Tutorial 2: Building a model

Introduction

This tutorial builds a Black Oil model from scratch using output generated by other SIS Simulation Software applications. It focuses on manipulation in each section in the Data Manager, but includes the submission of the simulation run, and viewing errors or warnings.

This tutorial takes about thirty-five minutes to complete.

Stages

Problem description

The example simulates the production life of an oil reservoir, 5000m by 5000m and 60m thick. The reservoir fluid consists of live oil and gas, with an aquifer of uncertain volume. The second layer has numerous shale components, introducing a lower permeability in this layer. The reservoir is subdivided into a 10x10x4 grid, and history matching concentrates on determining the aquifer contribution to total energy.

Data preparation

1 Create a working directory in a convenient place.

2 Copy all datafiles from the `Tutorial 2` directory, normally residing on `/ecl/2007.1/office/tutorials/example2`, to the current working directory.

Open new project

1 To begin, start ECLIPSE Office.

2 Select File | New Project.

3 Call the project `Build1`.

4 Select File | Save Project to save the project file to disk.

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

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

Model definition: data input

1 Click on Data to activate the Data Manager Module (DMM).

2 Select Case Definition.

3 Activate the following options:

Simulator	Black Oil
Title	Tutorial 2: Workflow
Simulation Start Date	1 Jan 1990
Model dimension (x,y,z)	10, 10, 4
Units	Metric
Run Type	Normal

4 Select the Reservoir tab.

5 Activate Aquifers and select Analytical

6 Grid Option: Cartesian

7 Geometry Option: CornerPoint

8 Select the PVT tab

9 Activate Water, Oil, Gas, Dissolved Gas.

Hint The Dissolved Gas option is only active if the OIL option has been selected.

Hint Input and output file formats are selected on the Run Manager panel, as well as the `NOSIM` option to initialize the model and perform data checking.

10 Select the Misc tab

11 Change the Stack size of previous search directions (`NSTACK`) to `50`

12 Accept the defaults for the remaining input parameters.

13 Click on OK to store the changes and exit.

14 Select File | Save Project from the main project panel to save the Case Definition Section.

Geological definition of model and grid building

1 Select Data Manager: Grid to open the grid definition section, where the geological properties are defined.

2 Select Grid Section: File | Import File | New... and import the `GRID1.GEC` file by selecting the file from the browser.

Note If the file contains the SPECGRID keyword then it is not necessary to specify the dimensions in the Case Definition Section, as ECLIPSE Office inserts the grid size it reads from the keyword. The SPECGRID keyword must appear before any array-based keywords.

3 Select Grid Section: File | Save As...

4 Check that the Directory for all the GRID Section Files: entry matches the selection you wish.

5 Accept the default root name, which should be Build1.

6 Click on the Advanced... button. The GRID header, geometry and parallel/operational INCLUDE files are saved. The Properties and Other INCLUDE files are not saved yet as they contain no keywords.

7 Click on Save.

8 Select Grid Section: Subsection | Grid Keywords.

9 Check that the data for the keywords COORD and ZCORN have been inserted correctly.

Hint Select Grid Keywords: View | Keywords to show the keyword names.

10 Select Properties from the Keyword Type list to insert porosity and permeability.

11 Select Grid Keyword Section: Edit | Insert Keyword, and from the list of keywords, select PORO.

12 Select Grid Keyword Section: Edit | Box to display the Array Box Selection panel.

Setting the porosity in the horizontal plane

1 Change the K-value range to be from 1 to 1, and enter .35 into the Data Value box. Make sure that the Operation is set to Equals.

