

Fluid-Structure Interaction

Introduction

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.

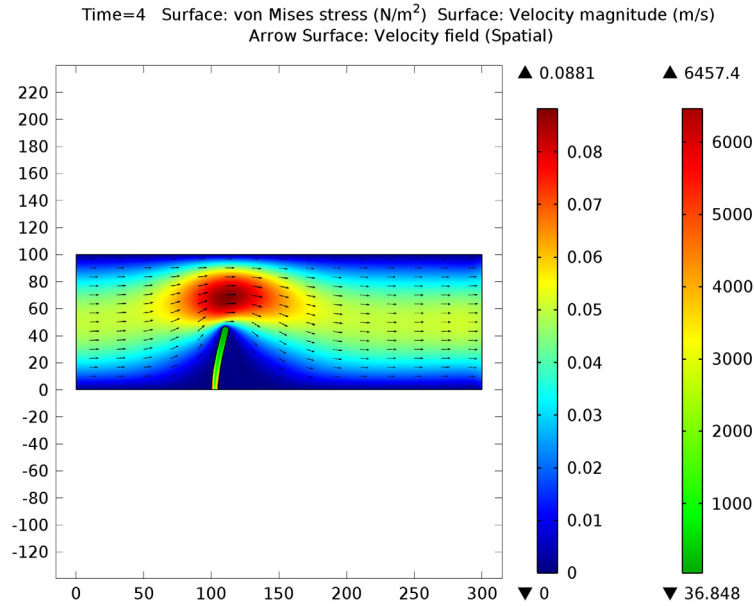


Figure 1: Fluid flows into this horizontal flow channel from the left, and it enters with a parabolic velocity profile. A narrow vertical structure in the channel (the straight vertical structure) forces the flow into a narrow path. Due to fluid pressure and viscous drag, the originally vertical structure 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 $\rho = 1000 \text{ kg/m}^3$ and dynamic viscosity $\eta = 0.001 \text{ Pa}\cdot\text{s}$. To demonstrate the desired techniques, assume the structure consists of a flexible material with a density $\rho = 7850 \text{ kg/m}^3$ and Young's modulus $E = 200 \text{ kPa}$.

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:

$$\begin{aligned} \rho \frac{\partial \mathbf{u}}{\partial t} - \nabla \cdot [-p \mathbf{I} + \eta (\nabla \mathbf{u} + (\nabla \mathbf{u})^T)] + \rho ((\mathbf{u} - \mathbf{u}_m) \cdot \nabla) \mathbf{u} &= \mathbf{F} \\ -\nabla \cdot \mathbf{u} &= 0 \end{aligned} \quad (1)$$

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

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

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

$$\mathbf{F}_T = -\mathbf{n} \cdot (-p \mathbf{I} + \eta (\nabla \mathbf{u} + (\nabla \mathbf{u})^T)) \quad (3)$$

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 Winslow smoothing. This is the default smoothing when using the Fluid-Structure Interaction interface. For more information, please refer to the Fluid-Structure Interaction interface in the *MEMS Module User's Guide*. Inside the obstacle, the moving mesh follows the deformations of the obstacle. At the exterior boundaries of the flow domain, the deformation is set to 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.

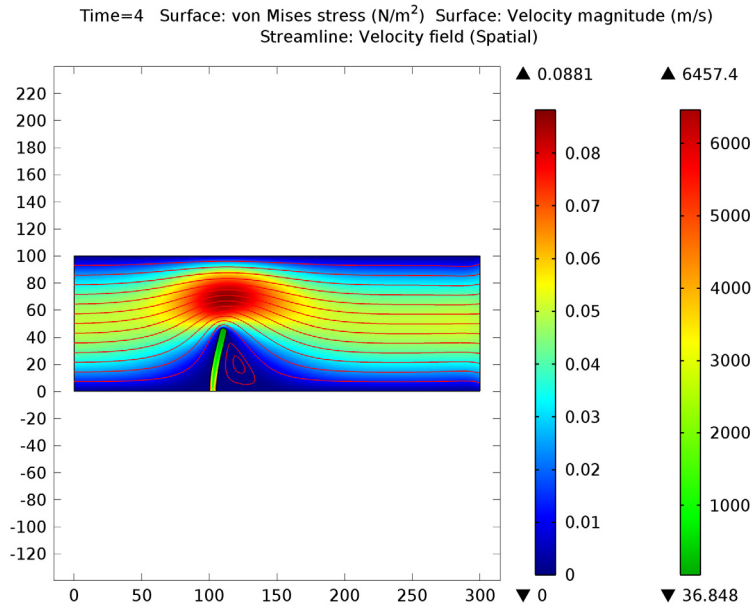


Figure 2: Flow velocity and geometry deformation at $t = 4$ s. The streamlines indicate the flow direction and the color indicates flow-velocity magnitude.

