



A01 • Required data files

RubGS01_FieldMap.bmp, RubGS01_TopHorizon.asc, RubGS01_ShaleThickness.asc, RubGS01_SandPermeability.asc, and RubGS01_SandPorosity.asc.

B01 • Model Construction: New Document Creation and PVT Initialization

B01.1 • New Document Creation

- Accept all default values.

B01.2 • PVT Initialization

- Click on the PVT icon in the Simulation page to edit the PVT definition, and set the fluid type to saturated oil with water.
- Proceed with the defaults.

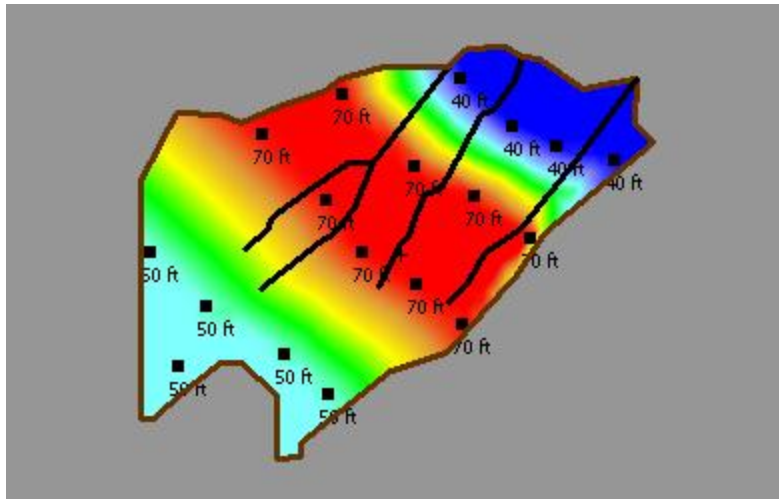
B02 • Model Construction

B02.1 • Reservoir Lateral Definition: Contour and Faults

- Load Bitmap: RubGS01_FieldMap.bmp.
- Set the scale.
- Set the reference point.
- Draw the contour.
- Draw the sealing faults.
- Untick the checkbox "image" in the display settings in order to visualize the current top view of the reservoir.

B02.2 • Reservoir Vertical Definition

- In the "Geometry" change the number of layers to 3, and in the table rename the top layer (Layer 1) to "Upper Sand", the middle layer to "Shale" and the bottom layer (Layer 3) to "Lower Sand".
- Change the top horizon of the "Sand" layer from Constant to Data Set.
- Enter the data points that will be used to define the surface of the top horizon from the file "RubGS01_TopHorizon.asc".
- The interpolation method should be Linear interpolation.
- Change the thickness of the Sand layer from "Constant" to "Data set", pick the thickness points on the screen. Impose a thickness of 50 ft in the south-west of the reservoir and 40 ft in the north-east, with a maximum thickness of 70 ft in the center, as displayed below:



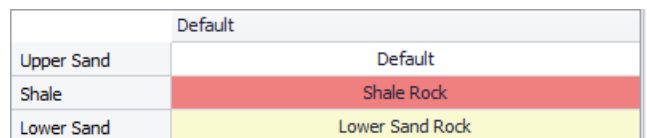
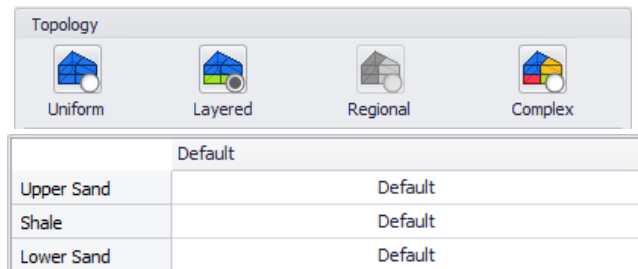
- Load the thickness of the Shale layer from the "RubGS01_ShaleThickness.asc" file.

A back up file at this point is "01 Geometry loaded.krp".

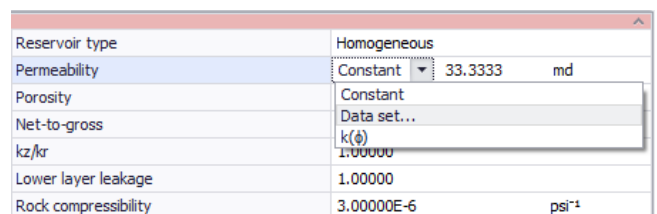
B03 • Model Construction: Properties Definition


B03.1 • Defining Petrophysical Properties

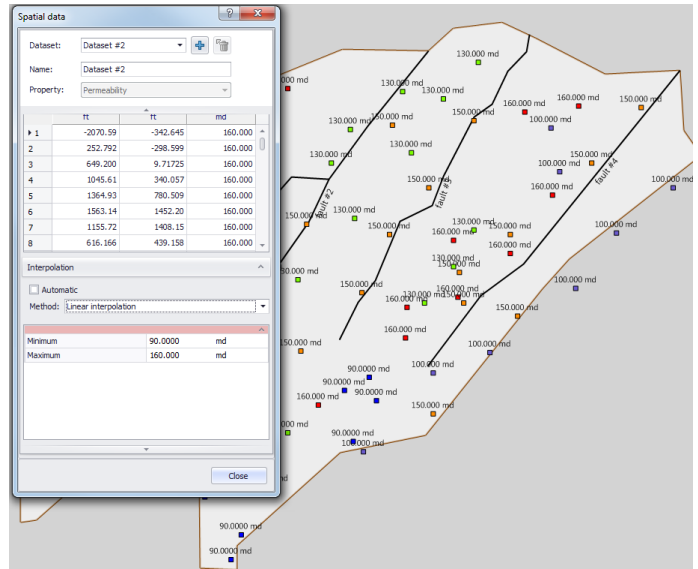
- Change the reservoir definition to "Layered" as shown opposite.
- Select the Shale layer in the table located on the left-hand part of the dialog:
- Create a new property set, and rename it "Shale Rock".
- Create a specific property set for the Bottom layer ("Lower Sand Rock").