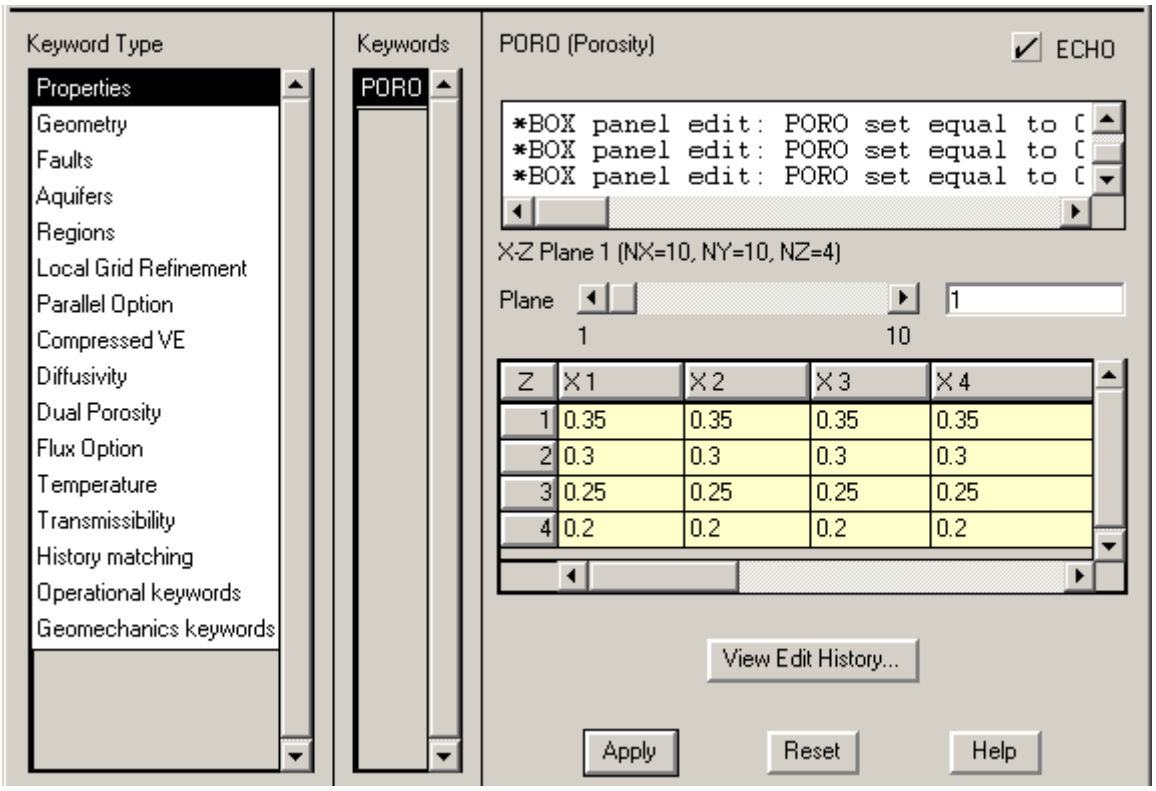
2 Click on Apply.

3 Repeat this action for the other 3 planes, with porosity values of .3, .25 and .2 for each of K = 2, 3 and 4, respectively.

4 Click on Array Box Selection: Close.

5 Select Grid Keyword Section: View | Grid Order | XZ-plane to see the following screen (Figure 4.7).

Figure 4.7 DMM - GRID Keyword section



6 Select View Edit History... in the PORO panel to display a table containing the edits which have been applied to the current keyword.

7 Select any row in the Edit History panel to display comments for a particular edit.

Note Comments which have been automatically added by Office are preceded with a "*". You can use this panel to edit the comments.

---

8 Select Cancel to close the Edit History panel.

9 To insert the permeability data, Grid Keywords Section: Edit | Insert Keyword, and from the list of keywords, select `PERMX`. The table for data input is displayed.

10 Select Grid Keyword Section: Edit | Box to display the Array Box Selection panel.

Setting permeability per plane

1 Change the K-value range to be from `1` to `1`.

2 Enter `1` into the Data Value box, and select D for Darcy from the Unit dropdown-list.

3 Make sure that the Operation is set to Equals.

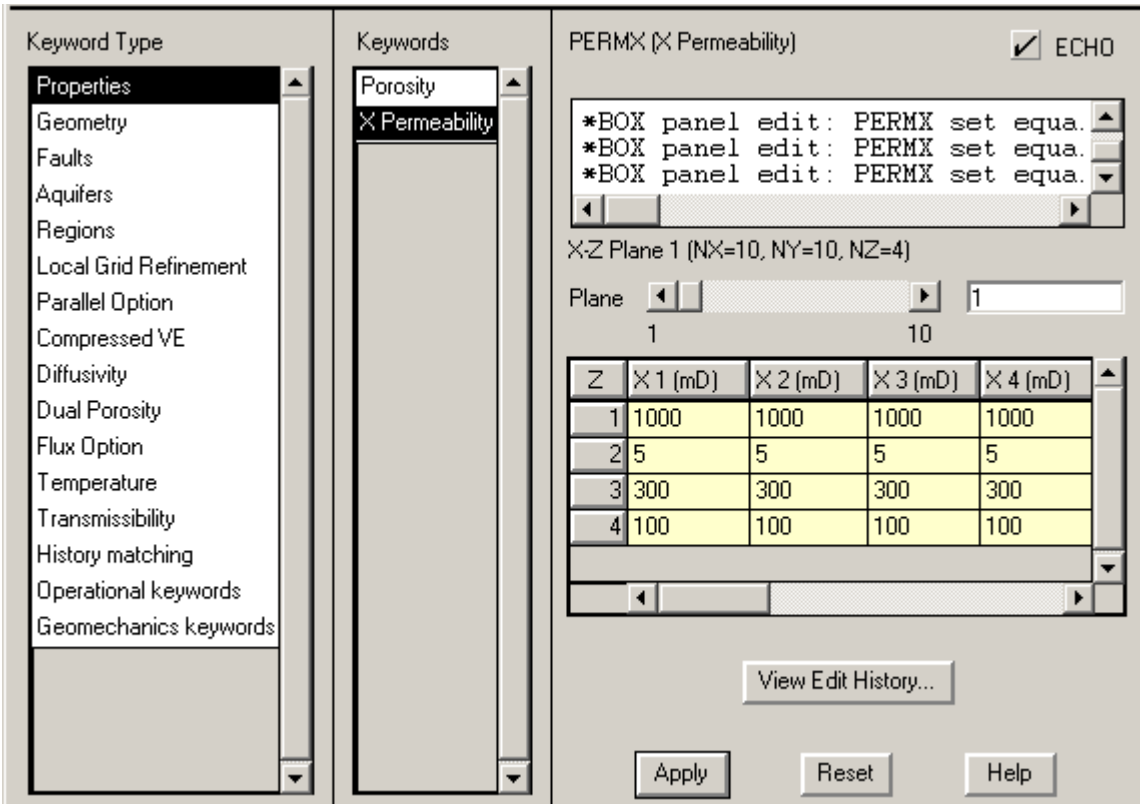
4 Click on Apply.