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.

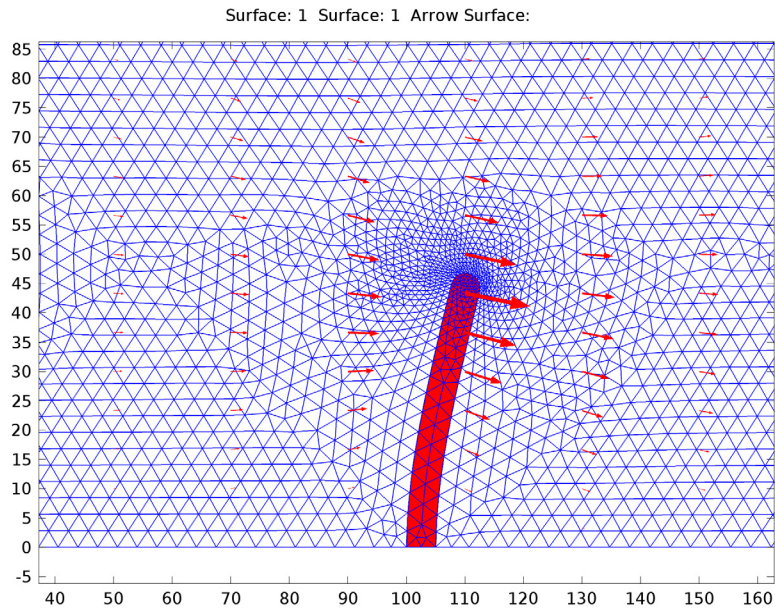


Figure 3: Mesh velocity (arrows) and mesh and geometry deformation at $t = 0.15$ s.

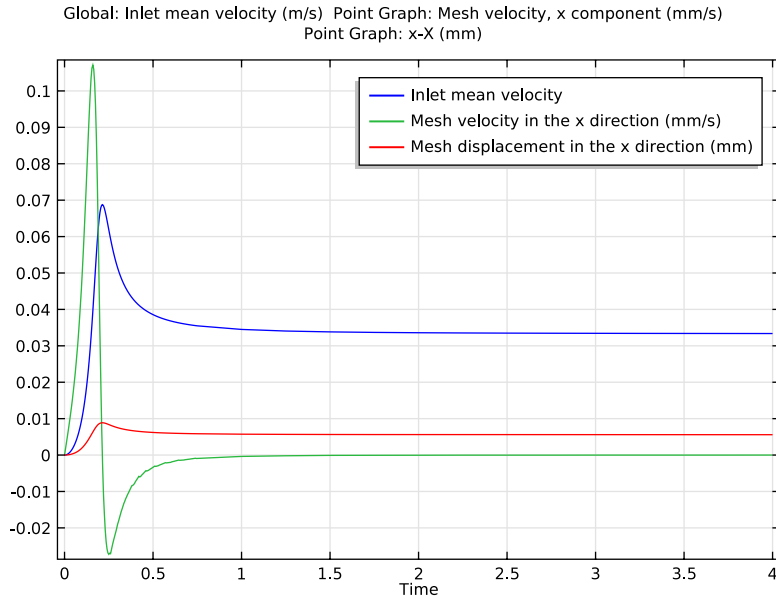


Figure 4: Inflow velocity, horizontal mesh velocity, and mesh deformation. The blue curve shows the average x direction velocity at the inflow boundary (m/s); the green shows $10^4 \times$ mesh displacement in the x direction (dx_{alg} ; m) at the geometry point $(1.05 \cdot 10^{-4}, 0.5 \cdot 10^{-4})$; 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.

