Peristaltic Pump

Introduction

In a peristaltic pump, rotating rollers squeeze a flexible tube. As the pushed-down rollers move along the tube, fluids in the tube follow the motion. The main advantage of the peristaltic pump is that no seals, valves, or other internal parts ever touch the fluid. Due to their cleanliness, peristaltic pumps have found many applications in the pharmaceutical, chemical, and food industries. Besides this, the action of a peristaltic pump is very gentle, which is important if the fluid is easily damaged. Peristaltic pumps are therefore used in medical applications, one of which is moving the blood through the body during open heart surgery. Other pumps would risk destroying the blood cells.

This COMSOL Multiphysics model of a peristaltic pump is a combination of structural mechanics (to model the squeezing of the tube) and fluid dynamics (to compute the fluid’s motion); that is, this is an example of fluid-structure interaction (FSI).

Model Definition

The model is set up in 2D axial symmetry (Figure 1). A vinyl tube 0.1 m long has an inner radius of 1 cm and an outer radius of 1.5 cm; it contains a gas of density \( \rho = 1 \text{ kg/m}^3 \) and viscosity \( \mu = 3 \cdot 10^{-5} \text{ Pa·s} \). A time- and position-dependent force density is applied to the outer wall of the tube, in the radial direction. This force density could have been taken from real data from a peristaltic pump, but for simplicity’s sake this example models it with a Gaussian distribution along the length of the tube. The Gaussian distribution has a width of 1 cm and is moving with the constant velocity \( 0.03 \text{ m/s} \) in the positive \( z \) direction. To represent the engagement of the roll, the force density, multiplied by a smoothed Heaviside function, kicks in at \( t = 0.1 \text{ s} \) and takes the force to its full development at \( t = 0.5 \text{ s} \). Likewise, the disengagement of the roll starts at \( t = 1.0 \text{ s} \) and ends at \( t = 1.4 \text{ s} \). The example models the tube’s deformation during a full cycle of 1.5 s.
The geometry of the peristaltic pump as it is deforming under the pressure of the roll. The tube is rotationally symmetric with respect to the z-axis. The color shows the deformation of the tube material.

**Domain Equations**

The structural mechanics computations use the assumption that the material is linear and elastic and take geometric nonlinearities into account. With the capabilities provided by the Structural Mechanics Module, you can easily extend this model for a more complicated material model.

The fluid flow is described by the incompressible Navier-Stokes equations:

\[
\rho \frac{\partial \mathbf{u}}{\partial t} - \nabla \cdot (\mathbf{u} (\nabla \mathbf{u})^T) + \mu \mathbf{u} \cdot \nabla \mathbf{u} + \nabla p = 0 \tag{1}
\]

\[
\nabla \cdot \mathbf{u} = 0
\]

where \( \rho \) denotes the density (SI unit: kg/m\(^3\)), \( \mathbf{u} \) the velocity (SI unit: m/s), \( \mu \) the viscosity (SI unit: Pa\(\cdot\)s), and \( p \) the pressure (SI unit: Pa). The equations are set up and solved in axial symmetry on a deformed mesh inside the tube.

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. Inside the wall of the tube, the moving mesh follows the deformations of the tube. For more information, please refer to the chapter The Fluid-Structure Interaction Interface in the Structural Mechanics Module User’s Guide.

**BOUNDARY CONDITIONS**

For the structural mechanics computations, the time- and coordinate-dependent load described serves as the boundary condition at the tube’s outer surface. The top and bottom ends of the tube are constrained along both coordinate axes.

For the fluid simulation, the boundary condition at the inlet and the outlet assumes that the total stress is zero, that is:

\[
\mathbf{n} \cdot [-p \mathbf{I} + \mu (\nabla \mathbf{u} + (\nabla \mathbf{u})^T)] = 0
\]  (2)

The mesh is fixed to a zero \(r\) displacement at the symmetry axis and a zero \(z\) displacement at the top and the bottom of the tube.

