Created in COMSOL Multiphysics 5.4

Two-Phase Flow in Column

Introduction

The following example analyzes two-phase flow in porous media. Describing how immiscible fluids move through porous media is key to answering many environmental and industrial questions. Unfortunately, multiphase analyses are complicated by the need to solve for multiple dependent variables along with a variety of unknowns. Among them are hydraulic properties that depend on the pressure and saturation levels of each fluid phase.

This problem demonstrates two-phase flow following a U.S. Environmental Protection Agency experimental setup (Ref. 1). This straightforward experiment matches observations for a laboratory column to numerical estimates of two-phase flow. With these column experiments, the researchers evaluated flow for varying fluid pairs (air-water, air-oil, and oil-water) and then match the experimental results to those from computer simulations that employ analytic expressions for retention and permeability. This discussion addresses their work for the Lincoln soil and use formulas from Mualem (Ref. 2) and van Genuchten (Ref. 3) to give hydraulic properties.

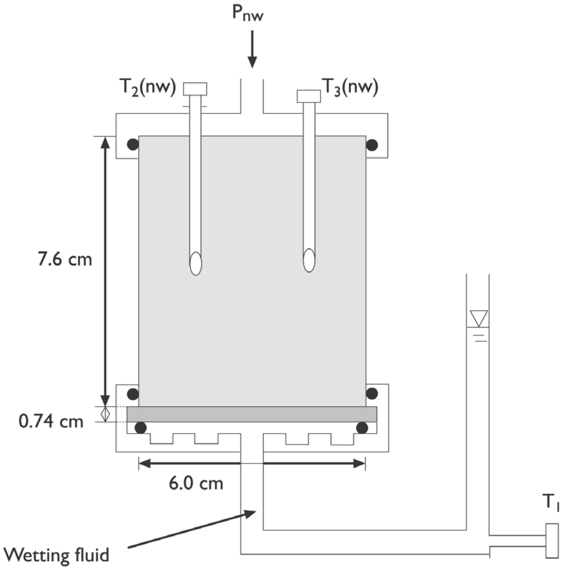


Figure 1: Geometry of the two-phase flow column experiments in Hopmans and others (Ref. 1).

This is a multipart example. The first part sets up the two-phase flow model for water and air; the equations solve for pressures. Saturation varies with the solution. An underlying assumption is that at least some residual air and water exist throughout the soil column at all times. The model tracks the gas front as it displaces a wetting fluid by observing saturation rather than assuming a discrete interface. The second part modifies the air-water simulation for air-oil and oil-water mixtures.

Model Definition

In the experimental setup for air and water, air enters from the top surface of a column made of water and sand. The incoming air (the nonwetting phase) forces the water (the wetting phase) toward the outlet at the base of the column. At the inlet, air pressure increases by steps in time, and no water exits through the column top. In moving to the outlet, the water passes through a disc that is impermeable to airflow. Neither the air nor the water can pass through the vertical column walls. The water pressure at the outlet, which changes in time, corresponds to the height of fluid rise in a receiving buret. The column has a total height of 8.34 cm, a 6-cm radius, and the disk is 0.74 cm thick. The experiment covers 170 hours.

GOVERNING EQUATIONS AND BOUNDARY CONDITIONS

Two-phase flow in porous media follows separate equations for the wetting (w) and nonwetting (nw) fluids:

$$(\theta_\mathrm{s}-\theta_\mathrm{r})\frac{\partial \mathrm{Se}_\mathrm{w}}{\partial t}+\nabla\cdot\left[-\frac{\kappa_\mathrm{int}k_\mathrm{r,\,w}}{\mu_\mathrm{w}}\nabla(p_\mathrm{w}+\rho_\mathrm{w}g\nabla D)\right]=0 \qquad (1)$$

$$(\theta_\mathrm{s}-\theta_\mathrm{r})\frac{\partial \mathrm{Se}_\mathrm{nw}}{\partial t}+\nabla\cdot\left[-\frac{\kappa_\mathrm{int}k_\mathrm{r,\,nw}}{\mu_\mathrm{nw}}\nabla(p_\mathrm{nw}+\rho_\mathrm{nw}g\nabla D)\right]=0 \qquad (2)$$

where θ_s is the total porosity or saturated volume fraction; θ_r is the residual volume fraction, so the difference θ_s-θ_r is the available pore space for phases to move; Se is the effective saturation; t is time; κ_int is the intrinsic permeability of the porous medium (m^2); k_r is the relative permeability (a function of saturation for a given fluid); μ is the fluid's dynamic viscosity (Pa·s); p is pressure (Pa); ρ is the fluid density ($\mathrm{kg/m^3}$); g is acceleration of gravity; and D is the coordinate (for example, x, y, or z) of vertical elevation (m).

When considering a continuous fluid field, neither phase ever completely fills the pore space, giving a volume fraction for the wetting phase, θ_w, and nonwetting phase, θ_nw. For the wetting phase, θ varies from zero or a small residual value θ_r to the total porosity, θ_s. The effective saturation, Se, comes from scaling θ with respect to θ_s and θ_r and so varies from 0 to 1. Both θ and Se are functions of the pressures of both fluids in the system.

The capillary pressure p_c is commonly defined as the difference between the pressure of the nonwetting and wetting phases

$$p_c = p_{nw} - p_w \tag{3}$$

The available pore space can be completely filled with one fluid at a given time, which relates the effective saturations for each phase

$$\mathrm{Se}_w + \mathrm{Se}_{nw} = 1 \tag{4}$$

The specific capacity of the wetting phase $C_{p,\,w}$ depends on changes in the effective saturation with respect to the capillary pressure as

$$C_{p,\,w} = (\theta_r - \theta_s)\frac{\partial \mathrm{Se}_w}{\partial p_c} \tag{5}$$

in the same way, the specific capacity of the nonwetting phase $C_{p,\,nw}$ is defined with the help of Equation 4 as

$$C_{p,\,nw} = (\theta_s - \theta_r)\frac{\partial \mathrm{Se}_{nw}}{\partial p_c} = (\theta_s - \theta_r)\frac{\partial(1 - \mathrm{Se}_w)}{\partial p_c} = -C_{p,\,w}$$

Since the specific capacity of the two phases is the same but with opposite sign, it is just denoted as C_p .