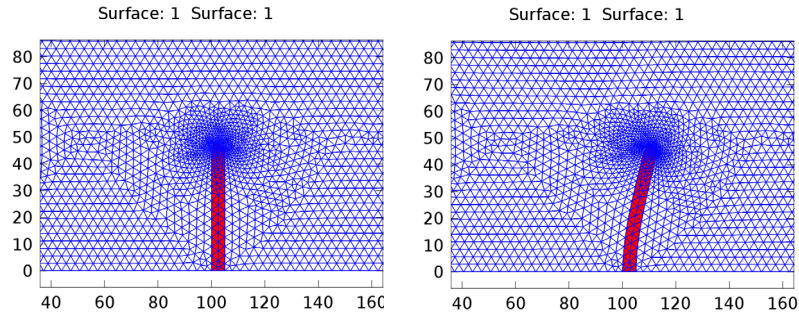


Figure 5: Geometry and mesh near the top of the structure at $t = 0$ s and 2 s.

Notes About the COMSOL Implementation

This example implements the model using Fluid-Structure Interaction interface. By default the Fluid-Structure Interaction interface treats all domains as fluid. Activate solid material model node in the area of the narrow structure. To get a more accurate computation of the large strains, large deformation analysis is the default setting. The interface automatically identifies the fluid-solid interaction boundaries and assigns the boundary condition to those boundaries.

Model Library path: MEMS_Module/Microfluidics/
fluid_structure_interaction

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click the **2D** button.
- 3 Click **Next**.
- 4 In the **Add physics** tree, select **Fluid Flow>Fluid-Structure Interaction (fsi)**.
- 5 Click **Add Selected**.
- 6 Click **Next**.

7 Find the **Studies** subsection. In the tree, select **Preset Studies>Time Dependent**.

8 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Description
U	3.33[cm/s]	Inlet mean velocity at steady state
H	100[um]	Channel height

Variables 1

1 Right-click **Global Definitions** and choose **Variables**.

2 In the **Variables** settings window, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Description
u_mean	$\frac{U \cdot t^2}{\sqrt{t^4 - 0.07[s^2] \cdot t^2 + 0.0016[s^4]}}$	Inlet mean velocity

GEOMETRY 1

1 In the **Model Builder** window, under **Model 1** click **Geometry 1**.

2 In the **Geometry** settings window, locate the **Units** section.

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

Rectangle 1

1 Right-click **Model 1>Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type 300.

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

5 Click the **Build All** button.

Rectangle 2

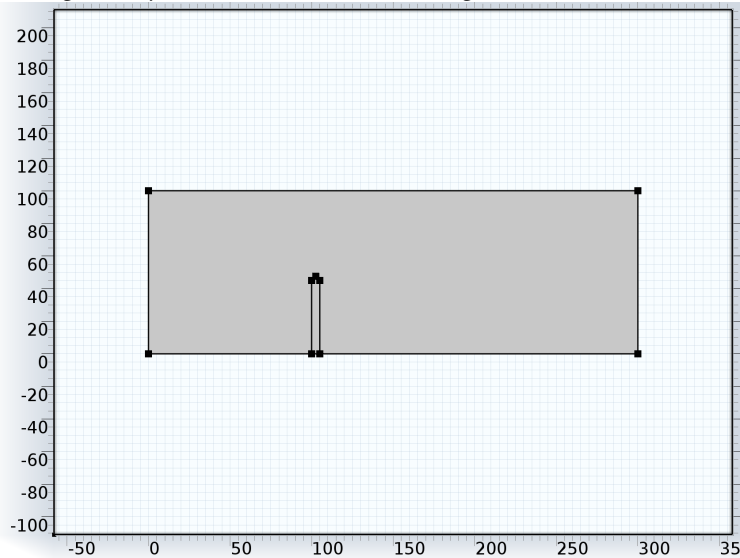
1 In the **Model Builder** window, right-click **Geometry 1** and choose **Rectangle**.

- 2 In the **Rectangle** settings window, locate the **Size** section.
- 3 In the **Width** edit field, type 5.
- 4 In the **Height** edit field, type 47.5.
- 5 Locate the **Position** section. In the **x** edit field, type 100.
- 6 Click the **Build All** button.

Fillet 1

- 1 Right-click **Geometry 1** and choose **Fillet**.
- 2 On the object **r2**, select Points 3 and 4 only.
- 3 In the **Fillet** settings window, locate the **Radius** section.
- 4 In the **Radius** edit field, type 2.5.
- 5 Click the **Build All** button.

The geometry should look like that in the figure below.



By default the Fluid-Structure Interaction interface treats all domains as fluid. Add the appropriate domain selection to the default node for the solid domain and proceed to the material specification.