Note The data is converted to the correct unit as needed for the simulator, based on the selection of model units in the Case Definition Section - in this case, `METRIC` - when the data is saved to disk.

---

5 Repeat this action for the remaining 3 planes, with permeability values of .005 D, .3 D and .1 D, for each of K = 2, 3 and 4, respectively. Data should appear as in Figure 4.8.

Figure 4.8 Data Manager module Grid Keyword Section



6 Use option Grid Keyword Section: Edit | Insert Keyword to display the Keyword Selection box. Select PERMY and PERMZ from the list.

7 Click on Properties in the Keyword Types list if it is not active already. This lists the current property keywords in the model.

8 Select PERMY from the list of keywords.

9 Select Edit | Box from the menubar.

Setting PERMY to equal PERMX

1 Select COPY on the Operation drop-down list.

2 Select PERMX from the Copy from drop-down list.

3 Leave the I,J,K ranges at their default values.

4 Click on Apply to perform the operation.

Setting PERMZ equal to .1*PERMX:

1 Click on the PERMZ keyword in the Keyword list, to activate it.

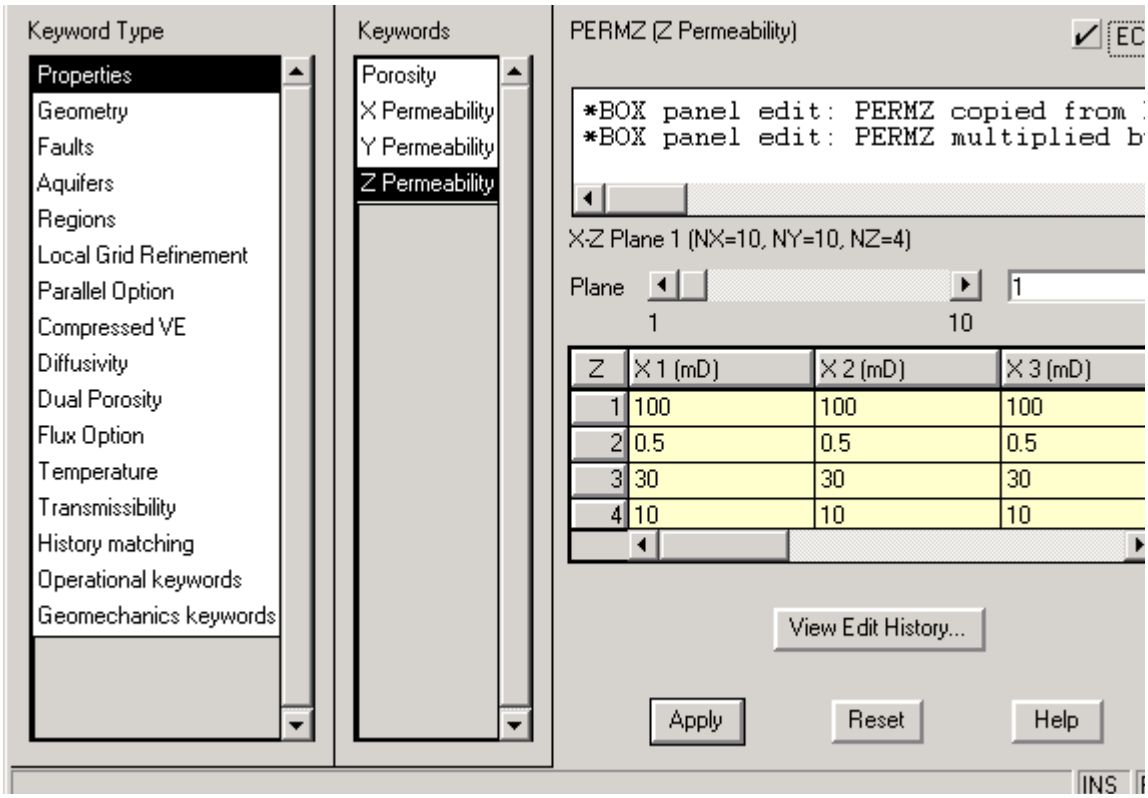
2 Repeat the Box | COPY operation to copy PERMX into PERMZ.

