Two-Phase Flow with Fluid-Structure Interaction
Introduction

The following example demonstrates techniques for modeling a fluid-structure interaction containing two fluid phases in COMSOL Multiphysics. It illustrates how a heavier fluid can induce movement in an obstacle modifying the fluid flow itself. This model uses the arbitrary Lagrangian-Eulerian (ALE) technique along with a Two-Phase Flow, Phase Field predefined multiphysics interface.

The model geometry consists of a small container, in the middle of which is a thin obstacle. Initially, a heavier fluid (water) is present in the left domain and air is present everywhere else. The model is similar to a classic dam break benchmark, except the obstacle disrupts the flow of the water into the right domain. The obstacle begins to bend due to the inertial force of the heavier fluid.

Figure 1: Water sloshing in a tank with a flexible baffle.

The ALE method handles the dynamics of the deforming geometry and the moving boundaries with a moving grid. The fluid-fluid boundary is tracked using the phase-field method. On the obstacle surface, an Interior Wetted Wall boundary condition is applied, which allows a contact angle to be specified on a deforming wall. COMSOL Multiphysics computes new mesh coordinates for the channel area based on the movement of the
structure’s boundaries and on mesh smoothing. Because of the small thickness of the obstacle, you can use shell elements to avoid adapting the mesh to the actual thickness.

**Model Definition**

Initially, the heavier fluid forms a dam. The container is 30 mm long, 5 mm wide, and 10 mm tall. The top of the soft obstacle is fixed to the container and it hangs freely inside the container. The obstacle is 9 mm in height, 4 mm wide, and 0.3 mm thick. An initial barrier of water is released, and when it reaches the obstacle, it pushes it away from its original position. The displaced air naturally feeds into the channel on the opposite side of the obstacle. If the obstacle would have been as wide as the channel, the water would not be able to penetrate into the right domain. In the real world, this effect is observed when pouring milk or orange juice from a container. The liquid tends to exit the container in a periodic motion. If the carton is pierced so that displaced air can reenter, a very smooth pour results. The two fluids are air and water, whose physical properties are defined at room temperature. The wetted wall contact angle is $\pi/2$ rad. The Young’s modulus of the obstacle is set to 2 MPa.

*Figure 2: Initial fluid density inside the container. The soft obstacle hangs down from the container and is free to move.*
Results and Discussion

Figure 3 shows the deformation of the nylon obstacle at $t = 0.08$ s. The water (indicated by the blue color) flows from left to right, and the return passage allows the displaced air to recirculate naturally between the two sides of the obstacle. After the initial release of the heavier fluid, the obstacle begins to relax back toward its original position and the liquid level starts to distribute evenly on both sides of the obstacle.

Figure 3: Plot of fluid density and deformed geometry at 0.08 s.

Figure 4 shows how the water has filled the box at the final computational time. The obstacle has nearly returned to its original position.

Figure 4: Plot of fluid density and deformed geometry at 0.08 s.
Figure 4: Plot of density at final solution time.

Figure 5 shows the solution at selected times during the computation.
TWO-PHASE FLOW WITH FLUID-STRUCTURE INTERACTION
Figure 5: Fluid propagation in the box and obstacle deformation (from right to left, top to bottom).

Figure 6 shows the structural displacement of the obstacle during the simulation period.

Figure 6: Horizontal (blue) and vertical (green) displacements of the obstacle.

Figure 7 shows the variation of the water volume. Note that the mass is well conserved.
Figure 7: Variation of the volume fraction of water during the analysis.

Modeling in COMSOL Multiphysics

This example implements the model using two predefined multiphysics coupling interfaces: Two-Phase Flow, Phase Field and Fluid-Shell Interaction. These multiphysics couplings add to the model a Laminar Flow interface, which computes the fluid’s velocity and pressure; a Phase Field interface, which computes the volume fraction of the two phases; and a Shell interface, which computes the structural displacement of the obstacle.

The fluid-domain deformations are obtained by computing a moving mesh that follows the structural deformation. The computational requirement is reduced by only computing the deformation of the mesh around the obstacle; in the rest of the fluid domain the mesh is considered to be rigid. The Fluid-Structure Interaction multiphysics node is fully coupled, the obstacle acts on the fluid as a moving wall, and the fluid applies face load on the structural part, see Figure 8.
Figure 8: Face load applied on the obstacle by the water

At walls in contact with the fluid interface, you can use the Wetted Wall condition. At the obstacle, whose geometry is not represented with a thickness, you can use the Interior Wetted Wall condition.

