Created in COMSOL Multiphysics 6.2

Two-Phase Flow over a Low Permeable Lens

Introduction

This example concerns two-phase flow in a porous medium that contains a low permeable lens. The heavier phase infiltrates the porous medium from above, and the low permeable lens is infiltrated only when a critical saturation at the outside of the lens is reached. As the saturation of the heavier phase is discontinuous at the boundary of the lens, this requires the use of the Porous Medium Discontinuity boundary condition.

Model Definition

The porous domain is assumed to be axially symmetric, with a radius of 0.5 m and a height of 0.65 m. The low permeable lens has radius of 0.32 m and a height of 0.12 m. The bottom boundary of the lens is located at a height of 0.35 m. Initially the porous domain, including the lens, is occupied with phase 1. Phase 2 flows into the porous medium at the top boundary through a circle with radius 0.07 m with a uniform and constant mass flux. See Figure 1 below for a graphic representation of the geometry.

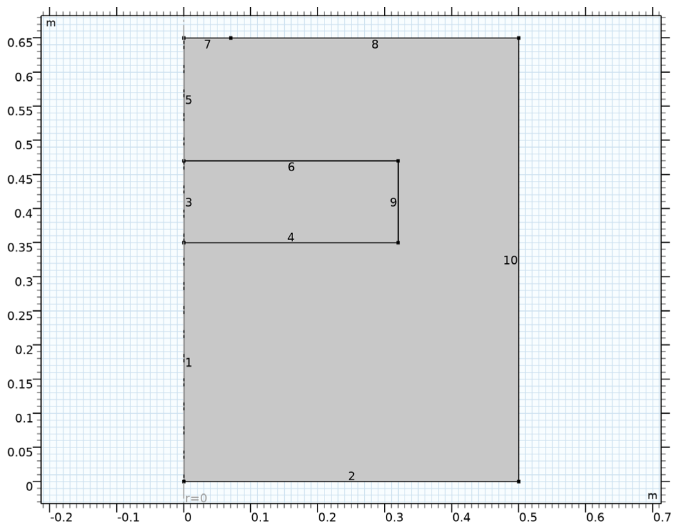

Figure 1: Cross section of the axially symmetric geometry.

Initially the porous domain, including the lens, is occupied with phase 1. Phase 2 flows into the domain with a constant mass flux. The properties of the two phases are given in Table 1.

TABLE 1: FLUID PROPERTIES.

QUANTITY	VALUE	DESCRIPTION
ρ_1	1000 kg/m^3	Density of phase 1
ρ_2	1460 kg/m^3	Density of phase 2
μ_1	10^{-3} Pa·s	Dynamics viscosity of phase 1
μ_2	$0.9 \cdot 10^{-3}$ Pa·s	Dynamics viscosity of phase 2

The properties of the solid matrix and the parameters for the constitutive relations for the relative permeabilities and capillary pressure curves, which are described by the Brooks and Corey model, are given in Table 2.

TABLE 2: SOLID MATRIX PROPERTIES AND BROOKS & COREY PARAMETERS.

QUANTITY	VALUE IN LENS	VALUE	DESCRIPTION
ε_p	0.39	0.4	Porosity
κ	$3.32 \cdot 10^{-11}$ m^2	$6.64 \cdot 10^{-11}$ m^2	Permeability
s_{r1}	0.12	0.1	Residual saturation of phase 1
s_{r2}	0	0	Residual saturation of phase 2
λ_p	2	2.7	Pore size distribution index
p_{ec}	1163.5 Pa	775 Pa	Entry capillary pressure

The initial values for the saturation of phase 1 and the pressure of phase 2 are given in Table 3.

TABLE 3: INITIAL VALUES.

QUANTITY	VALUE
s_2	0
p	`(0.65-z)*g_const*1000[kg/m^3]`

The boundary conditions are given in Table 4. In this table $q_{0,si}$ denotes the normal mass flux of phase i. The number of the boundaries refer to the numbers indicated in Figure 1. The time interval for the simulation is 100 minutes.

TABLE 4: BOUNDARY CONDITIONS.

BOUNDARY	CONDITION
1,3,5	axial symmetry
2	$s_2 = 0$, $q_{0,s1} = 0$

TABLE 4: BOUNDARY CONDITIONS.