3 For the next action, select Multiply as the Operation, Multiplier=.1.

4 Close the Array Box Selection edit panel.

The results should be the same as in Figure 4.9.

Figure 4.9 Grid Section keywords



5 Select View Edit History... to display a panel containing **all** edits which have been applied to property keywords.

Note Edit | Delete Edit History consolidates all the edits into a single keyword. This is identical to the pre-2002A behavior.

A `GRID` file is required by all sections in order to display region and property data during the model building phase of the project. It can be recognized by the file name extension `.GRID` or `.EGRID` for an unformatted or binary file, and `.FGRID` or `.FEGRID` for an `ASCII` or formatted file. The output format is selected on the main Run Manager panel.

6 To ensure that a `GRID` file is written as part of the output, select Operational Keywords from the Keyword Types list.

7 Confirm that the `GRIDFILE` keyword is in this list.

Note ECLIPSE Office writes `GRIDFILE 2` to the `_geom.inc` file in all cases to ensure that all data required in the Result Viewer is available.

---

8 Select GRID Keyword Section: Edit | Insert Keyword.

9 From the same list, insert `INIT` to write the file containing initial properties of the grid. File extension `.INIT` or `.FINIT`.

10 Click on Apply.

11 Click on File | Close to exit the Grid Keywords panel.

12 Select Grid Section: File | Save... and save the geometrical data.

13 Select Grid Section: GridView | From Keywords to generate a gridfile for 2D and 3D viewing.

14 Select YES to create the GRIDFILE.

15 Select Grid Section: GridView | 3D to see the 3D image of the simulation grid.

16 Close the 3D Viewer with File | Close.

17 Select Grid Section: File | Close to exit.

PVT section: fluid definitions

1 Select Data Manager: PVT to insert the PVT data.

2 Select File | Import | New... and insert the data from the pvt2.pvo file.

3 Select PVT Section: File | Save As, and use the default filename to save the data imported from the file to the Build1_pvt.inc file.

4 Update the project file at the same time, by selecting the Update Current Case and Save Project File options on the File Save As panel.

Hint Update Current Case updates the main ECLIPSE Office Case Manager panel with the link to the PVT data, and commits the data internally.
Save Project File saves the project data, with the link, to the main Build1.OFF file on disk.

