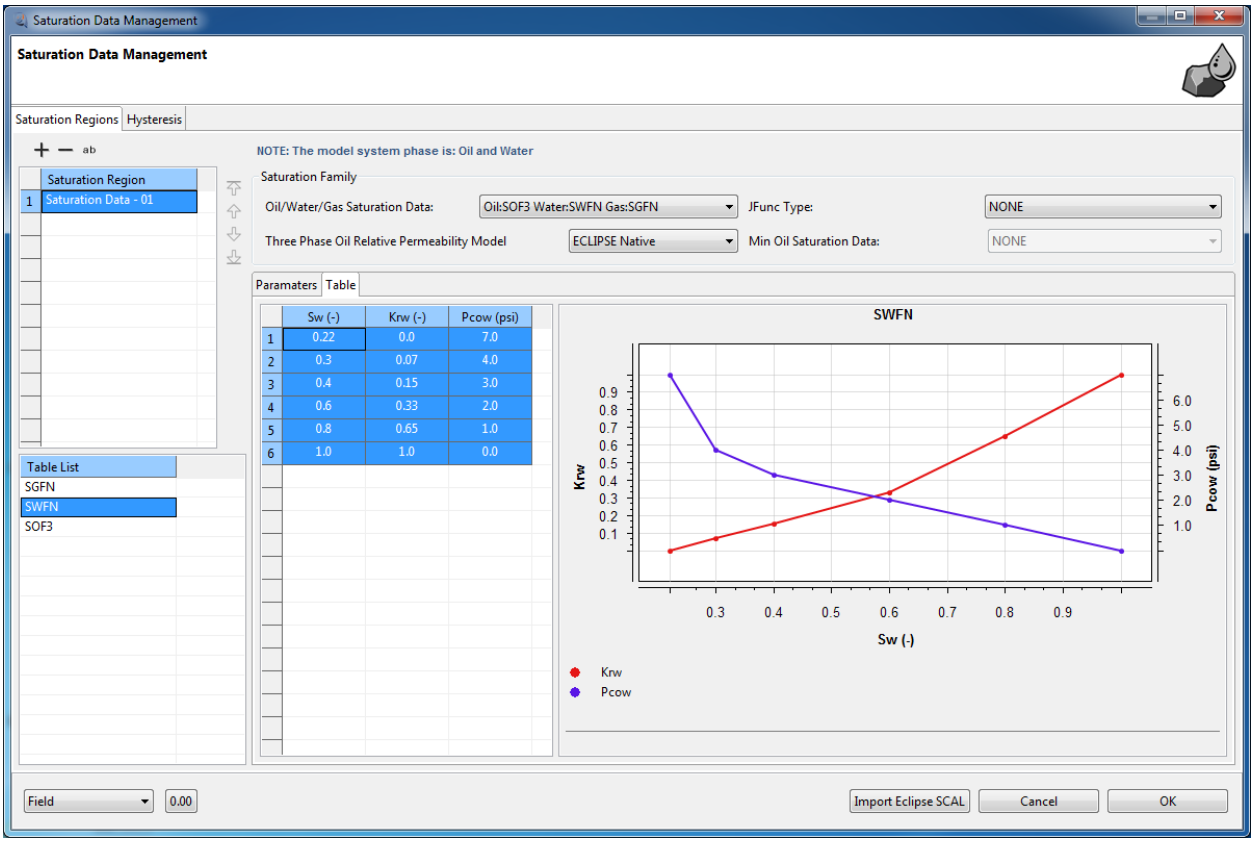


Figure 106 SWFN Saturation table and Plots

SGFN Table:

Each table of SGFN (Gas saturation functions) consists of the three columns of data. The first column includes gas saturation. Values of this column should be between 0 and 1 and should increase monotonically down the column. The second column includes the corresponding gas relative permeability. Values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be 0. The last column includes corresponding oil-gas capillary pressure. Values should be level or

increasing down the column. In gas-water systems (that is no active oil phase), the gas-water capillary pressure is entered in the SWFN keyword. The oil-gas capillary pressure in SGFN should be set to zero.

Default values (represented by 1*, or empty) may be inserted as required in columns 2 and 3. When the table is read in, defaults are replaced by values computed by linear interpolation. Each table must contain at least two rows. Each row must be complete, containing an entry in every column of the table. Figure 107 displays the SGFN table and plots.

Note: In gas-water runs the gas and water saturation functions may be input together using the SGWFN keyword as an alternative to using both SGFN and SWFN.

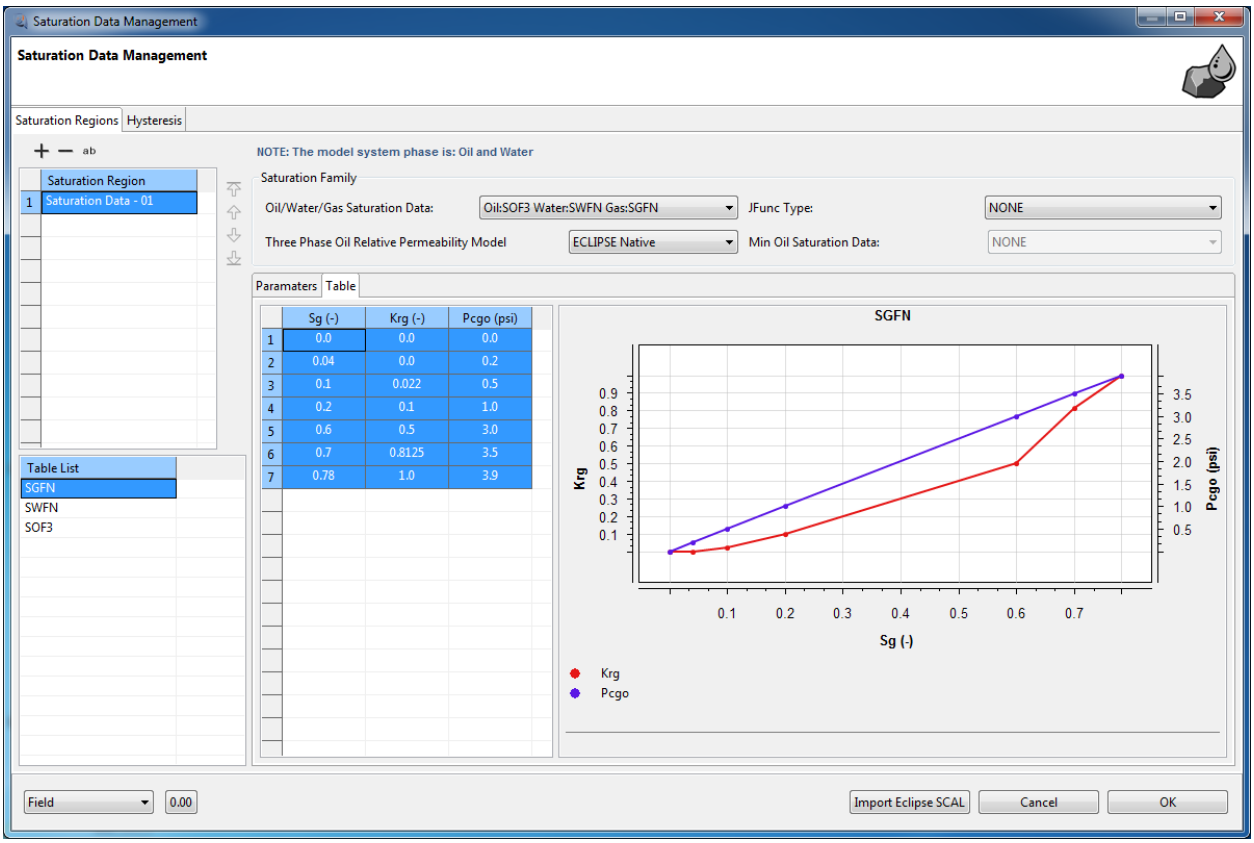


Figure 107 SGFN Saturation Table and Plots

Family 2: Oil: NONE, Water: SWOF, Gas: SGOF

SWOF Table:

The SWOF (Water / oil saturation functions versus water saturation) table may be used in runs containing both oil and water as active phases, to input tables of water relative

permeability, oil-in-water relative permeability and water-oil capillary pressure as functions of the water saturation. If gas is also an active phase in the run, the gas/oil saturation functions must be input with either keyword SGOF or SLGOF. Each SWOF table consists of four columns of data. The first column includes water saturation values. Values should be between 0 and 1 and should increase monotonically down the column. The first value in the column is interpreted as the connate water saturation. The last value in the column is interpreted as Sw = 1-Sor. The second column consists of the corresponding water relative permeability. Values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be 0 (zero). The third column includes the corresponding oil relative permeability when only oil and water are present. Values should be between 0 and 1 and should be level or decreasing down the column. If gas is also present in the run, the first value in the column (that is krow at So = Somax = 1-Swco) must be the same as krowg at Sg = 0 set in keyword SGOF or SLGOF. The last value in the column must be zero. Tha four and last column includes the corresponding water-oil capillary pressure. Values should be level or decreasing down the column.

Default values (represented by 1*, or empty) may be inserted as required in columns 2, 3 and 4. When the table is read in, defaults are replaced by values computed by linear interpolation. Each table must contain at least two rows. Each row must be complete, containing an entry in every column of the table. The Figure 108 shows the SWOF table and plots.

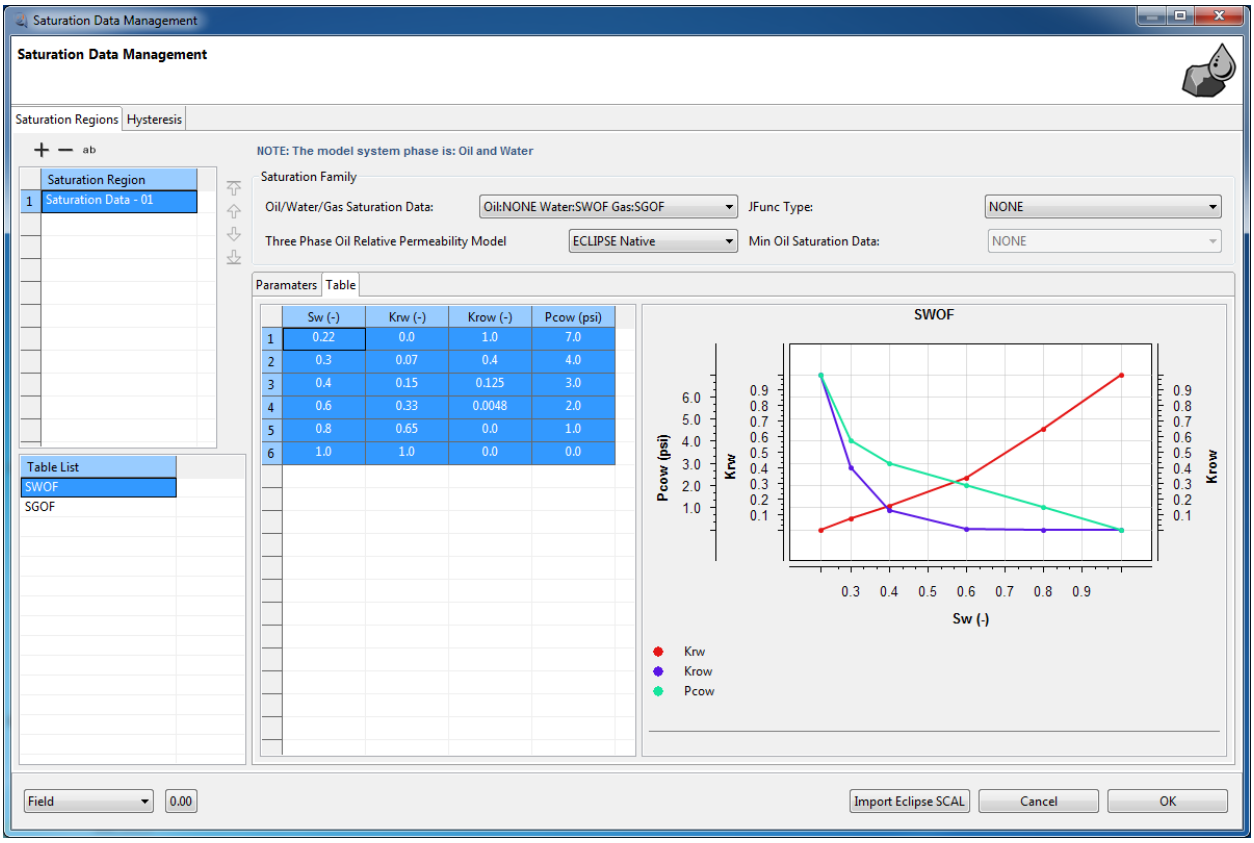


Figure 108 SWOF Saturation Table and Plots

SGOF Table:

The SGOF (Gas / oil saturation functions versus gas saturation) may be used in runs containing both oil and gas as active phases, to input tables of gas relative permeability, oil-in-gas relative permeability and oil-gas capillary pressure as functions of the gas saturation. If water is also an active phase in the run, the water/oil saturation functions must be input with SWOF. Each table consists of the four columns of data. The first column includes the gas saturation. Its values should be between 0 and 1 and should increase monotonically down the column. The first value in the column must be zero. The second column includes the corresponding gas relative permeability. The second column values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be zero. The third column consists the corresponding oil relative permeability when oil, gas and connate water are present. Its values should be between 0 and 1 and should be level or decreasing down the column. If water is present in the run, the first value in the column (krog at Sg = 0) must be the same as the

first value in the krow column in keyword SWOF (that is krow at So = 1-Swco). The last value in the column must be zero. The fourth column is the corresponding oil-gas capillary pressure. Values should be level or increasing down the column.

Default values (represented by 1*, or empty) may be inserted as required in columns 2, 3 and 4. When the table is read in, defaults are replaced by values computed by linear interpolation. Each table must contain at least two rows. Each row must be complete, containing an entry in every column of the table. The Figure 109 displays the SGOF table and plots.

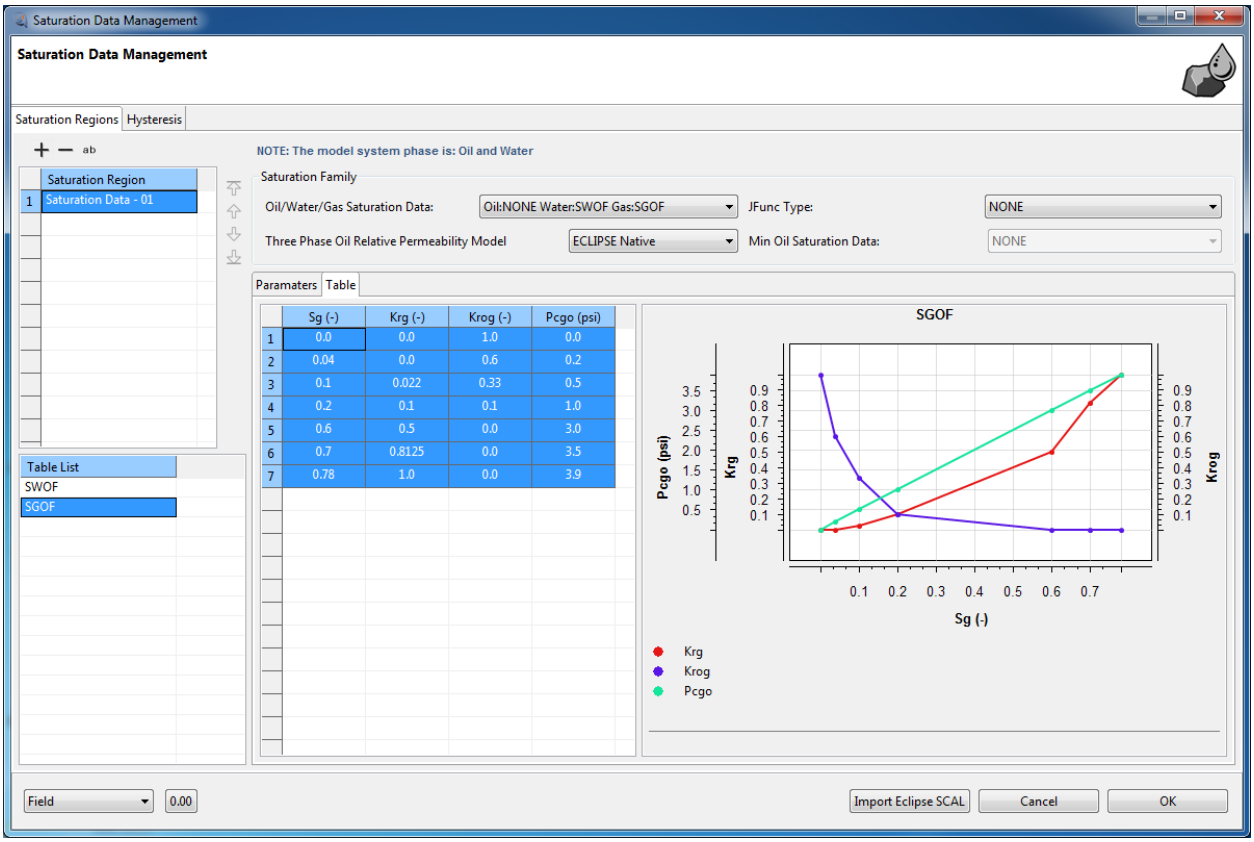


Figure 109 SGOF Saturation table and Plots

Family 3: Oil: NONE, Water: SWOF, Gas: SLGOF

SWOF Table:

The SWOF (Water / oil saturation functions versus water saturation) table may be used in runs containing both oil and water as active phases, to input tables of water relative permeability, oil-in-water relative permeability and water-oil capillary pressure as functions of the water saturation. If gas is also an active phase in the run, the gas/oil

saturation functions must be input with either keyword SGOF or SLGOF. Each SWOF table consists of four columns of data. The first column includes water saturation values. Values should be between 0 and 1 and should increase monotonically down the column. The first value in the column is interpreted as the connate water saturation. The last value in the column is interpreted as Sw = 1-Sor. The second column consists of the corresponding water relative permeability. Values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be 0 (zero). The third column includes the corresponding oil relative permeability when only oil and water are present. Values should be between 0 and 1 and should be level or decreasing down the column. If gas is also present in the run, the first value in the column (that is krow at So = Somax = 1-Swco) must be the same as krowg at Sg = 0 set in keyword SGOF or SLGOF. The last value in the column must be zero. Tha four and last column includes the corresponding water-oil capillary pressure. Values should be level or decreasing down the column.

Default values (represented by 1*, or empty) may be inserted as required in columns 2, 3 and 4. When the table is read in, defaults are replaced by values computed by linear interpolation. Each table must contain at least two rows. Each row must be complete, containing an entry in every column of the table.

SLGOF Table:

The SLGOF (Gas / oil saturation functions versus liquid saturation) may be used in runs containing both oil and gas as active phases, to input tables of gas relative permeability, oil-in-gas relative permeability and oil-gas capillary pressure as functions of the liquid saturation. If water is also an active phase in the run, the water/oil saturation functions must be input with SWOF. Each table consists of the four columns of data. The first column includes the liquid saturation. Values should be between 0 and 1 and should increase monotonically down the column. The first value in the column is interpreted as S1=Swco+Sor (Swco is the connate water saturation, the first saturation value in the SWOF table, or zero if there is no water in the run). The last value must be 1.0, which corresponds to a gas saturation of zero. The second column includes the corresponding gas relative permeability. Its values should be between 0 and 1 and should be level or decreasing down the column. The last value in the column must be zero. The values of the third column which includes the oil relative permeability when oil, gas and connate water are present should be between 0 and 1 and should be level or increasing down the

column. If water is present in the run, the last value in the column (krog at Sg=0) must be the same as the first value in the krow column in keyword SWOF (that is krow at So=1–Swco). The first value in the column must be zero. There are oil capillary pressure values in the fourth column of the SLGOF table. Its values should be level or decreasing down the column.

Default values (represented by 1*, or empty) may be inserted as required in columns 2, 3 and 4. When the table is read in, defaults are replaced by values computed by linear interpolation. Each table must contain at least two rows. Each row must be complete, containing an entry in every column of the table. The SLGOF table and plots are displayed in Figure 110.

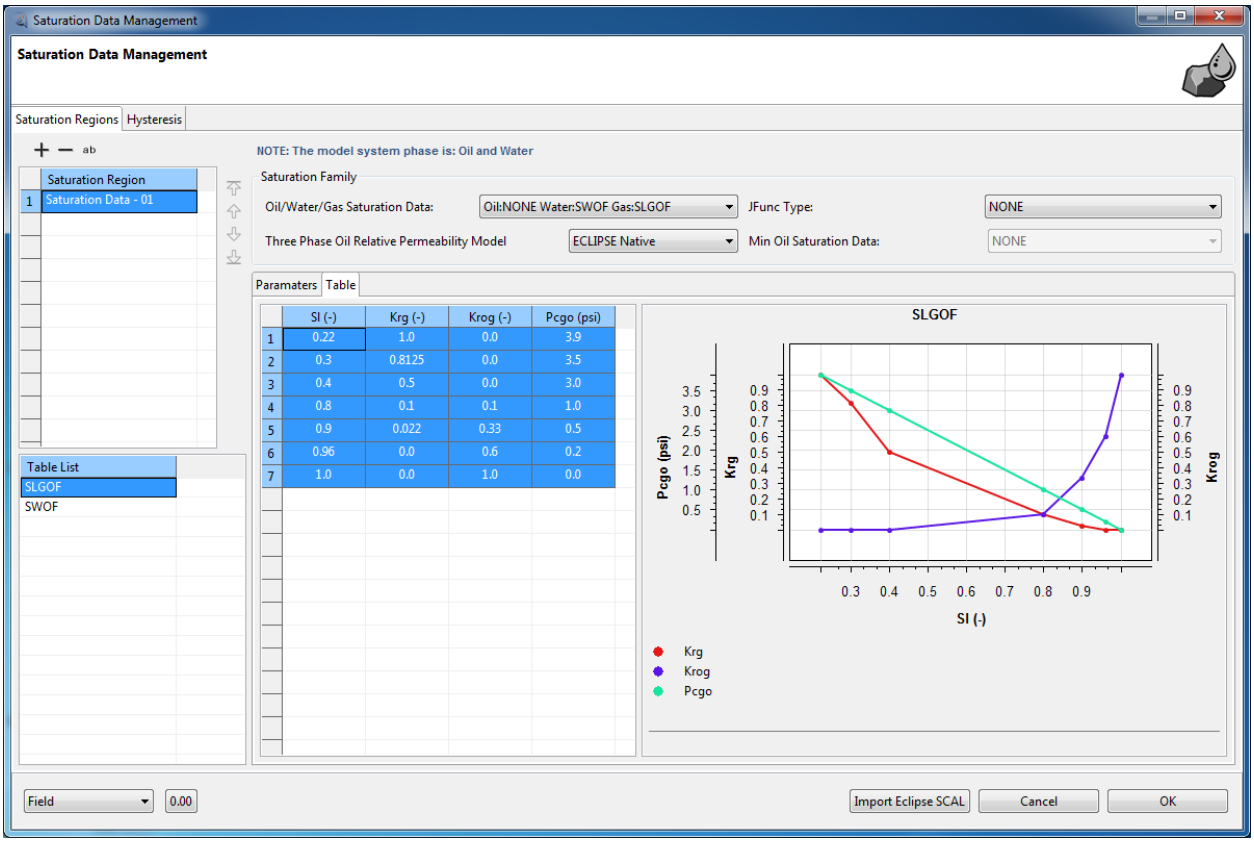


Figure 110 SLGOF Saturation Table and Plots

Family 4: Oil: SOF3, Water: SWFN, Gas: NONE

SOF3 Table:

There are three column for each SOF3 (Oil saturation functions (three-phase)) table. The first column includes the oil saturation. Its values should be between 0 and 1 and should

increase monotonically down the column. The maximum oil saturation should be equal to 1-Swco, where Swco is the connate water saturation. The second column includes corresponding oil relative permeability for regions where only oil and water are present. Its values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be 0. The third column consists the corresponding oil relative permeability for regions where only oil, gas and connate water are present. Values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be 0.

Default values (represented by 1*, or empty) may be inserted as required in columns 2 and 3. When the table is read in, defaults are replaced by values computed by linear interpolation. Each table must contain at least two rows. Each row must be complete, containing an entry in every column of the table. The maximum values in columns 2 and 3 both represent the oil relative permeability at maximum oil saturation and should therefore have the same value.

Note: SOF3 should be used only in three phase cases.

SWFN Table:

Each SWFN (Water saturation functions) table consists of three columns of data. The first column includes the water saturation. Its values should be between 0 and 1 and should increase monotonically down the column. The second column consists the corresponding water relative permeability. For this column the values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be 0. The third column includes the corresponding water-oil capillary pressure. Values should be level or decreasing down the column in this column of the data. If SWATINIT property is used, then the water-oil capillary pressures must be monotonically decreasing. In gas-water systems (that is with no active oil phase), the water-gas capillary pressure should be entered here.

Default values (represented by 1*, or empty) may be inserted as required in columns 2 and 3. When the table is read in, defaults are replaced by values computed by linear interpolation. Each table must contain at least two rows. Each row must be complete, containing an entry in every column of the table.

Family 5: Oil: SOF2, Water: SWFN, Gas: NONE

SOF2 Table:

The SOF2 (Oil saturation functions (two-phase)) table is used to specify the relative permeability of oil in two-phase models (oil-gas or oil-water). SOF2 should not be used to specify the oil's relative permeability in three-phase systems. Each SOF2 table consists of the two columns of data. The first column includes oil saturation data. Its values should be between 0 and 1 and should increase monotonically down the column. The second column includes the corresponding relative permeability kro. Its values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be 0.

Default values (represented by 1*, or empty) may be inserted as required in columns 2. When the table is read in, defaults are replaced by values computed by linear interpolation. Each table must contain at least two rows. Each row must be complete, containing an entry in every column of the table. The Figure 111 displays the SOF2 table and plot.

Note: SOF2 should be used only in two phase cases.

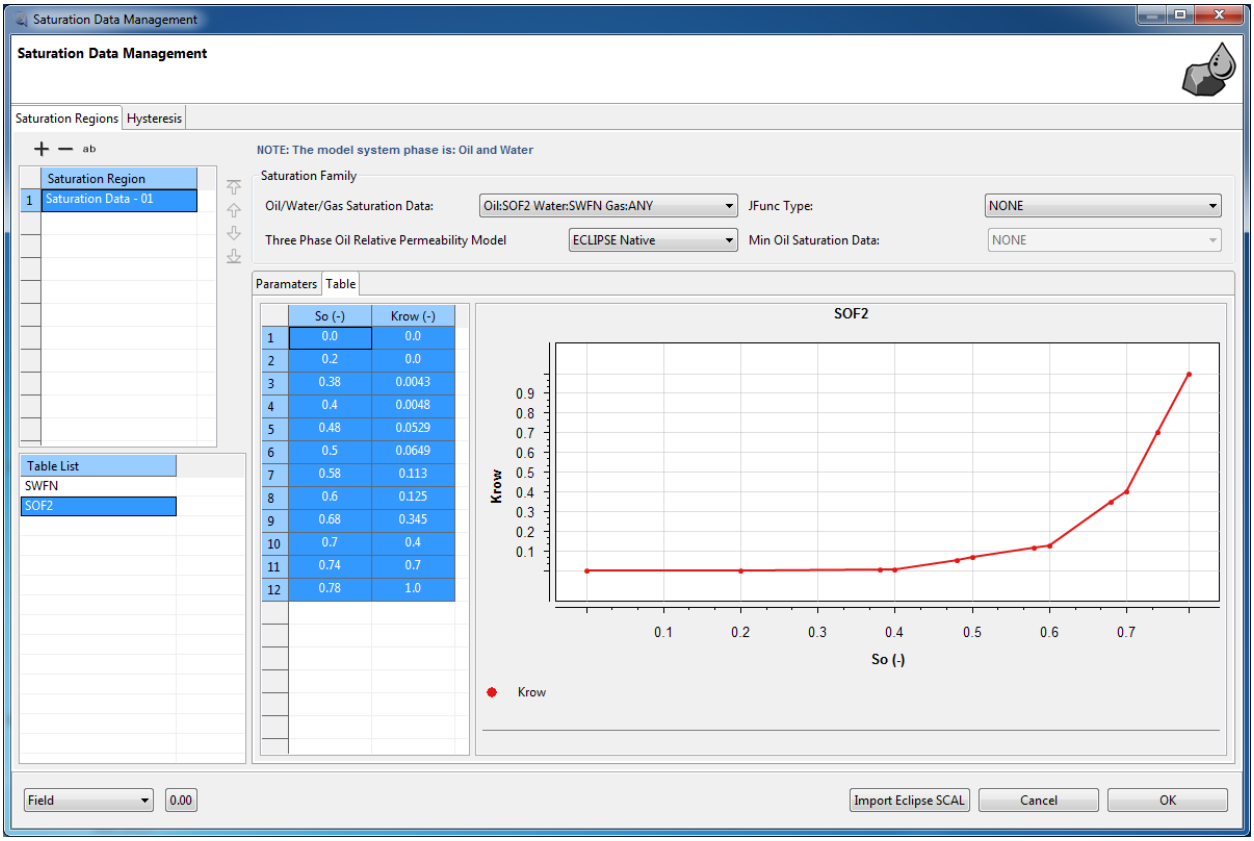


Figure 111 SOF2 Saturation Table and Plot

SWFN Table:

Each SWFN (Water saturation functions) table consists of three columns of data. The first column includes the water saturation. Its values should be between 0 and 1 and should increase monotonically down the column. The second column consists the corresponding water relative permeability. For this column the values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be 0. The third column includes the corresponding water-oil capillary pressure. Values should be level or decreasing down the column in this column of the data. If SWATINIT property is used, then the water-oil capillary pressures must be monotonically decreasing. In gas-water systems (that is with no active oil phase), the water-gas capillary pressure should be entered here.

Default values (represented by 1*, or empty) may be inserted as required in columns 2 and 3. When the table is read in, defaults are replaced by values computed by linear interpolation. Each table must contain at least two rows. Each row must be complete, containing an entry in every column of the table.

Family 6: Oil: NONE, Water: SWFN, Gas: NONE

SWFN Table:

Each SWFN (Water saturation functions) table consists of three columns of data. The first column includes the water saturation. Its values should be between 0 and 1 and should increase monotonically down the column. The second column consists the corresponding water relative permeability. For this column the values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be 0. The third column includes the corresponding water-oil capillary pressure. Values should be level or decreasing down the column in this column of the data. If SWATINIT property is used, then the water-oil capillary pressures must be monotonically decreasing. In gas-water systems (that is with no active oil phase), the water-gas capillary pressure should be entered here.

Default values (represented by 1*, or empty) may be inserted as required in columns 2 and 3. When the table is read in, defaults are replaced by values computed by linear interpolation. Each table must contain at least two rows. Each row must be complete, containing an entry in every column of the table.

Minimum Oil Saturation Data:

This field is activated when three phase oil relative permeability model set to the STONE-1. Three ways is available for defining this item:

1- NONE: needs no data.
2- SOMGAS: needs minimum oil saturation versus gas saturation table.
3- SOMWAT: needs minimum oil saturation versus water saturation table.

Moving mouse pointer on the plotted curves displays the related values and conditions (snapped or interpolated). User can use mouse right click on the plot to see setting which includes save, fit data, plot property. The zoom action is performed using mouse wheel. Clicking on "Properties" leads user to the "Chart Properties" window. In this window, User can edit title, description and related fonts, axis setting like axis minimum and maximum values, scales, marker types, colors and etc. If user wants to zoom in or out just one of the curves, he/she can click on the related axis and uses mouse wheel. There is an importer button which can import the Eclipse SCAL data to this window. User can also, use copy, paste, insert and delete some rows same as section 3-2-1.

J-FUNCTION:

This option can be used to activate the Leverett J-function option, which scales the water-oil and/or gasoil capillary pressure functions according to the grid block porosity and permeability. The option may only be used if end-point scaling is active. This option is for using J-function instead of capillary pressure in saturation functions for gas, water or both:

- Selecting "WATER" in JFunc combo box, requests the J-function option for the water-oil capillary pressure only.
- Selecting "GAS" in JFunc combo box, requests the J-function option for the gas-oil capillary pressure only.
- Selecting "BOTH" in JFunc combo box, requests the J-function option for both water-oil and gas-oil capillary pressure.

The following parameters (Figure 112) are added to the parameter tab in case of activating JFunc option:

- Oil-water or oil-gas surface tension,

  - The oil-water surface tension must be supplied if the first data item is set to either WATER or BOTH.
  - The oil-gas surface tension must be supplied if the first data item is set to either GAS or BOTH.
- JFunc power for the porosity term
- JFunc power for the permeability term
- JFunc Permeability Direction:
  - XY The average of PERMX and PERMY values
  - X PERMX value
  - Y PERMY value
  - Z PERMZ value

The J-function option scales the capillary pressure functions according to the rock porosity and permeability. The dimensionless J-function is entered in place of the capillary pressure function, specified using either the SWFN, SGFN or the SWOF, SGOF, SLGOF family of saturation data.

Note: If the SWATINIT property is used in conjunction with the JFunc option, then the J-function scaling is ignored in regions of the field where the SWATINIT array has been set, and will use PPCWMAX instead if set. If the PCW is used in conjunction with the JFunc, the input PCW array will be ignored when the J-function selects the water-oil capillary pressure. Similarly, if the PCG is input, the input PCG array is ignored when the J-function selects the gas-oil capillary pressure.

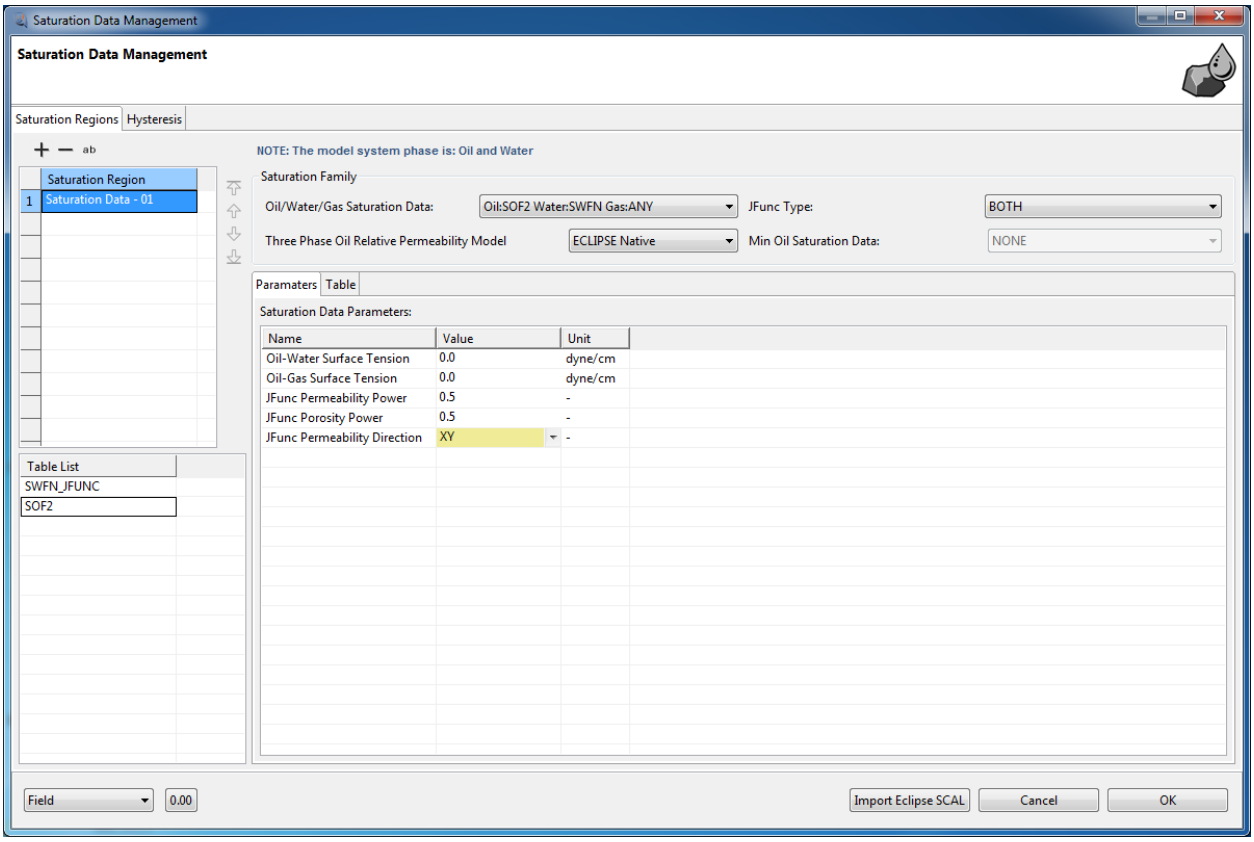


Figure 112 J-Function Corresponding Parameters Tab

There is a hysteresis tab in saturation data dialog which is shown in Figure 113. In this tab, user can define hysteresis type as capillary, relative permeability or both of them.

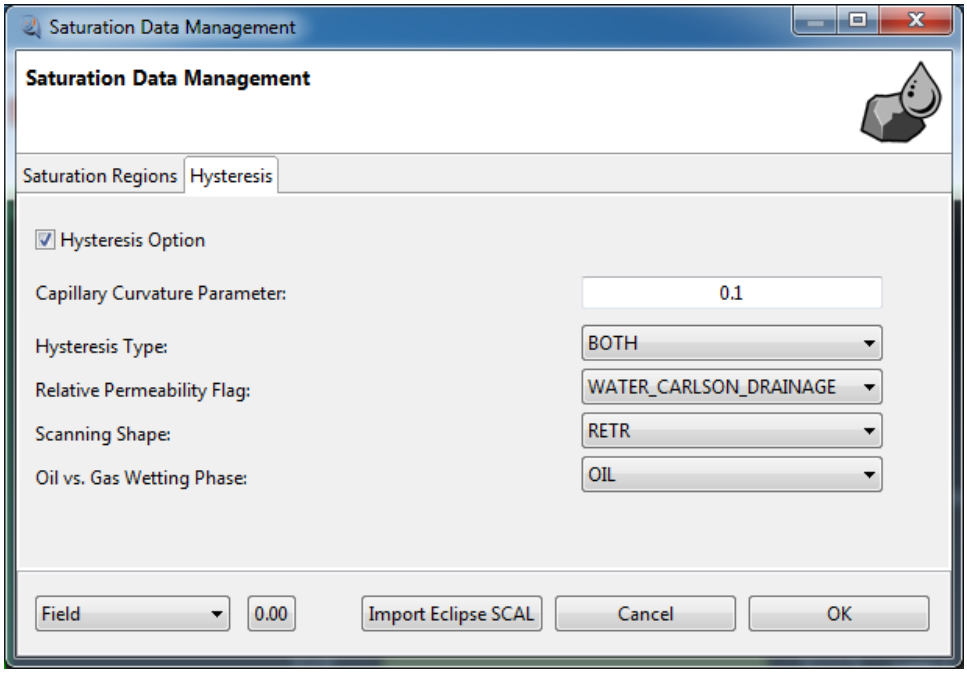


Figure 113- Hysteresis Tab

3-2-4- Equilibration Region Data

Clicking on the "Equilibration Region Data" button, user can define equilibration regions. The equilibration region window, which is shown in Figure 114, includes two tabs of "Equilibration Regions" and "Threshold Pressure".

"The "Equilibration Regions" tab includes two tabs of "Parameters" and "Tables". If there is active API Tracking option, a tab of API table is also added to this window.

Following parameters should be entered in equilibration window:

- Pb or Rs definition Method
- Pd or Rv Definition Method
- Reference Depth (since the RETINA default positive Z-direction is upward, all the depths are negative)
- Reference Pressure
- Water-Oil Contact Depth
- Gas-Oil contact Depth
- Oil-Water Capillary Pressure
- Gas-Oil Capillary Pressure
- Table Division Number
- Accuracy
- Performing Quiescence Correction

Bubble pressure (Pb) or solution gas ratio (Rs) should be defined versus depth (h) in related "Table" tab. In case of vaporized oil, the dew pressure (Pd) or vapor oil-gas ratio (Rv) should be defined also. All of the plot and table settings which are mentioned previously, are available also in this window.

There are four options for defining bubble pressure (Pb) or solution gas ratio (Rs):

1- Rs versus depth
2- Pb versus depth
3- Datum Depth = gas oil contact (GOC)
4- None (Dead oil)

There are four options for defining dew pressure (Pb) or vapor oil-gas ratio (Rv):

1- Rv versus depth
2- Pd versus depth
3- Datum Depth = gas oil contact (GOC)
4- None

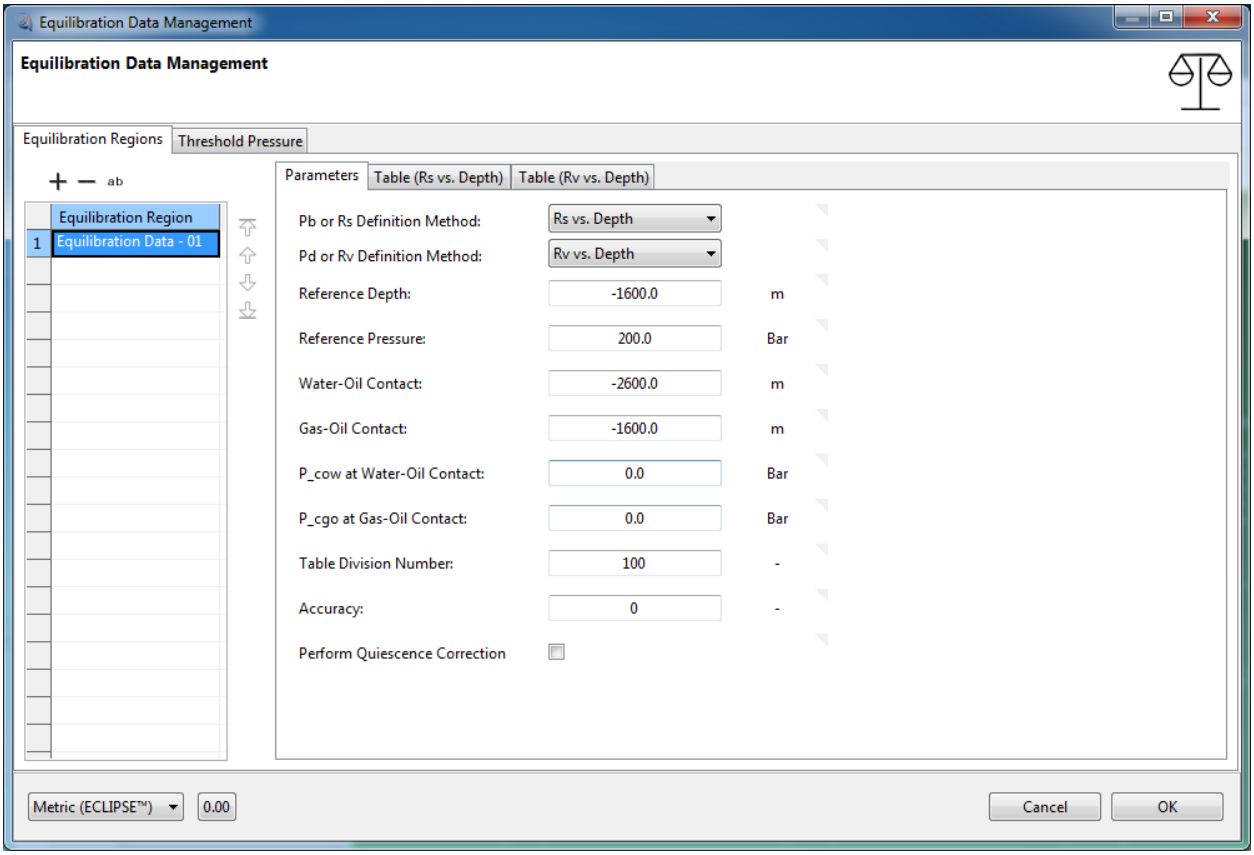


Figure 114- Equilibration Data Management

The water-oil contact depth, reference depth, gas-oil contact depth and Rs (Pb) tables are displayed in table tab (Figure 115). Similar tables are available for API versus depth if API tracking is activated, and for vaporized oil models.

"Threshold Pressure" tab which has been displayed in Figure 118 is used to define threshold pressure between two equilibration region connections.

