Fluid–Structure Interaction

The following example demonstrates techniques for modeling fluid–structure interactions in COMSOL Multiphysics. It illustrates how fluid flow can deform structures and how to solve for the flow in a continuously deforming geometry using the arbitrary Lagrangian–Eulerian (ALE) technique.

The model geometry consists of a horizontal flow channel in the middle of which is an obstacle, a narrow vertical structure (Figure 1). The fluid flows from left to right, except where the obstacle forces it into a narrow path in the upper part of the channel, and it imposes a force on the structure's walls resulting from the viscous drag and fluid pressure. The structure, being made of a deformable material, bends under the applied load. Consequently, the fluid flow also follows a new path, so solving the flow in the original geometry would generate incorrect results.

The ALE method handles the dynamics of the deforming geometry and the moving boundaries with a moving grid. COMSOL Multiphysics computes new mesh coordinates on the channel area based on the movement of the structure's boundaries and mesh smoothing. The Navier–Stokes equations that solve the flow are formulated for these moving coordinates.

The structural mechanics portion of the model does not require the ALE method, and COMSOL Multiphysics solves it in a fixed coordinate system as usual. However, the strains the model computes in this way are the only source for computing the deformed coordinates with ALE.
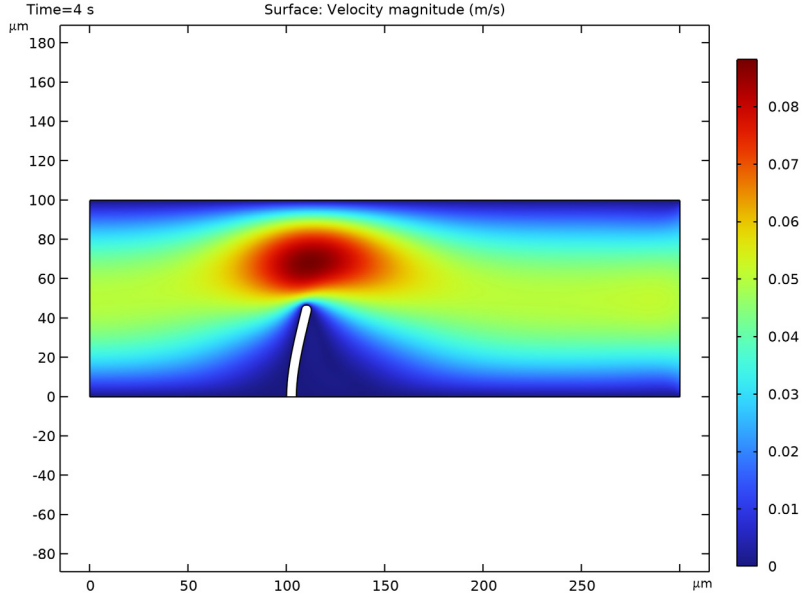
Figure 1: Fluid flows into this horizontal flow channel from the left, and it enters with a parabolic velocity profile. A narrow vertical structure in the channel (the straight vertical structure) forces the flow into a narrow path. Due to fluid pressure and viscous drag, the originally vertical structure bends. This simulation models the fluid flow in a deformed, moving mesh that follows the movement of the bending structure.

Model Definition

In this example the flow channel is 100 μm high and 300 μm long. The vertical structure — 5 μm wide, 50 μm high, and with a semicircular top — sits 100 μm away from the channel's left boundary. Assume that the structure is long in the direction perpendicular to the image.

The fluid is a water-like substance with a density $\rho = 1000$ kg/m^3 and dynamic viscosity $\eta = 0.001$ Pa·s. To demonstrate the desired techniques, assume the structure consists of a flexible material with a density $\rho = 7850$ kg/m^3 and Young's modulus $E = 200$ kPa.

FLUID FLOW

The fluid flow in the channel is described by the incompressible Navier–Stokes equations for the velocity field, $\mathbf{u} = (u, v)$, and the pressure, p, in the spatial (deformed) moving coordinate system:

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

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

In these equations, $\mathbf{I}$ denotes the unit diagonal matrix and $\mathbf{F}$ is the volume force affecting the fluid. Assume that no gravitation or other volume forces affect the fluid, so that $\mathbf{F} = 0$. The coordinate system velocity is $\mathbf{u}_m = (u_m, v_m)$.