Using Equation 3, Equation 4, and Equation 5 in Equation 1 and Equation 2 simplifies the numerical model. The governing equations become:

$$-C_{p,\,w}\frac{\partial}{\partial t}(p_{nw} - p_w) + \nabla \cdot \left[-\frac{\kappa_{int}k_{r,\,w}}{\mu_w}(\nabla p_w + \rho_w g \nabla D)\right] = 0$$

$$C_{p,\,w}\frac{\partial}{\partial t}(p_{nw} - p_w) + \nabla \cdot \left[-\frac{\kappa_{int}k_{r,\,nw}}{\mu_{nw}}(\nabla p_{nw} + \rho_{nw} g \nabla D)\right] = 0$$

You can solve this system of equations simultaneously for p_w and p_{nw}. In this example, the two fluids are incompressible, but that need not be the case. Rearranging terms, and adding the density of each fluid, we obtain two coupled Darcy's law equations, one for the wetting phase, and another for the nonwetting phase

$$\rho_w C_p \frac{\partial}{\partial t} p_w + \nabla \cdot \rho_w\left[-\frac{\kappa_{int}k_{r,\,w}}{\mu_w}(\nabla p_w + \rho_w g \nabla D)\right] = \rho_w C_p \frac{\partial}{\partial t} p_{nw}$$

$$\rho_{\mathrm{nw}}C_p\frac{\partial}{\partial t}p_{\mathrm{nw}}+\nabla\cdot\rho_{\mathrm{nw}}\left[-\frac{\kappa_{\mathrm{int}}k_{\mathrm{r,\,nw}}}{\mu_{\mathrm{nw}}}(\nabla p_{\mathrm{nw}}+\rho_{\mathrm{nw}}g\nabla D)\right]=\rho_{\mathrm{nw}}C_p\frac{\partial}{\partial t}p_{\mathrm{w}}$$

Initially, the water and air in the column follow hydrostatic distributions. The boundary conditions allow the water to exit only from the base of the soil column. For the wetting phase, the boundary conditions are

$$\begin{array}{ll}\mathbf{n}\cdot\rho_{\mathrm{w}}\left[-\frac{\kappa}{\mu}(\nabla p_{\mathrm{w}}+\rho_{\mathrm{w}}g\nabla D)\right]=0 & \partial\Omega\ \text{Inlet}\\ \mathbf{n}\cdot\rho_{\mathrm{w}}\left[-\frac{\kappa}{\mu}(\nabla p_{\mathrm{w}}+\rho_{\mathrm{w}}g\nabla D)\right]=0 & \partial\Omega\ \text{Sides}\\ p_{\mathrm{w}}=p_{\mathrm{w0}}(t) & \partial\Omega\ \text{Base}\end{array}$$

where **n** is the unit vector normal to the boundary.

Because air enters at the column top but never exits, the boundary conditions for the nonwetting phase are

$$\begin{array}{ll}\mathbf{n}\cdot\rho_{\mathrm{nw}}\left[-\frac{\kappa}{\mu}(\nabla p_{\mathrm{nw}}+\rho_{\mathrm{nw}}g\nabla D)\right]=0 & \partial\Omega\ \text{Surface}\\ \mathbf{n}\cdot\rho_{\mathrm{nw}}\left[-\frac{\kappa}{\mu}(\nabla p_{\mathrm{nw}}+\rho_{\mathrm{nw}}g\nabla D)\right]=0 & \partial\Omega\ \text{Sides}\\ p_{\mathrm{nw}}=p_{\mathrm{nw0}}(t) & \partial\Omega\ \text{Base}\end{array}$$

RETENTION AND PERMEABILITY RELATIONSHIPS

You can set up this two-phase flow analysis using interpolation from experimental data, arbitrary mathematical formulas, or results from other equations in the model to define how θ, C, Se, k_{r}, and p_{c} vary simultaneously. The existing model uses retention and permeability relationships from Ref. 2 and Ref. 3 that express changes in θ, C, Se, and k_{r} as a function of the capillary pressure p_{c}. Because p_{c} is large and because changes in θ, C, Se, and k_{r} are small, these expressions transform capillary pressure to the equivalent height of water or capillary pressure head as in $H_{\mathrm{c}}=p_{\mathrm{c}}/(\rho_{\mathrm{water}}\,g)$. The hydraulic properties relative to the wetting fluid in van Genuchten retention model are

$$\theta_{\mathrm{w}} = \begin{cases} \theta_{\mathrm{r,\,w}} + \mathrm{Se}_{\mathrm{w}}(\theta_{\mathrm{s,\,w}} - \theta_{\mathrm{r,\,w}}) & H_c > 0 \\ \theta_{\mathrm{s,\,w}} & H_c \le 0 \end{cases}$$

$$\mathrm{Se}_{\mathrm{w}} = \begin{cases} \dfrac{1}{[1 + |\alpha H_c|^n]^m} & H_c > 0 \\ 1 & H_c \le 0 \end{cases}$$

$$C_{\mathrm{w}} = \begin{cases} \dfrac{\alpha m}{1-m}(\theta_{\mathrm{s,\,w}} - \theta_{\mathrm{r,\,w}})\mathrm{Se}_{\mathrm{w}}^{\frac{1}{m}}\left(1 - \mathrm{Se}_{\mathrm{w}}^{\frac{1}{m}}\right)^{m} & H_c > 0 \\ 0 & H_c \le 0 \end{cases}$$

$$k_{\mathrm{r,\,w}} = \begin{cases} \mathrm{Se}_{\mathrm{w}}^{L}\left[1 - \left(1 - \mathrm{Se}_{\mathrm{w}}^{\frac{1}{m}}\right)^{m}\right]^{2} & H_c > 0 \\ 1 & H_c \le 0 \end{cases}$$

where α, n, m, and L denote soil characteristics. Note that with two-phase flow, the van Genuchten-Mualem formulas hinge on the value of H_{c}.

