Created in COMSOL Multiphysics 5.6

Capillary Filling — Phase Field Method

Introduction

Surface tension and wall adhesive forces are often used to transport fluid through microchannels in MEMS devices or to measure the transport and position of small amounts of fluid using micropipettes. Multiphase flow through a porous medium and droplets on solid walls are other examples where wall adhesion and surface tension strongly influence the dynamics of the flow.

This example studies a narrow vertical cylinder placed on top of a reservoir filled with water. Because of wall adhesion and surface tension at the air/water interface, water rises through the channel. The model calculates the velocity field, the pressure field, and the shape and position of the water surface.

This example demonstrates how to model the filling of a capillary channel using two multiphysics coupling features available in the CFD Module. You can use either the Two-Phase Flow, Level Set or the Two-Phase Flow, Phase Field multiphysics coupling feature. The Level Set interface uses a reinitialized level set method to represent the fluid interface between the air and the water. The Phase Field interface, on the other hand, uses a Cahn-Hilliard equation, including a chemical potential to represent a diffuse interface separating the two phases. The Navier-Stokes equations are used to describe the momentum transport and the conservation of mass.

Model Definition

The model consists of a capillary channel of radius 0.15 mm attached to a water reservoir. Water can flow freely into the reservoir. Because both the channel and the reservoir are cylindrical, you can use the axisymmetric geometry illustrated in Figure 1. Initially, the thin cylinder is filled with air. Wall adhesion causes water to creep up along the cylinder boundaries. The deformation of the water surface induces surface tension at the air/water interface, which in turn creates a pressure jump across the interface. The pressure variations cause water and air to move upward. The fluids continue to rise until the capillary forces are balanced by the gravity force building up as the water rises in the channel. In the present example, the capillary forces dominate over gravity throughout the simulation. Consequently, the interface moves upward during the entire simulation.

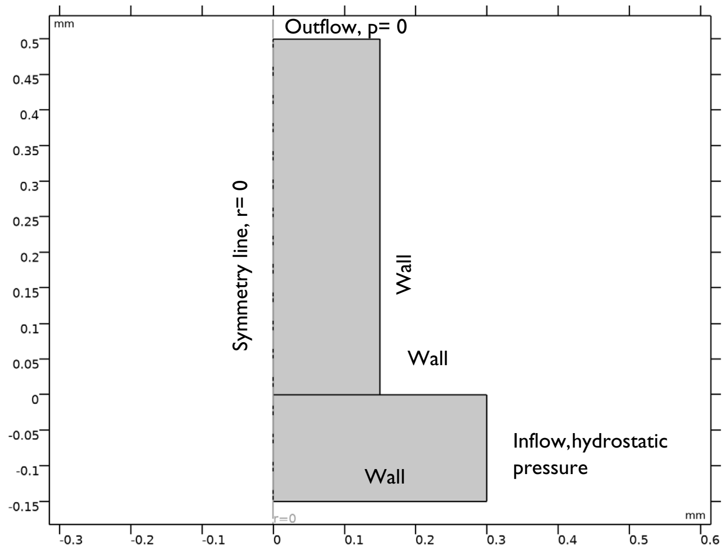


Figure 1: Axisymmetric geometry with boundary conditions.

REPRESENTATION AND CONVECTION OF THE FLUID INTERFACE

Level Set Method

The Level Set interface automatically sets up the equations for the convection of the interface. The fluid interface is represented by the 0.5 contour of the level set function ϕ. In air $\phi = 0$ and in water $\phi = 1$. The level set function can thus be thought of as the volume fraction of water. The transport of the fluid interface separating the two phases is given by

$$\frac{\partial\phi}{\partial t} + \mathbf{u}\cdot\nabla\phi = \gamma\nabla\cdot\left(\varepsilon\nabla\phi - \phi(1-\phi)\frac{\nabla\phi}{|\nabla\phi|}\right)$$

The ε parameter determines the thickness of the interface. When stabilization is used for the level set equation, you can typically use an interface thickness of $\varepsilon = h_c/2$, where h_c is the characteristic mesh size in the region passed by the interface. The γ parameter determines the amount of reinitialization. A suitable value for γ is the maximum velocity magnitude occurring in the model. The multiphysics coupling feature defines the density and viscosity according to:

$$\rho = \rho_{\mathrm{air}} + (\rho_{\mathrm{water}} - \rho_{\mathrm{air}})\phi$$
$$\mu = \mu_{\mathrm{air}} + (\mu_{\mathrm{water}} - \mu_{\mathrm{air}})\phi$$

