Tutorial 3: Constructing a PEBI simulation grid

Objective

The aim of this section is to demonstrate how a PEBI simulation grid can be constructed using the Unstructured Gridder option in ECLIPSE Office. The primary purpose of this tutorial is to familiarize you with the major aspects of the Unstructured Gridder; not all menu options are addressed, however.

This tutorial takes about one hour to complete.

Stages

The goal of this tutorial is to construct a PEBI simulation grid within ECLIPSE Office and then use this grid in a simulation run. This requires importing and managing a wide variety of data (that is tops, thickness, wells, porosity etc.). The tutorial stages are as follows:

Problem description

In this tutorial you construct a 4-layer reservoir model from a variety of data sources. First, the reservoir boundary and layers are defined. Mesh maps of top depth and thickness are then imported to construct a framework for the model. Next, well and fault locations are added to the model. Then, porosity and permeability data is imported. Once all the pieces are in place, the grid and its properties is generated using the PEBI gridder. Finally this grid is used in an ECLIPSE simulation.

Getting started

All the necessary data files are in the directory `$ECLARCH/$ECLVER/office/tutorial/example3`.

The following steps guide you into the Unstructured Gridder section of ECLIPSE Office.

1 Start ECLIPSE Office from the ECLIPSE Launcher (on UNIX: `@office`).

2 From the top menubar select File | New Project and give this project the name `Tut3`.

3 Choose File | Save Project

4 In the main ECLIPSE Office panel, choose View | Display Model in DM

5 In the main ECLIPSE Office panel, choose View | Display Model in Grid Section

6 Open the Data Manager module (now referred to as DMM) by either selecting the Data button in the left column or by choosing Module | Data Manager... from the top menubar.

7 Select DMM: Case Definition

8 In the General tab set the Units to metric.

9 In the PVT tab select the following phases: Water, Oil, Gas, Dissolved Gas.

10 In the Misc tab set `NSTACK` (Stack size of Previous search directions) to 50.

11 Click on OK to exit the Case Definition Section.

12 Select DMM: Grid Section. The grid is empty because no data has been imported yet.

13 Select DMM: Grid | Subsection | Unstructured Gridder... to open the Unstructured Gridder.

The Unstructured Gridder contains six major sections:

Reservoir data

This displays in a tree format the data that has been imported into the reservoir model. It contains both structural and property data. This is referred to throughout the tutorial as the Reservoir Data Tree.

Areal view

This displays an IJ slice through the reservoir. It is initially the active window of the Unstructured Gridder.

Navigation graph

This allows you to see what portion of the reservoir is currently displayed in the active window. This is most useful when the zoom option has been used.

Cross section

This displays a cross section through the reservoir. The line of the cross section can be seen and moved in the Areal View window.

Caption

This allows you to check the status of the gridding, see information about imported data, and see the number of geological and simulation layers in the model.

Legend

This displays the color legends when plotting various properties.

It is useful to hide all windows except the Unstructured Gridder so as to keep the desktop from becoming too cluttered. To do this:

1 Minimize the Unstructured Gridder

2 Minimize ECLIPSE Office

This causes all the other windows to minimize as well

3 Maximize the Unstructured Gridder window.

Importing reservoir boundary and layer data

Once inside the Unstructured Gridder, you must first define a volume for the reservoir model. This can be done in several ways:

- Create | Main Volume | Digitize 

  This digitizes a boundary for the reservoir.

- Create | Main Volume | Rectangular 

  This creates a rectangular reservoir boundary.

- Create | Main Volume | Circular 

  This creates a circular reservoir boundary.

- File | Import ASCII | Volumes...

  This reads in an `ASCII` file containing the coordinates of the boundary.

Importing a reservoir boundary

1 File | Import ASCII | Volumes...

2 Select the file `boundary.zon` and click on Open.

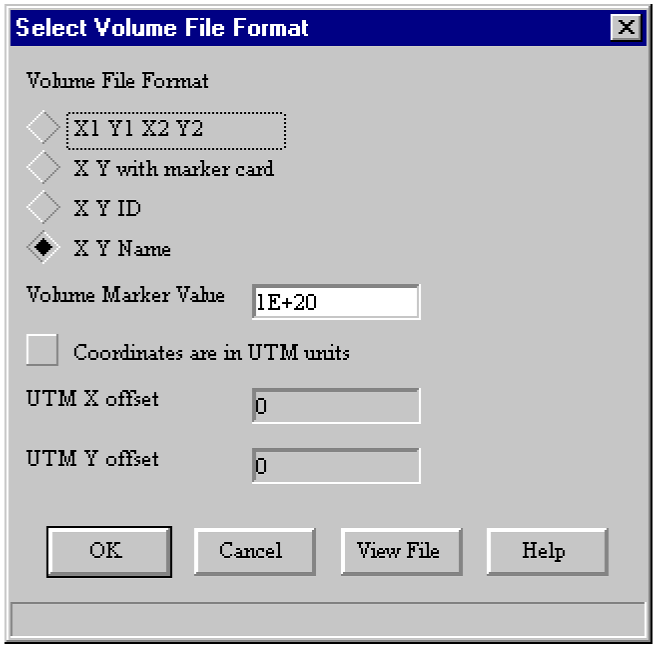
This opens the Select Volume File Format window.

The Volume File Format is X Y Name.

Note The outer boundary coordinate data is currently supported with four formats. Also, UTM offsets can be input for metric data.

Make sure your data is the same as that in Figure 4.13.

Figure 4.13 Select Volume File Format window



3 Once the data has been entered in the Select Volume File Format panel, click on OK.

4 Settings | Map Limits...

5 Click on the Reset button in the Set Display and Mesh Map Limits window to update the Areal View window.

6 Click on OK.

The primary volume now needs a constant properties table. This table defines the number of layers and constant property values for each layer. The maps attached later in the tutorial automatically override the values defined in the constant properties table.

7 Select the Boundary in the Reservoir Data Tree.

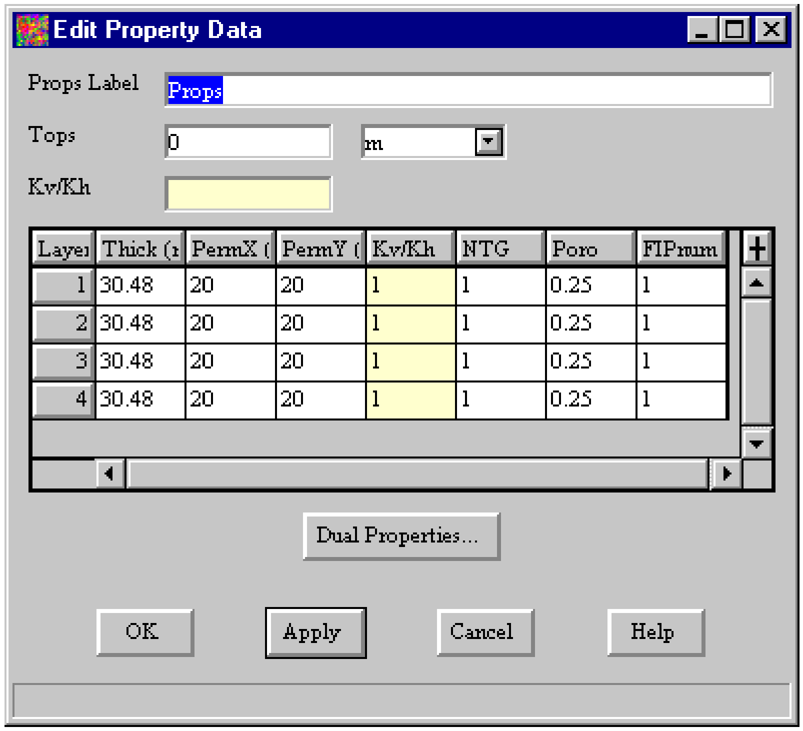
8 Create | Const. Props.

This opens the Edit Property Data window

9 Click on the + button in the top right corner of this table to create 4 layers

10 Compare with Figure 4.14.

Figure 4.14 Edit Property Data window



11 Select OK in the Edit Property Data window.

The Props box should now appear in the Reservoir Data Tree.

Creating layers and maps

Now that the boundary and constant properties table for the reservoir are defined, layers can be added.