For the nonwetting fluid, the properties

$$\begin{aligned} \theta_{\mathrm{nw}} &= \theta_{s,\,\mathrm{w}} - \theta_{\mathrm{w}} \\ \mathrm{Se}_{\mathrm{nw}} &= 1 - \mathrm{Se}_{\mathrm{w}} \\ C_{\mathrm{nw}} &= -C_{\mathrm{w}} \\ k_{\mathrm{r,\,nw}} &= (1 - \mathrm{Se}_{\mathrm{w}})^{L}\left(1 - \mathrm{Se}_{\mathrm{w}}^{\frac{1}{m}}\right)^{m2} \end{aligned}$$

arise from the definitions for the wetting phase.

DIFFERENT FLUID PAIRS

When switching between air-water, air-oil, and oil-water experiments, the authors used different scaling with interfacial tensions according to Leverett (Ref. 4). The Leverett scaling adjusts the nonwetting phase pressure at the column top to produce the same volume of wetting fluid outflow at the column bottom regardless of the fluid pair. With

Leverett scaling, switching between fluid pairs requires using the correct fluid properties ρ and μ for the fluid pair and adjusting the boundary and initial pressures according to

$$\sigma_{\mathrm{aw}} p_{\mathrm{c,\,aw}} = \sigma_{\mathrm{aw}} p_{\mathrm{c,\,aw}}$$
$$\sigma_{\mathrm{aw}} p_{\mathrm{c,\,aw}} = \sigma_{\mathrm{aw}} p_{\mathrm{c,\,ao}}$$
$$\sigma_{\mathrm{aw}} p_{\mathrm{c,\,aw}} = \sigma_{\mathrm{aw}} p_{\mathrm{c,\,ow}}$$

In these equations, σ represents the interfacial tension between the different fluids, and the subscripts denote the fluid pair. These values appear in a table at the end of this section. For example, $\sigma_{\mathrm{ao}}/\sigma_{\mathrm{aw}}$ equals 0.373, and $\sigma_{\mathrm{ow}}/\sigma_{\mathrm{aw}}$ equals 0.534; further, the first nonwetting phase pressure head (in meters of water) is 0.4 m for the air-water system, 0.1 m for the air-oil system, and 0.2 m for the water-oil system.

Because relative permeability and retention properties for a porous medium depend on the fluid moving through it, switching fluid pairs also requires switching the retention and permeability properties in the model. This requirement can mean inserting new experimental data or adjusting mathematical formulas. In this model, the authors assessed the permeability and retention parameters were assessed by curve fitting to analytic formulas. They adjusted the parameters α, n, κ_s, and θ_r to get the best fit for each fluid. A review of the data tables that follow reveals that the ratios in the α values for the different fluid pairs roughly equals the σ ratios just given.

Implementation: Step Change on a Boundary

The following step-by-step instructions define the timing of the stepped nonwetting phase pressures at the inlet by using an interpolation function. Interpolating in COMSOL Multiphysics is straightforward. Add an **Interpolation** function under **Global Definitions**, set up the table with the times and corresponding pressure heads, assign a name to the interpolation function, and use it where the function is needed. To activate the functions created, simply enter the function name (for example, `Hp_nw_t`) along with the argument, that is the time t in parenthesis. For example, you define the non wetting pressure as

```
p_nw = Hp_nw_t(t)*rho_water*g_const
```

The density of water appears in the equation because Ref. 1 defines the boundary pressure as a height of water.

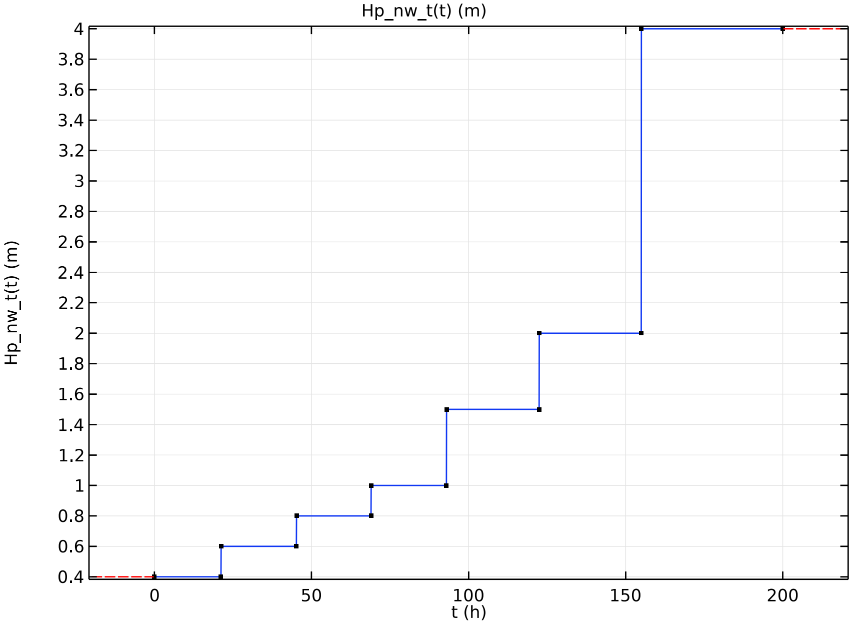


Figure 2: Interpolation function used for Hp_nw_t(t).

Data

The data used in this model correspond to the air-water experiments for the Lincoln sand as reported in Ref. 1:

VARIABLE	EXPRESSION	DESCRIPTION
ρ_{w}	1000 kg/m^3	Fluid density, water
μ_{w}	$1\cdot10^{-3}$ Pa·s	Dynamic viscosity, water
ρ_{a}	1.28 kg/m^3	Fluid density, air
μ_{a}	$1.81\cdot10^{-5}$ Pa·s	Dynamic viscosity, air
κ_{int}	2480 mD	Intrinsic permeability, column
κ_{s}	13.57 mD	Permeability, disc
θ_{s}	0.32	Saturated volume fraction, columnTab
$\theta_{\mathrm{s,u}}$	0.5	Saturated volume fraction, disc
$p_{\mathrm{nw,top}}$	0.2 m water	Initial pressure head at inlet, nonwetting phase

The van Genuchten parameters for the different fluid pairs are the following:

VARIABLE	DESCRIPTION	UNIT	AIR-WATER	AIR-OIL	OIL-WATER
$q_{\mathrm{r,w}}$	Residual volume fraction		0.021	0.00001	0.0072
α	alpha parameter	m^{-1}	1.89	5.29	3.58
n	n parameter, column		2.811	3.002	3.1365
L	L parameter, column		0.5	0.5	0.5
κ_{s}	Permeability, disc	m^{-2}	$2.48 \cdot 10^{-12}$	$1.09 \cdot 10^{-12}$	$0.94 \cdot 10^{-12}$

Pressure head at the air inlet increments in time according to the table below, see Figure 2.