Due to these definitions, the density and viscosity vary smoothly across the fluid interface. The delta function is approximated by

$$\delta = 6|\phi(1-\phi)||\nabla\phi|$$

and the interface normal is calculated from

$$\mathbf{n} = \frac{\nabla\phi}{|\nabla\phi|}$$

Phase Field Method

In the Phase Field interface the two-phase flow dynamics is governed by a Cahn-Hilliard equation. The equation tracks a diffuse interface separating the immiscible phases. The diffuse interface is defined as the region where the dimensionless phase field variable ϕ goes from -1 to 1. When solved in COMSOL Multiphysics, the Cahn-Hilliard equation is split up into two equations

$$\frac{\partial\phi}{\partial t} + \mathbf{u}\cdot\nabla\phi = \nabla\cdot\frac{\gamma\lambda}{\varepsilon^2}\nabla\psi$$

$$\psi = -\nabla\cdot\varepsilon^2\nabla\phi + (\phi^2 - 1)\phi$$

where $\mathbf{u}$ is the fluid velocity (m/s), γ is the mobility ($\mathrm{m^3 \cdot s/kg}$), λ is the mixing energy density (N) and ε (m) is the interface thickness parameter. The ψ variable is referred to as the phase field help variable. The following equation relates the mixing energy density and the interface thickness to the surface tension coefficient:

$$\sigma = \frac{2\sqrt{2}}{3}\frac{\lambda}{\varepsilon}$$

You can typically set the interface thickness parameter to $\varepsilon = h_c/2$, where h_c is the characteristic mesh size in the region passed by the interface. The mobility parameter γ determines the time scale of the Cahn-Hilliard diffusion and must be chosen judiciously. It must be large enough to retain a constant interfacial thickness but small enough so that the convective terms are not overly damped. The default value, $\gamma = \varepsilon^2$, is usually a good initial guess. This model uses a higher mobility to obtain the correct pressure variation over the interface.

In the Phase Field interface, the volume fractions of the individual fluids are

$$V_{f1} = \frac{1-\phi}{2}, \qquad V_{f2} = \frac{1+\phi}{2}$$

In the present model water is defined as Fluid 1 and air as Fluid 2.

The multiphysics coupling feature defines the density (kg/m^3) and the viscosity (Pa·s) of the mixture to vary smoothly over the interface by letting

$$\rho = \rho_w + (\rho_{air} - \rho_w)V_{f2}$$
$$\mu = \mu_w + (\mu_{air} - \mu_w)V_{f2}$$

where the single phase water properties are denoted w and the air properties air.

MASS AND MOMENTUM TRANSPORT

The Navier-Stokes equations describe the transport of mass and momentum for fluids of constant density. In order to account for capillary effects, it is crucial to include surface tension in the model. The Navier-Stokes equations are then

$$\rho\frac{\partial \mathbf{u}}{\partial t} + \rho(\mathbf{u}\cdot\nabla)\mathbf{u} = \nabla\cdot[-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T)] + \mathbf{F}_{st} + \rho\mathbf{g}$$

$$\nabla\cdot\mathbf{u} = 0$$

Here, ρ denotes the density (kg/m^3), μ equals the dynamic viscosity (Ns/m^2), $\mathbf{u}$ represents the velocity (m/s), p denotes the pressure (Pa), and $\mathbf{g}$ is the gravity vector (m/s^2). $\mathbf{F}_{st}$ is the surface tension force acting at the air/water interface.

Surface Tension

In the Level Set interface the surface tension force is computed as

$$\mathbf{F}_{st} = \sigma\delta\kappa\mathbf{n}$$

Here, $\mathbf{n}$ is the interface normal, σ is the surface tension coefficient (N/m), $\kappa = -\nabla\cdot\mathbf{n}$ is the curvature, and δ equals a Dirac delta function that is nonzero only at the fluid interface.

The following boundary force is added to enforce the contact angle:

$$\mathbf{F}_{\theta} = \sigma\delta(\mathbf{n}_{wall}\cdot\mathbf{n} - \cos\theta_w)\mathbf{n} \tag{1}$$

where θ is the contact angle (see Figure 2). If you apply a no slip boundary condition, the velocity vanishes on that boundary, and you cannot specify the contact angle. Instead, the interface remains fixed on the wall. However, if you allow a small amount of slip, it is possible to specify the contact angle. The Wetted Wall coupling feature adds the term given by Equation 1 and consequently makes it possible to set the contact angle.

In the Phase Field interface, the diffuse interface representation makes it possible to compute the surface tension by

$$\mathbf{F}_{\mathrm{st}} = G\nabla\phi$$

where ϕ is the phase field parameter, and G is the chemical potential ($\mathrm{J/m^3}$)

$$G = \lambda\left[-\nabla^2\phi + \frac{\phi(\phi^2-1)}{\varepsilon^2}\right] = \frac{\lambda}{\varepsilon^2}\psi$$

As seen above, the phase field surface tension is computed as a distributed force over the interface using only ψ and the gradient of the phase field variable. This computation avoids using the surface normal and the surface curvature, which are troublesome to represent numerically.