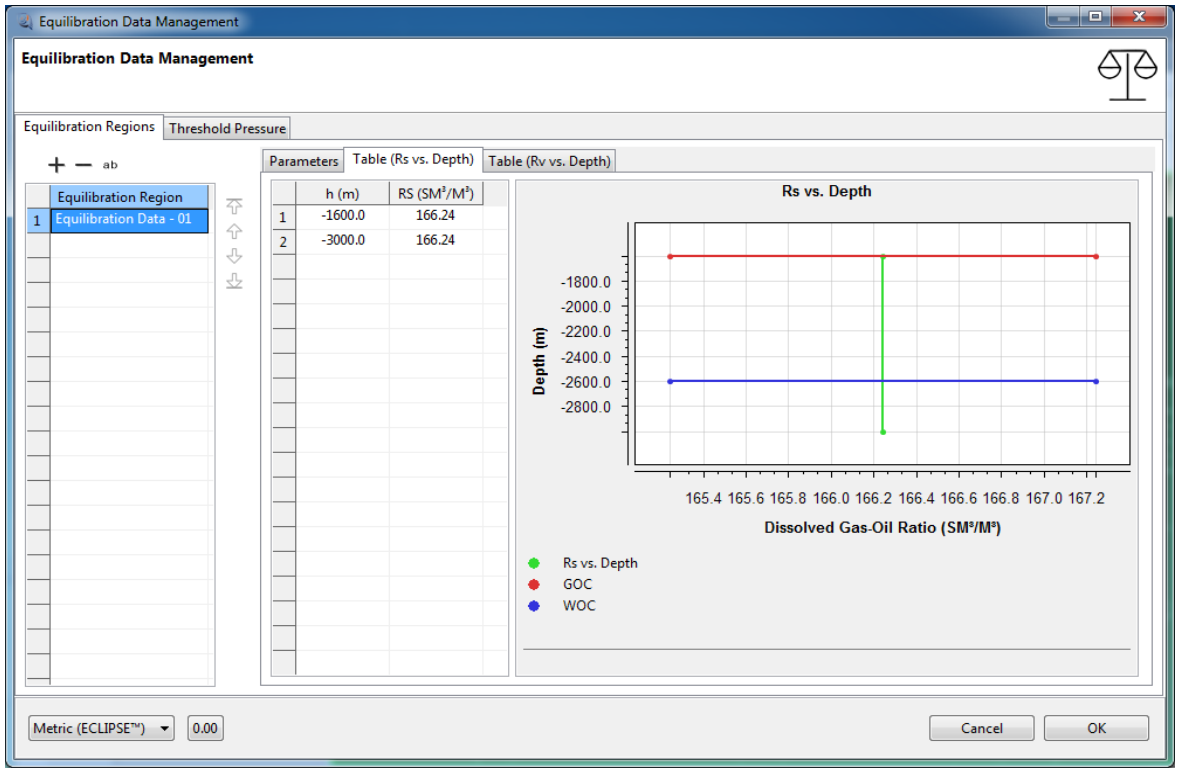


Figure 115- Typical Rs vs. Depth

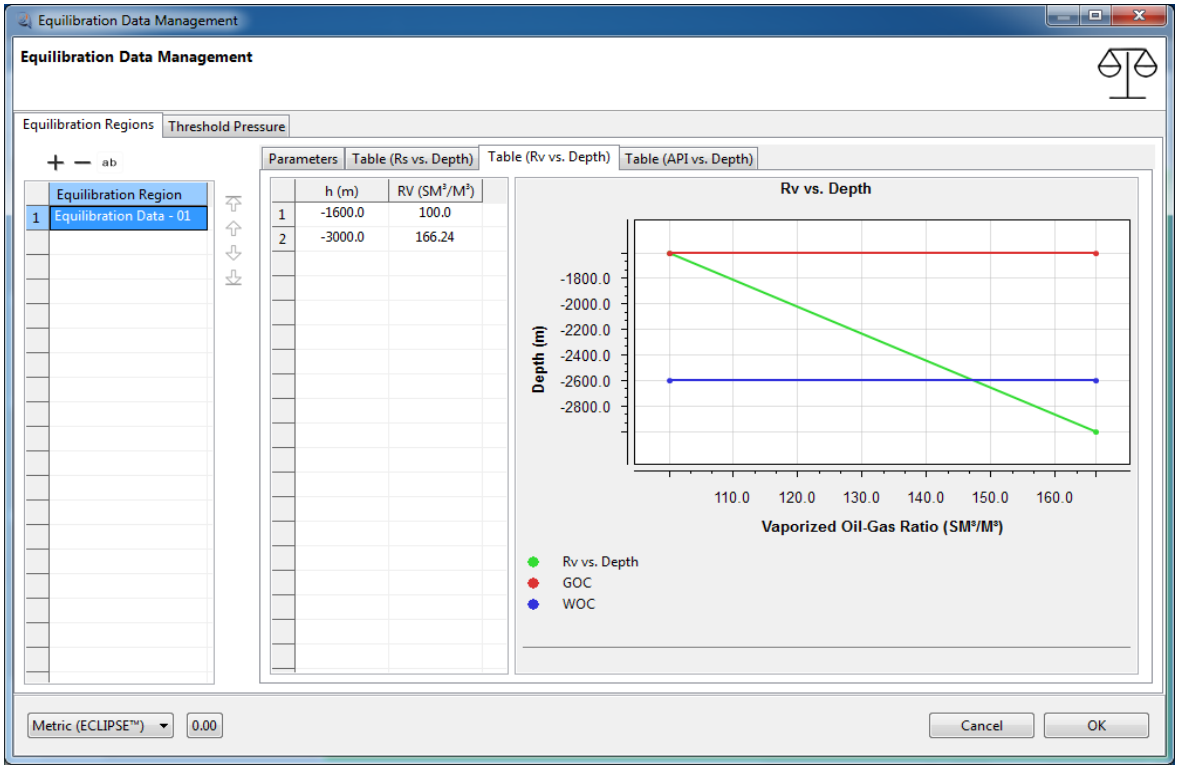


Figure 116 Typical Rv Vs. Depth

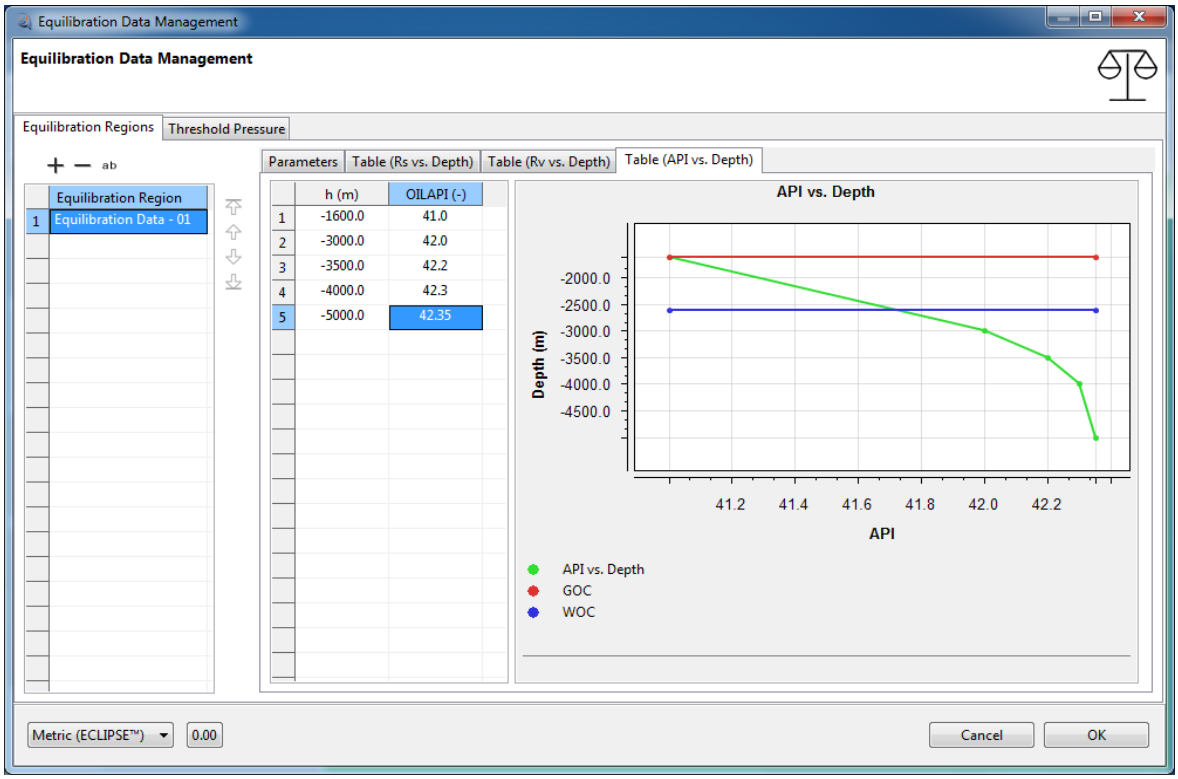


Figure 117- Typical API vs. Depth

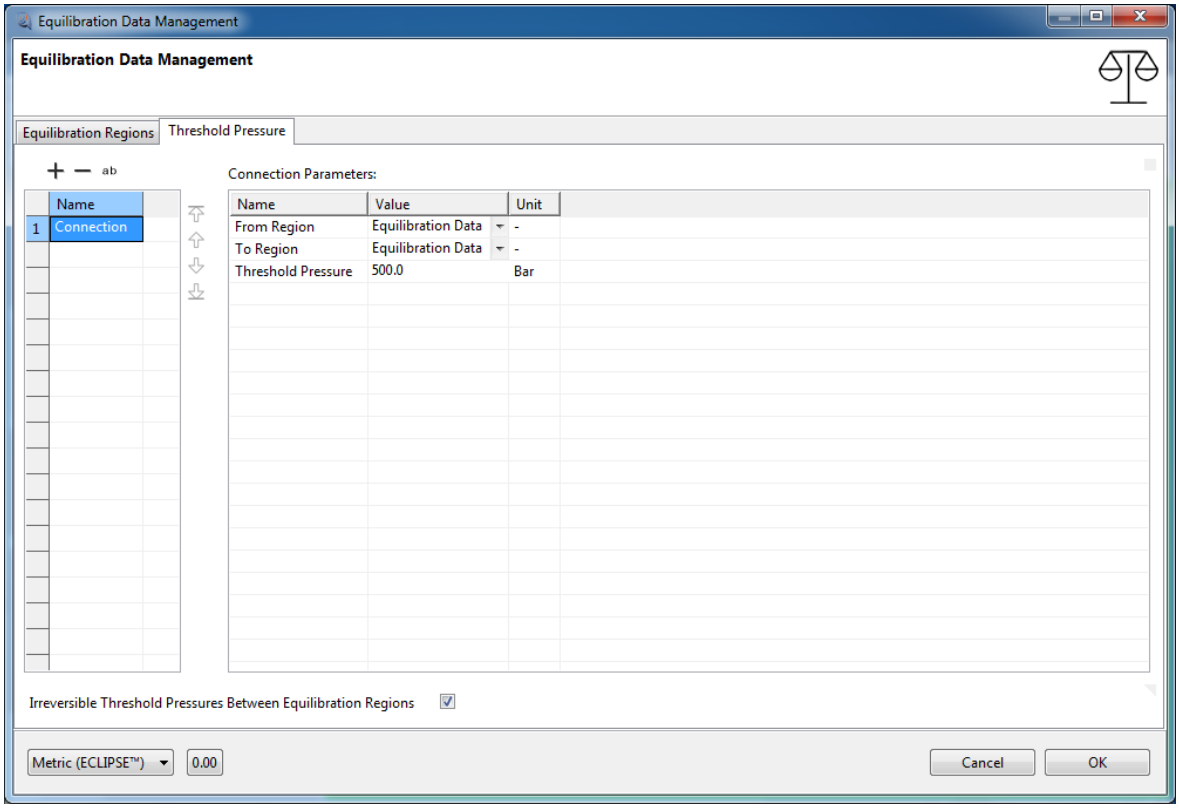


Figure 118- Typical Threshold Pressure Tab

3-2-5- End Point Scaling Region Data

The button of “End point Scaling Region Data” will be used to define end point scaling. At first scaling type must be selected using “End Point Scaling Type: dropdown list (Figure 119). The EPS type can be either “Two points” or “Three points”. If you select “None” a message of “End point scaling is disabled” is displayed and no scaling is performed. If you

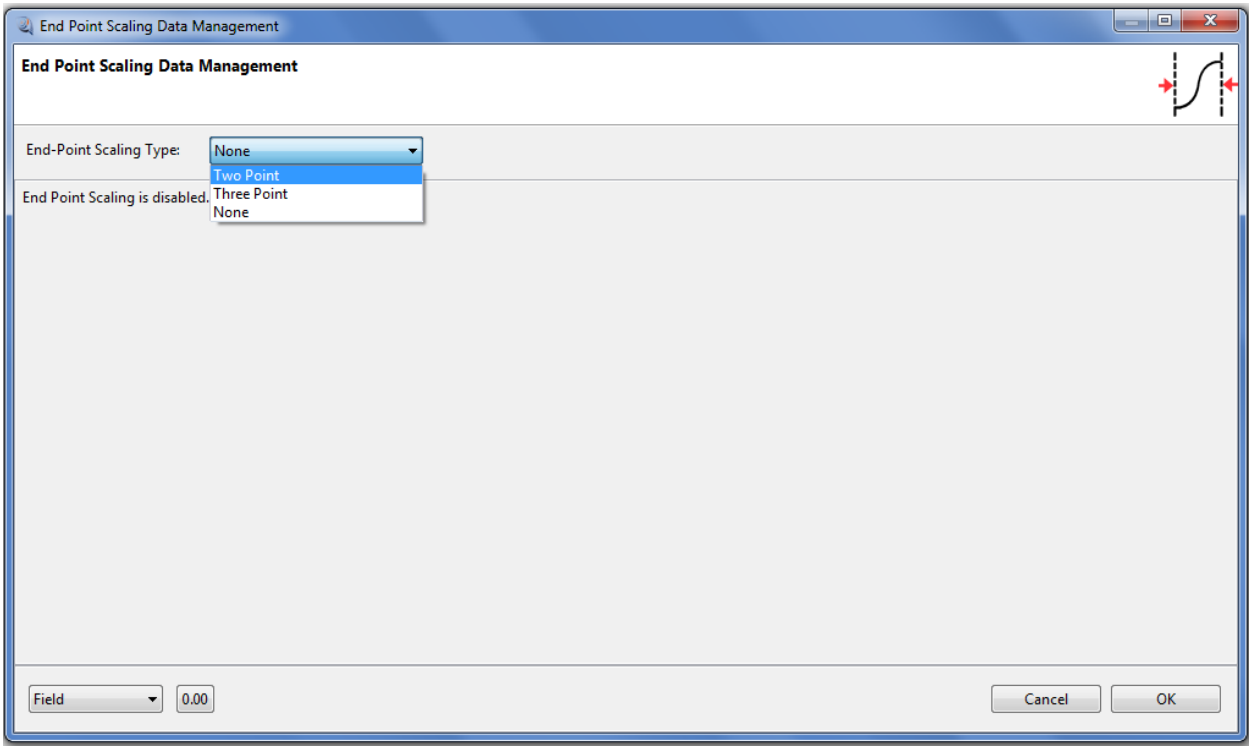


Figure 119- Define an End Point Scaling Type

If user chooses one of the two or three points scaling the Figure of 120 is opened. This window includes tables, parameters and EPS data types which are describe here.

For adding and removing some EPS regions, the + and - are used, respectively. The procedure is same as section 3-2-1. User can move defined EPS regions using navigating arrows. There is a button of ab in the EPS data management window, which is used to auto naming of the defined regions.

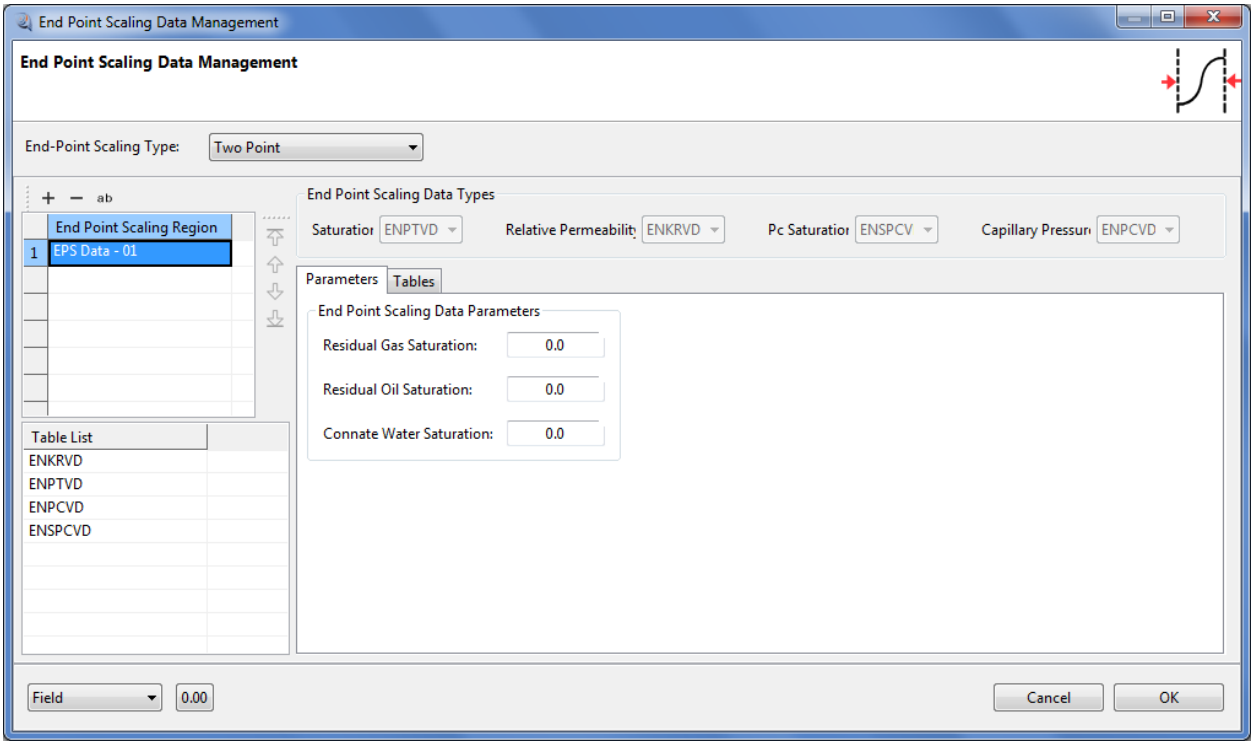


Figure 120- End Point Scaling Data Management

End Point Scaling Data Types

The types of the scaling such as saturation EPS type, relative permeability EPS type, capillary pressure EPS type Pc-saturation EPS type are presents in this section. Note that the EPS and Pc are abbreviations of End Point Scaling and Capillary Pressure, respectively.

ENPTVD (Saturation end-point versus depth tables):

This table specifies the depth variation of the saturation table end-points for different end-point scaling regions within the reservoir. The flow of each phase across each grid block face is calculated using transformed capillary pressure and relative permeability curves obtained by linearly scaling the tabulated curves between the end points specified using the ENPTVD table. The ENPTVD keyword also permits scaling of the relative permeability tables used in computing the flow of phases between grid cells and well connections and the scaling of capillary pressure functions in the equilibration algorithm.

Each table consists of 9 columns of data:

1- Depth values, the depth values should decrease monotonically down the column by default, since the positive z direction in RETINA is upward.
2- The corresponding values of the connate water saturation.
3- The corresponding values of the critical water saturation.
4- The corresponding values of the maximum water saturation.
5- The corresponding values of the connate gas saturation.
6- The corresponding values of the critical gas saturation.
7- The corresponding values of the maximum gas saturation.
8- The corresponding values of the critical oil-in-water saturation.
9- The corresponding values of the critical oil-in-gas saturation.

There must be the same number of entries in each column of a given table. Each saturation entry should be in the range 0.0 to 1.0 inclusive. If a saturation end-point is defaulted for all depth values, in a problem where a value could correctly be defined, then the end-point for each cell in the same scaling region will be the value in the appropriate saturation function table. However if a saturation end point is defaulted and the end-point has been defined at other depths, the end-point versus depth table will be filled in by linear interpolation within the table and constant extrapolation at the ends of the table. End point values entered for absent phases are ignored.

ENKRVD (Relative permeability end point versus depth tables):

Using this table you can specify the depth variation of the maximum relative permeability for the three phases. These tables complement the ENPTVD table, which specifies how the end point saturation values vary with depth.

Each ENKRVD table consists of eight columns of data:

1- Depth values, the depth values should decrease monotonically down the column by default, since the positive z direction in RETINA is upward.
2- The corresponding values of the maximum water relative permeability.
3- The corresponding values of the maximum gas relative permeability.
4- The corresponding values of the maximum oil relative permeability.
5- Water relative permeability at the critical oil (or gas) saturation.
6- Gas relative permeability at the critical oil (or water) saturation.
7- Oil relative permeability at the critical gas saturation.

8- Oil relative permeability at the critical water saturation

There must be the same number of entries in each column of a given table. If an end point is defaulted for all depth values, in a problem where a value could correctly be defined, then the end point for each cell in the same scaling region will be the value in the appropriate saturation function table. However if an end point is defaulted at a certain depth but has been defined at other depths, the end point vs. depth table will be 'filled in' by linear interpolation within the table and constant extrapolation at the ends of the table. End point values entered for absent phases are ignored.

ENSPCVD (Scaled saturations for Pc curves versus depth tables):

This table scales the connate water or gas end-point of the capillary pressure curves without scaling the corresponding oil relative permeability curves. The connate saturations are entered as functions of depth for different end-point scaling regions within the reservoir. The table can only be used when the end-point scaling option is active. Each table consists of 3 columns of data:

1- Depth values, the depth values should decrease monotonically down the column by default, since the positive z direction in RETINA is upward.
2- The corresponding values of connate gas saturation
3- The corresponding values of connate water saturation

There must be the same number of entries in each column of a given table. If a saturation end-point is defaulted for all depth values, in a problem where a value could correctly be defined, then the end-point for each cell in the same scaling region will be the value in the appropriate saturation function table. However, if a saturation end point is defaulted and the end-point has been defined at other depths, the end point versus depth table will be filled in by linear interpolation within the table and constant extrapolation at the ends of the table. End point values entered for absent phases are ignored.

ENPCVD (Maximum capillary pressure versus depth tables):

This table specifies the depth variation of the maximum capillary pressure for different end-point scaling regions within the reservoir. The table can only be used when the end-point scaling option is active. Each table consists of 3 columns of data:

1- Depth values, the depth values should decrease monotonically down the column by default, since the positive z direction in RETINA is upward.
2- The corresponding values of the maximum gas-oil capillary pressure
3- The corresponding values of the maximum water-oil capillary pressure

There must be the same number of entries in each column of a given table. If a maximum capillary pressure is defaulted for all depth values, in a problem where a value could correctly be defined, then the end point for each cell in the same scaling region will be the value in the appropriate saturation function table. However, if a maximum capillary pressure is defaulted and the end-point has been defined at other depths, the endpoint versus depth table will be filled in by linear interpolation within the table and constant extrapolation at the ends of the table. End-point values entered for absent phases are ignored.

Parameters/Tables

As you seen in Figure 120, there are two tabs of “Parameters” and “Tables” in the end point scaling dialog. User can enter the data of residual saturations of oil and gas and connate water saturation using the parameter tab. In the other tab (i.e. tables), the tables are displayed. “Tables” tab includes data of the EPS. For example Right clicking on the tables, user will be able to perform copy, paste, insert or delete table row. You can press “Enter” key for same action of insert row. There is plot of the data in this tab. The top list of the plot section shows the variables for plotting. Moving mouse pointer on the plotted curves will display the related values and conditions (snapped or interpolated).

3-2-6- Aquifer Data

Aquifer is defined using “Aquifer Data” button. Afterward the window of the Figure 121 is opened. There is an importer button which is used for import Eclipse aquifer data. The Aquifer Data window consists of 3 main sections: Aquifers, Aquifer Data and Aquifer Recurrent Parameters. Types of the Aquifers and related tables are defined in the “Aquifers” section. For defining a new Aquifer, + button is used (similarly - is used for removing a defined Aquifer). There are three types of aquifer to define in this window:

- Fetkovich Aquifer
- Carter-Tracy Aquifer
- Numerical Aquifer

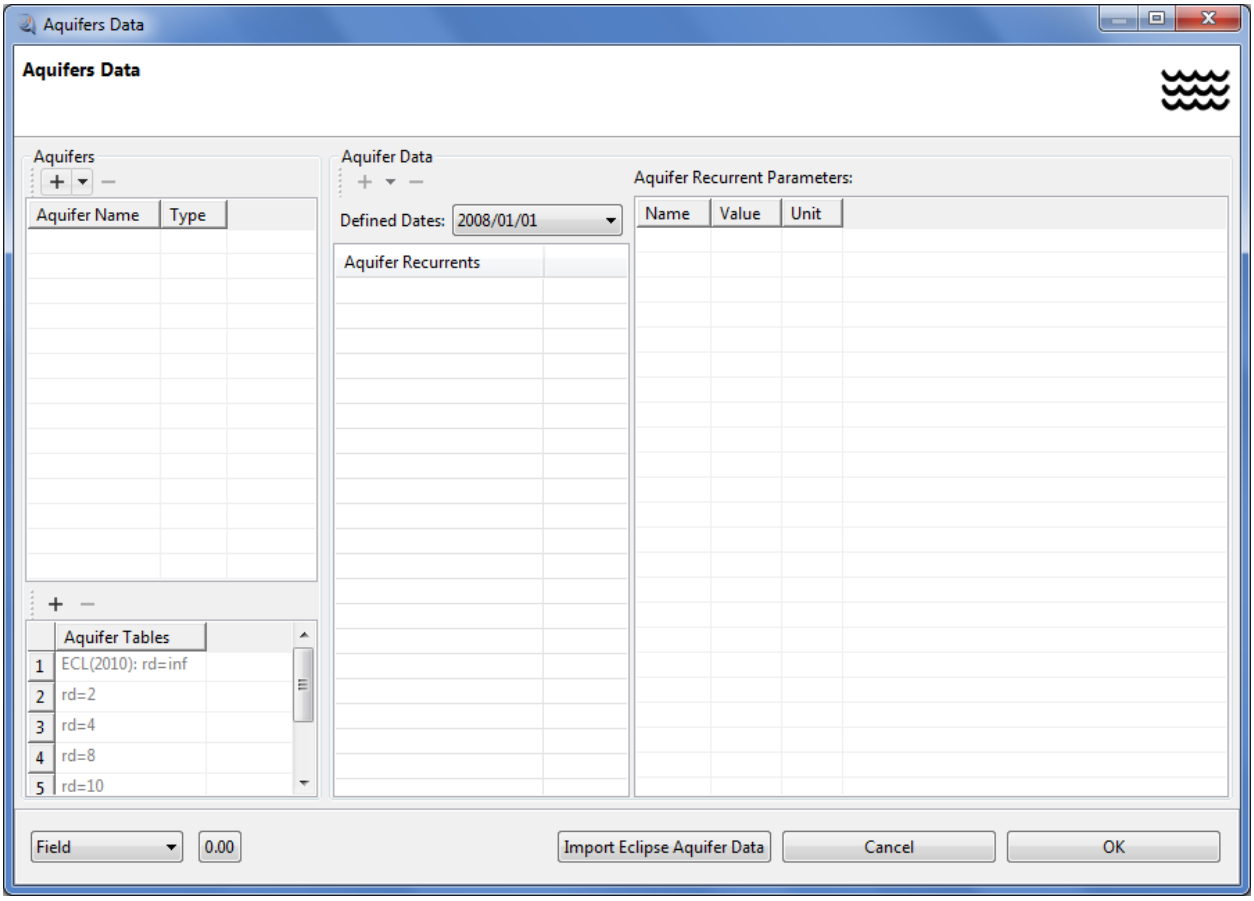


Figure 121- Aquifer Data

Fetkovich Aquifer:

User can define this type of aquifer using  button and selecting “Fetkovich Aquifer”. Afterward an Aquifer named”Aquifer-1” with Type of “Fetkovich” is inserted to the table. For renaming this aquifer, you can double click on it. Clicking on the aquifer name you can see the Aquifer Tables and Data (Figure 122).

There are some pre-defined aquifer tables, also you can click on + and define a new one using dimensionless time and pressure (Figure 123). Note that the dimensionless time must be monotonically increasing. All the table and plot settings are same as previous sections.

The “Aquifer Data” section is used for entering aquifer data. So, after selection a desired date (time) for aquifer definition and clicking on the + button, two options of “Add Aquifer Definitions” and “Add Aquifer Connection” are displayed. User must use both of these options to define aquifer.

Selecting "Add Aquifer Definition" the keyword of AQUFET is displayed. AQUFET needs following data which are displayed in the "Aquifer Recurrent Parameters" (Figure 124);

- Datum Depth
- Datum Depth initial Pressure
- Initial Water volume
- Total Compressibility (Rock + Water)
- Aquifer Productivity Index
- PVT Region

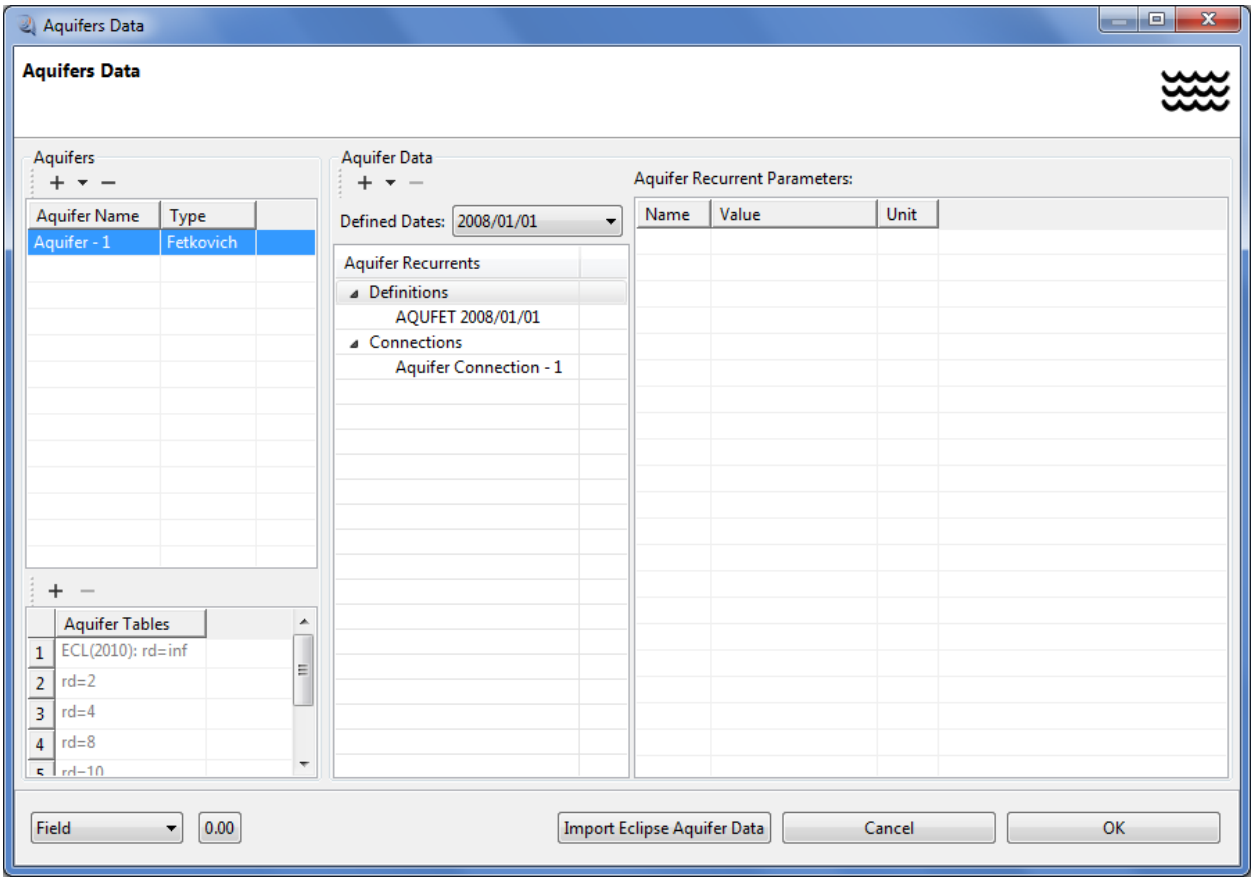


Figure 122- Fetkovich Aquifer Definition

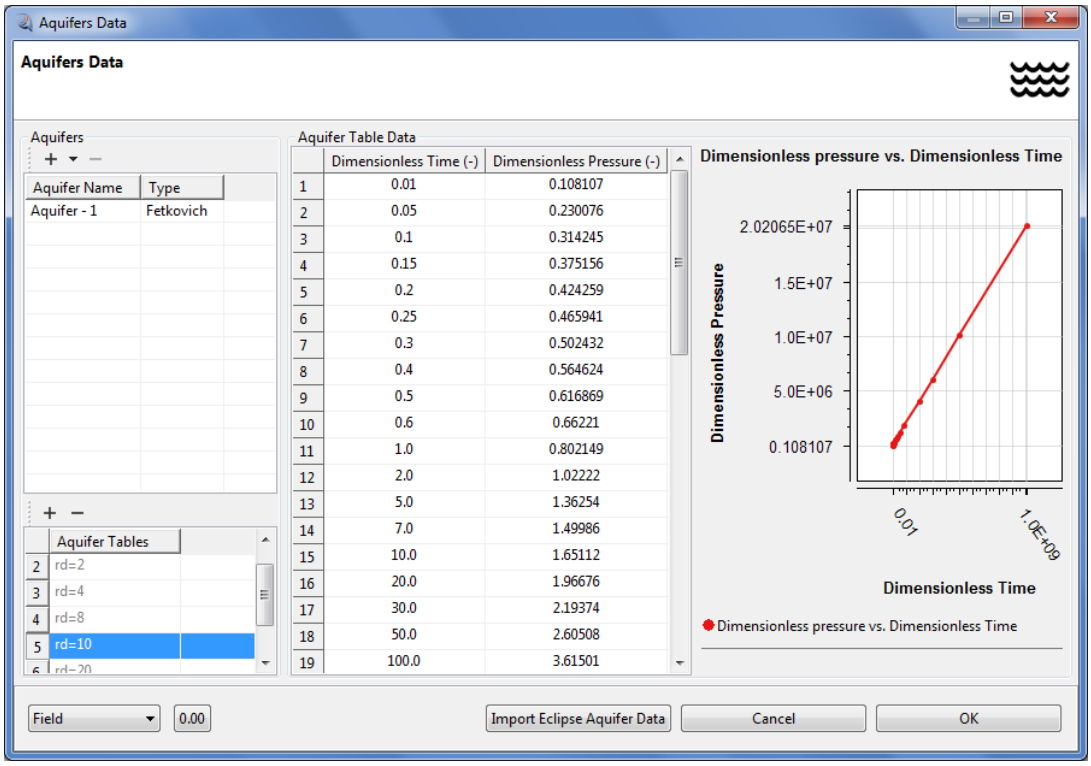


Figure 123- Aquifer Tables

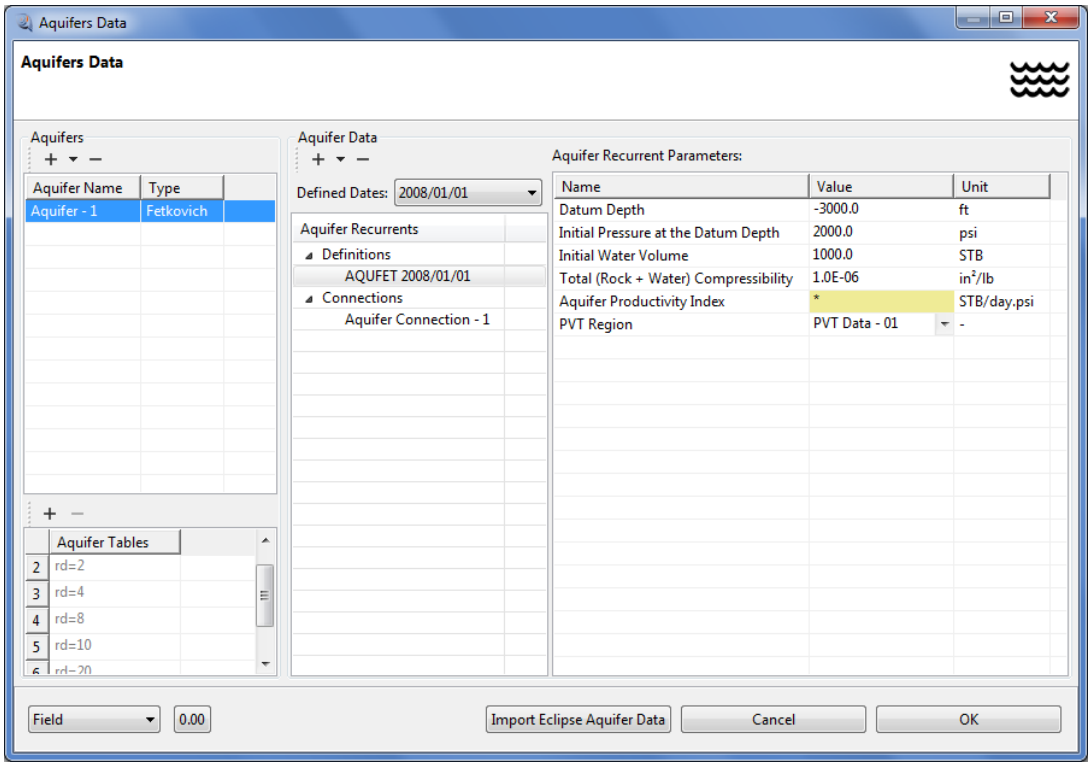


Figure 124- Fetkovich Aquifer Recurrent Parameters

There are some default values for aquifer recurrent parameters (yellow fields filed by *); user can see the values of the default by right clicking on the window and select “Show Default Values”. This option is available in all of the RETNA windows.

For defining any new connection for each aquifer, “Add Aquifer Connection” option is used, in which, the lower and upper indices in three directions, connection face, influx coefficient and influx coefficient multiplier should be entered. There is a checkbox which user can check it only when wants to allow aquifer connection to active neighbor one. Note that the above procedure must be repeated for defining a new aquifer connection.

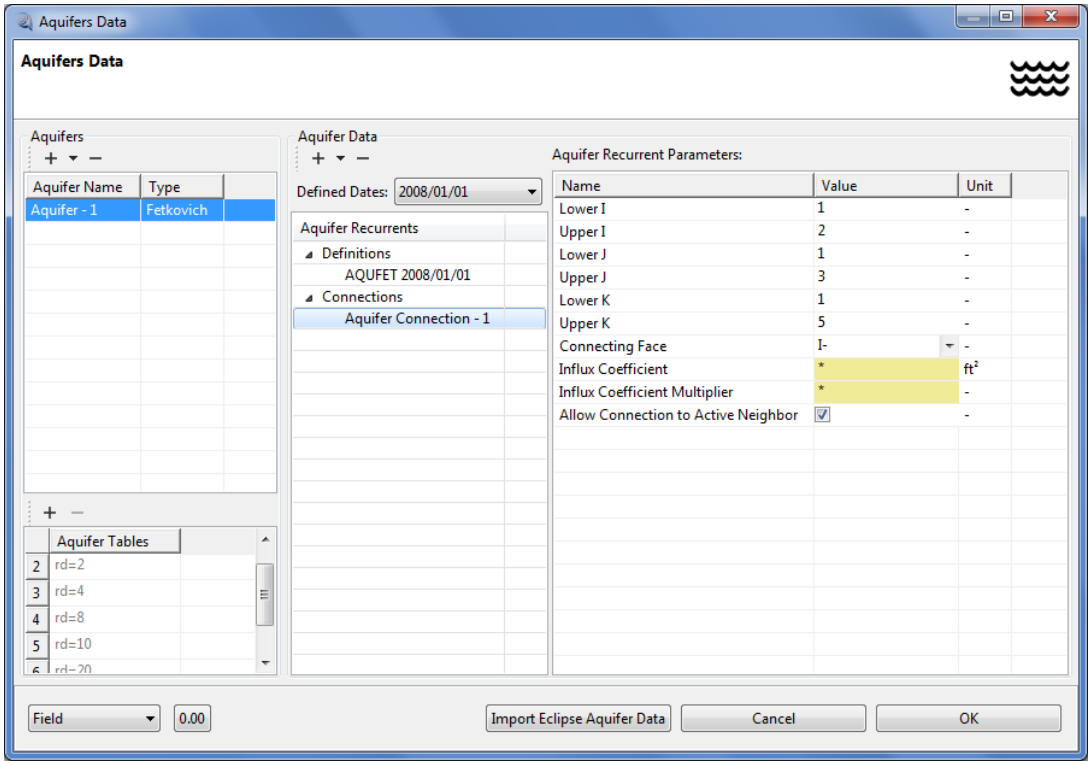


Figure 125- Aquifer Connections

Carter-Tracy Aquifer:

The Carter-Tracy aquifer definition procedure is same as Fetkovich aquifer. The only difference is the “Aquifer Recurrent Parameters” data (Figure 126). In the Carter-Tracy aquifer the following data are necessary:

- Datum Depth
- Initial Pressure
- Aquifer Permeability
- Aquifer Porosity

- Total Compressibility
- Radius
- Aquifer Thickness
- PVT Region
- Aquifer Table

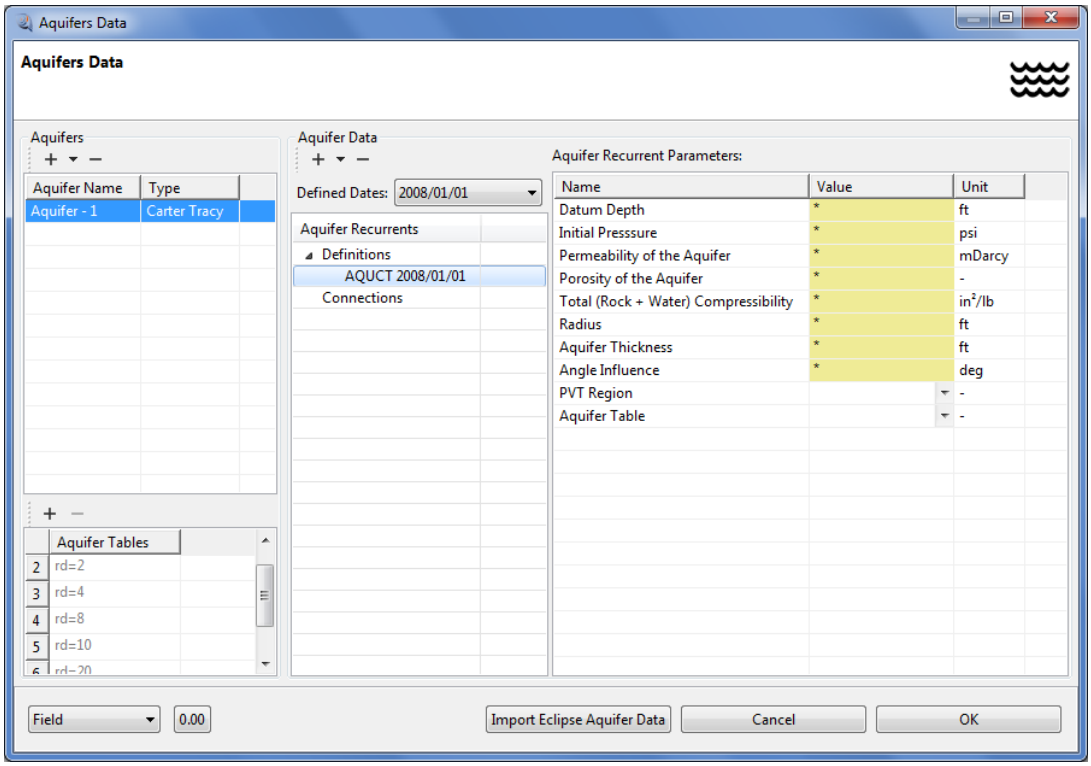


Figure 126- Carter-Tracy Aquifer

Numerical Aquifer:

The numerical aquifer definition procedure also is same as two previously mentioned aquifers. The only difference is the "Aquifer Recurrent Parameters" data (Figure 127). Following information are necessary for defining a numerical aquifer:

- Coordinates of 3 main directions of X, Y, Z for Aquifer Head
- Cross sectional Area
- Length of Aquifer
- Porosity
- Permeability
- Aquifer Depth
- Initial Pressure

- PVT Region
- Saturation Region

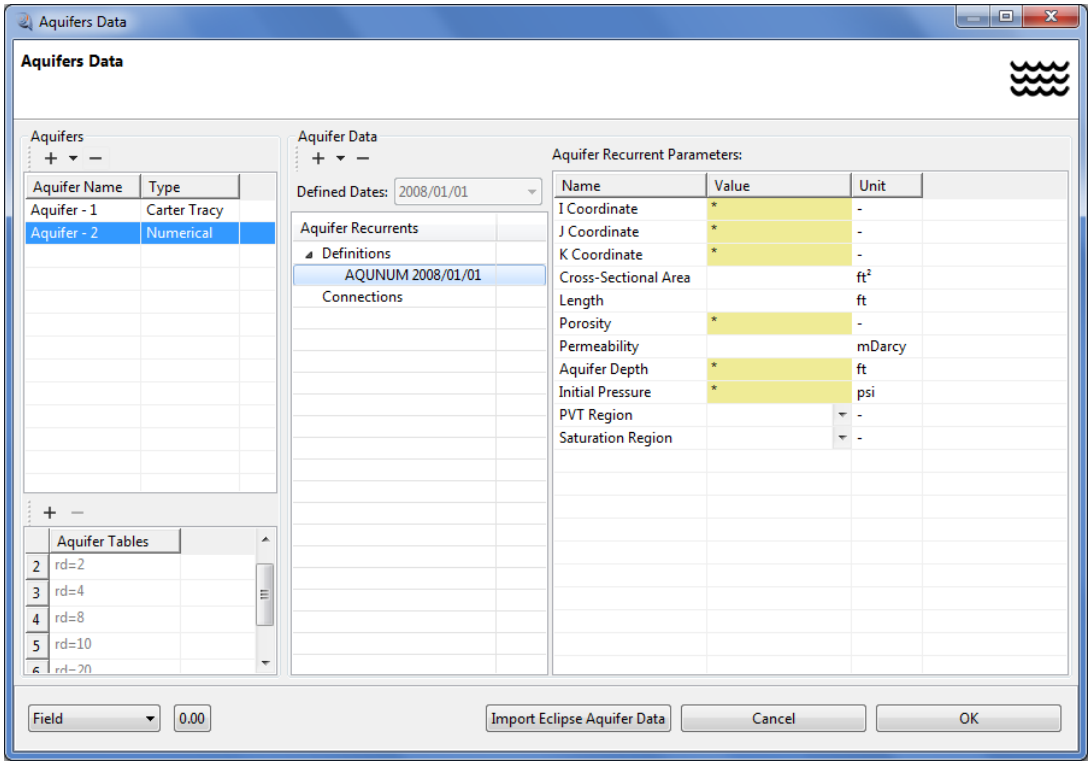


Figure 127- Numerical Aquifer Definition

3-3- Using Scheduling Data Tab

This tab includes steps, well and group scheduling, and recurrent data, which will be explained in following sections.

3-3-1-Date Manager

The button of "Open Date Manager" is used to manage time and date. User can define new steps by entering "Step Number", "Step Length" and time after start date.

Note: "Add Dates" button must be pressed for adding desired dates to provided list.

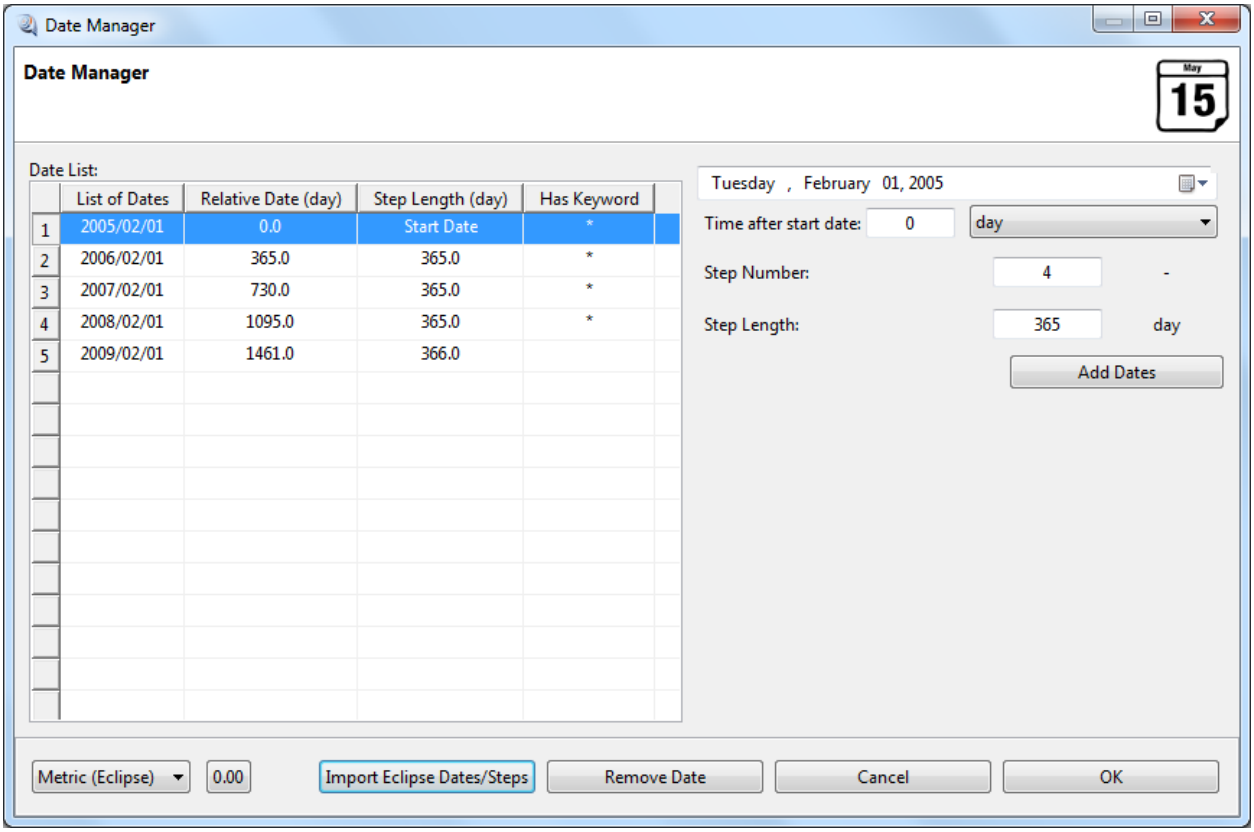


Figure 128- Date Manager

For defining new dates, the step number and length and the time after start date must be defined by user. Note that the start time may be different from project start time, if you click on any row of the date list, the related date will be set to start date. The unit of "Time after start date" can be selected from existed list.

There is a calendar in the right side of the date (Figure 129). User can select a desired date using calendar. The "Today" button will go to the current date. You can select the year and month using navigating arrows of calendar.

User can use *Ctrl* or *Shift* keys to select multiple dates for deleting or inserting. If you try to remove some dates, the warning message asking "Are you Sure?" will be displayed (Figure 130).

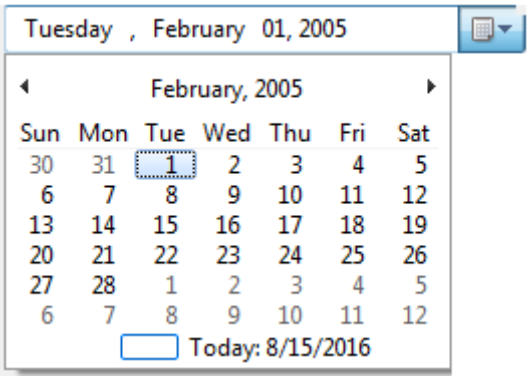


Figure 129- Select Date, Using Calendar

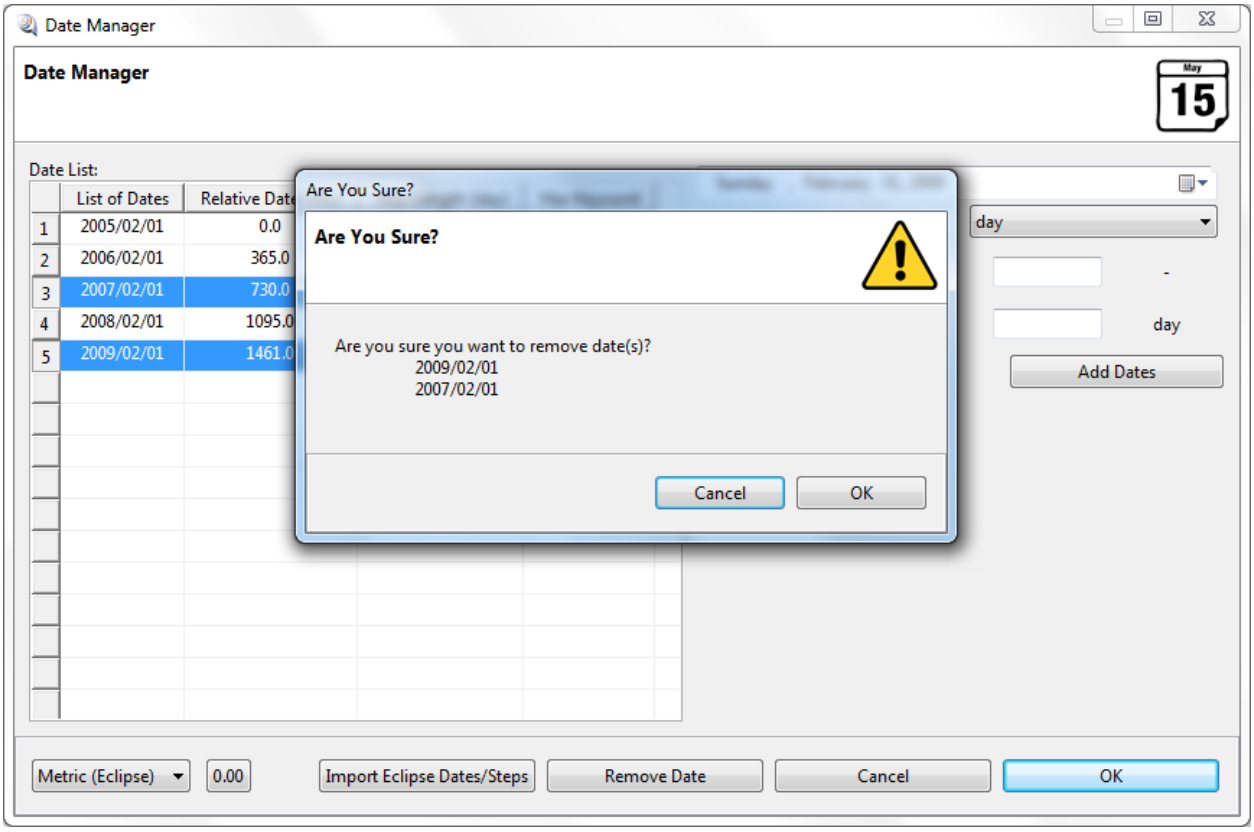


Figure 130- Warning Message of Remove Date

An error message is displayed if user wants to remove dates including keywords (Figure 131). In some cases the scheduling date/steps are numerous and have various time lengths which is an important issue, in such cases the dates/steps should be imported. So an importer of "Import Eclipse Dates/Steps" has been provided to resolve this issue. In

this importer, the start date and imported file unit must be defined by user before importing any files.

