Tutorial 4: History matching using ECLIPSE Office and SimOpt

Introduction

This tutorial examines the integration of an ECLIPSE Office project with the SIS history matching aid, SimOpt. In this example an existing simulation, along with associated historical well water-cut data, is imported into ECLIPSE Office to provide a basic project and examine the match between the simulation model and the well history.

The project is then exported into SimOpt, where the match can be analyzed using the ECLIPSE gradients option and a regression problem can be set up that is used to improve the history match.

The matched simulation model is then exported from SimOpt and associated with the ECLIPSE Office project as a new case.

Prediction simulations are made using the multiple runs options in ECLIPSE Office.

This tutorial has been broken down into stages. Each stage is self-contained and can be used independently of the full tutorial.

Hint If you are new to ECLIPSE Office or have not completed this tutorial before you are advised to complete all stages in the order in which they appear.

This tutorial takes about forty-five minutes to complete.

Stages

Setting up an ECLIPSE Office project from an existing simulation model

1. Start ECLIPSE Office.
2. ECLIPSE Office: File | New Project…
3. Call the new project `SNARKSIM`.
4. ECLIPSE Office: Case | Import…
5. Import the data-set `SNARKSIM.DATA`

6 ECLIPSE Office: Module | Run Manager…

7 Run Manager: Submit | Runs

8 ECLIPSE Office: Module | Results Viewer…

9 Result Viewer: File | Open Current Case | SUMMARY…

10 Select Read All Summary Vectors in the Extract/Load Summary Vectors panel.

11 Select Read All Reports in the Extract/Load Summary Vectors panel.

12 Click on Load in the Extract/Load Summary Vectors panel.

13 Results Viewer: LinePlot | User…

14 Select TIME in the X-Axis Vectors list of the User Templates panel.

15 Select WWCT:PROD1 in the Y-Axis list of the User Templates panel.

16 Click on Add to List.

17 Click OK in the User Templates panel.

Comparing simulation results and observed data

Now that the ECLIPSE Office project has been set up and the simulation run, the results can be compared with the observed well water-cut values.

1 Results Viewer: File | Open Observed | Column Format

2 Open the file `prod1.wwct`

3 Set the line numbers for the column format in the Column Format User Data panel from their original settings to the required settings as shown in the following table, and then click on OK .

Table 4.1 Line Numbers when importing Column Format User Data

Line Descriptions	Original Line Number	Required Line Number
Mnemonics	1	1
Units	2	2
Scale Factors	3	0
Well or Group Names	4	3
Lgr Name or Number	5	0
Local Cell Numbers	6	0
First Line of Data	7	4

4 Results Viewer: LinePlot | Observed…

5 Select TIME in the X-Axis Vectors list of the Select Observed Vectors for plotting panel.

6 Select WWCT:PROD1 in the Y-Axis Vectors list of the Select Observed Vectors for plotting panel.

7 Click on Add to List.

8 Select Add to Graph.

9 Click on OK.

Discussion

The match between the simulated and historical water-cut data for well PROD1 shows that the simulation model currently does a reasonable job over the period of the simulation, but has far too much water produced far to soon in the early history period and insufficient water produced in the late history period. This could be due to transmissibility across the faults being too high, or perhaps Z-transmissibility being too high. Stage 3 of this tutorial, "Exporting a SimOpt simulation model for import into ECLIPSE Office" on page 94, will use SimOpt to examine these parameters and attempt a better history match.

Before this can be done, however, the simulation model must be made ready for SimOpt."Exporting an ECLIPSE Office project for use in SimOpt" on page 88 of this tutorial examines the transition from an ECLIPSE Office project to a SimOpt project.

Exporting an ECLIPSE Office project for use in SimOpt

If you did not complete Stage 1 before starting this section of the tutorial, you will need to set up an ECLIPSE Office project with the data file `SNARKSIM.DATA`.

Hint If you are not sure how to set up an ECLIPSE Office project, see "Setting up an ECLIPSE Office project from an existing simulation model" on page 86

The first step in preparing a simulation model for SimOpt is to create a new case in the ECLIPSE Office project.

1 ECLIPSE Office: Case | Add Case | Clone

This creates a new ECLIPSE Office case that contains the same information as the default case. The New Case Name panel is displayed.

Note SimOpt appends the imported data set name with _1 in the file-root name (that is `BASE.DATA` becomes `BASE_1.DATA`). This is so that SimOpt versions of a simulation model are not confused with the original data set. It is important that the original dataset file-root name does not end with an underscore and a number; SimOpt rejects such datasets as they can be confused with the SimOpt versions of the simulation model.