PRESSURE HEAD (m water)	START TIME (hours)
0.4	0
0.6	21.25
0.8	45.25
1.0	69
1.5	93
2	122.5
4	155

At the water outlet, the fluid level in the receiving buret increases linearly in time from 0 m to 0.1 m.

Results and Discussion

Figure 3 shows an early-time snapshot from the COMSOL Multiphysics solution for two-phase flow in a laboratory column. The shading depicts the effective saturation of the nonwetting phase (air), while the arrows give the wetting phase (water) velocities.

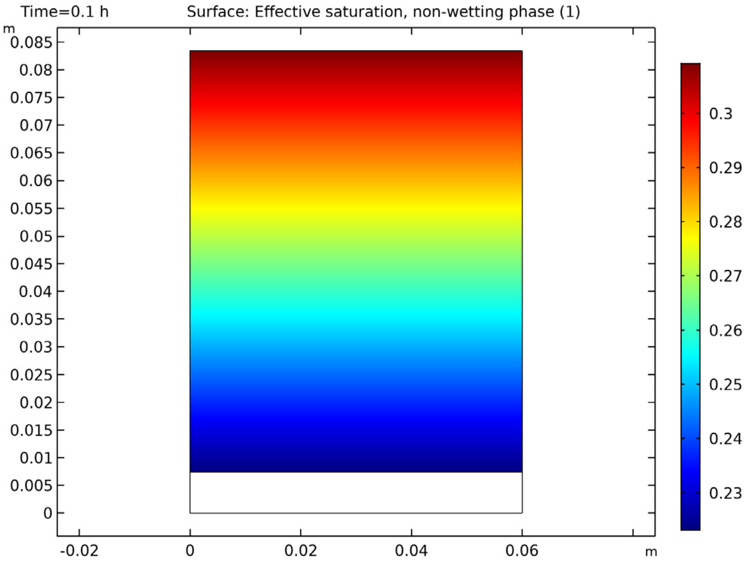


Figure 3: Solution to two-phase flow model at 0.1 hours: nonwetting phase saturation (surface plot). Results correspond to air-water experiment on Lincoln soil from the US EPA.

The image illustrates the nonwetting fluid entering the soil column and displacing the wetting fluid. The nonwetting phase enters because it is being forced into the inlet with a multi-step pressure change.

Figure 4 shows the stepped pressure head used at the inlet boundary along with the capillary pressure in the column at various elevations. Specifying the point locations during postprocessing circumvents the need to plan observation sites during input. The solution to the two-phase flow problem provided is an excellent match to the results of Ref. 1.

That the capillary pressure head and the air inlet pressure in Figure 4 track together is what made the laboratory setup successful. To get high resolution on the permeability and retention behaviors, the authors in Ref. 1 set the pressure steps large enough that the impact is instantaneous in the soil column. As shown in Figure 5, the permeability changes instantaneously throughout the column.

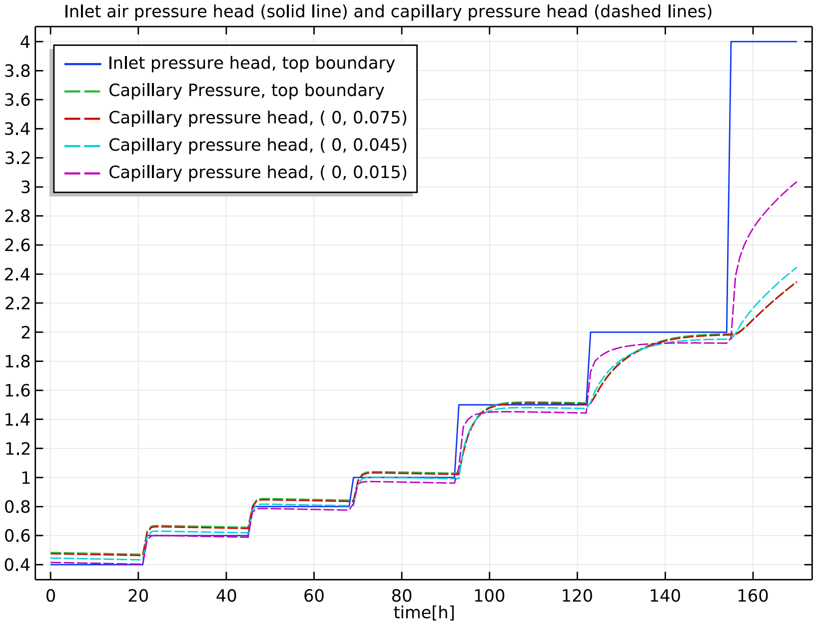


Figure 4: Inlet air pressure head (solid line) and capillary pressure head (dashed lines) for air-water flow in Lincoln soil.