For this data set, four layers need to be created. Each layer has a thickness map associated with it. In addition, the top layer has a top depth map associated with it.

To create the layers and associated maps:

1 Select the Boundary in the Reservoir Data Tree (once selected it should be highlighted in red)

2 Create | Layer 

Use this command four times to create four layers. These layers appear immediately in the Reservoir Data Tree.

Hint The name of a layer can be changed by double-clicking on the layer in the Reservoir Data Tree and modifying the Layer Name in the Edit Layer Data window.

---

3 Select Layer1 by clicking on it once in the Reservoir Data Tree.

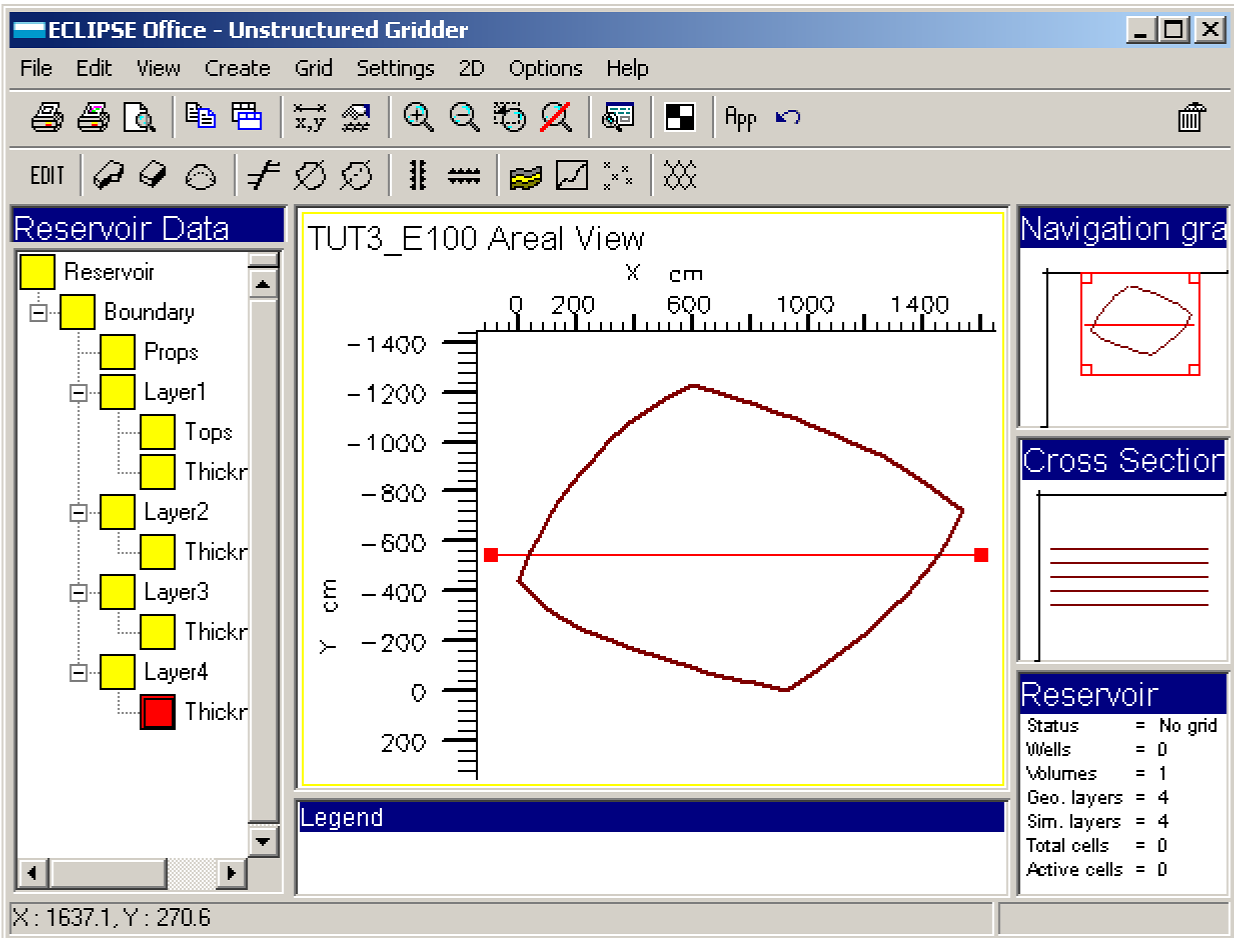
4 Select Create | Map | Tops and Create | Map | Thickness to add tops and thickness maps to Layer1.

5 Select the other layers (one at a time) and choose Create | Map | Thickness.

This associates a thickness map with each layer.

Now all the layers should have maps associated with them. The Unstructured Gridder should look like the one in Figure 4.15.

Figure 4.15 Unstructured Gridder showing the addition of 4 layers and associated maps



Importing map data

The maps have now been created, but there is no data associated with any of them. In this tutorial thickness and top depth data are imported as mesh maps. There are other options available such as contour data and scatter data that you may find useful.

When importing mesh maps the limits must first be defined by following these steps:

1 Settings | Map Limits...

This opens the Set display and mesh map limits panel.

2 Select the tab labeled Mesh Map.

3 Set the limits of the mesh maps so that -100<x<1600 and -1400<y<300.

4 Click on OK to commit the changes to mesh map limits and close the panel.

Hint It is important to set the mesh map limits before importing any mesh maps. Once a mesh map has been imported the mesh map limits become read-only. All mesh maps must be the same size.

5 Select the Tops Map in Layer 1 of the Unstructured Gridder Window | Reservoir Data Tree.

Hint If the Tops Map is not visible in the Reservoir Data Tree, click on the + button next to Layer 1 to expand this branch.

6 File | Import ASCII | Mesh Map...

7 Open the file `tops.L01`.

This opens the Select Mesh Map Format window in which you must specify the header format, the arrangement of the grid (ordered by rows or columns), and the location of the mesh origin.

Note The `ASCII` mesh map file is assumed to consist of a header area followed by a grid of NX*NY values. You can select from a number of industry standard header formats.

8 Enter the following data:

Mesh Map Property Unit	m
Mesh Map Format	ASCII NX*NY values
Data is ordered by rows	Checked
Scatter/Mesh Map Origin	First Pt in file is at Top Left
Coordinates are in UTM units	Not checked
UTM X offset	0
UTM Y offset	0
Null Value	1000000

9 Click on OK in the Select Mesh Map Format window to complete the import of data.

Note Maps which contain mesh map data are indicated by the letter *M* in the Reservoir Data Tree. The tops map that was just imported should now have an *M* to indicate that it is a mesh map. If it does not, click on another object in the Reservoir Data section and the *M* should appear.

10 Select the Thickness Map in Layer1.

Caution Make sure it is highlighted red, indicating it has been selected.

11 File | Import ASCII | Mesh Map...

12 Open `thick.L01`

This is the thickness data for Layer 1.

13 Click on OK in the Select Mesh Map Format window (any changes made before are now the defaults) to complete the import of thickness data for Layer 1.

14 Select the Thickness Map in Layer2

15 File | Import ASCII | Mesh Map...

16 Open `thick.L02`

This is the thickness data for Layer2.

17 Click OK in the Select Mesh Map Format window.

18 Import the mesh map thickness data for Layer 3 (`thick.L03`) and Layer 4 (`thick.L04`) following the same procedures as above.

Cross section window

The Cross Section window can be used to examine the newly imported data. It should now display all four layers. The location of the cross section line can be modified in several ways:

- Settings | Cross Section... opens the Cross Section Line Coordinates window in which the end points of the cross section line can be modified and applied.
- Edit | Edit Point; Edit | X Edit; and Edit | Y Edit can all be used to alter the location of the cross section line. Simply click on one endpoint of the cross section line and drag in to the desired location.