---

5 Select PVT Section: Section | Keywords from the menubar.

6 Select from the menubar PVT Keywords: Keyword Types | PVT Tables. This may be active already.

7 Observe the data inserted from the imports as in Figure 4.10:

   a PVDG, which contains the Dry Gas fluid properties,

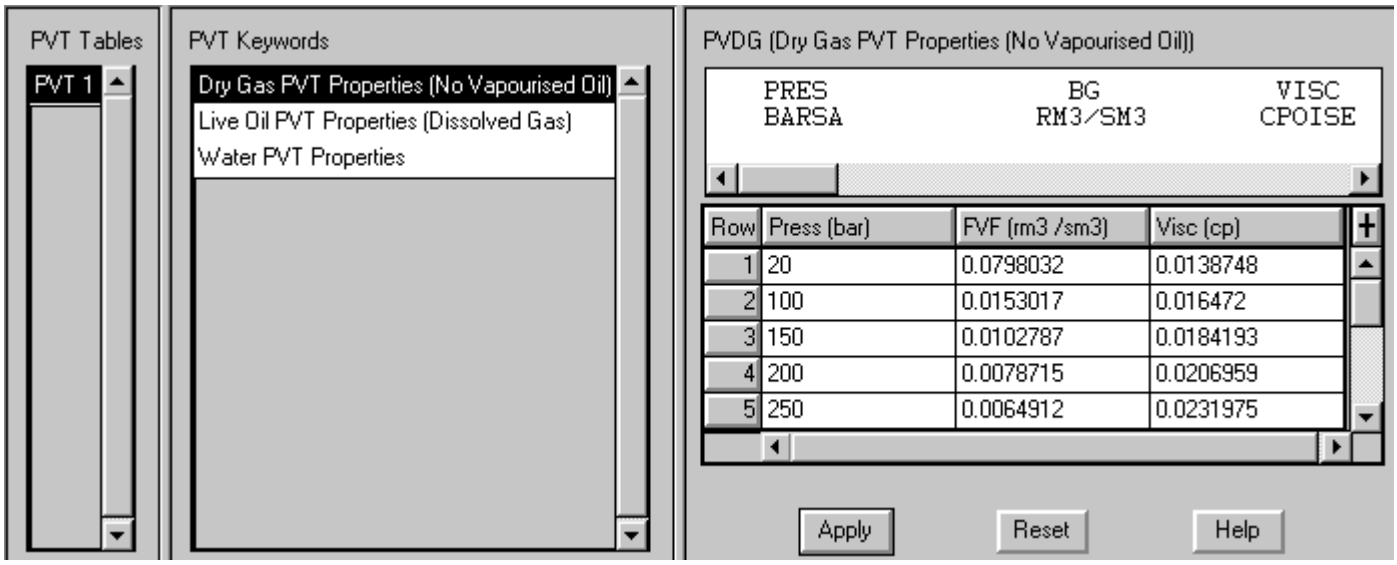
   b PVTO with the Oil properties, and

   c PVTW the water properties.

   d Only one PVT region has been specified: PVT1.

Hint Menu option PVT Keywords: View | Keywords displays keyword names.