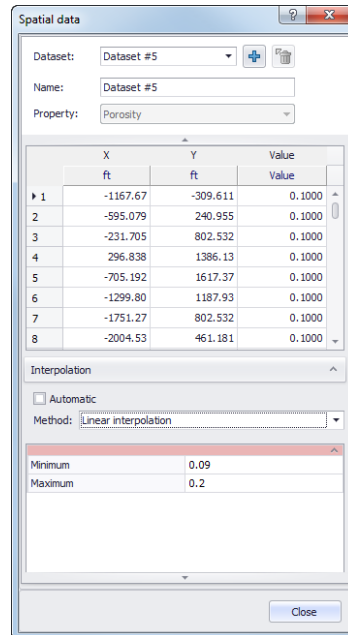
- In the Default property table change the permeability (k) type from "Constant" to "Data set":



- Load () "RubGS01_SandPermeability.asc".
- Select linear interpolation.



- Define the default porosity from "RubGS01_SandPorosity.asc" file.
- Select linear interpolation.



- Select the shale rock property set.

Default	
Upper Sand	Default
Shale	Shale Rock
Lower Sand	Lower Sand Rock

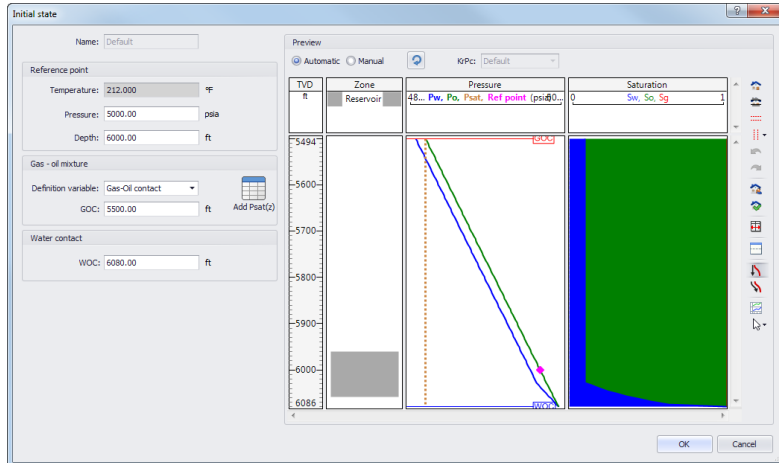
- Enter the following properties for this rock type: permeability of 0.001 mD, porosity equal to 0.05, and leakage factor (modeling a reduction of the layer to layer connection) of 0.05:

Reservoir type	Homogeneous	
Permeability	Constant	1.00000E-3 md
Porosity	Constant	0.05
Net-to-gross	Constant	1.00000
kz/kr	1.00000	
Lower layer leakage	0.05	
Rock compressibility	3.00000E-6	psi ⁻¹

- Define a description of the initial state and KrPc properties over the complete reservoir.

B03.2 • Defining the Initial State


- Select the Default property and use the “Initial State” button to edit the global reservoir initial state description.
- Set the reference initial pressure equal to 5000 psia at 6000 ft.
- Leave the reservoir temperature unchanged (212 °F).
- Select the definition type as ‘Gas-Oil contact’.
- Change the GOC to 5500 ft and the WOC to 6080 ft.

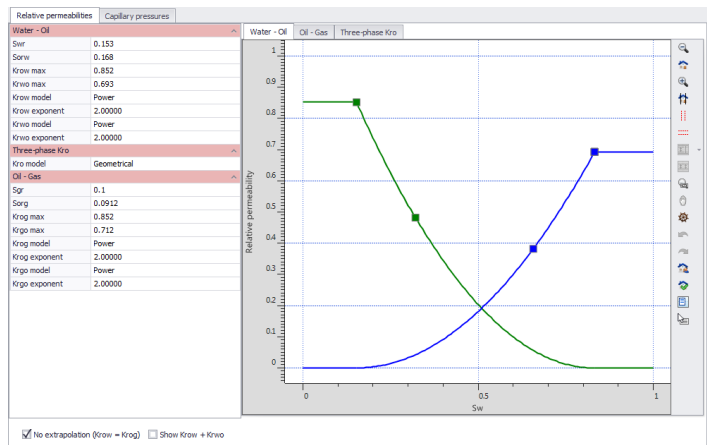


In this case there is no gascap.