INITIAL CONDITIONS

Initially, the reservoir is filled with water and the capillary channel is filled with air. The initial velocity is zero.

BOUNDARY CONDITIONS

Inlet

The hydrostatic pressure, $p = \rho g z$, gives the pressure at the inflow boundary. The pressure boundary condition automatically compensates for hydrostatic pressure so the actual value for the pressure is set to zero. Only water enters through the inlet, so the volume fraction of water is 1 here.

Outlet

At the outlet, the pressure is equal to zero, that is, equal to the pressure at the top of the inflow boundary. Because it is an outflow boundary, you do not have to set any condition on the level set function.

Walls

The Wetted Wall feature is suitable for solid walls in contact with a fluid interface. It sets the velocity component normal to the wall to zero; that is,

$$\mathbf{u}\cdot\mathbf{n}_{\mathrm{wall}} = 0$$

and adds a frictional boundary force

$$\mathbf{F}_{\mathrm{fr}} = -\frac{\mu}{\beta}\mathbf{u}$$

Here, β is the slip length. The boundary condition also allows you to specify the contact angle θ, that is, the angle between the wall and the fluid interface (see Figure 2). In this example, the contact angle is $67.5°$ and the slip length equals the mesh element size, h.

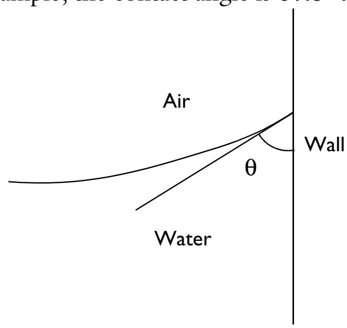


Figure 2: Definition of the contact angle θ.

Results and Discussion

The initial development of the fluid interface is shown in Figure 3. During this stage the surface changes drastically in order for it to obtain the prescribed contact angle with the wall. When this is achieved, the surface tension imposed by the surface curvature begins to pull water up through the vertical cylinder. Due to the instantaneous start, the surface oscillates slightly during the rise.

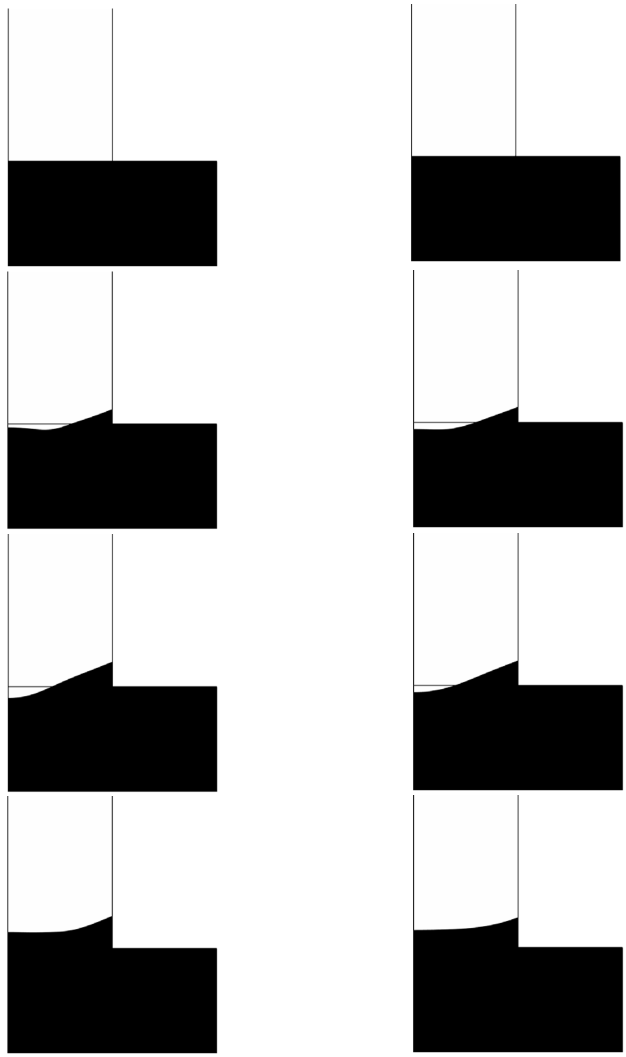


Figure 3: Snapshots of the position of the interface during the first 0.15 ms. Level Set (left) and Phase Field (right) model results.

Figure 4 shows the interface and the velocity field at three different times following the initial stage. After about 0.6 ms the shape of the water surface remains approximately constant and forms a rising concave meniscus. Comparing the velocity field in the Level Set and the Phase Field models, the Level Set results display a small velocity near the wall/

interface contact point, something that is not present in the Phase Field results. This is due to a difference in the wetted wall condition. The Level Set interface requires a wall slip length for the interface to move along the wall. As shown in Figure 4, the imposed slip velocity at the wall is small. In the Phase Field model a slip length is not necessary and the fluid velocity is truly zero on the wall.