2 Change the Case Name to `ORIGINAL`.

3 Click on OK.

4 Click on the `ORIGINAL` case to make it the active case.

5 ECLIPSE Office: File | Save Project.

Creating the INIT file

SimOpt requires `GRID` and `INIT` files as well as the dataset. By default, the `GRID` file is generated by this case. Before exporting the simulation model `ORIGINAL` from ECLIPSE Office, we need to create the `INIT` file.

1 ECLIPSE Office: Module | Data Manager…

2 Data Manager: Section | Grid…

3 Grid Section: Subsection | GRID keywords…
4 GRID Keyword Section: View | Keywords
5 Select Operational Keywords in the Keyword Type list.
6 GRID Keyword Section: Edit | Insert Keyword...
7 Select Output INIT file in the Keyword Selection panel.
8 Grid Section: File | Save
9 Click on Yes then Save to save the current contents of the Grid Section.
10 Grid Section: File | Close
11 Data Manager: File | Close
12 ECLIPSE Office: Module | Run Manager…

Hint This run is only to generate an `INIT` file (there were no changes to the simulation). The simulation can be turned off to save time.

---

13 Click on Turn Off Simulation (at the bottom right of the Run Manager).
14 Run Manager: Submit | Runs
15 After the run has finished, the file must be written without the `NOSIM` keyword, ready for SimOpt to read it.
16 Click on NOSIM (Turn Off Simulation) so that it is no longer checked.
17 Run Manager: File | Write Data.
18 Close the Run Manager and save the settings.

Discussion

SimOpt requires a working simulation model and a `GRID` and `INIT` file to set up a project. ECLIPSE Office can provide the data-checking and keyword editing facilities to create the files for a SimOpt history matching project.

Setting up a history matching project in SimOpt

1 Start SimOpt and call the project `SNARK`.
2 The Getting Started panel appears. Click on the Specify DATA file button.
3 Import the file `ORIGINAL_E100.DATA`.
4 In the Getting Started panel, click on the Specify GRID file button.
5 Import the file `ORIGINAL_E100.EGRID`.

Note The file `ORIGINAL_E100.INIT` has also been imported during this step.

---

6 SimOpt: Project | Import | Observed Data | Production | Graf User Vector Data Format…
7 Import the file `GRAF.WWCT`.

8 Click on the OK button on the message box that warns you that you must set measurement errors.

Now that the basic data for a SimOpt project has been entered, save the project.

9 SimOpt: Project | Save

Note The Well Water Cut data (`WWCT`) that has been imported has no estimate of the error in the measurement. This information is necessary for SimOpt to calculate the RMS (root-mean-square) fit between the observed and simulated data, and so must now be added manually.

---

10 SimOpt: Observed Data | Control...

11 Select Set Meas. Error on the Block Action (Selected Set) radio button.

12 Enter a value of `0.01` in the Meas. Error text entry field.

13 Click on Apply Block Action.

14 Click on Close.

Sensitivity model

The simulation model and observed data are now ready for sensitivity studies. In this case the three faults in the model and the Z-transmissibility of the entire grid will be used in the sensitivity model.

1 SimOpt: Parameter | Add…

2 Select ZTrans in the Parameter Type list and click on Apply.

3 Select Fault on the Domain radio button.

4 Select FaultTr in the Parameter Type list.

5 Click on FAULTS1 in the Domain Names list and then `Shift+Click` on FAULTS3 to select all three faults.

6 Click on Apply.

7 Click on Close

8 Click in the + sign to the left of the Version 1 node in the Parameter Versions hierarchy tree (that is, the central panel on the main screen).

Defining the transmissibilty multipliers

The fault transmissibility multipliers act along the entire length of each fault. The Z-transmissibility multiplier, however, acts on a number of grid cells and so its region of activity must be defined.

1 SimOpt: Parameter | IJK Region Editor...

2 IJK Region Editor: Region | Define | Using Box(es)...

Now we define the region to cover the entire grid:

3 Enter the following data in row 1 of the table and then click on OK:

I1(nI:14)	I2(nI:14)	J1(nJ:14)	J2(nJ:14)	K1(nK:5)	K2(nK:5)
1	14	1	14	1	5

4 Click on OK.

5 IJK Region Editor: File | Close

Defining starting values

Now that the parameters of interest have been defined, their starting values and upper and lower bounds must be set.