Viewing the active cross section

1 Settings | Cross Section...

2 Choose the Default button or make sure that the cross section line coordinates are the same as the following data:

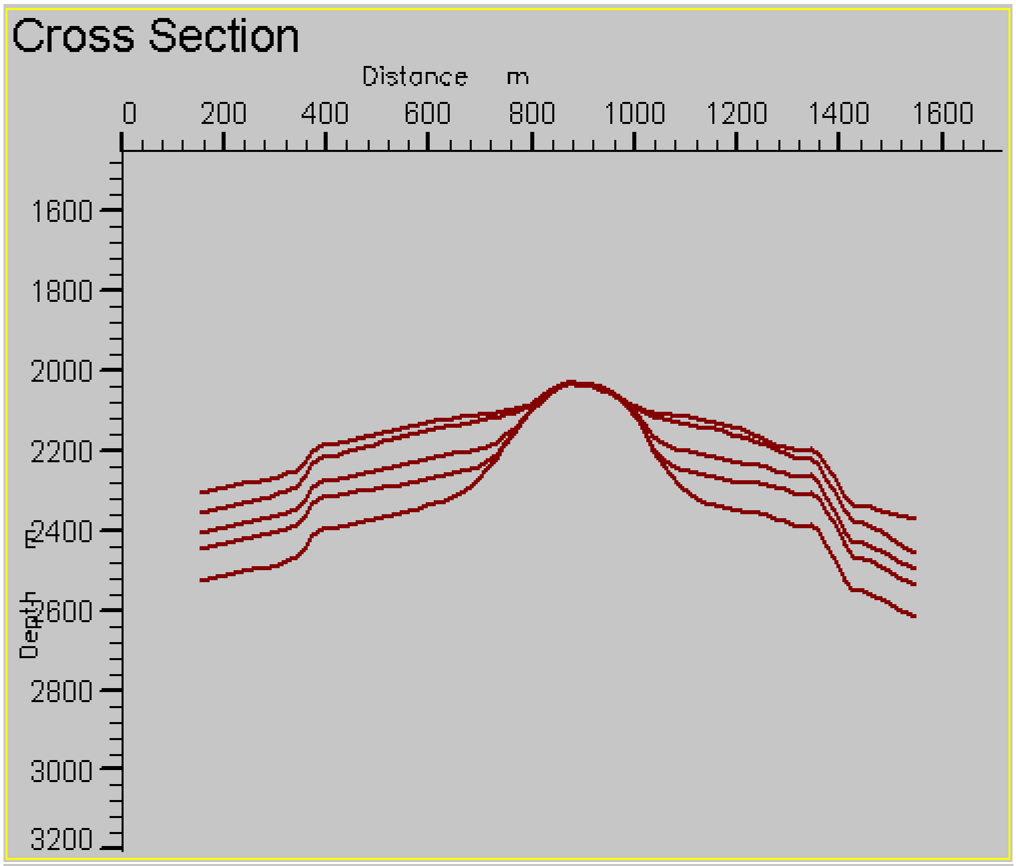
X (m)	Y(m)
-96.64	-550
1596.9	-550

3 Select OK to close the Cross Section Line Coordinate window

4 Double-click in the Cross Section window to bring it into the active frame

5 Compare with Figure 4.16 to make sure that all tops and thickness data has been imported correctly.

Figure 4.16 Cross Section displayed as active window



6 Make the Areal View active again by double-clicking on it.

Importing/manipulating well data

Follow the steps below to import two vertical wells into the Unstructured Gridder.

1 Select the Reservoir in the Reservoir Data Tree.

2 File | Import ASCII | Vertical Wells...

3 Open the file `VERT.WEL`.

4 In the Select Well Format panel click on the View File button to view the well data.

This option allows you to inspect the format of the file before importing the data. This is useful because there are several different formats that are acceptable for well data.

This data file is of the format X Y MD TVD

The figure `-999` is used as a marker to separate the wells in this data file

5 Choose Close View in the Select Well Format window to hide the well data.

6 Modify the Select Well Format window with the following data:

Well File Format	X Y MD TVD
Marker value	-999
UTM X Offset	0
UTM Y Offset	0

7 Click on OK in the Select Well File Format window to complete the import of the well data.

Note One limitation of the Unstructured Gridder is that deviated wells are not supported.

Now that the well data has been imported you can now manipulate it. If necessary, you can change the perforations

Changing well perforations

1 You can modify the well perforations in the following manner:

2 Double-click on Well1 in the Reservoir Data Tree.

This opens the Edit Well Data window.

3 Click on the Perforations... button to open the Edit Perforation Data window.

4 Edit the Start location of the perforations as follows:

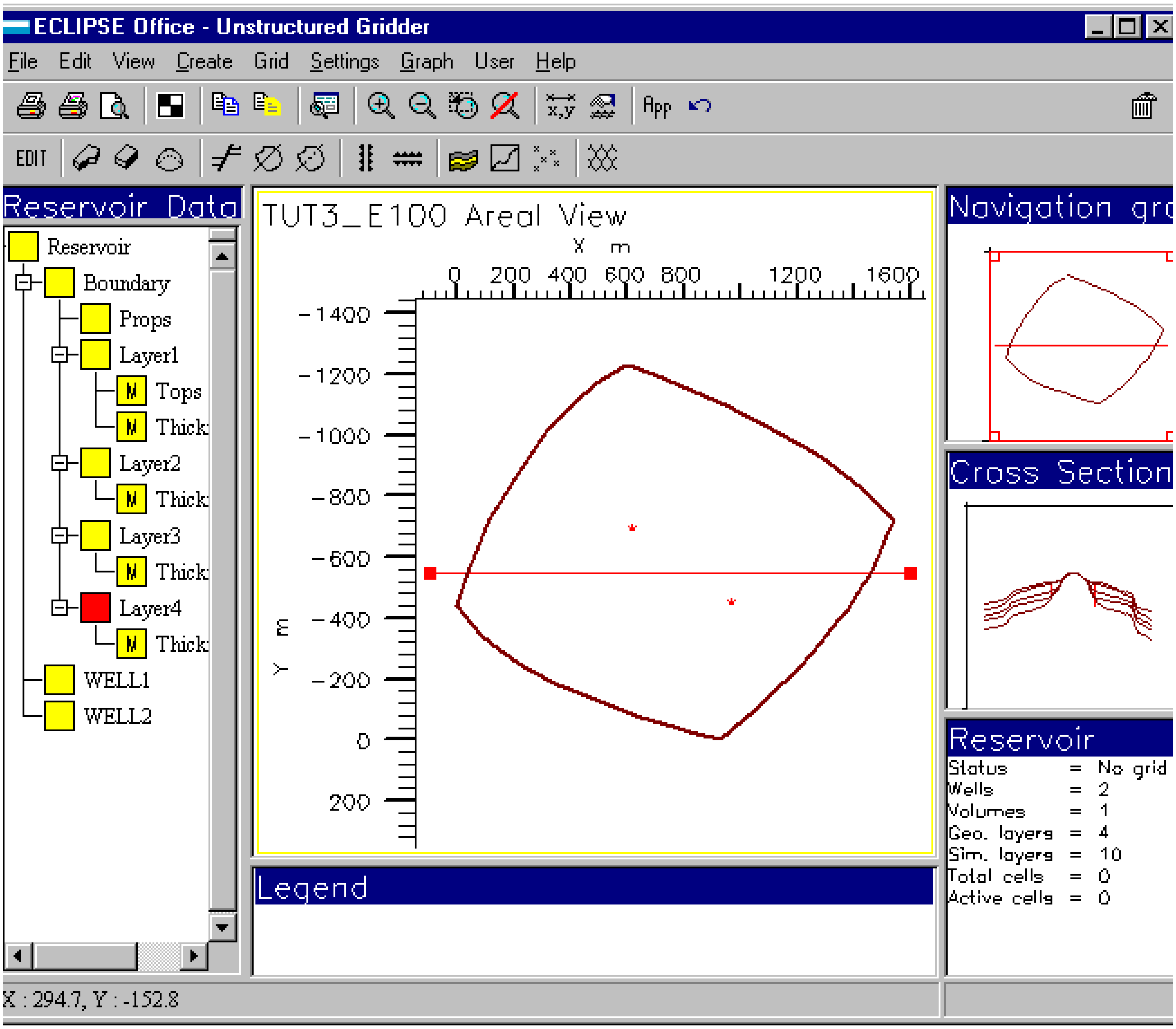
Start (m)	End (m)	Radius (m)	Skin	Active
10		0.088392	0	Yes

Hint The true vertical depth of the perforations can be seen by clicking on the True Vertical Depth... button in the Edit Perforation Data window. These values are read-only.

