Created in COMSOL Multiphysics 6.1

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.

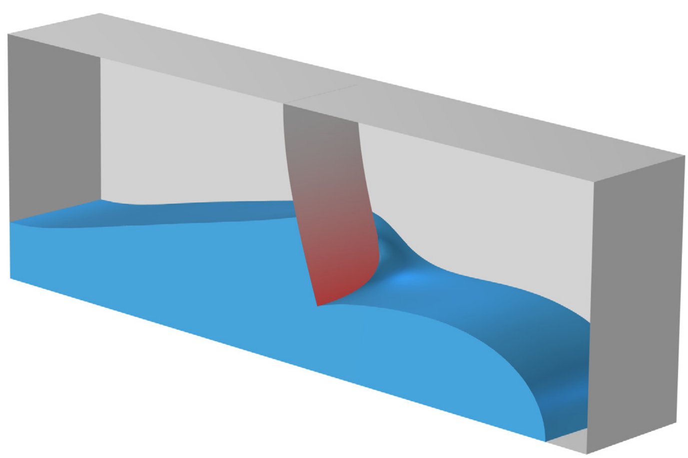


Figure 1: Water sloshing in a tank with a flexible baffle.

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.

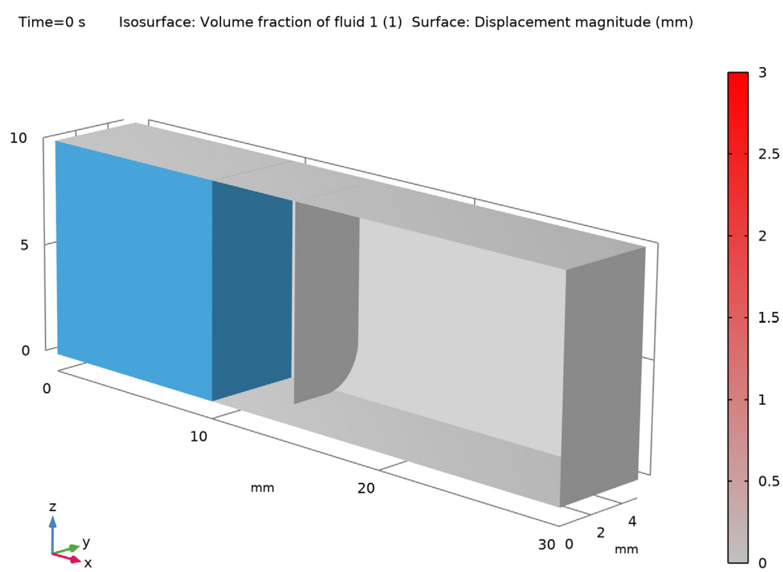


Figure 2: Initial fluid density inside the container. The soft obstacle hangs down from the container and is free to move.

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.

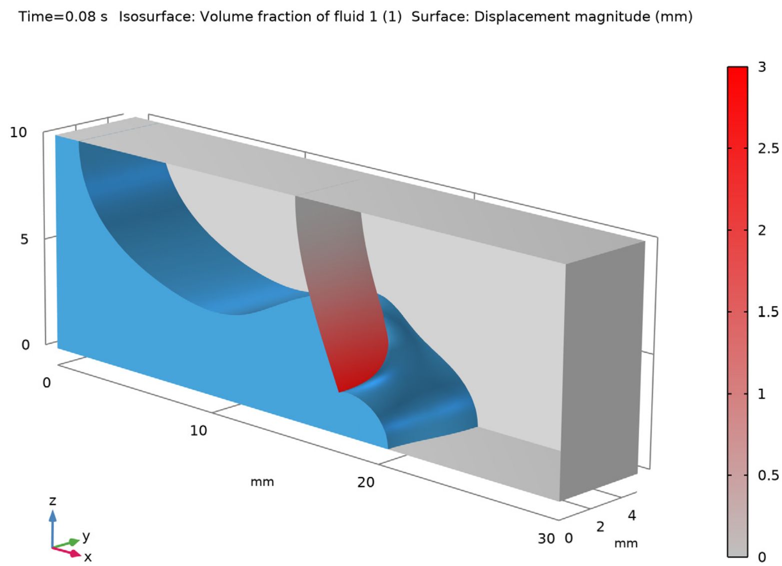


Figure 3: Plot of fluid density and deformed geometry at 0.08 s.