1 SimOpt: Parameter | Control...

Hint The fault transmissibility multiplier varies between 0 (a sealing fault) and 1 (an invisible fault). If there is no initial estimate of the multiplier's value, the usual situation, the following rule of thumb can be applied:
A change in the transmissibility of a near-sealing fault generally has a much greater effect on reservoir flow (and thus greater sensitivity) than a change in the transmissibility of a near-invisible fault. Consequently, it is usually useful to set the initial value of the fault modifiers to, say, 0.1 and allow the value to vary between 0 and 1.

Hint Vertical transmissibility estimates can be in error by a factor of 100 or more. Consequently it is usually wise to allow the Z transmissibility multiplier to vary from the original simulation model value (that is a multiplier of 1) down from and up to 100 hundred times the original value, that is from 0.01 to 100.

2 Enter the following data in the table in the Parameter Control panel:

Table 4.2 Parameter control starting values

Type	Domain	Modifier	Modifier Min.	Modifier Max.
ZTrans	Region1	1	0.01	100
FaultTr	FAULTS1	0.01	1E-5	1
FaultTr	FAULTS2	0.01	1E-5	1
FaultTr	FAULTS3	0.01	1E-5	1

Hint The most efficient way to change the modifier value, minimum value and maximum value for the fault transmissibility parameters is to use the Type filter to make the table display only the parameters of type FaultTr, then use the block actions Set Modifier Value, Set Minimum Value and Set Maximum Value to make the required changes to all the fault transmissibility parameters at the same time.

3 Click on Apply in the Parameter Control panel.

4 Click on Close.

Discussion

A project in SimOpt contains a simulation model, `GRID` and `INIT` files, some observed well histories (with standard errors for the observations), and parameters that are varied to try to improve the history match.

The project is now complete, and so sensitivities can be calculated. By taking advantage of the Gradient option in ECLIPSE, the sensitivity analysis can be performed in a single simulation run. The next section examines the analysis of sensitivities to provide a basis for regression to improve the history match.

Analyzing the current history match and designing a regression problem

1 SimOpt: Analysis | Simulate…

2 Select Gradient from the Run Mode radio buttons and then click on Start (and click on Yes when prompted to save the project).

Hint To view the simulation run while it is in progress, click, with the right mouse button, on the node for well PROD1 in the Observed Data menu and select Plot. This plots the observed data for well PROD1 with the simulation superimposed.

3 When the simulation has finished, look in the log window at the bottom of the main window.

The RMS fit between all the observed and simulated data is around 24.

Note The multipliers for FAULTS2 and FAULTS3 are marked as redundant. This means that they are not important to the fitting of the data and so can be turned off.

4 SimOpt: Parameter | Control…

5 Set the activity of FAULTS2 and FAULTS3 to Semi-active using the drop-down menus in the Activity column of the table.

Note The “semi-active” activity means that these parameters are no longer used in the analysis and their gradients are no longer calculated during a simulation, but their current modifier values are still used on the model. The only difference when using the alternative “inactive” activity is that the current modifier values are not used.

6 Click on Apply

7 Click on Close.

Reviewing the observed data and remaining parameters

Now that the redundant parameters have been removed from the analysis, the remaining parameters can be viewed.

1 SimOpt: Analysis | Match…

2 Click on Update.

The RMS folder shows that the RMS and sensitivities of well PROD2 are zero. This is because well PROD2 shuts at time zero so the simulation will always perfectly match the (zero water cut) observed values. Including this observed data in the history match is pointless; it only serves to make the overall match (as reported by the RMS) look better than it really is. For this reason, this observed data should be taken out of the history match problem.

Note In general, all observed data in the history matching project should be examined carefully to make sure that it really contributes to the history matching problem.

---

3 SimOpt: Observed Data | Control...

4 In the table, set the Status of the `WWCT` data for PROD2 to Inactive. Click on Apply.

Note This does not delete the data from the project. It merely removes it from the RMS calculation. The data can be added back into the RMS calculation at a later stage by setting its status to active.

---

5 Click on Close.

Match analysis panel

1 Click on Update.

This recalculates the analysis without including the PROD2 data.

2 Click on the Correlation folder.

The off-diagonal elements of the correlation matrix are around 0.33, and so the two remaining parameters are not strongly correlated. If parameters are strongly correlated, the problem may be mathematically ill-posed (which means that regression may not improve the match). In this case the problem seems well-posed and so it is possible to use non-linear regression to iteratively update the modifier values to improve the history match.

Note It is important to remember that the analysis is linear and that the response of the simulation model to changes in the multipliers will be non-linear. Consequently, it is unwise to regress for more than 3 or 4 iterations, as each iteration takes the simulation model further away from the point at which the analysis was valid.