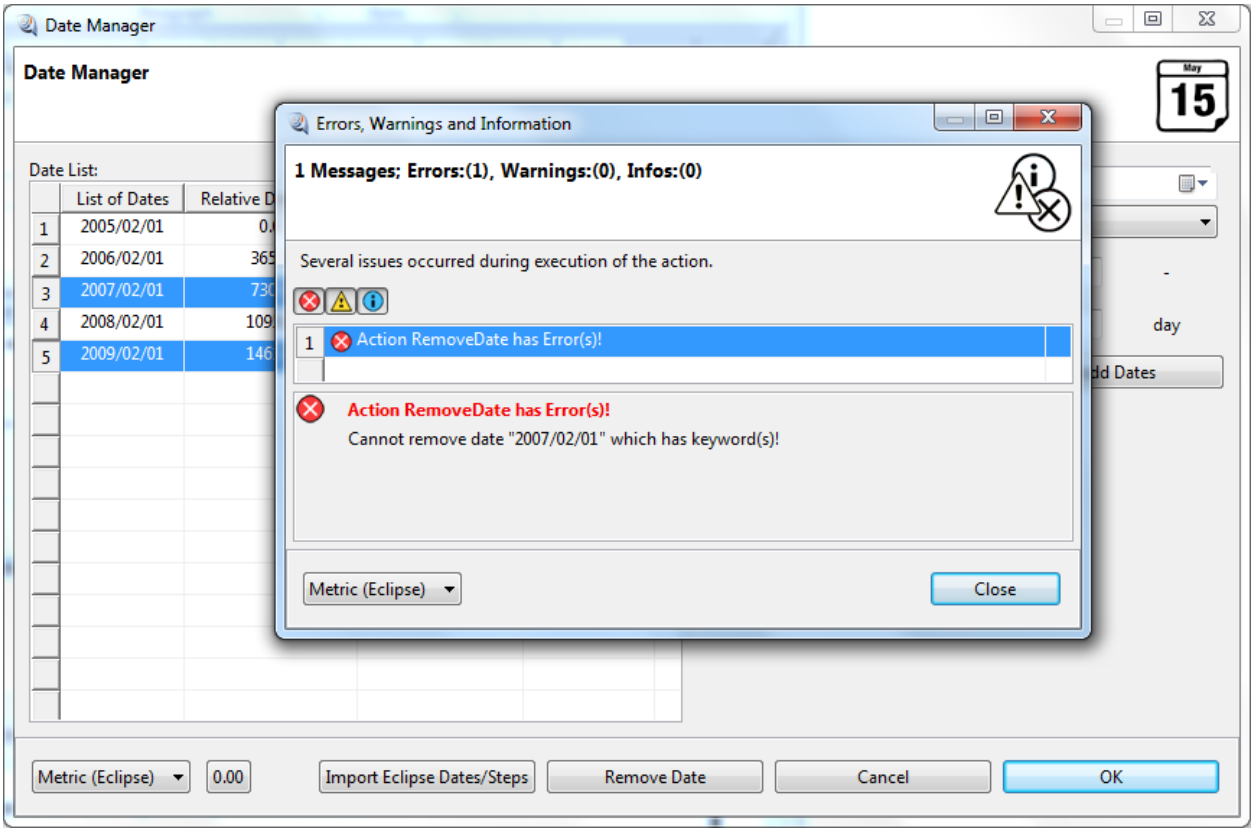


Figure 131- Removing Dates Including Keywords

3-3-2-Well Manager

User can start to define and manage wells using “Well Manager” button. Well manager window is shown on Figure 132.

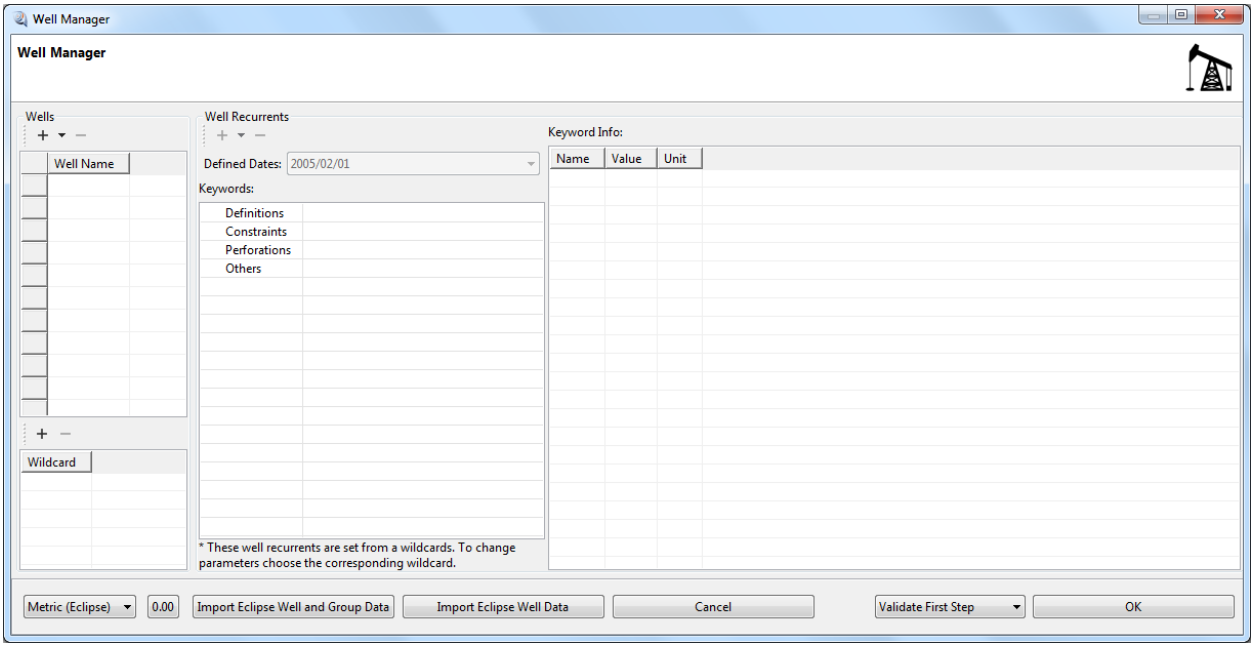


Figure 132- Well Manager Window

You can use **+** button for defining a new well using one of the “Add Well” or “Add Multiple Wells” options. Choosing “Add Well” a unique well is added to the well list. The **-** button is used to remove selected well. If user choose “Add Multiple Wells” the Figure of 133 is, in which, the name and the number of the wells are entered by user. Clicking on “Generate” button produces the well with entered base name. The head point coordination of all wells (I, J) must be entered by user, while the wells are added. User can either paste or enter these values one by one.

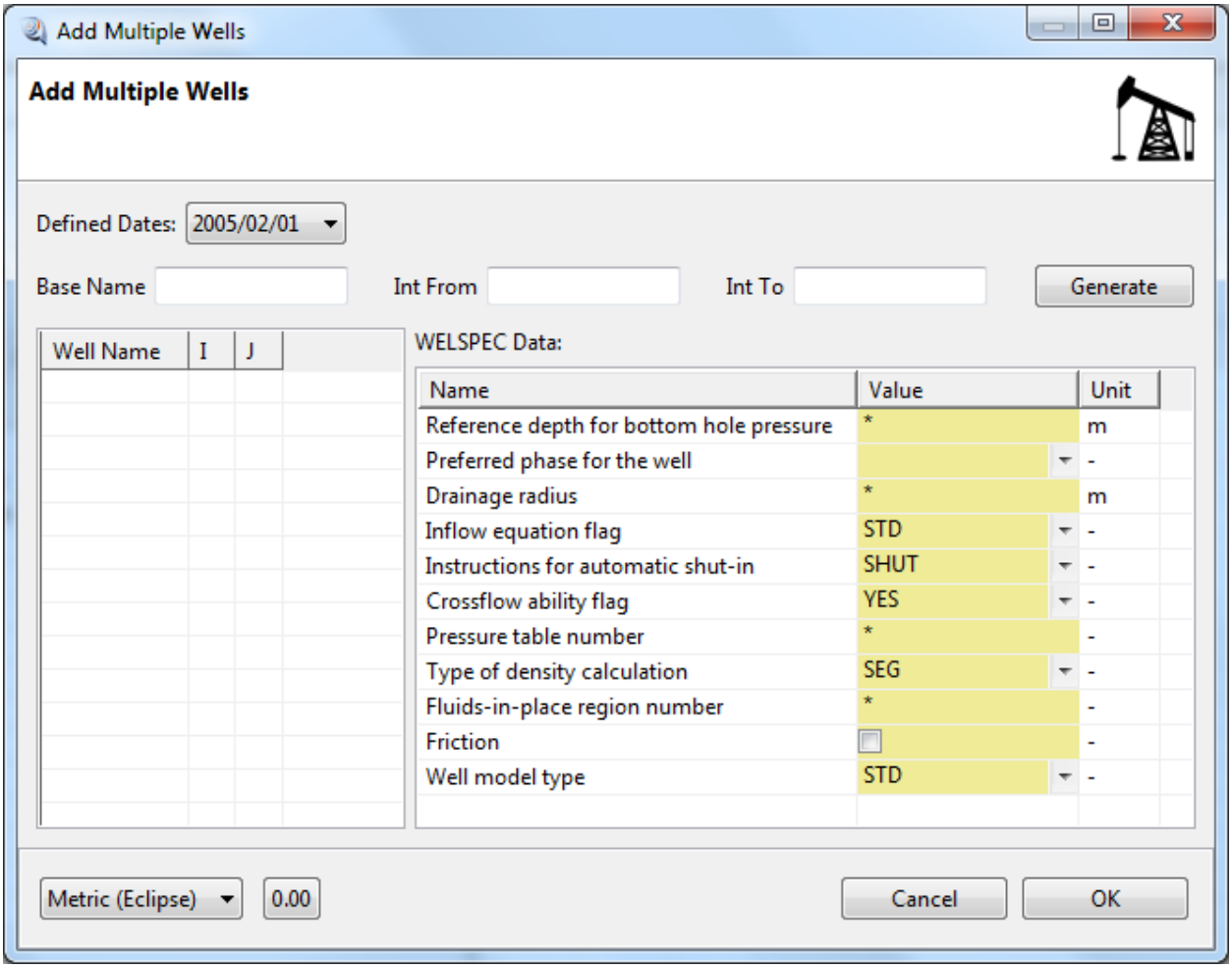


Figure 133- Add Multiple Wells

In the "Add Multiple Completions" dialog the range of I, J, K's are entered. Clicking on the button of  , the desired connections are added.

After adding wells the well recurrent data must be defined, there are 4 types of the keywords in the well recurrent section:

- Definition
- Constraints
- Perforations
- Others

1- Definition:

1-1- WELSPECS (General specification data for wells):

The WELSPECS introduces a new well, defining the position of the wellhead, its bottom hole reference depth and other specification data. Figure 134 displays the WELSPECS data. It consists of the following data:

- I-location of the well head or heel
  - For horizontal wells, this item defines the I-location of the heel of the well. If all connections are vertical the well can still connect with grid blocks having different I-locations.
- J-location of the well head or heel
  - For horizontal wells, this item defines the J-location of the heel of the well. If all connections are vertical the well can still connect with grid blocks having different J-locations.
- Reference depth for bottom hole pressure
  - This should be situated in the vicinity of the perforations. A recommended location is the topmost perforation of the well.
- Preferred phase for the well
  - OIL: The well is primarily an oil well
  - WATER: The well is primarily a water well
  - GAS: The well is primarily a gas well
  - LIQ: May be used to output the oil + water phase P.I
  - TOTAL: May be used to output the oil + water + gas phase P.I
- Instructions for the automatic shut-in
  - STOP: Stop well above formation (allowing cross-flow)
  - SHUT: Isolate well from the formation.
  - The well can be automatically shut or stopped if a well or group economic limit, or a group flow limit, is violated.
- Cross-flow ability flag
  - YES: Cross flow allowed in the well
  - NO: Cross flow not allowed; the connections act as one-way valves, preventing flow if there is a reverse drawdown
- Type of density calculation

  - SEG: Segmented density calculation. The flowing mixture density is calculated between each adjacent pair of connections, and depends on the relative quantities of the phases flowing in these particular sections of the wellbore. This calculation is more accurate than the other choice if different mixtures of fluids enter the well through each connection, but it is treated explicitly in the numerical scheme.
  - AVG: Averaged density calculation. The flowing mixture density is treated as uniform throughout the wellbore at formation level, and depends on the total inflow rates of each phase and the well's bottom hole pressure. This calculation is performed implicitly in the numerical scheme

- Fluid in place region number
  - It supplies the reservoir conditions used in calculating the well's reservoir fluid volume rate. (The reservoir fluid volume rate is based on the volume that the produced or injected mixture would occupy at the average conditions - hydrocarbon pressure, Rs, Rv - within the specified fluids-in place region). If a zero value is entered, the average hydrocarbon conditions in the field are used. If a negative value is entered, the region number is set equal to the fluids-in place region number of the lowest grid block in which the well is completed. If a positive value is entered, it is interpreted as the required fluids-in-place region number.
- Friction
  - Enabling Friction well option

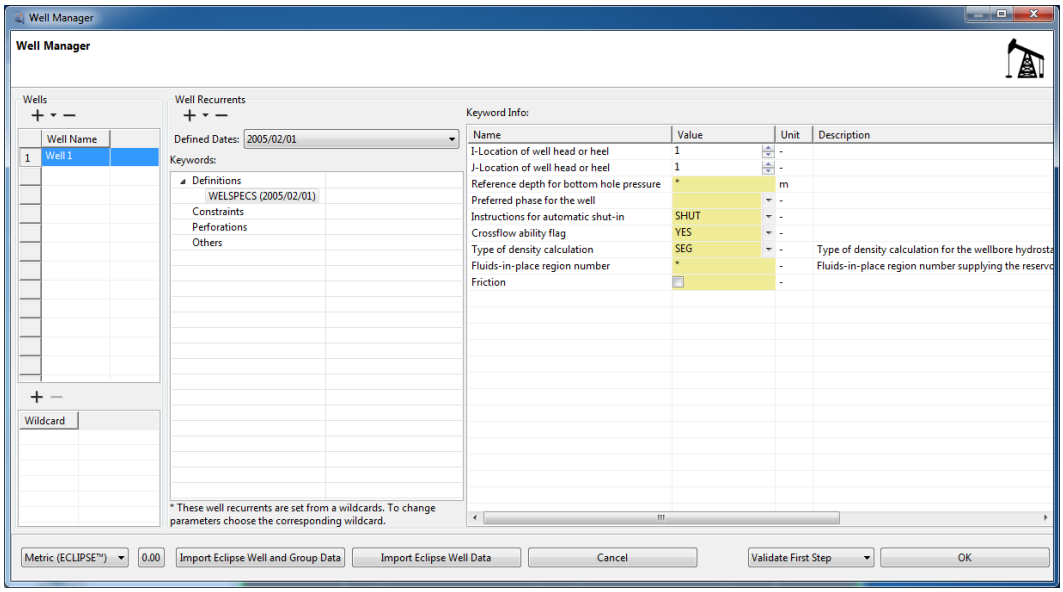


Figure 134 WELSPECS Data

2- *Constraints:*

2-1- *WGRUPCON (Sets well guide rates for group control):*

The WGRUPCON data has been displayed in Figure 135. It contains the following data:

- Available for group control
  - Check: The well is automatically placed under group control if a rate target is imposed on its group, or on a higher level group to which it connects, or on the field.
  - Uncheck: The well is prevented from coming under group control it flows according to its own target or limits.
- Guide Rate
  - It is a dimensionless number governing the well's share of the group's target rate. The target rates of the wells under group control are set in proportion to their guide rates, to meet the group or field production target. If the guide rate phase differs from the phase under control, the guide rate is translated into a guide rate for the controlled phase using the well's flow rates at the beginning of each time step. If a zero or negative guide rate is specified, the well's guide rate is set at the beginning of each time step from the general formula

specified with keyword GUIDERAT. If a general formula has not been specified, or for injection wells (for which there is no general formula), the well's guide rate is set equal to its production or injection potential (that is the flow rate the well would instantaneously achieve in the absence of any rate constraints.

- Scaling Factor
  - The well's guide rate is multiplied by the number specified here.
- Target Phase
  - OIL: Guide rate applies to water phase
  - WAT: Guide rate applies to oil phase
  - GAS: Guide rate applies to gas phase
  - LIQ: Guide rate applies to liquid phase (for production wells only)
  - RAT: Guide rate applies to the surface flow rate of the phase that the well is injecting (for injectors only)
  - RES: Guide rate applies to the reservoir fluid volume rate

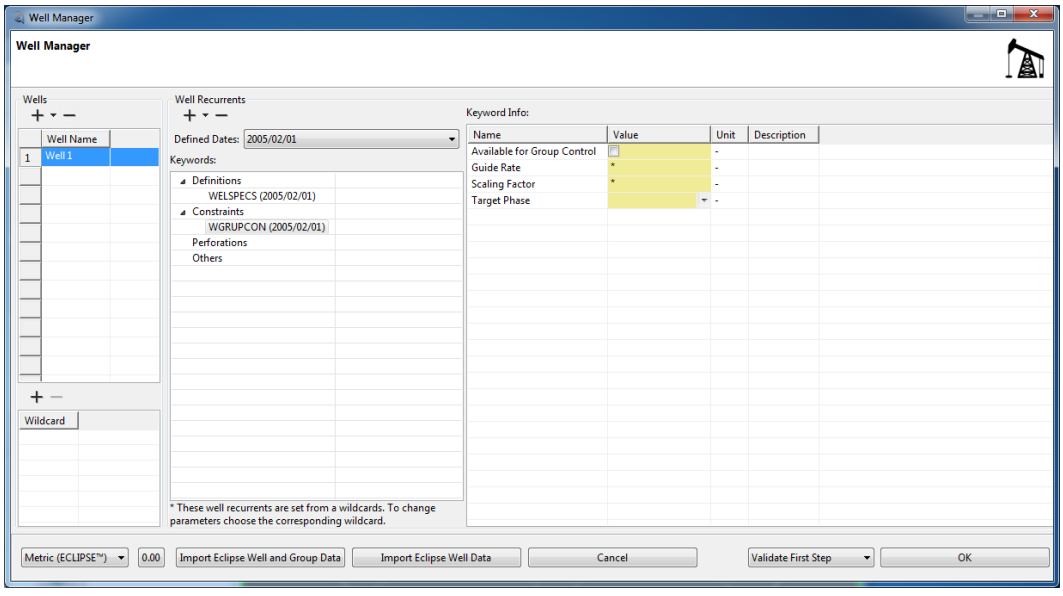


Figure 135 WGRUPCON Data

2-2- *WCONHIST (Observed rates for history matching production wells):*

WCONHIST is used in place of WCONPROD to declare production wells as special history matching wells, and to enter their observed flow rates (and

optionally their measured BHP and THP values). The equivalent constraint for defining history matching injection wells is WCONINJH. Wells can be declared as history matching wells during the history matching process, when their oil, water and gas production rates are known. Figure 136 displays the WCONHIST data. It contains the following data:

- Open/Shut Flag
  - OPEN: Well open for production
  - STOP: Well stopped off above the formation
  - SHUT: Well completely isolated from the formation
- Control Mode
  - ORAT: Controlled by the observed oil rate
  - WRAT: Controlled by the observed water rate
  - GRAT: Controlled by the observed gas rate LRAT: Controlled by the observed liquid rate (oil + water)
  - RESV: Controlled by the reservoir fluid volume rate calculated from the observed phase flow rates
- Observed Oil Production
- Observed Gas Production
- Production Well VFP Table
  - Reset: The well's THP is calculated, but the lower limit check is disabled so that the well does not go on to THP control.
  - Unchanged: does not require the THP to be calculated.
- Artificial Lift Quantity
  - This should be defaulted to 0, if you do not require the THP to be calculated.
- Observed Tubing Head Pressure
- Observed Bottom Hole Pressure

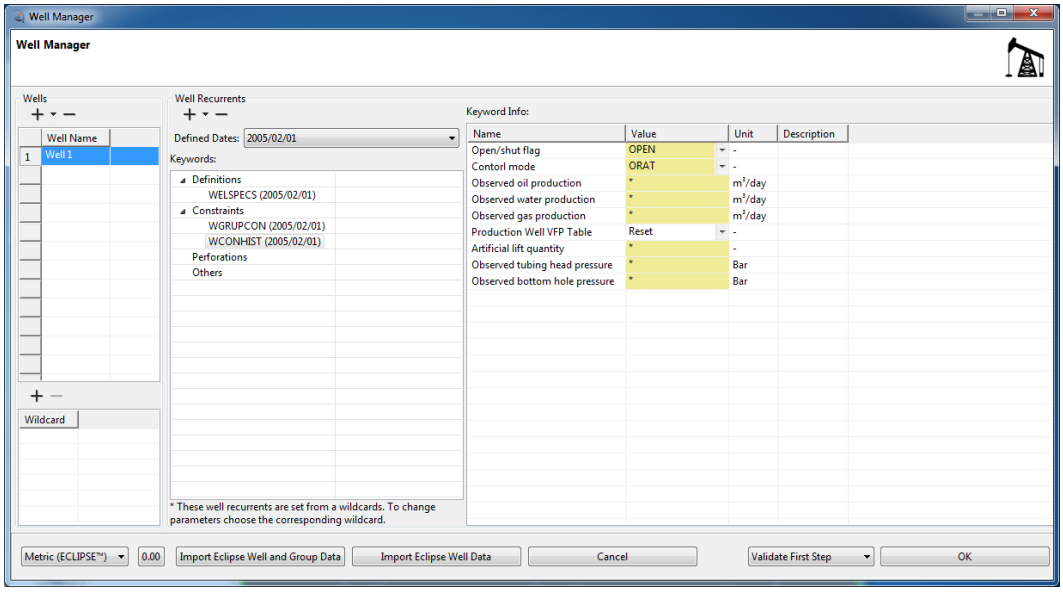


Figure 136 WCONHIST Data

2-3- *WHISTCTL (Control mode instructions for history matching wells):*

This constraint contains instructions that influence the control of all history matching wells. Figure 137 displays the WHISTCTL data. It contains the following data:

- Control Mode Override
  - ORAT: Wells are controlled by the observed oil rate
  - WRAT: Wells are controlled by the observed water rate
  - GRAT: Wells are controlled by the observed gas rate
  - LRAT: Wells are controlled by the observed liquid rate (oil +
  - RESV: Wells are controlled by the reservoir fluid volume rate calculated from the observed phase flow rates
  - NONE No control mode override is applied; the control modes specified in subsequent WCONHIST constraint will be honored
- Terminate Simulation
  - Check: Terminate the run
  - Uncheck: Continue the simulation with the well on BHP control

Note: This constraint will only change the control mode of a well when the well is in a subsequent WCONHIST constraint. It has no effect on previous WCONHIST constraints.

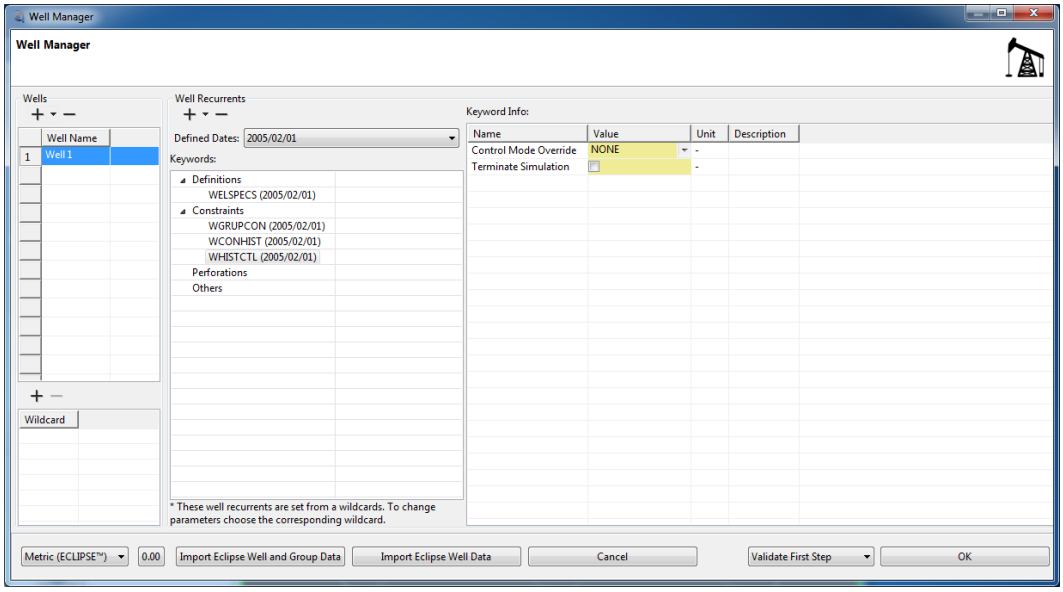


Figure 137 WHISTCTL Data

2-4- *WCONINJH (Observed rates for history matching injection wells):*

This constraint is used in place of WCONINJE to declare injection wells as special history matching wells, and to enter their observed flow rates (and optionally their measured BHP and THP values). The equivalent constraint for defining history matching production wells is WCONHIST. Wells can be declared as history matching wells during the history matching process, when their injection rates are known. During the simulation these wells are normally be constrained to operate at their observed injection rates, and their calculated BHP and/or THP values are compared against the measured values. Figure 138 displays the WCONINJH data. It contains the following data:

- Open/Shut Flag
  - OPEN: Well open for injection
  - STOP: Well stopped off above the formation
  - SHUT: Well completely isolated from the formation

When the well is shut or stopped, set the observed flow rate to zero. If the flow rates are set to zero while the well is open the behavior depends on whether cross flow is permitted in WELSPECS constraint. If cross flow is allowed the well behaves as if stopped. If cross flow is banned the well will be automatically shut.

- Injector Type
  - WATER: Water injector
  - GAS: Gas injector
  - OIL: Oil injector
- Observed Injection Rate
  - For a multiphase injector, this is the surface flow rate of the preferred phase of the well.
- Observed Bottom Hole Pressure
- Observed Well Head Pressure
- Injection Well VFP Table
  - Reset: The well's THP is calculated, but the upper limit check is disabled so that the well does not go on to THP control.
  - Unchanged: does not require the THP to be calculated.

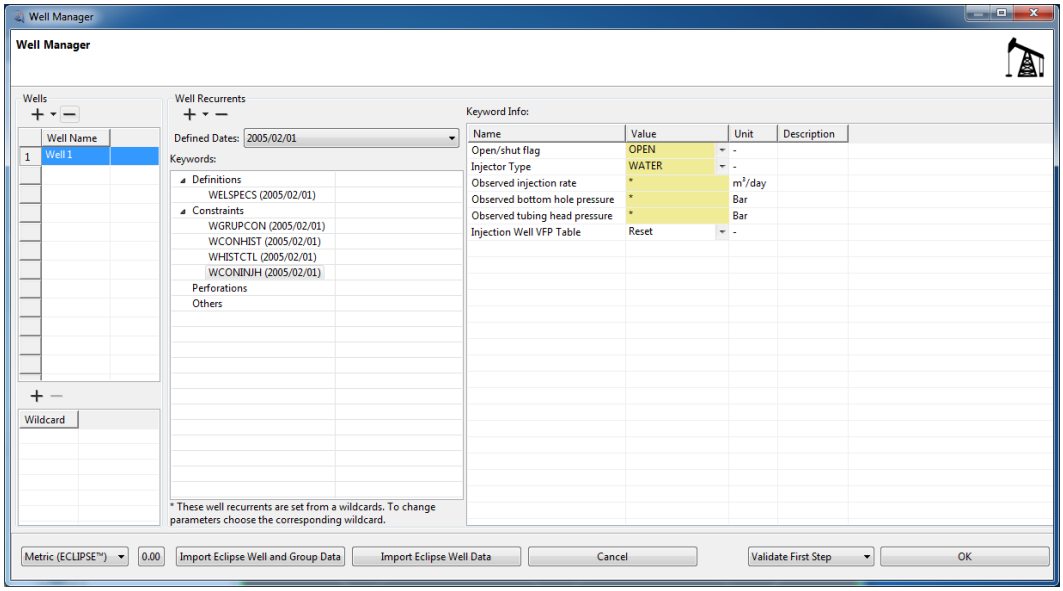


Figure 138 WCONINJH Data

2-5- *WELOPEN (Shuts or reopens wells or well connections):*

This constraint can be used to shut or reopen wells or well connections, without having to specify the rest of the well control data or connection data. Figure 139 displays the WELOPEN data. It contains the following data:

- Open/Shut Flag
  - OPEN: Well or connections open to flow
  - SHUT: Well or connections completely isolated from formation

- STOP: Well stopped off above formation. (If the record applies to a connection, it has the same effect as SHUT.)
- AUTO: Well or connections initially SHUT but open automatically.

- I-Location of Connecting Grid Block(s)
- J-Location of Connecting Grid Block(s)
- K-Location of Connecting Grid Block(s)
- Number of First Completion in Range
- Number of First Completion in Range

Note: If this constraint is used to shut a well during a history matching simulation, the actual production or injection rates are set to zero from this point in time, but all historical well rates remain unaltered. If the well was actually shut at this point in time, then the WCONHIST or WCONINJH constraints should be used to shut the well and thus zero the historical well rates.

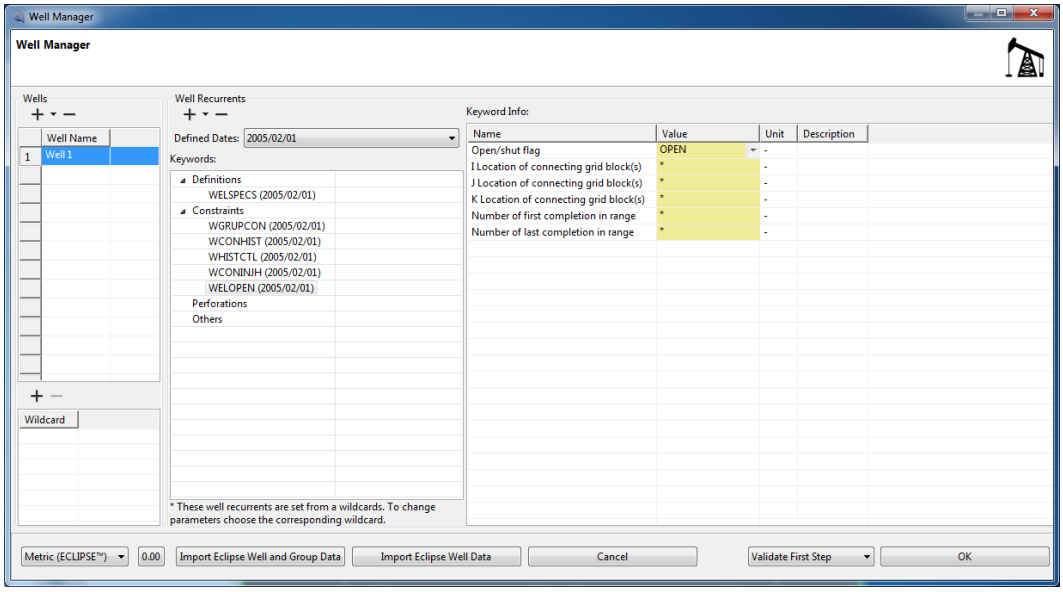


Figure 139 WELOPEN DATA

2-6- *WECON (Economic limit data for production wells):*

This constraint can be used to put an economic limit data only for the production wells. Figure 140 displays the WECON data. It contains the following data:

- Minimum Oil Production rate

  - If the oil production rate falls below the specified minimum, the well is shut or stopped
- Minimum Gas Production Rate
  - If the gas production rate falls below the specified minimum, the well is shut or stopped
- Maximum Water Cut
  - The water cut is the ratio of water to total liquid (oil + water) surface volume production rates
- Maximum Gas Oil Ratio
- Maximum Water Gas Ratio
- Work-Over Procedure
  - NONE: No action to be taken
  - CON: Shut worst-offending connection
  - PLUS_CON: Shut worst-offending connection and all those below it.
  - WELL: Shut or stop the well (refer to WELSPECS)
- Other Well
  - Name of well to be opened when this well is shut. If a well name is entered here, the named well is opened when the current well is closed automatically for any reason (for example on violating an economic limit). The named well must be fully specified as a SHUT producer or injector. Manual closure of the current well (for example using WCONPROD or WELOPEN) does not cause the named well to be opened.
- Target Quantity
  - RATE: The limits are applied to the well's actual flow rate. The minimum rate checks are bypassed for wells whose rate target is currently cut back under prioritization, to prevent them being closed while temporarily flowing with a small target rate. For wells subordinate to groups under guide rate control (GCONPROD), the minimum rate limits will be applied unless the group has a zero production rate target.
- Secondary Maximum Water Cut Limit

  - If this value is set > 0.0 (and < 1.0), a secondary water cut limit is applied to the well.
- Secondary Work-Over Procedure
  - NONE: Do nothing
  - CON: Shut worst-offending connection
  - PLUS_CON: Shut worst-offending connection and all those below it (see COMPORD)
  - WELL: Shut or stop the well (see WELSPECS)
  - LAST: The secondary water-cut is applied only to the last open connection in the well. (Connections on automatic opening class as open in this context.) This action differs from the ones above in that, when the well has only one connection left open and it begins to exceed its primary water-cut limit again, the well remains open until it exceeds its secondary water-cut limit but the other connections closed in previous work overs will not be reopened.
  - RED: The well is worked over each time it violates its primary water-cut limit, as normal. But each work over attempts to close sufficient connections to reduce the well's water-cut to below the specified secondary limit. Clearly, in this case, the secondary value should be less than the primary value. Assuming the well can be worked over to achieve this secondary value, it is not worked over again until the primary value is again violated. This option has the effect of requiring fewer work overs but closing more connections per work over.
- Maximum Gas Liquid Ratio
- Minimum Liquid Production Rate
  - If the liquid production rate falls below the specified minimum, the well is shut or stopped

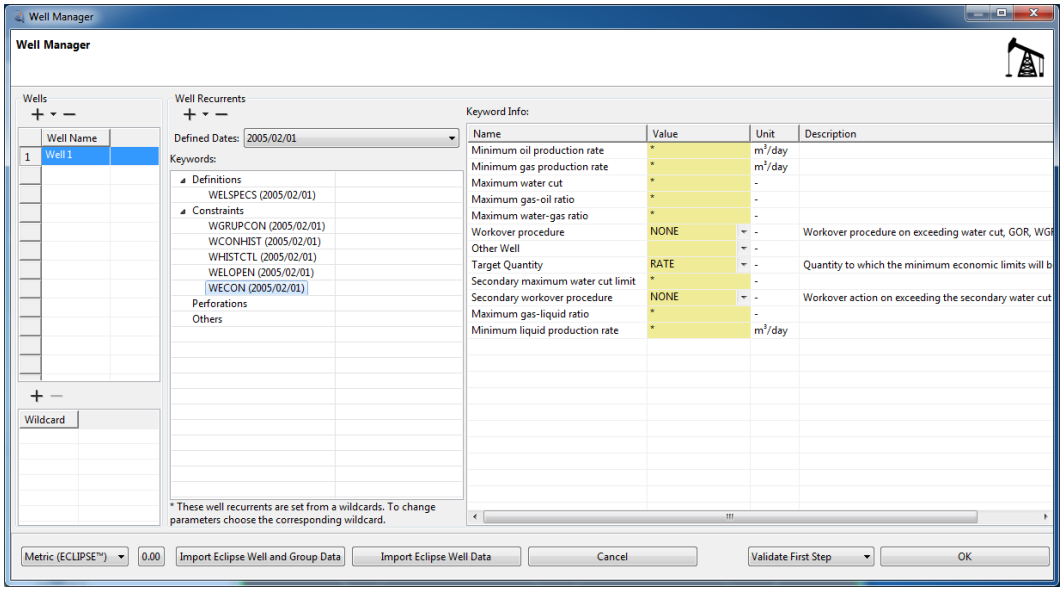


Figure 140 WECON Data

2-7- *WTEST (Instructions for periodic testing of closed wells):*

This constraint can be used for periodic testing of closed wells. Figure 141 displays the WTEST data. It contains the following data:

- Time Interval
  - The well is tested at the beginning of the first time step that starts after the specified interval has elapsed since it was closed. Subsequent tests are performed at each time step that starts after the specified interval has elapsed since the previous test. If the testing interval is very small (for example, 1 day), the well is tested at the beginning of each time step, unless of course the time steps are smaller than this.
- Test Number
  - After the well has been tested this number of times (regardless of whether any tests are successful), it is not tested any more unless this item is reset in a subsequent use of the WTEST constraint.
- Start-up Time
  - The well's efficiency factor is steadily ramped up over the specified start-up time when it is reopened after a successful test. Thus if the start-up time is large compared to the time step size, the well is brought on gradually, avoiding the time

step convergence difficulties sometimes observed when a high-rate well is suddenly opened.

- P
  - The well is tested if it has been closed for any physical reason (for example failure to operate at its BHP or THP limit). It is opened if the test shows that it can operate again.
- E
  - The well is tested if it has been closed due to an economic limit. If it has at least one open connection, the well is opened if it obeys all the limits set in WECON. Any closed connections will not be reopened. However, if the well has no open connections, it must have been closed after a workover resulting from a water cut, GOR or WGR limit violation. In this case, all connections that were closed by automatic workovers are tested individually. A connection is reopened if its water cut, GOR and WGR do not exceed the economic limits. Connections that were closed manually are not tested. The well is opened if at least one connection is reopened.
- G
  - The well is opened if it has been closed due to a GROUP or FIELD limit. The well is opened unconditionally, since it cannot be tested in isolation. If it causes a group or field limit to be broken, the remedial action is performed at the end of the time step in the usual manner. If the well has no open connections, all connections closed by automatic workovers are reopened.
- D
  - The well is tested if it has been closed on exceeding its maximum THP design limit. It is reopened if its THP has fallen below a maximum limit for reopening, when flowing at a specified rate (subject to obeying its other rate and pressure limits).
- C
  - Connections closed in automatic workovers are tested individually for reopening while the well is still open. A

connection is reopened if it does not violate any of the limits set in WECON or CECON. Any connections that were closed manually are not reopened. The difference between this type of test and the test triggered by an E is that the E test is only performed after the whole well has been closed by an economic limit, while the C test is performed while one or more connections have been closed but the well is still producing through other connections. The testing interval prevents testing until the specified number of days has elapsed since the most recent automatic closure of any connection in that well. Since the well itself does not have to be reopened after a successful C test, the start-up time does not apply. Wells (and, with the C option, connections) that were closed manually are not tested.

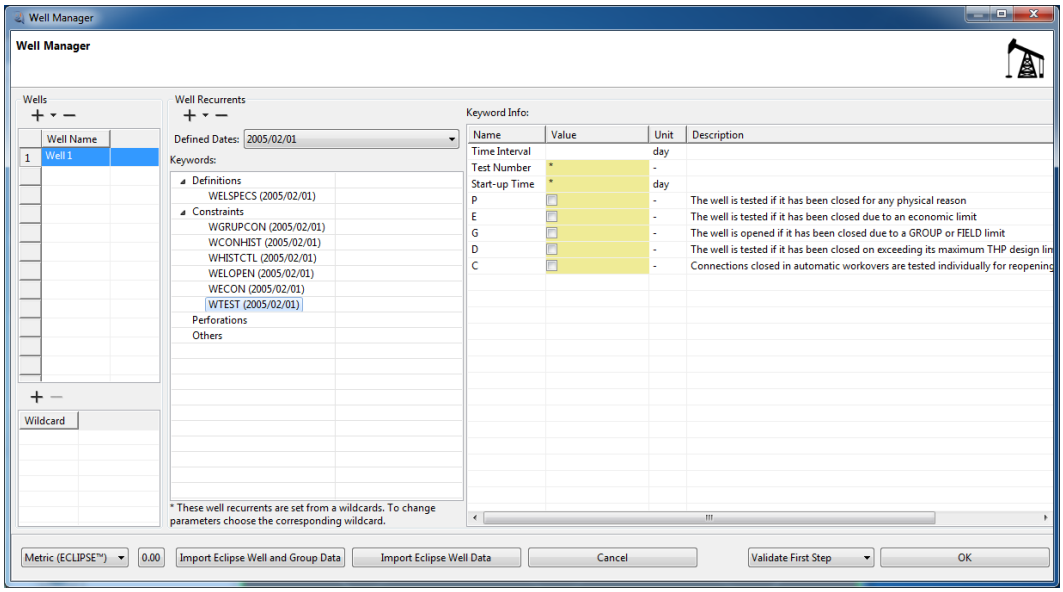


Figure 141 WTEST Data

2-8- *WVFPDP (Adjusts a well's BHP obtained from the VFP tables):*

This constraint specifies a pressure adjustment, which is added to the value of the well BHP obtained by interpolating the VFP tables. It can be used, for example, to help match a well's flow rate at a given THP, by adjusting the effective pressure loss between the bottom hole and the tubing head. A positive adjustment increases the BHP and makes a production well less

productive. Conversely, a negative adjustment improves the production. The reverse holds for injection wells. Figure 142 displays the WVFPDP data. It contains the following data:

- BHP Pressure Adjustment
- Tubing Pressure Loss Scaling Factor

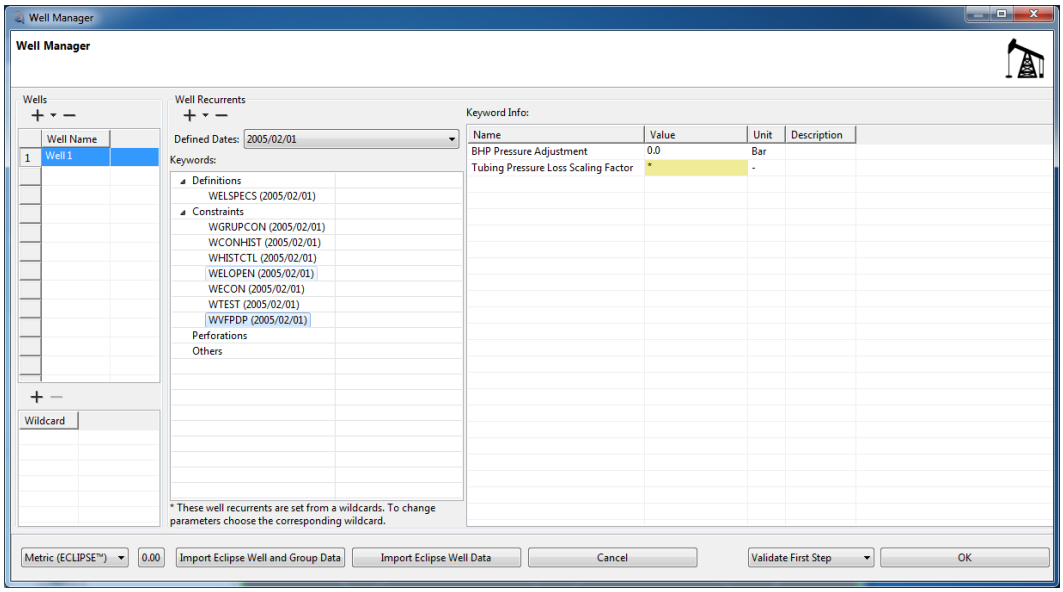


Figure 142 WVFPDP Data

2-9- *WCUTBACK (Well cutback limits):*

WCUTBACK is used to define well cutback limits for both production and injection wells. Figure 143 displays the WCUTBACK data. It contains the following data:

- Water Cut Upper Limit
- Gas Oil Ratio Upper Limit
- Gas Liquid Ratio Upper Limit
- Water Gas Ratio Upper Limit
- Rate Cutback Factor (Fraction)
  - This is the ratio of the reduced flow rate to the current flow rate. It should be less than 1.0
- Control Phase
  - OIL: Reduced oil rate
  - WAT: Reduced water rate
  - GAS: Reduced gas rate

  - LIQ: Reduced liquid rate
  - RESV: Reduced reservoir fluid volume rate
- Grid Block Pressure Limit
  - It is applied to any block containing an open connection to the well. This is treated as a lower limit for production wells, and an upper limit for injection wells.
- Grid Block Pressure Limit of Reversing
  - It is applied to any block containing an open connection to the well. This is treated as an upper limit for production wells, and a lower limit for injection wells.
- Water Cut Lower Limit for Reversing
- Gas Oil Ratio Lower Limit for Reversing
- Gas Liquid Ratio Lower Limit for Reversing
- Water Gas Ratio Upper Limit For Reversing
- Remove cutbacks when the well is worked over?
  - Checked: When the well is worked over after violating an economic limit, any cutbacks performed on the well are completely reversed by removing the rate limit on the control phase. Cutbacks are only reversed for workovers (that is connection alterations) and not if the well is shut or stopped.
  - Unchecked: Any cutbacks made to the well remain in force (unless reversed by the application of cutback reversal limits).

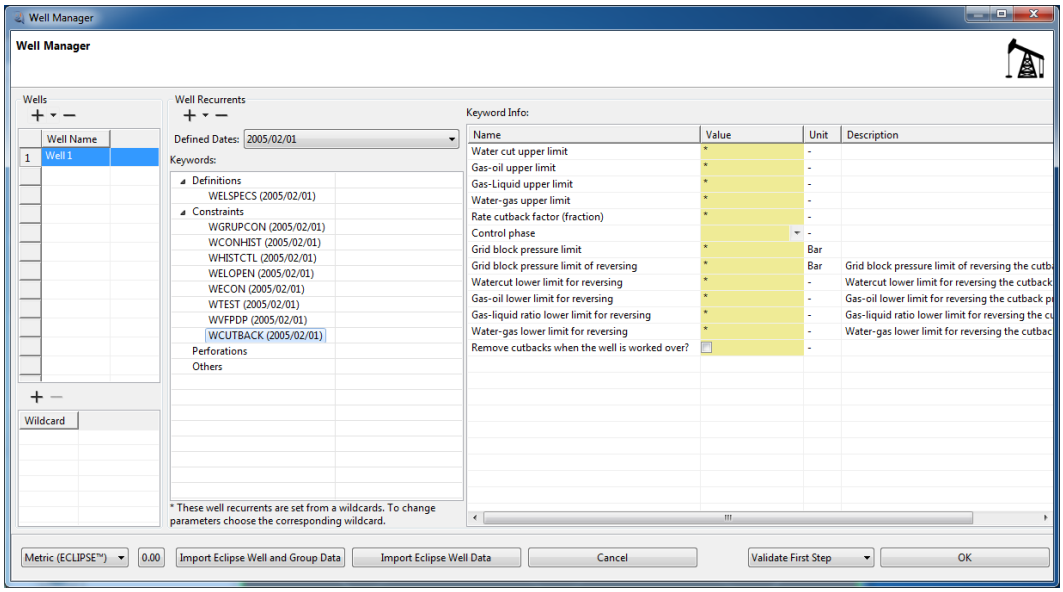


Figure 143 WCUTBACK Data

2-10- *WCONPROD (Control data for production wells):*

WCONPROD is used to put control data for production wells. Figure 144 displays the WCONPROD data. It contains the following data:

- Open/Shut Flag
  - OPEN: Well open for production
  - STOP: Well stopped off above the formation
  - SHUT: Well completely isolated from the formation
  - AUTO: Well initially SHUT, but is opened automatically as soon as constraints on drilling rate (WDRILTIM), drilling rig availability, and the maximum number of open wells per group allow. If no such constraints are specified, the well will open immediately. Wells on AUTO are opened in the order in which they are first specified in WELSPECS, unless any belong to a group that already has its maximum number of open wells or has all its available drilling rigs already occupied. It is preferable to shut or stop the well rather than give it a zero flow rate target. In particular, a zero flow rate target applied to a well with a THP limit may cause hunting between control modes.
- Control Mode

  - ORAT: Controlled by oil rate target
  - WRAT: Controlled by water rate target
  - GRAT: Controlled by gas rate target
  - LRAT: Controlled by liquid rate target
  - RESV: Controlled by reservoir fluid volume rate target
  - BHP: Controlled by BHP target
  - THP: Controlled by THP target
  - GRUP: The well is immediately under group control, to produce its share of a group or field target rate set with constraint GCONPROD.
- Oil Rate Target or Upper Limit
- Water Rate Target or Upper Limit
- Gas Rate Target or Upper Limit
- Liquid Rate Target or Upper Limit
- Reservoir Fluid Volume Rate Target or Upper Limit
- BHP Target or Lower Limit
- THP Target or Lower Limit
- Production Well VFP Table
  - Reset: The well's THP is calculated, but the upper limit check is disabled so that the well does not go on to THP control.
  - Unchanged: does not require the THP to be calculated
- Artificial Lift Quantity

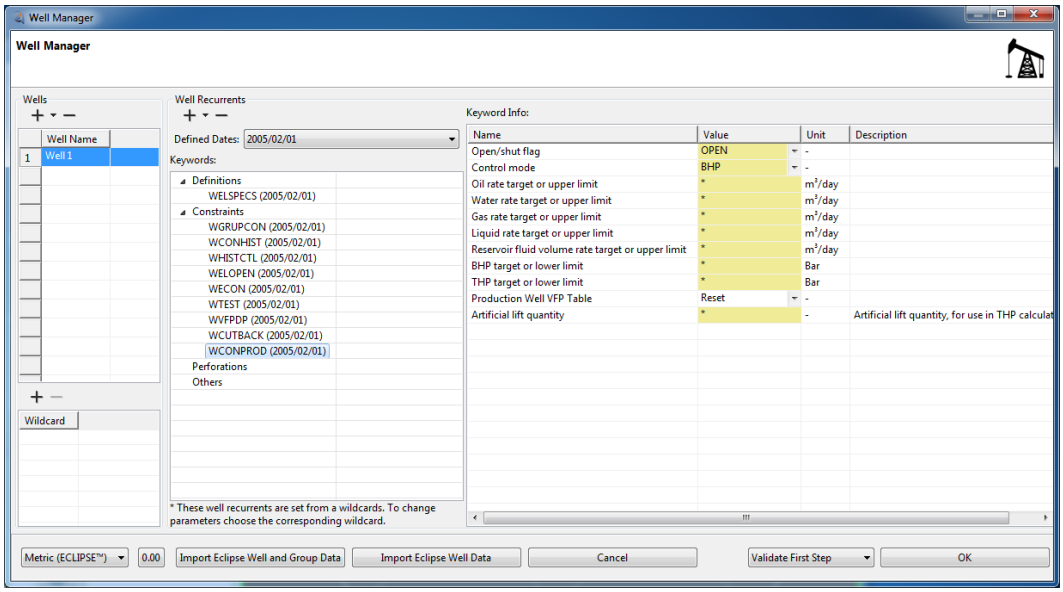


Figure 144 WCONPROD Data

2-11- *WCONINJE (Control data for injection wells):*

WCONINJE is used to put control data for production wells. Figure 145 displays the WCONINJE data. It contains the following data:

- Injector Type
  - WATER: Water injector
  - GAS: Gas injector
  - OIL: Oil injector
- Open/Shut Flag
  - OPEN: Well open for injection.
  - STOP: Well stopped off above the formation.
  - SHUT: Well completely isolated from the formation
  - AUTO: Well initially SHUT, but is opened automatically as soon as constraints on drilling rate (WDRILTIM), drilling rig availability (GRUPRIG), and the maximum number of open wells per group allow. If no such constraints are specified, the well opens immediately. Wells on AUTO are opened in the order in which they are first specified in WELSPECS, unless any belong to a group that already has its maximum number of open wells or has all its available drilling rigs already occupied.