Note Although multiperforated wells are allowed in the Unstructured Gridder, these wells appear to be fully completed through the reservoir.

Now the Unstructured Gridder should look similar to Figure 4.17.

Figure 4.17 The Unstructured Gridder after the import of mesh maps and well data



Importing/manipulating fault data

Fault data is now be added to the model. There are four faults in this model. The importation is similar to well and map imports:

1 Select the Reservoir in the Reservoir Data Tree

2 File | Import ASCII | Faults...

3 Open `Fault1.FLT`.

This opens the Select Fault Format window.

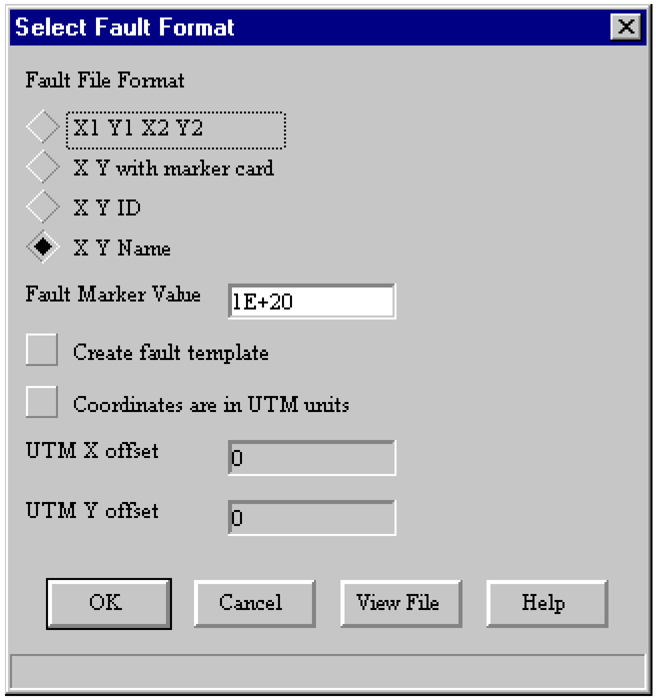
4 Select View File in this window to check the format of the data.

This data is ‘X Y Name’.

5 Click on Close View in the Select Fault Format window to hide the fault data.

6 Compare the Select Fault Format window with that shown in Figure 4.18.

Figure 4.18 Select Fault Format window



7 Select OK to complete the import of fault data.

8 Repeat the above procedure to import the other three faults (`fault2.flt`, `fault3.flt`, and `fault4.flt`).

Note Sloping fault geometries are not supported.

Digitizing a fault

Some of the faults are zig-zag in appearance. In this section we will change one of these faults to a template and digitize a new, smoother fault over the top of the old one.

1 Select Fault 4 in the Reservoir Data Tree.

2 Edit | Feature EDIT opens the Edit Fault Data window.

3 Check the Fault is a template box.

4 Click on OK to make Fault 4 into a template and close the Edit Fault Data window.

Hint Faults that are templates appear as a faint line rather than as a bold line. Fault 4 should now appear as a faint green line in the Areal View window of the Unstructured Gridder. Also, templates are not saved upon exiting the Unstructured Gridder.

---

5 Select Create | Fault (or use the appropriate icon).

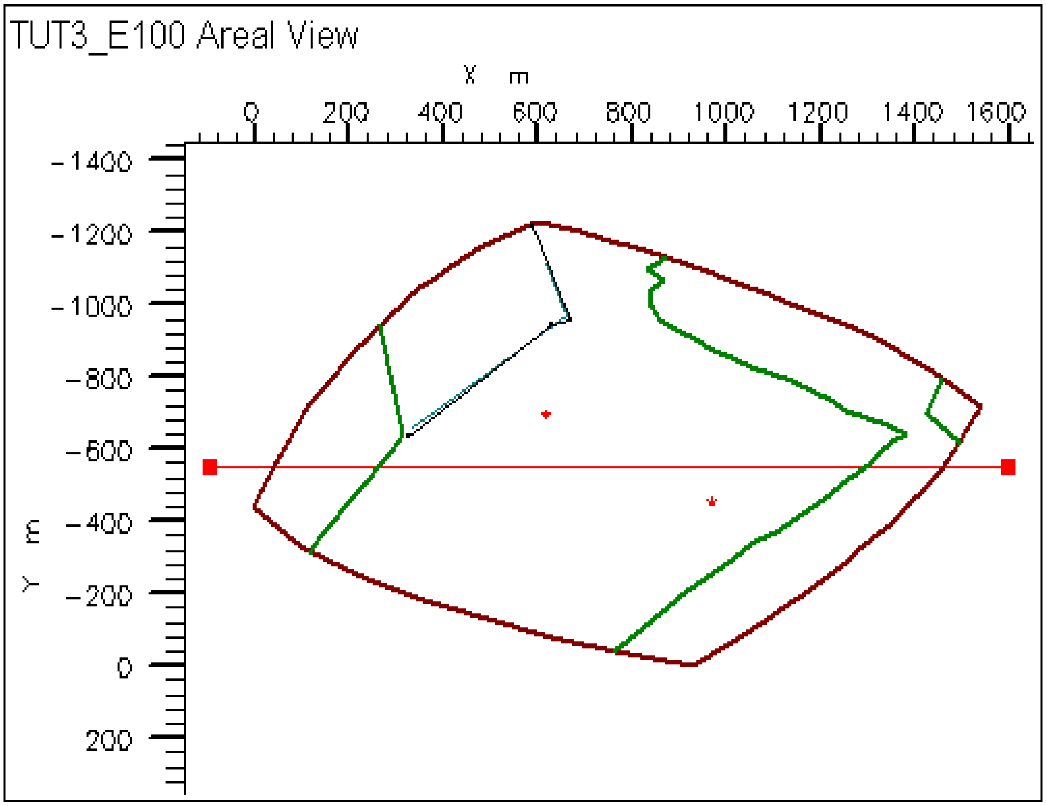
In the Areal View window, the cursor is now pencil shaped to indicate that you are ready to digitize.

6 Start digitizing by clicking the left mouse button on one end of the fault.

7 Click the left mouse button on several more locations along the template to make fault segments.

As you can see in Figure 4.19 a black square appears every time you create a segment of the fault.

Figure 4.19 Unstructured Gridder window during the digitization of a fault



8 Double-click or press `Return` when you reach the other end of the fault, to complete the fault.

The black squares disappear.

If you are not satisfied with this new fault then delete it in the following manner:

9 In the Reservoir Data Tree, select Fault 5

10 Select Edit | Delete Feature.

Once you have digitized a new fault, you can get rid of the old one. To do this:

11 In the Reservoir Data Tree, select Fault 4.

12 Select Edit | Delete Feature.

Hint An alternative way to delete features is to first select them in the Reservoir Data Tree and then select Delete feature from the right mouse button menu to remove them.

Smoothing faults

The Unstructured Gridder also has an automatic fault smoothing option. To implement this option:

1 Double-click on Fault 5 in the Reservoir Data Tree to open the Edit Fault Data window

2 Click on the Smooth button to open the Smoothing Parameters window

3 Edit the Smoothing Parameters window with the following data:

Identification mode	Least squares linear
Noise level	155 m
Keep end points of faults	Checked