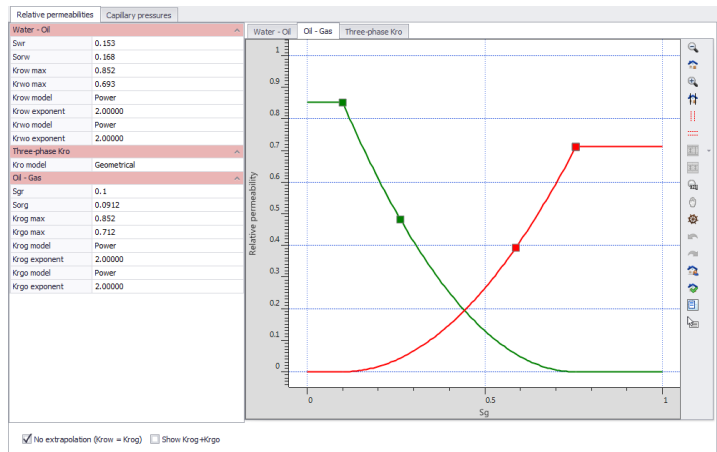
B03.3 • Defining the KrPc

- Edit the reservoir relative permeabilities and capillary pressures.
- Check the “No extrapolation” option.

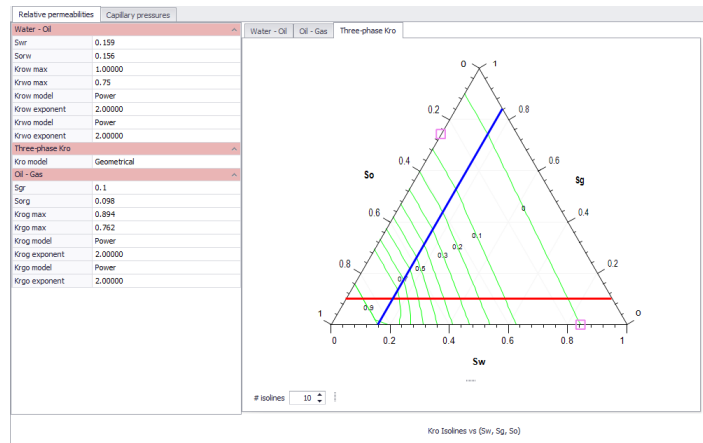
- On the tab “Relative permeabilities” graphically impose the (approximate) shape to the “Water-Oil” system relative permeabilities by dragging the end points () with the mouse:



- Select the "Oil-Gas" tab to proceed similarly with this new set of curves:



- Move to the "Kr Three-Phase" tab to edit the total oil relative permeability, and change the Three-phase model to "Geometrical":



- For the definition of the water-oil and oil-gas capillary pressure curves, we will keep the default curves.

A back up file at this point is "02 Properties loaded.krp".

B04 • Model Construction: Building the Wells

B04.1 • Wells Creation and Naming

- In the 2DMap display the reservoir bitmap ("show bitmap"):

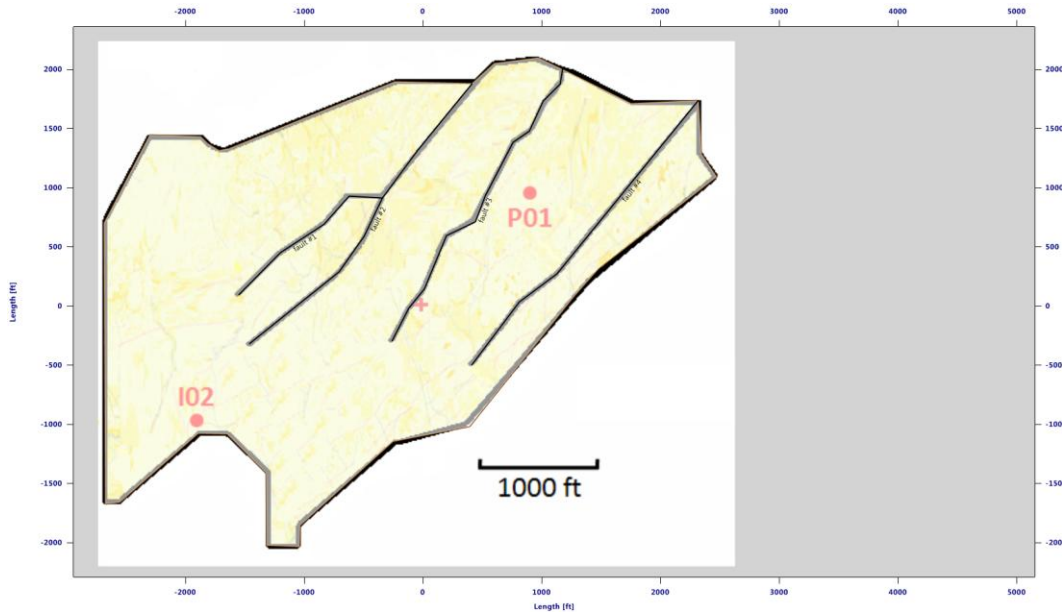


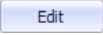


Fig. B04.1 • Bitmap display before wells creation

- Create the vertical wells in the positions P01 and I02.
- Open the Well definition dialog.
- Rename the wells with the names displayed in the bitmap- P01 for the eastern well, and I02 for the one in the western part of the reservoir.