At the channel entrance on the left, the flow has fully developed laminar characteristics with a parabolic velocity profile but its amplitude changes with time. At first flow increases rapidly, reaching its peak value at 0.215 s; thereafter it gradually decreases to a steady-state value of 5 cm/s. The centerline velocity in the x direction, u_{in} (see Figure 4), with the steady-state amplitude U comes from the equation

$$u_{in} = \frac{U \cdot t^2}{\sqrt{(0.04 - t^2)^2 + (0.1t)^2}}$$

where t must be expressed in seconds.

At the outflow (right-hand boundary), the condition is $p = 0$. On the solid (nondeforming) walls, no slip conditions are imposed, $u = 0$, $v = 0$, while on the deforming interface the velocities equal the deformation rate, $u_0 = u_t$ and $v_0 = v_t$ (the default condition; note that u and v on the right-hand sides refer to the displacement components).

STRUCTURAL MECHANICS

The structural deformations are solved for using an elastic formulation and a nonlinear geometry formulation to allow large deformations.

The obstacle is fixed to the bottom of the fluid channel. All other object boundaries experience a load from the fluid, given by

$$\mathbf{F}_T = -\mathbf{n} \cdot (-p\mathbf{I} + \eta(\nabla\mathbf{u} + (\nabla\mathbf{u})^T))$$

where $\mathbf{n}$ is the normal vector to the boundary. This load represents a sum of pressure and viscous forces.

MOVING MESH

The Navier–Stokes equations are solved on a freely moving deformed mesh, which constitutes the fluid domain. The deformation of this mesh relative to the initial shape of the domain is computed using Yeoh smoothing. Inside the obstacle, the moving mesh

follows the deformations of the obstacle. At the exterior boundaries of the flow domain, the deformation is zero in all directions.

Results and Discussion

Figure 2 shows the geometry deformation and flow at $t = 4$ s when the system is close to its steady state. Due to the channel's small dimensions, the Reynolds number of the flow is small ($Re \ll 100$), and the flow stays laminar in most of the area. The swirls are restricted to a small area behind the structure. The amount of deformation as well as the size and location of the swirls depend on the magnitude of the inflow velocity.

Time=4 s   Surface: Velocity magnitude (m/s) Streamline: Velocity field (spatial frame)
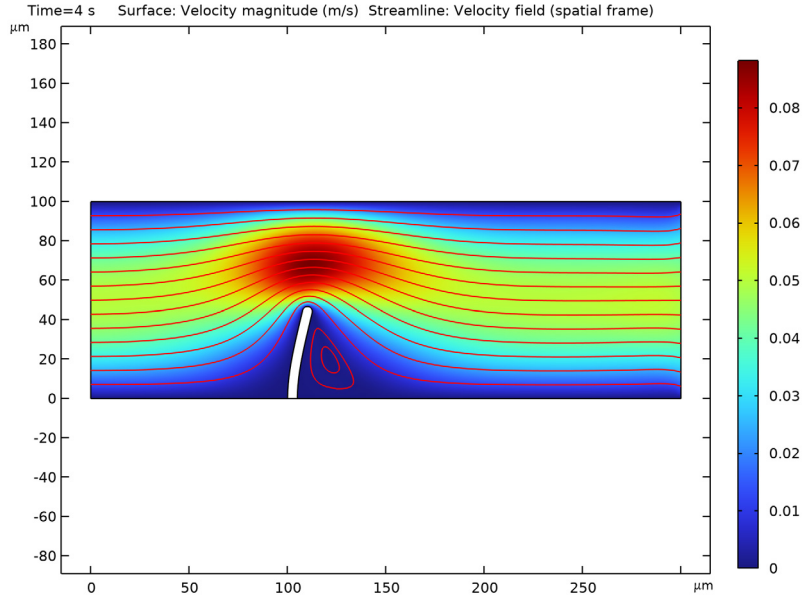
Figure 2: Flow velocity and geometry deformation at t = 4 s. The streamlines indicate the flow direction and the color indicates flow-velocity magnitude.