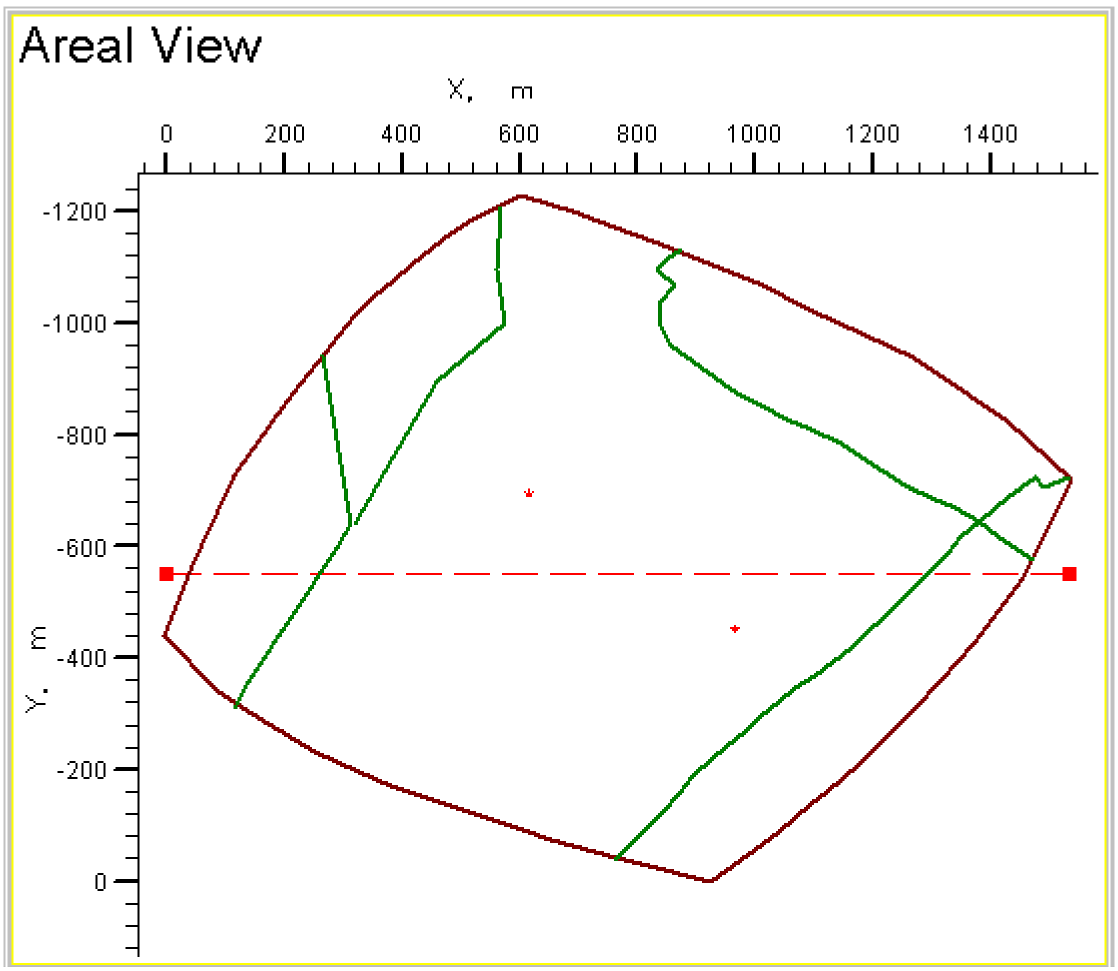
Hint The amount of smoothing can be increased by increasing the noise level or switching to the least-squares linear mode.

4 Click on Smooth

5 Click on Accept to accept the new fault.

The Unstructured Gridder should now appear similar to Figure 4.20.

Figure 4.20 The Unstructured Gridder displaying the smoothed faults



Now all that is left to complete the model is to import the properties data.

Importing porosity and permeability data

The final step in creating this model is importing porosity and permeability data. For this tutorial the porosity and permeability data are scatter data.

1 Select Layer1 in the Reservoir Data Tree.

2 Create | Map | Perm_X and Create | Map | Porosity

Hint If no Perm-Y map is defined, then both X and Y permeability are obtained from the Perm-X map. In this tutorial Perm-Z is constant and is defined in the constant properties table.

3 Repeat these steps for the other 3 layers

4 Select the Perm-X map in Layer1

5 File | Import ASCII | Scatter Data...

6 Load the permeability file `permx1.cnt`

Markers, indicating the locations of the scatter data, appear in the Areal View window.The Select Scatter Data Format dialog box opens up.

a Accept the defaults on the Select Scatter Data Format Window and click OK.

7 Select Scatter Data Format dialog box opens up and click OK.

8 Select the Porosity map in Layer1

9 File | Import ASCII | Scatter Data...

10 Load the porosity file `poro1.cnt`

11 Repeat these import steps for the other three layers.

Note In general, PEBI grids can only be used with isotropic models. In the Unstructured Gridder, anisotropic models with constant K_x:K_y ratio are supported. Fully anisotropic models are supported only for single-phase models.

Viewing input data

Now that all the data has been imported into the Reservoir Data Tree, it is useful to look at some of this data.

1 Select View | Options to open the Edit View Options window.

2 Under the Line/Text tab, deselect the Scatter/Contour data button.

This turns off the markers for the scatter data.

3 Under the Color Fill tab, select Property Display | Input Map and Layer1 Tops.

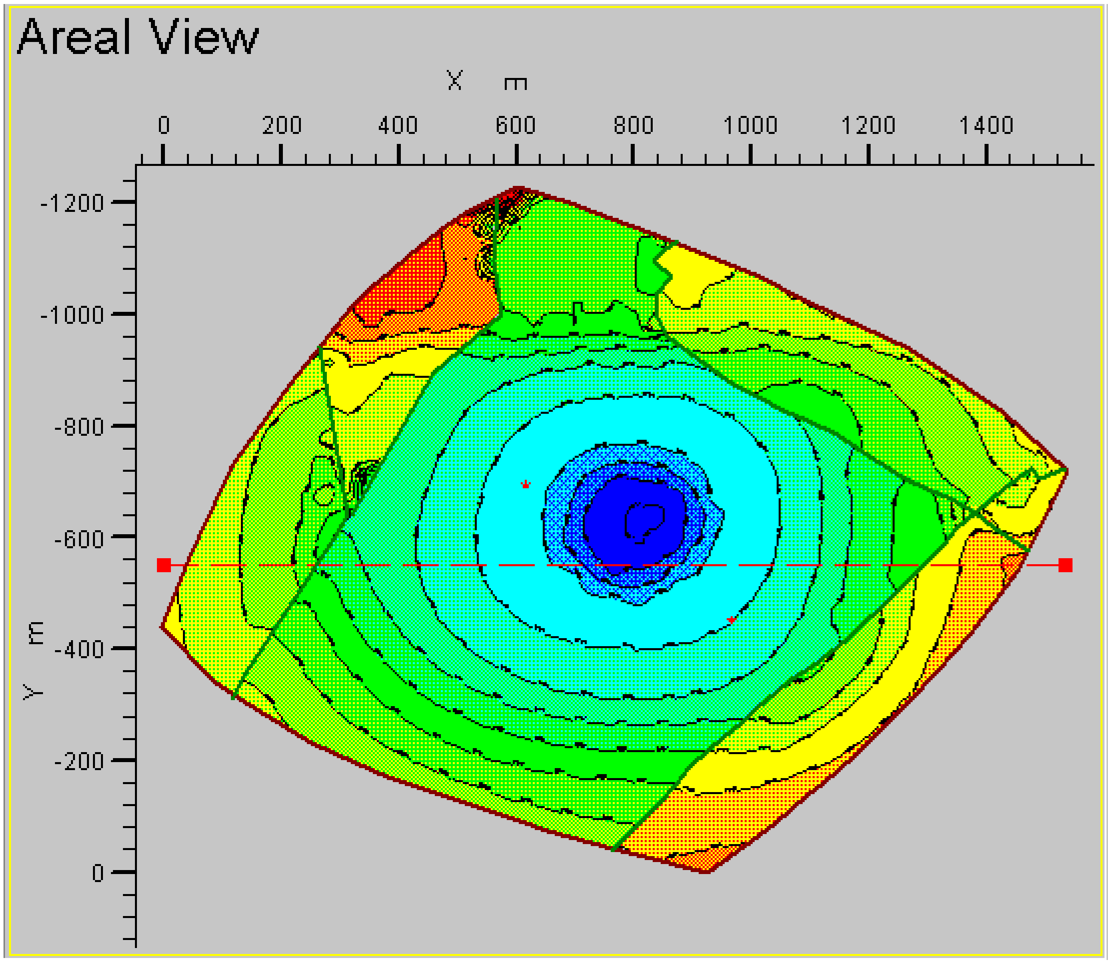
Select Contour Lines option.

4 Click on Apply.

5 Click on OK in the Edit View Options window to close the window.

The Unstructured Gridder Window | Areal View should now contain a color map of the tops data, similar to Figure 4.21.