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

Application Library path: CFD_Module/Fluid-Structure_Interaction/twophase_flow_fsi

Modeling Instructions

From the File menu, choose New.
NEW
In the New window, click \(\text{Model Wizard}\).

MODEL WIZARD
1 In the Model Wizard window, click \(\text{3D}\).
2 In the Select Physics tree, select \(\text{Fluid Flow}>\text{Multiphase Flow}>\text{Two-Phase Flow, Phase Field}>\text{Laminar Flow}\).
3 Click Add.
4 In the Select Physics tree, select \(\text{Fluid Flow}>\text{Fluid-Structure Interaction}>\text{Fluid-Shell Interaction}\).
5 Click Add.
6 In the Added physics interfaces tree, select \(\text{Laminar Flow (spf2)}\).
7 Right-click and choose Remove.
8 Click \(\text{Study}\).
9 In the Select Study tree, select \(\text{Preset Studies for Selected Multiphysics}>\text{Time Dependent with Phase Initialization}\).
10 Click \(\text{Done}\).

GLOBAL DEFINITIONS

Parameters 1
1 In the Model Builder window, under Global Definitions click Parameters 1.
2 In the Settings window for Parameters, locate the Parameters section.
3 In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Name</th>
<th>Expression</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hb</td>
<td>10[mm]</td>
<td>0.01 m</td>
<td>Box height</td>
</tr>
<tr>
<td>Wb</td>
<td>5[mm]</td>
<td>0.005 m</td>
<td>Box width</td>
</tr>
<tr>
<td>Lb</td>
<td>30[mm]</td>
<td>0.03 m</td>
<td>Box length</td>
</tr>
<tr>
<td>Xo</td>
<td>15[mm]</td>
<td>0.015 m</td>
<td>Position of obstacle</td>
</tr>
<tr>
<td>do</td>
<td>0.3[mm]</td>
<td>3E-4 m</td>
<td>Obstacle thickness</td>
</tr>
<tr>
<td>Ho</td>
<td>9[mm]</td>
<td>0.009 m</td>
<td>Obstacle height</td>
</tr>
<tr>
<td>Wo</td>
<td>4[mm]</td>
<td>0.004 m</td>
<td>Obstacle width</td>
</tr>
<tr>
<td>Xi</td>
<td>10[mm]</td>
<td>0.01 m</td>
<td>Initial position of the interface</td>
</tr>
</tbody>
</table>
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.

Block 1 (blk1)
1 In the Geometry toolbar, click Block.
2 In the Settings window for Block, locate the Size and Shape section.
3 In the Width text field, type Lb.
4 In the Depth text field, type Wb.
5 In the Height text field, type Hb.
6 Click to expand the Layers section. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Layer name</th>
<th>Thickness (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Layer 1</td>
<td>Xi</td>
</tr>
<tr>
<td>Layer 2</td>
<td>Xi</td>
</tr>
</tbody>
</table>

7 Find the Layer position subsection. Select the Left check box.
8 Clear the Bottom check box.

Obstacle
1 In the Geometry toolbar, click Work Plane.
2 In the Settings window for Work Plane, type Obstacle in the Label text field.
3 Locate the Plane Definition section. From the Plane list, choose yz-plane.
4 In the x-coordinate text field, type Xo.
5 Locate the Selections of Resulting Entities section. Select the Resulting objects selection check box.

Obstacle (wp1)>Plane Geometry
In the Model Builder window, click Plane Geometry.

Obstacle (wp1)>Rectangle 1 (r1)
1 In the Work Plane toolbar, click Rectangle.
2 In the Settings window for Rectangle, locate the Size and Shape section.
3 In the Width text field, type Wo.
4 In the Height text field, type Ho.
5 Locate the Position section. In the yw text field, type Hb-Ho.
Obstacle (wp1)>Fillet 1 (fil1)
1 In the Work Plane toolbar, click Fillet.
2 On the object r1, select Point 2 only.
3 In the Settings window for Fillet, locate the Radius section.
4 In the Radius text field, type Wo/2.

Definitions

Symmetry
1 In the Definitions toolbar, click Explicit.
2 In the Settings window for Explicit, locate the Input Entities section.
3 From the Geometric entity level list, choose Boundary.
4 Select Boundaries 2, 7, and 13 only.
5 In the Label text field, type Symmetry.

Wall
1 In the Definitions toolbar, click Explicit.
2 In the Settings window for Explicit, locate the Input Entities section.
3 From the Geometric entity level list, choose Boundary.
4 Select Boundaries 1, 3–5, 8–10, and 14–17 only.
5 In the Label text field, type Wall.