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

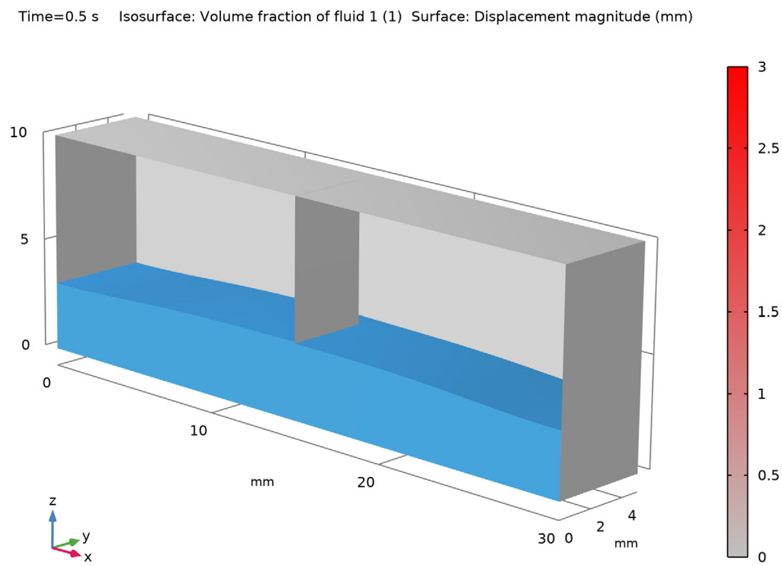
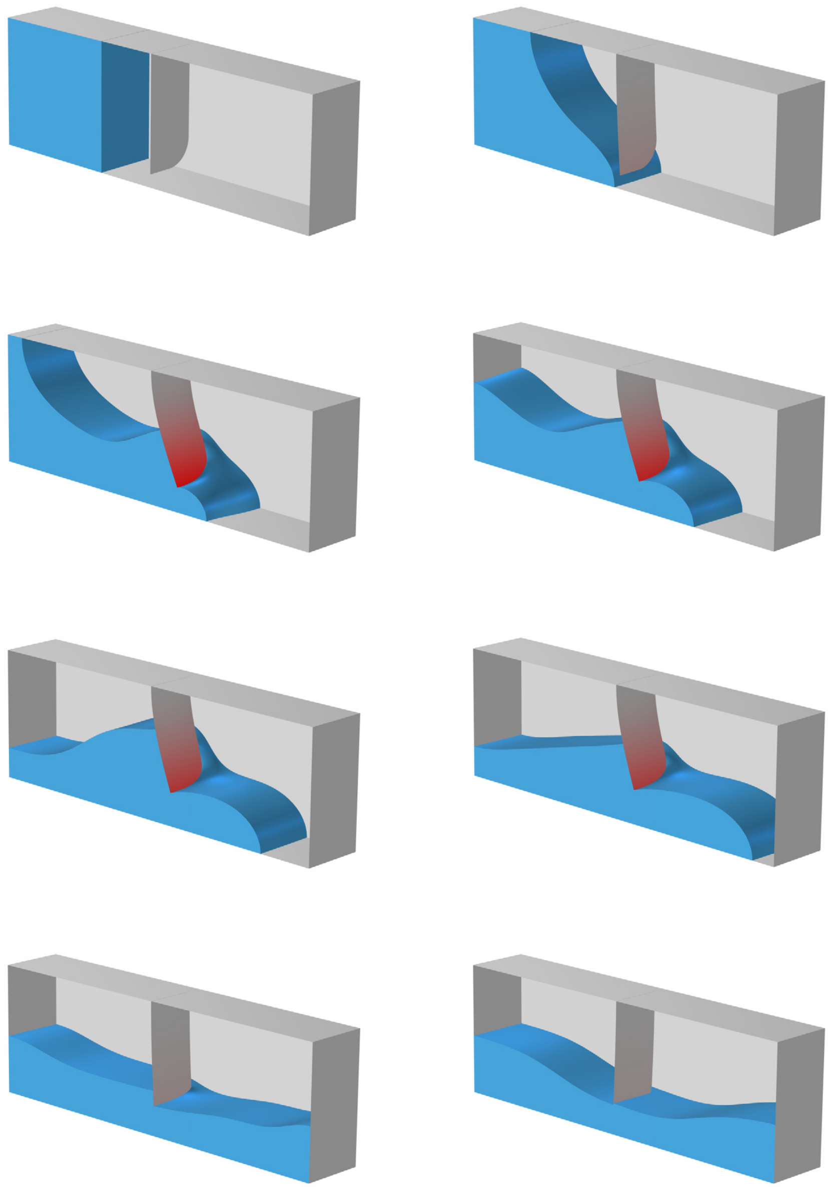