Figure 4.21 Unstructured Gridder window displaying Layer 1 Tops Data



Generating a grid and properties

We can now generate both a grid and grid properties.

Setting grid style and grid resolution

You can control the grid style and resolution by adjusting the grid controls. This is described below.

1 Double-click on Boundary in the Reservoir Data Tree.

This opens the Edit Primary Volume Data panel.

2 Select the Grid Controls... button.

The Volume Grid Controls panel now appears.

The Grid Style should be set to Variable (this is the default)

Hint Various PEBI gridding styles are supported. These can be selected in the drop-down menu for Grid Style.

3 Modify the following elements of the Volume Grid Controls.

a Relative minimum cell size = 0.02

b Relative maximum cell size = 0.05

Note For variable gridding style, you should specify either the relative maximum and minimum cell size or the absolute maximum and minimum cell size.

4 Click on OK in the Volume Grid Controls to apply the changes and close the panel

5 Click on OK in the Edit Primary Volume Data window.

Note The gridding around faults and wells can also be modified by you. This is done in the Grid | Grid Controls menu.

Generating the grid

Now that the grid style and resolution have been edited, the grid can be generated.

1 Make sure Grid | Show grid report is selected (if it is selected a check mark should appear next to it in the drop-down menu).

2 Grid | Generate Grid

This generates the unstructured grid.

The Grid Report window appears to inform you which items have not been fully gridded. You can choose to either halve the bulk cell size to try to fully grid these items, or ignore the items that cannot be fully gridded.

3 Opt to ignore the items that cannot be fully gridded.

4 Click on OK.

This completes the grid generation.

After generating a grid the zoom options allow you to examine the grid in detail to decide if it is honoring the structural data.

Zoom options

1 Select Settings | Cross Section... | Default to make sure your cross section is the same as the tutorial, (recalled in the following table:

X (m)	Y(m)
-96.64	-550
1596.9	-550

2 Bring the cross section into the active window by double-clicking on it.

3 Use the Rubberband Zoom In button  for zooming in on the region around the fault. Zooming allows you to inspect faults and wells more closely. Select the part of the plot you wish to inspect by click-dragging around it.

4 Now unzoom completely  and click on  to deselect the rubberband zoom.

5 Double-click on the Areal View window to make it the active window again.

Generating properties

Now the property data can be assigned.

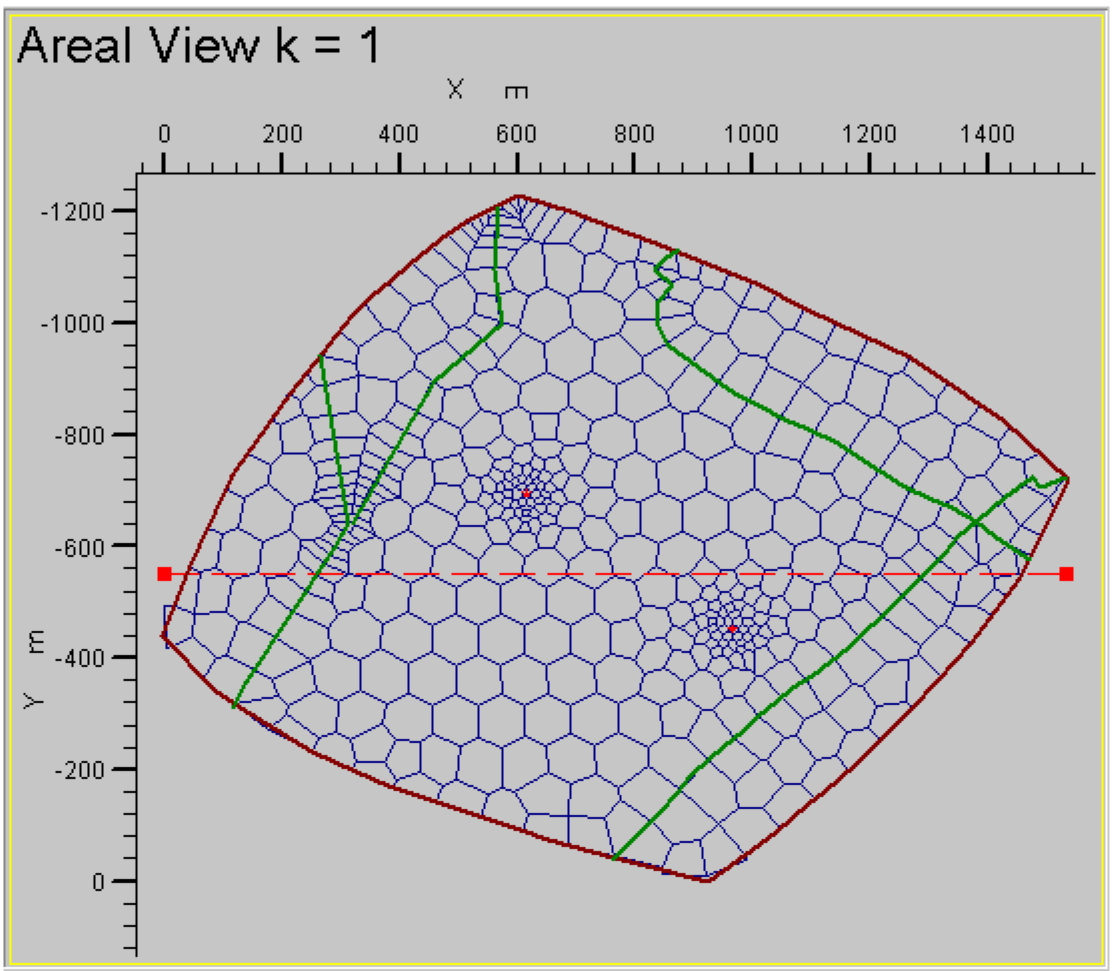
1 Grid | Generate Properties

This assigns permeability and porosity values to the unstructured grid.

Hint Do not be alarmed if the Generating Grid box or the Generating Properties box appears to freeze. Grid generation can take some time.

The unstructured grid that is generated should appear similar to Figure 4.22.

Figure 4.22 Unstructured Gridder after generating a grid.



Viewing properties of the unstructured grid

Now that the grid has been generated you can view the permeability and porosity data for this grid as described below.

1 View | Options

This opens the Edit View Options window

2 In the Color Fill tab choose Initial property and Porosity.

3 Select Apply and OK

4 The Unstructured Gridder | Areal Viewer now displays a color map of the porosity values.

Modifying the range of the color map

1 In the Areal Viewer, double-click on the numbers beneath the color legend.

2 In the Color Legend Editing dialog box, change the range to go from `0.1` to `0.2`.

3 Click on OK.

Editing properties of the unstructured grid

You can view and edit the properties of individual grid blocks.

1 View | Options

This opens the Edit View Options window

2 Select the Color Fill tab

3 Choose Initial Property and Porosity

4 Click on the Edit Properties button in the Edit View Options window.

This opens the Selected Cells window

5 Deselect Unstructured Gridder: 2D | Show cell probe so that a new window is not displayed when using the cell probe

6 In the Unstructured Gridder choose 2D | Pick. 

This activates the cell probe.

7 Select a cell in the Areal View window.

The porosity value and the index for this cell should now appear in the Selected Cells window.

In the Selected Cells window the porosity value of the highlighted cell can be changed.

Hint Multiple cells can be selected by holding down the `Shift` key and then selecting cells with the left mouse button. The Selected Cells button in the Edit Initial Property window can then be used to view or edit the values for this set of cells.

---

8 Click on Cancel in the Selected Cells window.

9 Click on Cancel in the Edit View Options window.

Saving and exiting the Unstructured Gridder

Now that a grid has been created it can be used by the rest of ECLIPSE Office.

1 Unstructured Gridder: File | Save opens the Save File window

The grid for visualization is saved in `Tut3.pgrid`.

2 Select Save to complete the save.

3 File | Close closes the Unstructured Gridder.

4 If the other ECLIPSE Office windows are minimized, maximize them now

5 Select File | Close to close the Grid Section window.

The active window should now be the Data Manager.

Running an ECLIPSE simulation

Now the remaining data is imported and an ECLIPSE simulation is performed.