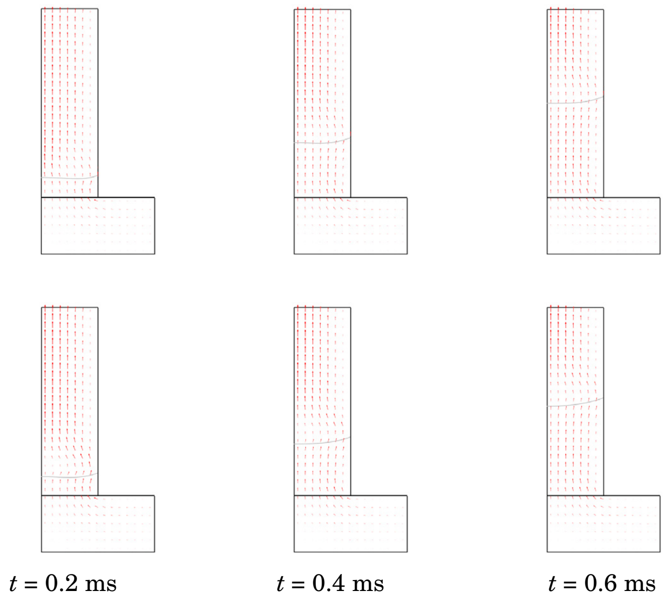


Figure 4: Interface and velocity field at different times. Level Set (top) and Phase Field (bottom) model results.

Figure 5 shows surface plots of the pressure at $t = 0.6$ ms. At the fluid interface there is a pressure jump of roughly 300 Pa. The jump is caused by the surface tension and forces the water and air to rise through the vertical cylinder.

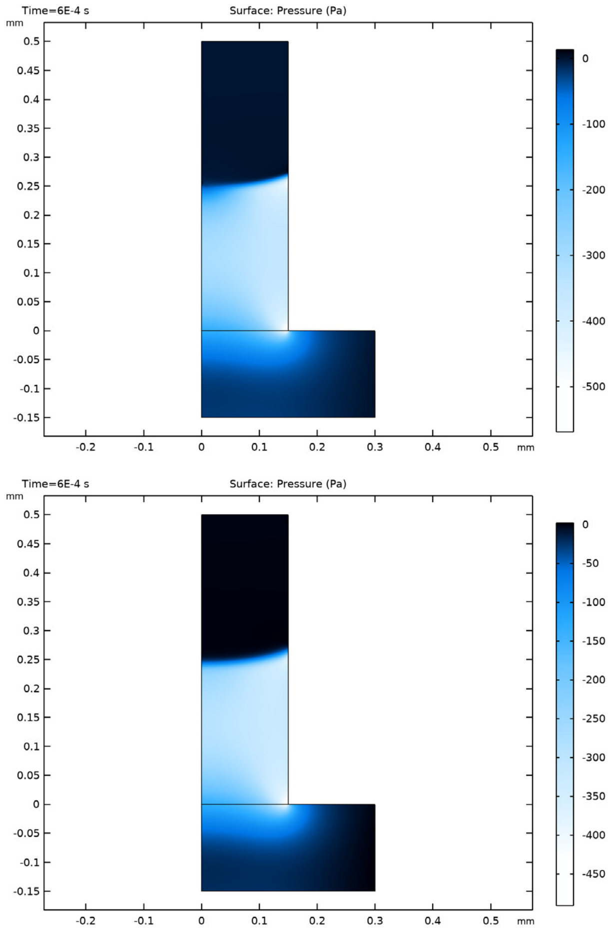


Figure 5: Pressure at t = 0.6 ms. Level Set (top) and Phase Field (bottom) model results.

You can easily calculate the position of the interface/wall contact point by integrating the level set function along the thin cylinder wall. Figure 6 shows the position of the contact point as a function of time. The slight oscillations of the water surface noted above are seen here also in the contact point plot. The contact plots from the Level Set and Phase Field models compare very well, except for two minor points. The surface oscillation is a bit more pronounced in the Level Set model, and the surface endpoint is somewhat higher up in this case. Both these differences are small and are most likely related to the different implementations of the wetted wall boundary condition.