- Control Mode
    - RATE: Controlled by surface flow rate target
    - RESV: Controlled by reservoir volume rate target
    - BHP: Controlled by BHP target
    - THP: Controlled by THP target. The control mode is not available with multiphase injectors.
    - GRUP: The well is immediately under group control, to inject its share of a group or field target set with the GCONINJE constraint.
- Surface Flow Rate Target or Upper Limit
    - For a multiphase injector, this is the surface flow rate of the preferred phase of the well.
- Surface Fluid Volume Rate Target or Upper Limit
- BHP Target or Upper Limit
    - It is recommended that this quantity should be set to a value not exceeding the highest pressure in the PVT tables, to ensure the tables are not extrapolated in the well. It is best not to default this quantity, as its value is used in calculating the well potential.
- THP Target or Upper Limit
- Injection Well VFP Table
    - Reset: The well's THP is calculated, but the upper limit check is disabled so that the well does not go on to THP control.
    - Unchanged: does not require the THP to be calculated

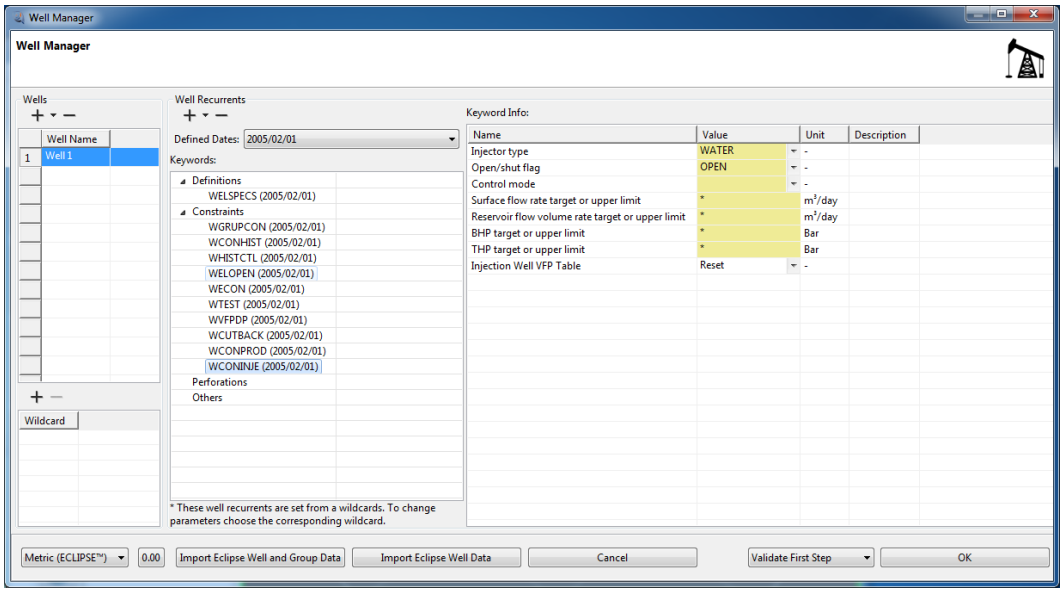


Figure 145 WCONINJE Data

2-12- *WELTARG (Resets a well operating target or limit):*

This constraint can be used to reset a target or limit value for a well, without having to re-specify all the other quantities required by the control constraints WCONPROD or WCONINJE. These other quantities are left unchanged, including the open/shut status of the well. The well control data must initially have been fully specified using WCONPROD or WCONINJE.

If the well has been declared a history matching well (see WCONHIST and WCONINJH) the WELTARG constraint may be used to modify its BHP limit, VFP table number, and artificial lift quantity. The other quantities should not be modified with this keyword. Figure 146 displays the WELTARG data. It contains the following data:

- Control to be Changed
  - ORAT: Oil rate
  - WRAT: Water rate
  - GRAT: Gas rate
  - LRAT: Liquid rate
  - RESV: Reservoir fluid volume rate
  - BHP: Bottom hole pressure

- THP: Tubing head pressure (A non-zero VFP table number must first have been set)
- VFP: VFP table number
- LIFT: Artificial lift quantity (A non-zero VFP table number must first have been set)

- Value
  - If the quantity is VFP or LIFT, then the value here may not be defaulted. For all other quantities, a defaulted value here will result in RETINA using the corresponding flowing value for the well at the end of the previous time step as the target or limit for the control or constraint in the first item.

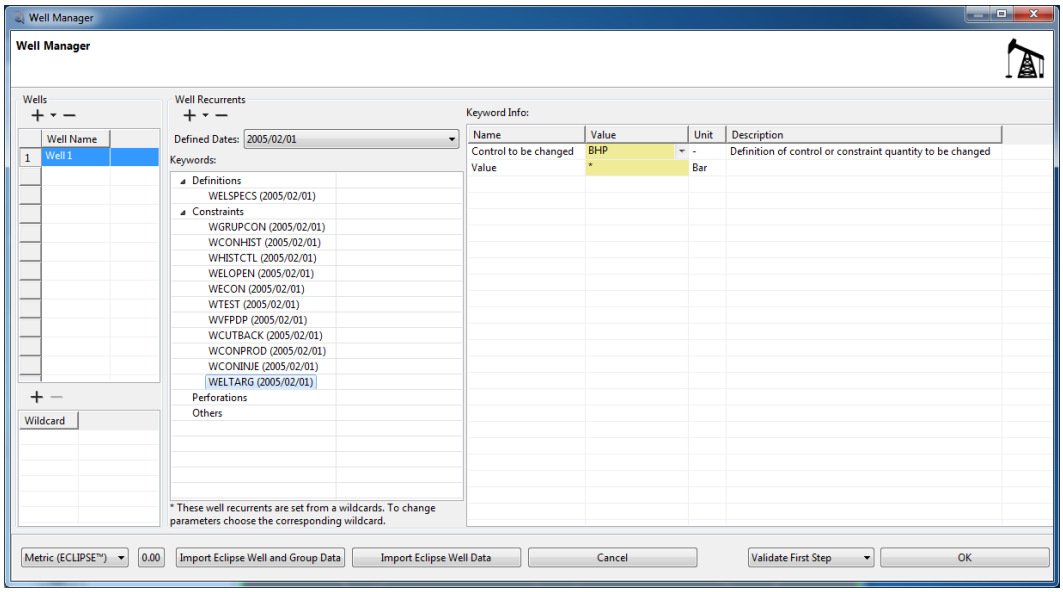


Figure 146 WELTARG Data

2-13- *WELPI (Sets well productivity/injectivity index values):*

This constraint can be used to set the productivity/injectivity index (PI) value of one or more wells to a specified value. The well connection properties must be defined in the usual manner, with COMPDAT, but the connection transmissibility factors are adjusted proportionately so that the calculated PI of the well equals the specified value. The PI is calculated using the phase and drainage radius defined in WELSPECS. Note that the calculation is not suitable for application to horizontal wells. The adjustment of the connection factors is performed at the time the WELPI constraint is entered, using the current grid block solution. If the fluid

mobility values in the grid blocks subsequently change, the well's actual PI also changes. Figure 147 displays the WELPI data. It contains the following data:

- Well Productivity Index for Liquid Phase
- Well Productivity Index for Gas

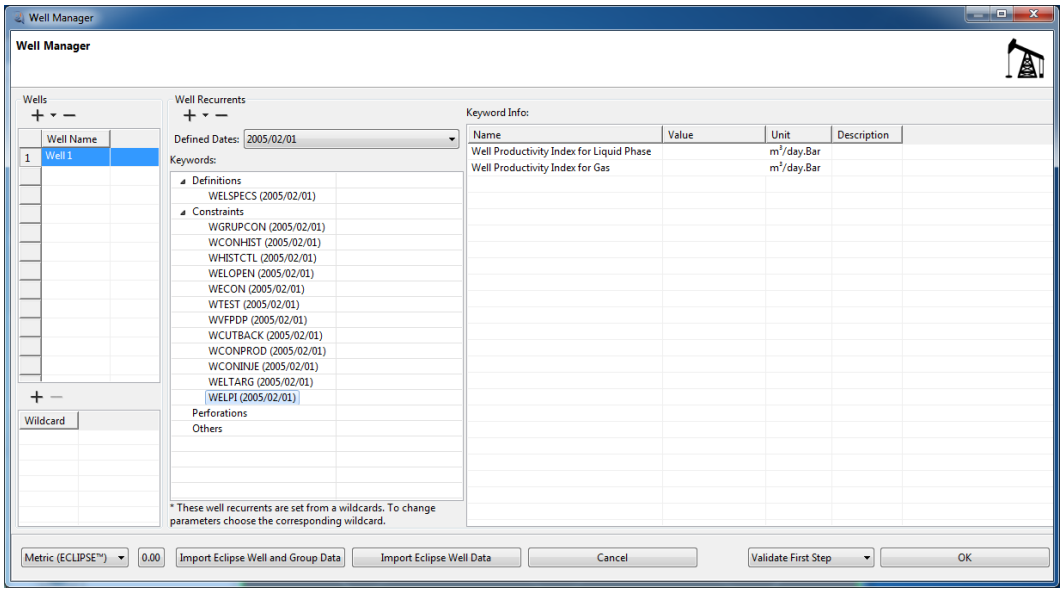


Figure 147 WELPI Data

2-14- *WEFAC (Sets well efficiency factors):*

This constraint is used to set well efficiency factors for down time of the wells. Figure 148 displays the WEFAC data. It contains the following data:

- Well Efficiency Factor
  - This is the fractional time for which the well is operational. For example, if the well is 'down' for 20 percent of the time, its efficiency factor will be 0.8. The efficiency factor must be greater than zero.

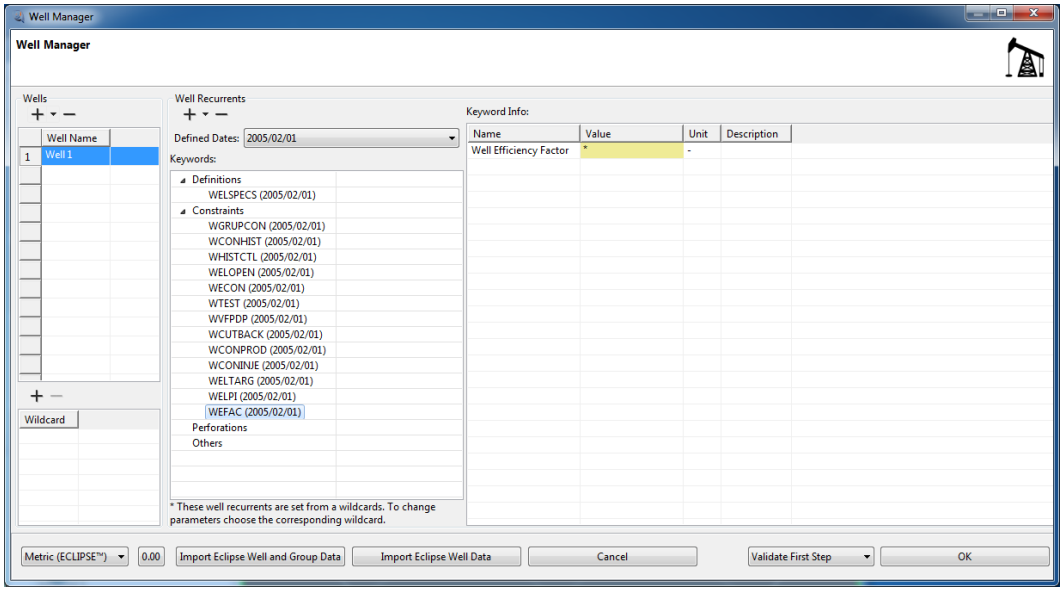


Figure 148 WEFAC Data

2-15- *WWPAVE (Individual well block average pressure controls):*

This constraint controls the calculation of well block average pressures for individual wells. They represent the average pressure of the grid blocks containing connections to a given well, and optionally their adjacent and diagonal neighbors. Neighbor contributions can be weighted according to either the connection transmissibility factors or the grid block pore volumes. The averages relate only to a Cartesian grid. Figure 149 displays the WWPAVE data. It contains the following data:

- F1
  - The weighting factor between the inner block and the outer ring of neighboring blocks, in the connection factor weighted average.

  Note: For one-block averages, F1 is effectively 1.0, whatever the value entered here.

- F2
  - The weighting factor between the connection factor weighted average and the pore volume weighted average.
- Average Density Calculation Method

- WELL: The hydrostatic head is calculated using the density of the fluid in the wellbore at the well connections.
- RES: The hydrostatic head is calculated using a representative density for the fluid in the reservoir. This density is calculated by averaging over fluid density for all the grid blocks associated with the well's connections. The average over phases is weighted by the phase saturation, and the average over grid blocks is weighted by pore volume. Whether the averaging is performed over all grid blocks with declared connections to the well or only those with currently open connections is determined by this item.
- NONE: Grid block pressures are not depth corrected.

Note: The wellbore fluid density is set to 0 whenever the well is shut. If WELL is selected there is a discontinuity in the reported pressure average when the well's status changes between shut and open/stopped.

- Well Connection Recognition Method

  These flag controls whether the grid blocks associated with all the well's declared connections contribute to the average pressure, or just those associated with the currently open connections.
  - OPEN: Only grid blocks associated with currently open connections are included in the averaging calculation.
  - ALL: Grid blocks associated with all currently defined connections (whether open or closed) are included in the averaging calculation.

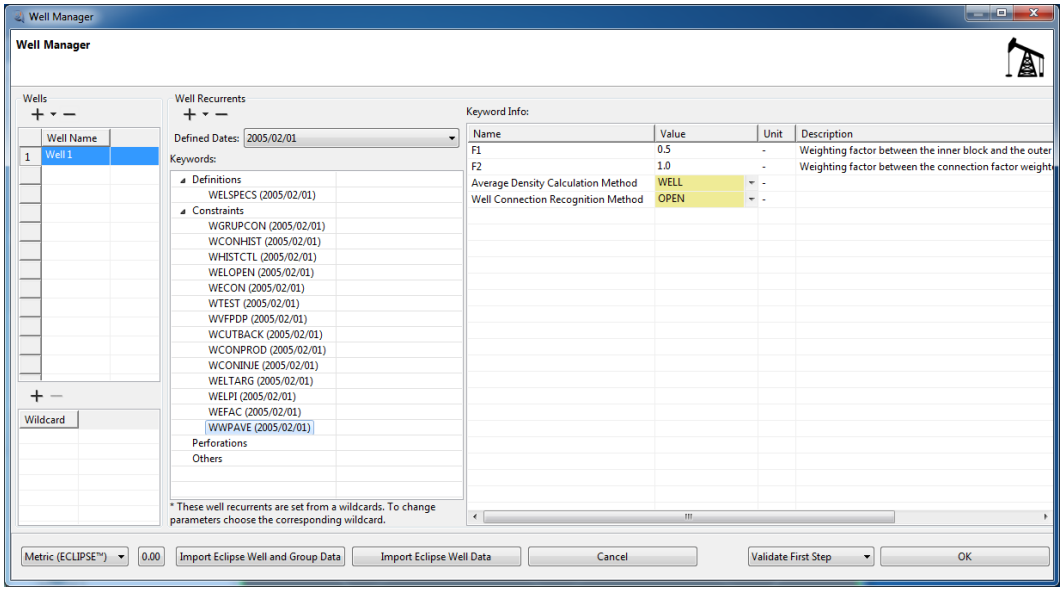


Figure 149 WWPAVE Data

2-16- *WPAVEDEP (Reference depth for well block average pressure calculation):*

This constraint may be used to modify the reference depth for the calculation of well block average pressures. By default, the grid block pressures are corrected to the well's bottom hole reference depth (see WELSPECS) by applying a hydrostatic head. The WPAVEDEP constraint enables the use of an alternative reference depth, which is used solely for this calculation. Figure 150 displays the WPAVE data. It contains the following data:

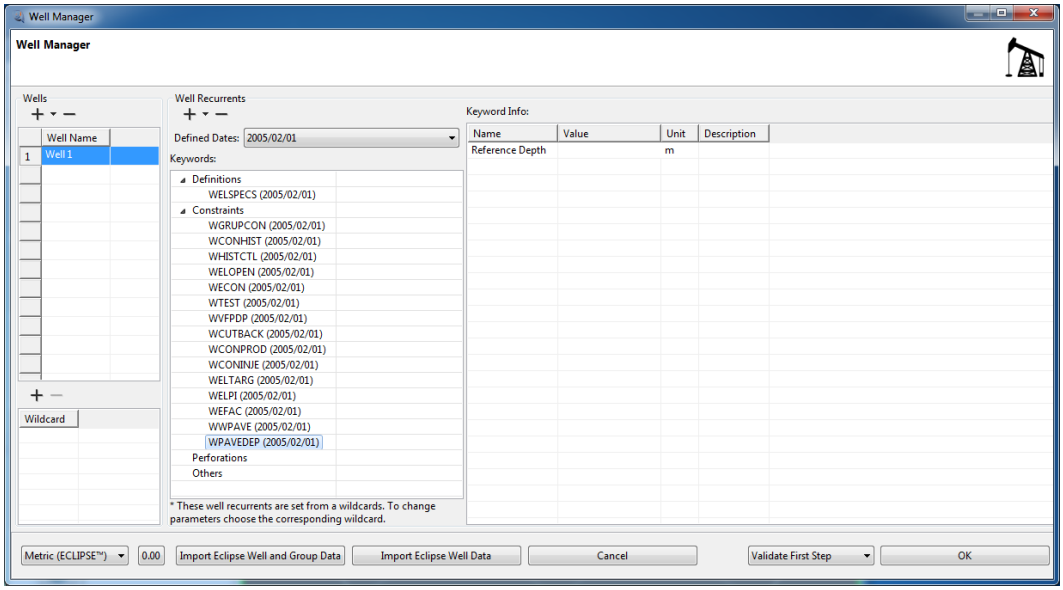


Figure 150 WPAVEDEP Data

3- *Perforations:*

3-1- Completion (Well completion specification data)

This specifies the position and properties of one or more well completions. Figure 151 displays the Completion data. It contains the following data:

- I-Location of the Completion
  - If set to 0 or defaulted, the I-location of the well head (entered in WELSPECS) will be used
- J-Location of the Completion
  - If set to 0 or defaulted, the J-location of the well head (entered in WELSPECS) will be used
- K-Location of the Completion
  - If set to 0 or defaulted, the K-location of the well head (entered in WELSPECS) will be used
- Open/Shut Flag
  - OPEN: Connection open to flow
  - SHUT: Connection closed off
  - AUTO: Connection initially closed, but will be opened automatically when another connection in the well is closed during an automatic workover. A connection on AUTO is opened each time the well is worked over, in the order in which the connections are first defined in Completion.
- Transmissibility Factor
- Wellbore Diameter
- Effective Kh
  - If a default, zero or negative value is entered, the Kh value is calculated from the grid block data. If a positive value for the connection transmissibility factor is entered in item 5, the calculation of the denominator term depends on whether the Kh value is set or defaulted. This quantity is required for calculating the connection transmissibility factor (if a default or zero value was supplied for item 5). It is ignored if not required.
- Skin Factor

- Large negative skin factors can cause problems if they increase the effective wellbore radius to approach the pressure equivalent radius of the grid block. Very high values of the connection transmissibility factor may result, which could cause convergence problems. In these circumstances, it may be preferable to change the grid block data near the well to model the physical cause of the negative skin (for example enhance the permeability out to a certain distance from the well). If the effective wellbore radius is increased beyond the pressure equivalent radius, the connection factor becomes negative and an error message is issued.

- Penetration Direction
  - X: the well penetrates the grid block in the X direction
  - Y: the well penetrates the grid block in the Y direction
  - Z: the well penetrates the grid block in the Z direction
- Pressure Equivalent Radius
- P Length
- Geo Fac
- W Frac
- Tubing Diameter
- Tubing Absolute Roughness

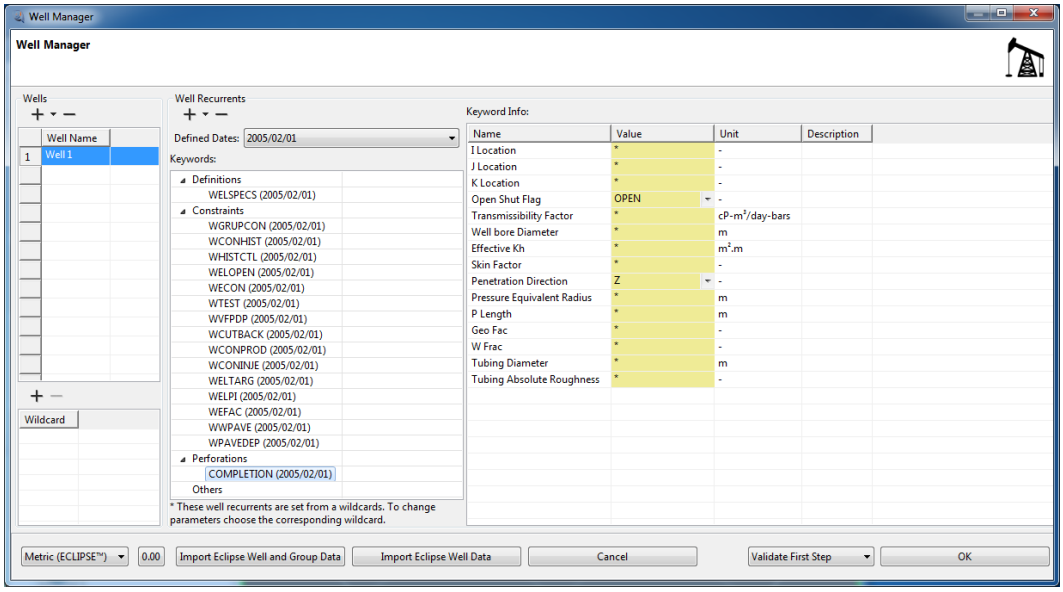


Figure 151 Completion Data

3-2- WPIMULT (Multiplies well connection factors by a given value)

This can be used to multiply the connection transmissibility factors of selected well connections by a specified value. To multiply the transmissibility factors of all the connections in a well, leave items 2-6 defaulted. To multiply the transmissibility factors of a subset of connections in a well, you can identify the subset by their I,J,K location (items 2-4). A subset of connections can also be identified by their completion number (items 5 and 6). If any of items 2-6 are set positive, the multiplying factor applies only to a subset of the connections in the well. The connections in the subset are those with I,J,K locations matching the specified location indices in items 2-4 and a completion number in the range specified by items 5-6. A zero or negative value for a location index (I, J or K) matches any value of that location index. Similarly a zero or negative value for a completion range lower/upper end-point indicates that there is no lower/upper bound on the completion numbers. Figure 152 displays the WPIMULT data. It contains the following data:

- Well Connection Factor Multiplier
  - Multiplier to act on the well's connection transmissibility factors and Kh values
- I-Location of Connecting Grid Block
- J-Location of Connecting Grid Block
- K-Location of Connecting Grid Block
- Index of First Completion
- Index of Last Completion

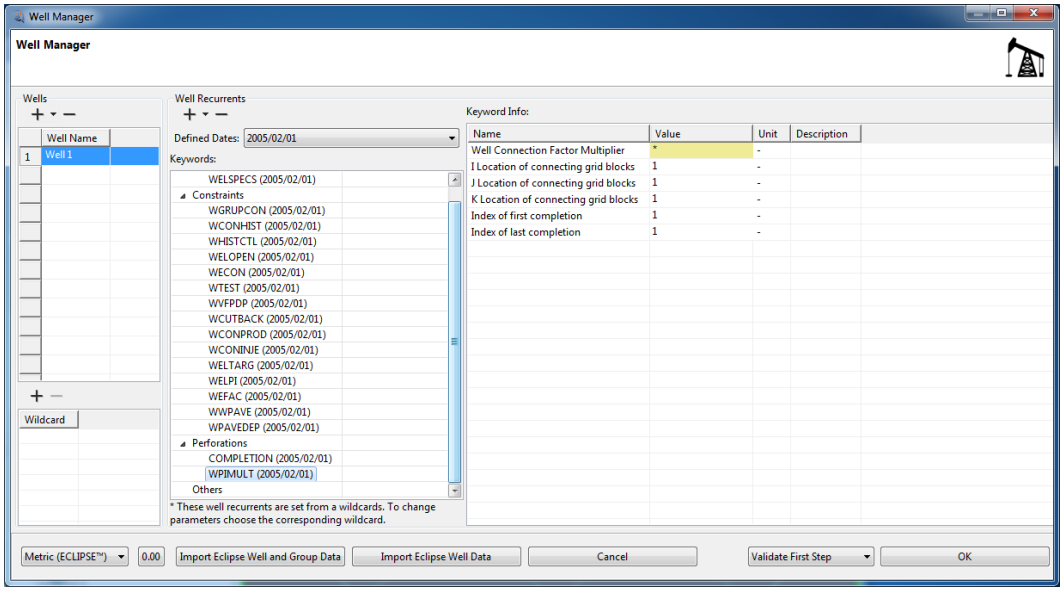


Figure 152 WPIMULT Data

4- *Others:*

4-1- COMPORD (Defines the ordering of well connections)

The COMPORD is used to define the ordering of well connections. Figure 153 displays the COMPORD data. It contains the completion ordering method in which three possibly are available:

- DEPTH: The connections will be ordered according to their vertical depth, from the top downwards. For connections at the same vertical depth, they will be ordered in the sequence they are first declared in Completion.
- INPUT: The connections will be ordered in the sequence that they are first declared in Completion. If you use this option, you must declare the connections in their correct sequence, starting with the connection nearest the wellhead then working along the wellbore towards the bottom or toe of the well.
- TRACK: RETINA will determine the order of the connections by attempting to trace the well track through the grid from the grid blocks in which the connections are located. If this fails, or if all the connections in the well are vertical, DEPTH ordering will be used.

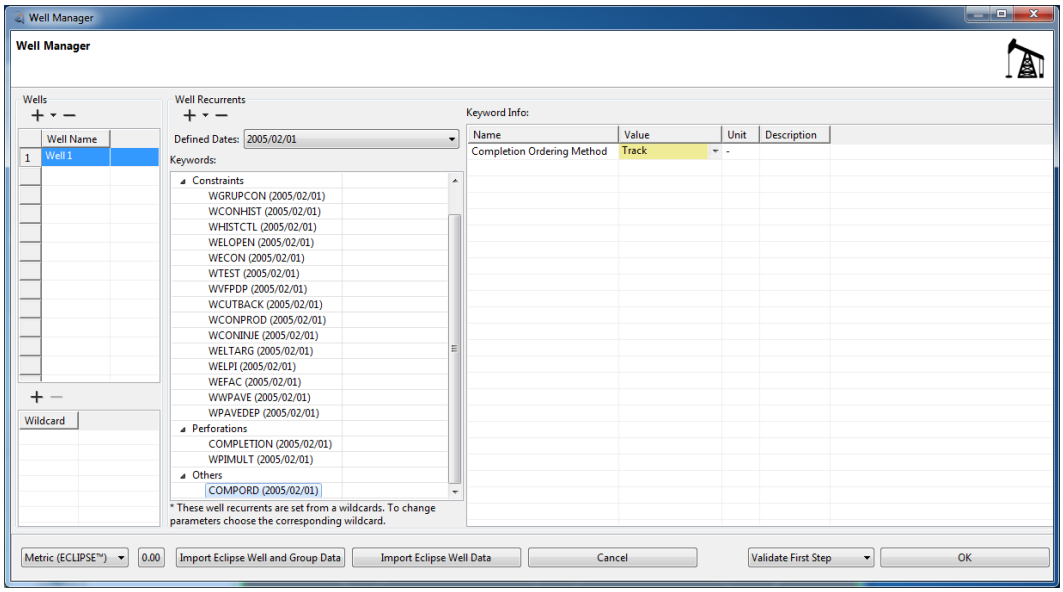


Figure 153 COMPORD Data

For defining a new well recurrent (after choosing a desired date) the + button must be used. Clicking on this button the Figure 86 list is displayed. User can select and define each of the well recurrent data with a user friendly method just using one click on them. User can define a specific well recurrent for all of the wells with a desired well base name, in the wildcard section of the Figure 154. The button of the + and - are used to define a wildcard and remove related properties, respectively.

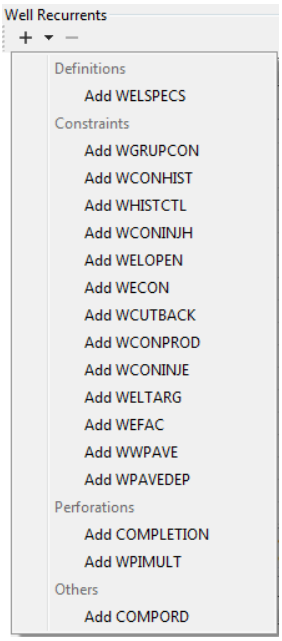


Figure 154- Adding Well Recurrents

Two importer buttons exist in this window:

- Import Eclipse Well and Group Data: Both well and group data are imported using this importer. In this case, you can see added groups in “Well Group Manager” and wells in “Well Manager” dialogs.
- Import Eclipse Well Data: Only wells and their properties are added, no change is made on the group manager window.

A validator is also placed in this window, in which user can choose one of three exist validation options for scheduling data.

3-3-3-Well Group Manager

The window of Figure 155 will be opened when selecting “Well Group Manager” button, from the data tab. User can start to define group from “Group Dates” section using + button. In the “Group Tree” section, the new group can be defined by selecting the + button (Figure 156). After defining desired groups, the wells might be attached to them using “Group Well Assignment” section (Figure 157).

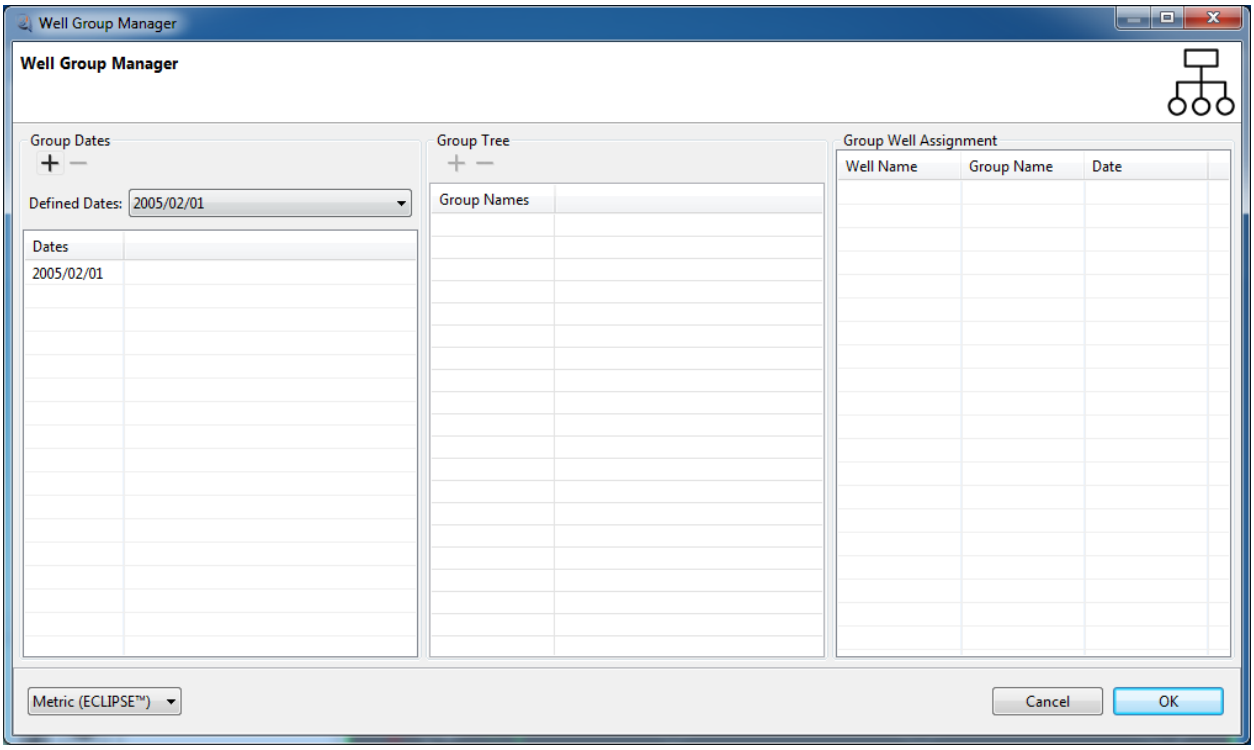


Figure 155- Well Group Manager

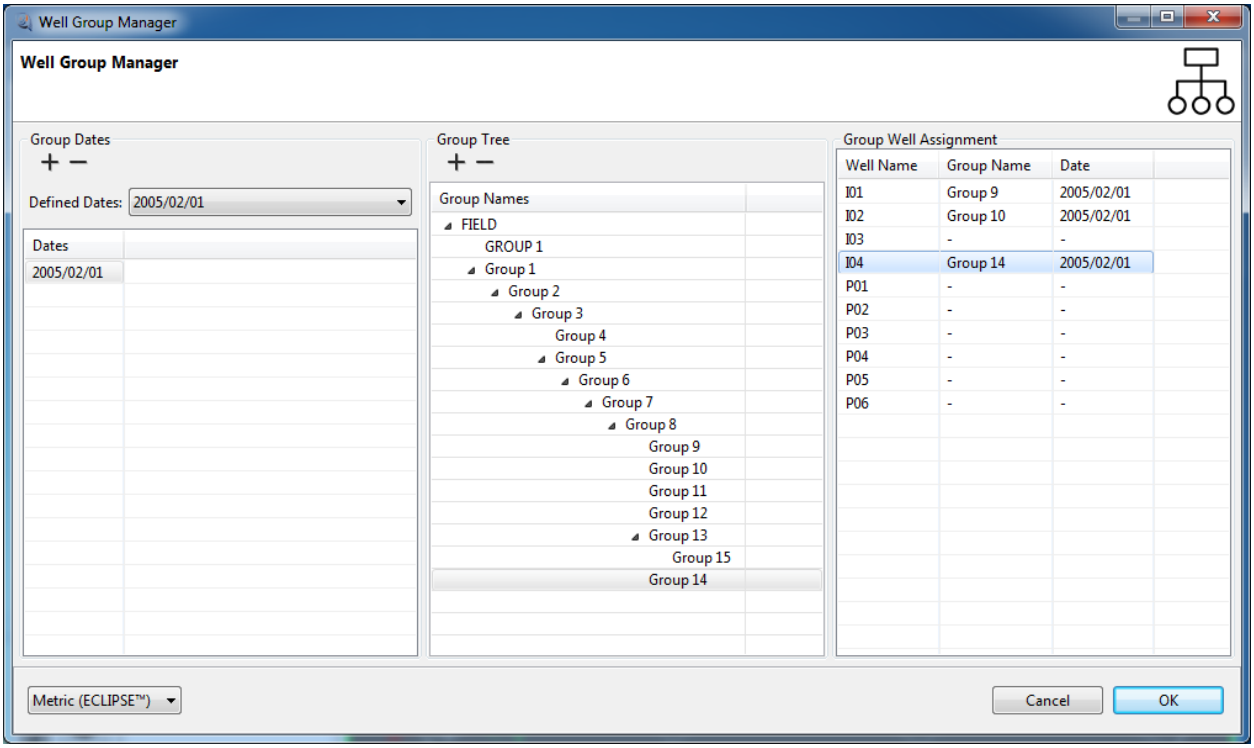


Figure 156- Define Group

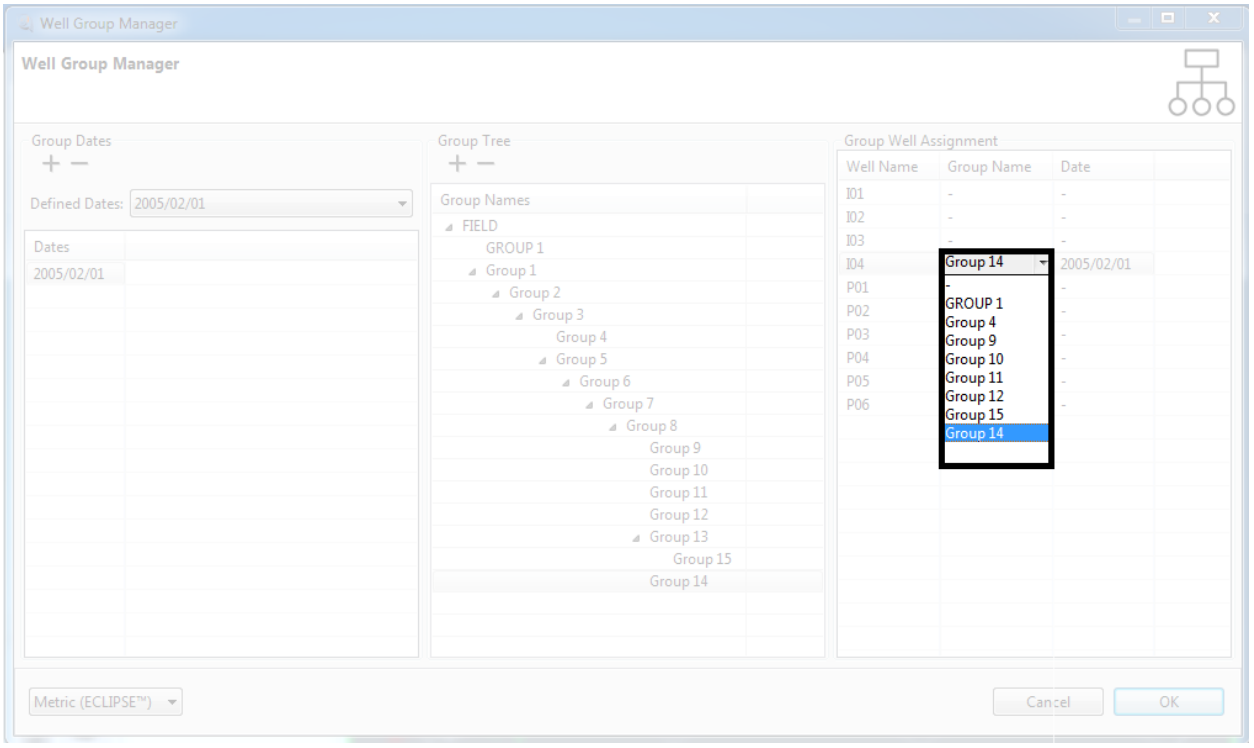


Figure 157- Well Assignment to Groups

3-3-4-Group Recurrent Manager

The button of the “Group Recurrent Manger” is used to manage group recurrent (Figure 158). This window consists of three tables of “Well Group List”, “Group Recurrent” and “Group constraint”. The desired group and date are selected from “Well Group List” and “Group Recurrent“, respectively. Afterward user can enter necessary constraints and provide their values (Figure 159).

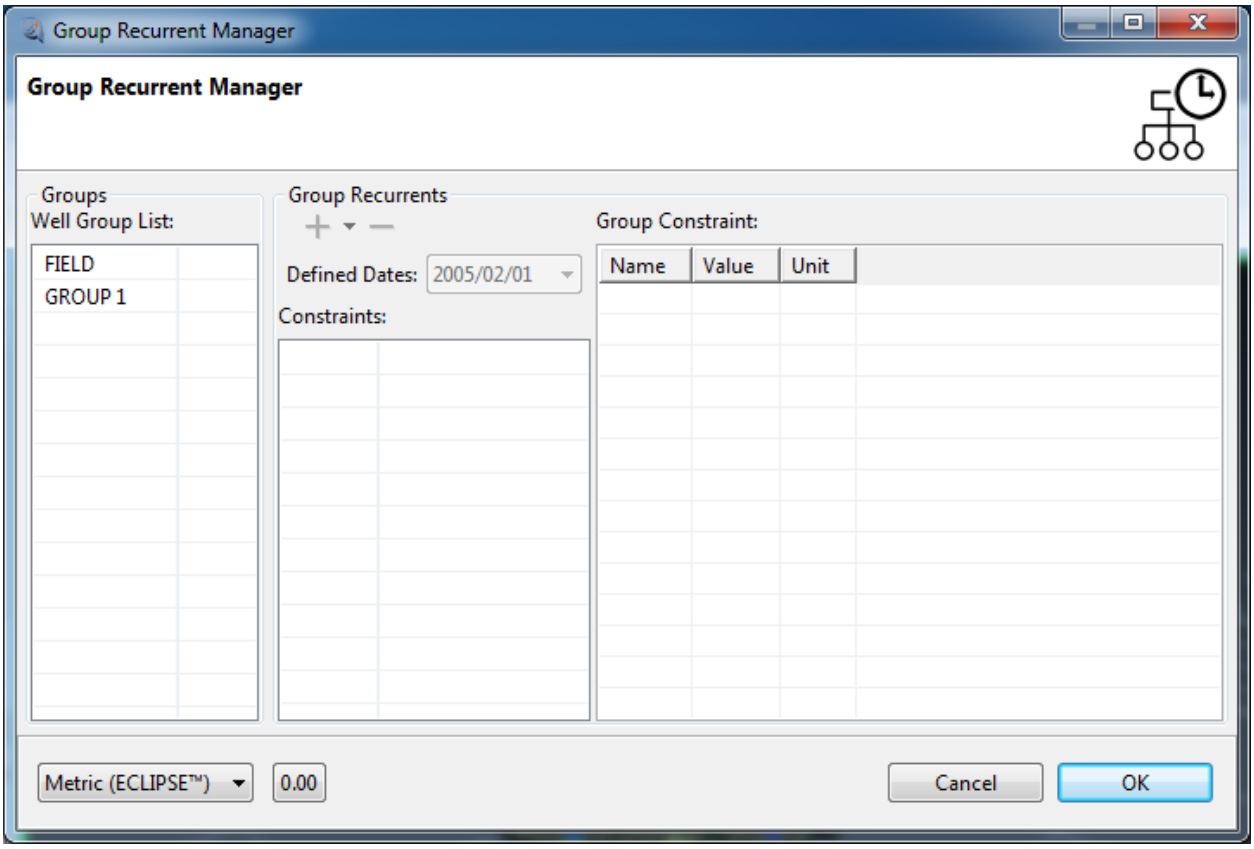


Figure 158- Group Recurrent Manager

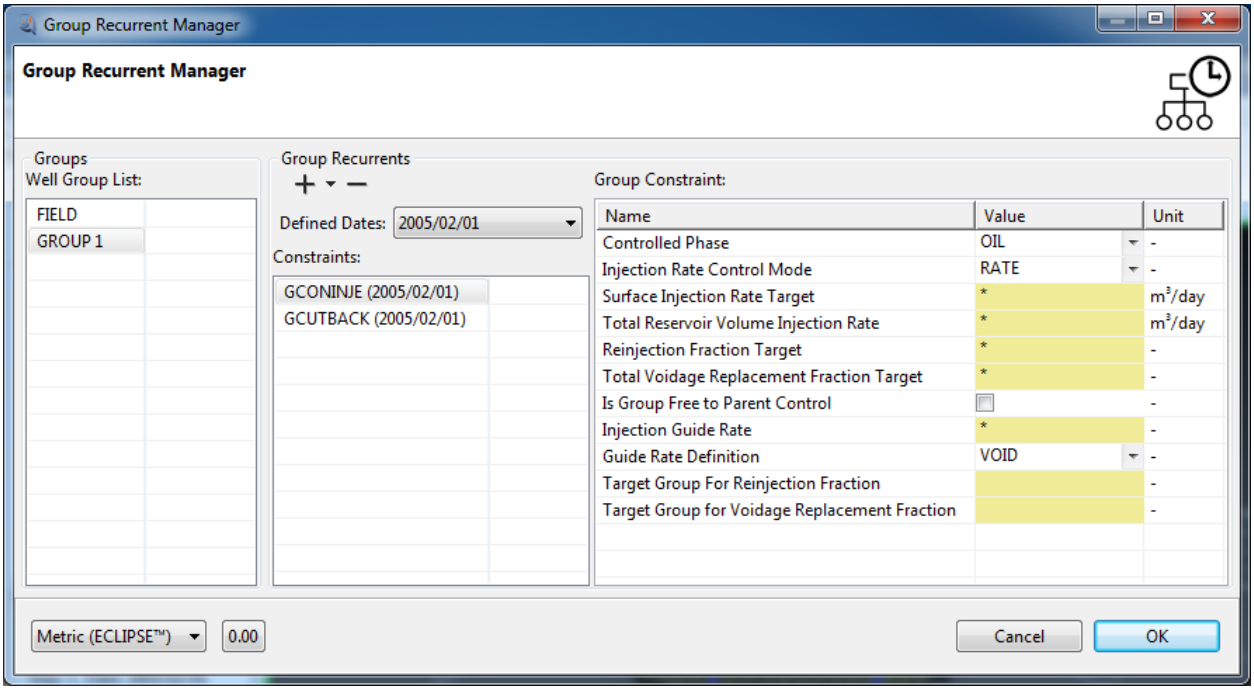


Figure 159- Group Constraints

Group Recurrents:

1- *GCONINJE (Injection rate controls/limits for groups/field)*

This specifies the injection targets and limits for groups. Figure 160 displays the GCONINJE data. It contains following data:

- Controlled Phase
  - WATER: Water injection controls
  - GAS: Gas injection controls
  - OIL: Oil injection controls
- Injection Rate Control Mode
  - NONE: No immediate control of injection rate
  - RATE: The group/field surface injection rate of the controlled phase
  - RESV: The group/field reservoir volume injection rate of the controlled phase will be controlled so that the total reservoir volume injection rate of the group/field meets the related target
  - REIN: The group/field surface injection rate of the controlled phase is controlled to equal the group/field production rate of the phase times the reinjection fraction

  - VREP: The group/field reservoir volume injection rate of the controlled phase is controlled so that the total reservoir volume injection rate of the group/field equals its production voidage rate times the voidage replacement fraction
  - FLD: The group is immediately under control from a higher level group or the field, injecting its share of the higher group's or field's target rate, according to its guide rate
- Surface Injection Rate Target
  - Surface injection rate target or upper limit for the controlled phase.
- Total Reservoir Volume Injection Rate
  - Total reservoir volume injection rate target or upper limit. If a non-default value is specified here, the controlled phase in item 1 is declared the top-up phase. Its target or limiting reservoir volume injection rate will be equal to the value specified here minus the reservoir volume injection rate of the other phases. The phase injection rate is calculated to top up the total group or field injection to the required reservoir volume rate, after allowing for any injection of the other phases. There can be only one top-up phase at any given time in the simulation run.
- Reinjection Fraction Target
  - Reinjection fraction target or upper limit for the controlled phase. The reinjection fraction is applied to the group or field production rate of the phase in item 1.
- Total Voidage Replacement Fraction Target
  - If a non-default value is specified here, the phase in item 1 is declared the 'top-up' phase. Its target or limiting reservoir volume injection rate will be equal to the value specified here times the group/field production voidage rate, minus the reservoir volume injection rate of the other phases. The phase injection rate is calculated to 'top up' the total group or field injection to the required voidage replacement fraction, after allowing for any injection of the other phases. There can be only one 'top up' phase at any given time in the simulation run.
- Is Group Free to Parent Control?

- Checked: Injection from the group is cut back as necessary to meet any rate target/limit imposed at a higher level
- Unchecked: The group injects at its own capacity or target irrespective of any higher level rate target or limit

- Injection Guide Rate
  - Group's injection guide rate for the controlled phase, a dimensionless number governing the group's share of a higher level injection target.

    A group needs a guide rate only if it is required to inject a specified proportion of a higher level rate target. The group is then placed under rate control, with a target rate set in proportion to its guide rate. It can either be set to a fixed surface rate or reservoir volume rate, or adjusted each time step to reflect the group's voidage replacement requirements. If no guide rate is specified, the group's share of a higher level injection target is governed by the guide rates of its subordinate injection wells, or any subordinate groups with injection guide rates.

    The group will essentially be transparent to higher level injection targets, which will be apportioned directly down to its subordinate wells or groups with guide rates.
- Guide Rate Definition
  - RATE: The guide rate applies to group's surface injection rate (so the surface injection rate target will be in proportion to the guide rate).
  - RESV: The guide rate applies to group's reservoir volume injection rate (so the reservoir volume injection target is in proportion to the guide rate).
  - VOID: The guide rate is set at the beginning of each time step equal to the group's voidage production rate. This setting can be used to apportion an injection target between groups in proportion to their voidage production rates.
  - NETV: The guide rate is set at the beginning of each time step equal to the group's net voidage rate (that is its voidage production rate minus the reservoir volume injection rate of any other phases). This setting can be used to apportion an injection target between groups in proportion to their top-up needs for voidage replacement. It can

only be applied if the controlled phase in item 1 is the 'top-up' phase.

- NONE: the group has a injection guide rate for this phase.

