Created in COMSOL Multiphysics 6.2

Reservoir with Horizontal Wells

Introduction

This example models a thin oil reservoir with two horizontal wells, as described in the Seventh SPE Comparative Solution Projects, case 1a (see Ref. 1). This project is about oil recovery by injecting water through a bottom well. The model is used to compute the oil production rate and the water-oil production ratio over time.

Model Definition

The reservoir measures 2700 ft by 2700 ft by 160 ft, and consist of 6 horizontal layers with different initial oil saturations. The thickness of each layer and the initial oil saturation is given in Table 1. The porosity of the reservoir is 0.2 and the horizontal and vertical permeabilities are 300 mD and 30 mD, respectively. The density and the viscosity of both phases are given in Table 2, while the relative permeabilities and capillary pressure (as functions of the water saturation) are given in Table 3.

In the bottom layer water is injected into the reservoir through a horizontal well with a length of 2700 ft. The water pressure at this well is maintained at 3700 psi. The injection well is located at the following coordinates: x = 1350 ft, 0 ft < y < 2700 ft, z = 25 ft.

In the top layer a horizontal well, with a length of 900 ft, produces the fluids at a constant mass flow rate of 5.4181 kg/s, which is the equivalent of 3000 STB of water per day (1 STB, or stock tank barrel, is 0.159 m^3). This production well is located at the coordinates x = 1350 ft, 300 ft < y < 1200 ft, z = 150 ft.

The simulation time is 1500 days.

TABLE 1: LAYER THICKNESSES AND INITIAL SATURATIONS.

Layer	Thickness	Initial oil saturation
1 (top)	20 ft	0.711
2	20 ft	0.652
3	20 ft	0.527
4	20 ft	0.351
5	30 ft	0.131
6 (bottom)	50 ft	0.000

TABLE 2: FLUID PROPERTIES.

Value	Description
9814 kg/m^3	Water density
8975 kg/m^3	Oil density

TABLE 2: FLUID PROPERTIES.

Value	Description
$0.96 \cdot 10^{-3}$ Pa·s	Dynamics viscosity of water
$0.954 \cdot 10^{-3}$ Pa·s	Dynamics viscosity of oil

TABLE 3: RELATIVE PERMEABILITIES AND CAPILLARY PRESSURE AS A FUNCTION OF WATER SATURATION.

Water saturation	Rel. perm. water	Rel. perm. oil	Capillary pressure (psi)
0.22	0.00	1.0000	6.30
0.30	0.07	0.4000	3.60
0.40	0.15	0.1250	2.70
0.50	0.24	0.0649	2.25
0.60	0.33	0.0048	1.80
0.80	0.65	0.0000	0.90
0.90	0.83	0.0000	0.45
1.00	1.00	0.0000	0.00

Results and Discussion

As water is injected through the bottom well, water will start to infiltrate the higher, more oil saturated reservoir layers. This happens especially near the two horizontal wells, as can be seen in Figure 1, where the oil saturation is plotted after the production period of 1500 days: the water table is much higher in between the two wells. This phenomenon is known as coning, and it usually causes a reduction of the oil production rate and an increase in the water-oil production ratio over time.