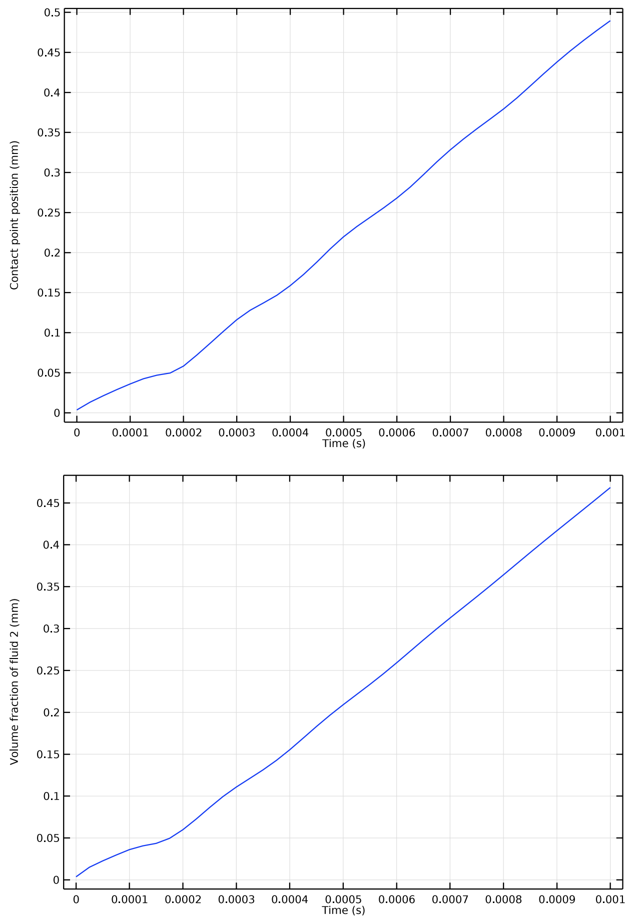


Figure 6: Position of the interface/wall contact point as a function of time. Level Set (top) and Phase Field (bottom) model result. The velocity is approximately constant after t = 0.6 ms.

Finally, you can verify the obtained contact angle. It is defined by $\cos\theta = \mathbf{n}^T\mathbf{n}_{\mathrm{wall}}$. In this case, the normal to the wall is $\mathbf{n}_{\mathrm{wall}} = \mathbf{e}_r$. The contact angle is thus $\theta = \mathrm{acos}\, n_r$, where n_r is the radial component of the interface normal. Due to the slight oscillations of the surface, the contact angle varies during the rise. As Figure 7 shows, at $t = 0.6$ ms the contact angle is approximately 69° for the Level Set model and approximately 68° for the

Phase Field model. Both results are close to the imposed contact angle of $3\pi/8 = 1.18$ rad $= 67.5°$. The contact angle further approaches the imposed value if the mesh is refined.

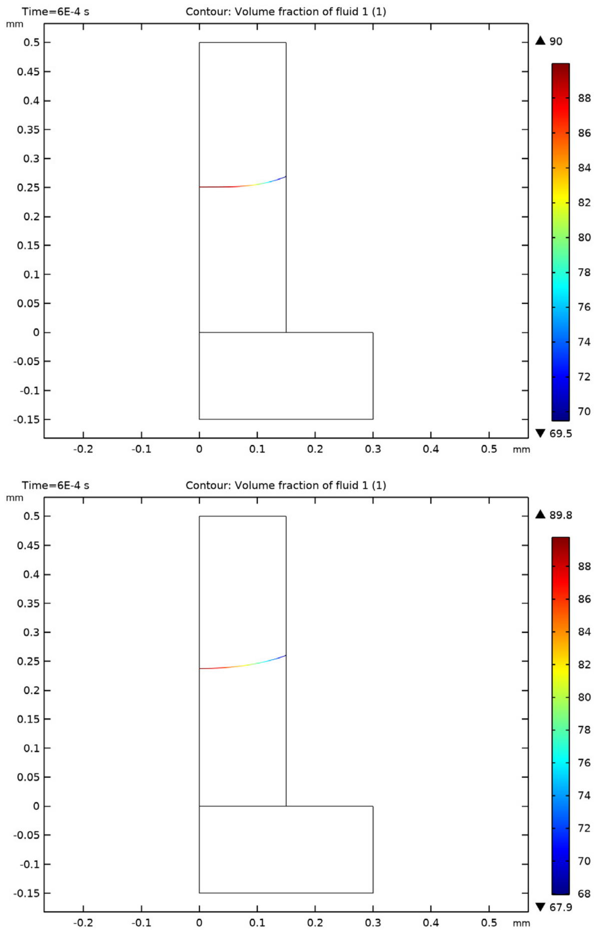


Figure 7: Plot of $acos(n_r)$*. At the wall, this gives the contact angle. In the Level Set model (top) the wall angle is approximately 69° and in the Phase Field model (bottom) it is approximately 68°.*

Notes About the COMSOL Implementation

The model is straightforward to set up using either the Level Set or the Phase Field interface. At walls in contact with the fluid interface, you can use the Wetted Wall coupling feature.