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.
4 Specify the \( \mathbf{r}_{\text{ref}} \) vector as

<table>
<thead>
<tr>
<th>( x )</th>
<th>( y )</th>
<th>( z )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>( H_b )</td>
</tr>
</tbody>
</table>

Initial Values 1
1 In the Model Builder window, under Component 1 (comp1)>Laminar Flow (spf) click Initial Values 1.
2 In the Settings window for Initial Values, locate the Initial Values section.
3 In the \( p \) text field, type \( \text{spf.rho} \cdot \text{g\_const} \cdot (H_b - z) \).

4 Clear the **Compensate for hydrostatic pressure** check box.

**Symmetry** 
1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Symmetry**.

**Interior Wall** 
1 In the **Physics** toolbar, click **Boundaries** and choose **Interior Wall**.
2 In the **Settings** window for **Interior Wall**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Obstacle**.

**Wall** 
1 In the **Physics** toolbar, click **Boundaries** and choose **Wall**.
2 Select Boundaries 8 and 10 only.
3 In the **Settings** window for **Wall**, click to expand the **Wall Movement** section.
4 From the **Translational velocity** list, choose **Zero (Fixed wall)**.

**Pressure Point Constraint** 
1 In the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.
2 Select Point 20 only.
3 In the **Settings** window for **Pressure Point Constraint**, locate the **Pressure Constraint** section.
4 Clear the **Compensate for hydrostatic pressure** check box.

**Phase Field (PF)**

**Phase Field Model** 
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 \( \chi \) text field, type 5.

**Initial Values, Fluid 2** 
1 In the **Model Builder** window, click **Initial Values, Fluid 2**.
2 Select Domains 2 and 3 only.
Symmetry 1
1 In the Physics toolbar, click Boundaries and choose Symmetry.
2 In the Settings window for Symmetry, locate the Boundary Selection section.
3 From the Selection list, choose Symmetry.

Interior Wetted Wall 1
1 In the Physics toolbar, click Boundaries and choose Interior Wetted Wall.
2 In the Settings window for Interior Wetted Wall, locate the Boundary Selection section.
3 From the Selection list, choose Obstacle.

SHELL (SHELL)
1 In the Model Builder window, under Component 1 (comp1) click Shell (shell).
2 In the Settings window for Shell, locate the Boundary Selection section.
3 From the Selection list, choose Obstacle.

Thickness and Offset 1
1 In the Model Builder window, under Component 1 (comp1)>Shell (shell) click Thickness and Offset 1.
2 In the Settings window for Thickness and Offset, locate the Thickness and Offset section.
3 In the \( d_0 \) text field, type \( d_0 \).

Fixed Constraint 1
1 In the Physics toolbar, click Edges and choose Fixed Constraint.
2 Select Edge 19 only.

Symmetry 1
1 In the Physics toolbar, click Edges and choose Symmetry.
2 Select Edge 17 only.

Now, create materials because they will be required for the settings of the Two-Phase Flow, Phase Field node.

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 Right-click and choose Add to Component 1 (comp1).
5 In the tree, select Built-in>Water, liquid.
6 Right-click and choose Add to Component 1 (comp1).
7 In the Home toolbar, click Add Material to close the Add Material window.

MATERIALS

Nylon
1 In the Materials toolbar, click Blank Material.
2 In the Settings window for Material, type Nylon in the Label text field.
3 Locate the Geometric Entity Selection section. From the Geometric entity level list, choose Boundary.
4 From the Selection list, choose Obstacle.
5 Locate the Material Contents section. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Property</th>
<th>Variable</th>
<th>Value</th>
<th>Unit</th>
<th>Property group</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young's modulus</td>
<td>$E$</td>
<td>2[MPa]</td>
<td>Pa</td>
<td>Young's modulus and Poisson's ratio</td>
</tr>
<tr>
<td>Poisson's ratio</td>
<td>$\nu$</td>
<td>0.4</td>
<td>1</td>
<td>Young's modulus and Poisson's ratio</td>
</tr>
<tr>
<td>Density</td>
<td>$\rho$</td>
<td>1000</td>
<td>kg/m³</td>
<td>Basic</td>
</tr>
</tbody>
</table>

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 Water, liquid (mat2).
4 Locate the Fluid 2 Properties section. From the Fluid 2 list, choose Air (mat1).

MESH 1
1 In the Model Builder window, under Component 1 (comp1) click Mesh 1.
2 In the Settings window for Mesh, locate the Physics-Controlled Mesh section.
3 From the Element size list, choose Coarse.
4 Click Build All.
**STUDY 1**

*Step 1: Phase Initialization*

The **Phase Initialization** study step will compute initial values for the phase field variables.