- Target Group for Reinjection Fraction
- Target Group for Voidage Replacement Fraction

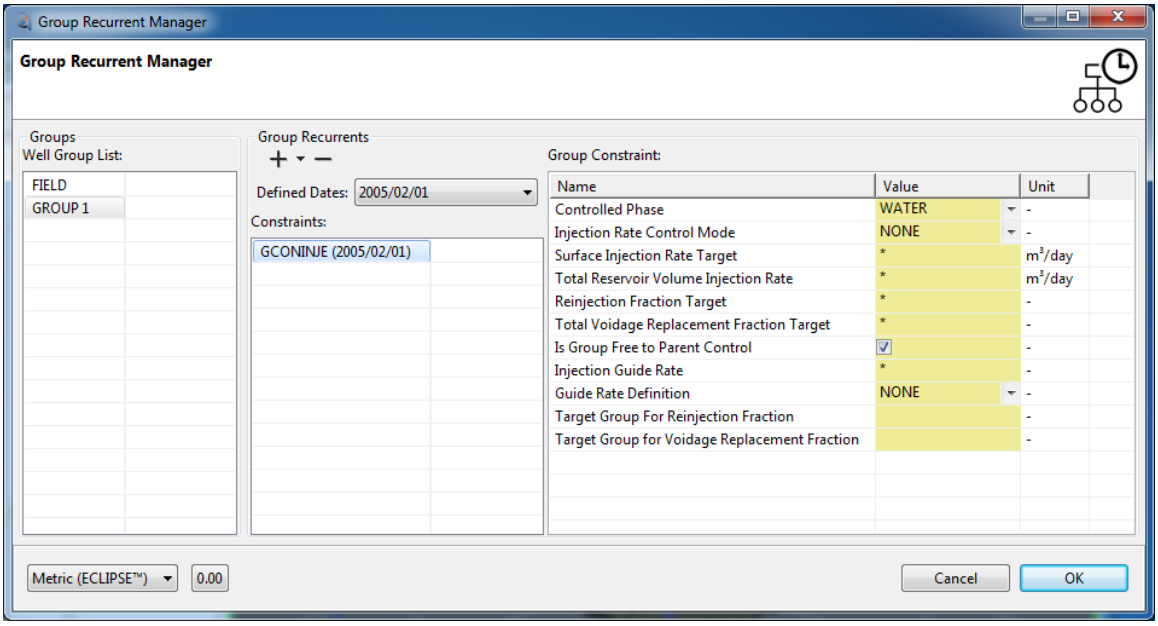


Figure 160 GCONINJE Data

2- *GCONPROD (Production rate controls/limits for groups/field)*

This sets the production rate targets and limits for groups using the guide rate method of group control. Figure 161 displays the GCONPROD data. It contains following data:

- Production Rate Control Mode
  - NONE: No immediate control of production rate
  - ORAT: Group/field oil production rate is controlled to meet the target
  - WRAT: Group/field water production rate is controlled to meet the target
  - GRAT: Group/field gas production rate is controlled to meet the target

  - LRAT: Group/field liquid production rate is controlled to meet the target
  - RESV: Group/field reservoir fluid volume production rate is controlled to meet the target
  - PRBL: Group/field reservoir fluid volume production rate is controlled to meet the target production balancing fraction
  - FLD: Group is immediately under control from a higher level group or the field, producing its share of the higher group's or field's target rate, according to its guide rate
- Oil Production Rate Target
- Water Production Rate Target
- Gas Production Rate Target
- Liquid Production Rate Target
- Procedure on Exceeding a Limit
  - NONE: Do nothing
  - CON: Shut worst-offending connection in worst-offending well
  - PLUS_CON: Shut worst-offending connection and all below it in worst-offending well (see COMPORD)
  - WELL: Shut or stop worst-offending well
  - RATE: Control group/field production rate to equal the violated upper limit
- Is Group Free to Higher Control?
  - Checked: production from the group is cut back as necessary to meet any rate target imposed at a higher level
  - Unchecked: the group produces at its own capacity or target irrespective of any higher level rate target
- Group's Production Guide Rate
  - This is a dimensionless number governing the group's share of a higher level production rate target. A group needs a guide rate only if it is required to produce a specified proportion of a higher level rate target. The group is then placed under rate control, with a target rate set in proportion to its guide rate. It can either be set for a specified phase, or calculated from the group's production potentials or its injection rate. If the guide rate phase differs from

the phase under control, the guide rate is translated into a guide rate for the controlled phase using the group's production ratios at the beginning of each time step. If no guide rate is specified, the group's share of a higher level rate target is governed by the guide rates of its subordinate wells, or any subordinate groups with guide rates. (The well guide rates are set by default equal to the wells' production potentials, but can be specified directly by using the WGRUPCON constraint). The group is essentially 'transparent' to higher level production rate targets, which will be apportioned directly down to its subordinate wells or groups with guide rates.

- Definition of Group's Guide Rate
  - OIL: The group's guide rate applies to the oil phase.
  - WAT: The group's guide rate applies to the water phase.
  - GAS: The group's guide rate applies to the gas phase.
  - LIQ: The group's guide rate applies to the liquid phase.
  - RESV: The group's guide rate applies to the Reservoir Fluid.
  - INJV The group's guide rate is set at the beginning of each time step equal to the group's reservoir volume injection rate. This can be useful `when the field has a production balancing target that is apportioned among the groups in proportion to their injection rates.
  - POTN: The group's guide rate is set at the beginning of each time step equal to the group's production potential - which is the sum of the production potentials of all its subordinate open producers;
  - FORM: The group's guide rate is calculated at specified intervals from a formula involving its production potentials, defined with the GUIDERAT. This option can be used to decrease the contribution from groups with high GOR or water cut.
  - NONE: the group has a production guide rate. Higher level production targets will be apportioned directly down to its subordinate wells, or any subordinate groups with guide rates.
- Procedure on Exceeding Water Rate
  - NONE: Do nothing
  - CON: Shut worst-offending connection in worst-offending well
  - PLUS_CON: Shut worst-offending connection and all below it in worst-offending well (see COMPORD)

  - WELL: Shut or stop worst-offending well
  - RATE: Control group/field production rate to equal the violated upper limit
- Procedure on Exceeding Gas Rate
  - NONE: Do nothing
  - CON: Shut worst-offending connection in worst-offending well
  - PLUS_CON: Shut worst-offending connection and all below it in worst-offending well (see COMPORD)
  - WELL: Shut or stop worst-offending well
  - RATE: Control group/field production rate to equal the violated upper limit
- Procedure on Exceeding Liquid Rate
  - NONE: Do nothing
  - CON: Shut worst-offending connection in worst-offending well
  - PLUS_CON: Shut worst-offending connection and all below it in worst-offending well (see COMPORD)
  - WELL: Shut or stop worst-offending well
  - RATE: Control group/field production rate to equal the violated upper limit
- Reservoir Fluid Volume Production Rate Target
  - The procedure on exceeding this limit is always RATE.
- Production Balancing Fraction Target
  - This limits the group's reservoir fluid volume production rate to the specified fraction (or multiple) of the group's reservoir fluid volume injection rate.

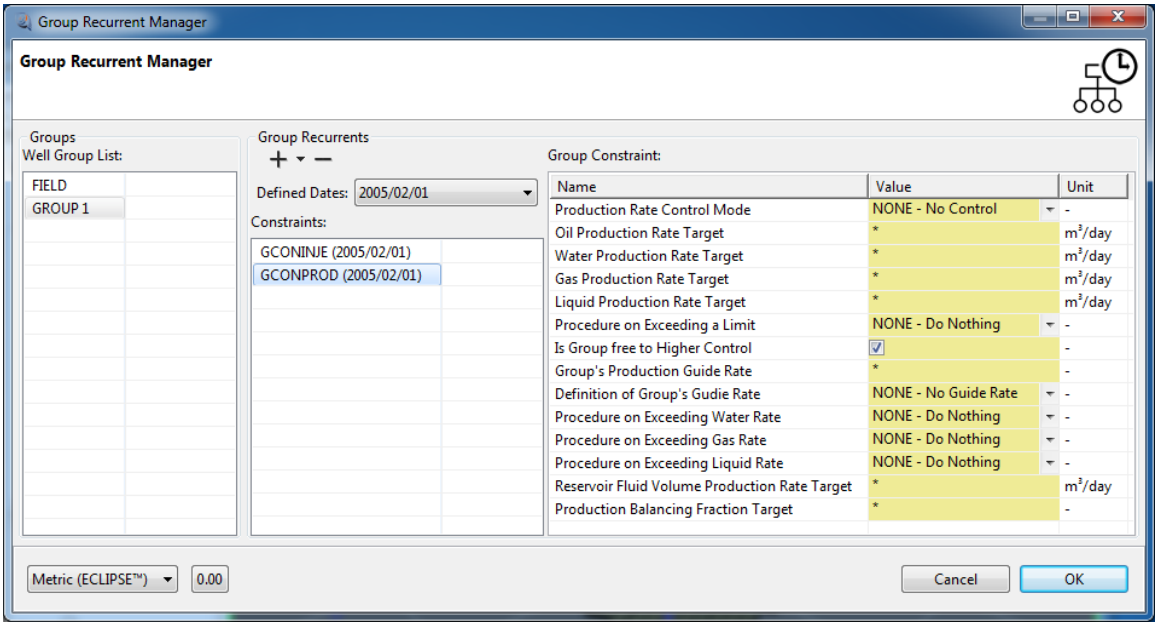


Figure 161 GCONPROD Data

3- *GECON (Economic limit data for groups and the field)*

This is used to put an economic limit data for groups and field. Figure 162 displays the GECON data. It contains following data:

- Minimum Oil Production Rate
  - If the group (or field) oil production rate falls below the specified minimum, all the producers in the group (or field) are shut or stopped (see WELSPECS). A zero or negative value switches off this constraint.
- Minimum Gas Production Rate
  - If the group (or field) gas production rate falls below the specified minimum, all the producers in the group (or field) are shut or stopped (see WELSPECS). A zero or negative value switches off this constraint.
- Maximum Group or Field Water Cut
- Maximum Group or Field Gas Oil Ratio
- Maximum Group or Field Water Gas Ratio
- Workover Procedure
  - NONE: Do nothing
  - CON: Shut worst-offending connection in worst-offending well
  - PLUS_CON: Shut worst-offending connection and all below it in worst-offending well

- WELL: Shut or stop worst-offending well (see WELSPECS)
- ALL: Shut all producers subordinate to the group

- Maximum Open Well
  - This option prevents wells from being opened automatically (AUTO in WCONPROD, WCONINJE, and WELOPEN) or from the drilling queue (QDRILL). if the number of subordinate wells would violate this limit. The limit does not prevent wells being opened manually ('OPEN' in WCONPROD and WCONINJE for example), but a warning message is issued if they violate this limit.

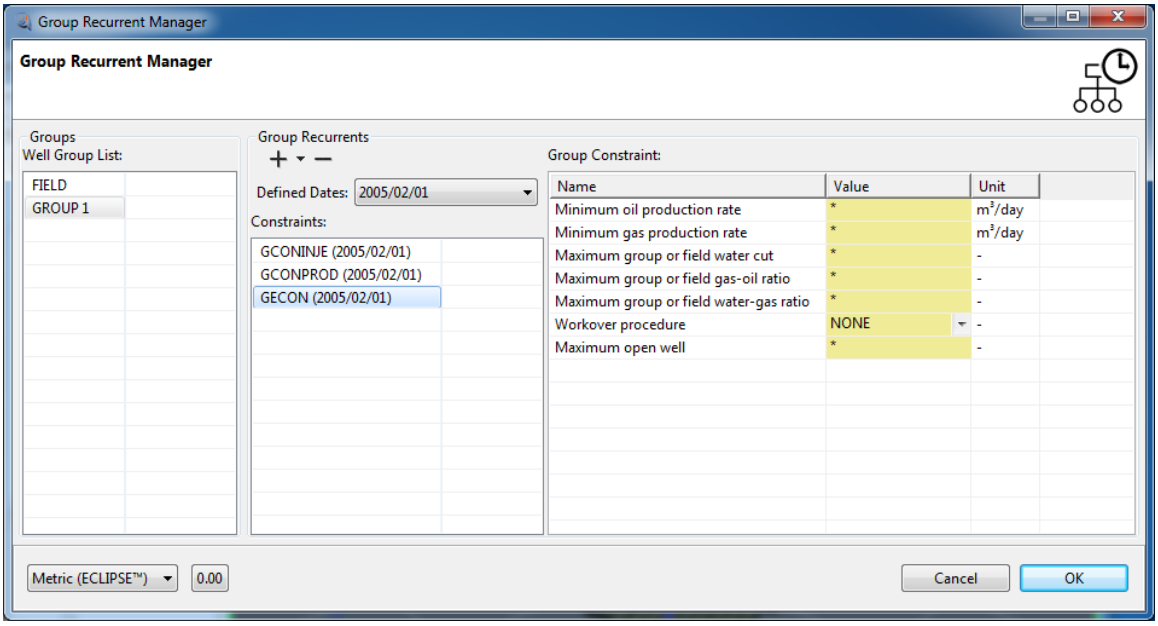


Figure 162 GECON Data

4- *GCUTBACK (Group Cutback Limit)*

This is used to put a cutback limit for group or filed. Figure 163 displays the GCUTBACK data. It contains following data:

- Water Cut Upper Limit
- Gas Oil Ratio Upper Limit
- Gas Liquid Ratio Upper Limit
- Water Gas Ratio Upper Limit
- Rate Cutback Factor (fraction)
  - This is the ratio of the reduced flow rate to the current flow rate. It should be less than 1.0

- Control Phase
  - OIL: Reduced oil rate
  - WAT: Reduced water rate
  - GAS: Reduced gas rate
  - LIQ: Reduced liquid rate
  - RESV: Reduced reservoir fluid volume rate

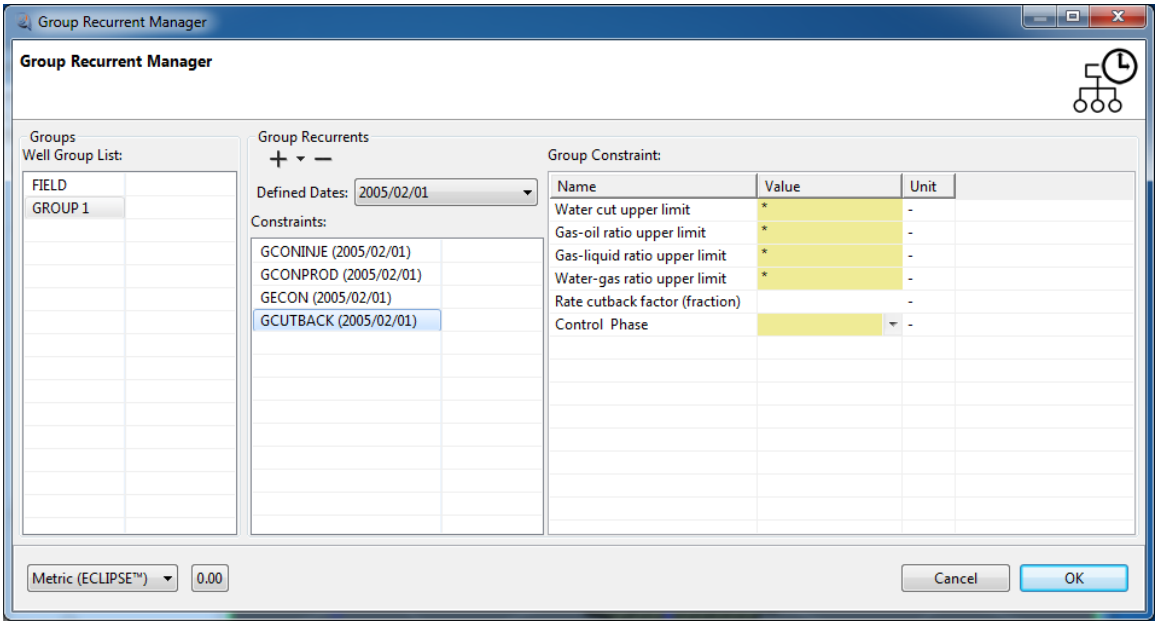


Figure 163 GCUTBACK Data

3-3-5-VFP Manager

VFP (Vertical Flow Performance) manager is available by clicking "Vertical Flow Performance Manager" button, in which user can define VFP tables. The data can be entered by user, one by one or using Eclipse importer facility.

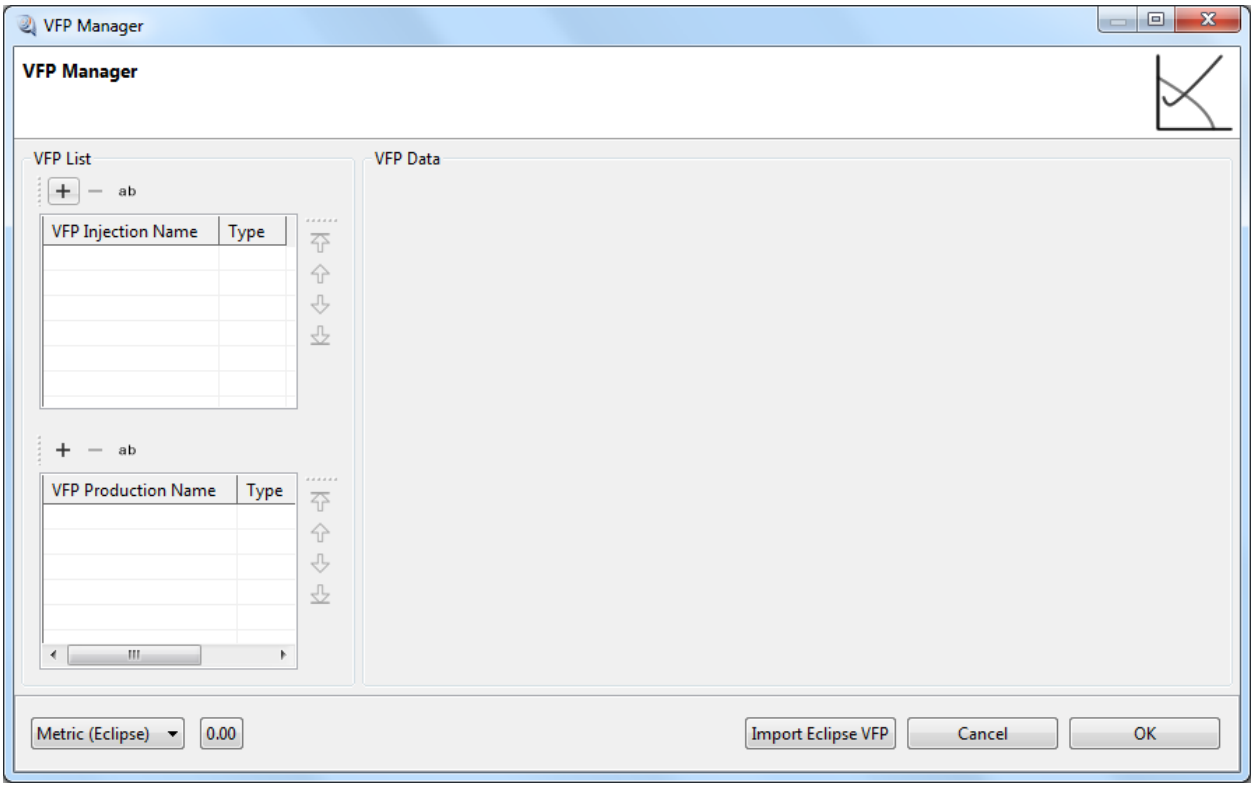


Figure 164- VFP Manager

There are two separate tables of production and injection wells in the VFP table dialog (Figure 164).

For adding a new injection VFP tables (Figure 165) use + button same as RETINA other windows, note that, the auto naming ability exist for this window, also. User must choose the fluid type from “Flow Rate”, pressure type from “Fixed Pressure” and tabulated parameter from “Tabulated Quantity” lists. Then the depth of the well and table values are entered. After entering the required values and updating table, the data VFP tables are created (Figure 166); afterward, user must enter the tabulated BHP values. Note that there are copy and paste facility for the VFP tables. The VFP table defining procedure for production wells are same as the injection one, except in input data (Figure 167).

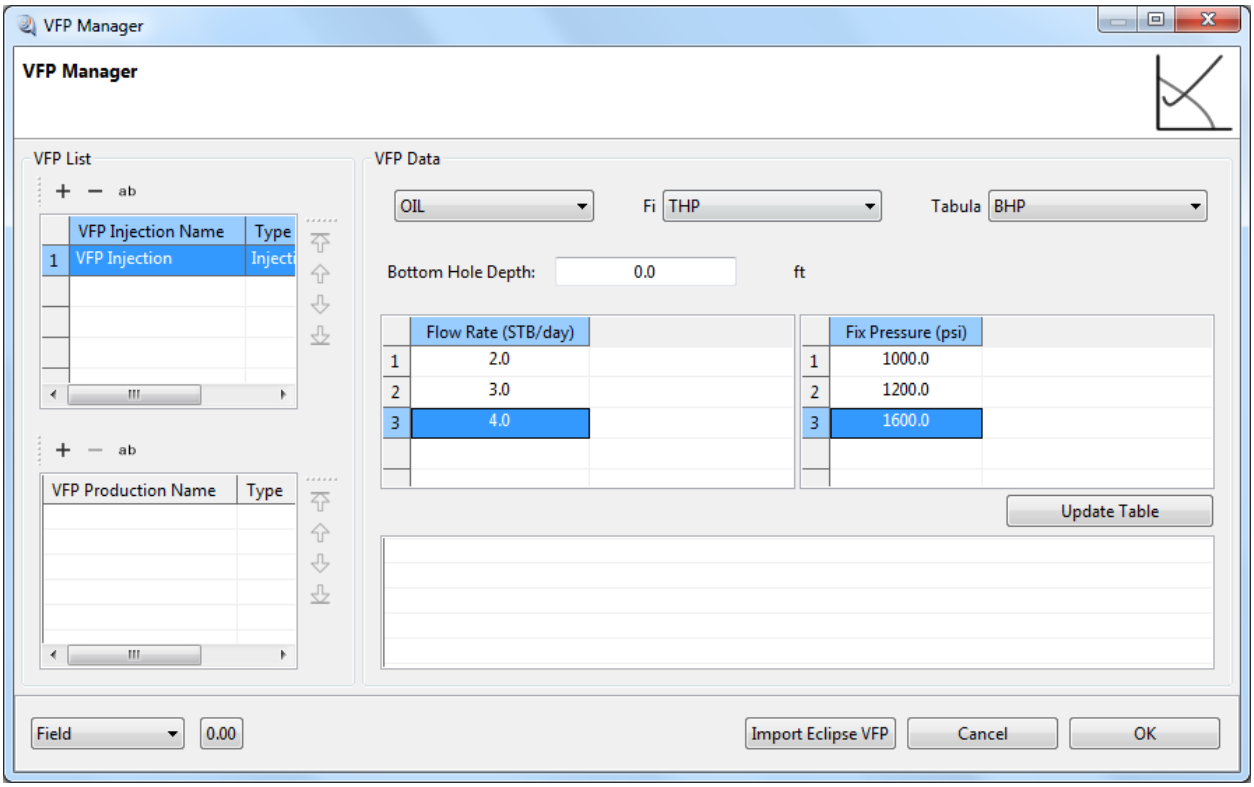


Figure 165- Define VFP Tables for Injection well

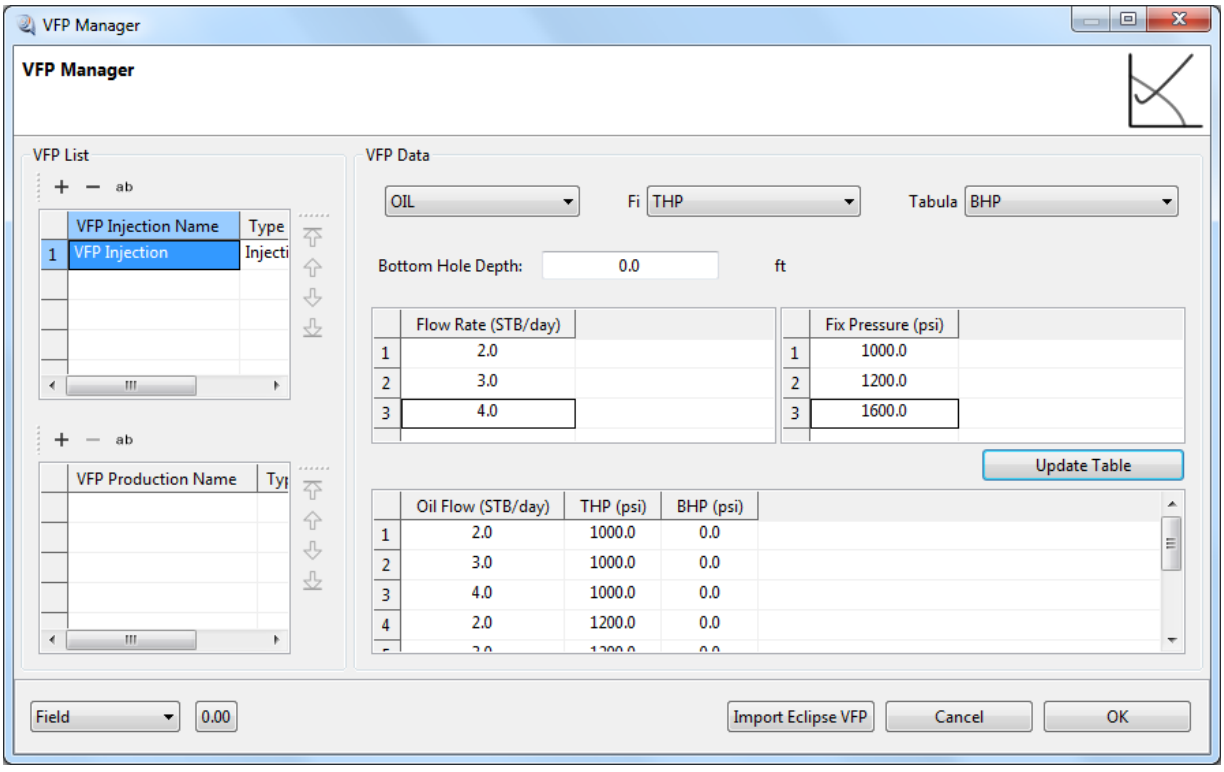


Figure 166- Injection Well VFP Table

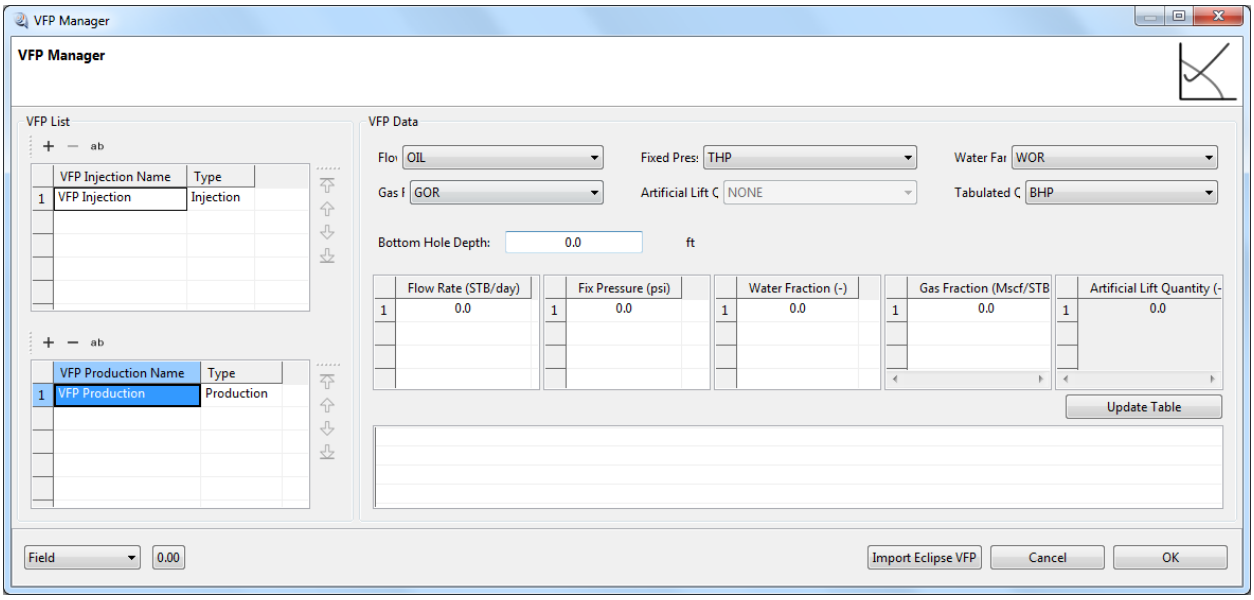


Figure 167- Production Well VFP Table

3-3-6-Well Recurrent Manager

Clicking on the "Well Recurrent Manager", user can observe well recurrents (Figure 168). There are three choices for observing the well recurrent using button:

1- Sort by Well
2- Sort by Well Recurrent
3- Sort by Date (Time)

All of the recurrents will be displayed if user chooses . If there is no need to observe well recurrents the button of should be selected. For example if user selects "Sort by Well" and then clicks on , the Figure of 98 is shown. It should be noted that, the data in this section are all read only.

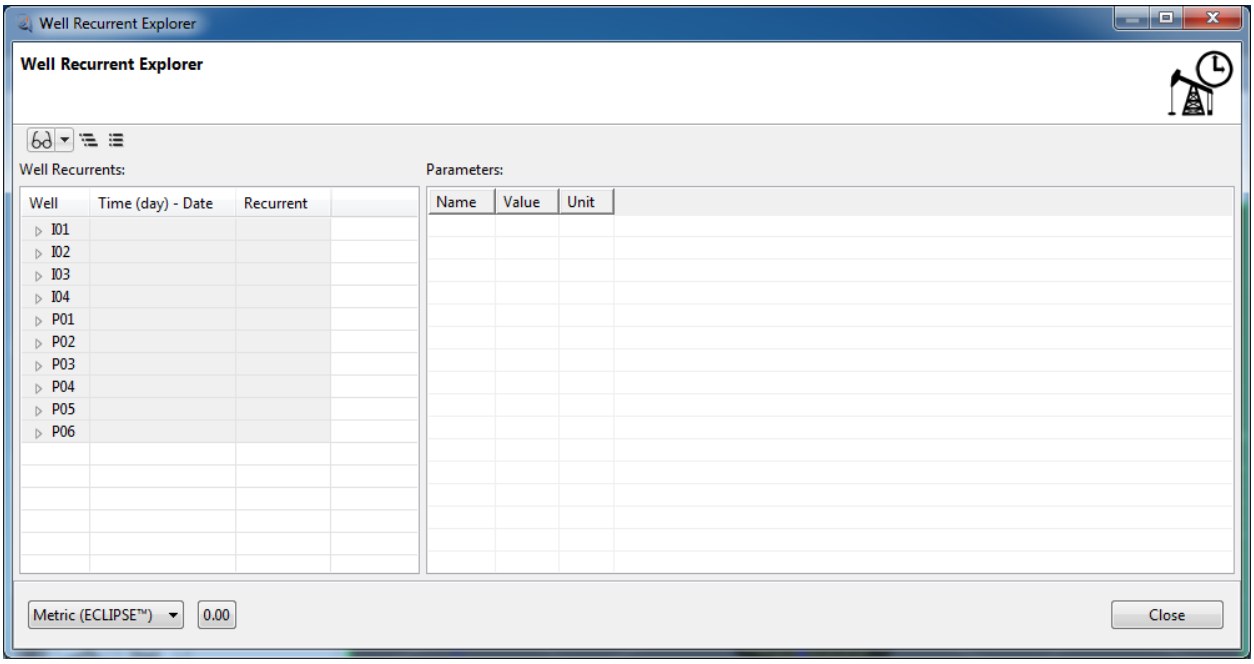


Figure 168- Well Recurrent Manager

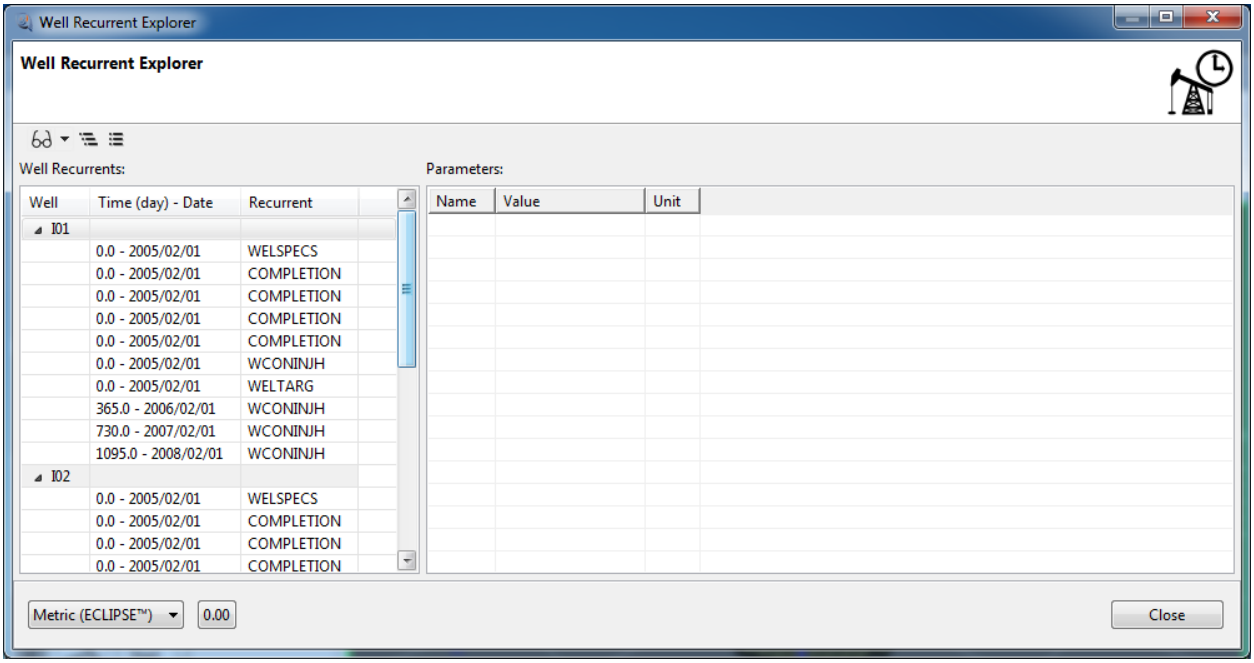


Figure 169- Displaying Options of Well Recurrents

3-3-7-Reservoir Recurrent Manager

The Reservoir recurrent manager window which has been shown in Figure 170 includes following recurrents:

- DRSDT

- MULTFLT
- TUNING
- MULTREGT

Like other windows of RETINA, the + and – buttons are used to define and remove defined recurrents. The recurrents can be defined in different defined dates, which can be selected from "Defined Dates" list; in such cases the date added to recurrent nod as a child (Figure 171).

The DRSDT controls the rate at which the solution gas-oil ratio is allowed to rise; two parameters of "Gas Resolution Maximum Rate" and "Option Flag" should be entered (Figure 171). The first one represents the maximum rate at which the solution gas-oil ratio in any grid block is allowed to increase. The second item is a flag including FREE and ALL options:

- "ALL": Apply the DRSDT limit to all cells
- "FREE": Apply the DRSDT limit only to cells containing free gas

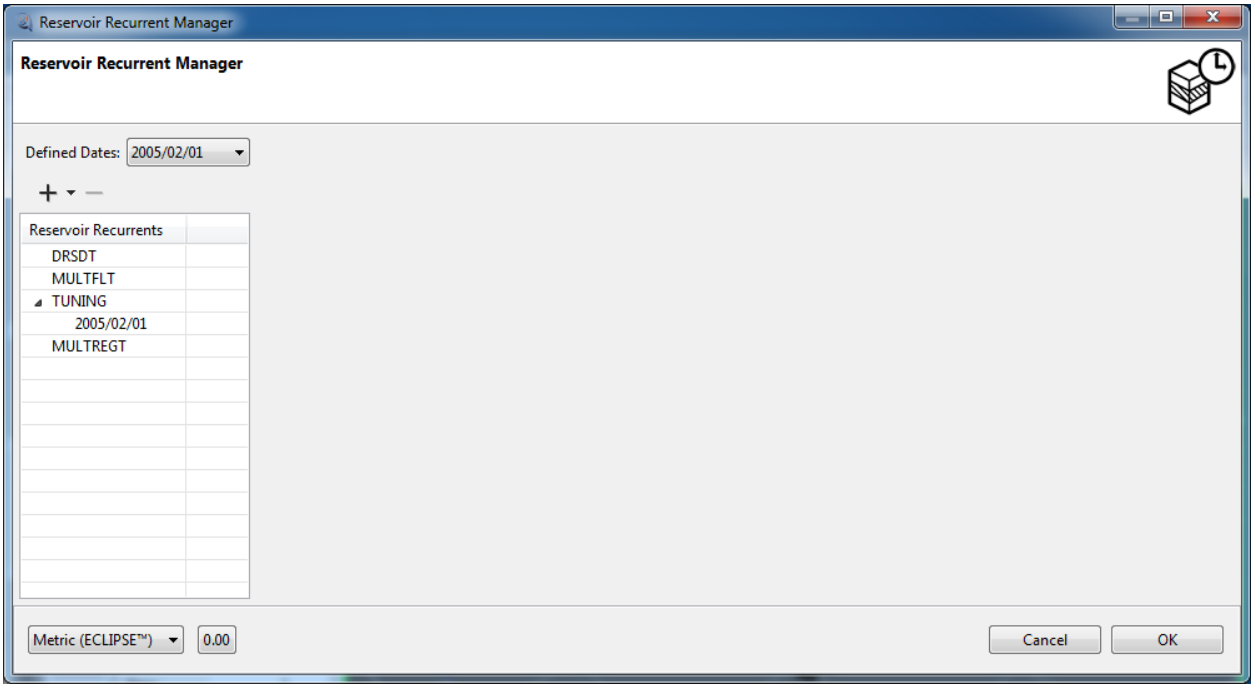


Figure 170- Reservoir Recurrent Manager

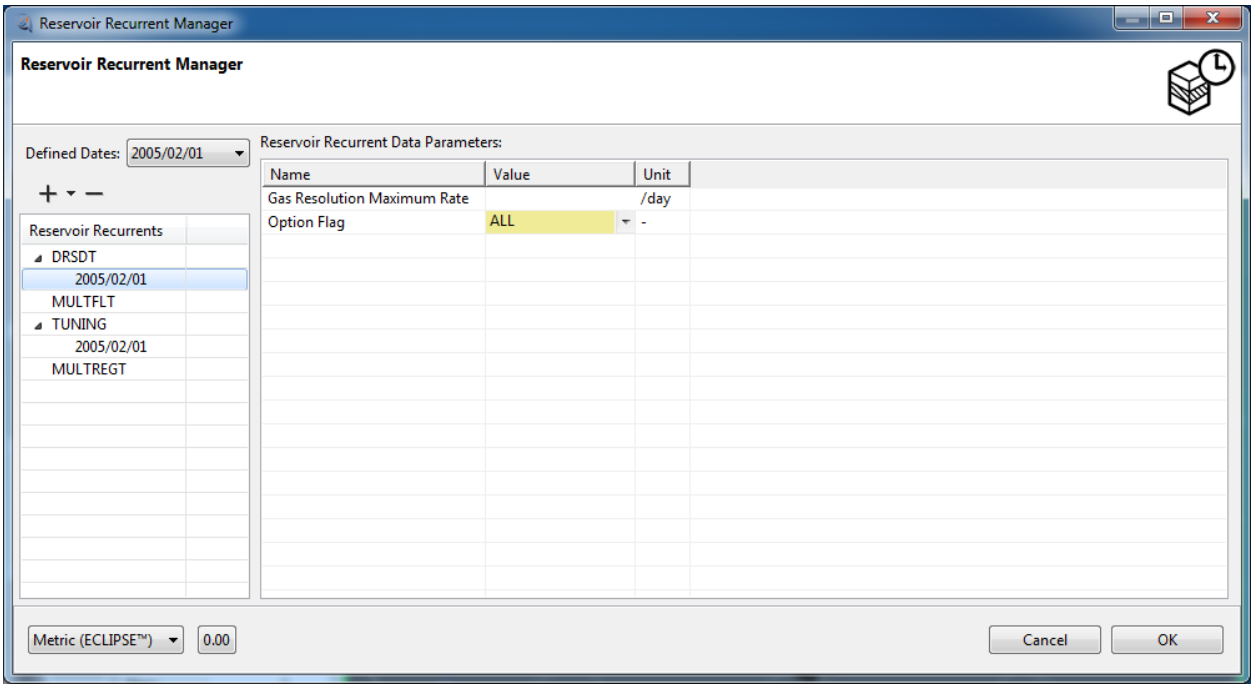


Figure 171- Data of DRSDT

The MULTFLT can be used to modify the transmissibility across a fault using either the FAULTS keyword or "Fault Manager" window. According to Figure 172 the essential data for MULTFLT are fault name and transmissibility multiplier.

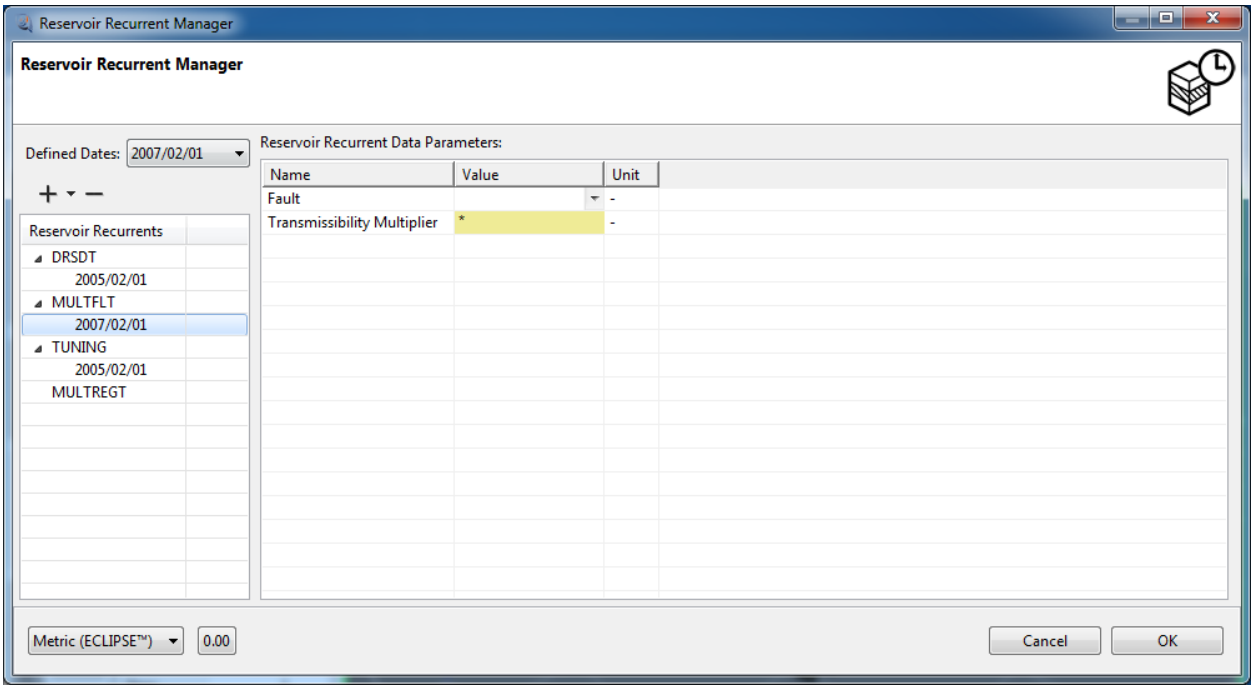


Figure 172- Data of MULTFLT

The TUNING is used to set simulator control parameters which are displayed in the Figure 173. The core number management for parallel simulating is available here. This dialog should be used with great care.

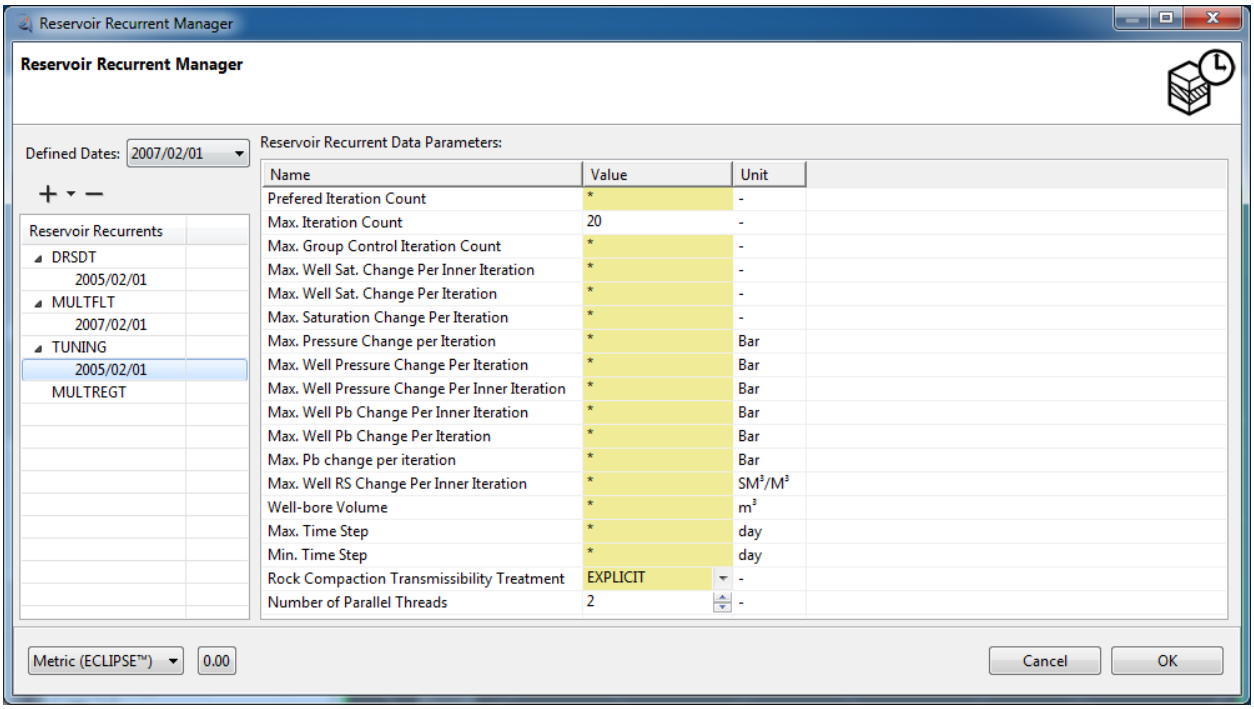


Figure 173- Data of TUNING

The MULTREGT can be used to set a transmissibility multiplier between regions. The regions must previously have been defined in the GRID properties section. Following four inputs are necessary to define a MULTREGT keyword (Figure 174):

- From Region Number
- To Region Number
- Transmissibility multiplier for all transmissibilities connecting regions I and J
- Direction(s) in which to apply the multiplier

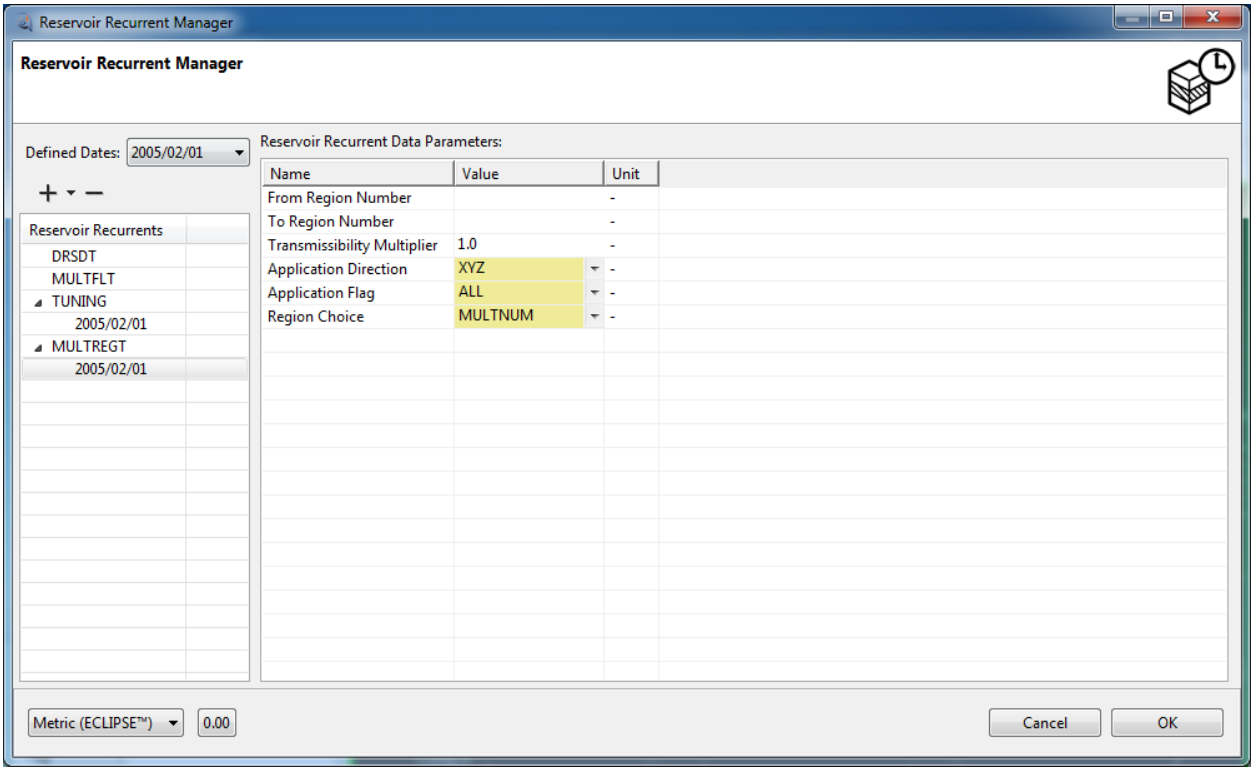


Figure 174- Data of MULTREGT

3-3-8-Drilling Management

The "Drilling Management" is used to manage auto drilling operation. Following recurrents are available to define drilling which has been displayed in Figure 175:

- QUEUE
- DRILPRI
- WELSOMIN
- PRORDER
- WDRILTIM
- WORKLIM

There is a filtering facility in this window which user can use it to filter recurrents either by type or by date.

For defining QUEUE, the Queue type which can be one of the sequential or priority options, must be selected. The arrows are used to move selected wells to queue or vice versa.

Note: the priority value is not defined for sequential queue type.