BOUNDARY	CONDITION
7	$q_{0,s1}$=0, $q_{0,s2}$=0.25 kg/(m^2·s)
8	$q_{0,s1}$=0, $q_{0,s2}$=0
10	s_2 = 0, p = `(0.65-z)*g_const*1000[kg/m^3]`

Results and Discussion

Due to gravity, the heavier phase 2 infiltrates the porous domain and flows down over the low permeable lens. Since the entry capillary pressure of the lens is higher than the entry capillary pressure of the surrounding material, phase 2 will not enter the lens directly when it reaches the lens. Phase 2 will only enter the lens when a critical saturation is reached. This condition, which applies at boundaries where the porous medium properties, and especially the capillary pressure curves, are discontinuous, is implemented in the model using a Porous Medium Discontinuity boundary condition. This condition allows for a discontinuity in the saturation of phase 2 and determines the critical saturation at which phase 2 enters the low permeable domain. Figure 2 and Figure 3 below show that this happens after around 12 minutes. After approximately 60 minutes, phase 2 has reached the bottom of the lens.

Figure 4 shows the how the isosurface of volume fraction of phase 2 being 20% evolves with time.

This simulation is inspired by a very similar model as discussed in Ref. 1and Ref. 2.

Notes About the COMSOL Implementation

In the present implementation of the model, the dependent variables are the saturation of phase 2, s_2, and the pressure of phase 1, p. The equation for the saturation takes as boundary flux the mass flux of phase 2, and the equation for the pressure takes as boundary flux the total mass flux (mass fluxes of phase 1 and 2 added together). The boundary condition at the bottom boundary prescribes the saturation of phase 2 and the mass flux of phase 1. To be able to prescribe the total mass flux in the equation for p, the mass flux of phase 2 is also needed. This mass flux is computed automatically if the saturation condition for phase 2 is implemented as a weak constraint, see the instructions in the Modeling Instructions section.

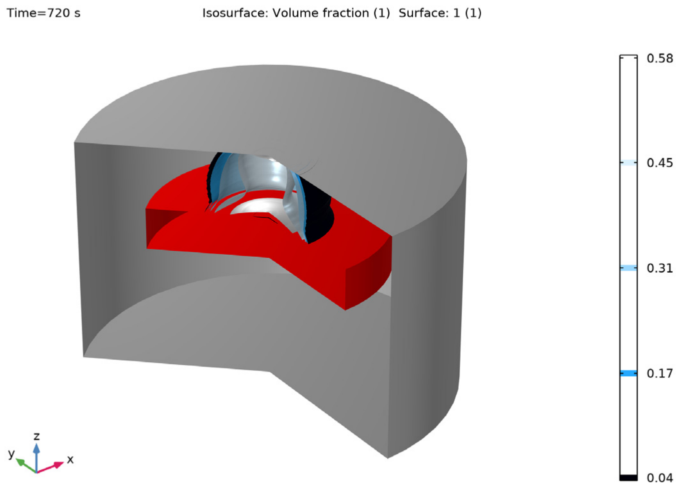

Figure 2: Isosurfaces of the penetrating phase 2 after 12 minutes. Phase 2 just starts entering the low permeable lens at this instant in time.

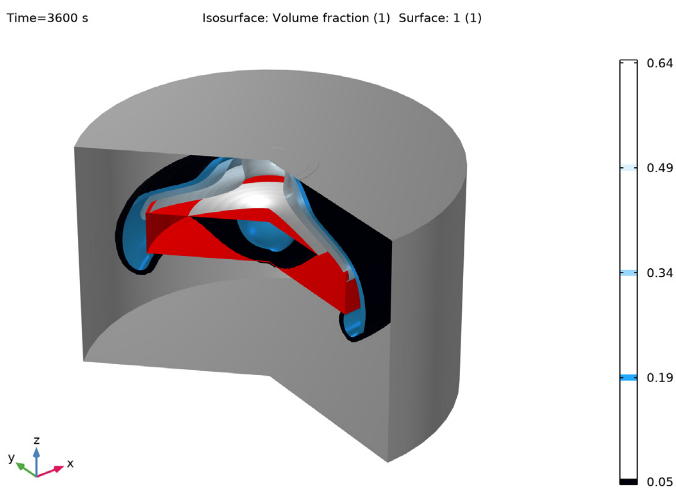

Figure 3: Isosurfaces of the penetrating phase 2 after 60 minutes. Phase 2 has now reached the bottom of the low permeable lens.