1. In the **Model Builder** window, under **Study 1** click **Step 1: Phase Initialization**.

2. In the **Settings** window for **Phase Initialization**, locate the **Physics and Variables Selection** section.

3. In the table, clear the **Solve for** check boxes for **Shell (shell)** and **Moving mesh (Component 1)**.

4. In the table, clear the **Solve for** check boxes for **Two-Phase Flow, Phase Field 1 (tpf1)** and **Fluid-Structure Interaction 1 (fsi1)**.

**MOVING MESH**

*Deforming Domain 1*

1. In the **Model Builder** window, under **Component 1 (comp1)>Moving Mesh** click **Deforming Domain 1**.

2. Select Domain 2 only.

**COMPONENT 1 (COMPI)**

*Symmetry/Roller 1*

1. In the **Definitions** toolbar, click **Moving Mesh** and choose **Boundaries>Symmetry/Roller**.

2. Select Boundaries 7, 8, and 10 only.

**STUDY 1**

In the **Study** toolbar, click $U_{t=0}$ **Get Initial Value**.

**RESULTS**

*Volume Fraction of Fluid 1 (pf)*

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

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

3. Clear the **Plot dataset edges** check box.

*Slice 1*

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

2. Right-click **Results>Volume Fraction of Fluid 1 (pf)>Slice 1** and choose **Delete**.
Isosurface 1
1. In the Model Builder window, under Results>Volume Fraction of Fluid 1 (\(pf\)) click Isosurface 1.
2. In the Settings window for Isosurface, locate the Coloring and Style section.
3. From the Color list, choose Custom.
4. On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the Color button.
5. Click Define custom colors.
6. Set the RGB values to 54, 140, and 203, respectively.
7. Click Add to custom colors.
8. Click Show color palette only or OK on the cross-platform desktop.

Surface 1
1. In the Model Builder window, right-click Volume Fraction of Fluid 1 (\(pf\)) and choose Surface.
2. In the Settings window for Surface, locate the Expression section.
3. In the Expression text field, type 1.
4. Click to expand the Title section. From the Title type list, choose None.
5. Locate the Coloring and Style section. From the Coloring list, choose Uniform.
6. From the Color list, choose Gray.

Selection 1
1. Right-click Surface 1 and choose Selection.
2. In the Settings window for Selection, locate the Selection section.
3. From the Selection list, choose Wall.

Surface 2
1. In the Model Builder window, right-click Volume Fraction of Fluid 1 (\(pf\)) 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 Title section. From the Title type list, choose None.
5. Click to expand the Range section. Select the Manual color range check box.
6. In the Minimum text field, type 0.5.
7. In the Maximum text field, type 0.5.
8 Select the Manual data range check box.

9 In the Minimum text field, type 0.5.

10 In the Maximum text field, type 1.

11 Click to expand the Quality section. From the Smoothing list, choose None.

12 Click to expand the Inherit Style section. From the Plot list, choose Isosurface 1.

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

2 Select Boundaries 2, 7, and 13 only.

Surface 3
1 In the Model Builder window, right-click Volume Fraction of Fluid 1 (pf) 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 (comp1)>Shell>
Displacement>shell.disp - Displacement magnitude - m.

3 Locate the Coloring and Style section. From the Coloring list, choose Gradient.

4 From the Top color list, choose Red.

5 From the Bottom color list, choose Gray.

6 Locate the Range section. Select the Manual color range check box.

7 In the Maximum text field, type 3.

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(0,5e-3,0.5).

4 From the Tolerance list, choose User controlled.

5 In the Relative tolerance text field, type 0.01.

6 Click to expand the Results While Solving section. Select the Plot check box.

7 From the Plot group list, choose Volume Fraction of Fluid 1 (pf).

Solver Configurations
In the Model Builder window, expand the Study 1>Solver Configurations node.
Solution 1 (sol1)

1. In the Model Builder window, expand the Study 1>Solver Configurations>
   Solution 1 (sol1)>Dependent Variables 2 node, then click Pressure (comp1.p).

2. In the Settings window for Field, locate the Scaling section.

3. From the Method list, choose Manual.

4. In the Scale text field, type 100.

5. In the Model Builder window, click Spatial mesh displacement (comp1.spatial.disp).

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

7. In the Scale text field, type 3e-3.

8. In the Model Builder window, click Velocity field (spatial frame) (comp1.u).

9. In the Settings window for Field, locate the Scaling section.

10. From the Method list, choose Manual.

11. In the Scale text field, type 1.

12. In the Model Builder window, click Displacement field (comp1.u_shell).

13. In the Settings window for Field, locate the Scaling section.

14. In the Scale text field, type 3e-3.

15. In the Model Builder window, expand the Study 1>Solver Configurations>
    Solution 1 (sol1)>Time-Dependent Solver 1>Segregated 1 node, then click Displacement field.

