Created in COMSOL Multiphysics 5.6

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. The reservoir contains two phases, water and oil. The oil is recovered by injecting water through the 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
$0.96 \cdot 10^{-3}$ 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 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.

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 an Edge ODEs and DAEs interface.

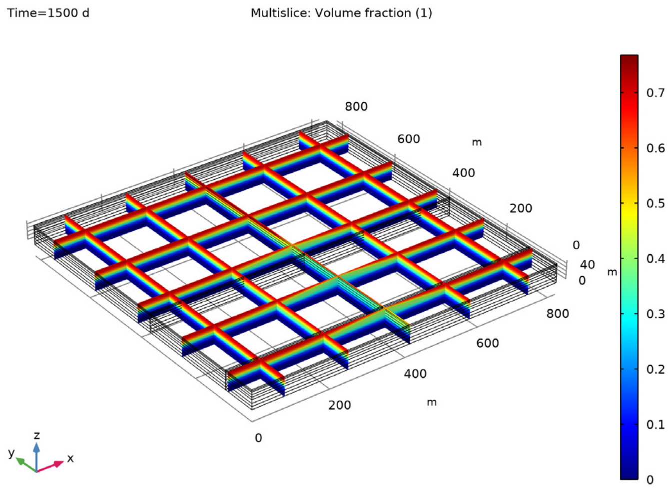

Figure 1: Oil saturation at the end of the 1500 day production period. Note the elevated water table near the production well.

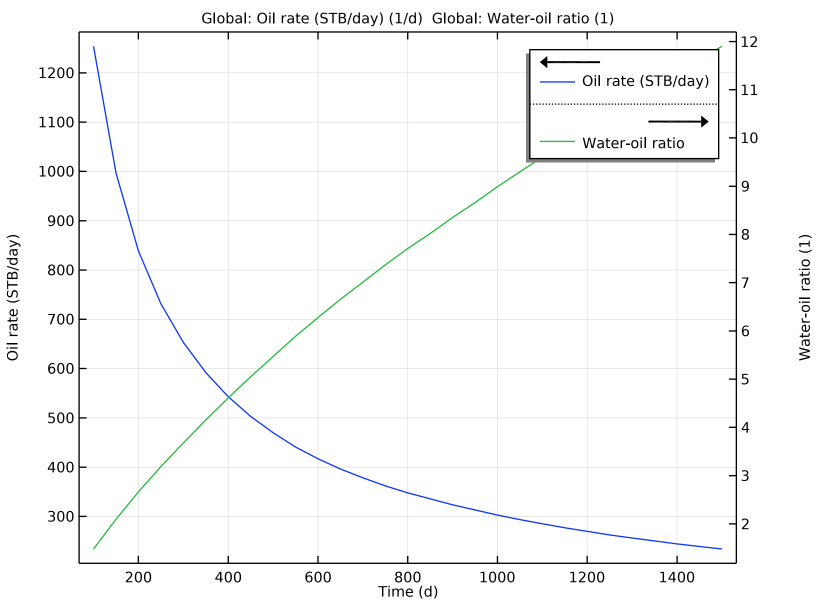

Figure 2: The oil production rate (left y-axis), and the water-oil production ratio (right y-axis) over time.

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 **Select Physics** tree, select **Mathematics>ODE and DAE Interfaces>Edge ODEs and DAEs (eode)**.

5 Click **Add**.

6 In the **Added physics interfaces** tree, select **Phase Transport in Porous Media (phtr)**.

7 In the **Volume fractions** table, enter the following settings:

sw
sn

8 In the **Added physics interfaces** tree, select **Darcy's Law (dl)**.

9 In the **Pressure** text field, type pw.

10 In the **Added physics interfaces** tree, select **Edge ODEs and DAEs (eode)**.

11 In the **Field name** text field, type `massflowoil`.

12 In the **Dependent variables** table, enter the following settings:

massflowoil

13 Click **Study**.

14 In the **Select Study** tree, select **General Studies>Time Dependent**.

15 Click **Done**.

GEOMETRY 1

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[ft]`.

4 In the **Depth** text field, type `2700[ft]`.

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

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 (m)
Layer 1	10[ft]
Layer 2	10[ft]
Layer 3	20[ft]
Layer 4	20[ft]
Layer 5	20[ft]
Layer 6	30[ft]
Layer 7	25[ft]

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.

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

3 From the **Plane** list, choose **xz-plane**.

4 In the **y-coordinate** text field, type `300[ft]`.

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1)>Cross Section 1 (cro1)

In the **Work Plane** toolbar, click **Cross Section**.

Work Plane 2 (wp2)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Work Plane**.
2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.
3 From the **Plane** list, choose **zy-plane**.
4 In the **x-coordinate** text field, type `1350[ft]`.

Work Plane 2 (wp2)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 2 (wp2)>Cross Section 1 (cro1)

In the **Work Plane** toolbar, click **Cross Section**.

Work Plane 3 (wp3)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Work Plane**.
2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.
3 From the **Plane** list, choose **xz-plane**.
4 In the **y-coordinate** text field, type `1200[ft]`.