Figure 3 shows the mesh velocity at $t = 0.15$ s. The boundaries of the narrow structure are the only moving boundaries of the flow channel. Therefore the mesh velocity also has its largest values near the structure. Depending on the current state of the deformation — whether it is increasing, decreasing or stationary — the mesh velocity can have a very different distribution. Figure 4 further illustrates this point; it compares the average inflow velocity to the horizontal mesh velocity and the horizontal mesh displacement just beside the top of the structure. Most of the time the deformation follows the inflow velocity quite closely. Whenever the inflow velocity starts to decrease, the deformation also decreases,

which you can observe as the negative values on the horizontal mesh velocity. Toward the end of the simulation, when inflow and structure deformation approach their steady-state values, the mesh velocity also decreases to zero.

Time=0.15 s Surface: 1 (1) Surface: 1 (1) Arrow Surface: Arrow Surface: Velocity field (spatial frame)
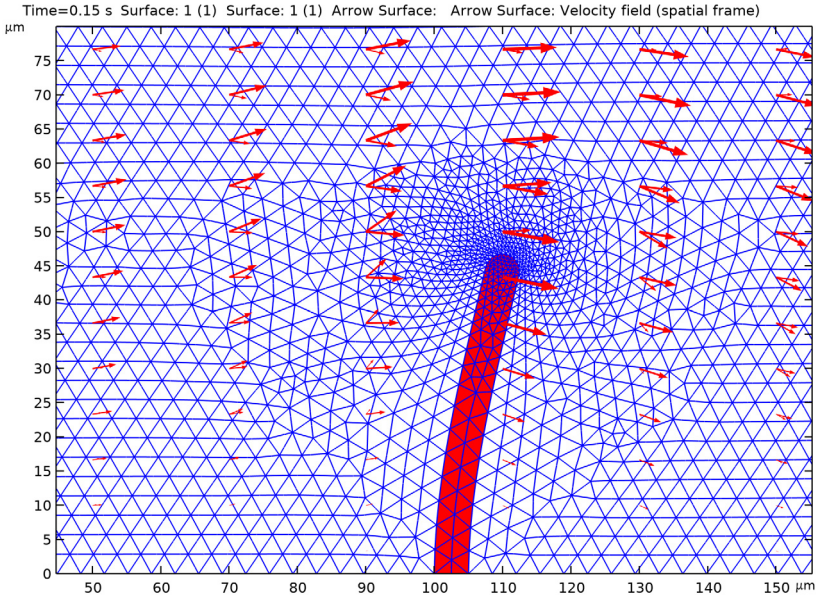
Figure 3: Mesh velocity (arrows) and mesh and geometry deformation at t = 0.15 s.

Global: Inlet mean velocity (m/s) Point Graph: Mesh velocity, x-component (mm/s) Point Graph: x-X (mm)
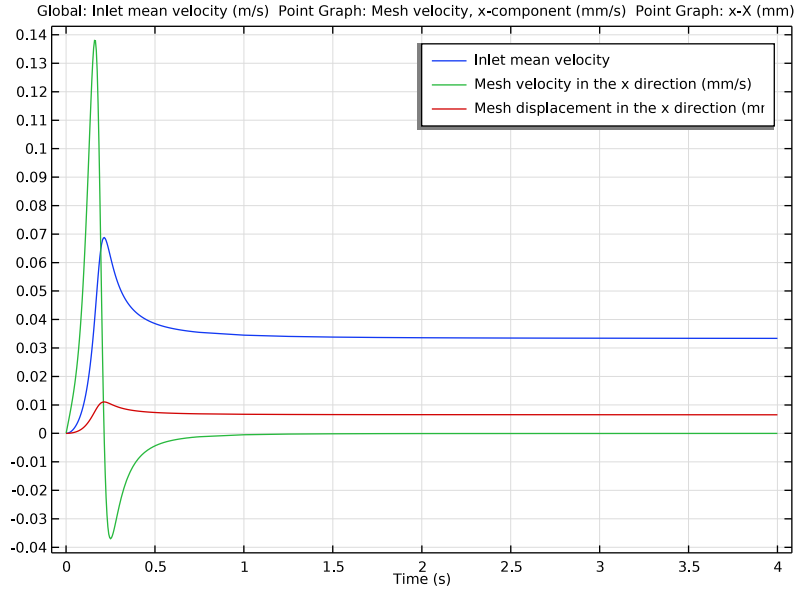
Figure 4: Inflow velocity, horizontal mesh velocity, and mesh deformation. The blue curve shows the average x direction velocity at the inflow boundary (m/s); the green shows $10^4 \times$ mesh displacement in the x direction (dx_ale; m) at the geometry point ($1.05 \cdot 10^{-4}$, $0.5 \cdot 10^{-4}$); and the red curve shows $10^5 \times$ mesh velocity in the x direction (xt; m/s), also at the point ($1.05 \cdot 10^{-4}$, $0.5 \cdot 10^{-4}$).