---

3 SimOpt: Analysis | Simulate…

4 Select Regression in the Run Mode radio button.

5 Set the Max Runs Simulator to 3

6 Click on Update parameters (and click on Yes in the warning dialog that appears). This updates the parameter values using the gradients already calculated for the parameters.

7 Click on Start (and click on Yes when prompted to save the project).

Hint If you want to view the progress of the RMS during the regression, click on the Plot RMS button. (This updates after each simulation run). As before, runtime monitoring of well data is available.

The final RMS fit is 11.7, a big improvement over the ORIGINAL data-set imported into SimOpt.

8 To see the match between observed and simulation data, select the PROD1 WWCT node on the `Observed Data` hierarchy tree.

9 Select Observed Data | Plot to see the improved history match

Discussion

Following a sensitivity run, the gradients were analyzed using tools available in SimOpt. This analysis facilitated the design of a well-conditioned regression problem. SimOpt was then used to run the ECLIPSE simulator, iteratively updating the selected parameters to provide an improved history match.

Exporting a SimOpt simulation model for import into ECLIPSE Office

The new simulation model that resulted from history matching can now be exported from SimOpt and used to make prediction runs using the multiple runs feature in ECLIPSE Office.

1 SimOpt: Project | Export | Simulation Model…

2 Click on OK in the Project Export Comments panel.

3 Save the data-set with the name `MATCHED.DATA`

4 Close the file viewer displaying the exported DATA file.

5 SimOpt: Project | Exit

6 Click on Yes to save the project.

Discussion

The matched simulation (that is with the multipliers applied) can be exported from SimOpt. This is the original simulation with multiplier keywords added so that, at a later stage, the original simulation model can, if required, be retrieved.

Multiple prediction runs in ECLIPSE Office

1 ECLIPSE Office: File | New Project…

2 ECLIPSE Office: Case | Add Case | New

3 Change the Case Name to `PREDICTION` and select OK.

4 Click on the PREDICTION case to make it the active case.

5 ECLIPSE Office: Case | Import...

6 Import `MATCHED.DATA`.

Adding a prediction period

Now that the history matched version of the simulation model is in ECLIPSE Office, the next step is to increase the length of the simulation (that is add a prediction period).

1 ECLIPSE Office: Module | Data Manager…

2 Data Manager: Section | Schedule…

3 Select the last date in the data-set (1 January 2030) in the Time - Dates list.

4 Schedule Section: Time | Insert…

5 Select Time Step on the New Time Entry Choice radio button.

6 Enter `365 days` as the Time Step.

7 Enter `5` in the Num entry box of the New Time panel.

8 Click on OK.

This adds five simulation timesteps, each a year in length, to the existing data-set.

Defining the rate controls

The next step is to define rate controls for the prediction period.

1 Schedule Section: Event | New…

2 Select Well Controls and Limits in the Event Types list

3 Select Production Well Control in the Events list

4 Click on OK.

5 Enter the following information in the Schedule Section fields for the new `WCONPROD` keyword:

Well	PROD1
Open/Shut flag	Open
Control	ORAT
Oil Rate	1000 stb/day

6 Click on the Multiple Runs button.

7 In the Oil Rate column, enter a value of `3000 stb/day` in the second row of the Production Well Control table.

8 Click on OK in the Production Well Control panel.

9 Click on Apply in the Schedule Section panel.

10 Schedule Section: File | Save

11 Save the Schedule and Multiple Sensitivities file.

12 Schedule Section: File | Close.

Prediction runs

The multiple prediction runs are defined, so the prediction runs can now be made.

1 Data Manager: File | Close

2 ECLIPSE Office: Module | Run Manager…

3 Ensure that the NOSIM (Turn off Simulation) checkbox is not checked because the simulation run is required for generating Summary Vector Reports.

4 Run Manager: Submit | Runs

5 Run Manager: File | Close

Viewing results

Once the simulations have finished, the results can be seen using the Result Viewer.

1 ECLIPSE Office: Module | Results Viewer…

2 Results Viewer: File | Open Current Case | Summary…

3 Select both cases (to select a case, click on it and then click on the >> button).

4 Click on OK.

5 Select Read All Summary Vectors and Read All Reports in the Extract/Load Summary Vectors panel

6 Click on Load

7 Results Viewer: Tools | Calculate Totals | Well Production…

8 Select All in the Names list

9 Select PROD1 in the WGNames list

10 Click on Add to List

11 Click on OK.

The graph shows the oil and water production totals for the two prediction runs. It can be seen that improved total production is achieved in scenario 2 (3000 stb/day).

Discussion

In this final stage, multiple prediction runs were made in ECLIPSE Office so that the effects of different well controls on future production could be determined.