The simulation procedure consists of two steps. First the phase field and level set functions are initialized, then the time-dependent calculation starts. This is automatically set up by the software. You only need to specify appropriate times for the initialization step and the time-dependent analysis.

Application Library path: `CFD_Module/Multiphase_Flow/capillary_filling_pf`

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>Multiphase Flow>Two-Phase Flow, Phase Field>Laminar Flow**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Multiphysics> Time Dependent with Phase Initialization**.

6 Click **Done**.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

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 `0.3`.

4 In the **Height** text field, type `0.15`.

5 Locate the **Position** section. In the **z** text field, type `-0.15`.

6 Click Build Selected.

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 `0.15`.

4 In the **Height** text field, type `0.5`.

5 Click **Build Selected**.

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

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in>Air**.

4 Click **Add to Component** in the window toolbar.

5 In the tree, select **Built-in>Water, liquid**.

6 Click **Add to Component** in the window toolbar.

7 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MULTIPHYSICS

Two-Phase Flow, Phase Field 1 (tpf1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Two-Phase Flow, Phase Field 1 (tpf1)**.

2 In the **Settings** window for **Two-Phase Flow, Phase Field**, locate the **Fluid 1 Properties** section.

3 From the **Fluid 1** list, choose **Air (mat1)**.

4 Locate the **Fluid 2 Properties** section. From the **Fluid 2** list, choose **Water, liquid (mat2)**.

PHASE FIELD (PF)

Phase Field Model 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Phase Field (pf)** click **Phase Field Model 1**.
2. In the **Settings** window for **Phase Field Model**, locate the **Phase Field Parameters** section.
3. In the χ text field, type `50`.
4. In the ε_{pf} text field, type `6.5e-6`.

Initial Values, Fluid 2

1. In the **Model Builder** window, click **Initial Values, Fluid 2**.
2. Select Domain 1 only.

LAMINAR FLOW (SPF)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.
2. In the **Settings** window for **Laminar Flow**, locate the **Physical Model** section.
3. Select the **Include gravity** check box.

Inlet 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
2. Select Boundary 8 only.
3. In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.
4. From the list, choose **Pressure**.

Outlet 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.
2. Select Boundary 5 only.

PHASE FIELD (PF)

In the **Model Builder** window, under **Component 1 (comp1)** click **Phase Field (pf)**.

Inlet 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
2. Select Boundary 8 only.
3. In the **Settings** window for **Inlet**, locate the **Phase Field Condition** section.
4. From the list, choose **Fluid 2 (φ = 1)**.

Outlet 1

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

2 Select Boundary 5 only.

Wetted Wall 2

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

2 Select Boundaries 6 and 7 only.

3 In the **Settings** window for **Wetted Wall**, locate the **Wetted Wall** section.

4 In the θ_w text field, type `(3*pi/8)[rad]`.

Next, define a variable for the contact angle.

DEFINITIONS

Variables 1

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

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`intnormr`	`d(phipf,r)/sqrt(d(phipf,r)^2+d(phipf,z)^2+eps)`		`Interface normal, r component`
`theta`	`(acos(intnormr))[1/deg]`		`Contact angle`

MESH 1

Mapped 1

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

Size

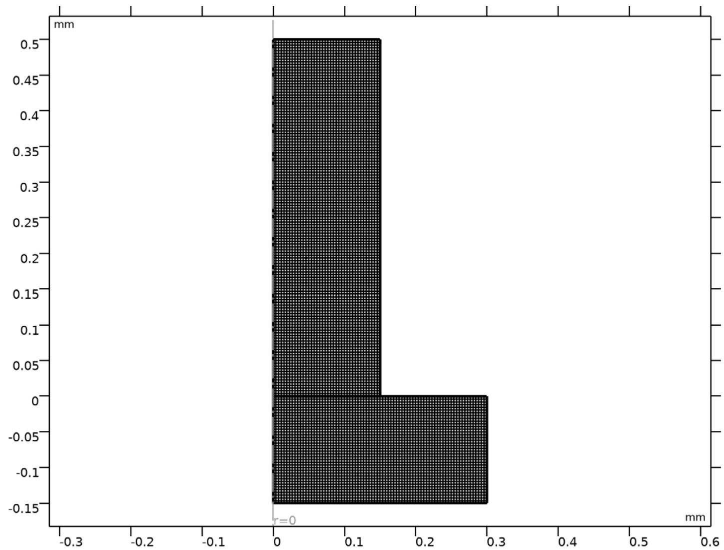
1 In the **Model Builder** window, click **Size**.

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

3 From the **Calibrate for** list, choose **Fluid dynamics**.

4 From the **Predefined** list, choose **Extra fine**.

5 Click **Build All**.



STUDY 1

Step 2: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 2: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type `range(0,0.25e-4,1e-3)`.