Importing PVT data

The PVT data is in the file `PVT.data`

1 In the DMM choose the PVT button to open the PVT Section

2 Select File | Import | New to import the file `PVT.data`

3 File | Save As... saves this data to the file `tut3_pvt.inc`

4 File | Close

Importing SCAL data

The SCAL Section data is imported as follows:

1 In the DMM choose the SCAL button to open the SCAL Section

2 Select File | Import New to import the file `SCAL.data`

3 File | Save As... saves this data to the file `tut3_scal.inc`

4 File | Close

Entering initialization data

The initialization data is entered as follows:

5 In the DMM choose the Initialization button to open the Initialization Section.

Hint It may take some time for the Initialization Section to open because all the information for the PEBI grid must be read.

6 Choose Edit | Insert Keyword to insert the keyword `EQUIL`

7 Enter the following data for the EQUIL (Equilibrium Data Specification) keyword:

Depth	2000 m
Pressure	200 bar
WOC Depth	2300 m
OW Cap Pressure	0 bar
GOC Depth	2000 m
GO Cap Pressure	0 bar
Rs/Pb v Depth Table	0
Rw/Pdw v Depth Table	0
Accuracy	0

8 Select Apply to finalize the changes to the initialization data

9 Select File | Save As... to save the data to the files `tut3_init.inc.`

10 Choose File | Close to close the Initialization section

Entering Schedule data

The Schedule Section takes a little more time to update. The well location (`WELSPECS`) and well connection (`COMPDAT`) keywords have already been input by the Unstructured Gridder. These keywords need to be checked.

1 `WELSPECS`: check that the preferred phase is correct

`Well1`: should have OIL as its preferred phase

`Well2`: should have OIL as its preferred phase

2 `COMPDAT`: check that perforations of all wells are OPEN

Specifying the well control keywords (WCONPROD)

1 In the Events All window select the last event

2 From the menu choose Event | New.

This opens the New Event panel.

3 Choose Event Types | Well Controls and Limits in the left window

4 Choose Events | Production Well Control in the right window

5 Click OK.

The keyword `WCONPROD` should now appear in the Events-All box in the Schedule Section.

6 Enter the following data production well control (`WCONPROD`):

Well	WELL2
Open/Shut flag	OPEN
Control	ORAT
Oil Rate	2000 m^3/day
BHP Target	150 bar

7 Click Apply.

Hint It is important to check that the Well Name for the WCONPROD keyword matches exactly with that in the WELSPECS keyword. These names are case sensitive.

Specifying the well control keyword (WCONINJE)

1 From the menu choose Event | New.

This opens the New Event panel.

2 Choose Event Types | Well Controls and Limits in the left window

3 Choose Events | Injection Well Control in the right window

4 Select OK.

5 In the DMM: Schedule Section input the following data for the keyword WCONINJE:

Well	WELL1
Injection type	WATER
Open/Shut Flag	OPEN
Control Mode	RATE
Liquid Surface Rate	2000 m^3/day

6 Click on Apply.

The Print File Output Control keyword (RPTSCHED) is entered next

7 From the menu choose Event | New.

8 Choose Event Types | Output in the left window

9 Choose Events | Print File Output Control in the right window

10 Click on OK.

11 In the DMM | Schedule Section, input the following data for RPTSCHED keyword:

Grid Block Pressure	Checked
Grid Block Oil Saturations	Checked
Grid Block Water Saturations	Checked
Grid Block Gas Saturations	Checked
Grid Block Solution Gas-Oil Ratios	Checked
Restarts	Every Report
FIP Reports	FIP Report
VFP Tables	No VFP Table Output

The simulator control keyword (TUNING) is now entered

12 From the menu choose Event | New.

13 Choose Event Types | Simulator Controls in the left window

14 Choose Events | Simulator Control Parameters in the right window

15 Select OK

16 In the DMM: Schedule Section, go to page 3 of the TUNING keyword and change the Max Linear Iterations in Newton Iteration to 100.

17 Click on Apply.

Finally more dates must be added to the simulation.

18 From the top menubar choose Time | Insert to open the New Time panel

19 Choose a Date of `1 Jun 1983`.

20 Use Time | Insert to enter the following dates: `1 Jan 1984` and `1 Jun 1984`.

Select the first date; the Schedule Section should be similar to Figure 4.23.

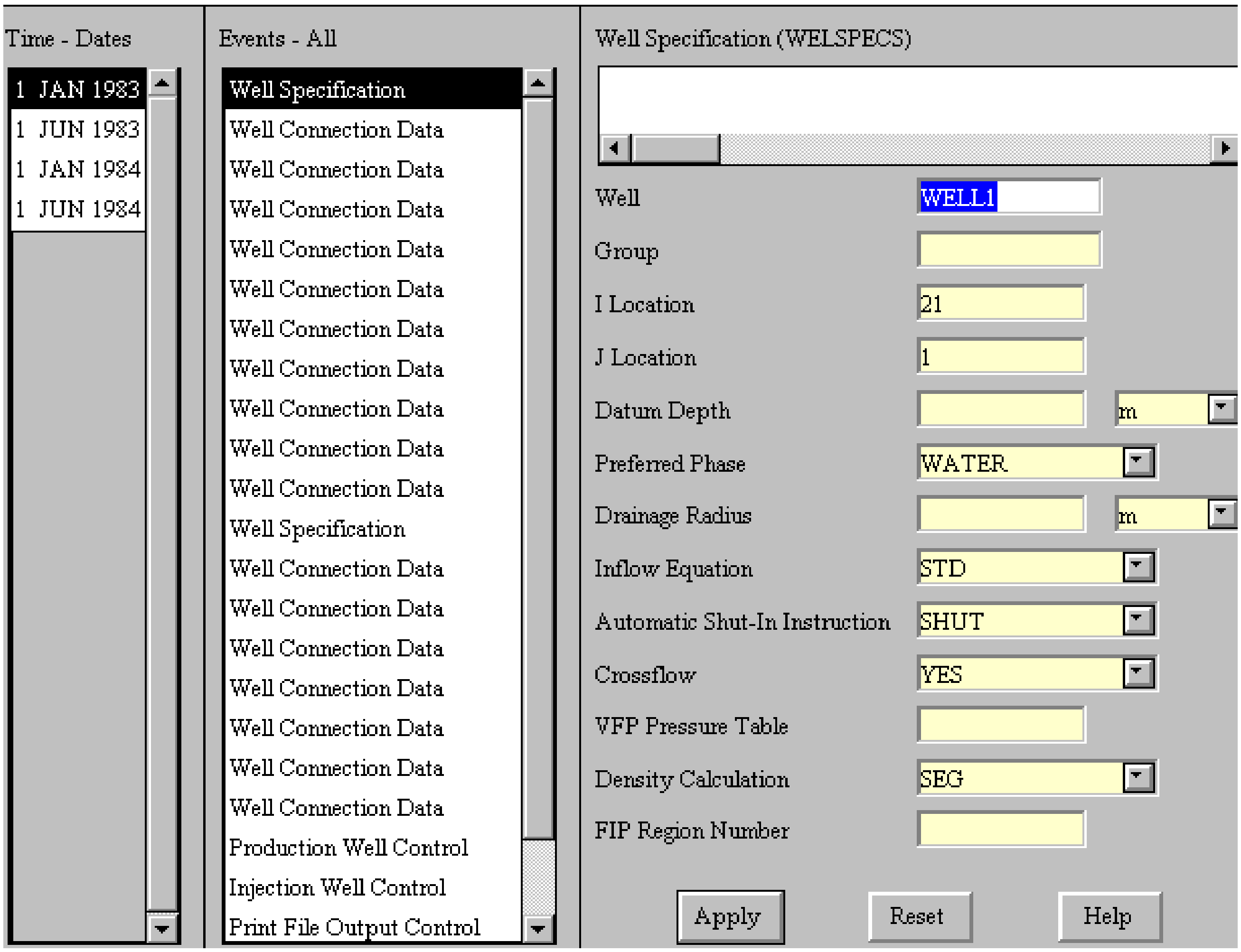


Figure 4.23 The Schedule Section after all necessary keywords and dates have been input

21 Now choose File | Save As to save the file `tut3_sch.inc` and then use File | Close to close the Schedule Section.

Importing summary data

The summary data is in the file `SUM.data`

1 In the DMM choose the Summary button

2 Select File | Import | New... to import the file `SUM.data`

3 File | Save As saves this data to the file `tut3_sum.inc`

4 File | Close.

Note If the Generic Keywords option is ON:
1. In Data Manager > Section > Dimension Overrides, select Keyword Table Dimensions.
2. Change the 3rd, 4th and 6th fields to 22 each. Click on Apply and close the Dimension Overrides Section.