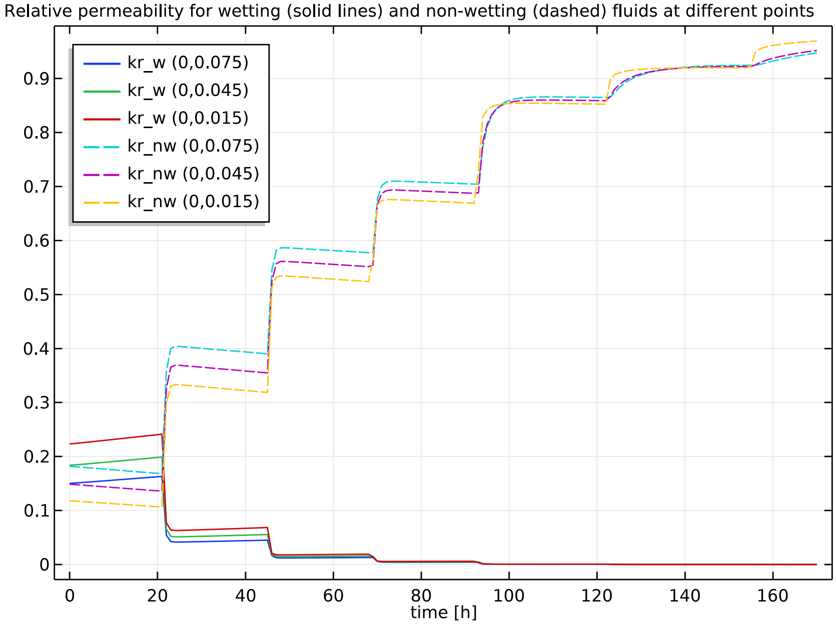


Figure 5: Relative permeability functions for water (solid lines) and air (dashed lines) for Lincoln soil at three different points.

Solutions for two-phase flow for the air-oil and oil-water systems appear in Figure 6 and Figure 7, respectively.

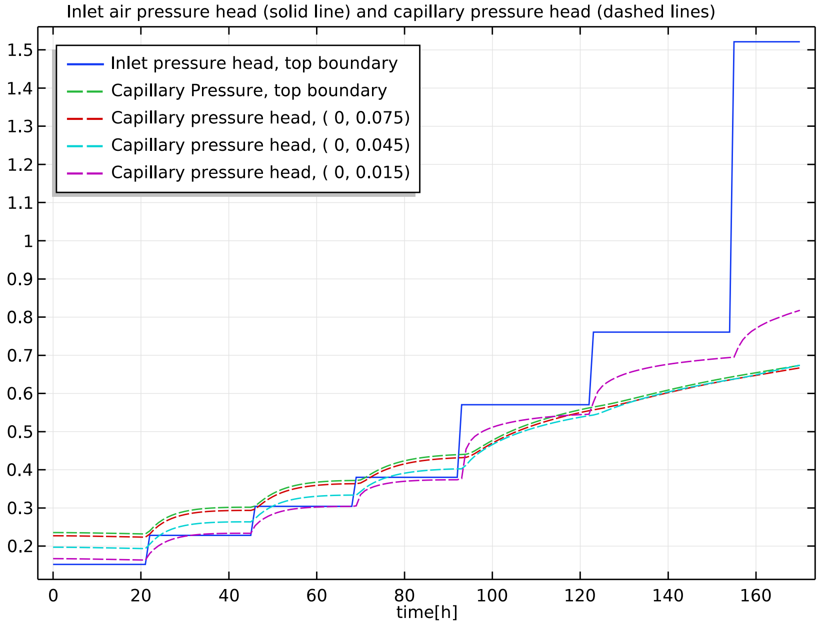


Figure 6: Inlet-air pressure head (solid line) and capillary pressure head (dashed lines) for air-oil flow in Lincoln soil.

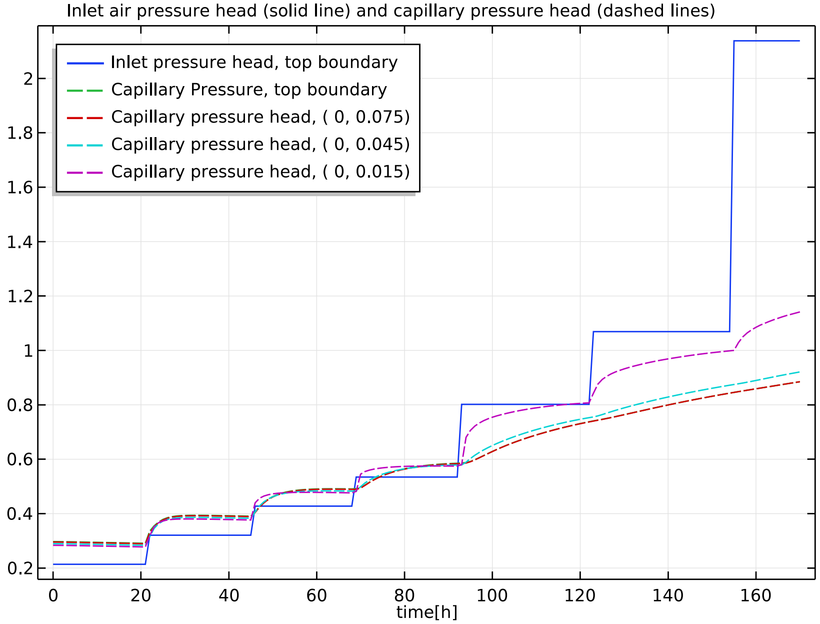


Figure 7: Inlet-air pressure head (solid line) and capillary pressure head (dashed lines) for oil-water flow in Lincoln soil.

The COMSOL Multiphysics results for the air-oil and oil-water two-phase flow problems prove to be in agreement to the results shown in Ref. 1. Through Leverett scaling you set the inlet pressure so that the air-oil and oil-water systems would produce the volume outflow rate from the air-water experiment. As with the air-water system, the capillary pressure head and air-inlet pressure for the air-oil experiment track instantaneously. For the water-oil system, however, there is a lag between the nonwetting and wetting phase pressures.

References

1. J.W. Hopmans, M.E. Grismer, J. Chen, and Y.P. Liu, *Parameter Estimation of Two-fluid Capillary Pressure Saturation and Permeability Functions*, U.S. Environmental Protection Agency EPA/600/R-98/046, Cincinnati, Ohio, 1998.

2. Y. Mualem, “A new model for predicting the hydraulic permeability of unsaturated porous media”, *Water Res. Research*, vol. 12, pp. 513–522, 1976.

3. M.Th. van Genuchten, “A closed-form equation for predicting the hydraulic of conductivity of unsaturated soils”, *Soil Sci. Soc. Am. J.*, vol. 44, pp. 892–898, 1980.

4. M.C. Leverett, “Capillary behavior in porous solids”, *Trans. AIME*, vol. 142, pp. 152–169, 1941.

Application Library path: `Subsurface_Flow_Module/Fluid_Flow/twophase_flow_column`

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**.