FLUID-STRUCTURE INTERACTION*Linear Elastic Material 1*

- 1 In the **Model Builder** window, expand the **Model 1>Fluid-Structure Interaction** node, then click **Linear Elastic Material 1**.
- 2 Select Domain 2 only.

MATERIALS*Material 1*

- 1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Material**.
- 2 Select Domain 1 only.
- 3 In the **Material** settings window, locate the **Material Contents** section.
- 4 In the table, enter the following settings:

Property	Name	Value
Density	rho	1e3
Dynamic viscosity	mu	1e-3

Material 2

- 1 In the **Model Builder** window, right-click **Materials** and choose **Material**.
- 2 Select Domain 2 only.
- 3 In the **Material** settings window, locate the **Material Contents** section.
- 4 In the table, enter the following settings:

Property	Name	Value
Young's modulus	E	2e5
Poisson's ratio	nu	0.33
Density	rho	7850

FLUID-STRUCTURE INTERACTION

- 1 In the **Model Builder** window, under **Model 1** click **Fluid-Structure Interaction**.
- 2 In the **Fluid-Structure Interaction** settings window, locate the **Physical Model** section.
- 3 From the **Compressibility** list, choose **Incompressible flow**.
- 4 Locate the **Free Deformation Settings** section. From the **Mesh smoothing type** list, choose **Hyperelastic**.

Inlet 1

- 1 In the **Model Builder** window, right-click **Fluid-Structure Interaction** and choose the boundary condition **Laminar Flow>Inlet**.
- 2 Select Boundary 1 only.
- 3 In the **Inlet** settings window, locate the **Velocity** section.
- 4 In the U_0 edit field, type $u_mean*6*(H-Y)*Y/H^2$.

This gives a parabolic velocity profile with the specified mean velocity appropriate for laminar inflow. If you have a license for the CFD Module or Microfluidics Module, you can use the predefined Laminar inflow boundary condition with average velocity u_mean , a boundary condition that works for general inlet shapes.

Outlet 1

- 1 Right-click **Fluid-Structure Interaction** and choose the boundary condition **Laminar Flow>Outlet**.
- 2 Select Boundary 8 only.

Fixed Constraint 1

- 1 Right-click **Fluid-Structure Interaction** and choose the boundary condition **Solid Mechanics>Fixed Constraint**.
- 2 Select Boundary 5 only.

MESH 1*Free Triangular 1*

- 1 In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **Free Triangular**.
- 2 In the **Free Triangular** settings window, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Entire geometry**.

Size

- 1 In the **Model Builder** window, under **Model 1>Mesh 1** click **Size**.
- 2 In the **Size** settings window, locate the **Element Size** section.
- 3 From the **Predefined** list, choose **Fine**.
- 4 From the **Calibrate for** list, choose **Fluid dynamics**.
- 5 Click the **Build All** button.

STUDY I

Step 1: Time Dependent

- 1 In the **Model Builder** window, expand the **Study I** node, then click **Step 1: Time Dependent**.
- 2 In the **Time Dependent** settings window, locate the **Study Settings** section.
- 3 In the **Times** edit field, type range (0,0.005,0.75) range (1,0.25,4).
- 4 Select the **Relative tolerance** check box.
- 5 In the associated edit field, type 0.0001.
- 6 In the **Model Builder** window, right-click **Study I** and choose **Compute**.

RESULTS

Flow and Stress (fsi)

The first default plot shows the velocity field together with the von Mises stress.

- 1 In the **Model Builder** window, under **Results** right-click **Flow and Stress (fsi)** and choose **Plot**.
- 2 Click the **Zoom Extents** button on the Graphics toolbar.
Proceed to reproduce Figure 2, which shows the streamlines instead of the arrows.
- 3 Right-click **Results>Flow and Stress (fsi)>Arrow Surface 1** and choose **Disable**.
- 4 Right-click **Flow and Stress (fsi)** and choose **Streamline**.
- 5 In the **Streamline** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Fluid-Structure Interaction (Laminar Flow)>Velocity field (Spatial) (u_fluid,...,v_fluid)**.
- 6 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Start point controlled**.
- 7 From the **Entry method** list, choose **Coordinates**.
- 8 In the **x** edit field, type $0^{(\text{range}(1,15))} 125 \cdot 1^{(\text{range}(1,2))}$.
- 9 In the **y** edit field, type range(0,100/14,100) 20 5.
- 10 Locate the **Coloring and Style** section. From the **Color** list, choose **Red**.
- 11 Click the **Plot** button.

