Model created in COMSOL Multiphysics 6.4

Two-Phase Flow in a Porous Medium: Buckley–Leverett Model

Introduction

This example uses the Multiphase Flow in Porous Media multiphysics interface to model an immiscible displacement process in a porous medium. You can think of the displacement of oil by water in a reservoir. In a one-dimensional setting and under certain assumptions, the equations for the saturations and pressures of the two phases reduce to a single conservation equation for the saturation of one of the phases, the Buckley–Leverett equation. In the model setup described below this equation allows for an analytical solution, and as such, this example also serves as a benchmark model.

Model Definition

A 1D porous medium with a length of 1 meter is considered. It is assumed that both phases present are incompressible, that there are no sources or sinks, and that gravity plays no role. Furthermore, zero capillary pressure and (relative) permeabilities and porosity independent of time and space are assumed.

Initially the porous medium is completely filled with phase 1. At $x = 0$ phase 2 enters the porous medium with a volumetric flux of 0.001 m/s and displaces phase 1, which is allowed to flow out of the porous domain at $x = 1$. The volumetric flux of phase 1 is equal to 0 at $x = 0$. The pressure at $x = 1$ is set to be equal to 0 Pa. Table 1 collects the relevant material properties. The process in simulated for a time interval of 300 seconds.

From the assumptions and boundary conditions mentioned above it follows that the total volumetric flux $\mathbf{u} = \mathbf{u}_1 + \mathbf{u}_2$ is constant in time and space, and that the two-phase flow equations implemented in the Multiphase Flow in Porous Media interface for the saturations and pressures reduce to a single equation for the saturation s_1:

$$\frac{\partial}{\partial t}(\varepsilon_{\mathrm{p}} s_1) + \frac{\partial}{\partial x}\left(\frac{\lambda_1}{\lambda_1 + \lambda_2}\mathbf{u}\right) = 0 \qquad (1)$$

where ε_{p} denotes the porosity and $\lambda_i = \kappa_{ri}/\mu_i$, with κ_{ri} and μ_i the relative permeability and dynamic viscosity of phase i, respectively. This equation allows for an analytical solution (see Ref. 1 for the construction of the analytical solution to this Buckley–Leverett equation). In the Results and Discussion section below, the analytical solution of the Buckley–Leverett equation is compared to the solution obtained with the Multiphase Flow in Porous Media interface.

TABLE 1: MODEL DATA.

QUANTITY	VALUE	DESCRIPTION
$\rho_1 = \rho_2$	1000 kg/m^3	Density of both phases
$\mu_1 = \mu_2$	0.001 kg/(m·s)	Dynamic viscosity of both phases
ε_p	0.5	Porosity
κ	10^{-9} m^2	Permeability
κ_{r1}	$s_1{}^2$	Relative permeability of phase 1
κ_{r2}	$s_2{}^2$	Relative permeability of phase 2

Results and Discussion

In Figure 1, the profiles of the saturation s_2 are shown for different instances in time (20 seconds intervals). Phase 1 is displaced by phase 2, and the solution exhibits a shock traveling through the porous domain. The solution of the two-phase flow equations (solid lines) shows a good agreement with the analytical solution of the Buckley–Leverett equation in Equation 1(dotted lines).