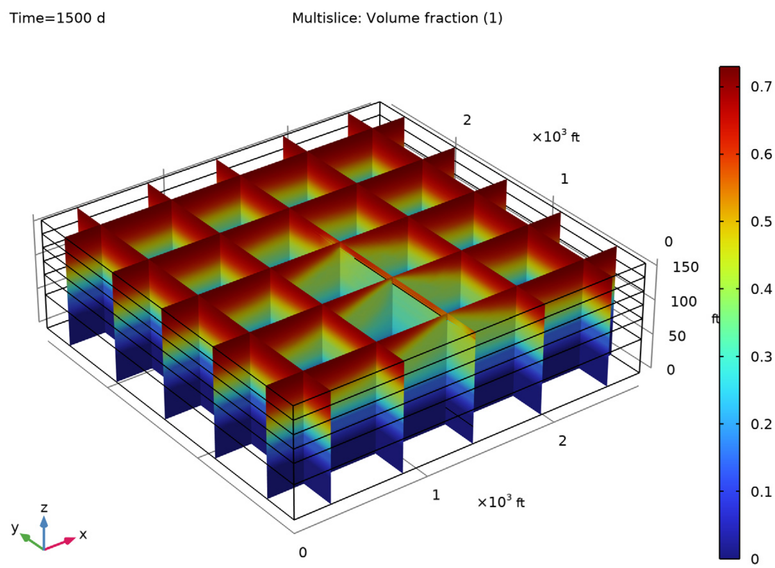


Figure 1: Oil saturation at the end of the 1500 day production period. Note the elevated water table near the production well.

Figure 2 illustrates this and clearly shows these trends in the present simulation. The results for the oil production and the water-oil ratio agree nicely with the simulation results reported in Ref. 1.

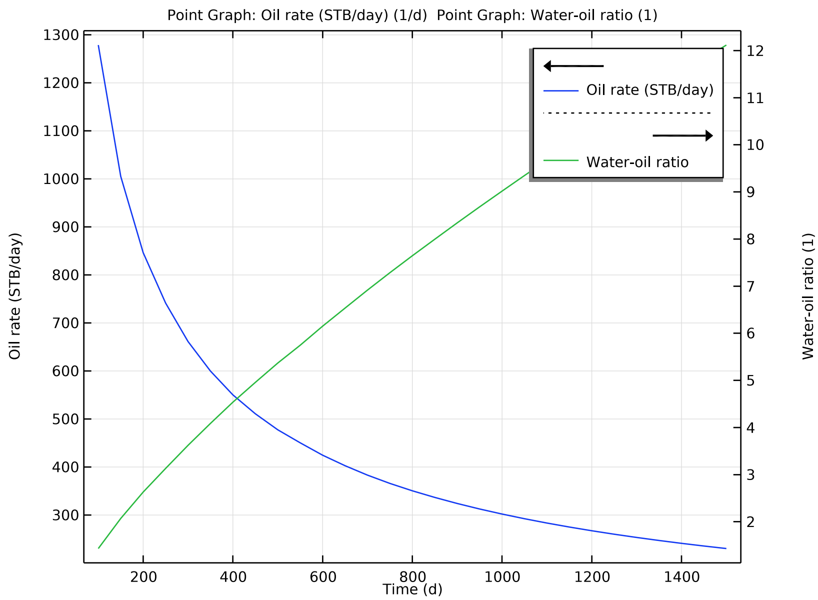


Figure 2: The oil production rate (left y-axis), and the water-oil production ratio (right y-axis) over time.

Notes About the COMSOL Implementation

The prescribed production rate at the production well of 3000 STB/day constitutes a boundary condition for the total mass flow rate, but to solve the model also a boundary condition for the mass flow rate of the oil phase is needed. In the COMSOL model, the oil mass flow rate is determined by requiring that the oil saturation at the production well equals the residual oil saturation (which is equal to 0.2, see Table 3). This condition is implemented using a Well multiphysics edge feature.

Reference

1. L.S. Nghiem, D.A. Collins, and R. Sharma (1991), *Seventh SPE Comparative Solution Project: Modelling of Horizontal Wells in Reservoir Simulation*, SPE 21221, The 11th SPE Symposium on Reservoir Simulation, Anaheim, CA.

Application Library path: `Subsurface_Flow_Module/Fluid_Flow/reservoir_horizontal_wells`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.
2 In the **Select Physics** tree, select **Fluid Flow>Porous Media and Subsurface Flow>Multiphase Flow in Porous Media**.
3 Click **Add**.
4 In the **Added physics interfaces** tree, select **Phase Transport in Porous Media (phtr)**.
5 In the **Volume fractions (1)** table, enter the following settings:

sw
sn

6 Click **Study**.
7 In the **Select Study** tree, select **General Studies>Time Dependent**.
8 Click **Done**.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2 In the **Settings** window for **Geometry**, locate the **Units** section.
3 From the **Length unit** list, choose **ft**.