Figure 5 compares the meshes at different times. The first image shows the initial mesh, which you generate prior to solving the model. This mesh is equally distributed around the top of the structure. The second image shows the mesh in its deformed form. Because the structure deforms more in the horizontal direction, the mesh also changes more in this

direction: On the left, the mesh elements are stretched; on the right, they are compressed in the x direction.
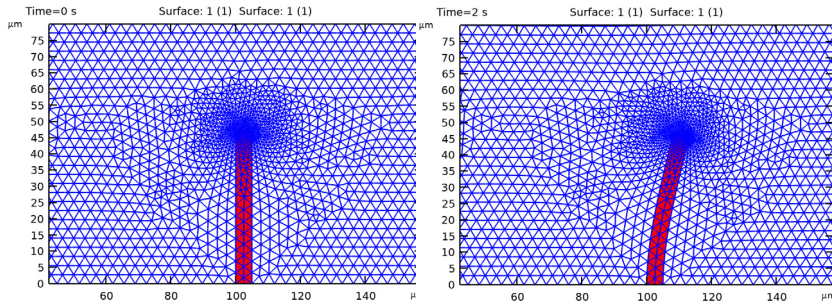
Figure 5: Geometry and mesh near the top of the structure at t = 0 s and 2 s.

Notes About the COMSOL Implementation

This example implements the model using Fluid–Solid Interaction multiphysics interface. The interface includes Laminar Flow and Solid Mechanics interfaces to model fluid and solid domains. The Deforming Domain feature is added to represent the domain and boundaries where the mesh can deform. By default, the Deforming Domain node has an empty selection. In this model, the fluid domain deforms significantly due to the deformation of the solid. Therefore, add a fluid domain to the selection list in the Deforming Domain node.

To model the interaction at the fluid–solid boundaries, The Fluid–Structure Interaction coupling feature is included in the interface. The feature automatically identifies the fluid–solid interaction boundaries and assigns the boundary condition to those boundaries.

To get a more accurate computation of the large strains, large deformation analysis is the default setting.

Application Library path: MEMS_Module/Fluid-Structure_Interaction/
fluid_structure_interaction

Modeling Instructions

From the **File** menu, choose **New**.

In the **New** window, click ◈ **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click Ω **2D**.

2 In the **Select Physics** tree, select **Fluid Flow>Fluid–Structure Interaction>Fluid–Solid Interaction**.

3 Click **Add**.

4 Click → **Study**.

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

6 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
U	3.33[cm/s]	0.0333 m/s	Inlet mean velocity at steady state
H	100[um]	1E-4 m	Channel height

DEFINITIONS

Analytic 1 (an1)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Local>Analytic**.

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

3 In the **Arguments** text field, type t.

4 In the **Expression** text field, type t^2/sqrt((0.04[s^2]-t^2)^2+(0.1[s]*t)^2).

5 Locate the **Units** section. In the table, enter the following settings:

Argument	Unit
t	s

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 **µm**.

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 300.

4 In the **Height** text field, type H.

5 Click ▦ **Build All Objects**.

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 5.