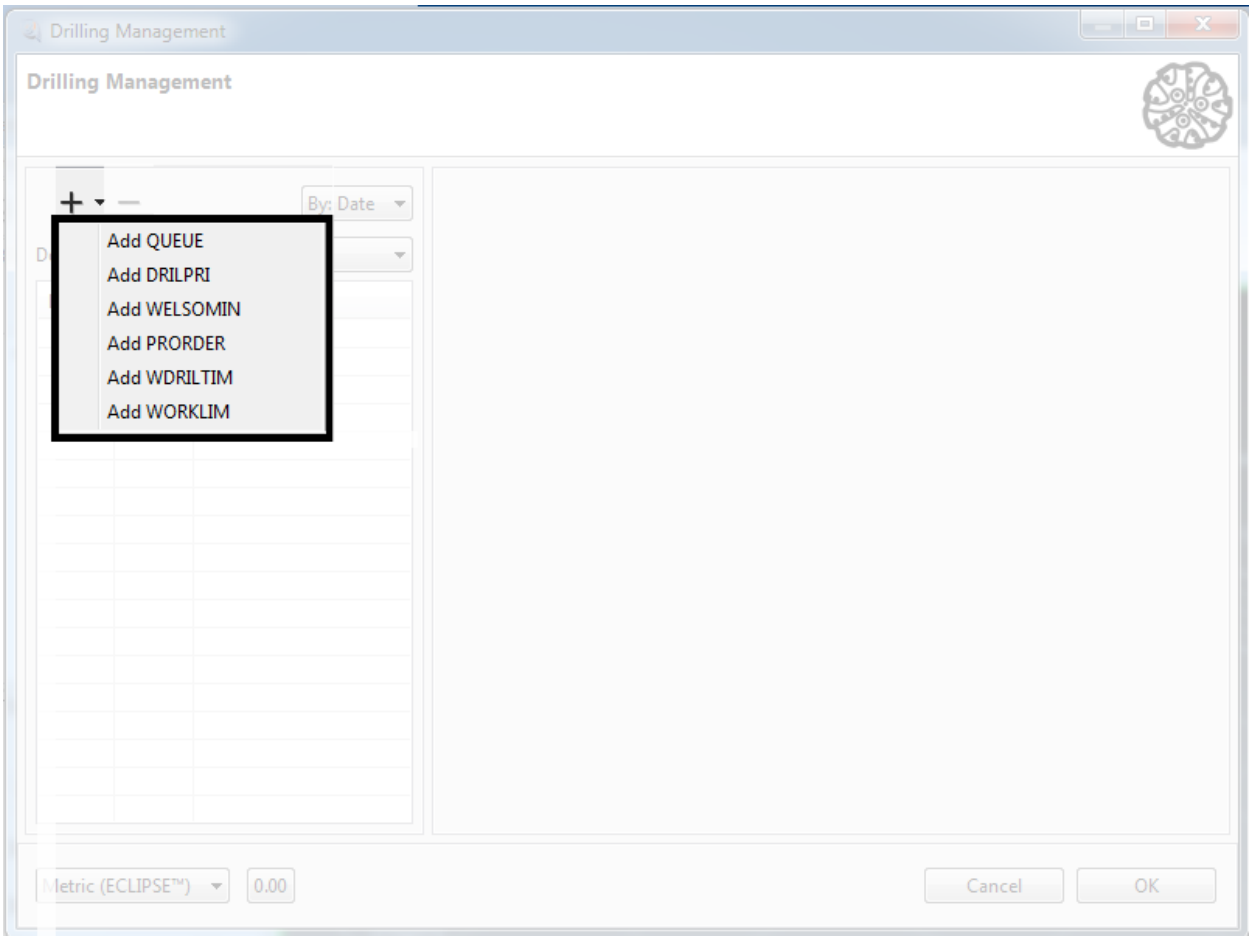


Figure 175- Drilling Management

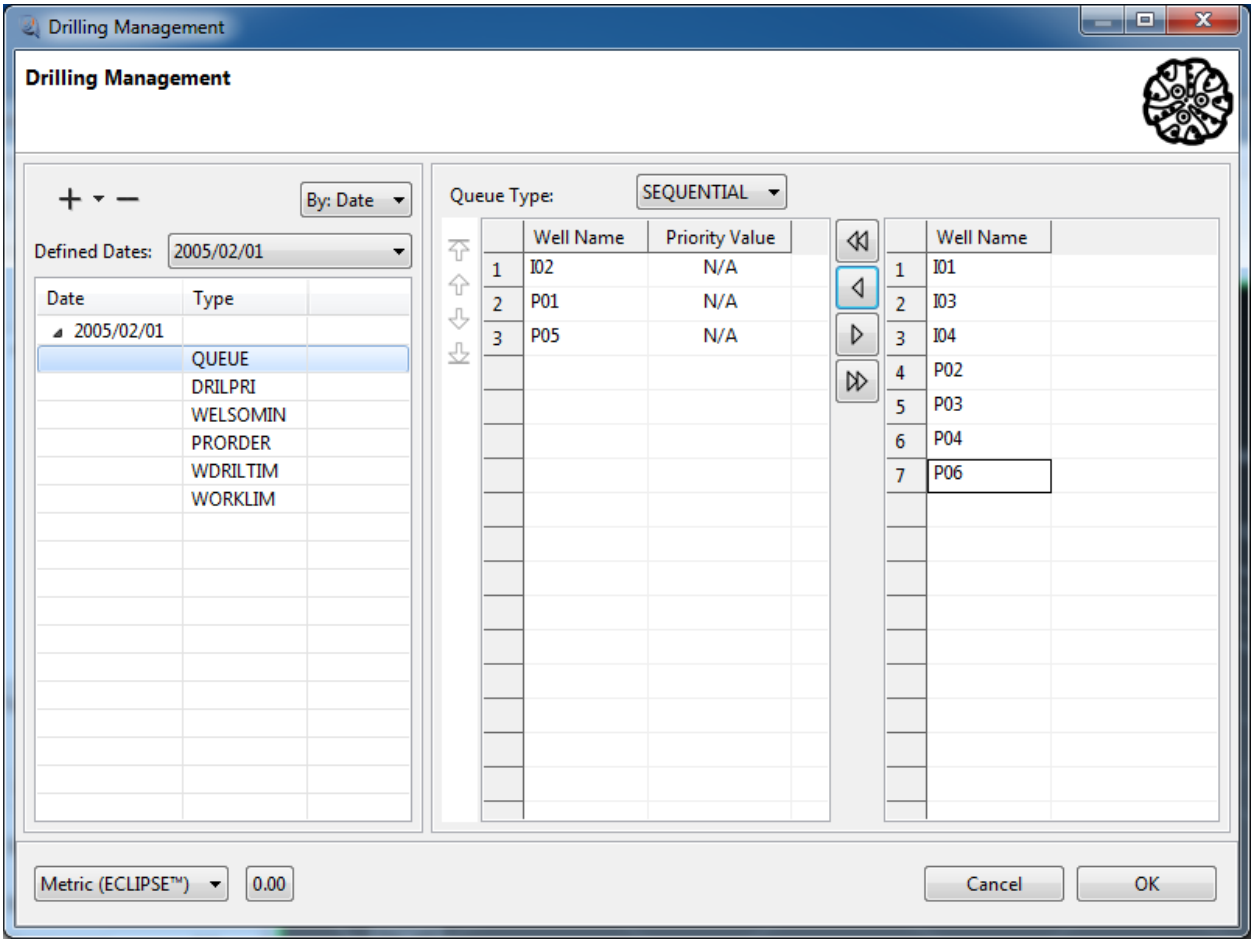


Figure 176- Moving well to Drilling Queue

The DRILPRI is used to set the coefficients that define the default priority formula for the prioritized drilling queue. It must be used if any wells are placed in this queue with the WDRILPRI keyword without a fixed priority set in WDRILPRI.

The minimum oil saturation in a grid block necessary to allow a well connection to be opened automatically is entered in WELSOMIN.

The PRORDER keyword can optionally be used to define a specific sequence of actions to be performed if a group fails to meet its production rate target.

The WDRILTIM controls the automatic drilling of new wells. The well name and time taken to drill the well are two inputs of this recurrent.

The time taken for each automatic workover is defined in WORKLIM keyword.

3-3-9-Rig Management

The Rig Management window which is displayed in Figure 177 and 178 includes two tabs of “Drilling Rig Assignments” and “Workover Rig Assignment”. The + and – buttons are used to add or remove rig assignment for both tabs. The number of rigs must be specified before adding any rig assignment.

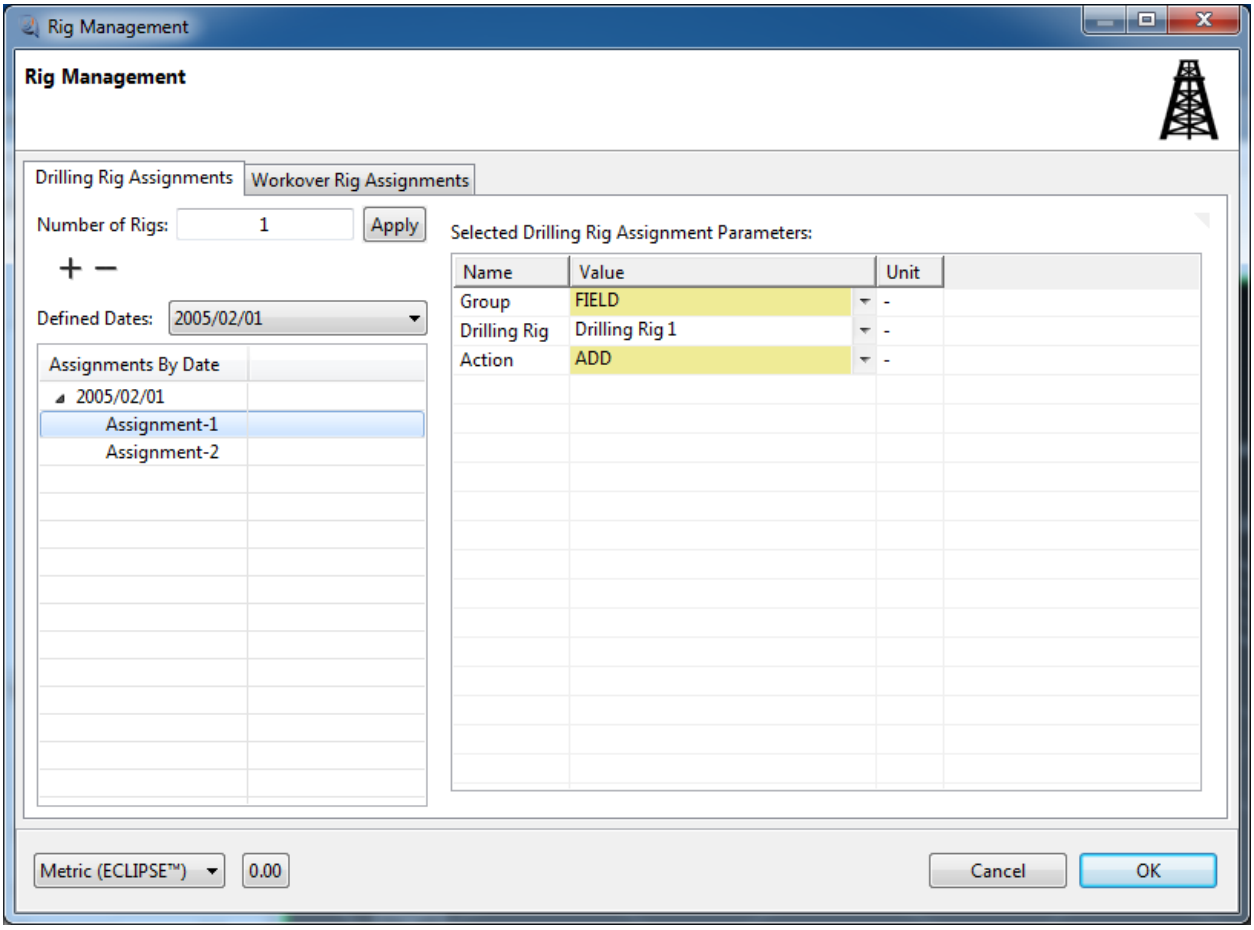


Figure 177-Drilling Rig Assignment

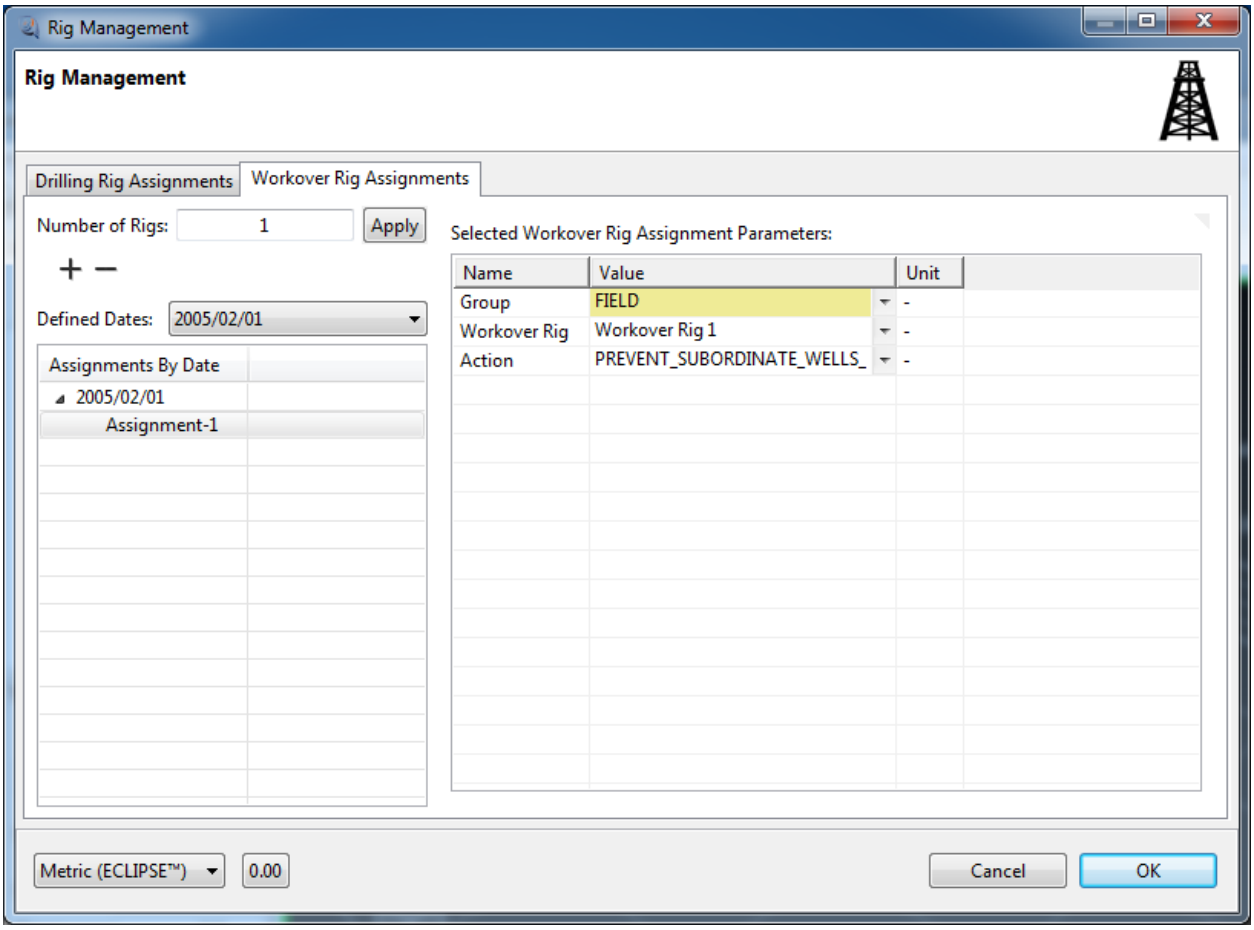


Figure 178- Workover Rig Assignment

3-4- Using Numerical Data Tab

3-4-1-Numerical Parameters

Clicking on “Numerical Parametes” button, the window of Figure 179 is opened. In this window the numerical parameters of solver options such as solution method, system phase, allowable errors for some parameters, bubble point and saturation switch triggers, preconditioner type, minimum and maximum pressure, using well inner iteration, and cascade analysis are entered. In this window user can enter Crank-Nicolson Multiplier also.

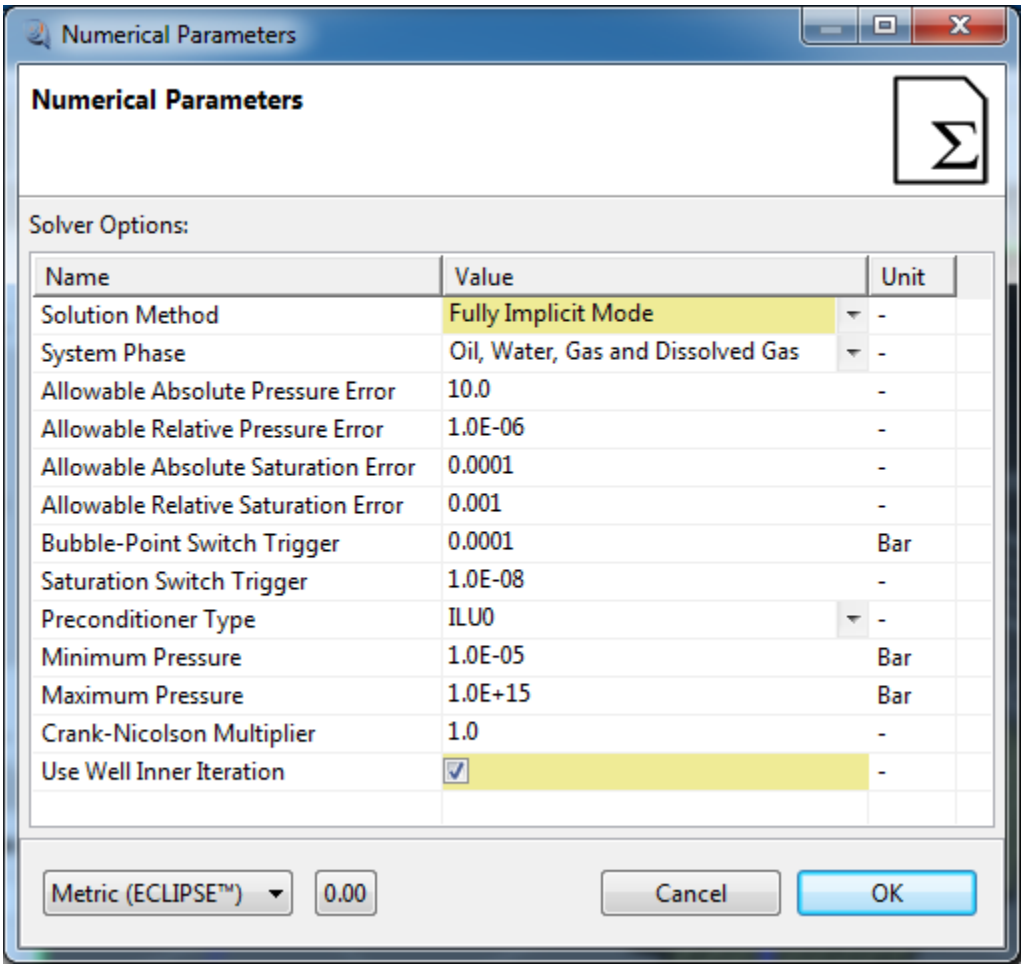


Figure 179- Numerical Parameters window

3-4-2-Output Data

The "output data" button, will lead user to the output data window (Figure 180). In this window the desired parameters for each block, region, groups, wells, connections and reservoir are selected for displaying next to the simulating. User can see run 3D animation just if select some data of the "3D Visualization Variables".

The summary vectors can be selected using "Summary Variables" tab of Figure 181. There is also a summary importer in "Output Data" window.

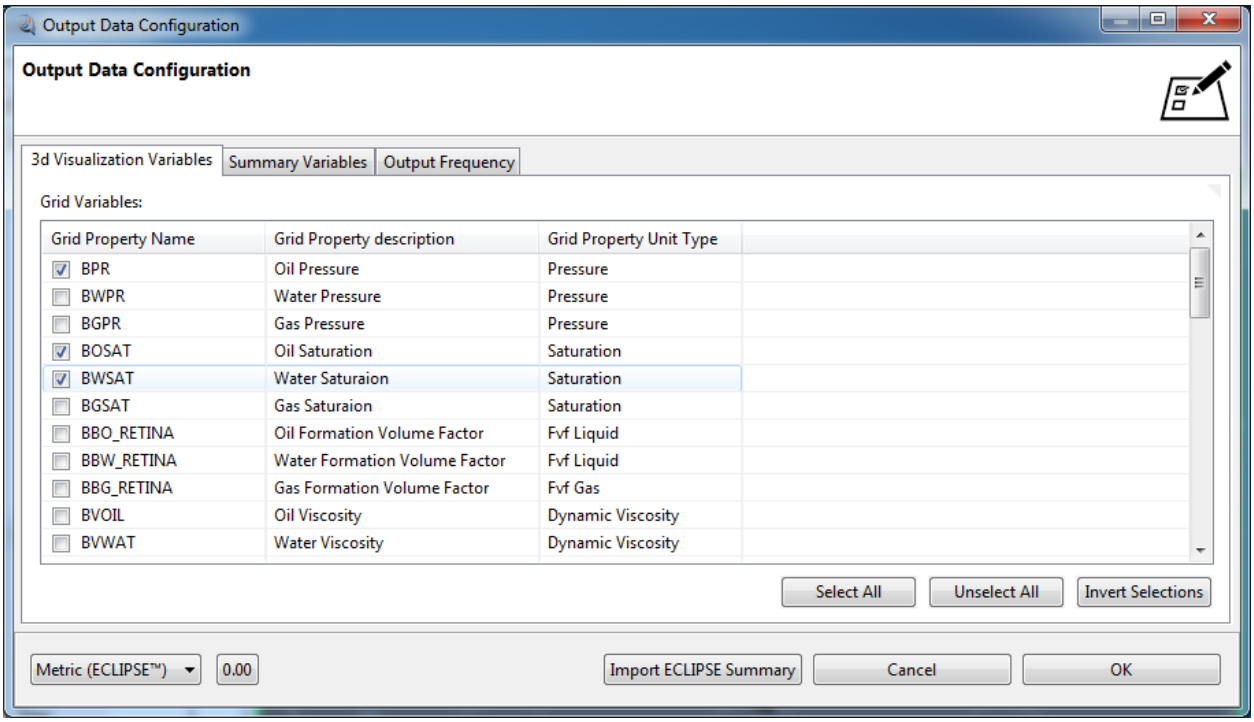


Figure 180- Output Data Window

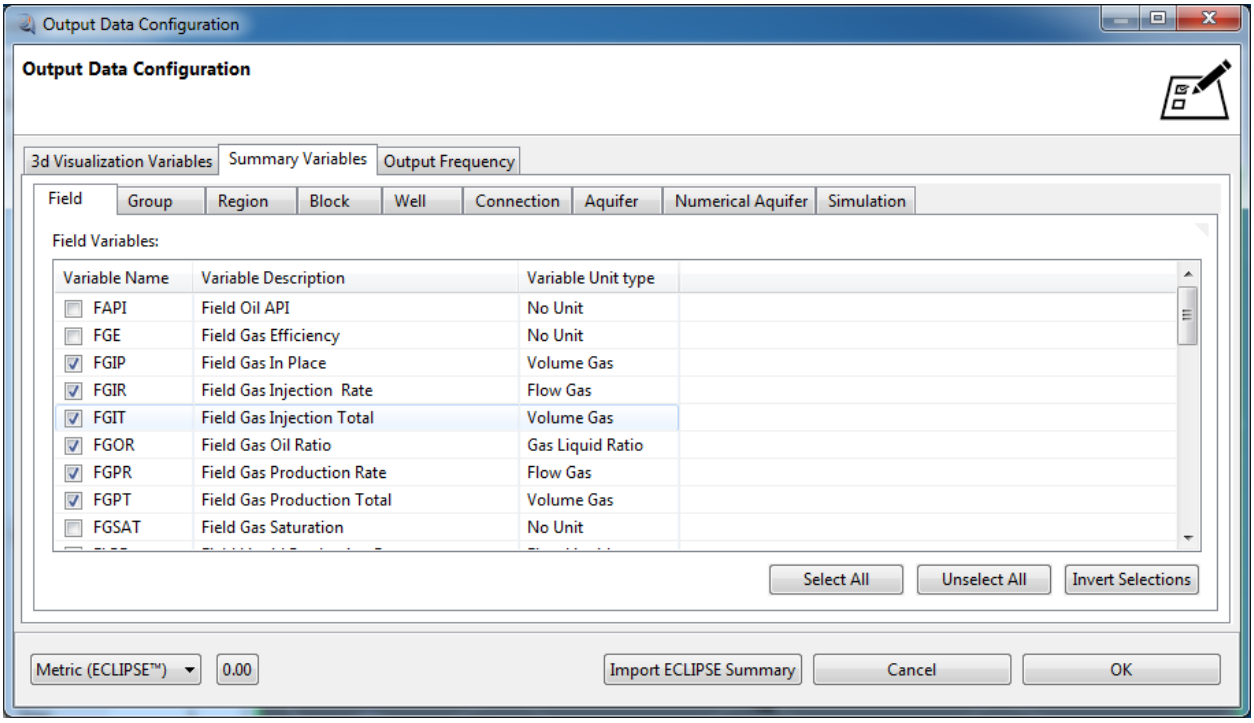


Figure 181- Summary Variables Tab

User can choose one or more time for having Restart Point in the output frequency tab, which could be performed after selecting desired time (Date) by clicking on the row and specifying plot dates (Figure 182). The values of Plot Counts can be edited by user one by

one (this specifies the count of the outputs in the desired interval). Also there is a capability of multiple Plot count using “Constant Plot Interval” selection in this window.

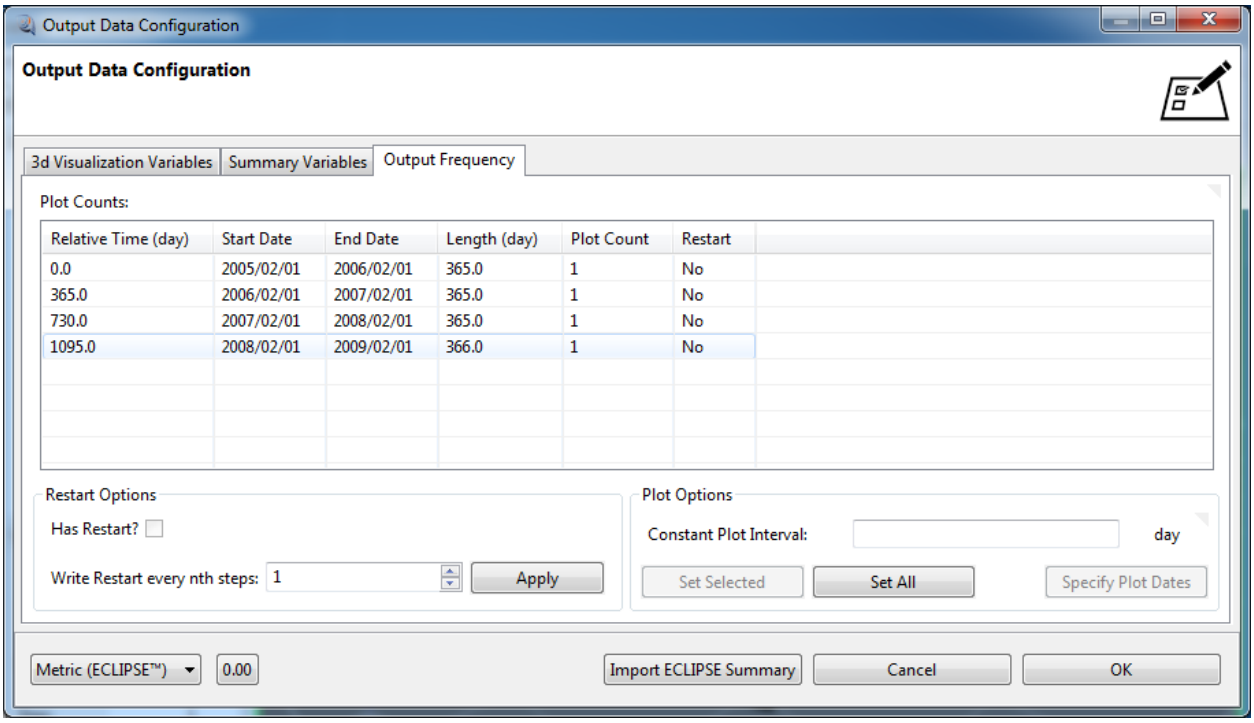


Figure 182- Output Frequency Tab

The “Specify Plot Count” button will display the Figure 183, in which the selected dates are shown. For adding new dates, use “Add Plot Dates” button.

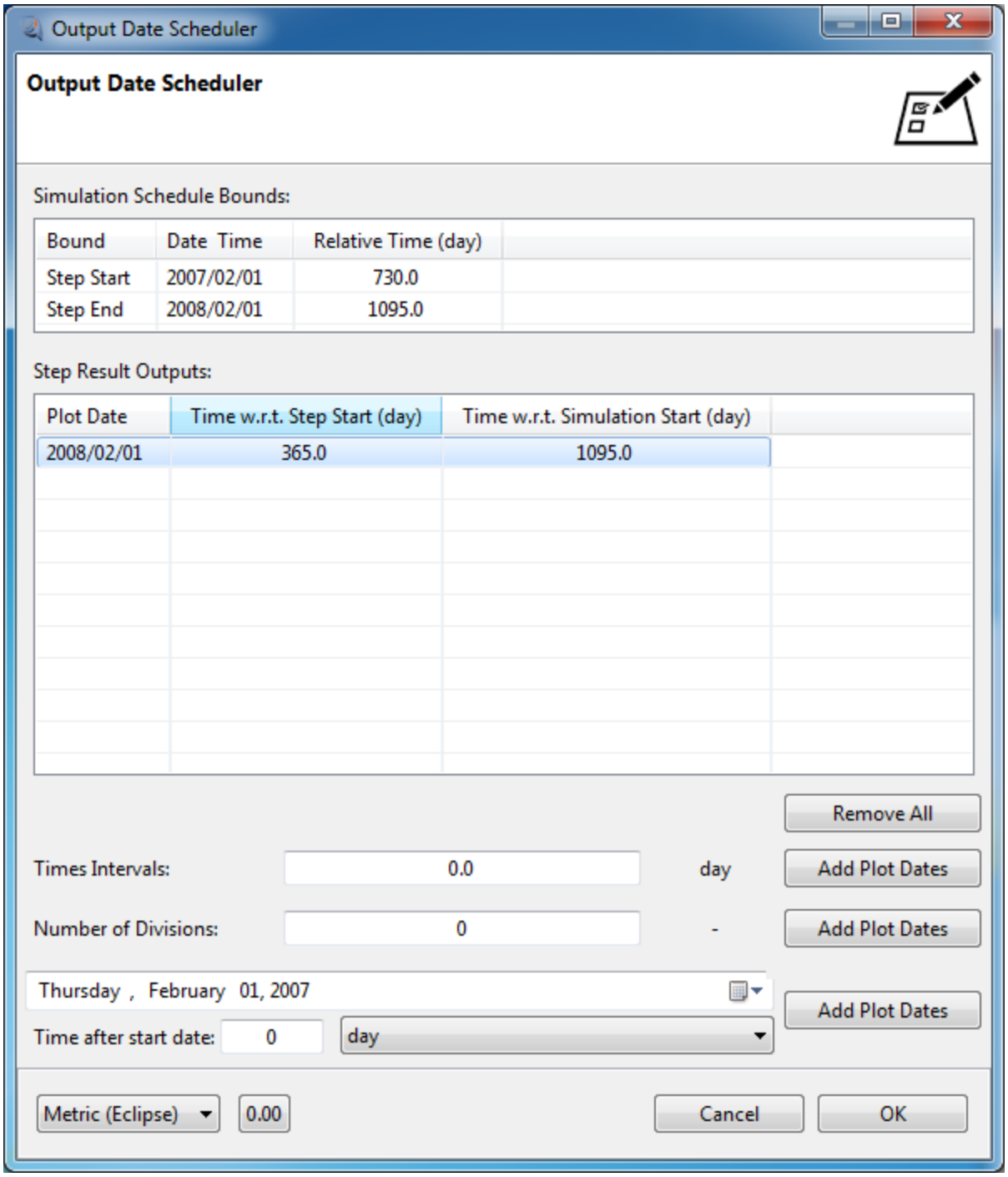


Figure 183- Specify Plot Count, Output Data Manager

4- How to Use 3D Toolbar

4-1- MTRX, FRAC, Dual

If the model is fractured, then three view options of matrix (M), fracture (F) and dual (M/F) are activated. Clicking on each of them will lead user to desired view.

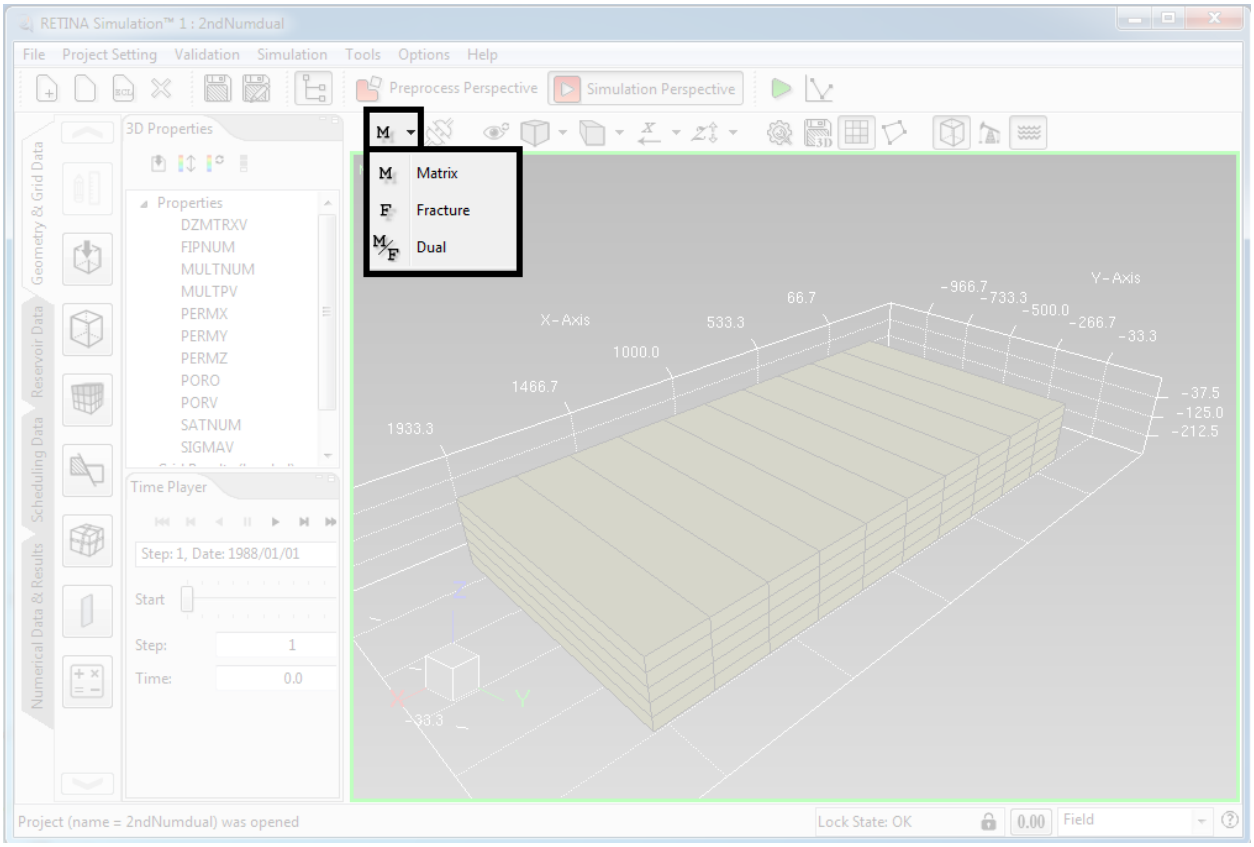


Figure 184- MATRIX, FRACTURE, and DUAL view

The name of active view is written in top left side of the 3D view. There is also a green border indicating active view. Figure 185 displays DUAL view for a sample mode.

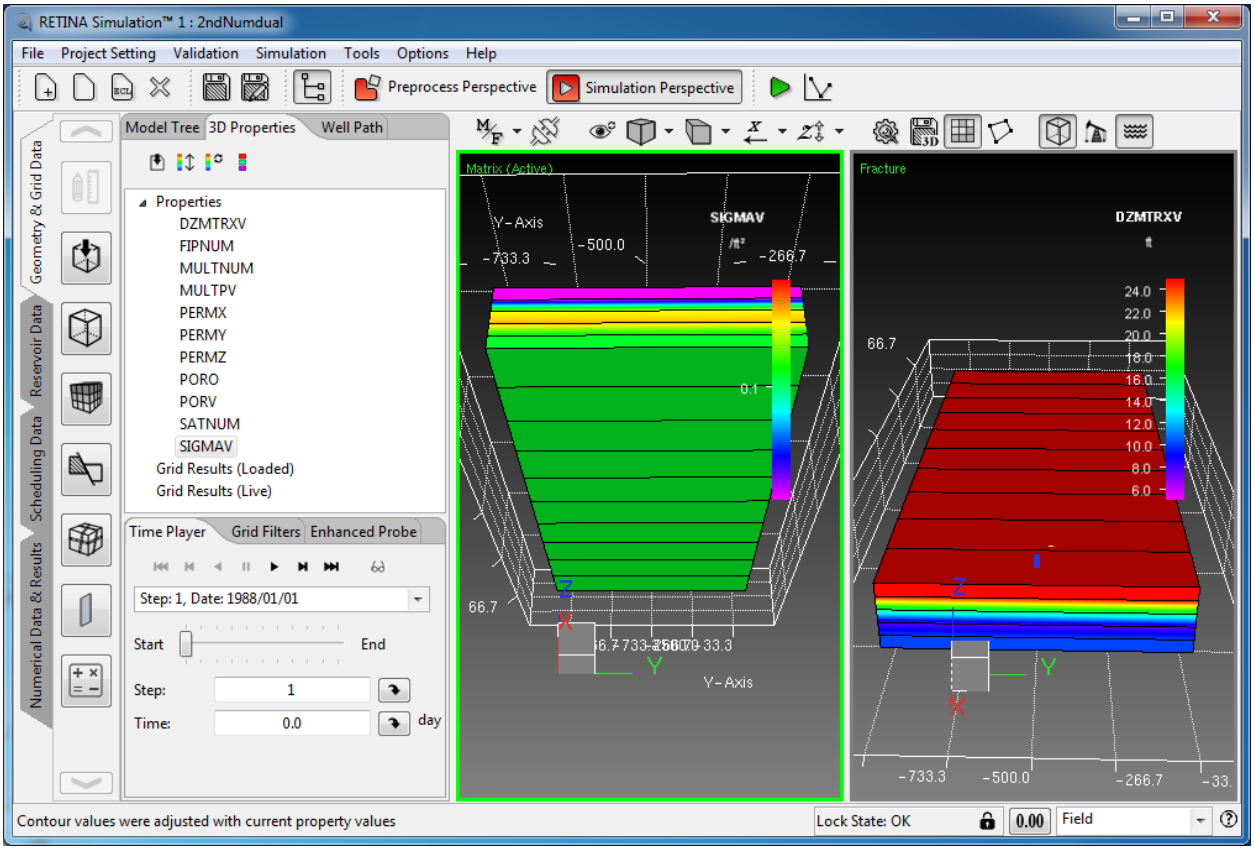


Figure 185- DUAL view fora Sample Model

The button is used for synchronizing both Matrix and fracture views. In synchronized mode (), the model properties and 3D zooming, rotating and etc. for active view is applied for inactive one, immediately. Note that inactive view can be rotated, zoomed and etc. independently. Figure 186 shows a rotation of active fracture view.

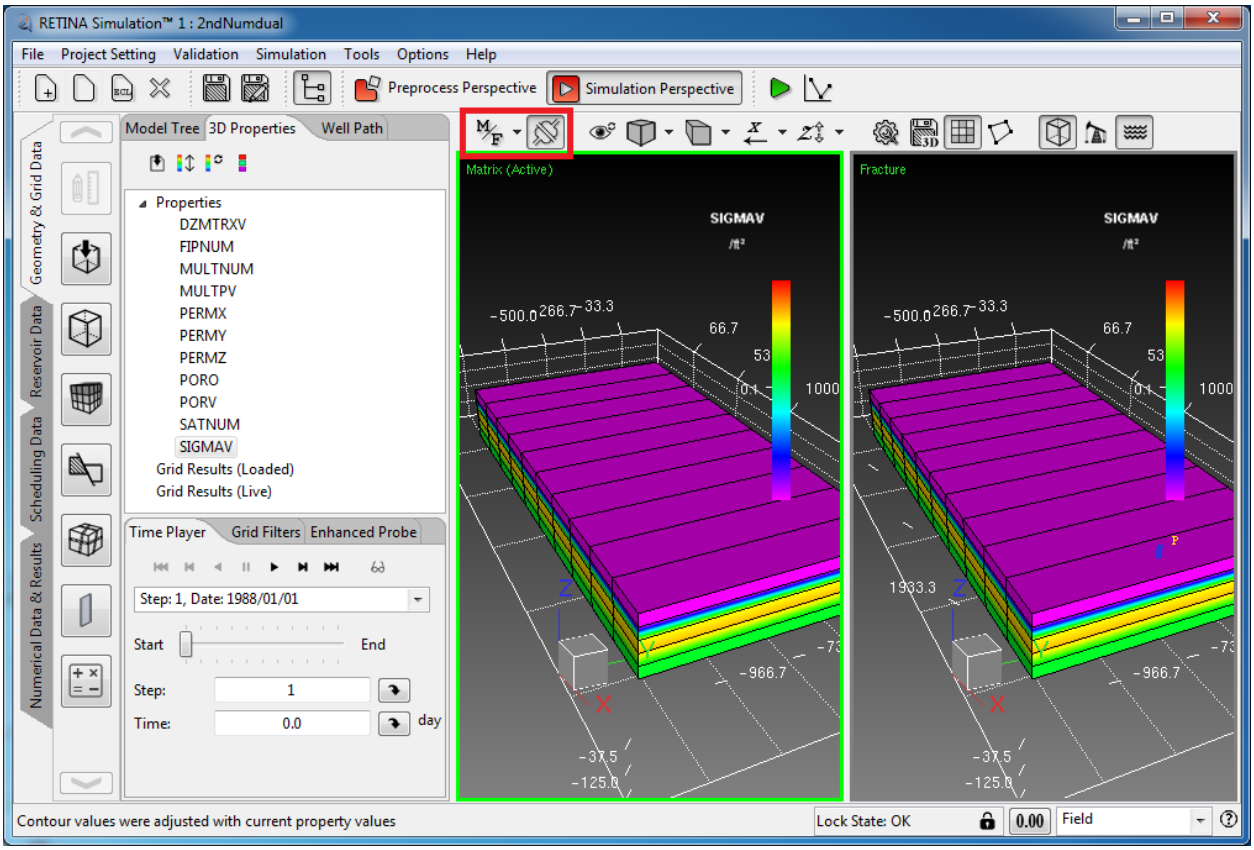


Figure 186- Synchronized Dual View

4-2- Displaying Options

The button of is used for resetting the 3D view to its default size and direction. The render option includes three options of render point, render wireframe, and render solid, which are shown in Figure 166. using render points option, the model is displayed by 8 points for each grid block (corner points of each grid block); whereas, the render wireframe option, is used to view each grid block by 12 lines (coordination lines); Finally if user wants to view the solid grid blocks, the render solid option must be selected; Figure 188 is shown above descriptions.

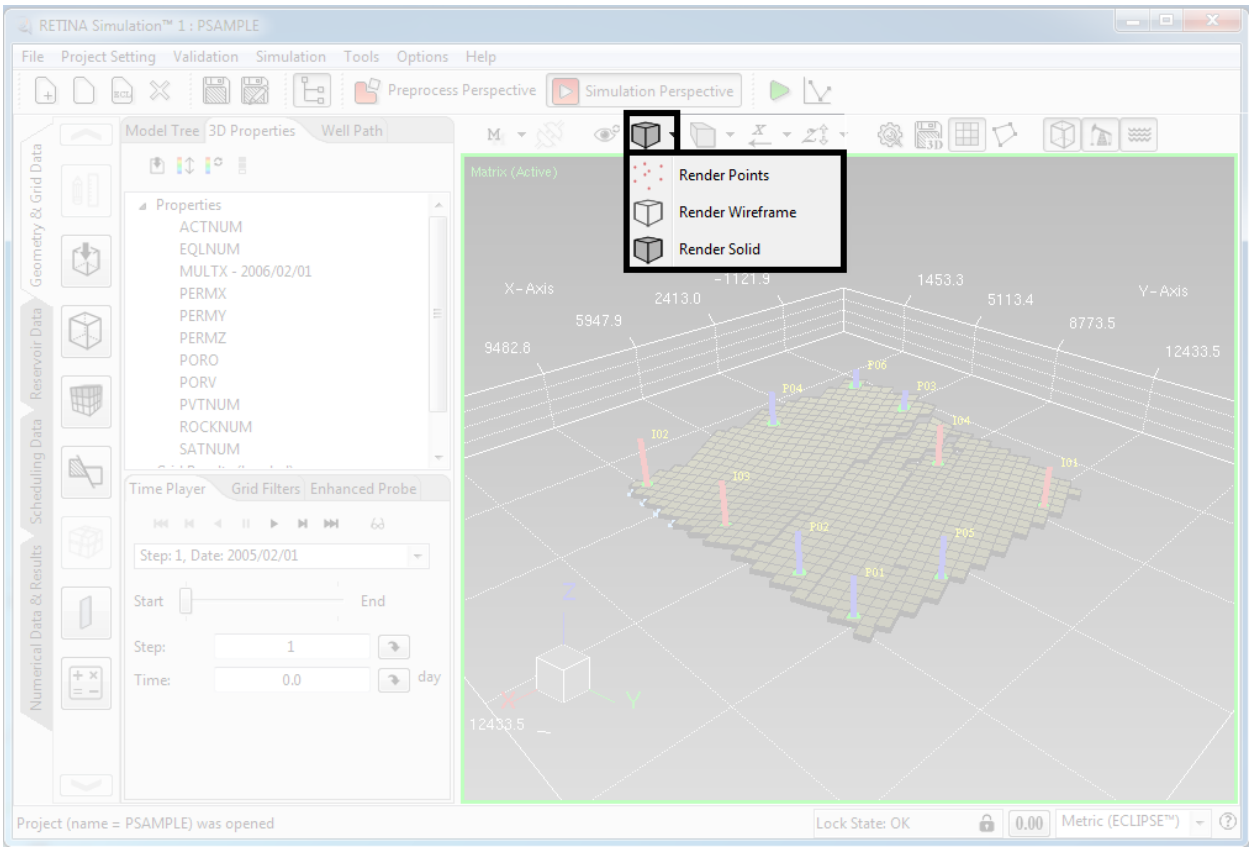
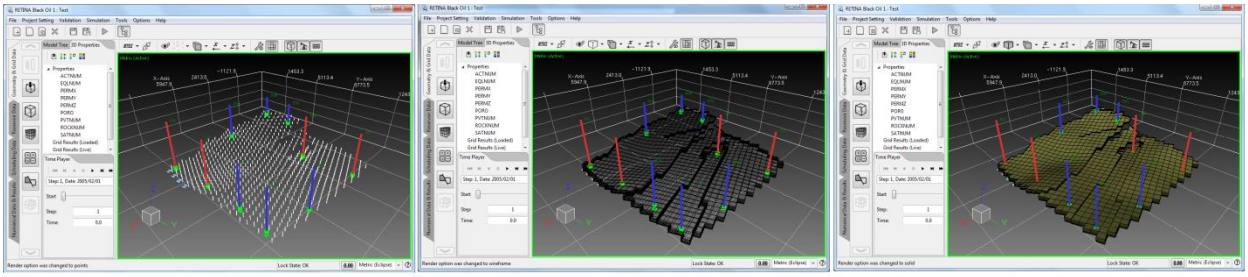


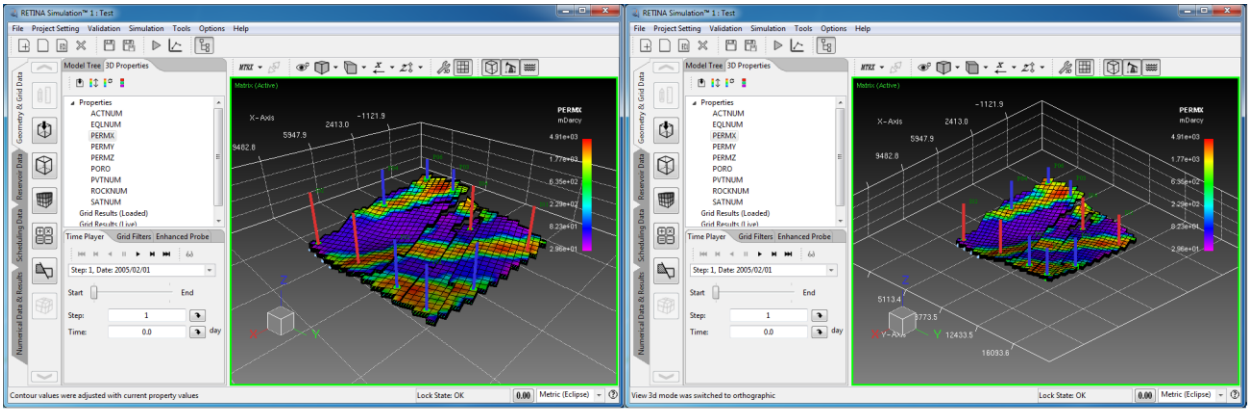
Figure 187- Render Options

The camera mode option includes two options of perspective and Orthographic. The perspective is default option. Both of the displaying options are shown in the Figure 189.