2 In the **Select Physics** tree, select **Fluid Flow>Porous Media and Subsurface Flow>Darcy’s Law (dl)**.

3 Click **Add**.

4 In the **Pressure** text field, type `p_w`.

5 In the **Select Physics** tree, select **Fluid Flow>Porous Media and Subsurface Flow>Darcy’s Law (dl)**.

6 Click **Add**.

7 In the **Pressure** text field, type `p_nw`.

8 Click **Study**.

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

10 Click **Done**.

GLOBAL DEFINITIONS

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 `twophase_flow_column_parameters.txt`.

Define an interpolation function for the stepped pressure on the non-wetting phase boundary using the data available in a file.

Interpolation 1 (int1)

1 In the **Home** toolbar, click **Functions** and choose **Global>Interpolation**.

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

3 From the **Data source** list, choose **File**.

4 Click **Browse**.

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

6 Click **Import**.

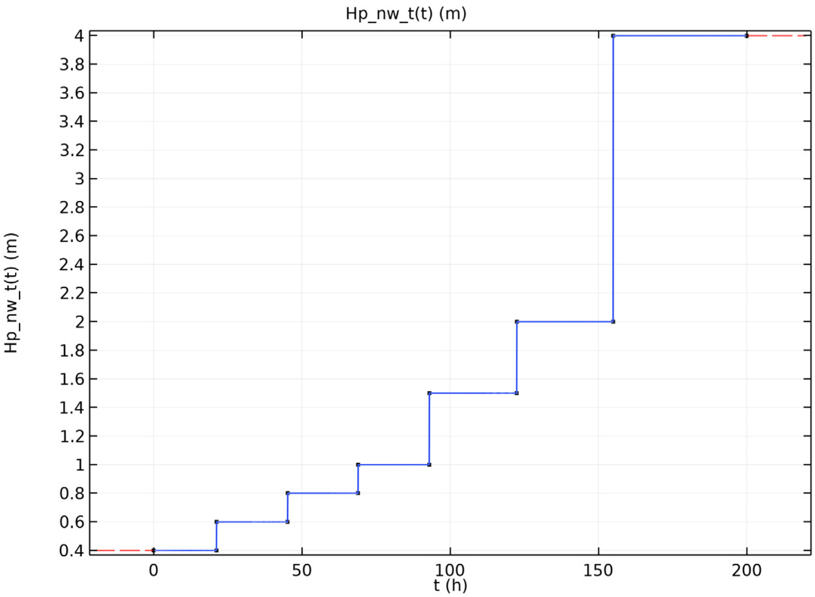
7 In the **Function name** text field, type `Hp_nw_t`.

8 Locate the **Interpolation and Extrapolation** section. From the **Extrapolation** list, choose **Nearest function**.

9 Locate the **Units** section. In the **Arguments** text field, type `h`.

10 In the **Function** text field, type `m`.

11 Click **Plot**.



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 **Height** text field, type `0.0834`.
4. In the **Width** text field, type `0.06`.
5. Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`0.0074`

6. Click **Build All Objects**.

DEFINITIONS

The following modeling steps create a step function that is used later to create a smooth ramp for the effective saturation.

Ramp 1 (rm1)

1. In the **Home** toolbar, click **Functions** and choose **Local>Ramp**.
2. In the **Settings** window for **Ramp**, click to expand the **Smoothing** section.
3. Select the **Size of transition zone at start** check box.
4. In the associated text field, type `1e-3`.

   Add variables to simulate two-phase flow for the air-water system.

Variables 1

1. In the **Home** toolbar, click **Variables** and choose **Local Variables**.
2. In the **Settings** window for **Variables**, type `Air-water experiment` in the **Label** text field.
3. Locate the **Variables** section. Click **Load from File**.
4. Browse to the model's Application Libraries folder and double-click the file `twophase_flow_column_air_water.txt`.

Import the variables that define the Van Genuchten retention model, initial and boundary conditions as well as the material properties of the lower and the upper layers. The parameters are presented in the Data section.

Variables 2

1. In the **Home** toolbar, click **Variables** and choose **Local Variables**.
2. In the **Settings** window for **Variables**, type `Van Genuchten retention model` in the **Label** text field.
3. Locate the **Variables** section. Click **Load from File**.
4. Browse to the model's Application Libraries folder and double-click the file `twophase_flow_column_retention_model.txt`.

Variables 3

1. In the **Home** toolbar, click **Variables** and choose **Local Variables**.
2. In the **Settings** window for **Variables**, type `Initial and boundary conditions` in the **Label** text field.
3. Locate the **Variables** section. Click **Load from File**.
4. Browse to the model's Application Libraries folder and double-click the file `twophase_flow_column_initial_conditions.txt`.

Variables 4

1. In the **Home** toolbar, click **Variables** and choose **Local Variables**.

2. In the **Settings** window for **Variables**, type `Lower layer` in the **Label** text field.
3. Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.
4. Select Domain 1 only.
5. Locate the **Variables** section. Click **Load from File**.
6. Browse to the model's Application Libraries folder and double-click the file `twophase_flow_column_lower_layer.txt`.

Variables 5

1. In the **Home** toolbar, click **Variables** and choose **Local Variables**.
2. In the **Settings** window for **Variables**, type `Upper layer` in the **Label** text field.
3. Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.
4. Select Domain 2 only.
5. Locate the **Variables** section. Click **Load from File**.
6. Browse to the model's Application Libraries folder and double-click the file `twophase_flow_column_upper_layer.txt`.

MATERIALS

Material 1 (mat1)

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, type `Wetting fluid` in the **Label** text field.

Material 2 (mat2)

1. Right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, type `Non-wetting fluid` in the **Label** text field.

   Before defining the material properties, configure the physics selecting wetting and non-wetting domains. This allows COMSOL Multiphysics to indicate what material properties you need to enter.

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.

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 **Fluid Properties** section.
3. From the **Fluid material** list, choose **Wetting fluid (mat1)**.
4. Locate the **Matrix Properties** section. From the ε_p list, choose **User defined**. In the associated text field, type `0.25`.
5. From the κ list, choose **User defined**. In the associated text field, type `kap_s`.