Well name
P01
I02

B04.2 • Perforations and Controls for P01

- Select the P01 well in the "Wells" dialog. Click on the button .
- Define the "Perforations" in the tab "Definition" - "Cross-Section View" in the top (Upper Sand) layer.
- Click on  to modify the perforation interval: drag the bottom node to set it in the top layer only, right click on the perforation to span the entire top layer.
- Activate the checkbox "Define intake" to edit the wellbore model that will be applied to compute the pressure drop along P01.
- Set the use of "All trajectory".
- Define the production schedule for the well P01 (tab "Schedule").
- "Add" a new control for this well .
- Change the target type to "P(Surf)".
- Enter 2000 psia.
- Keep the constraint type to "QTot(BH)" (total bottomhole rate).
- Enter 15000 B/D for the value of this constraint.

B04.3 • Perforations and Controls for I02

- Select I02 and click on "Info" to add a perforation for this well.
- Change the well type to complex: change MD start (starting measured depth for the perforation) to 6083 ft, and MD end to 6096 ft (in case of the vertical well $zw=17.0357\text{ft}$, Perforation length=13ft).
- Check that the perforation is located in the lower part of the reservoir in the "Cross-Section View".
- We do not use a wellbore model for this well.
- Click now on "Schedule" tab to create the following water injection control.
- Change the mode from Producer to Injector.
- The target type to "QWat(BH)" to a value of -5000 STB/D.
- Controlled by a maximum surface pressure of 10000 psia.
- Finally, click on OK to exit the "Reservoir – Wells" dialog: both wells are defined.

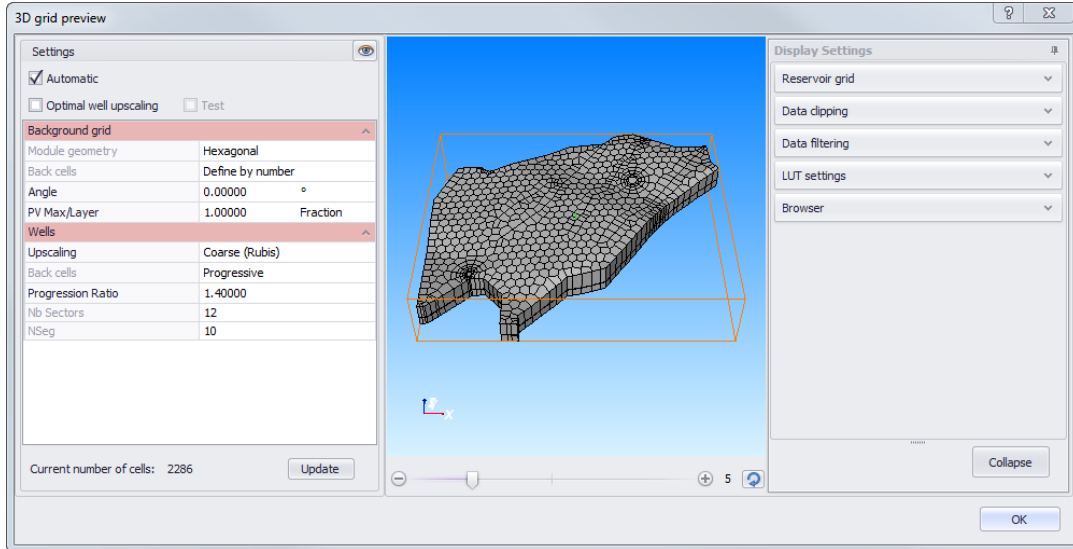
A back up file at this point is "03 Wells loaded.krp".

C01 • Simulation: Grid Construction

C01.1 • Constructing the Grid



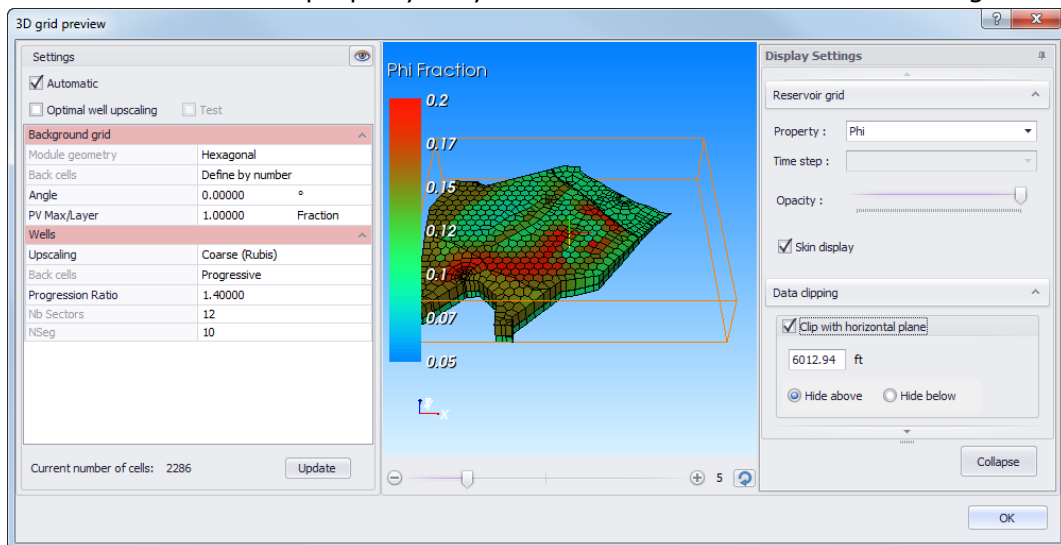
- Click on the "Grid" button on the top ribbon to edit the "3D grid preview" dialog.
- Note: check the "PV Max/layer" to 1 in order to remove the automatic sublayering:



- Increase the vertical gain.
- Change now the Well "Upscaling" from Course (Rubis) to Fine (Saphir), click on Update
- Zoom on the first grid cells around the I02 well in the plot (explain grid geometrical settings).