**COMPUTATION OF VOLUMETRIC FLOW RATES AND TOTAL VOLUME OF PUMPED FLUID**

The model’s dependent variables (those solved for) are the displacements of the tube wall as well as the fluid’s velocity \(\mathbf{u} = (u, v)\) and pressure \(p\). To get the volumetric flow rate of the fluid \(V\) in \(\text{m}^3/\text{s}\) and the total volume of pumped fluid, you must perform some additional calculations. To get the volumetric flow rate at any instant \(t\), compute a boundary integral over the pipe’s inlet or outlet boundary:

\[
\dot{V}_{\text{in}} = -\int_{s_{\text{in}}} 2\pi r (\mathbf{n} \cdot \mathbf{u}) ds
\]

\[
\dot{V}_{\text{out}} = \int_{s_{\text{out}}} 2\pi r (\mathbf{n} \cdot \mathbf{u}) ds
\]

where \(\mathbf{n}\) is the outward-pointing unit normal of the boundary, \(\mathbf{u}\) equals the velocity vector, and \(s\) is the boundary length parameter along which you integrate. In this particular model, the inlet and outlet boundaries are horizontal so \(\mathbf{n} \cdot \mathbf{u} = n_x u + n_y v\) simplifies to \(v\) or \(-v\) depending on the direction of flow.

It is of interest to track how much volume of fluid, \(V_{\text{pump}}(t)\) (SI unit: \(\text{m}^3\)), that is conveyed out of the outlet during a peristaltic cycle, which is equal to the following time integral:
\[ V_{\text{pump}}(t) = \int_0^t V_{\text{out}}(t') \, dt'. \]

To compute this integral, specify the corresponding ODE in COMSOL Multiphysics

\[ \frac{dV_{\text{pump}}}{dt} = V_{\text{out}} \]

with proper initial conditions; the software then integrates this equation.

**Results**

**Figure 2** shows three snapshots from the peristaltic pump in action.

**Figure 2:** Snapshots of the velocity field and the shape of the inside of the tube at \( t = 0.3 \) s, \( t = 0.7 \) s, and \( t = 1.2 \) s. The colors represent the magnitude of the velocity, and the arrows its direction.

**Figure 3** shows the inner volume of the tube as a function of time. At \( t = 0.3 \) s, the roll has begun its engagement phase and is increasing its pressure on the tube. As less space is left for the gas, it is streaming out of the tube, through both the inlet and the outlet. At \( t = 0.7 \) s, the roll has been fully engaged for a while. As it is moving up the tube, so is the gas, both at the inlet and at the outlet. This is where most of the net flow in the direction from the inlet to the outlet is created. Finally, at \( t = 1.2 \) s, the engagement
process is reversed as the roll is disengaging. Fluid is streaming into the tube from both ends.

\[ V_{\text{in}} - V_{\text{out}} \]

and integrating over time, you generate Figure 3.

**Figure 3:** The inner volume (m$^3$) of the tube as a function of time (s).

**Figure 4** shows the inlet and outlet flows, and it confirms the overall behavior indicated in the velocity snapshots. As the roll is engaged, it squeezes gas out through both the inlet and outlet. When the fully engaged roll moves along the tube, gas flows steadily in through the inlet and out through the outlet. As the roll is disengaging, the gas is being sucked back into the tube. Note that a real peristaltic pump usually removes or minimizes the peaks associated with volume changes with the help of a second roll that engages at the same time as the first roll disengages. This way there are hardly any volume changes, and the fluid flows forward all the time. Also note from Figure 4 that by taking the difference of the curves, $V_{\text{in}} - V_{\text{out}}$ and integrating over time, you generate Figure 3.
Figure 4: Inlet and outlet flow in m³/s as functions of time. Positive values indicate that the gas is flowing in through the inlet and out through the outlet.
Figure 5 sums up the process, plotting the accumulated net flow versus time. It is worth noting that although the accumulated flow during the first 0.5 s of the cycle is zero or negative, it is well above zero after the full cycle.