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_w text field, type `p_w_init`.

Storage Model 1

1. In the **Physics** toolbar, click **Domains** and choose **Storage Model**.
2. Select Domain 2 only.
3. In the **Settings** window for **Storage Model**, locate the **Fluid Properties** section.
4. From the **Fluid material** list, choose **Wetting fluid (mat1)**.
5. Locate the **Matrix Properties** section. From the ε_p list, choose **User defined**. In the associated text field, type `0.25`.
6. From the κ list, choose **User defined**. In the associated text field, type `kap_s*kr_w`.
7. Locate the **Storage Model** section. From the **Storage** list, choose **User defined**. In the S text field, type `Cp`.

Pressure 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Pressure**.
2. Select Boundary 2 only.
3. In the **Settings** window for **Pressure**, locate the **Pressure** section.
4. In the p_0 text field, type `p_w0`.

Mass Source 1

1. In the **Physics** toolbar, click **Domains** and choose **Mass Source**.
2. Select Domain 2 only.
3. In the **Settings** window for **Mass Source**, locate the **Mass Source** section.

4. In the Q_m text field, type `Cp*p_nwt*rho_w`.

DARCY'S LAW 2 (DL2)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Darcy's Law 2 (dl2)**.
2. Select Domain 2 only.
3. In the **Settings** window for **Darcy's Law**, locate the **Gravity Effects** section.
4. Select the **Include gravity** check box.

Initial Values 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Darcy's Law 2 (dl2)** click **Initial Values 1**.
2. In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
3. In the p_{nw} text field, type `p_nw_init`.
4. In the **Model Builder** window, click **Darcy's Law 2 (dl2)**.

Storage Model 1

1. In the **Physics** toolbar, click **Domains** and choose **Storage Model**.
2. Select Domain 2 only.
3. In the **Settings** window for **Storage Model**, locate the **Fluid Properties** section.
4. From the **Fluid material** list, choose **Non-wetting fluid (mat2)**.
5. Locate the **Matrix Properties** section. From the ε_p list, choose **User defined**. In the associated text field, type `0.25`.

MATERIALS

Non-wetting fluid (mat2)

1. From the κ list, choose **User defined**. In the associated text field, type `kap_s*kr_nw`.
2. Locate the **Storage Model** section. From the **Storage** list, choose **User defined**. In the S text field, type `Cp`.

DARCY'S LAW 2 (DL2)

Pressure 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Pressure**.
2. Select Boundary 5 only.
3. In the **Settings** window for **Pressure**, locate the **Pressure** section.
4. In the p_0 text field, type `p_nw_top`.

Mass Source 1

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

2 Select Domain 2 only.

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

4 In the Q_{m} text field, type `Cp*p_wt*rho_nw`.

MATERIALS

Wetting fluid (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Wetting fluid (mat1)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Density	rho	`rho_w`	kg/m³	Basic
Dynamic viscosity	mu	`mu_w`	Pa·s	Basic

Non-wetting fluid (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Non-wetting fluid (mat2)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Density	rho	`rho_nw`	kg/m³	Basic
Dynamic viscosity	mu	`mu_nw`	Pa·s	Basic

MESH 1

Mapped 1

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

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

3 From the **Geometric entity level** list, choose **Entire geometry**.

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 **Extra fine**.

4 Click **Build All**.

STUDY 1

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

2 In the **Settings** window for **Study**, type `Air-water` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

AIR-WATER

Step 1: Time Dependent

1 In the **Model Builder** window, under **Air-water** 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 **h**.

4 In the **Times** text field, type `0 0.001 0.01 0.1 range(1,170)`.

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

RESULTS

Data Sets

Create point data sets for plotting the capillary pressure head and the relative permeabilities at several points.

Cut Point 2D 1

1 In the **Results** toolbar, click **Cut Point 2D**.

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

3 In the **X** text field, type `0 0 0`.

4 In the **Y** text field, type `0.075 0.045 0.015`.

5 Click **Plot**.

Next, reproduce the plot in Figure 4.

1D Plot Group 1

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Inlet air pressure and capillary pressure, air-water` in the **Label** text field.

3 Locate the **Plot Settings** section. Select the **x-axis label** check box.

4 In the associated text field, type `time[h]`.

5 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

6 In the **Title** text area, type `Inlet air pressure head (solid line) and capillary pressure head (dashed lines)`.

7 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Point Graph 1

1 Right-click **Inlet air pressure and capillary pressure, air-water** and choose **Point Graph**.

2 Select Point 6 only.

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

4 In the **Expression** text field, type `Hp_nw_t(t)`.

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

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

7 In the table, enter the following settings:

Legends
`Inlet pressure head, top boundary`

8 In the **Inlet air pressure and capillary pressure, air-water** toolbar, click **Plot**.

Point Graph 2

1 Right-click **Results>Inlet air pressure and capillary pressure, air-water>Point Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Definitions>Variables>Hc - Capillary pressure head - m**.

3 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Capillary Pressure, top boundary`

5 In the **Inlet air pressure and capillary pressure, air-water** toolbar, click **Plot**.

Point Graph 3

1 Right-click **Results>Inlet air pressure and capillary pressure, air-water>Point Graph 2** and choose **Duplicate**.

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

3 From the **Data set** list, choose **Cut Point 2D 1**.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Capillary pressure head, ( 0, 0.075)`
`Capillary pressure head, ( 0, 0.045)`
`Capillary pressure head, ( 0, 0.015)`

5 In the **Inlet air pressure and capillary pressure, air-water** toolbar, click **Plot**.

To generate the plots in Figure 5 and Figure 3, continue with the steps below.

1D Plot Group 2

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

2 In the **Settings** window for **1D Plot Group**, type `Relative permeabilities at 3 points` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Cut Point 2D 1**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Relative permeability for wetting (solid lines) and non-wetting (dashed) fluids at different points`.

6 Locate the **Plot Settings** section. Select the **x-axis label** check box.

7 In the associated text field, type `time [h]`.

8 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Point Graph 1

1 Right-click **Relative permeabilities at 3 points** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Definitions>Variables>kr_w - Relative permeability, wetting phase**.