C01.2 • Visualizing the Grid and the Petrophysical Properties

- In the "Display Settings"- "Reservoir grid" section change the property to be displayed to "Phi"
- Go to the section "Data clipping" to show a horizontal cross-section of the reservoir at a defined depth.
- The color scale of each property may be edited in the section "LUT settings".



C02 • Simulation: Run Settings and Problem Initialization

C02.1 • Initialization

- “Initialize” the run: define the type of results to be output. The initial state is computed, and a numerical upscaling is performed around all the wells.
- Maximize the 3D plot to display the initial reservoir pressure and saturation fields.
- Check in the Cross-section plot: switch to 2DX-Y, draw the polyline going through the wells (Vertical cross-section), 2D X-Z and select the polyline from the list, increase the vertical gain and select “Sw” as the property to be displayed.

A back up file at this point is “Initialized.krp”.

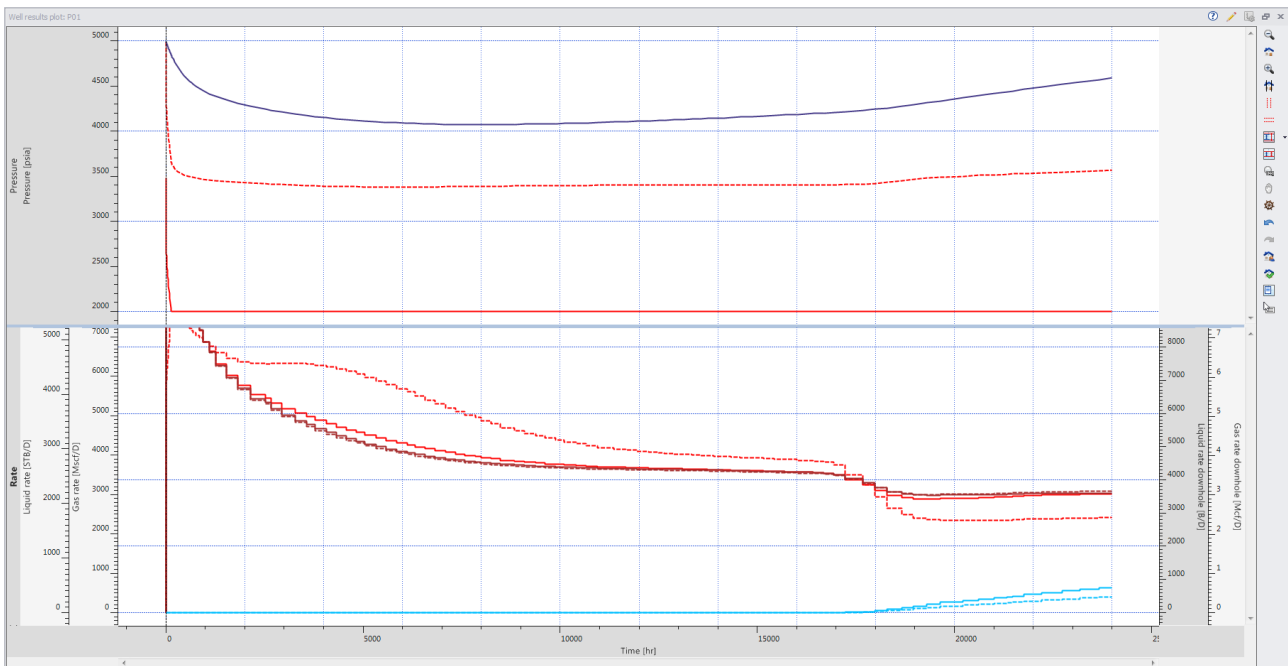
C03 • Simulation: Run Simulation and Results Visualization


C03.1 • Run Simulation

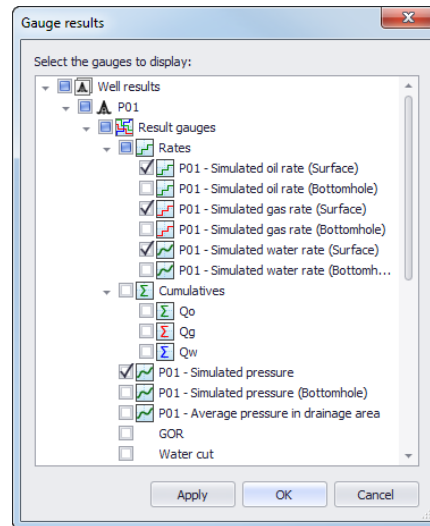
- Simulate: keep all defaults offered in Rubis, except for the run duration that needs to be reset to 1000 days (slightly less than 3 years).
- Start the simulation thread. It should take few minutes.