Block 1 (blk1)

1 In the **Geometry** toolbar, click **Block**.
2 In the **Settings** window for **Block**, locate the **Size and Shape** section.
3 In the **Width** text field, type 2700.
4 In the **Depth** text field, type 2700.

5 In the **Height** text field, type `160`.

6 Click to expand the **Layers** section. Find the **Layer position** subsection. Clear the **Bottom** check box.

7 Select the **Top** check box.

8 In the table, enter the following settings:

Layer name	**Thickness (ft)**
`Layer 1`	20
`Layer 2`	20
`Layer 3`	20
`Layer 4`	20
`Layer 5`	30

9 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **x** text field, type `1350`.

5 In the **z** text field, type `25`.

6 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

7 In the **x** text field, type `1350`.

8 In the **y** text field, type `2700`.

9 In the **z** text field, type `25`.

Line Segment 2 (ls2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **x** text field, type `1350`.

5 In the **y** text field, type `300`.

6 In the **z** text field, type `150`.

7 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

8 In the **x** text field, type `1350`.

9 In the **y** text field, type `1200`.

10 In the **z** text field, type `150`.

Point 1 (pt1)

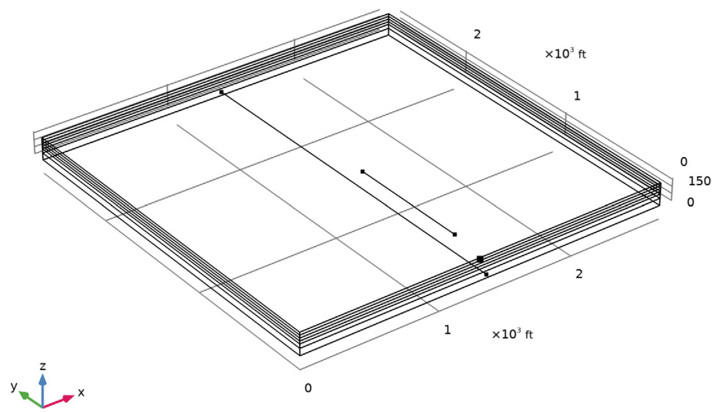
1 In the **Geometry** toolbar, click **More Primitives** and choose **Point**.

2 In the **Settings** window for **Point**, locate the **Point** section.

3 In the **x** text field, type `1300`.

4 In the **z** text field, type `150`.

5 Click **Build Selected**, and compare with the image below.



The point is only used later to create an appropriate mesh. In the subsequent step, designate it as a mesh control vertex. This point will then exclusively be part of the mesh sequence and will remain hidden otherwise.

Mesh Control Vertices 1 (mcv1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Mesh Control Vertices**.

2 On the object **fin**, select Point 15 only.

3 In the **Geometry** toolbar, click **Build All**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `reservoir_horizontal_wells_parameters.txt`.

Interpolation 1 (int1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `reservoir_horizontal_wells_krw.txt`.

5 In the **Function name** text field, type `krw`.

6 Locate the **Interpolation and Extrapolation** section. From the **Interpolation** list, choose **Piecewise cubic**.

Interpolation 2 (int2)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `reservoir_horizontal_wells_krn.txt`.

5 In the **Function name** text field, type `krn`.

6 Locate the **Interpolation and Extrapolation** section. From the **Interpolation** list, choose **Piecewise cubic**.

Interpolation 3 (int3)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `reservoir_horizontal_wells_pc.txt`.

5 In the **Function name** text field, type `pc`.

6 Locate the **Interpolation and Extrapolation** section. From the **Interpolation** list, choose **Piecewise cubic**.

PHASE TRANSPORT IN POROUS MEDIA (PHTR)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Phase Transport in Porous Media (phtr)**.

2 In the **Settings** window for **Phase Transport in Porous Media**, locate the **Gravity Effects** section.

3 Select the **Include gravity** check box.

Phase and Porous Media Transport Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Phase Transport in Porous Media (phtr)** click **Phase and Porous Media Transport Properties 1**.