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

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

5 In the table, enter the following settings:

Legends
`kr_w (0,0.075)`

Legends
`kr_w (0,0.045)`
`kr_w (0,0.015)`

Point Graph 2

1 Right-click **Results>Relative permeabilities at 3 points>Point Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Definitions> Variables>kr_nw - Relative permeability, non-wetting phase**.

3 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`kr_nw (0,0.075)`
`kr_nw (0,0.045)`
`kr_nw (0,0.015)`

5 In the **Relative permeabilities at 3 points** toolbar, click **Plot**.

2D Plot Group 3

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

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

3 From the **Time (h)** list, choose **0.1**.

Surface 1

1 Right-click **2D Plot Group 3** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Definitions>Variables> Se_nw - Effective saturation, non-wetting phase**.

3 In the **2D Plot Group 3** toolbar, click **Plot**.

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

2D Plot Group 3

1 In the **Model Builder** window, under **Results** click **2D Plot Group 3**.

2 In the **Settings** window for **2D Plot Group**, type `Effective Saturation, non-wetting phase` in the **Label** text field.

Perform similar studies with air-oil and oil-water to generate the plots in Figure 6 and Figure 7.

DEFINITIONS

Variables 6

1. In the **Home** toolbar, click **Variables** and choose **Local Variables**.
2. In the **Settings** window for **Variables**, type `Air-oil experiment` in the **Label** text field.
3. Locate the **Variables** section. Click **Load from File**.
4. Browse to the model's Application Libraries folder and double-click the file `twophase_flow_column_air_oil.txt`.

Variables 7

1. In the **Home** toolbar, click **Variables** and choose **Local Variables**.
2. In the **Settings** window for **Variables**, type `Oil-water experiment` in the **Label** text field.
3. Locate the **Variables** section. Click **Load from File**.
4. Browse to the model's Application Libraries folder and double-click the file `twophase_flow_column_oil_water.txt`.

AIR-WATER

Step 1: Time Dependent

1. In the **Model Builder** window, under **Air-water** click **Step 1: Time Dependent**.
2. In the **Settings** window for **Time Dependent**, locate the **Physics and Variables Selection** section.
3. Select the **Modify model configuration for study step** check box.
4. In the **Physics and variables selection** tree, select **Component 1 (comp1)>Definitions>Air-oil experiment** and **Component 1 (comp1)>Definitions>Oil-water experiment**.
5. Click **Disable**.

ADD STUDY

1. In the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2. Go to the **Add Study** window.
3. Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Time Dependent**.
4. Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

1 In the **Settings** window for **Study**, type `Air-oil` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

AIR-OIL

Step 1: Time Dependent

1 In the **Model Builder** window, under **Air-oil** 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 **h**.

4 In the **Times** text field, type `0 0.001 0.01 0.1 range(1,170)`.

5 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

6 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Definitions>Air-water experiment** and **Component 1 (comp1)>Definitions>Oil-water experiment**.

7 Click **Disable**.

8 In the **Home** toolbar, click **Compute**.

RESULTS

Cut Point 2D 2

1 In the **Model Builder** window, under **Results>Data Sets** right-click **Cut Point 2D 1** and choose **Duplicate**.

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

3 From the **Data set** list, choose **Air-oil/Solution 2 (sol2)**.

Inlet air pressure and capillary pressure, air-water 1

1 In the **Model Builder** window, under **Results** right-click **Inlet air pressure and capillary pressure, air-water** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Inlet air pressure and capillary pressure, air-oil` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Air-oil/Solution 2 (sol2)**.

Point Graph 1

1 In the **Model Builder** window, expand the **Results> Inlet air pressure and capillary pressure, air-oil** node, then click **Point Graph 1**.

2. In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
3. In the **Expression** text field, type `Hp_nw_t(t)*sigma`.

Point Graph 3

1. In the **Model Builder** window, under **Results>Inlet air pressure and capillary pressure, air-oil** click **Point Graph 3**.
2. In the **Settings** window for **Point Graph**, locate the **Data** section.
3. From the **Data set** list, choose **Cut Point 2D 2**.
4. In the **Inlet air pressure and capillary pressure, air-oil** toolbar, click **Plot**.

ADD STUDY

1. In the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2. Go to the **Add Study** window.
3. Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Time Dependent**.
4. Click **Add Study** in the window toolbar.
5. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 3

1. In the **Settings** window for **Study**, type `Oil-water` in the **Label** text field.
2. Locate the **Study Settings** section. Clear the **Generate default plots** check box.

OIL-WATER

Step 1: Time Dependent

1. In the **Model Builder** window, under **Oil-water** 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 **h**.
4. In the **Times** text field, type `0 0.001 0.01 0.1 range(1,170)`.
5. Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.
6. In the **Physics and variables selection** tree, select **Component 1 (comp1)>Definitions>Air-water experiment** and **Component 1 (comp1)>Definitions>Air-oil experiment**.
7. Click **Disable**.
8. In the **Home** toolbar, click **Compute**.

RESULTS

Cut Point 2D 3

1. In the **Model Builder** window, under **Results>Data Sets** right-click **Cut Point 2D 2** and choose **Duplicate**.
2. In the **Settings** window for **Cut Point 2D**, locate the **Data** section.
3. From the **Data set** list, choose **Oil-water/Solution 3 (sol3)**.

Inlet air pressure and capillary pressure, air-oil 1

1. In the **Model Builder** window, under **Results** right-click **Inlet air pressure and capillary pressure, air-oil** and choose **Duplicate**.
2. In the **Settings** window for **1D Plot Group**, type `Inlet air pressure and capillary pressure, oil-water` in the **Label** text field.
3. Locate the **Data** section. From the **Data set** list, choose **Oil-water/Solution 3 (sol3)**.

Point Graph 3

1. In the **Model Builder** window, expand the **Results>Inlet air pressure and capillary pressure, oil-water** node, then click **Point Graph 3**.
2. In the **Settings** window for **Point Graph**, locate the **Data** section.
3. From the **Data set** list, choose **Cut Point 2D 3**.
4. In the **Inlet air pressure and capillary pressure, oil-water** toolbar, click **Plot**.