Figure 4: Plot of density at final solution time.

Figure 5 shows the solution at selected times during the computation.



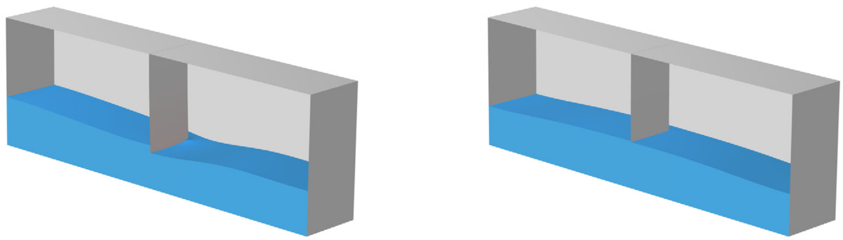


Figure 5: Fluid propagation in the box and obstacle deformation (from right to left, top to bottom).

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

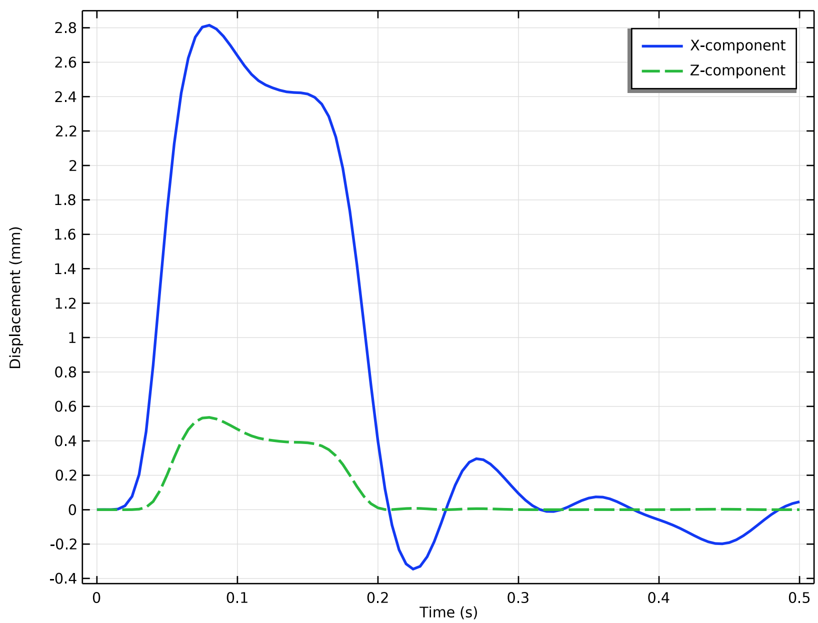


Figure 6: Horizontal (blue) and vertical (green) displacements of the obstacle.

Figure 7 shows the variation of the water volume. Note that the mass is well conserved.