4 In the **Height** text field, type 47.5.

5 Locate the **Position** section. In the **x** text field, type 100.

6 Click ▦ **Build All Objects**.

Fillet 1 (fil1)

1 In the **Geometry** toolbar, click ◢ **Fillet**.

2 On the object **r2**, select Points 3 and 4 only.

It might be easier to select the points by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type 2.5.

5 Click ⊞ **Build All Objects**.

The geometry should look like that in the figure below.
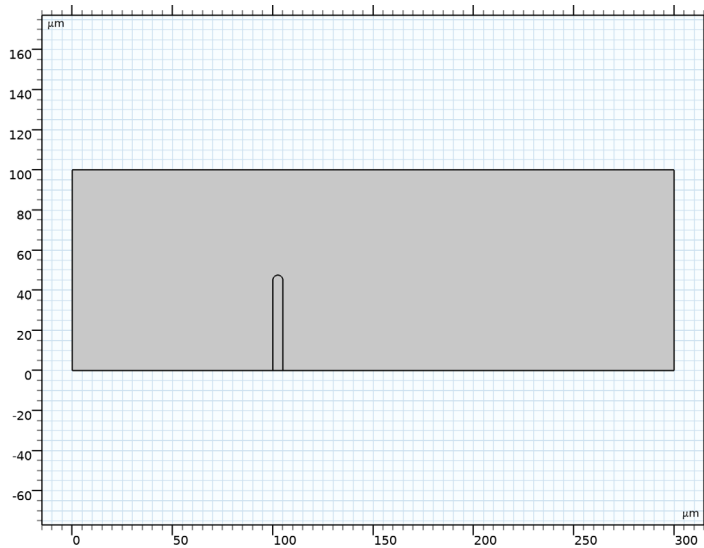
Modify the domain selections for the **Laminar Flow** and **Solid Mechanics** interface.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 Select Domain 1 only.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 Select Domain 2 only.

MOVING MESH

Deforming Domain 1

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

2 Select Domain 1 only.

LAMINAR FLOW (SPF)

In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

Inlet 1

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

2 Select Boundary 1 only.

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

4 From the list, choose **Fully developed flow**.

5 Locate the **Fully Developed Flow** section. In the U_{av} text field, type U*an1(t).

This gives a parabolic velocity profile with the specified mean velocity.

Outlet 1

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

2 Select Boundary 8 only.

SOLID MECHANICS (SOLID)

In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

Fixed Constraint 1

1 In the **Physics** toolbar, click ⬭ **Boundaries** and choose **Fixed Constraint**.

2 Select Boundary 5 only.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 Select Domain 1 only.

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

4 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	1e3	kg/m³	Basic
Dynamic viscosity	mu	1e-3	Pa·s	Basic

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

2 Select Domain 2 only.

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

4 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	2e5	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.33	1	Young's modulus and Poisson's ratio
Density	rho	7850	kg/m³	Basic

MESH 1

Free Triangular 1

1 In the **Mesh** toolbar, click ⬡ **Free Triangular**.

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

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

Size

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

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

3 From the **Predefined** list, choose **Fine**.

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

5 Click ▦ **Build All**.

STUDY 1

Step 1: Time Dependent

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

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

3 In the **Output times** text field, type range(0,0.005,0.75) range(1,0.25,4).

4 From the **Tolerance** list, choose **User controlled**.

5 In the **Relative tolerance** text field, type 0.0001.

6 In the **Home** toolbar, click ▬ **Compute**.

RESULTS

Velocity (spf)

The first default plot shows the velocity field. The solution is shown on the material frame. Switch to the spatial frame to plot the results in the deformed geometry.

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results>Datasets** node.

Velocity (spf)

1 In the **Model Builder** window, expand the **Results>Velocity (spf)** node, then click **Velocity (spf)**.

2 In the **Velocity (spf)** toolbar, click ⊡ **Plot**.

3 Click the ⊹ **Zoom Extents** button in the **Graphics** toolbar.

Proceed to reproduce Figure 2.

