Created in COMSOL Multiphysics 5.6

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.

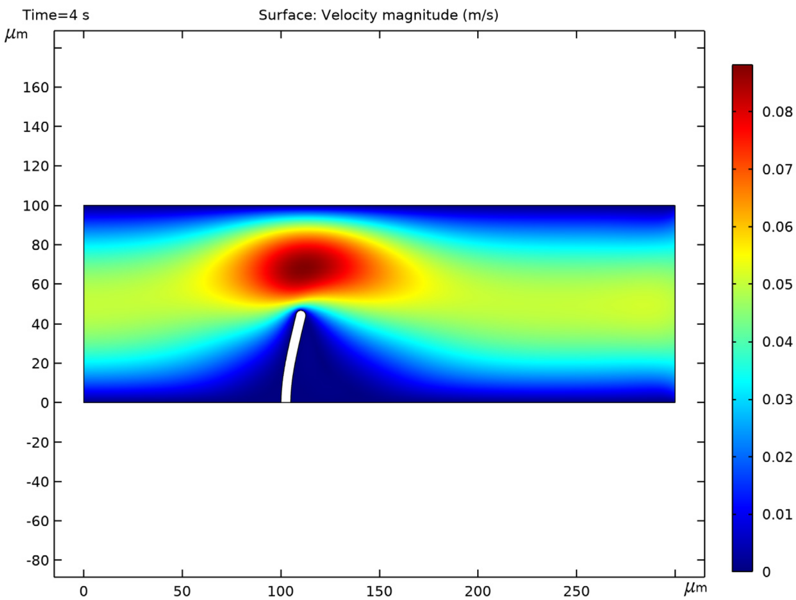


Figure 1: Fluid flows into this horizontal flow channel from the left, and it enters with a parabolic velocity profile. A narrow vertical structure in the channel (the straight vertical structure) forces the flow into a narrow path. Due to fluid pressure and viscous drag, the originally vertical structure 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 ρ = 1000 kg/m^3 and dynamic viscosity η = 0.001 Pa·s. To demonstrate the desired techniques, assume the structure consists of a flexible material with a density ρ = 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

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

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

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

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 Hyperelastic smoothing. 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 << 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.

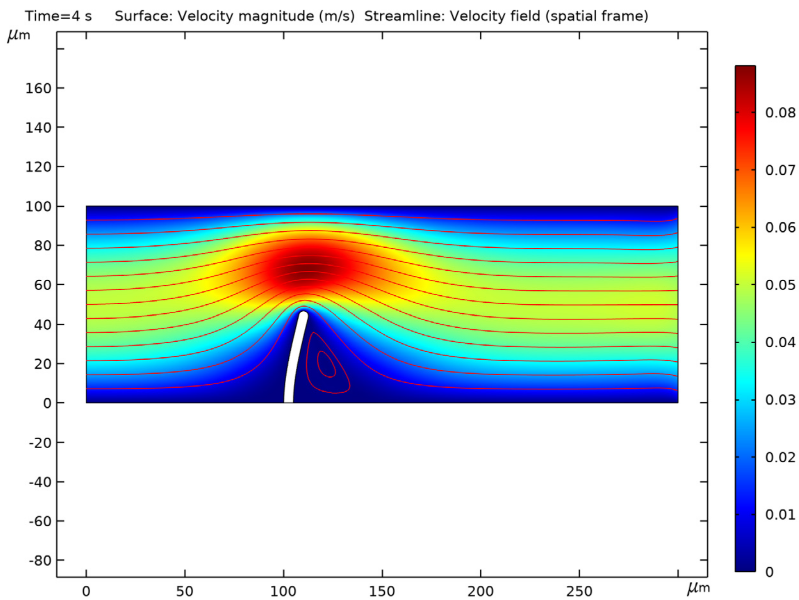


Figure 2: Flow velocity and geometry deformation at t = 4 s. The streamlines indicate the flow direction and the color indicates flow-velocity magnitude.

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.

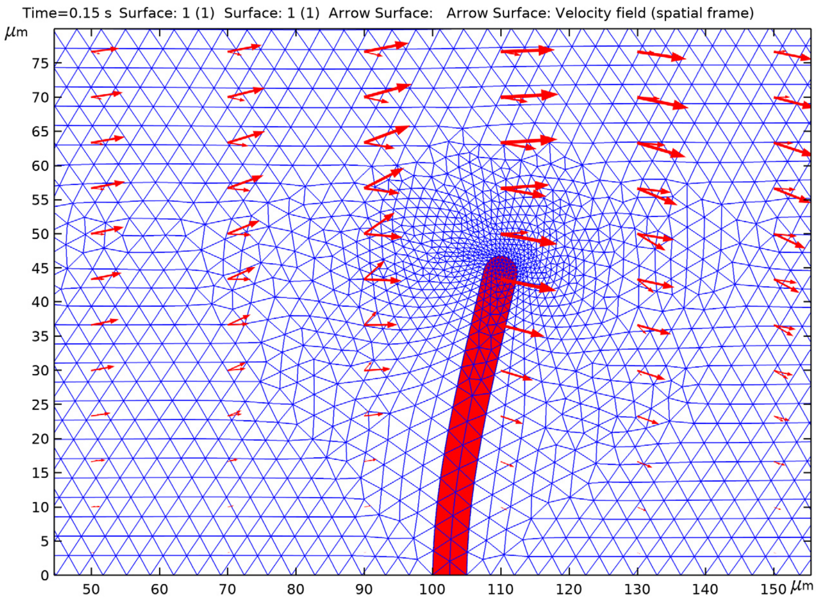


Figure 3: Mesh velocity (arrows) and mesh and geometry deformation at t = 0.15 s.