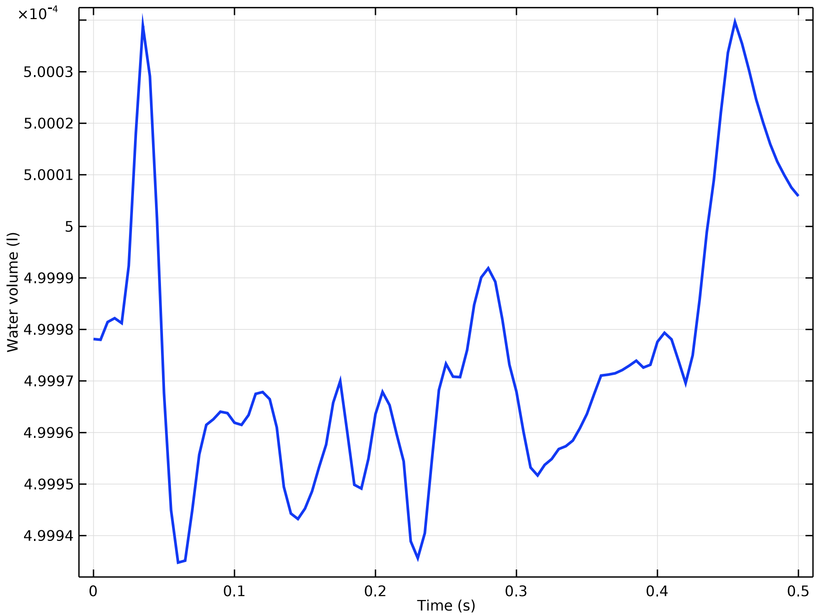


Figure 7: Variation of the volume fraction of water during the analysis.

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.

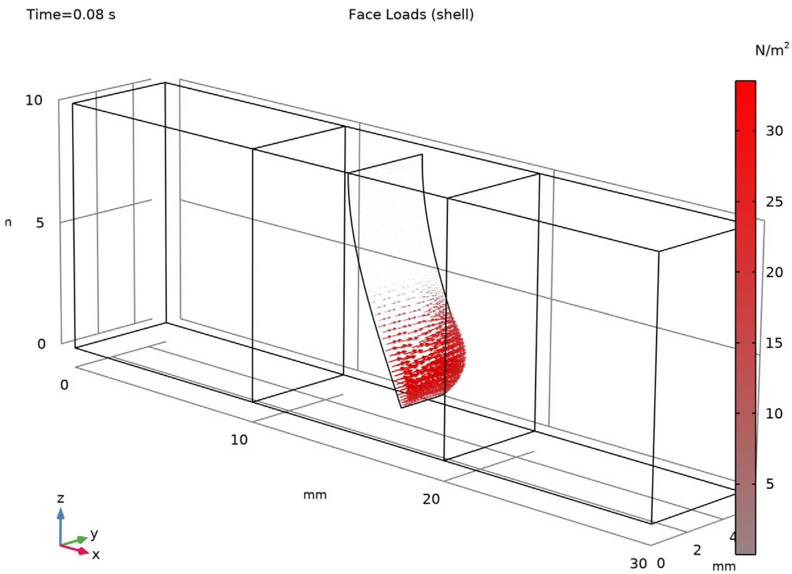


Figure 8: Face load applied on the obstacle by the water

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 **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Fluid Flow>Multiphase Flow>Two-Phase Flow, Phase Field>Laminar Flow**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Fluid Flow>Fluid-Structure Interaction>Fluid-Shell Interaction**.

5 Click **Add**.

6 In the **Added physics interfaces** tree, select **Laminar Flow (spf2)**.

7 Right-click and choose **Remove**.

8 Click **Study**.

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

10 Click **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:

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

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:

Layer name	Thickness (mm)
`Layer 1`	`Xi`
`Layer 2`	`Xi`

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 $\mathbf{r}_{\mathrm{ref}}$ vector as

0	x
0	y
Hb	z

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 `spf.rho*g_const*(Hb-z)`.

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

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 Wall 1

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 2

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

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

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 `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 `do`.

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:

Property	Variable	Value	Unit	Property group
Young's modulus	E	2[MPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.4	1	Young's modulus and Poisson's ratio
Density	rho	1000	kg/m³	Basic

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 (COMP1)

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 **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.

5 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:

Legends
`X-component`

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:

Legends
`Z-component`

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

Volume Integration 1

1 In the **Results** toolbar, click 8.85 e-12 **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:

Expression	**Unit**	**Description**
`phipf`	`1`	`Phase field variable`

5 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**.