Created in COMSOL Multiphysics 5.4

Two-Phase Flow over a Low Permeable Lens

Introduction

This example concerns two-phase flow in a porous medium which 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.

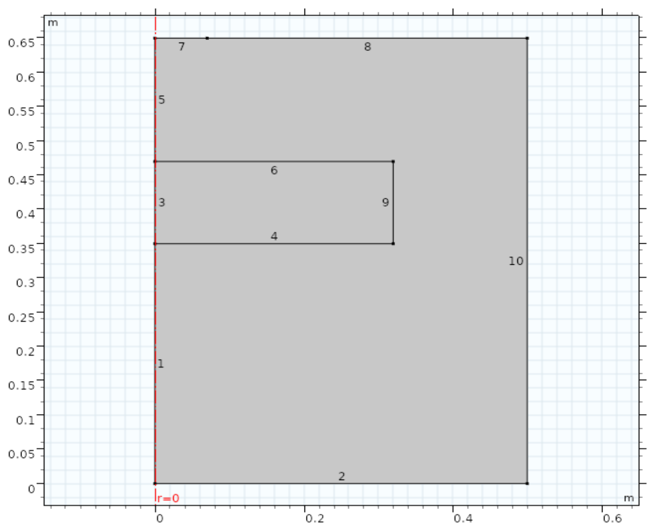

Figure 1: Cross section of the axially symmetric geometry.

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. The figures below show that this happens after around 12 minutes. After approximately 60 minutes, phase 2 has reached the bottom of the lens.

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.

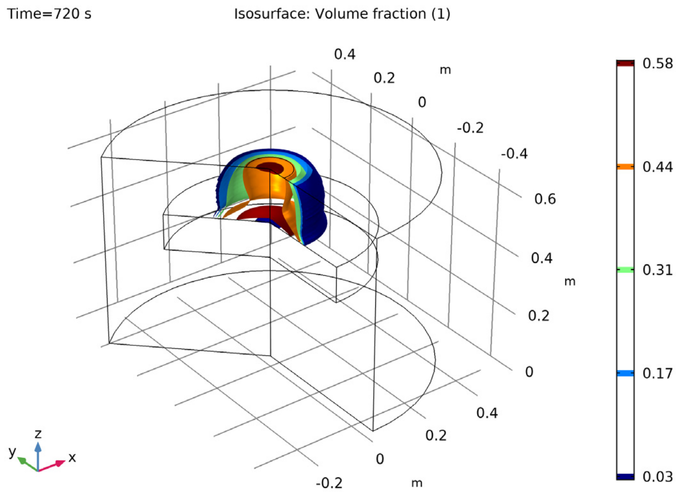

Figure 2: Isosurfaces of the penetrating phase 2 after 12 minutes. Phase 2 just starts entering the low permeable lens at this instant in time.

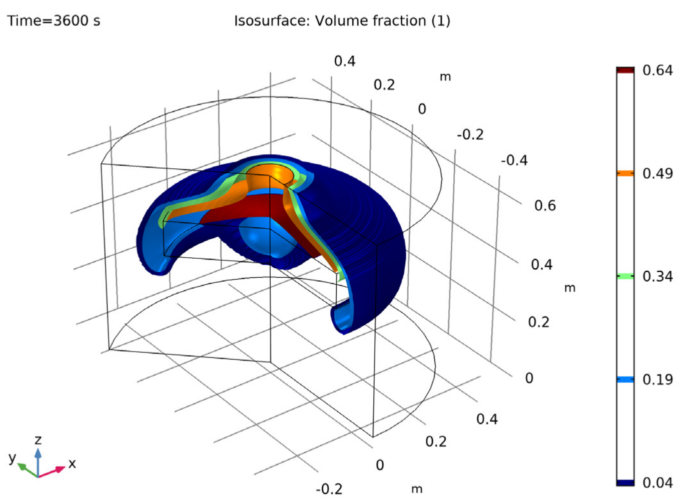

Figure 3: Isosurfaces of the penetrating phase 2 after 60minutes. Phase 2 has now reached the bottom of the low permeable lens.

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 Univeristät Kiel, 1999.

Application Library path: `Subsurface_Flow_Module/Fluid_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 **Primitives** and choose **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 **Primitives** and choose **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 **Primitives** and choose **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 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

6 Click to expand the **Constraint Settings** section. Select the **Use weak constraints** check box.

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)

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 `0.39`.

4 From the κ list, choose **User defined**. In the associated text field, type `3.32e-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 In the p text field, type `(0.65-z)*g_const*1000[kg/m^3]`.

4 In the **Model Builder** window, click **Darcy's Law (dl)**.

Fluid and Matrix Properties 2

1 In the **Physics** toolbar, click **Domains** and choose **Fluid and Matrix Properties**.

2 Select Domain 1 only.

3 In the **Settings** window for **Fluid and Matrix Properties**, locate the **Matrix Properties** section.

4 From the ε_p list, choose **User defined**. In the associated text field, type `0.4`.

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

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 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]`.

Mass Flux 2

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

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

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 **Times** text field, type `range(0,60,6000)`.

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

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

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

3 Find the **Algebraic variable settings** subsection. From the **Error estimation** list, choose **Exclude algebraic**.

4 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Time-Dependent Solver 1** node, then click **Fully Coupled 1**.

5 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

6 From the **Jacobian update** list, choose **On every iteration**.

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

RESULTS

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

Isosurface 1

1 In the **Model Builder** window, under **Results** right-click **3D Plot Group 5** and choose **Isosurface**.

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

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

4 In the **3D Plot Group 5** toolbar, click **Plot**.