Running the simulation

1 In the ECLIPSE Office window choose Module | Run Manager from the top menubar or click on the box labeled Run on the left and Submit | Runs.

Viewing the results

Once the run has ended, close the Run Manager and then, in the ECLIPSE Office window, click on the Result button in the right menubar to open the Result Viewer.

1 File | Open Current Case | Summary

2 In the Result Viewer Module: Extract/Load Summary Vectors window, check the boxes to Read All Summary Vectors and Read All Reports.

3 Click on Load

4 Choose LinePlot | User to open the User Template window

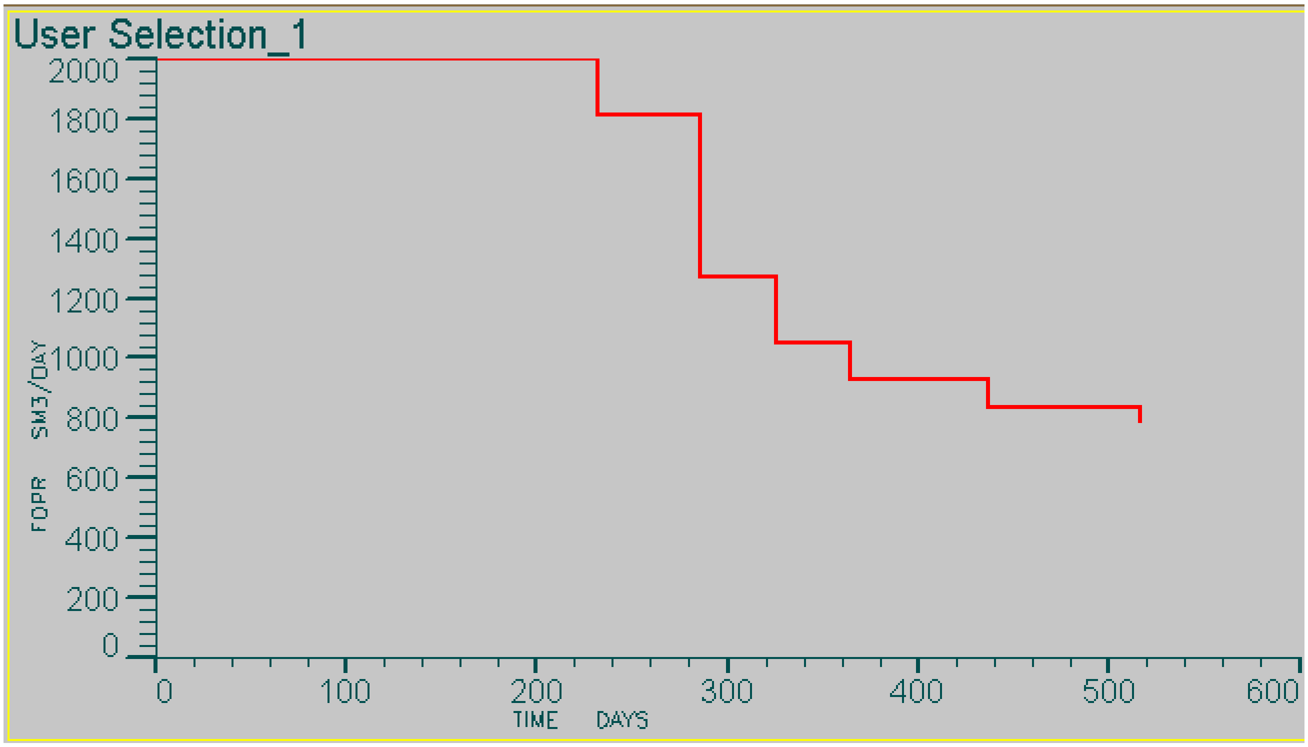
5 Select Time vs. FOPR in this window.

6 Click on Add to List

7 Click OK

The resulting plot of Field Oil Production Rate versus Time is shown in Figure 4.24.

Figure 4.24 Plot of Field Oil Production Rate versus Time



Viewing grid data

In the Result Viewer:

1 Choose File | Open Current Case | GRID... to open `tut3_E100.pgrid`.

In the Extract/Load Solutions window:

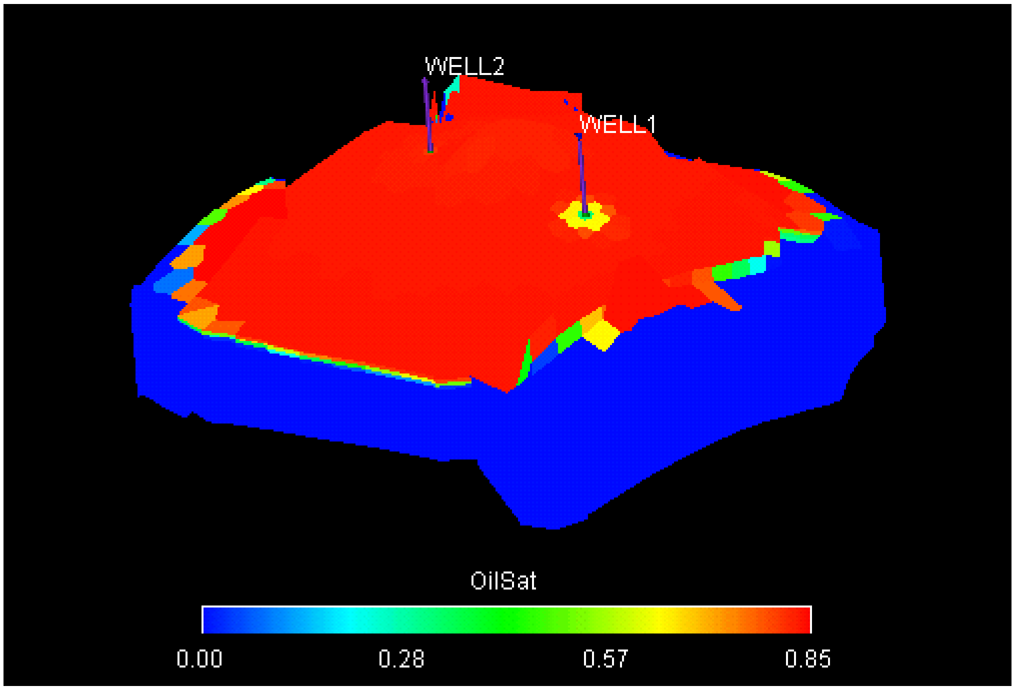
2 Check Read INIT file

3 Check Read All Reports

4 Click on Load.

5 Result Viewer | View | 3D.

Figure 4.25 The 3D Result Viewer displaying the initial oil saturation data



The 3D Viewer window should currently be displaying the oil saturation (SOIL): see Figure 4.25.

To view the gas saturation:

6 Scene | Grid | Property... This opens the Property Display window.

7 Choose SGAS, and the 3D Viewer should automatically update.

For recurrent properties the values at different timesteps can also be viewed. The timestep being displayed is recorded in the bottom right corner of the 3D Viewer window. "Tutorial 1: Standard usage" on page 29 contains more details about viewing results from simulation runs.

Importing a structured grid

This tutorial demonstrated the steps to create an unstructured grid from scratch using ECLIPSE Office. It should be noted that existing data sets can be imported into ECLIPSE Office, and the existing grid file can be converted to a PEBI grid using the Unstructured Gridder. When a structured grid is imported into the Unstructured Gridder, certain data, primarily from the Initialization and SCAL Sections, are lost. Other keywords such as WELSPECS and COMPDAT is updated by the Unstructured Gridder. After converting from a structured to an unstructured grid, all sections must be re-visited and saved to ensure that any incompatible keywords are removed. Only then should a simulation run be attempted.