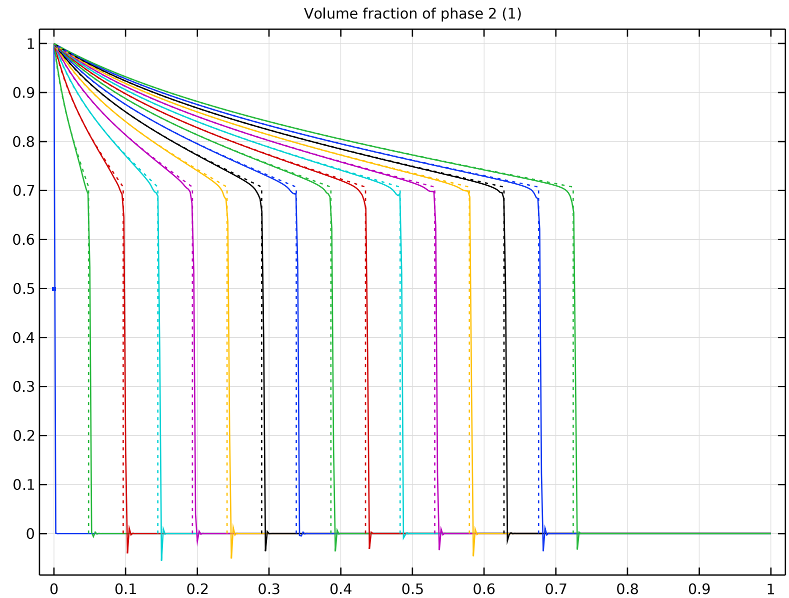

Figure 1: Saturation profiles for phase 2 shown at intervals of 20 seconds, as computed using the Multiphase Flow in Porous Media interface (solid lines), and as obtained analytically as solution of the Buckley–Leverett equation.

Reference

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

Application Library path: `Porous_Media_Flow_Module/Verification_Examples/buckley_leverett_model`

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 **1D**.
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

Interval 1 (i1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Geometry 1** and choose **Interval**.
2 In the **Settings** window for **Interval**, click **Build All Objects**.

DEFINITIONS

Add a piecewise analytic function to visualize the analytic solution of the Buckley–Leverett equation and to compare it to the solution computed using the Multiphase Flow in Porous Media interface.

Piecewise 1 (pw1)

1 In the **Definitions** toolbar, click **Piecewise**.
2 In the **Settings** window for **Piecewise**, locate the **Definition** section.
3 Find the **Intervals** subsection. In the table, enter the following settings:

Start	End	Function
0.7071	1	d(x^2/(x^2+(1-x)^2),x)

PHASE TRANSPORT IN POROUS MEDIA (PHTR)

Fluid 1

1 In the **Model Builder** window, under **Component 1 (comp1) > Phase Transport in Porous Media (phtr) > Porous Medium 1** click **Fluid 1**.
2 In the **Settings** window for **Fluid**, locate the **Phase 1 Properties** section.
3 From the ρ_{s1} list, choose **User defined**. From the μ_{s1} list, choose **User defined**. Locate the **Phase 2 Properties** section. From the ρ_{s2} list, choose **User defined**. From the μ_{s2} list,

choose **User defined**. This sets the density and dynamic viscosity of both phases to the default values $\rho = 1000 \text{ kg/m}^3$ and $\mu = 10^{-3}$ Pa·s.

Volume Fraction 1

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

2 Select Boundary 1 only.

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

4 Select the **Phase s2** checkbox.

5 In the $s_{0,s2}$ text field, type `1`.

Outflow 1

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

2 Select Boundary 2 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**, click to expand the **Discretization** section.

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

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 `0.5`.

4 From the κ list, choose **User defined**. In the associated text field, type `1e-9[m^2]`.

Inlet 1

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

2 Select Boundary 1 only.

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

4 In the U_0 text field, type `0.001`.

Pressure 1

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

2 Select Boundary 2 only.

MESH 1

Edge 1

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

Distribution 1

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

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

3 In the **Number of elements** text field, type `400`.

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(0,20,300)`.

Solution 1 (sol1)

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

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

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

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

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

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

7 Click **Run**.

RESULTS

Volume Fraction (phtr)

Two default plots are created automatically — one for the volume fraction and one for the pressure distribution. Add a plot of the analytical solution to the volume fraction plot as follows.

1 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

2 From the **Title type** list, choose **Manual**.

3 In the **Title** text area, type `Volume fraction of phase 2 (1)`.

Line Graph 2

1. Right-click **Volume Fraction (phtr)** and choose **Line Graph**.
2. Select Domain 1 only.
3. In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
4. In the **Expression** text field, type `x`.
5. Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
6. In the **Expression** text field, type `pw1(x)*(0.001*t)/0.5`.
7. Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.
8. From the **Color** list, choose **Cycle (reset)**.
9. In the **Volume Fraction (phtr)** toolbar, click **Plot** and compare with Figure 1.