Figure 5: Accumulated flow (m$^3$) through the pump and volume of fluid conveyed out of the outlet versus time (s).

Notes About the COMSOL Implementation

This model is primarily intended to demonstrate the use of the Fluid-Structure Interaction interface, but it also shows some features for results analysis. It defines integration coupling operators to integrate the flow rate. An ordinary differential equation calculates the accumulated fluid volume that has passed through the pump at certain points in time. The smooth step function in this model is called $flc2hs$ (a $C^2$-continuous step).

You solve this model as a coupled problem. However, the reaction forces from the fluid on the tube are very small compared to the force applied by the rollers. In this case, you can solve first for structural mechanics and then for the fluid dynamics, thereby reducing memory consumption.
**Model Library path:** Structural_Mechanics_Module/Fluid-Structure Interaction/peristaltic_pump

---

**Modeling Instructions**

**MODEL WIZARD**
1. Go to the Model Wizard window.
2. Click the 2D axisymmetric button.
3. Click Next.
5. Click Add Selected.
6. Click Next.
7. In the Studies tree, select Preset Studies>Time Dependent.
8. Click Finish.

**GEOMETRY 1**

**Rectangle 1**
1. In the Model Builder window, right-click Geometry 1 and choose Rectangle.
2. Go to the Settings window for Rectangle.
3. Locate the Size section. In the Width edit field, type 0.01.
4. In the Height edit field, type 0.1.
5. Click the Build All button.

**Rectangle 2**
1. In the Model Builder window, right-click Geometry 1 and choose Rectangle.
2. Go to the Settings window for Rectangle.
3. Locate the Size section. In the Width edit field, type 5e-3.
4. In the Height edit field, type 0.1.
5. Locate the Position section. In the r edit field, type 0.01.
6. Click the Build All button.
GLOBAL DEFINITIONS

Parameters

1. In the Model Builder window, right-click Global Definitions and choose Parameters.
2. Go to the Settings window for Parameters.
3. Locate the Parameters section. In the Parameters table, enter the following settings:

<table>
<thead>
<tr>
<th>NAME</th>
<th>EXPRESSION</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>t_on</td>
<td>0.3[s]</td>
<td>Time when roll is engaged</td>
</tr>
<tr>
<td>t_off</td>
<td>1.2[s]</td>
<td>Time when roll is disengaged</td>
</tr>
<tr>
<td>dt</td>
<td>0.2[s]</td>
<td>Time to reach full force</td>
</tr>
<tr>
<td>z0</td>
<td>0.03[m]</td>
<td>z coordinate where roll starts</td>
</tr>
<tr>
<td>v0</td>
<td>0.03[m/s]</td>
<td>Vertical velocity of roll</td>
</tr>
<tr>
<td>width</td>
<td>0.01[m]</td>
<td>Width of Gaussian force distribution</td>
</tr>
<tr>
<td>Ttot</td>
<td>1.5[s]</td>
<td>Total time for a pump cycle</td>
</tr>
<tr>
<td>Lmax</td>
<td>1.e8[N/m^2]</td>
<td>Max load</td>
</tr>
</tbody>
</table>

DEFINITIONS

To define the force density for load applied to the outer wall of the tube, follow the steps given below.

Analytic 1

1. In the Model Builder window, right-click Model 1>Definitions and choose Functions>Analytic.
2. Go to the Settings window for Analytic.
3. Locate the Function Name section. In the Function name edit field, type load.
4. Locate the Parameters section. In the Expression edit field, type flc2hs(t_off/dt-ts,1)*flc2hs(ts-t_on/dt,1)*exp(-(zs-(z0+v0*ts*dt)/width)^2/2).
5. In the Arguments edit field, type zs,ts.

   Note that the function arguments are made dimensionless by setting zs = z/width and ts = t/dt.

To compute inflow/outflow rates, define the integration over the relevant boundaries.