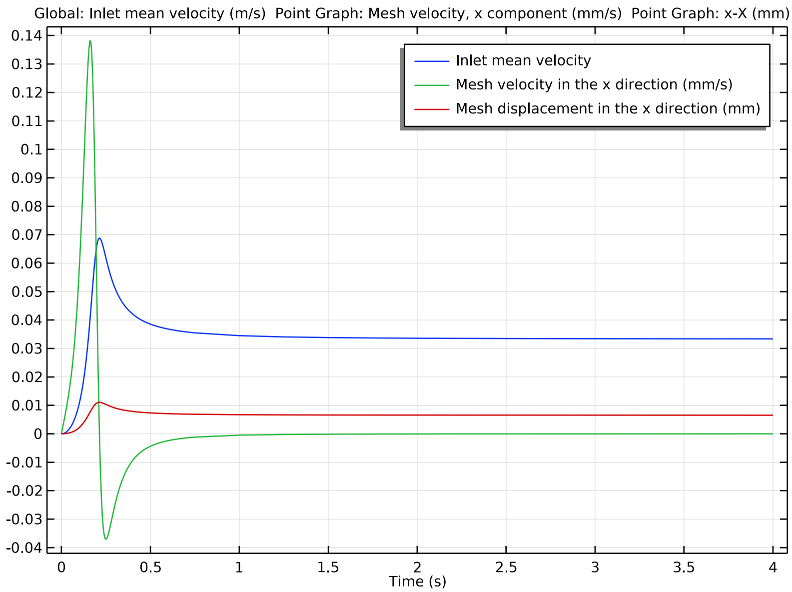


Figure 4: Inflow velocity, horizontal mesh velocity, and mesh deformation. The blue curve shows the average x direction velocity at the inflow boundary (m/s); the green shows 10^4×mesh displacement in the x direction (dx_ale; m) at the geometry point ($1.05\cdot10^{-4}$, $0.5\cdot10^{-4}$); and the red curve shows 10^3×mesh velocity in the x direction (xt; m/s), also at the point ($1.05\cdot10^{-4}$, $0.5\cdot10^{-4}$).

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.

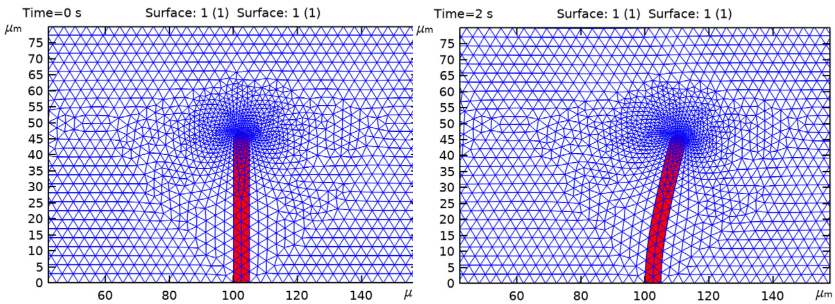


Figure 5: Geometry and mesh near the top of the structure at t = 0 s and 2 s.

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.

Application Library path: `MEMS_Module/Fluid-Structure_Interaction/fluid_structure_interaction`

Modeling Instructions

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

NEW

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

MODEL WIZARD

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

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

Variables 1

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

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
u_mean	U*t^2/sqrt(t^4-0.07[s^2]*t^2+0.0016[s^4])	m/s	Inlet mean velocity

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 **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.

DEFINITIONS

Deforming Domain 1

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

2 Select Domain 1 only.

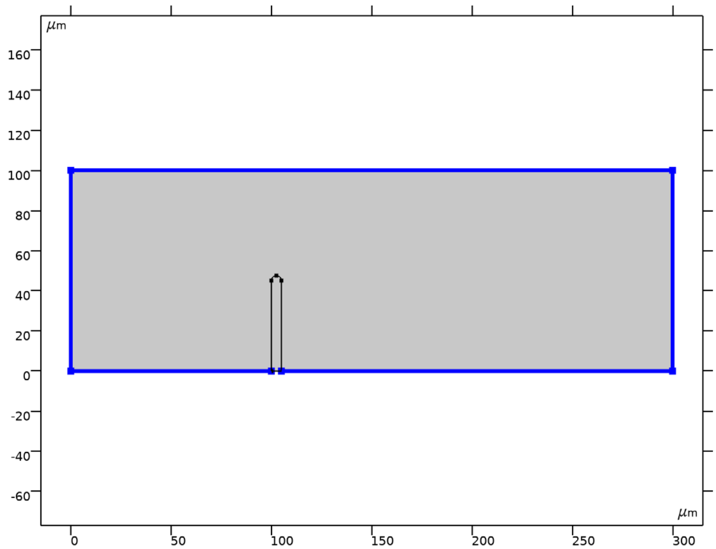
Fixed Boundary 1

1 In the **Definitions** toolbar, click **Moving Mesh** and choose **Fixed Boundary**.

2 In the **Settings** window for **Fixed Boundary**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

4 Select Boundaries 1–3, 5, 7, and 8 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 **Velocity** section.

4 In the U_0 text 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 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	Basic
Poisson's ratio	nu	0.33	1	Basic
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 Right-click **Animation 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.

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, 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**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Global definitions>Variables>u_mean - Inlet mean velocity - m/s**.

3 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)`

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 `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**.