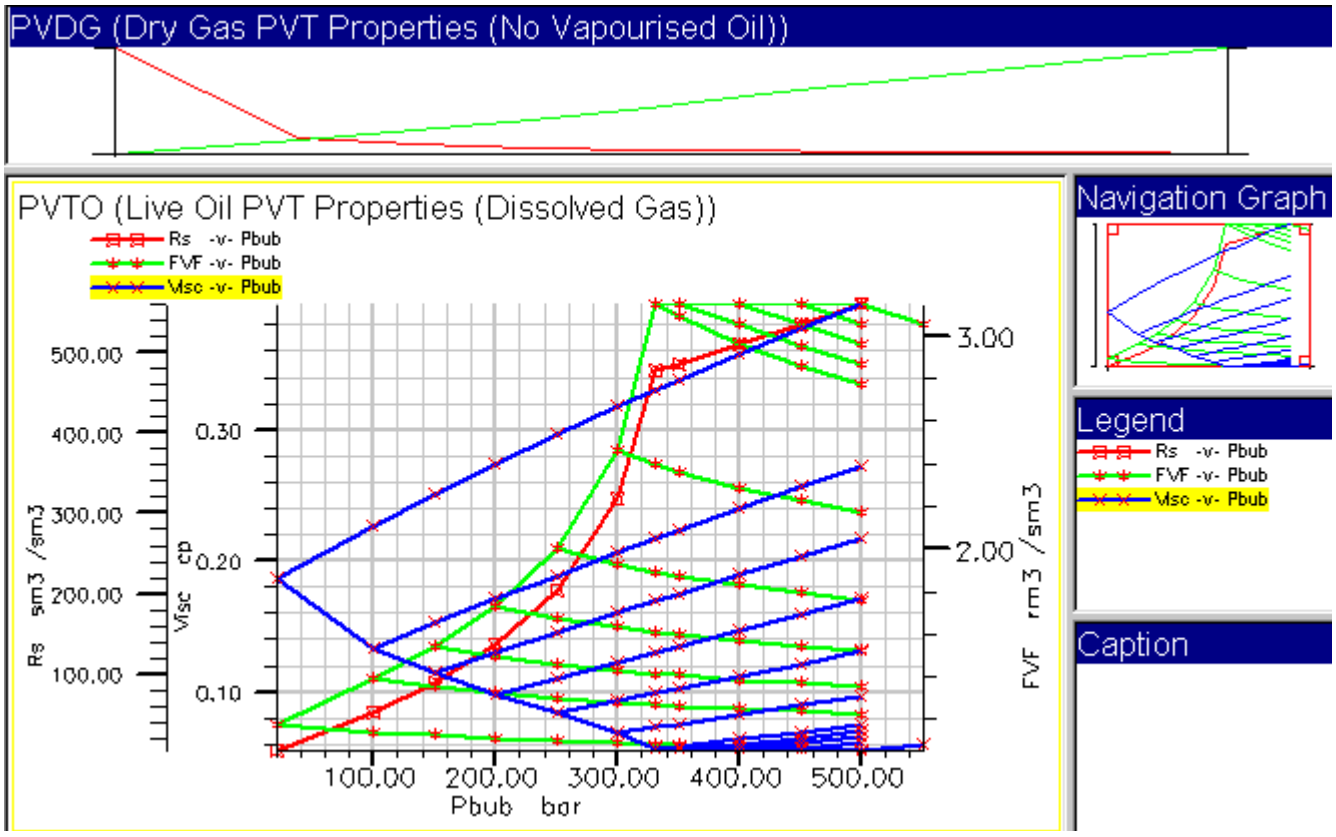
Figure 4.10 DMM - PVT keywords



8 Select PVT Keywords: View | Plot to view the data graphically.
This shows both Oil and Gas properties.

9 Double-click on the top section workspace to toggle between the two graphs.

Figure 4.11 PVDG and PVTO properties for the BUILD1_E100 model



10 Select File | Close to return to the PVT Keywords panel.

11 Select PVT Keywords: Edit | Insert keyword

12 Select DENSITY from the list of Keywords.

13 Insert the following values:

Oil density	749.389 kg/m^3
Water density	1000 kg/m^3
Gas density	1.11242 kg/m^3

14 Click on Apply to commit the data.

15 Select Edit | Explore keyword|Toggle Desc/Keys and then select the ROCK keyword.

16 Enter:

Reference pressure	400 bar
Rock compressibility	4.0E-5 /bar.

17 Click on Apply to commit the data.

18 Select PVT Keywords: Keyword Types | Miscellaneous from the menu bar.

19 Select Edit | Explore keyword|Toggle Desc/Keys and then select the RPTPROPS keyword.

20 Switch on the outputs of Oil PVT tables and Gas PVT tables.

21 Click on Apply to commit.

22 Select File | Close to exit from the PVT Keywords section.

23 Select PVT Section: File | Save to save the data to the same file that initial save was done.

Note If any regions have been specified using PVTNUM, the Region File Save panel appears, to allow these to be written to the Build1_reg.inc file.

---

24 Select PVT: File | Close to exit.

SCAL section: saturation definitions

SCAL keywords

1 Select Data Manager: SCAL.

2 Select File | Import | New... and read the saturation keyword file scal2.rcp.

Hint Change the File Browser filter to search for *.rcp if required

---

3 Select SCAL Section: Section | Keywords.

4 Use SCAL Keywords: View | Plot to graphically view the Saturation versus Relative Permeability curves.

Values may be changed in the table until the graphs are more acceptable. The graph is updated automatically when you click on the Apply button.

5 Select File | Close to close the Graph panel.

6 To return to the main SCAL Section panel, select SCAL Keywords: File | Close

7 Select SCAL Section: File | Save As... to save the data to disk, as well as the project. This section is saved to `Build1_scal.inc`.

8 Accept the default filename.

9 Select SCAL Section: File | Close to exit.

Initialization section

1 Select Data Manager: Initialization.

2 Select Initialization Section: Edit | Insert Keyword.

3 Select `EQUIL` to specify the initial equilibration conditions.

4 Insert the following data for `EQUIL`:

Datum depth	3000 m
Pressure at datum depth	331.65 bar
WOC depth (Oil-Water contact depth)	3085 m
OW Cap Pressure (Oil-water Pc at OWC)	0 bar
GOC (Depth of Gas contact)	3000 m
GO Cap Pressure (Gas-Oil Pc at GOC)	0 bar
RS/Pb v Depth table number	1
Rv/Pd v Depth table number	1

To indicate that the default value should be used in the simulation, the data field should be left empty, in which case ECLIPSE Office writes "1*" as a placeholder in the file.

Leave the Accuracy field empty, so that the default value is used in the simulator.

5 Click on Apply to commit the data.

6 Insert the `RSVD` keyword.

7 Insert data for `RSVD`, using the `TAB` key to move between columns, and the + symbol on the top right corner of the table to add a row.

3000m	477.91 sm^3/sm^3
4000m	486.6 sm^3/sm^3

8 Click on Apply.

Aquifer data

1 Select Initialization Section: Keyword Types | Aquifer to display aquifer related keywords.

2 Select Initialization Section: Edit | Insert Keyword.

3 Click on `AQUFETP` to define the general aquifer parameters for a Fetkovich aquifer, and `AQUANCON` to define the aquifer connections with the reservoir.

4 Click on Fetkovich Aquifer and insert data for `AQUFETP`:

Note To change the unit of a column, click on the column with the right-mouse-button and select Change Units from the menu displayed. The Initialization: Aquifer Connections panel appears on which pressure units kPa can be selected from the drop-down lists. Save to disk converts the units as needed.

5 Enter the following data:

Id of Aquifer	1
Datum depth	3113.2 m
Initial Aquifer Pressure	337.582 bar
Initial Aquifer Volume	18.0E9 m3
Total Compressibility (Rock + Water)	5.0 E-5 /kPa
Aquifer Productivity Index	500 sm3/day/bar
PVT Water Property Table	1

6 Leave the Salt Water Concentration field empty.

7 Click on Apply to commit.

Hint Table edit functions are displayed when the right mouse button is pressed with the cursor on the table.