To animate flow around the structure, do the following:

Export

- 1 In the **Model Builder** window, under **Results** right-click **Export** and choose **Player**.

- 2 In the **Player** settings window, locate the **Scene** section.
- 3 From the **Subject** list, choose **Flow and Stress (fsi)**.
- 4 Locate the **Animation Editing** section. From the **Time selection** list, choose **Interpolated**.
- 5 In the **Times** edit field, type range (0.025, 0.025, 0.5).
- 6 Right-click **Results>Export>Player 1** and choose **Play**.

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

Data Sets

- 1 In the **Model Builder** window, under **Results** right-click **Data Sets** and choose **Solution**.
- 2 Right-click **Results>Data Sets>Solution 2** and choose **Add Selection**.
- 3 In the **Selection** settings window, locate the **Geometric Entity Selection** section.
- 4 From the **Geometric entity level** list, choose **Domain**.
- 5 Select Domain 2 only.

2D Plot Group 3

- 1 In the **Model Builder** window, right-click **Results** and choose **2D Plot Group**.
- 2 In the **2D Plot Group** settings window, locate the **Plot Settings** section.
- 3 From the **Frame** list, choose **Spatial (x, y, z)**.
- 4 Locate the **Data** section. From the **Time** list, choose **0**.
- 5 Right-click **Results>2D Plot Group 3** and choose **Surface**.
- 6 In the **Surface** settings window, locate the **Expression** section.
- 7 In the **Expression** edit field, type 1.
- 8 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
- 9 From the **Color** list, choose **Blue**.
- 10 Select the **Wireframe** check box.
- 11 Click to expand the **Quality** section. Click the **Plot** button.
- 12 In the **Model Builder** window, right-click **2D Plot Group 3** and choose **Surface**.
- 13 In the **Surface** settings window, locate the **Data** section.
- 14 From the **Data set** list, choose **Solution 2**.
- 15 From the **Time** list, choose **0**.
- 16 Locate the **Expression** section. In the **Expression** edit field, type 1.

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

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

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

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

3 From the **Time** list, choose **2**.

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

5 In the **Surface** settings window, locate the **Data** section.

6 From the **Time** list, choose **2**.

7 Click the **Plot** button.

Add the arrow plot, to reproduce [Figure 3](#).

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

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

3 From the **Time** list, choose **0.15**.

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

5 In the **Surface** settings window, locate the **Data** section.

6 From the **Time** list, choose **0.15**.

7 In the **Model Builder** window, right-click **2D Plot Group 3** and choose **Arrow Surface**.

8 In the **Arrow Surface** settings window, locate the **Expression** section.

9 In the **x component** edit field, type x_t .

10 In the **y component** edit field, type y_t .

11 Locate the **Coloring and Style** section. Select the **Scale factor** check box.

12 In the associated edit field, type 80000.

13 Click the **Plot** button.

Finally, plot the horizontal mesh velocity, the mesh deformation at the point beside the top of the structure, and inflow velocity ([Figure 4](#)).

1D Plot Group 4

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

2 Right-click **1D Plot Group 4** and choose **Global**.

- 3 In the **Global** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Definitions>Inlet mean velocity (u_mean)**.
- 4 Click the **Plot** button.

Data Sets

- 1 In the **Model Builder** window, under **Results** right-click **Data Sets** and choose **Cut Point 2D**.
- 2 In the **Cut Point 2D** settings window, locate the **Point Data** section.
- 3 In the **x** edit field, type 105.
- 4 In the **y** edit field, type 50.

ID Plot Group 4

- 1 In the **Model Builder** window, under **Results** right-click **ID Plot Group 4** and choose **Point Graph**.
- 2 In the **Point Graph** settings window, locate the **Data** section.
- 3 From the **Data set** list, choose **Cut Point 2D 1**.
- 4 Locate the **y-Axis Data** section. In the **Expression** edit field, type x_t .
- 5 From the **Unit** list, choose **mm/s**.
- 6 Locate 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 **Results>ID Plot Group 4>Point Graph 1** and choose **Duplicate**.
- 10 In the **Point Graph** settings window, locate the **y-Axis Data** section.
- 11 In the **Expression** edit field, type $x-X$.
- 12 From the **Unit** list, choose **mm**.
- 13 Locate the **Legends** section. In the table, enter the following settings:

Legends
Mesh displacement in the x direction (mm)

- 14 Click the **Plot** button.