C03.2 • Looking at Well Results

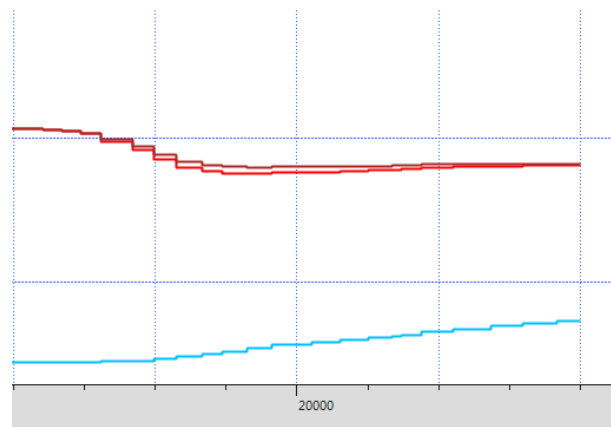
Well results plot (one can switch the well at the ‘Plot options ribbon’) contains the evolution of pressure and rates simulated at the wells. Maximize the “Well results plot P01”:




- Click on  in the 'Plot options' ribbon. In the following dialog choose to display only the surface oil, water and gas rates, along with the surface pressure:

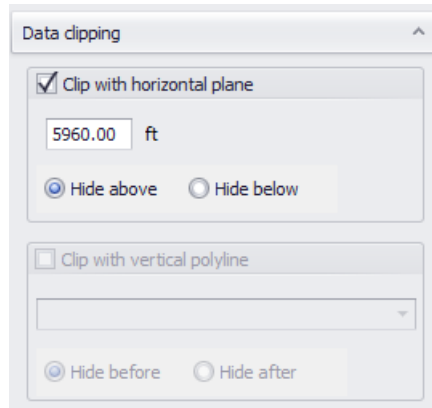


The curves remaining on the screen show that water breaks through at the producer after about 17000 hrs of production:

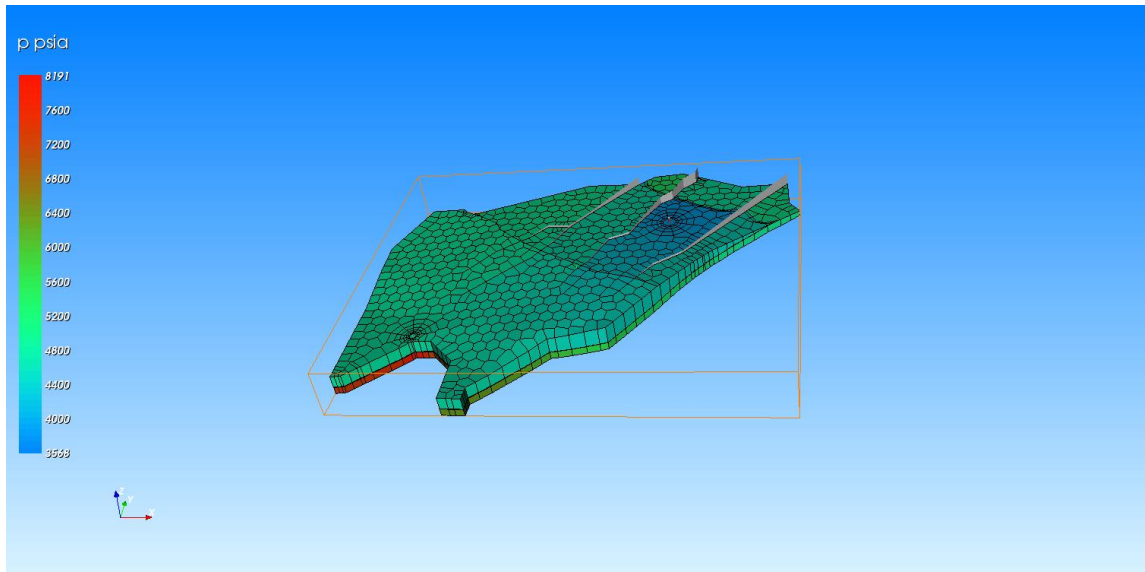


C03.3 • Looking at Field Results


- Maximize the 3D plot, and display the initial pressure field (Display Settings – Reservoir grid section- Property:P) by selecting the first output time ( in the "Navigation" ribbon).In the "Data clipping" section tick the checkbox 'Clip with horizontal plane', and input the depth of 5960 ft at which the reservoir will be cut:



- Display the last stored pressure field by clicking on . The final result should look like:

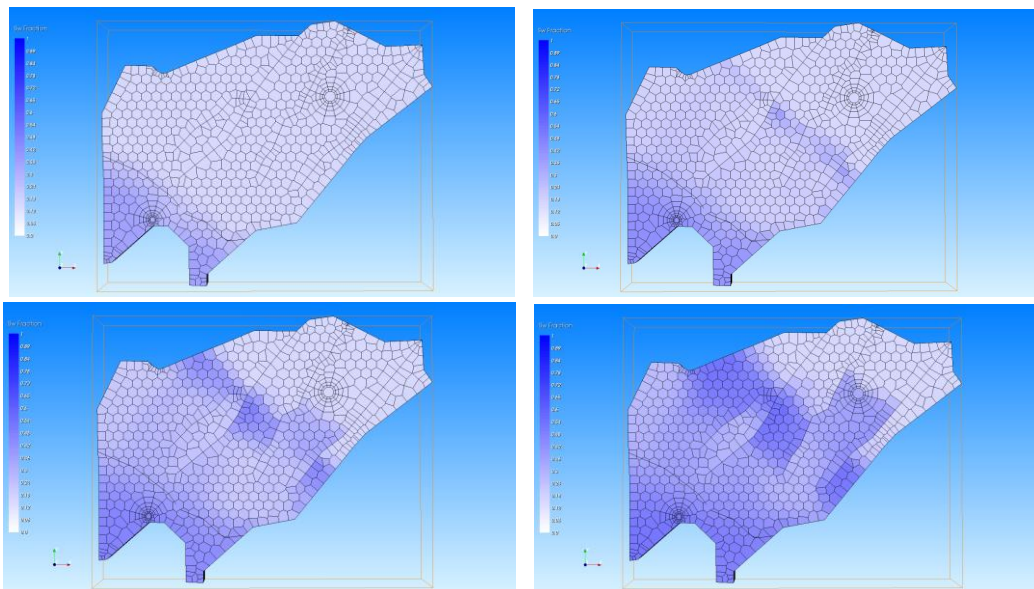


➤ Remove the horizontal clipping

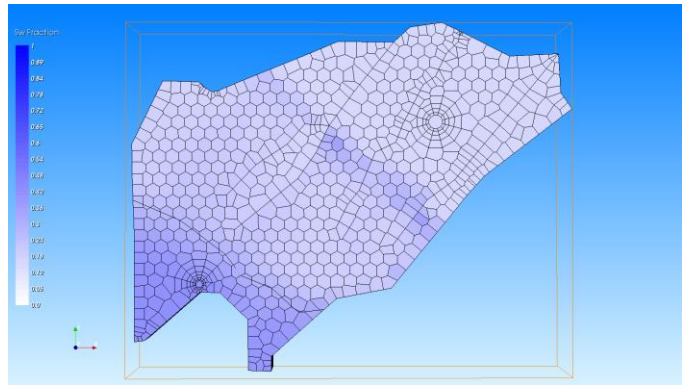
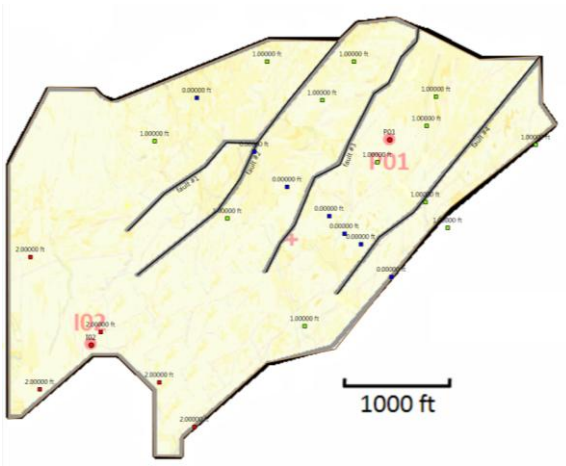
➤ Set "Sw" as the displayed property, show the initial condition with  , and click on to



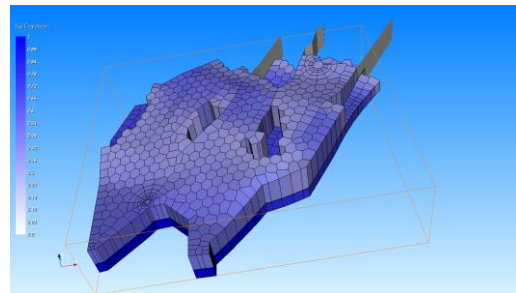
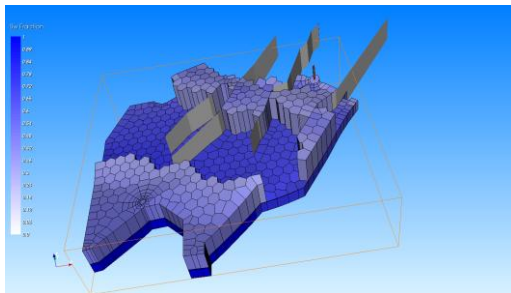
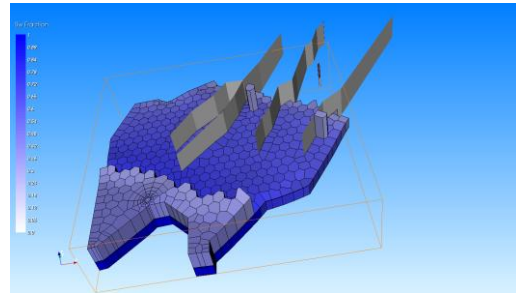
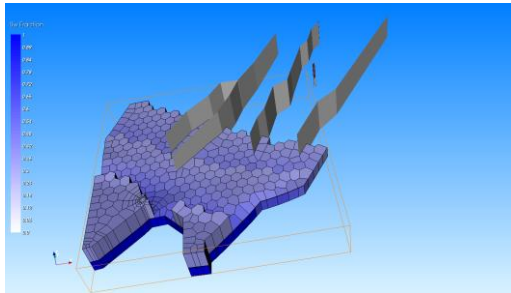
play-back the evolution of Sw with time:



➤ The animation and the final saturation field show that water invades the top layer by passing across the null-thickness region of the shale layer:



- To filter the cells within a certain gap of property values:
 - Go to the section "Data filtering" and select Sw to filter on
 - Input the bottom limit value ("Above min") equal to 0.3
 - Navigate through the different time steps in order to visualize water front propagation