8 Insert data the following data for `AQUANCON`, using the `TAB` key to move between columns. Click on the + sign on the top right corner of the table with the cursor in last column to add a row.

The connection face (J/I) is selected from a dropdown list that becomes active once you click in the cell.

Aquifer ID	Lower I Con	Upper I Con	Lower J Con	Upper J Con	Lower K Con	Upper K Con	Connection Face
1	1	10	1	1	4	4	J-
1	1	10	10	10	4	4	J+
1	1	1	1	10	4	4	I-
1	10	10	1	10	4	4	I+

9 Click on Apply to commit the data.

Inserting keywords

1 Select Initialization Section: Keyword Types | Miscellaneous and then Edit | Insert Keyword to insert keyword `RPTSOL`.

2 Check the parameters:

a Grid Block Pressures, Oil, Water and Gas Saturations

b Restarts/Initial Restart and Fluid In Place Reports with Balance Sheet.

3 Click on Apply to commit the options.

4 Use File | Save As... to write the data to the file `Build1_INIT.INC` and update the project with the section.

5 Select Initialize Model | Run Simulation. This runs the simulator to the first time step. An initial restart file is created which contains details of grid block pressure and saturations at the beginning of the run.

6 Select Initialize Model | FIP Report to see initial fluids in place. Select this option again to return to the keywords.

Note If more than one Fluid-In-Place region has been defined using FIPNUM, then the panel will contain data on a region-by-region basis as well as at field level.

---

7 Select Initialization Section: File | Close to exit from the Initialization Section.

Schedule section: production schedule

1 Select Schedule from the Data Manager list.

2 Use the Schedule Section: File | Import | New... to import the file `sched2`. All initial well specifications, well completion and well controls are specified in this file.

Hint Change the File Browser filter to `*.sch`, if required, to make it visible in the browser window.

---

3 Use File | Import | Append... to add the history data from the file `wconhist`.

Note Ignore the warning that the `SCHEDULE` keyword does not exist, and continue.

---

4 Select File | Save As... to save the data to the default ECLIPSE Office file `Build1_SCH.INC`. Update the project with the data at the same time.

5 Click on each time/date to see the events connected to it.

6 Toggle between keywords and descriptions of events using View | Description and View | Keyword.

7 Select Schedule Section: File | Close to close the section.

Note If the Generic Keywords option is ON in the Dimension Overrides section:
1. In Data Manager > Section > Dimension Overrides, select Keyword Well Dimensions.
2. Change the 4 dimensions to 4, 3, 2, and 5 respectively. Click on Apply and close the Dimension Overrides Section.

Summary section

1 Select Data Manager: Summary

Summary variables contain vector data, such as well production rates, for each report step, and are saved in the `.Snnnn` files during the simulation run.

2 Select Summary Section: File | Import | New... to import the summary vectors from the file `summary.dat`.

3 Click on the Selected button to see the vectors which have been loaded.

Adding summary vectors

1 Click on the Keywords button to return to the panel where new output mnemonics may be added.

2 Select the Field tab on the Summary Section panel

3 Select others from the Phases list, and Pressure from the Types list. FPR will be listed in the Keyword selection box.

4 Click on FPR in the Keywords list to make it active.

5 Click on Add to List to add this keyword to the selected list of summary vectors.

Producing bottomhole pressures for all wells

1 Select the Well tab.

2 Select others from the Phases list, and Pressure from the Types list.

3 Select WBHP from the Keyword list.

4 Select All wells in the Wells list.

5 Click on Add to List to insert the keyword into the selected list.

6 Use Summary Section: File | Save As... to save the list to the project and the summary include file Build1_SUM.INC, on disk.