(a) (b) (c)

Figure 188- A) Render Pint, B) Render Wireframe, C) Render Solid



(a) *(b)*

Figure 189- A) Perspective, B) Orthographic

User can use Figure 190 options to change view in desired directions of x, y or z axis. There are two portions for each axis which are either in positive or negative directions. In fact, this is a 2D view option.

Sometimes and especially in thin reservoirs, detecting layers in the Z direction is hard. So user can increase or decrease the Z-Scale of the model using "Z-Scale Action" options. There are 10 scale factors of 0.1 to 100 for scaling Z ( )

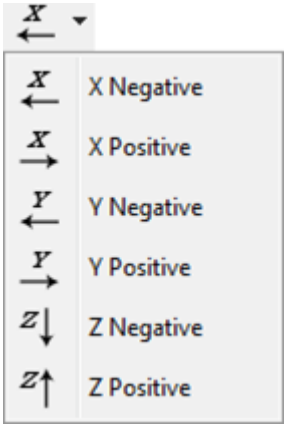
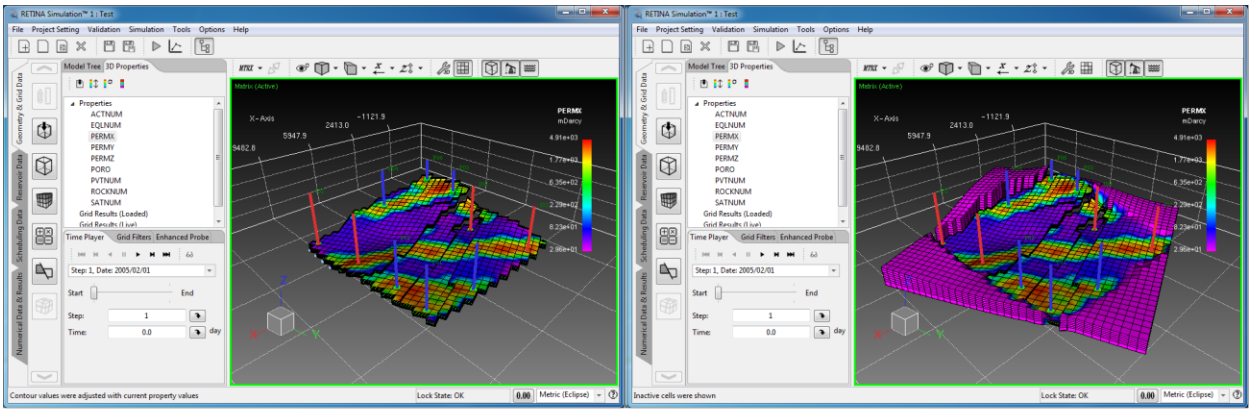


Figure 190- View Modes

Four buttons are available for show/hide some components of the model:

- Show or hide inactive cells
- Show or hide model grids
- Show or hide model wells
- Show or hide aquifers

All of the above buttons except inactive cells option are in show mode when the model loaded. For changing status to the hide, just click on the related button. For example Figure 191 shows the model in two status of: A) Show wells and Hide Inactive Cells and B) Hide Wells and Show Inactive Cells.



(a) *(b)*

Figure 191- A) Show wells and Hide Inactive Cells, B) Hide Wells and Show Inactive Cells

4-3- 3D View Setting and Save

3D View setting is available using  button (Figure 192). There are two tables in this window, which include items and their information. There are two buttons of “Close” and “Reset” in the bottom of the setting window, close will implement changes and reset will cancel any changes you made in setting. You can save a screen shot of the 3D view in .png format using the  button.

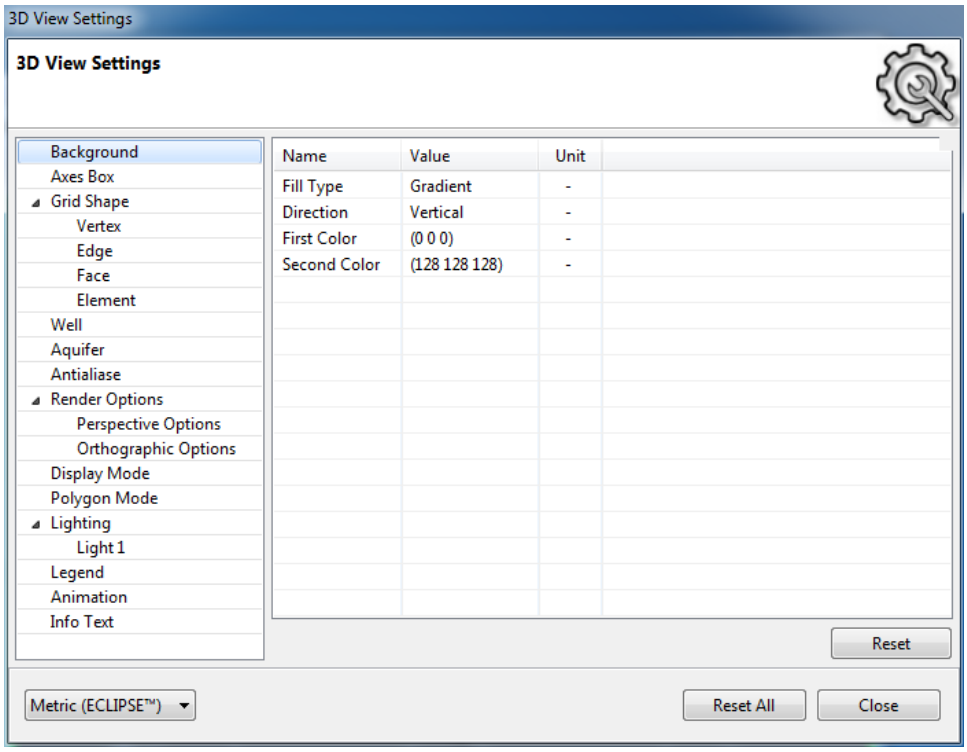


Figure 192- 3D View Setting

4-3-1-Background Settings

The background settings includes following items:

Fill Type: it can be chosen either Solid or Gradient. There is no color gradient in the solid one.

Direction: the direction of the color gradient will be either horizontal or vertical.

Fist Color: user can choose first color of the background from Figure 193.

Second Color: it is the second color of the background which can be select from Figure 193.

Note that, in the gradient fill type of the background, the gradient lies between first and second colors (Figure 194).

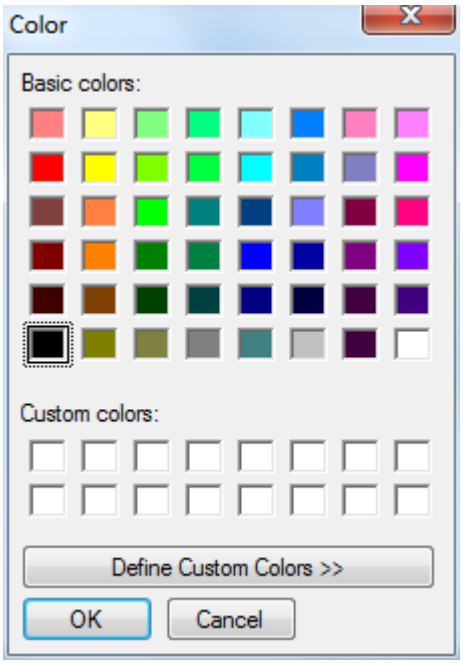


Figure 193- Background Colors

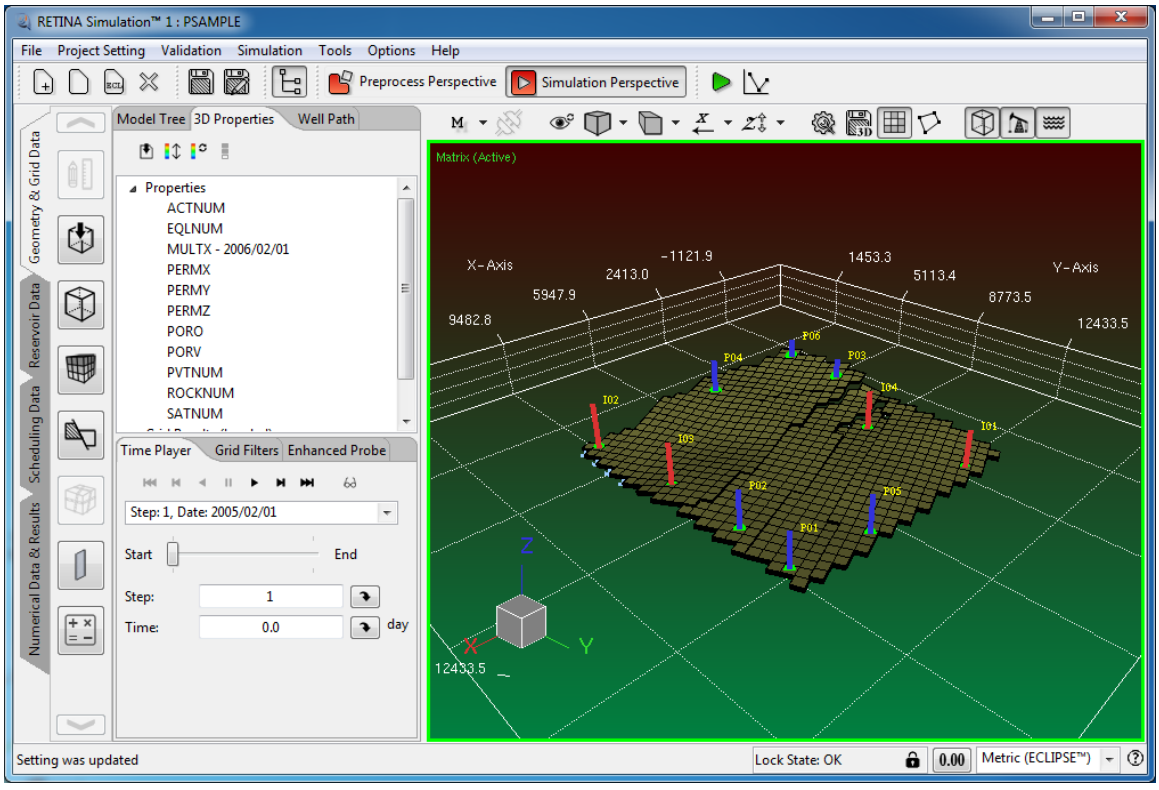


Figure 194- Background Color

4-3-2-Axes Box Settings

In this part of the setting window, the visibility, scale, line width, colors and precision will be defined by user. If you want remove axes from 3D view, just change the “false” to “True” in the value column.

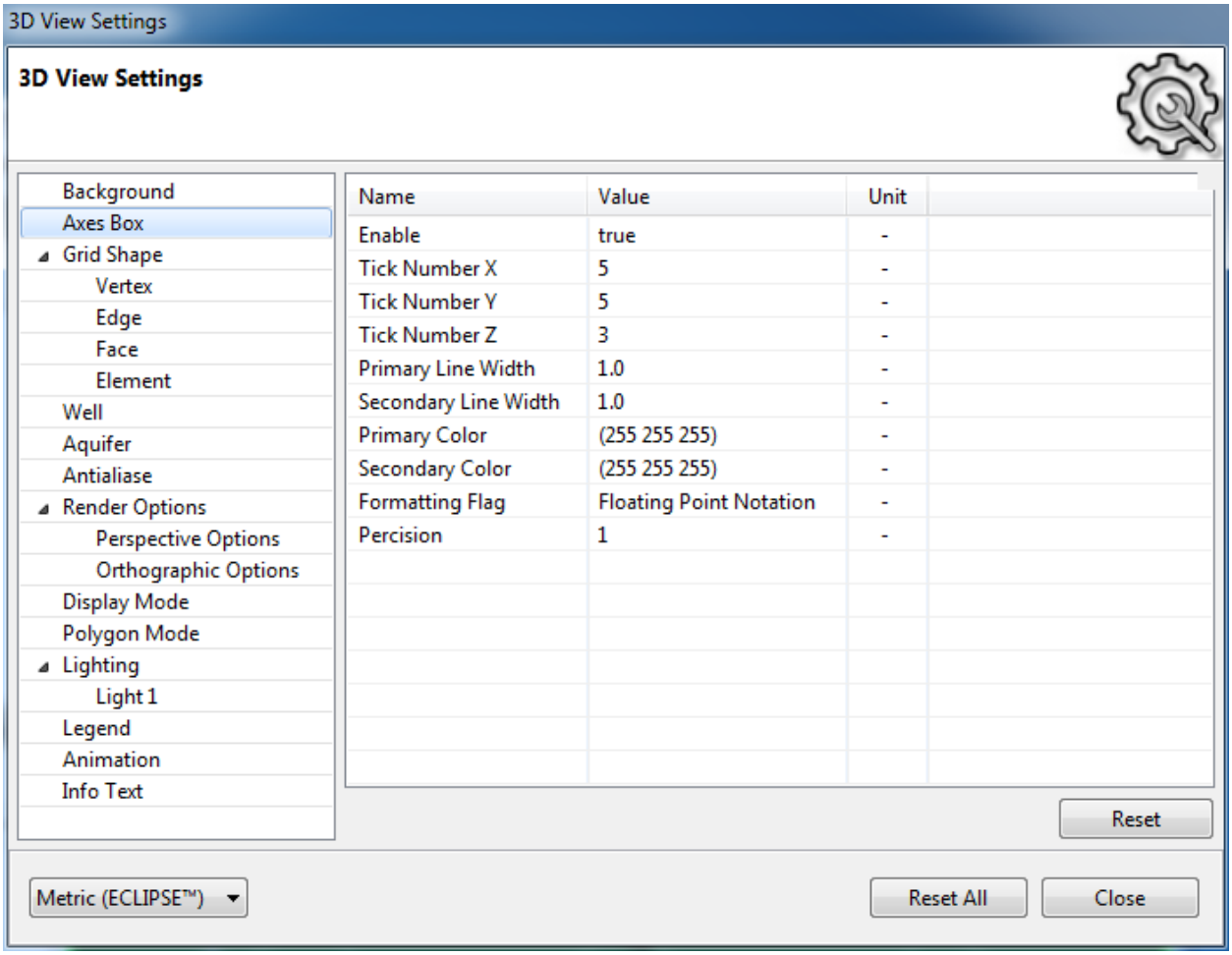


Figure 195- Axes Box Settings

4-3-3-Mesh Shape Settings

User can use this window for the color and size settings in the following statuses:

- *Vertex*: available for “Render Point” option.
- *Edge*: available for “Render Wireframe” option
- *Face*: available for “Render Solid”
- *Element*: available for “Render Solid”

4-3-4-Well Settings

In this part of the setting window, the visibility, well radius, Injection and production default color, injection and production selection color, well name text color and offset can be defined by user. The wells are visible while the "Value" column is true. For example Figure 197 shows the model, in which the colors of the injection and production wells are white and blue, respectively. The well names text color is yellow and the radius of the wells defined to be twice of default value.

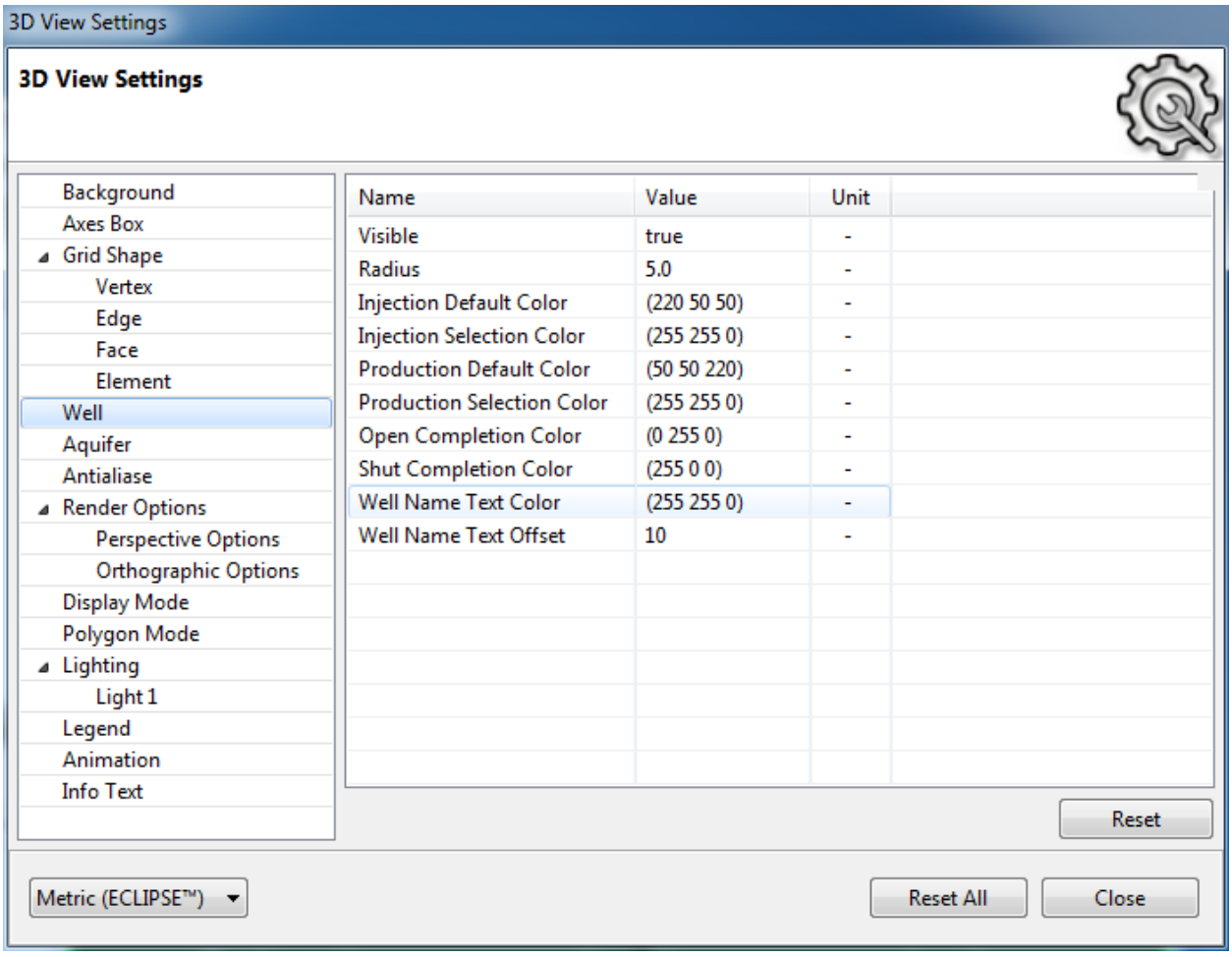


Figure 196- Well Settings

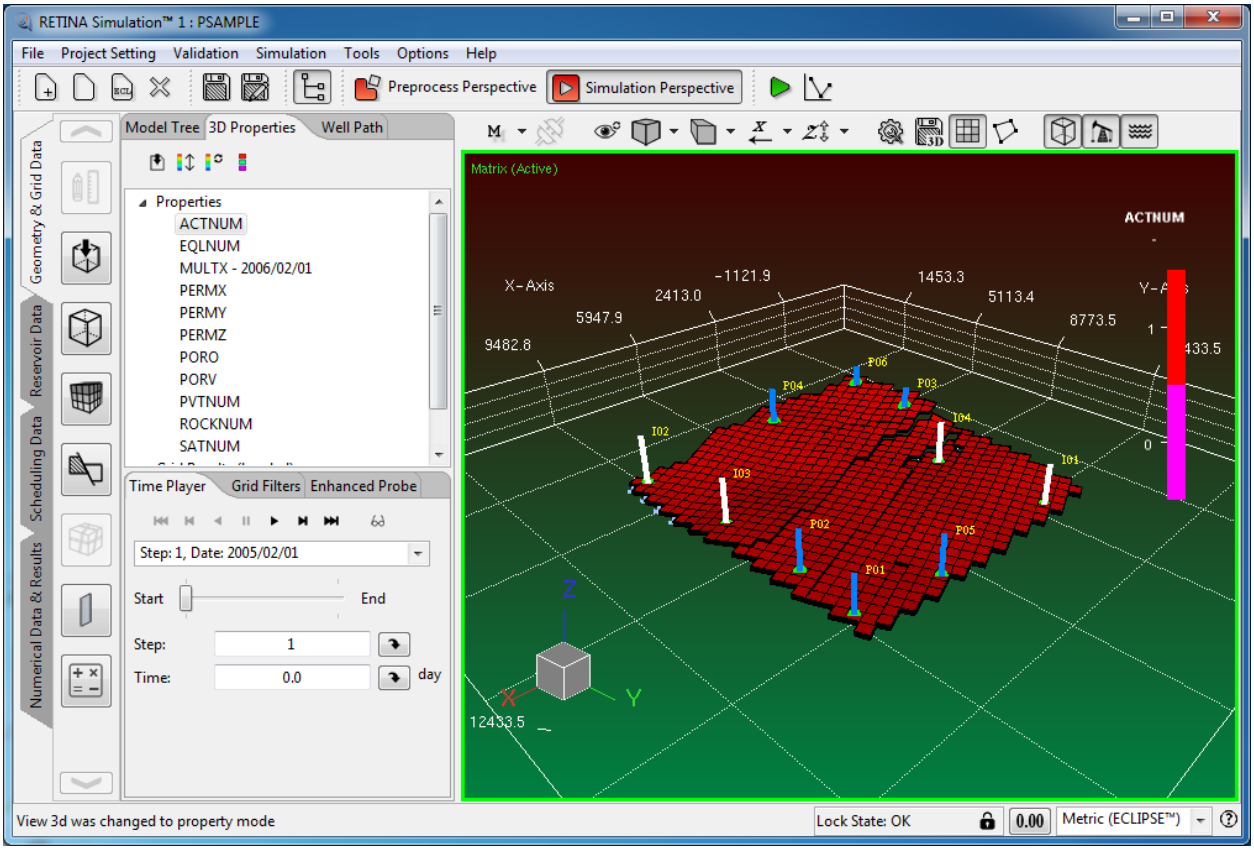


Figure 197- Well Visual Settings

4-3-5-Aquifer Settings

The Aquifer settings are displayed in the Figure 198. The visibility, default and selected colors and aquifer size should be defined here.

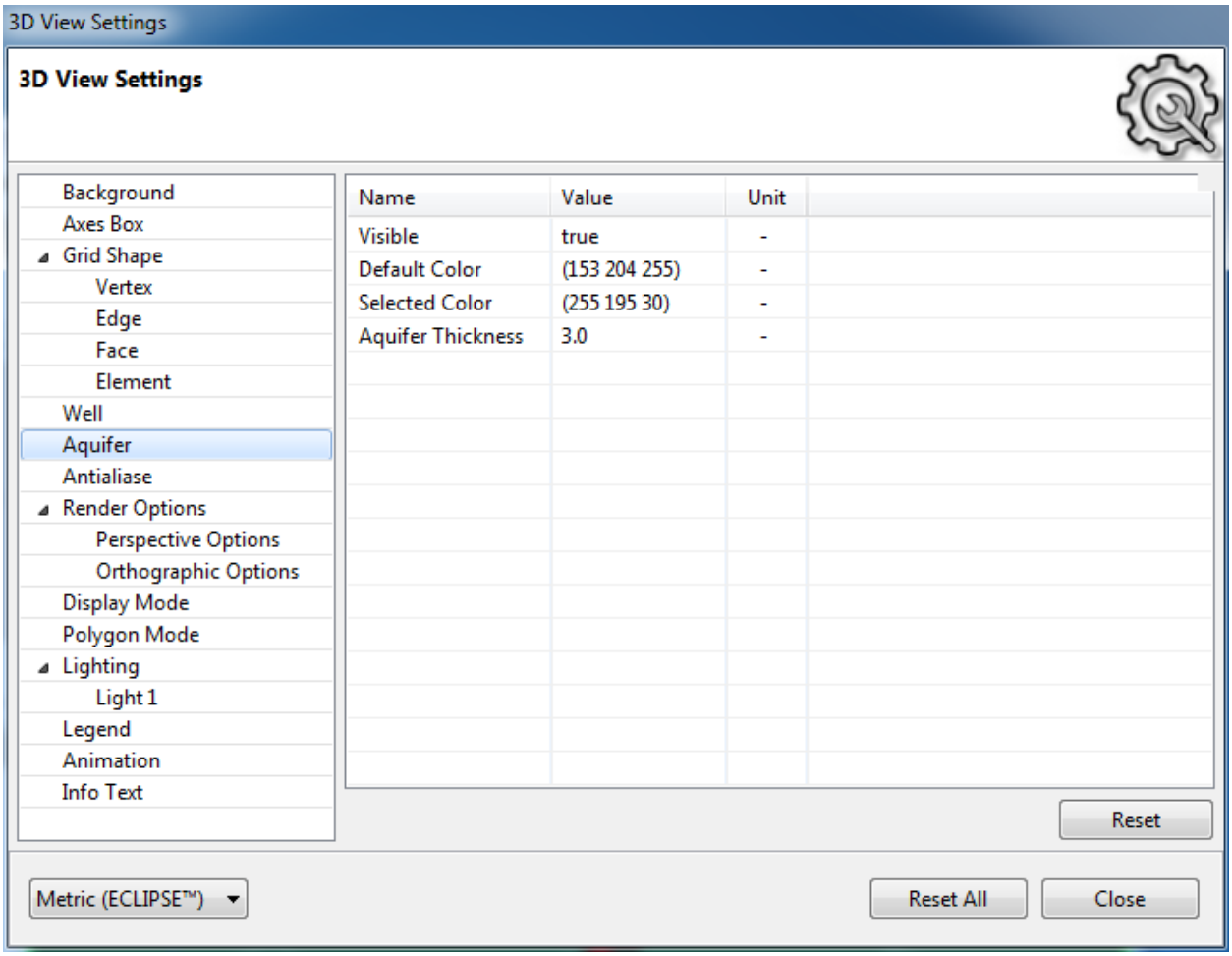


Figure 198- Aquifer Settings

4-3-6-Antialiasing Settings

The point and line smoothing setting is done here. There are three options for line and point smoothing:

- *Nicest*: the quality of point and lines are in the highest level
- *Fastest:* the quality of point and lines are in the intermediate level
- *Don't Care*: the quality of point and lines are in the lowest level

4-3-7-Rendering Settings

There are four types of settings in this section:

- *Camera Mode*: it will be either perspective or orthographic
- *Scale*: there are three numbers in "Value" column, which are scale multipliers in x, y, z directions. For example if the first number changes to 2, the length of model in the x direction will be multiply in 2. The Scale with (3, 2, 10) is shown with the original view of the model in the Figure 200.

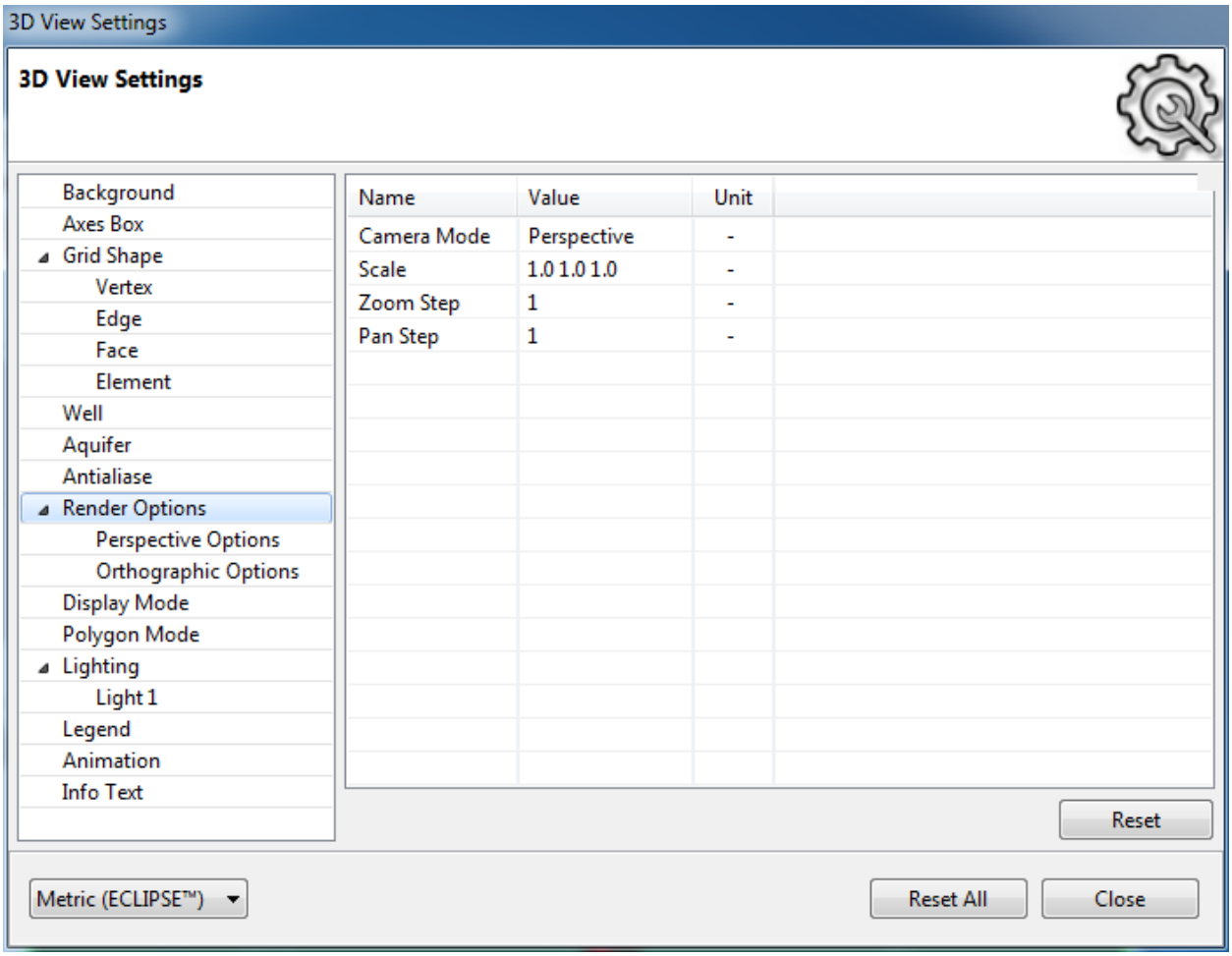
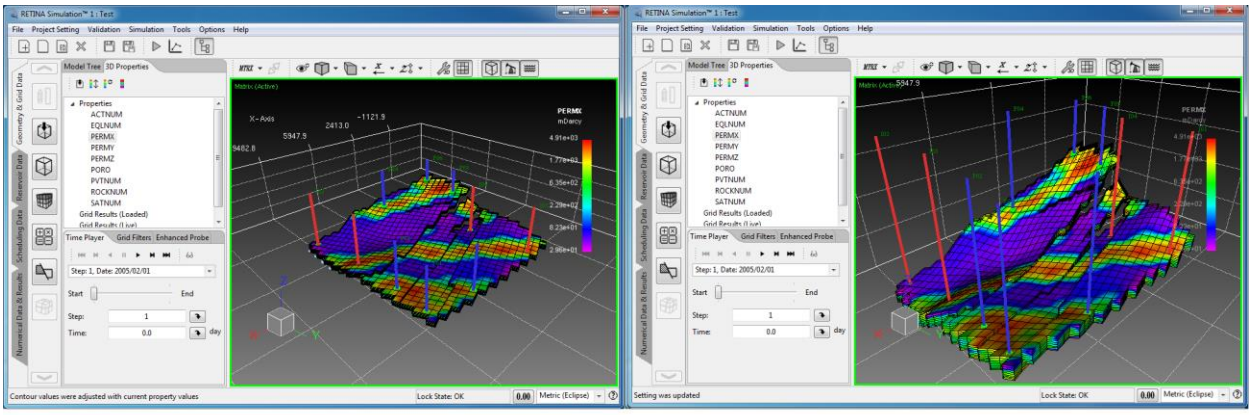


Figure 199- Rendering Settings



(a) (b)

Figure 200- A) Original Scale, B) Scaled to (3, 2, 10)

Perspective Settings:

Following items which are displayed in Figure 201 should be defined by user:

- *Fovy:* y direction angel
- *Aspect*: the ratio of width to height in each projection
- *Z Near Perspective*: distance of the watcher from near plane
- *Z Far Perspective*: distance of the watcher from far plane

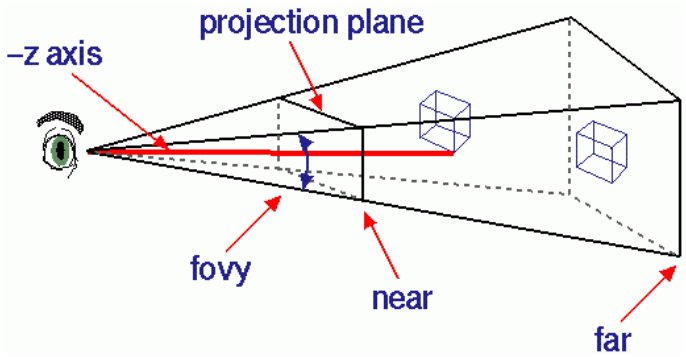


Figure 201- Perspective Parameters

Orthographic Settings:

Following items which are displayed in Figure 202 should be defined by user:

- *Screen Width*
- *Screen Height*
- *Z Near Orthographic*

- *Z Far Orthographic*

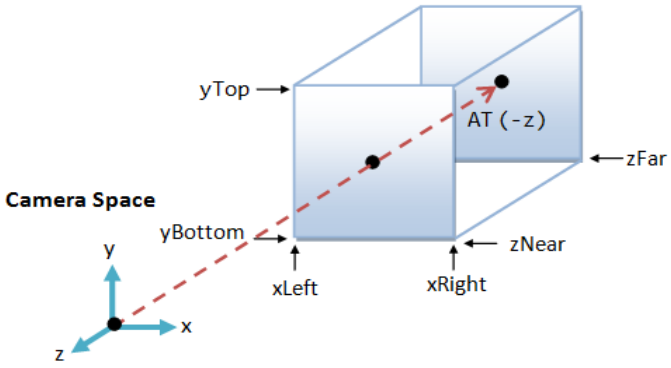


Figure 202- Orthographic Parameters

4-3-8-Display Mode Settings

The type and transparency Ratio of the blocks are defined here. Only when the type is "Transparent" the transparency ratio will have effect.

4-3-9-Polygon Mode Settings

It lets user to choose the wireframe, point or solid modes.

4-3-10- Lighting Settings

The lighting settings window is shown in Figure 203. Initially the lighting is active by default. If user wants to inactive this option, should change "true" to "false" in the "Value" column.

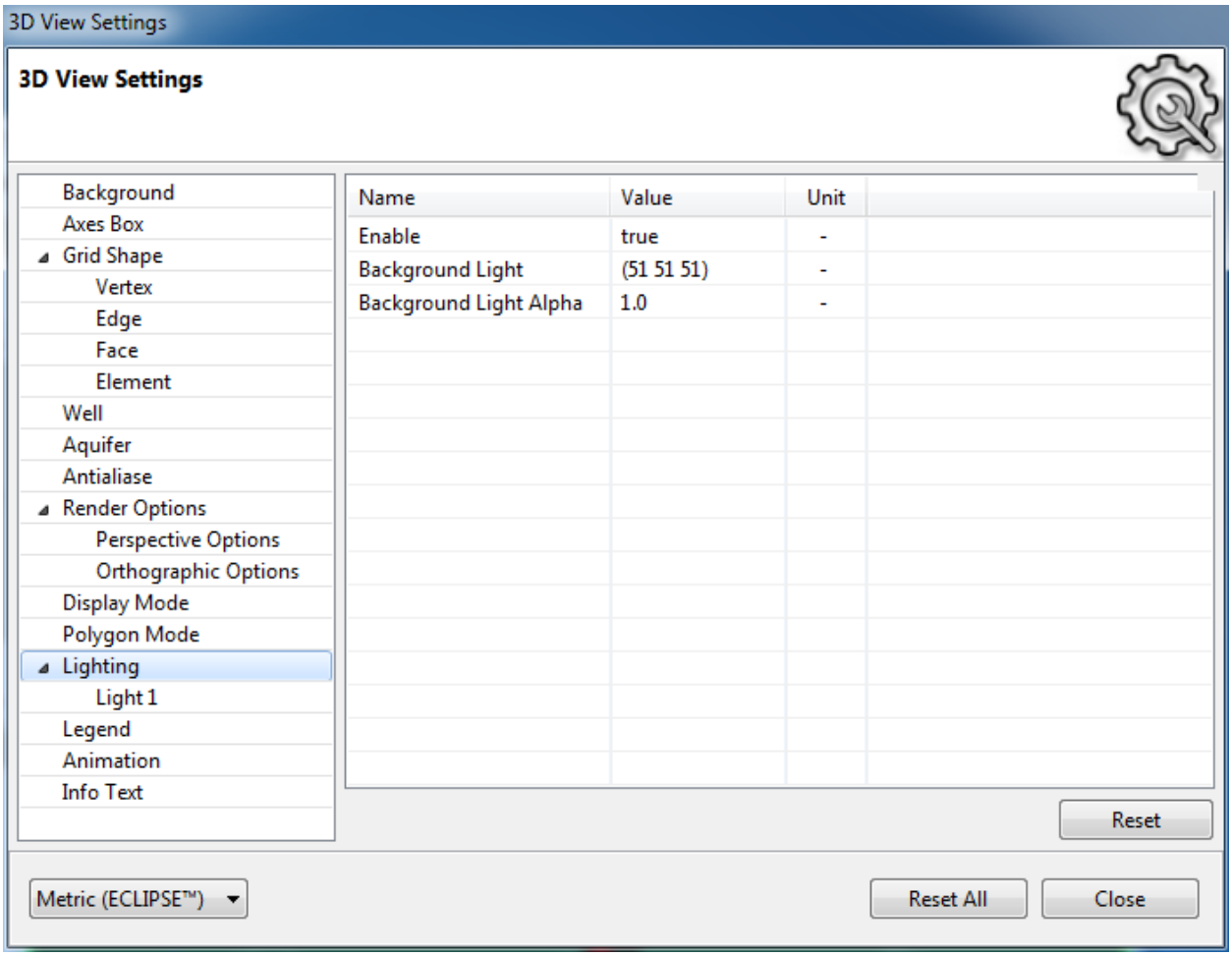


Figure 203-Lightning Settings

4-3-11- Property Settings

Following settings are available here:

- *Type*: the coloring type can be either continuous color or block color.
- *Level Number*: the number of the colors is defined here.
- *Grid Enable*: it is initially active, if user try to inactivate this option, the grids won't be shown any more.
- *Color Map*: there are four color maps in this setting dialog as: Extended rainbow, Gray Scale, Rainbow and two colors.

Legend:

Legend setting includes show/hide, coloring and its type, header font and size, numbers font and size, legend background settings and font colors. The Figure of 204 displays the legend for porosity in a typical model.

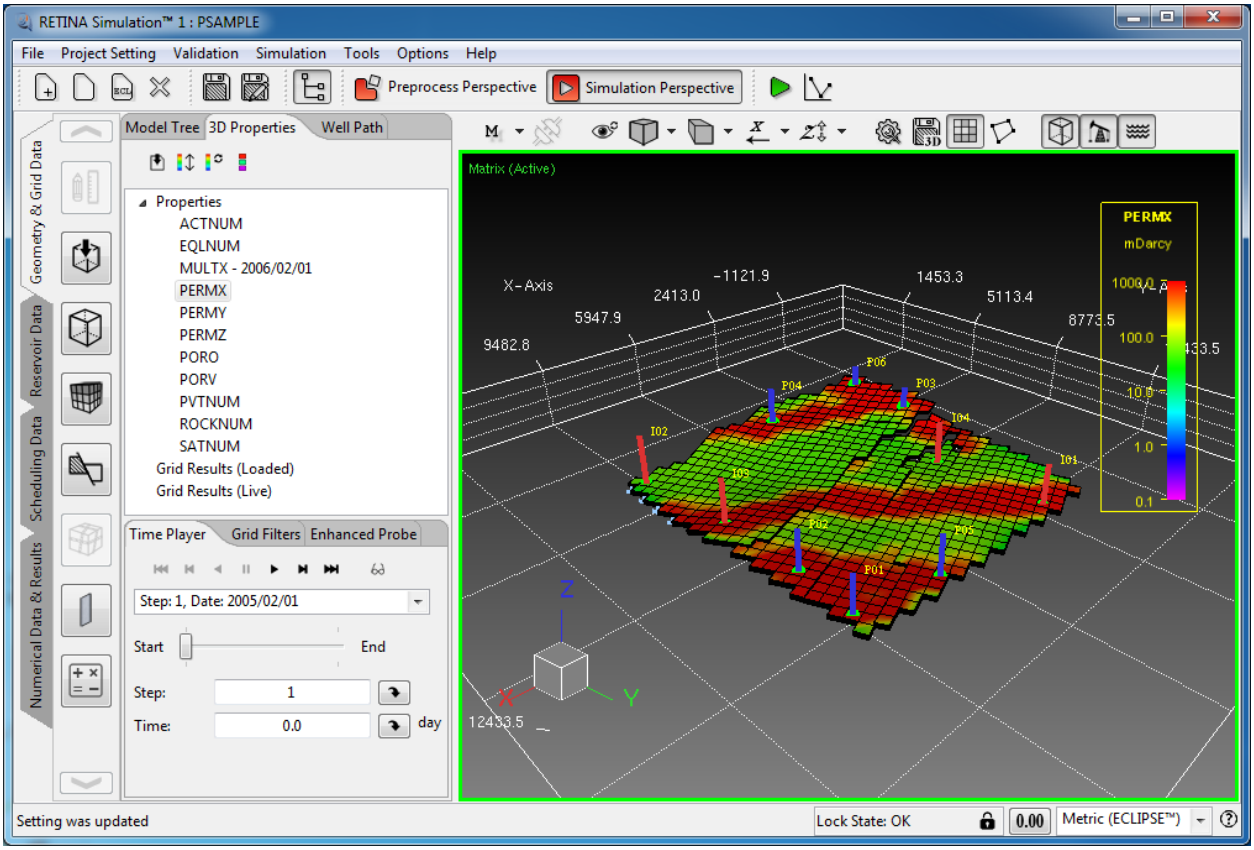


Figure 204- Legend View

4-3-12- Animation and Info Text Settings

The time settings of animating, font and color of the info text (displayed in the bottom of the windows) are available here.

4-4- Mouse Buttons Action and Keyboard Shortcuts

4-4-1-Rotation

Right button of the mouse can be used to rotate desired object in the 3D view. You should hold this button down and try to rotate the object. The rotation will performed around the normal vector of the plate.

4-4-2-Movement

Middle button of the mouse or “Ctrl+left button” of the mouse, can be used to move desired object in the 3D view. You should hold this buttons down and try to move the object. The movement will performed in the direction which mouse is moved.

4-4-3-Zooming in and out

The wheel of the mouse is used to zoom in or zoom out. In addition to this the “Ctrl+middle button” of mouse will be used to zoom in or zoom out the model.

4-5- Keyboard Shortcuts

There are two types of shortcuts:

- Menu shortcuts (Figure 205)
- General shortcuts (Figure 206)

	Binding
New Project	Ctrl + N
Open Project	Ctrl + O
Open Eclipse Data File	Ctrl + E
Save Project	Ctrl + S
Save As Project	Ctrl + D
Close Project	Ctrl + W
Project Information	Ctrl + I
Open Eclipse Import Messages	Ctrl + M
Full Project Validation	Ctrl + F
Start Simulation	Ctrl + R
Simulation Configuration	Ctrl + G
Open Plot Result Viewer	Ctrl + T
Open History Schedule Builder	Ctrl + B
Help	Ctrl + H
Technical Document	Ctrl + J

Figure 205- Menu Shortcuts

	Binding
Increase Z Scale	Ctrl + =
Decrease Z Scale	Ctrl + -
Reset View	Ctrl + Shift + R
Matrix View	Ctrl + Shift + M
Fracture View	Ctrl + Shift + F
Dual View	Ctrl + Shift + D
X Negative	Ctrl + Shift + X
Y Negative	Ctrl + Shift + Y
Z Negative	Ctrl + Shift + Z
Show Hide Left Panel	Ctrl + Shift + P
Show Hide Inactive Cell	Ctrl + Shift + C
Show Hide Grid	Ctrl + Shift + G
Show Hide Wells	Ctrl + Shift + W
Show Hide Aquifers	Ctrl + Shift + A
View 3 D Setting	Ctrl + Shift + S
In Sync	Ctrl + Shift + I
Rotate Right View 3 D Model	Ctrl + Shift + K
Rotate Left View 3 D Model	Ctrl + Shift + H
Numerical Data	Ctrl + Alt + N
Rotate Up View 3 D Model	Ctrl + Shift + U
Rotate Down View 3 D Model	Ctrl + Shift + J
Zoom View 3 D Model In	Ctrl + Shift + =
Zoom View 3 D Model Out	Ctrl + Shift + -
Pan Right View 3 D Model	Ctrl + Shift + Right
Pan Left View 3 D Model	Ctrl + Shift + Left
Pan Up View 3 D Model	Ctrl + Shift + Up
Pan Down View 3 D Model	Ctrl + Shift + Down
X Positive	Ctrl + Alt + X
Y Positive	Ctrl + Alt + Y
Z Positive	Ctrl + Alt + Z
Z Positive	Ctrl + Alt + Z
Model Tree	Ctrl + Alt + M
View 3 D Properties	Ctrl + Alt + 3
Time Player	Ctrl + Alt + T
Grid Filters	Ctrl + Alt + F
Enhanced Probe	Ctrl + Alt + P
Geometry And Grid Data	Ctrl + Alt + G
Reservoir Data	Ctrl + Alt + R
Schedule Data	Ctrl + Alt + S

Figure 206- General Shortcuts

5- Program Standard Toolbar

This toolbar consists of the following buttons:

- File
- Project Setting
- Validation
- Simulation
- Tools
- Options
- Help

5-1- File

The file button includes following options:

 Create a New Project
 Open a RETINA Saved Project
 Open an Eclipse Data File
 Close Project
 Save Current Project
 Save Project as a New One
 Exit

The first three items ( ,  , ), were described in previous sections (section 1).

The Button of  is used to close the project. The  button is used to save the project to the current file, the  button is also used to save project as new one, note that all of the RETINA projects are saved as a *.retina files.

5-2- Project Setting

The “Project Setting” includes following two components:

 Project Information
 Open Eclipse Import Messages

User can enter project name, validator engineer name, supervisor, reservoir and engineer names in “Project Information” window (Figure 207). The Grid Type and project start date field are not editable.

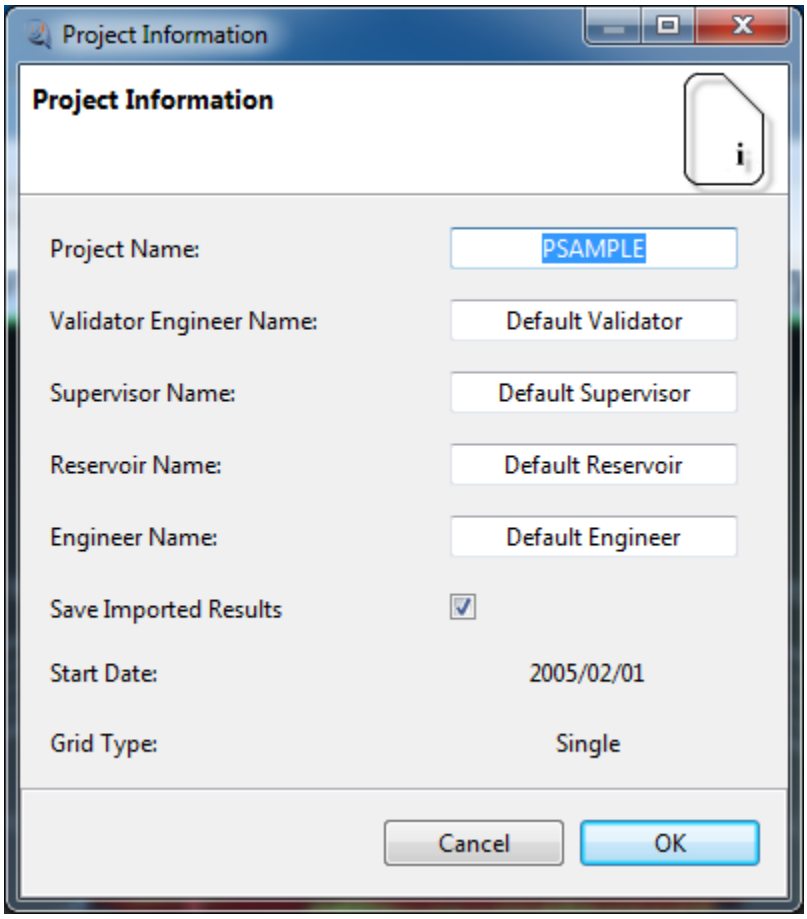


Figure 207- Project Information

Open Eclipse import message, displays all messages of error, warning and info levels (Figure 208).

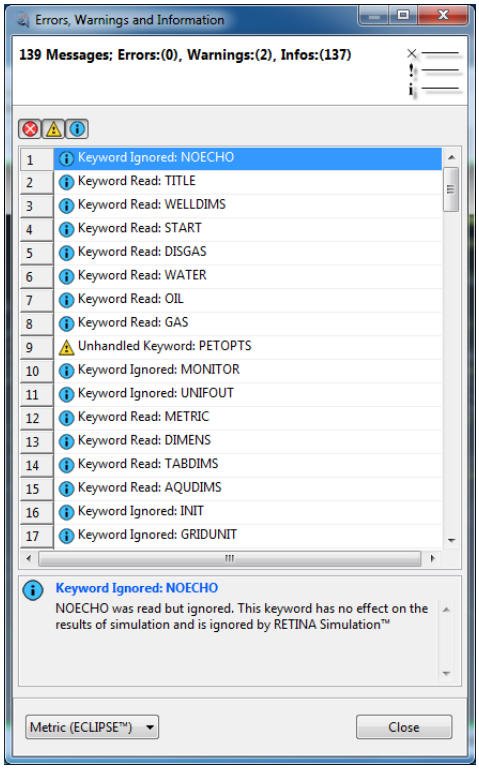


Figure 208 - Local Coordinate System

5-3- Validation

There are 9 validation options in the RETINA Validation button:

- ☑ Full Project Validation
- ☑ Grid Validation
- ☑ Function Validation
- ☑ Initialization Validation
- ☑ End-Point Scaling Validation
- ☑ Aquifers Validation
- ☑ Scheduling Validation
- ☑ Numerical Validation
- ☑ Output Validation

Each validation option validates just a specific portion of the RETINA, except first one. The first option is used to validate full project. For observing the validation messages, go to the "Validation Messages" tab of "Model Tree" tab which exists in Left Panel.