- Create the vertical cross-section going through both wells:



- Click on 2D X-Y at the plots option ribbon
- Create the polyline going through 2 wells ("Vertical cross-section"- "Polyline editor")

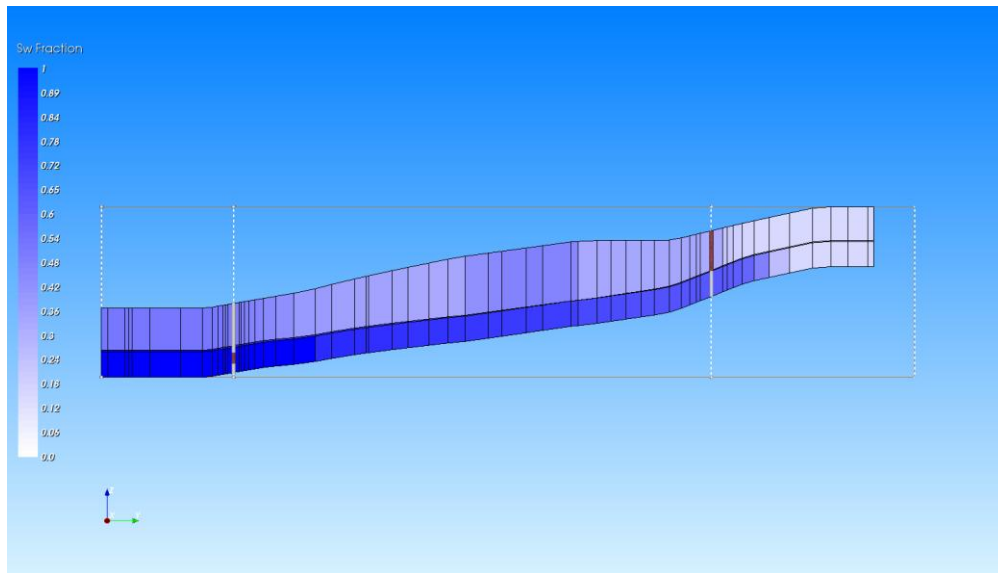


- Press on 2D X-Z and select Polyline#1 from the list

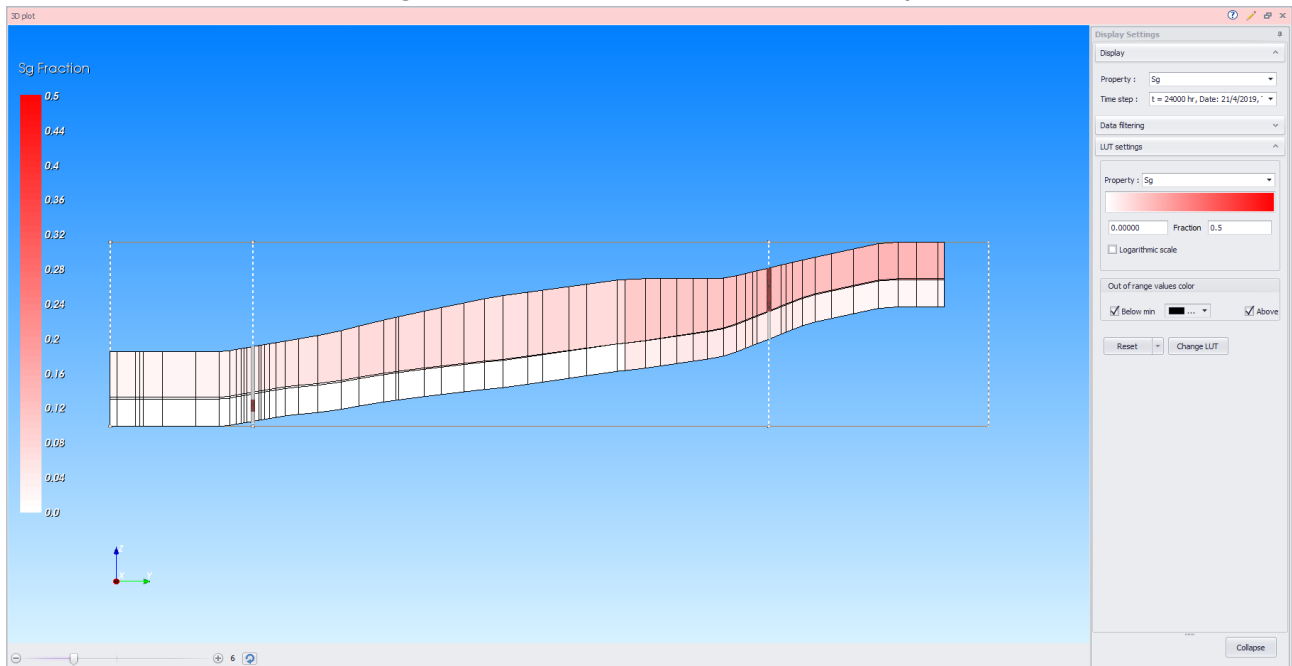
- Make sure that the display is set to water saturation and replay the simulation using



again.



- When the animation is complete, change the display property to Sg (modify the upper limit in the 'LUT settings' section for the better visualization):



As can be seen above, some free gas appears by pressure depletion around P01 during the simulation.