16. In the Settings window for Segregated Step, click to expand the Method and Termination section.

17. From the Jacobian update list, choose On every iteration.

18. In the Study toolbar, click Compute.

RESULTS

Surface

1. In the Model Builder window, expand the Pressure (spf) node, then click Surface.

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

3. From the Solution parameters list, choose From parent.

Surface Slit 1

1. In the Model Builder window, click Surface Slit 1.

2. In the Settings window for Surface Slit, locate the Data section.
3 From the Solution parameters list, choose From parent.

Pressure (spf)
1 In the Model Builder window, click Pressure (spf).
2 In the Settings window for 3D Plot Group, locate the Data section.
3 From the Time (s) list, choose 0.08.
4 In the Pressure (spf) toolbar, click Plot.

Stress (shell)
1 In the Model Builder window, click Stress (shell).
2 In the Settings window for 3D Plot Group, locate the Plot Settings section.
3 From the Frame list, choose Spatial (x, y, z).
4 Locate the Data section. From the Time (s) list, choose 0.08.
5 In the Stress (shell) toolbar, click Plot.
6 In the Home toolbar, click Add Predefined Plot.

ADD PREDEFINED PLOT
1 Go to the Add Predefined Plot window.
2 In the tree, select Study 1/Solution 1 (sol1)>Shell>Applied Loads (shell)>Face Loads (shell).
3 Click Add Plot in the window toolbar.
4 In the Home toolbar, click Add Predefined Plot.

RESULTS

Face Loads (shell)
1 In the Model Builder window, under Results click Face Loads (shell).
2 In the Settings window for 3D Plot Group, locate the Data section.
3 From the Time (s) list, choose 0.08.
4 In the Face Loads (shell) toolbar, click Plot.

Obstacle Displacement
1 In the Home toolbar, click Add Plot Group and choose 1D Plot Group.
2 In the Settings window for 1D Plot Group, type Obstacle Displacement in the Label text field.
3 Click to expand the Title section. From the Title type list, choose None.
4 Locate the Plot Settings section.
Select the **y-axis label** check box. In the associated text field, type **Displacement (mm)**.

**Point Graph 1**
1. Right-click **Obstacle Displacement** and choose **Point Graph**.
2. Select Point 9 only.
3. In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
4. In the **Expression** text field, type `u_shell`.
5. Click to expand the **Coloring and Style** section. From the **Width** list, choose `2`.
6. Click to expand the **Legends** section. Select the **Show legends** check box.
7. From the **Legends** list, choose **Manual**.
8. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Legends</th>
<th>X-component</th>
</tr>
</thead>
</table>

**Point Graph 2**
1. Right-click **Point Graph 1** and choose **Duplicate**.
2. In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
3. In the **Expression** text field, type `w_shell`.
4. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.
5. Locate the **Legends** section. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Legends</th>
<th>Z-component</th>
</tr>
</thead>
</table>

6. In the **Obstacle Displacement** toolbar, click `Plot`.

**Volume Integration 1**
1. In the **Results** toolbar, click `More Derived Values` and choose **Integration> Volume Integration**.
2. Click in the **Graphics** window and then press Ctrl+A to select all domains.
3. In the **Settings** window for **Volume Integration**, locate the **Expressions** section.
4. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Expression</th>
<th>Unit</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>phipf</td>
<td>1 E-12</td>
<td>Phase field variable</td>
</tr>
</tbody>
</table>
Click Evaluate.

**TABLE**
1. Go to the Table window.
2. Click Table Graph in the window toolbar.

**RESULTS**

*Water Volume*
1. In the Model Builder window, under Results click 1D Plot Group 7.
2. In the Settings window for 1D Plot Group, type Water Volume in the Label text field.

*Table Graph 1*
1. In the Model Builder window, click Table Graph 1.
2. In the Settings window for Table Graph, locate the Coloring and Style section.
3. From the Width list, choose 2.

*Water Volume*
1. In the Model Builder window, click Water Volume.
2. In the Settings window for 1D Plot Group, locate the Plot Settings section.
3. Select the y-axis label check box. In the associated text field, type Water volume (l).
4. In the Water Volume toolbar, click Plot.