5-4- Simulation

There are two options for simulating:

- Start Simulation
- Simulation Configuration

After defining all reservoir and model properties, you can start to simulate the model using "Start Simulation Button". Choosing this button will lead user to the window of Figure 209. There are six buttons of "Start", "Stop", "Restart", "Pause", "Live Results" and "Cancel" in the simulation window. Initially only three of them are active (Start, Restart and Cancel). Clicking on start button simulation will start (Figure 210). If you have restart file, you can restart the simulation using restart button. You can unlock the window using  to see desired part of simulation log during run. An estimation of remain time is displayed during run, in this window. User can filter the messages which want to see during run using Message level list.

There are 8 choices for user in this list:

- Off
- Fatal
- Error
- Warn
- Info
- Debug
- Trace
- All

The number of displayed messages is increased from Off to All.

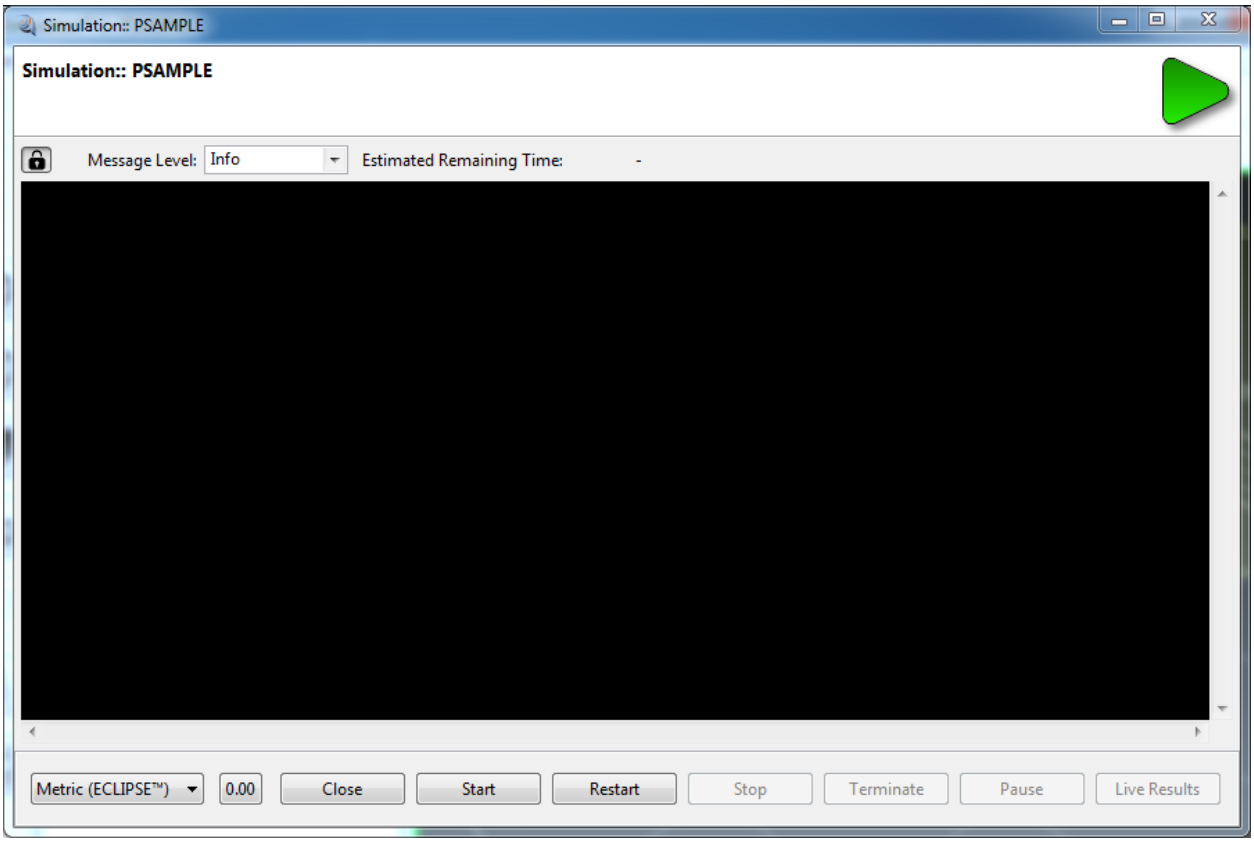


Figure 209- Simulation Window

During simulating, at first, inactive buttons are activated, using “Stop” button will stop simulation and reset to initial state. The “Pause” button pause the simulation. In this case if you start the simulation again the simulation will continue from paused time.

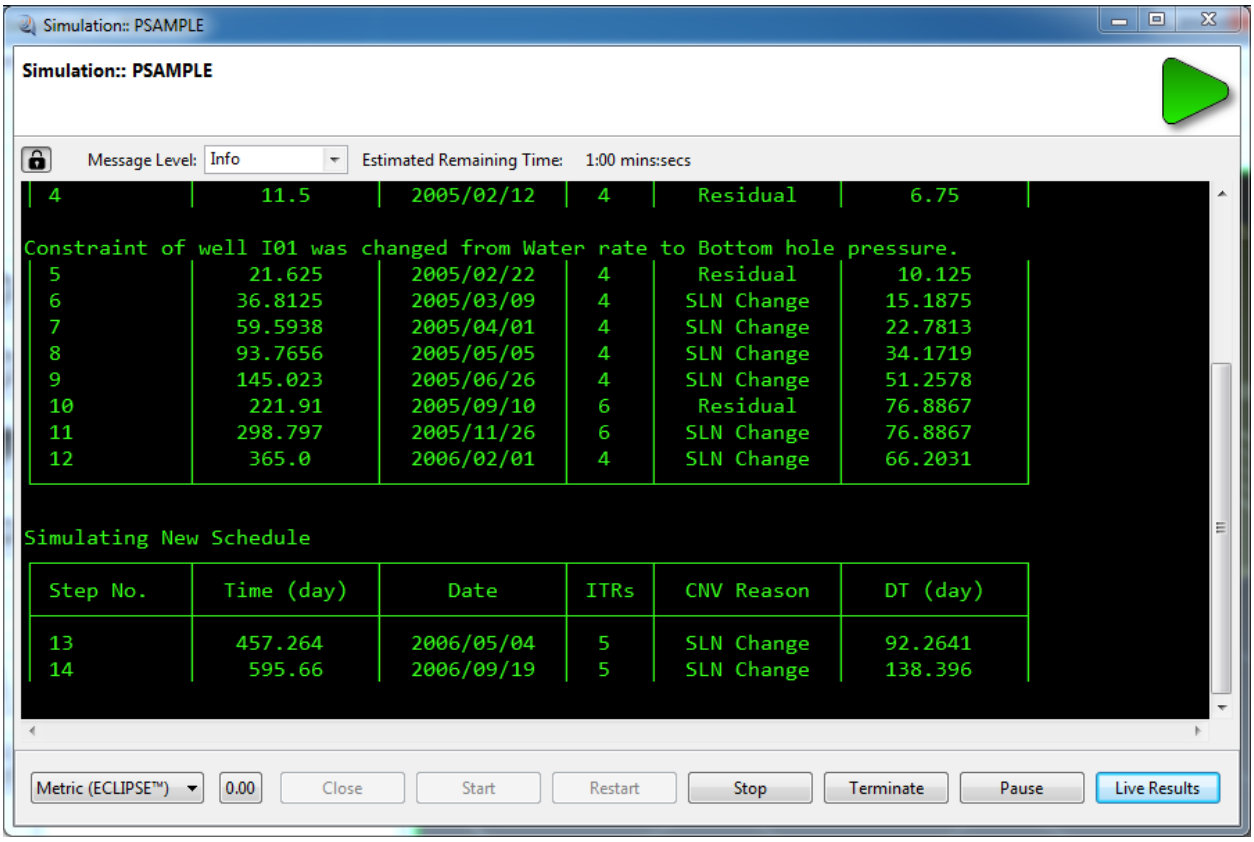


Figure 210-Simulating the Sample Model

The button of "Live Results" is used to display results during simulation, lively. In this window user can define refreshing time of the results from 1s to 10s (Figure 211). All options of plot and axis settings are available for this plot (like RETINA other plots) using right click on plot.

The  button is used to configure simulation. The Connector type, result folder name format, validation type and file connector steps can be edited by user in this window (Figure 212).

User can select "Simulation log section" parameters to write their values to the log file in the project path. Note log file and console files are written in the project path and are available for various purposes.

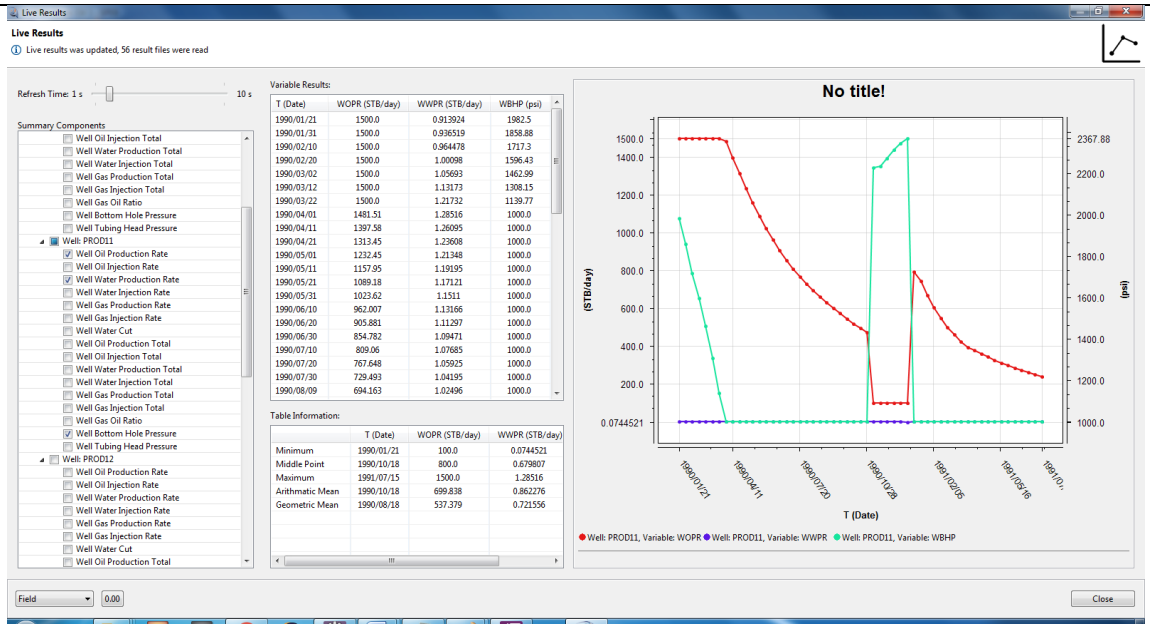


Figure 211- Live Results

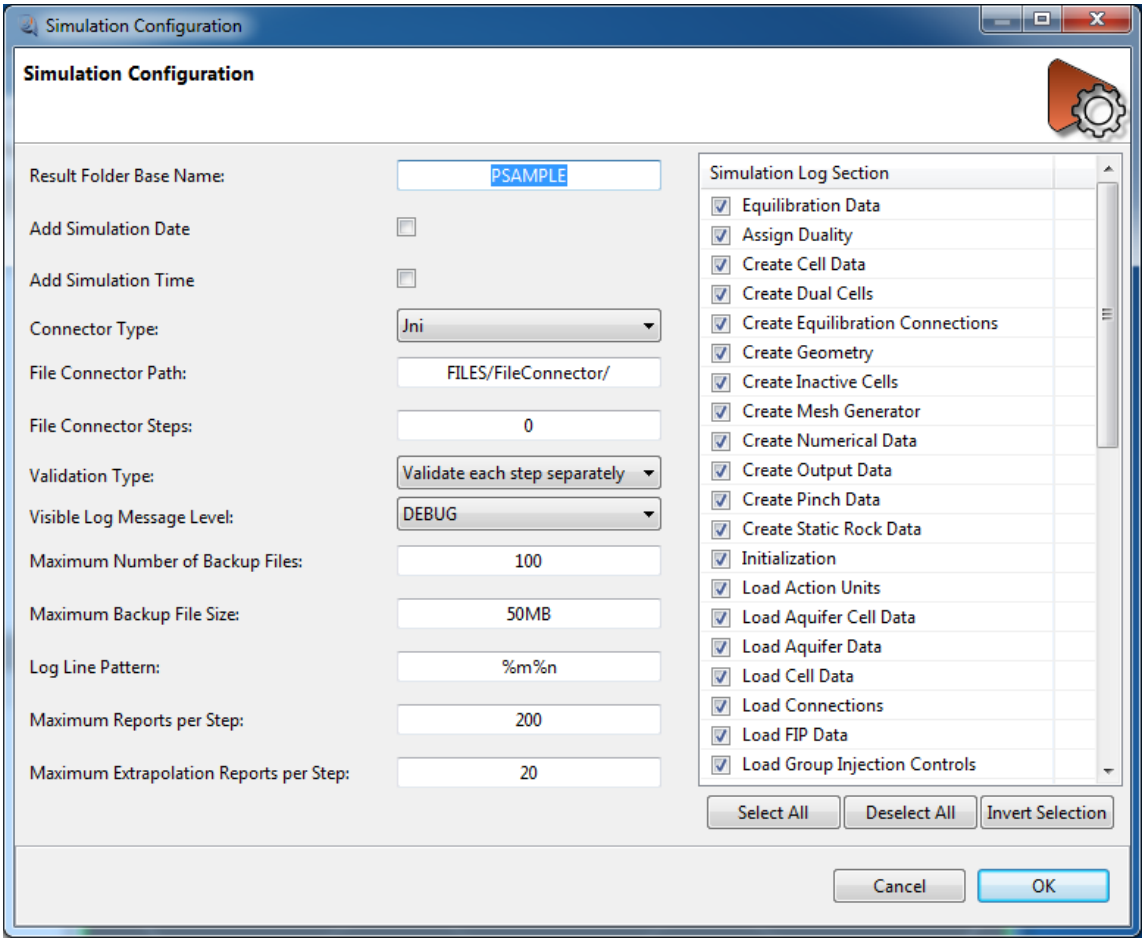


Figure 212- Simulation Configuration

5-5- Tools (Result Viewer)

Open Results Viewer

User can use “Open Result Viewer” for viewing multiple results of RETINA or ECLIPSE. The plot result viewer window is displayed in Figure 213.

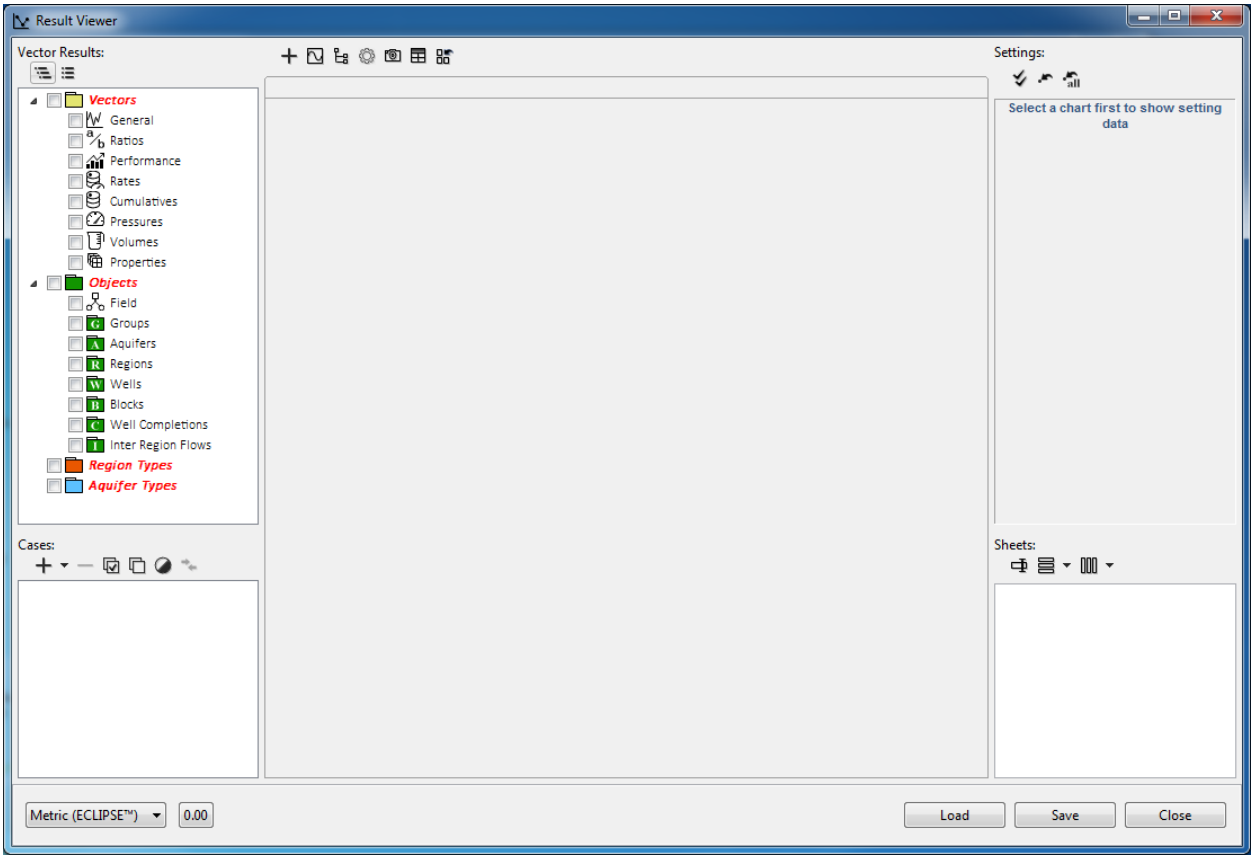


Figure 213- Result Viewer

There are five separate sections in the result viewer:

- Vector Results
- Cases
- Setting
- Sheets
- Main View

1) *Vector Results:*

There are four main parent nodes of: Vectors, Objects, Region Types, and Aquifer Types with red font color in "Vector Results" section. Vector parent node includes the following child nodes:

- General
- Ratios
- Performance
- Rates
- Cumulatives
- Pressures
- Volumes
- Properties

Objects parent node contains following child nodes:

- Field
- Groups
- Aquifers
- Regions
- Wells
- Blocks
- Well Completions
- Inter Region Floes

Aquifer types include the analytical and numerical aquifers and region type includes FIP… regions.

There are "expand all" and "collapse all" buttons in this section which user can use them.

2) *Cases:*

Three options are available in this section, for importing the requested results using "+" button. The imported case names are editable by clicking on each of them.

- Import RETINA Result file
- Import ECLIPSE Summary File (SMSPEC)
    - This option can be used for all multiple, unified, formatted and formatted ECLIPSE binary result files.

- Import ECLIPSE RSM File

a) *Import RETINA Result File:*

In case of importing RETINA result file, a dialog is opened in which the result file path must be entered; note that the extension of the retina result file is ".rri", so user must select a desired .rri file for importing to the result viewer. After importing the desired RETINA result file, a RETINA case and blank sheet are added to the result viewer (Figure 214). For the .rri files, a prefix of ***RETINA*** is added to the result file name in cases section. User can import multiple results and select them. There is a remove icon of "-"in this section; user can remove cases one by one or multiple using this button. The multiple removing options are available using either "Ctrl+right Click" on cases or "Shift+right Click" and then "-";

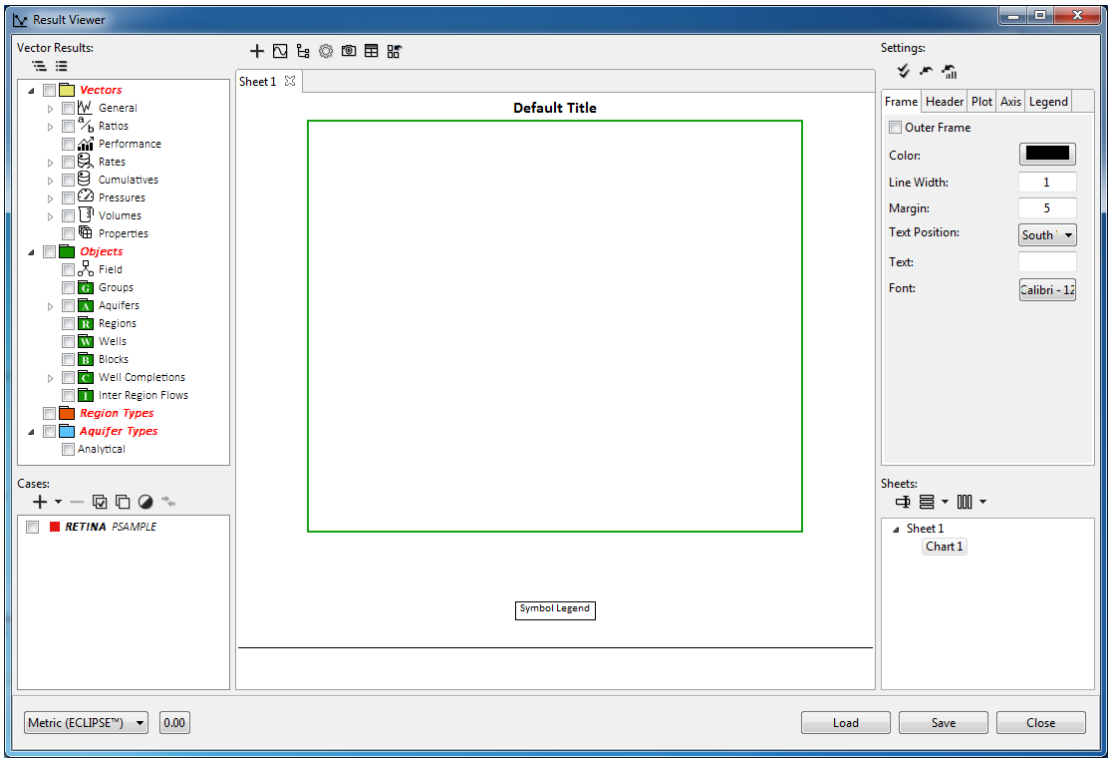


Figure 214 Result Viewer in Case of Imported RETINA Result File

b) *Import ECLIPSE Summary File (SMSPEC):*

In case of importing ECLIPSE summary result file (binary result files), a dialog is opened in which the result file path must be entered; in this dialog the unit of the result file must be specified using the related combo box. Note that the

extension of the ECLIPSE binary files is either SMSPEC or FSMSPEC. So user must select a desired ECLIPSE binary result file for importing to the result viewer. After importing the result file, a ECLIPSE case and blank sheet are added to the result viewer. For such result files, a prefix of ***SMSPEC*** is added to the result file name in cases section. User can import multiple results and select them. There is a remove icon of "-"in this section; user can remove cases one by one or multiple using this button. The multiple removing options are available using either "Ctrl+right Click" on cases or "Shift+right Click" and then "-";

c) *Import ECLIPSE RSM File:*

In case of importing ECLIPSE RSM file, a dialog is opened in which the result file path, simulation start date and model dimension, must be entered. If result viewer opens from RETINA opened project, the dimension and start date will be get from project data. Check the unit of imported result file and the importing dialog. Note that the extension of the ECLIPSE RSM files .rsm. So user must select a desired file for importing to the result viewer. After importing the result file, an ECLIPSE case and blank sheet are added to the result viewer. For such result files, a prefix of ***RSM*** is added to the result file name in cases section. User can import multiple results and select them. There is a remove icon of "-"in this section; user can remove cases one by one or multiple using this button. The multiple removing options are available using either "Ctrl+right Click" on cases or "Shift+right Click" and then "-";

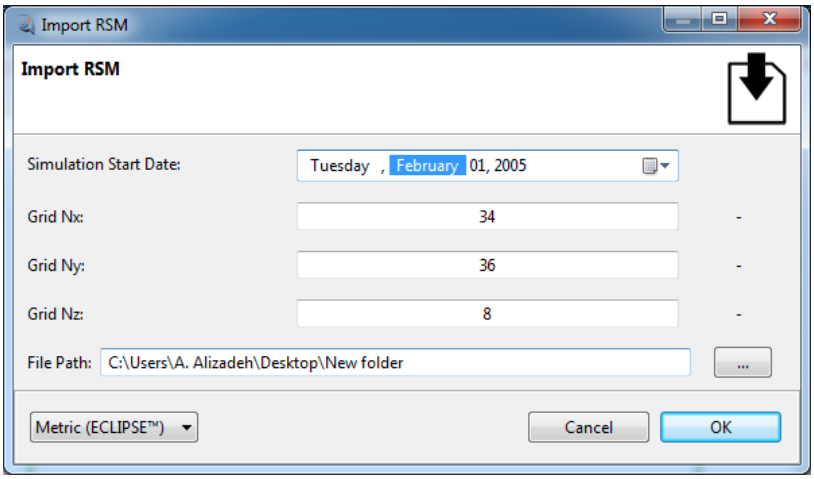


Figure 215 Import RSM File

There are options of select all, unselect all, and invers selection in the cases section. There is a useful facility for comparing results. The comparison buttons

are inactive initially; it will be activated just in case of selecting two cases. You can use comparison windows to compare two selected results (Figure 216). The relative error, minimum and maximum differences and average differences for each point is reported and plotted in comparison window.

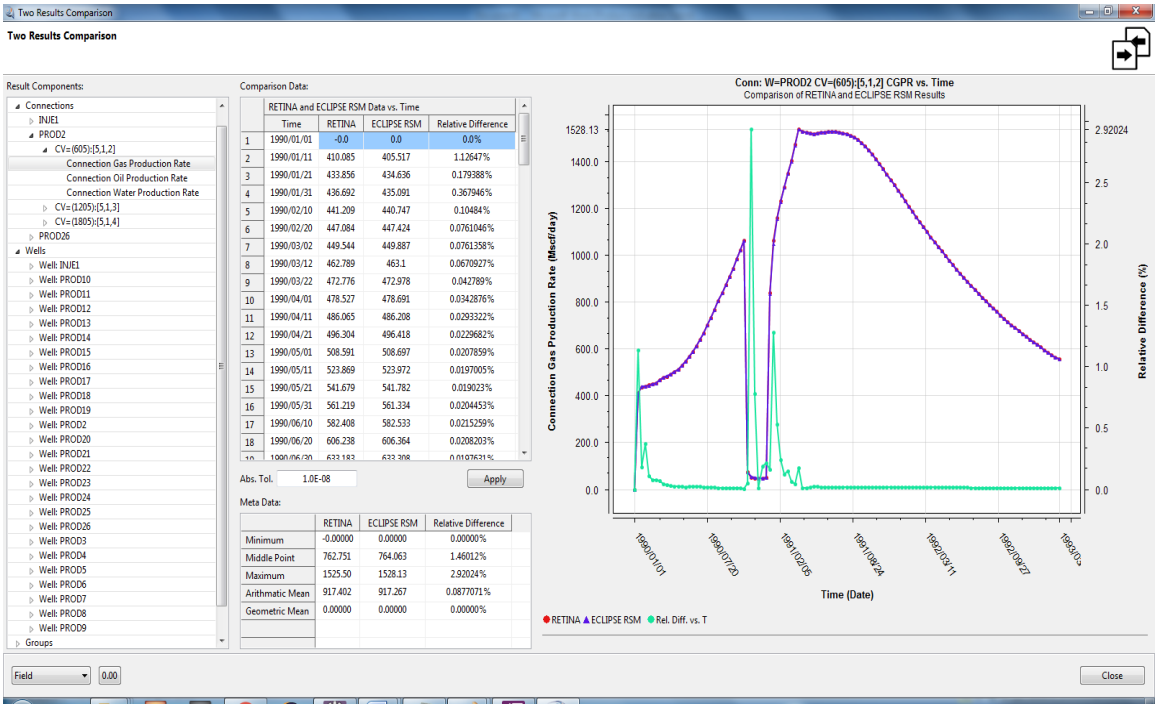


Figure 216 Comparison Window

3) *Settings:*

There are five tabs in this section:

- Frame
- Header
- Plot
- Axis
- Legend

Frame Setting:. Show or hide actions for the frame of the observable sheet can be set using the "Outer Frame" check box. The color, frame line width, margin, text position, text, and font of the text, are editable in this tab.

Header Setting: Show or hide actions for the each chart header and frame can be set using the "Visible" and "Frame" check boxes, respectively. The position, offset, inner space, color, font, text, and chart description text and font are editable in this tab.

Plot Setting: the content of this setting is only available in case of adding plots in the result viewer. The name of plots (the default format of the plot name in RETINA is [(Case Name) Object-Vector vs. Time/Date]), color, line width, symbol type and size, are configurable in this tab.

Axis Setting: there are two tabs of "General" and "Detail" in the Axis setting; in the general tab user can show/hide the axis frame, change it color, change the align of the axis, and define the minor tick numbers. In the detail tab user can configure the following settings for each axis:

- Name
- Side
- Minimum value
- Maximum value
- Maximum tick number
- Auto step
- Step value
- Inver axis facility

Note: there are multiple Y-axis features in the RETINA result viewer.

Plot Legend Setting: the visibility, position, offset, font size, font type, font color, legend frame show/hide, legend frame color, legend background color, and title are configurable using this tab.

There are three options of "Apply Settings to All" (All means all of the charts), "Reset Setting" and "Reset All Settings are available in "Setting" section.

4) *Sheets:*

User can define up to 6 charts in a sheet assigning the row and columns in the sheets section. Also user can edit name of each sheet. For example there are 4 charts in sheet1 in the Figure 217. Each chart will activate by clicking on it and displayed in green color.

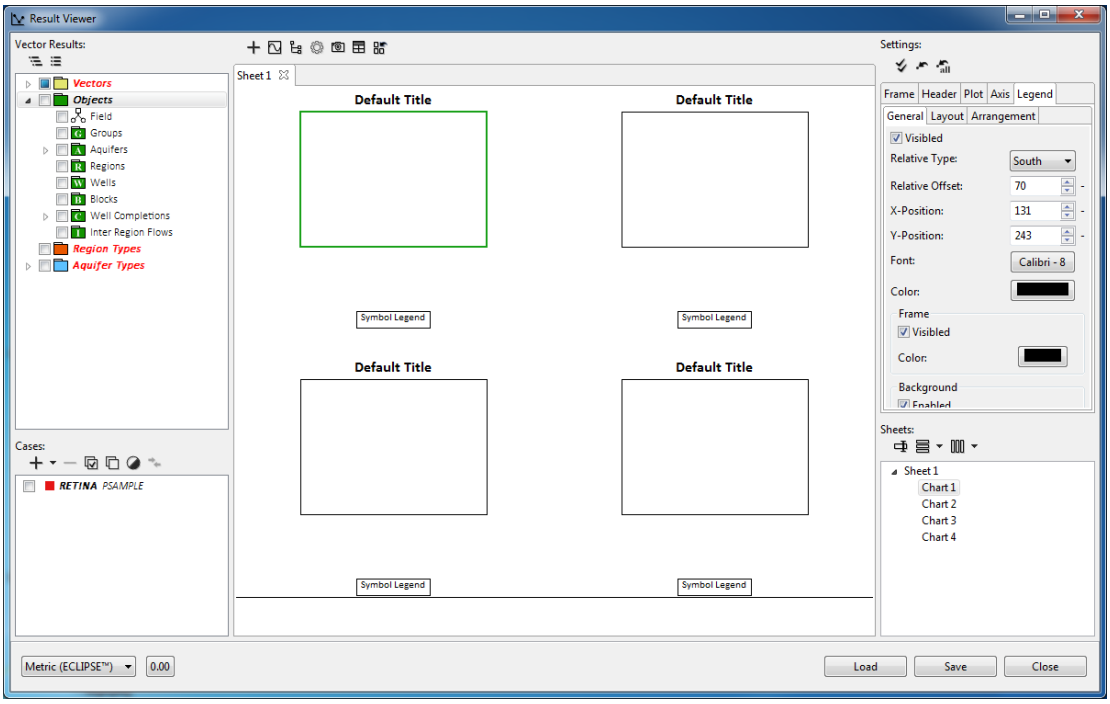


Figure 217 Defining Multiple Charts in a Sheet

5) *Main View:*

User can observe the charts and sheets in this view; it contains the following facilities:

- Add new sheet
- Fit diagrams
- Show/hide left panel
- Show/hide setting panel
- Save chart as .png file
- Show table
- Re-layout sheet

If user hides left panel and/or setting panel, the re-layout facility can be used to extend the charts to full view.

5-5-1-How to Use Result Viewer

Follow the following instructions to use the result viewer:

1- Import the result(s) data using import button of the case section

2- Set the numbers of chart in sheet (by default 1 chart in displayed in each sheet)
3- Check the box of the each case which you want to see the results
4- Select desired case(s)
5- Select the desired object(s)
6- Select desired vector(s)
7- Assign a desired X-Axis by right clicking on each vector
8- Save the result viewer (.rrf file)
9- Load .rrf file if you need

All charts are freely moved by the mouse, zooming in or out and band zoom facility are available.

The Figure 218 displays the results for multiple vectors and multiple objects for one case.

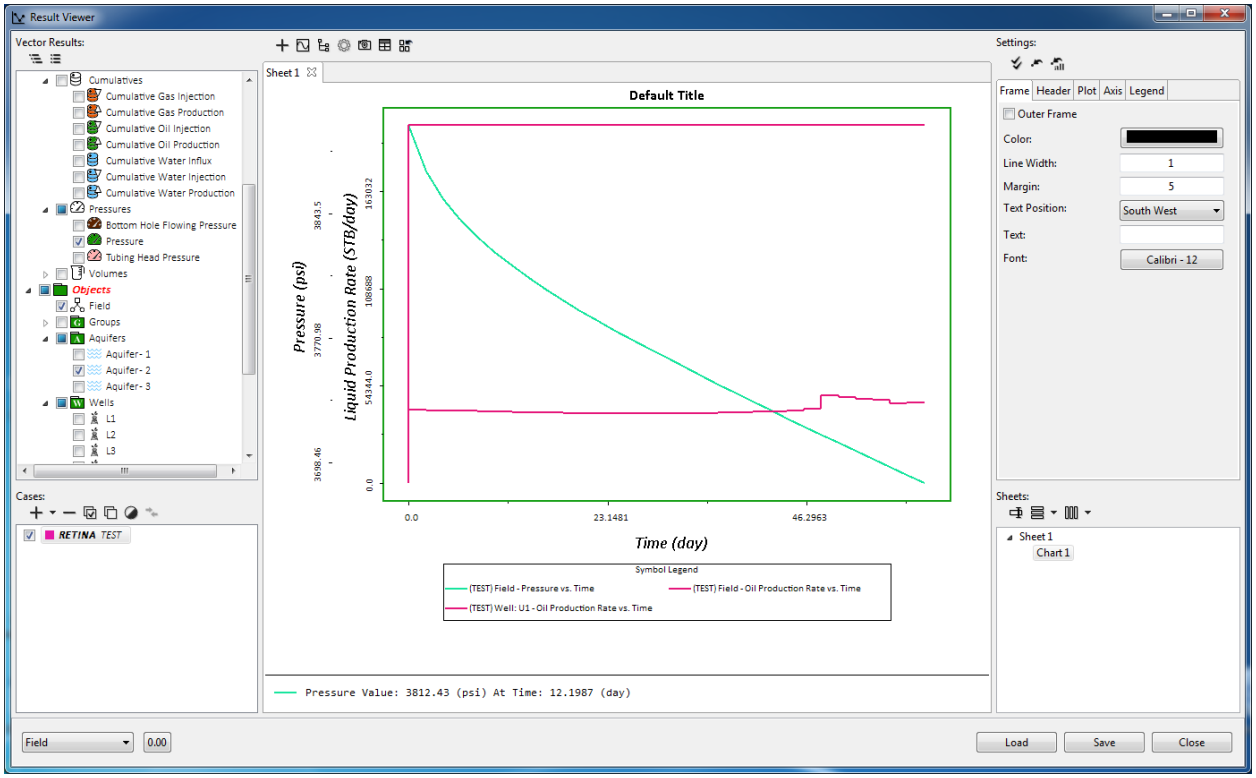


Figure 218 Result Viewer for One Case and One Chart

The Figure 219 displays the results for multiple cases in multiple charts.

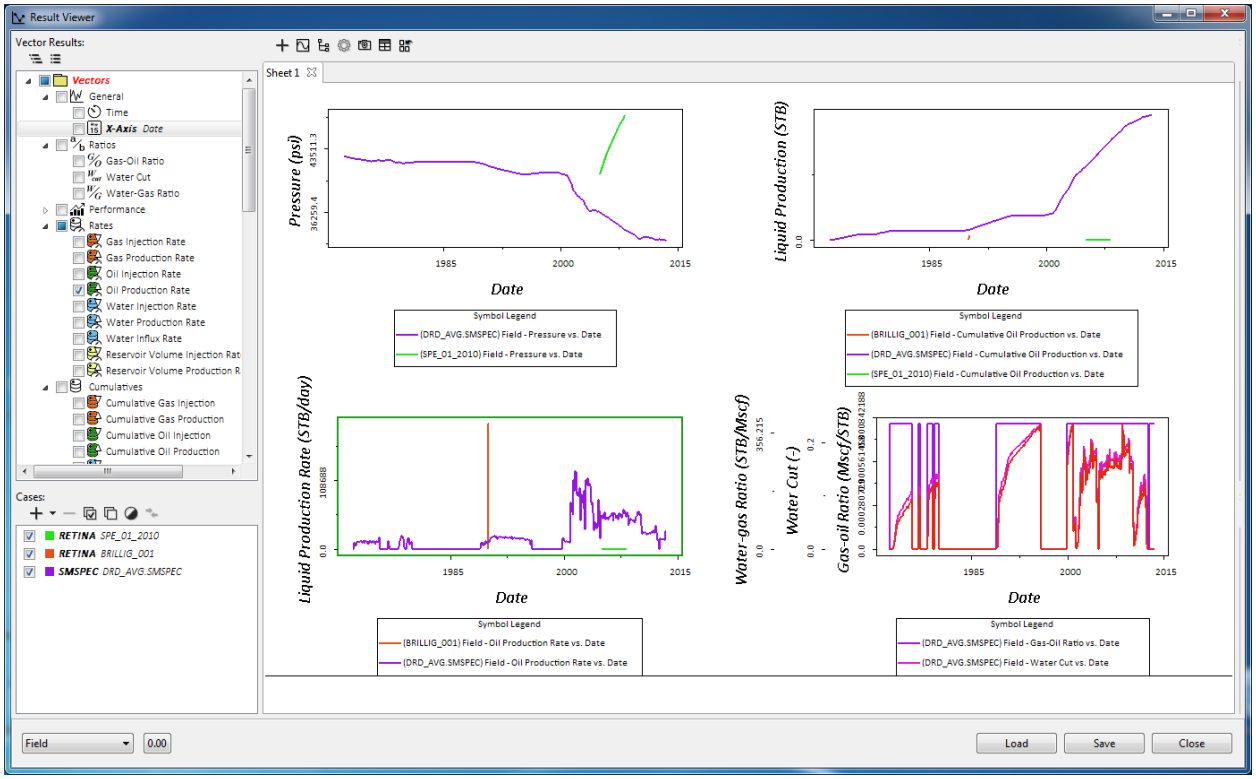


Figure 219 Result Viewer for Multi Cases and Multi Charts

5-6- Options

 Input/output Path

 Validation

 Property Template

The  button is used for defining input/output options (Figure 220). User can select one of the following I/O options in this window:

- Last User Path
- Project Path
- Custom Path

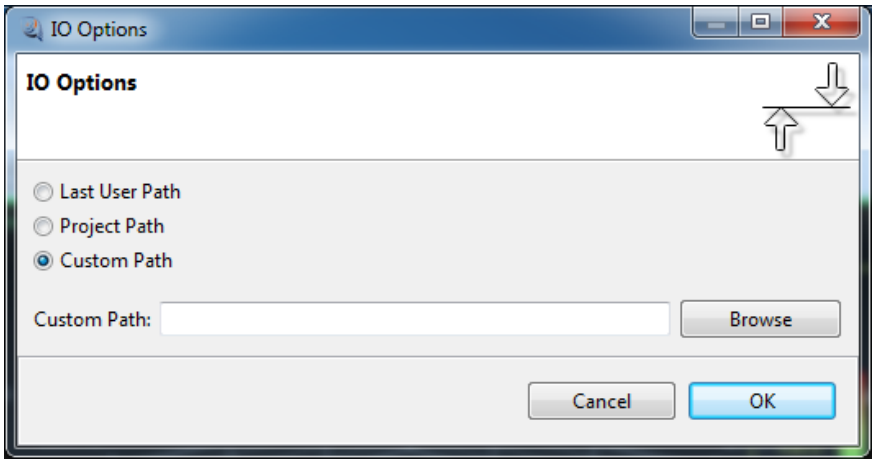


Figure 220- IO Options

User can set validation state (Figure 221) to "off", if he/she does not want to see validations during creating a model, note that if there is validation error in created or loaded model the simulation is prevented from starting regardless to Validation option state.

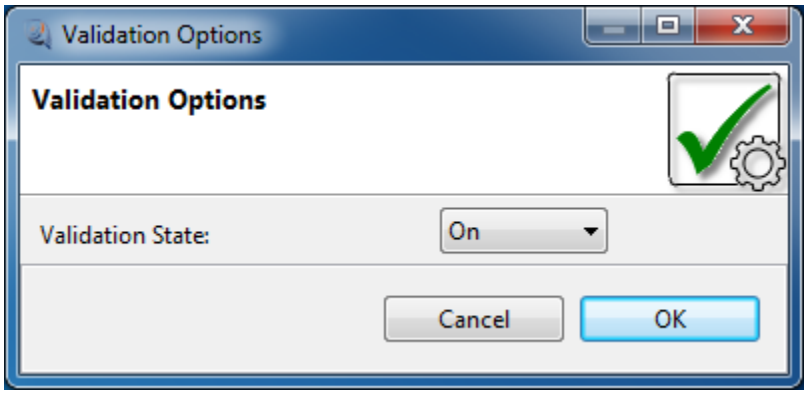


Figure 221-Validation Options

The property template ( ) button is used for defining template for each property which is applied to default legend of the 3D view. It contains two tabs of Info and color for each property.

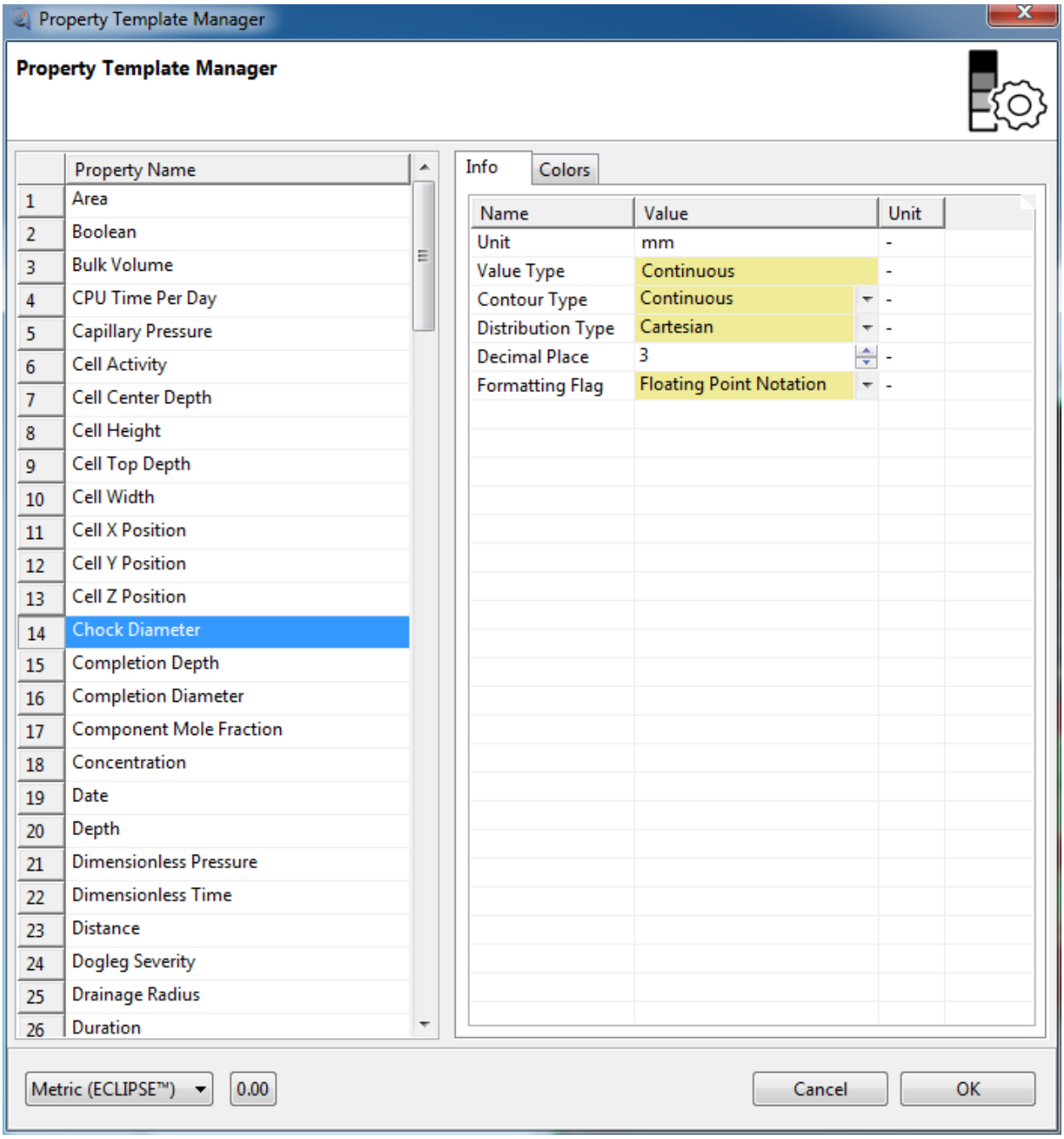


Figure 222- Property Template Manager Info Tab

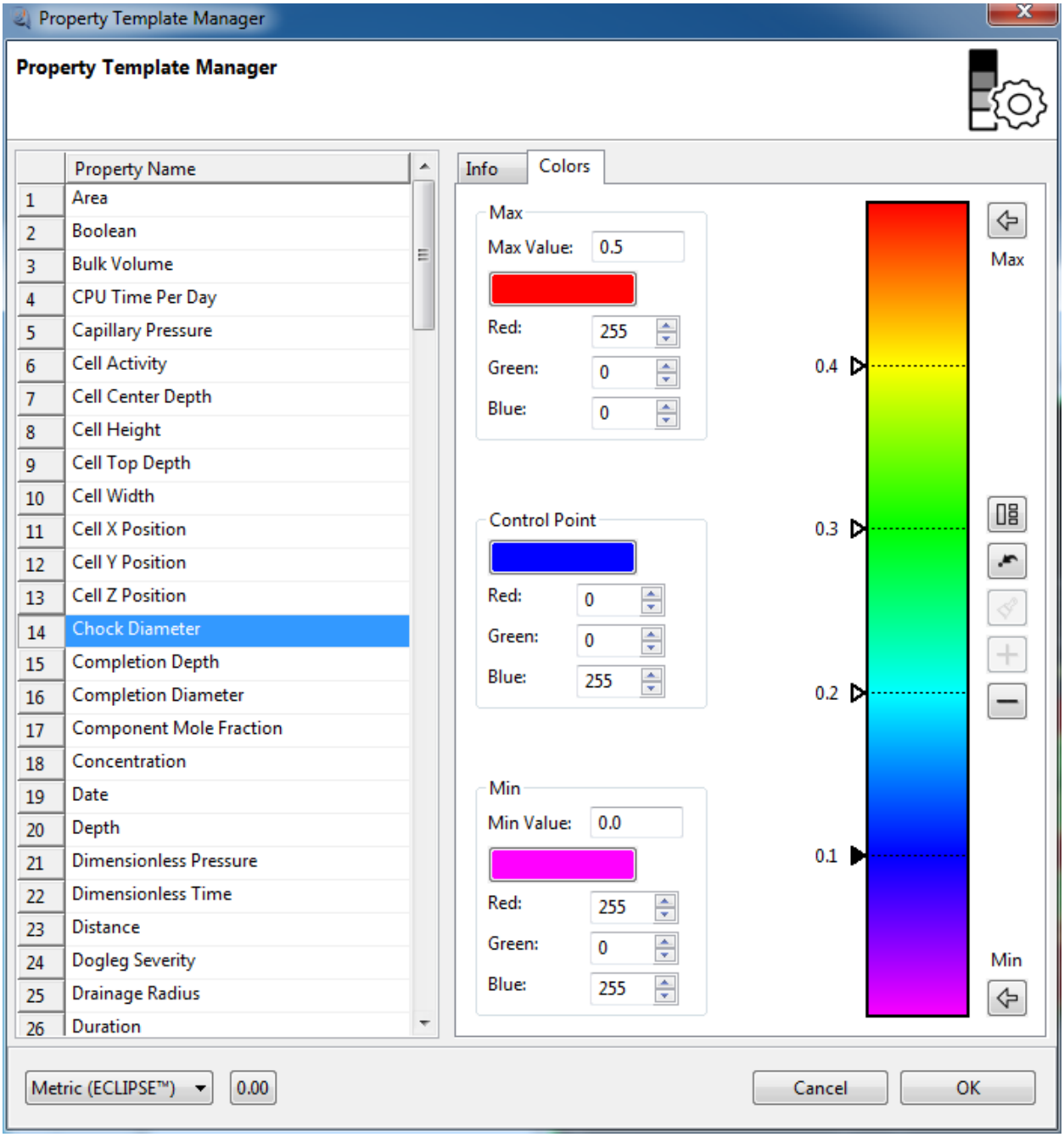


Figure 223 Property Template Manager Colors Tab

5-7- Help


Help Document


Technical Document


Keyboard Shortcuts