7 Summary Section: File | Close closes the Summary Section and returns you to the main Data Manager panel.

8 Select Data Manager: File | Close to return to the main ECLIPSE Office project panel.

Run manager

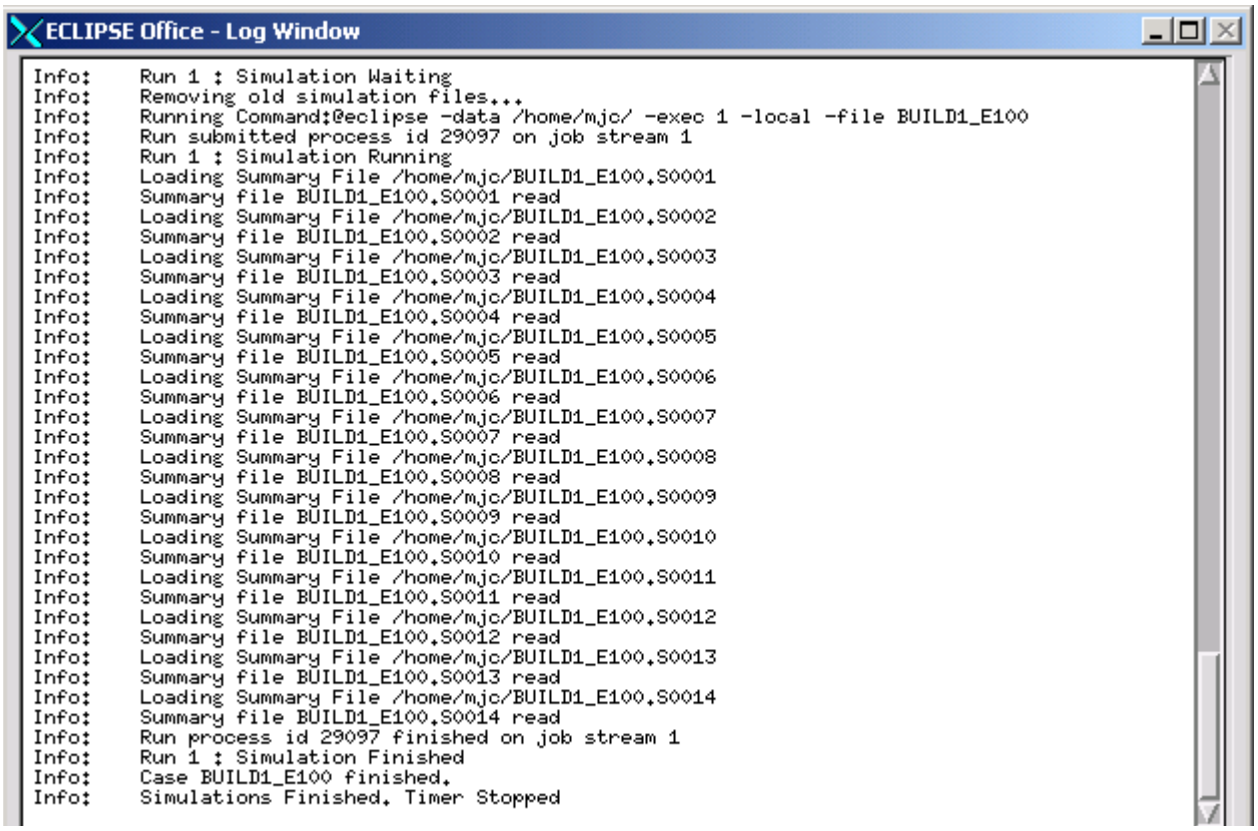
1 Click on Run on the main ECLIPSE Office panel to activate the Run Manager.

2 Select Run Manager: Submit | Runs to submit the simulation run. 

Hint If a run has been attempted on a previous occasion with the same project, the file Build1_E100.DATA might exist. In such a case the program gives you the opportunity to either use the previous data file, or overwrite it with the current data. Choose the YES option to overwrite the existing file.

3 Check the Log window to check if submission was successful and to monitor the progress of the files written to disk at each reportstep. (See Figure 4.12).

Figure 4.12 Log window with information on runs



4 Select Run Manager: Monitor | Control Simulations for information about the current run. This option is only activated once the simulation started.

5 Check the Log window for messages about the simulation.

6 Select Run Manager: File | Close once the simulation run is finished.

Report generator

1 Click on Report on the main ECLIPSE Office project panel, to activate the Report Generator.

2 Select Report Generator: File | Open Current Case | PRT... to open the `PRT` file created by the simulation run.

Note The `BUILD1_E100.PRT` file contains all output requested by the `PRT*` keywords selected in each data section of the data model; as well as a report on the errors or problems found during the simulation run.

---

3 Select Errors... from the Report drop-down.

Note It is sometimes useful to look at the complete `.PRT` file in order to resolve errors with regard to erroneous data in keyword arguments. In the `.PRT` file the error is generally directly after the keyword involved. The Util | Text Editor can be used at any point in time to view this file.

4 Tick ERRORS and WARNINGS then click on Generate Report.

If there are no errors, the ERRORS option is not available for selection.

Note If errors appear with a `Simulation Failed` message in the Log window, the errors listed in the report have to be resolved in the Data Manager before attempting a simulation run again.

5 Select Output to see the report.

6 Exit from the Report Generator module with the File | Close option.

7 Save the project on the main ECLIPSE Office project panel, using the menu option File | Save Project.

Note See "Report Generator" on page 316 and "Result Viewer" on page 284 to view the simulation results. "Tutorial 4: History matching using ECLIPSE Office and SimOpt" on page 86 shows the basic procedure of a History Matching process, using the same model as above.

8 Select File | Exit to exit.