Integration 1

1. In the Model Builder window, right-click Definitions and choose Model Couplings>Integration.
2. Go to the Settings window for Integration.
3 Locate the **Source Selection** section. From the **Geometric entity level** list, select **Boundary**.

4 Select Boundary 2 only.

**Integration 2**

1 In the **Model Builder** window, right-click **Definitions** and choose **Model Couplings>Integration**.

2 Go to the **Settings** window for Integration.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, select **Boundary**.

4 Select Boundary 3 only.

**Variables 1**

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

2 Go to the **Settings** window for Variables.

3 Locate the **Variables** section. In the **Variables** table, enter the following settings:

<table>
<thead>
<tr>
<th>NAME</th>
<th>EXPRESSION</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>inflow</td>
<td>in(top1(2<em>pi</em>r*w_fluid))</td>
<td>Inflow</td>
</tr>
<tr>
<td>outflow</td>
<td>in(top2(2<em>pi</em>r*w_fluid))</td>
<td>Outflow</td>
</tr>
</tbody>
</table>

**MATERIALS**

1 In the **Model Builder** window, right-click **Model 1>Materials** and choose **Open Material Browser**.

2 Go to the **Material Browser** window.

3 Locate the **Materials** section. In the **Materials** tree, select **Built-In>Nylon**.

4 Right-click and choose **Add Material to Model** from the menu.

**Nylon**

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

2 Select Domain 2 only.

3 Go to the **Settings** window for Material.

4 Locate the **Material Contents** section. In the **Material contents** table, enter the following settings:

<table>
<thead>
<tr>
<th>PROPERTY</th>
<th>NAME</th>
<th>VALUE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Poisson's ratio</td>
<td>nu</td>
<td>0.33</td>
</tr>
</tbody>
</table>
Material 2
1 In the Model Builder window, right-click Materials and choose Material.
2 Select Domain 1 only.
3 Go to the Settings window for Material.
4 Locate the Material Contents section. In the Material contents table, enter the following settings:

<table>
<thead>
<tr>
<th>PROPERTY</th>
<th>NAME</th>
<th>VALUE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>rho</td>
<td>1</td>
</tr>
<tr>
<td>Dynamic viscosity</td>
<td>mu</td>
<td>1e-3</td>
</tr>
</tbody>
</table>

Fluid-Structure Interaction
1 In the Model Builder window, click Model 1>Fluid-Structure Interaction.
2 Go to the Settings window for Fluid-Structure Interaction.
3 Locate the Physical Model section. From the Compressibility list, select Incompressible flow.

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

Boundary Load 1
1 In the Model Builder window, right-click Fluid-Structure Interaction and choose the boundary condition Solid Mechanics>Boundary Load.
2 Select Boundary 7 only.
3 Go to the Settings window for Boundary Load.
4 Locate the Force section. Specify the $\mathbf{F}_A$ vector as

<table>
<thead>
<tr>
<th></th>
<th>r</th>
<th>z</th>
</tr>
</thead>
<tbody>
<tr>
<td>$-L_{max}\cdot \text{load}(z/\text{width}, t/dt)$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>0</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Fixed Constraint 1
1 In the Model Builder window, right-click Fluid-Structure Interaction and choose the boundary condition Solid Mechanics>Fixed Constraint.
2 Select Boundaries 5 and 6 only.
Open Boundary 1
1 In the Model Builder window, right-click Fluid-Structure Interaction and choose the boundary condition Laminar Flow>Open Boundary.
2 Select Boundaries 2 and 3 only.

Prescribed Mesh Displacement 2
1 In the Model Builder window, right-click Fluid-Structure Interaction and choose Prescribed Mesh Displacement.
2 Select Boundaries 2 and 3 only.
3 Go to the Settings window for Prescribed Mesh Displacement.
4 Locate the Prescribed Mesh Displacement section. Clear the Prescribed r displacement check box.