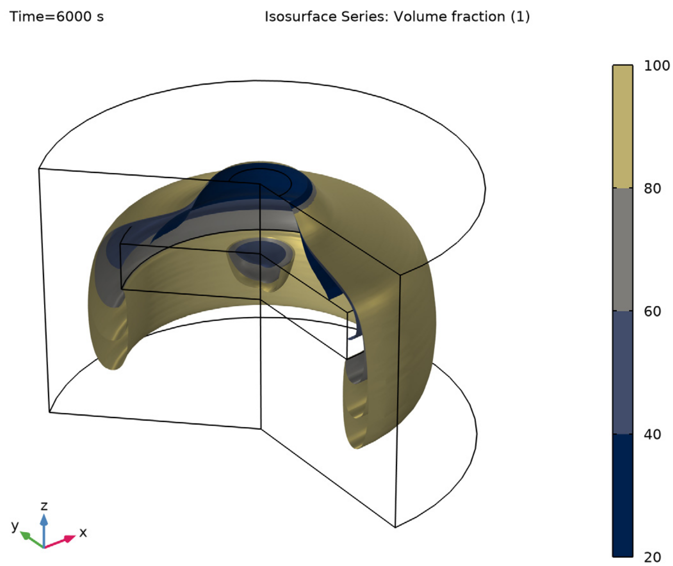

Figure 4: Evolution of isosurface of volume fraction of phase 2, s2= 0.2, with time (plotted every 20 minutes).

References

1. R. Helmig, *Multiphase Flow and Transport Processes in the Subsurface – A Contribution to the Modeling of Hydrosystems*, Springer Verlag, 1997.

2. P. Bastian, *Numerical Computation of Multiphase Flows in Porous Media*, Habilitationsschrift Universität Kiel, 1999.

Application Library path: `CFD_Module/Multiphase_Flow/low_permeable_lens`

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 **2D Axisymmetric**.
2 In the **Select Physics** tree, select **Fluid Flow>Porous Media and Subsurface Flow>Multiphase Flow in Porous Media**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Time Dependent**.
6 Click **Done**.

GEOMETRY 1

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `0.5`.
4 In the **Height** text field, type `0.65`.

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `0.32`.
4 In the **Height** text field, type `0.12`.
5 Locate the **Position** section. In the **z** text field, type `0.35`.

Point 1 (pt1)

1 In the **Geometry** toolbar, click **Point**.
2 In the **Settings** window for **Point**, locate the **Point** section.
3 In the **r** text field, type `0.07`.
4 In the **z** text field, type `0.65`.
5 Click **Build All Objects**.

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 From the **Capillary pressure model** list, choose **Brooks and Corey**.

4 In the p_{ec} text field, type `1163.5`.

5 Locate the **Phase 1 Properties** section. From the ρ_{s1} list, choose **User defined**. From the μ_{s1} list, choose **User defined**. In the s_{rs1} text field, type `0.12`.

6 Locate the **Phase 2 Properties** section. From the ρ_{s2} list, choose **User defined**. In the associated text field, type `1460[kg/m^3]`.

7 From the μ_{s2} list, choose **User defined**. In the associated text field, type `0.0009[Pa*s]`.

Phase and Porous Media Transport Properties 2

1 In the **Physics** toolbar, click **Domains** and choose **Phase and Porous Media Transport Properties**.

2 Select Domain 1 only.

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

4 From the **Capillary pressure model** list, choose **Brooks and Corey**.

5 In the p_{ec} text field, type `755`.

6 In the λ_p text field, type `2.7`.

7 Locate the **Phase 1 Properties** section. From the ρ_{s1} list, choose **User defined**. From the μ_{s1} list, choose **User defined**. In the s_{rs1} text field, type `0.1`.

8 Locate the **Phase 2 Properties** section. From the ρ_{s2} list, choose **User defined**. In the associated text field, type `1460[kg/m^3]`.

9 From the μ_{s2} list, choose **User defined**. In the associated text field, type `0.0009[Pa*s]`.

Mass Flux 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Mass Flux**.

2 Select Boundary 7 only.

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

4 Select the **Phase s2** check box.

5 In the $q_{0,\mathrm{s2}}$ text field, type `0.25`.

Volume Fraction 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Volume Fraction**.

2 Select Boundaries 2 and 10 only.

3 In the **Settings** window for **Volume Fraction**, locate the **Volume Fraction** section.

4 Select the **Phase s2** check box.

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

6 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.

7 Click **OK**.

8 In the **Settings** window for **Volume Fraction**, click to expand the **Constraint Settings** section.

9 From the **Constraint** list, choose **Weak constraints**.

Porous Medium Discontinuity 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Porous Medium Discontinuity**.

2 Select Boundaries 4, 6, and 9 only.

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**, locate the **Gravity Effects** section.

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

Gravity 1

In the Gravity node specify a reference position to ensure that the pressure is zero at the upper boundary of the model.

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

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

3 Select the **Specify reference position** check box.

4 Specify the $\mathbf{r}_{\mathrm{ref}}$ vector as

r	r
0.65	z

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 `0.39`.