Use manual scaling of variables to improve time stepping efficiency.

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 2**.

3 In the **Settings** window for **Dependent Variables**, locate the **Scaling** section.

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

5 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 2** node, then click **Phase field help variable (comp1.psi)**.

6 In the **Settings** window for **Field**, locate the **Scaling** section.

7 From the **Method** list, choose **Manual**.

8 In the **Scale** text field, type `0.01`.

9 In the **Model Builder** window, click **Pressure (comp1.p)**.

10 In the **Settings** window for **Field**, locate the **Scaling** section.

11 From the **Method** list, choose **Manual**.

12 In the **Scale** text field, type `100`.

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

RESULTS

Volume Fraction of Fluid 1 (pf)

The third default plot group shows the volume fraction of air. While the position of the air/water interface appears clearly, you can obtain an even sharper interface by plotting the 0.5 level of the same quantity using a filled contour plot, as in Figure 3.

Volume Fraction of Fluid 1

1 In the **Model Builder** window, expand the **Volume Fraction of Fluid 1 (pf)** node.

2 Right-click **Volume Fraction of Fluid 1** and choose **Delete**.

Volume Fraction of Fluid 1 (pf)

1 In the **Model Builder** window, click **Volume Fraction of Fluid 1 (pf)**.

2 Click **Yes** to confirm.

Volume Fraction of Fluid 1.1

1 In the **Model Builder** window, click **Volume Fraction of Fluid 1.1**.

2 In the **Settings** window for **Contour**, locate the **Coloring and Style** section.

3 From the **Contour type** list, choose **Filled**.

4 From the **Coloring** list, choose **Color table**.

5 From the **Color table** list, choose **GrayScale**.

6 Select the **Color legend** check box.

7 From the **Legend type** list, choose **Line**.

Volume Fraction of Fluid 1 (pf)

1 In the **Model Builder** window, click **Volume Fraction of Fluid 1 (pf)**.

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

3 From the **Time (s)** list, choose **0**.

4 In the **Volume Fraction of Fluid 1 (pf)** toolbar, click **Plot**.

5 In the **Model Builder** window, click **Volume Fraction of Fluid 1 (pf)**.

6 Click the **Zoom Box** button in the **Graphics** toolbar, then zoom in on the lower part of the capillary. Compare the resulting plot with that in the upper-right panel of Figure 3.

Velocity (spf)

The first default plot shows a surface plot of the velocity magnitude of the fluids. This plot can be changed to reproduce the combined velocity-field arrows and air/water-interface plot shown in Figure 4.

Surface

1. In the **Model Builder** window, expand the **Velocity (spf)** node.
2. Right-click **Surface** and choose **Delete**.
3. Click **Yes** to confirm.

Contour 1

1. In the **Model Builder** window, right-click **Velocity (spf)** and choose **Contour**.
2. In the **Settings** window for **Contour**, locate the **Expression** section.
3. In the **Expression** text field, type `pf.Vf1`.
4. Locate the **Levels** section. From the **Entry method** list, choose **Levels**.
5. In the **Levels** text field, type `0.5`.
6. Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
7. From the **Color** list, choose **Gray**.
8. In the **Velocity (spf)** toolbar, click **Plot**.
9. Click the **Zoom Extents** button in the **Graphics** toolbar.

Arrow Surface 1

1. Right-click **Velocity (spf)** and choose **Arrow Surface**.
2. In the **Settings** window for **Arrow Surface**, locate the **Arrow Positioning** section.
3. Find the **z grid points** subsection. In the **Points** text field, type `30`.

Velocity (spf)

1. In the **Model Builder** window, click **Velocity (spf)**.
2. In the **Settings** window for **2D Plot Group**, locate the **Data** section.
3. From the **Time (s)** list, choose **2E-4**.
4. In the **Velocity (spf)** toolbar, click **Plot**.

   The resulting plot should closely resemble the upper-right plot in Figure 4.

   Generate the remaining two plots by choosing the values 4e-4 and 6e-4 from the Time list.

Volume Fraction of Fluid 1 (pf) 1

The fifth default plot group shows the air/water-interface as an isosurface plot using a revolved dataset.

Edge 2D 1

1 In the **Model Builder** window, expand the **Results>Views** node.
2 Right-click **Datasets** and choose **Edge 2D**.
3 Select Boundaries 6 and 7 only.

Revolution 2D 3

1 In the **Results** toolbar, click **More Datasets** and choose **Revolution 2D**.
2 In the **Settings** window for **Revolution 2D**, locate the **Data** section.
3 From the **Dataset** list, choose **Edge 2D 1**.
4 Click to expand the **Revolution Layers** section. In the **Start angle** text field, type `-90`.
5 In the **Revolution angle** text field, type `225`.