2 In the **Settings** window for **Phase and Porous Media Transport Properties**, locate the **Capillary Pressure** section.

3 In the p_{csn} text field, type `pc(sw)[psi]`.

4 Locate the **Phase 1 Properties** section. From the ρ_{sw} list, choose **User defined**. In the associated text field, type `rhow`.

5 From the μ_{sw} list, choose **User defined**. In the associated text field, type `muw`.

6 In the κ_{rsw} text field, type `krw(sw)`.

7 Locate the **Phase 2 Properties** section. From the ρ_{sn} list, choose **User defined**. In the associated text field, type `rhoo`.

8 From the μ_{sn} list, choose **User defined**. In the associated text field, type `muo`.

9 In the κ_{rsn} text field, type `krn(sw)`.

Initial Values 2

1 In the **Physics** toolbar, click **Domains** and choose **Initial Values**.

2 Select Domain 6 only.

3 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

4 In the $s_{0,\mathrm{sn}}$ text field, type `0.711`. This is the top layer.

Initial Values 3

1 In the **Physics** toolbar, click **Domains** and choose **Initial Values**.

2 Select Domain 5 only.

3 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

4 In the $s_{0,\mathrm{sn}}$ text field, type `0.652`.

Initial Values 4

1 In the **Physics** toolbar, click **Domains** and choose **Initial Values**.

2 Select Domain 4 only.

3 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

4 In the $s_{0,\mathrm{sn}}$ text field, type `0.527`.

Initial Values 5

1 In the **Physics** toolbar, click **Domains** and choose **Initial Values**.

2 Select Domain 3 only.

3 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

4 In the $s_{0,\mathrm{sn}}$ text field, type `0.351`.

Initial Values 6

1 In the **Physics** toolbar, click **Domains** and choose **Initial Values**.

2 Select Domain 2 only.

3 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

4 In the $s_{0,\mathrm{sn}}$ text field, type `0.131`.

The bottom layer has an initial oil saturation of zero, a condition set by the default **Initial Values 1** node.

DARCY'S LAW (DL)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Darcy's Law (dl)**.

2 In the **Settings** window for **Darcy's Law**, click to expand the **Discretization** section.

3 From the **Pressure** list, choose **Linear**.

Porous Matrix 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Darcy's Law (dl)> Porous Medium 1** click **Porous Matrix 1**.

2 In the **Settings** window for **Porous Matrix**, locate the **Matrix Properties** section.

3 From the ε_p list, choose **User defined**. In the associated text field, type `0.2`.

4 From the κ list, choose **User defined**. From the list, choose **Diagonal**.

5 In the κ table, enter the following settings:

300[mD]	0	0
0	300[mD]	0
0	0	30[mD]

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Darcy's Law (dl)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the p text field, type `3700[psi]`.

MULTIPHYSICS

Well 1 (wellmpe1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Edge>Well**.

2 Select Edge 34 only.

3 In the **Settings** window for **Well**, locate the **Well** section.

4 From the **Specify** list, choose **Pressure**.

5 In the p_0 text field, type `3700[psi]`.

Well 2 (wellmpe2)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Edge>Well**.

2 Select Edge 35 only.

3 In the **Settings** window for **Well**, locate the **Well** section.

4 From the **Well type** list, choose **Production**.

5 In the M_0 text field, type `massflow`.

6 Locate the **Phase 2** section. From the **Specify** list, choose **Volume fraction**.

7 In the $s_{0,\text{sn}}$ text field, type `0.2`.

MESH 1

Free Triangular 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Triangular**.

2 In the **Settings** window for **Free Triangular**, click to expand the **Scale Geometry** section.

3 In the **z-direction scale** text field, type `10`.

4 Select Boundaries 2, 5, 8, 11, 14, and 17 only.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Point**.

4 Select Point 33 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** check box. In the associated text field, type `6`.

8 Select the **Minimum element size** check box. In the associated text field, type `0.1`.

9 Select the **Maximum element growth rate** check box. In the associated text field, type `1.1`.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Finer**.