Streamline 1

1 Right-click **Velocity (spf)** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

3 From the **Positioning** list, choose **Starting-point controlled**.

4 From the **Entry method** list, choose **Coordinates**.

5 In the **x** text field, type `0^(range(1,15)) 125*1^(range(1,2))`.

6 In the **y** text field, type `range(0,100/14,100) 20 5`.

7 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Color** list, choose **Red**.

8 In the **Velocity (spf)** toolbar, click ⊡ **Plot**.

To animate flow around the structure, do the following:

Animation 1

1 In the **Results** toolbar, click ▦ **Animation** and choose **File**.

2 In the **Settings** window for **Animation**, locate the **Target** section.

3 From the **Target** list, choose **Player**.

4 Locate the **Animation Editing** section. From the **Time selection** list, choose **Interpolated**.

5 In the **Times (s)** text field, type `range(0.025,0.025,0.5)`.

6 Click the ▶ **Play** button in the **Graphics** toolbar.

To inspect the deformed geometry and deformed mesh near the top of the structure, Figure 5, proceed with the following steps.

Deformed Mesh and Geometry

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

2 In the **Settings** window for **2D Plot Group**, type Deformed Mesh and Geometry in the **Label** text field.

3 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

4 Locate the **Data** section. From the **Time (s)** list, choose **0**.

Surface 1

1 Right-click **Deformed Mesh and Geometry** and choose **Surface**.

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

3 In the **Expression** text field, type 1.

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

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

6 Select the **Wireframe** check box.

7 In the **Deformed Mesh and Geometry** toolbar, click **Plot**.

Surface 2

1 In the **Model Builder** window, right-click **Deformed Mesh and Geometry** and choose **Surface**.

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

3 In the **Expression** text field, type 1.

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

Selection 1

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

2 Select Domain 2 only.

3 In the **Deformed Mesh and Geometry** toolbar, click **Plot**.

Deformed Mesh and Geometry

1 In the **Model Builder** window, under **Results** click **Deformed Mesh and Geometry**.

2 Click the **Zoom Box** button in the **Graphics** toolbar and then use the mouse to zoom in on the obstacle.

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

4 From the **Time (s)** list, choose **2**.

Surface 2

1 In the **Model Builder** window, click **Surface 2**.

2 In the **Deformed Mesh and Geometry** toolbar, click **Plot**.

Add the arrow plot, to reproduce Figure 3.

Arrow Surface 1

1 In the **Model Builder** window, right-click **Deformed Mesh and Geometry** and choose **Arrow Surface**.

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

3 In the **x-component** text field, type xt.

4 In the **y-component** text field, type yt.

Deformed Mesh and Geometry

1 In the **Model Builder** window, click **Deformed Mesh and Geometry**.

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

3 From the **Time (s)** list, choose **0.15**.

Arrow Surface 2

Right-click **Deformed Mesh and Geometry** and choose **Arrow Surface**.

Arrow Surface 1

1 In the **Deformed Mesh and Geometry** toolbar, click ▣ **Plot**.

Finally, plot the horizontal mesh velocity, the mesh deformation at the point beside the top of the structure, and inflow velocity -- see Figure 4.

Mesh Velocity

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

2 In the **Settings** window for **1D Plot Group**, type Mesh Velocity in the **Label** text field.

Global 1

1 Right-click **Mesh Velocity** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
U*an1(t)	m/s	Inlet mean velocity

4 In the **Mesh Velocity** toolbar, click ▣ **Plot**.

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 105.

4 In the **y** text field, type 50.

Point Graph 1

1 In the **Model Builder** window, right-click **Mesh Velocity** and choose **Point Graph**.

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

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

4 Locate the **y-Axis Data** section. In the **Expression** text field, type xt.

5 From the **Unit** list, choose **mm/s**.

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
Mesh velocity in the x direction (mm/s)

9 Right-click **Point Graph 1** and choose **Duplicate**.

Point Graph 2

1 In the **Model Builder** window, click **Point Graph 2**.

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

3 In the **Expression** text field, type x-X.

4 From the **Unit** list, choose **mm**.

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

Legends
Mesh displacement in the x direction (mm)

6 In the **Mesh Velocity** toolbar, click **Plot**.