Edge 2D 2

1 In the **Results** toolbar, click **More Datasets** and choose **Edge 2D**.
2 Select Boundary 2 only.

Revolution 2D 4

1 In the **Results** toolbar, click **More Datasets** and choose **Revolution 2D**.
2 In the **Settings** window for **Revolution 2D**, locate the **Data** section.
3 From the **Dataset** list, choose **Edge 2D 2**.

Volume Fraction of Fluid 1 (pf) 1

1 In the **Model Builder** window, expand the **Results>Volume Fraction of Fluid 1 (pf) 1** node, then click **Volume Fraction of Fluid 1 (pf) 1**.
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3 From the **Dataset** list, choose **Revolution 2D**.
4 From the **Time (s)** list, choose **3E-4**.
5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.
6 In the **Volume Fraction of Fluid 1 (pf) 1** toolbar, click **Plot**.

Isosurface 1

1 In the **Model Builder** window, click **Isosurface 1**.
2 In the **Settings** window for **Isosurface**, locate the **Data** section.

3 From the **Dataset** list, choose **Revolution 2D 1**.

4 From the **Time (s)** list, choose **3E-4**.

5 Locate the **Coloring and Style** section. From the **Color** list, choose **White**.

Surface 1

1 In the **Model Builder** window, right-click **Volume Fraction of Fluid 1 (pf) 1** and choose **Surface**.

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

3 In the **Expression** text field, type `pf.Vf1`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Cividis**.

5 Locate the **Data** section. From the **Dataset** list, choose **Revolution 2D**.

6 From the **Time (s)** list, choose **3E-4**.

7 In the **Volume Fraction of Fluid 1 (pf) 1** toolbar, click **Plot**.

Surface 2

1 Right-click **Volume Fraction of Fluid 1 (pf) 1** and choose **Surface**.

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

3 From the **Dataset** list, choose **Revolution 2D 3**.

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

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

Surface 3

1 Right-click **Volume Fraction of Fluid 1 (pf) 1** and choose **Surface**.

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

3 From the **Dataset** list, choose **Revolution 2D 4**.

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

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

6 In the **Volume Fraction of Fluid 1 (pf) 1** toolbar, click **Plot**.

Next, plot the pressure at t = 0.6 ms. Compare the result with the upper plot in Figure 5.

2D Plot Group 6

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 (s)** list, choose **6E-4**.

Surface 1

1 Right-click **2D Plot Group 6** and choose **Surface**.

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

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

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **JupiterAuroraBorealis**.

5 Select the **Reverse color table** check box.

6 In the **2D Plot Group 6** toolbar, click **Plot**.

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

Pressure Surface (spf)

1 In the **Model Builder** window, right-click **2D Plot Group 6** and choose **Rename**.

2 In the **Rename 2D Plot Group** dialog box, type `Pressure Surface (spf)` in the **New label** text field.

3 Click **OK**.

Go on to compute and plot the position of the interface/wall contact point.

Line Integration 1

1 In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Integration>Line Integration**.

2 Select Boundary 6 only.

3 In the **Settings** window for **Line Integration**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit	Description
`pf.Vf2`	`mm`	`Volume fraction of fluid 2`

5 Locate the **Integration Settings** section. Clear the **Compute surface integral** check box.

6 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

2 Click **Table Graph** in the window toolbar.

RESULTS

Table Graph 1

Compare this graph with that in the lower panel of Figure 6.

Meniscus position

1 In the **Model Builder** window, right-click **1D Plot Group 7** and choose **Rename**.

2 In the **Rename 1D Plot Group** dialog box, type `Meniscus position` in the **New label** text field.

3 Click **OK**.

Finally, check the value of the contact angle at t = 0.6 ms (Figure 7).

2D Plot Group 8

1 In the **Results** toolbar, click **2D Plot Group**.

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

3 From the **Time (s)** list, choose **6E-4**.

Contour 1

1 Right-click **2D Plot Group 8** and choose **Contour**.

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

3 In the **Expression** text field, type `pf.Vf1`.

4 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

5 In the **Levels** text field, type `0.5`.

Color Expression 1

1 Right-click **Contour 1** and choose **Color Expression**.

2 In the **Settings** window for **Color Expression**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>theta - Contact angle**.

Meniscus angle

1 In the **Model Builder** window, click **2D Plot Group 8**.

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

3 Select the **Show maximum and minimum values** check box.

4 In the **2D Plot Group 8** toolbar, click **Plot**.

At this instance the contact angle is approximately 68 degrees which can be found by expanding the **Range** section in the **Settings Window** of the **Color Expression** node created.

5 Right-click **2D Plot Group 8** and choose **Rename**.
6 In the **Rename 2D Plot Group** dialog box, type `Meniscus angle` in the **New label** text field.
7 Click **OK**.