Swept 1

1 In the **Mesh** toolbar, click **Swept**.

2 In the **Settings** window for **Swept**, click **Build All**.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 From the **Time unit** list, choose **d**.

4 In the **Output times** text field, type `range(0,50,1500)`.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Dependent Variables 1**.

3 In the **Settings** window for **Dependent Variables**, locate the **Scaling** section.

4 From the **Method** list, choose **Initial value based**.

5 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)** click **Time-Dependent Solver 1**.

6 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

7 From the **Steps taken by solver** list, choose **Strict**.

8 Click **Compute**.

RESULTS

Slice

1 In the **Model Builder** window, expand the **Volume Fraction (phtr)** node.

2 Right-click **Slice** and choose **Disable**.

Volume Fraction (phtr)

In the **Model Builder** window, click **Volume Fraction (phtr)**.

Multislice 1

1 In the **Volume Fraction (phtr)** toolbar, click **More Plots** and choose **Multislice**.

2 In the **Settings** window for **Multislice**, locate the **Expression** section.

3 In the **Expression** text field, type `sn`.

4 Locate the **Multiplane Data** section. Find the **x-planes** subsection. In the **Planes** text field, type `5`.

5 Find the **y-planes** subsection. In the **Planes** text field, type `5`.

6 Find the **z-planes** subsection. In the **Planes** text field, type `0`.

7 In the **Volume Fraction (phtr)** toolbar, click **Plot**.

For a more comprehensive view of the results, employ a scaled view by following these steps.

8 Click the **Show More Options** button in the **Model Builder** toolbar.

9 In the **Show More Options** dialog box, select **Results>Views** in the tree.

10 In the tree, select the check box for the node **Results>Views**.

11 Click **OK**.

View 3D 2

In the **Model Builder** window, under **Results** right-click **Views** and choose **View 3D**.

Camera

1 In the **Model Builder** window, expand the **View 3D 2** node, then click **Camera**.

2 In the **Settings** window for **Camera**, locate the **Camera** section.

3 From the **View scale** list, choose **Manual**.

4 In the **z scale** text field, type `5`.

Volume Fraction (phtr)

1 In the **Model Builder** window, under **Results** click **Volume Fraction (phtr)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 From the **View** list, choose **View 3D 2**.

4 In the **Volume Fraction (phtr)** toolbar, click **Plot**.

5 Click the **Zoom Extents** button in the **Graphics** toolbar.

1D Plot Group 5

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Time selection** list, choose **Interpolated**.

4 In the **Times (d)** text field, type `range(100,50,1500)`.

Point Graph 1

1 Right-click **1D Plot Group 5** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `17` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

7 In the **Expression** text field, type `wellmpe2.MO_sn/rhoo/STB`.

8 In the **Unit** field, type `1/d`.

9 Select the **Description** check box. In the associated text field, type `Oil rate (STB/day)`.

10 Click to expand the **Legends** section. Select the **Show legends** check box.

11 From the **Legends** list, choose **Manual**.

12 In the table, enter the following settings:

Legends
`Oil rate (STB/day)`

Point Graph 2

1 In the **Model Builder** window, right-click **1D Plot Group 5** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `17` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

7 In the **Expression** text field, type `(massflow-wellmpe2.MO_sn)/rhow/(wellmpe2.MO_sn/rhoo)`.

8 Select the **Description** check box. In the associated text field, type `Water-oil ratio`.

9 Locate the **Legends** section. Select the **Show legends** check box.

10 From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
Water-oil ratio

Oil Rate and Water-Oil Ratio

1 In the **Model Builder** window, under **Results** click **1D Plot Group 5**.

2 In the **Settings** window for **1D Plot Group**, type `Oil Rate and Water-Oil Ratio` in the **Label** text field.

3 Locate the **Plot Settings** section. Select the **Two y-axes** check box.

4 In the table, select the **Plot on secondary y-axis** check box for **Point Graph 2**.

5 Select the **y-axis label** check box. In the associated text field, type `Oil rate (STB/day)`.

6 In the **Oil Rate and Water-Oil Ratio** toolbar, click **Plot**. Compare with Figure 2.