4 From the κ list, choose **User defined**. In the associated text field, type `3.32e-11[m^2]`.

Porous Medium 2

1 In the **Physics** toolbar, click **Domains** and choose **Porous Medium**.

2 Select Domain 1 only.

Porous Matrix 1

1 In the **Model Builder** window, 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 `0.4`.

4 From the κ list, choose **User defined**. In the associated text field, type `6.64e-11[m^2]`.

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 Click the **Hydraulic head** button. This way the initial pressure field is forced to equal the hydraulic pressure.

Inlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 7 only.

3 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.

4 From the **Boundary condition** list, choose **Mass flow**.

5 Locate the **Mass Flow** section. From the **Mass flow type** list, choose **Pointwise mass flux**.

6 In the N_0 text field, type `0.25`.

Pressure 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Pressure**.

2 Select Boundary 10 only.

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

4 In the p_0 text field, type `(0.65-z)*g_const*1000[kg/m^3]` to compensate for hydrostatic pressure.

Mass Flux 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Mass Flux**.

2 Select Boundary 2 only.

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

4 In the N_0 text field, type `s2_lm`.

MESH 1

Free Triangular 1

In the **Mesh** toolbar, click **Free Triangular**.

Size

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

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.01`.

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 In the **Output times** text field, type `range(0,60,6000)`.

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**. This setting ensures that the variable scaling is based on the supplied initial hydrostatic pressure profile, which in this case gives a better scaling than the automatic setting. Scaling is important to obtain well weighted error estimates and avoid ill-conditioned matrices which may hamper or slow down the solution procedure.

5 In the **Study** toolbar, click **Compute**.

RESULTS

Follow the instructions below to obtain the plots as shown in the Results and Discussion section above.

Volume Fraction of Phase 2

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 From the **Time (s)** list, choose **3600**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

5 In the **Label** text field, type `Volume Fraction of Phase 2`.

6 Click the **Show Grid** button in the **Graphics** toolbar.

Isosurface 1

1 Right-click **Volume Fraction of Phase 2** and choose **Isosurface**.

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

3 In the **Expression** text field, type `s2`.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Aurora>JupiterAuroraBorealis** in the tree.

6 Click **OK**.

Surface 1

1 In the **Model Builder** window, right-click **Volume Fraction of Phase 2** and choose **Surface**.

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

3 In the **Expression** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Gray**.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

2 Select Domain 2 only.

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

4 Clear the **Evaluate the start cap** check box.

5 Clear the **Evaluate the end cap** check box.

6 In the **Volume Fraction of Phase 2** toolbar, click **Plot**.

7 Locate the **Selection** section. Click to select the **Activate Selection** toggle button.

8 Select Domain 1 only.

9 In the **Volume Fraction of Phase 2** toolbar, click **Plot**.

Surface 2

1 In the **Model Builder** window, right-click **Volume Fraction of Phase 2** and choose **Surface**.

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

3 In the **Expression** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Selection 1

1 Right-click **Surface 2** and choose **Selection**.

2 Select Domain 2 only.

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

4 Clear the **Evaluate the start cap** check box.

5 Clear the **Evaluate the end cap** check box.

6 In the **Volume Fraction of Phase 2** toolbar, click **Plot**.

To plot the volume fraction of phase 2 for different times in one plot, follow the steps below.

Volume fraction, isosurface series

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Volume fraction, isosurface series` in the **Label** text field.

Isosurface Series 1

1 In the **Volume fraction, isosurface series** toolbar, click **More Plots** and choose **Isosurface Series**.

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

3 In the **Expression** text field, type `s2`.

4 Locate the **Levels** section. In the **Level** text field, type `0.2`.

5 Locate the **Data** section. From the **Time selection** list, choose **Manual**.

6 Click **Range**.

7 In the **Integer Range** dialog box, type `10` in the **Step** text field.

8 In the **Stop** text field, type `101`.

9 In the **Step** text field, type `20`.

10 Click **Replace**.

Color Expression 1

1 Right-click **Isosurface Series 1** and choose **Color Expression**.

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

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

4 From the **Unit** list, choose **min**.

5 Locate the **Coloring and Style** section. From the **Color table type** list, choose **Discrete**.

6 Click **Change Color Table**.

7 In the **Color Table** dialog box, select **Linear>Cividis** in the tree.

8 Click **OK**.

9 In the **Settings** window for **Color Expression**, locate the **Coloring and Style** section.

10 In the **Number of bands** text field, type `4`.

11 In the **Volume fraction, isosurface series** toolbar, click **Plot**.