Work Plane 3 (wp3)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 3 (wp3)>Cross Section 1 (cro1)

1 In the **Work Plane** toolbar, click **Cross Section**.
2 Right-click **Cross Section 1 (cro1)** and choose **Build All Objects**.

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**.

DEFINITIONS

Integration 1 (intop1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Definitions** and choose **Nonlocal Couplings>Integration**.

3 In the **Settings** window for **Integration**, locate the **Source Selection** section.

4 From the **Geometric entity level** list, choose **Edge**.

5 Select Edge 144 only.

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 Domains 7, 8, 15, 16, 23, 24, 31, 32, 39, 40, 47, and 48 only.

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

4 In the $s_{0,sn}$ text field, type `0.711`.

Initial Values 3

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

2 Select Domains 6, 14, 22, 30, 38, and 46 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 Domains 5, 13, 21, 29, 37, and 45 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 Domains 4, 12, 20, 28, 36, and 44 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 Domains 3, 11, 19, 27, 35, and 43 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`.

Line Mass Source 1

1 In the **Physics** toolbar, click **Edges** and choose **Line Mass Source**.

2 Select Edge 144 only.

3 In the **Settings** window for **Line Mass Source**, locate the **Mass Flux** section.

4 Select the **Phase sn** check box.

5 In the $q_{0,\mathrm{sn}}$ text field, type `-massflowoil`.

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**.

Fluid and Matrix Properties 1

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

2 In the **Settings** window for **Fluid and Matrix Properties**, 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, click **Initial Values 1**.

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

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

Well 1

1 In the **Physics** toolbar, click **Edges** and choose **Well**.

2 Select Edges 100, 126, and 152 only.

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

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

Line Mass Source 1

1 In the **Physics** toolbar, click **Edges** and choose **Line Mass Source**.

2 Select Edge 144 only.

3 In the **Settings** window for **Line Mass Source**, locate the **Line Mass Source** section.

4 In the N_0 text field, type -massflow.

EDGE ODES AND DAES (EODE)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Edge ODEs and DAEs (eode)**.

2 In the **Settings** window for **Edge ODEs and DAEs**, locate the **Edge Selection** section.

3 Click **Clear Selection**.

4 Select Edge 144 only.

5 Locate the **Units** section. Click **Define Dependent Variable Unit**.

6 In the **Dependent variable quantity** table, enter the following settings:

Dependent variable quantity	Unit
Custom unit	kg/m/s

7 In the **Source term quantity** table, enter the following settings:

Source term quantity	Unit
Custom unit	1

8 Click to expand the **Discretization** section. From the **Shape function type** list, choose **Lagrange**.

9 From the **Element order** list, choose **Linear**.

Distributed ODE 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Edge ODEs and DAEs (eode)** click **Distributed ODE 1**.

2 In the **Settings** window for **Distributed ODE**, locate the **Source Term** section.

3 In the *f* text field, type `sn-0.2`.

4 Locate the **Damping or Mass Coefficient** section. In the d_a text field, type `0`.

MESH 1

Free Triangular 1

1 In the **Mesh** toolbar, click **Boundary** 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, 17, 20, 23, 85, 88, 91, 94, 97, 100, 103, and 106 only.

Size

1 In the **Model Builder** window, click **Size**.

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

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

Size 1

1 In the **Model Builder** window, 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 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `44` in the **Selection** text field.

6 Click **OK**.

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

8 Click the **Custom** button.

9 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

10 In the associated text field, type `2`.

11 Select the **Minimum element size** check box.

12 In the associated text field, type `0.04`.

13 Select the **Maximum element growth rate** check box.

14 In the associated text field, type `1.1`.

Swept 1

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

Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `1-8, 25-32` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Distribution**, locate the **Distribution** section.

8 In the **Number of elements** text field, type `2`.

Distribution 2

1 In the **Model Builder** window, right-click **Swept 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `9-24, 33-48` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Distribution**, locate the **Distribution** section.

8 In the **Number of elements** text field, type `8`.

9 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, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** node.
6 Right-click **Time-Dependent Solver 1** and choose **Fully Coupled**.
7 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.
8 From the **Nonlinear method** list, choose **Automatic (Newton)**.
9 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**.

1D Plot Group 6

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)`.

Global 1

1 Right-click **1D Plot Group 6** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`intop1(massflowoil/rhoo/STB)`	`1/d`	`Oil rate (STB/day)`

Global 2

1 In the **Model Builder** window, right-click **1D Plot Group 6** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`(intop1(massflow)-intop1(massflowoil))/ rhow/(intop1(massflowoil/rhoo))`	`1`	`Water-oil ratio`

1D Plot Group 6

1 In the **Model Builder** window, click **1D Plot Group 6**.

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

3 Select the **Two y-axes** check box.

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

5 Select the **y-axis label** check box.

6 In the associated text field, type `Oil rate (STB/day)`.

7 In the **1D Plot Group 6** toolbar, click **Plot**.