Prescribed Mesh Displacement 3
1 In the Model Builder window, right-click Fluid-Structure Interaction and choose Prescribed Mesh Displacement.
2 Select Boundary 1 only.
3 Go to the Settings window for Prescribed Mesh Displacement.
4 Locate the Prescribed Mesh Displacement section. Clear the Prescribed z displacement check box.

Define the ordinary differential equations to calculate volume of the pumped fluid and the accumulated flow.

In the Model Builder window’s toolbar, click the Show button and select Advanced Physics Interface Options in the menu.

Global Equations 1
1 In the Model Builder window, right-click Fluid-Structure Interaction and choose Global>Global Equations.
2 Go to the Settings window for Global Equations.
3 Locate the Global Equations section. In the associated table, enter the following settings:

<table>
<thead>
<tr>
<th>NAME</th>
<th>F(U,UT,UTT,T)</th>
</tr>
</thead>
<tbody>
<tr>
<td>netflow</td>
<td>netflowt-(outflow+inflow)/2</td>
</tr>
</tbody>
</table>

Global Equations 2
1 In the Model Builder window, right-click Fluid-Structure Interaction and choose Global>Global Equations.
2 Go to the **Settings** window for Global Equations.

3 Locate the **Global Equations** section. In the associated table, enter the following settings:

<table>
<thead>
<tr>
<th>NAME</th>
<th>F(U,UT,UTT,T)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vpump</td>
<td>Vpumpt-outflow</td>
</tr>
</tbody>
</table>

**STUDY 1**

**Step 1: Time Dependent**

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.

2 Go to the **Settings** window for Time Dependent.

3 Locate the **Study Settings** section. In the **Times** edit field, type `range(0, 0.01, 1.5)`.

4 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

**RESULTS**

**Flow and Stress (fsi)**

The first default plot shows the stress distribution together with the velocity field at $t = 1.5$ s. To plot the total displacement at $t = 0.7$ s (**Figure 1**), follow these steps:

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

2 Go to the **Settings** window for Surface.

3 In the upper-right corner of the **Expression** section, click **Replace Expression**.

4 From the menu, choose **Fluid-Structure Interaction (Solid Mechanics)>Total displacement (fsi.disp)**.

5 Locate the **Expression** section. From the **Unit** list, select **mm**.

6 In the **Model Builder** window, right-click **Surface 2** and choose **Disable**.

7 Right-click **Arrow Surface 1** and choose **Disable**.

8 In the **Model Builder** window, click **Flow and Stress (fsi)**.

9 Go to the **Settings** window for 2D Plot Group.

10 Locate the **Data** section. From the **Time** list, select **0.7**.

11 Click the **Plot** button.

12 Click the **Zoom Extents** button on the Graphics toolbar.
To produce the series of snapshots of the velocity field shown in Figure 2, proceed with the following steps:

1. In the **Model Builder** window, right-click **Surface 1** and choose **Disable**.
2. Right-click **Surface 2** and choose **Enable**.
3. Right-click **Arrow Surface 1** and choose **Enable**.
4. In the **Model Builder** window, click **Flow and Stress (fsi)**.
5. Go to the **Settings** window for 2D Plot Group.
6. Locate the **Data** section. From the **Time** list, select **0.3**.
7. Click the **Plot** button.
8. In the **Model Builder** window, click **Flow and Stress (fsi)**.
9. Go to the **Settings** window for 2D Plot Group.
10. Locate the **Data** section. From the **Time** list, select **0.7**.
11. Click the **Plot** button.
12. In the **Model Builder** window, click **Flow and Stress (fsi)**.
13. Go to the **Settings** window for 2D Plot Group.
14. Locate the **Data** section. From the **Time** list, select **1.2**.
15. Click the **Plot** button.

**Export**

To animate the velocity field as a function of time, do the following:

1. In the **Model Builder** window, right-click **Results>Export** and choose **Player**.
2. Go to the **Settings** window for Player.
3. Locate the **Scene** section. From the **Subject** list, select **Flow and Stress (fsi)**.
4. Click the **Generate Frame** button.
5. Right-click **Player 1** and choose **Play**.

To plot the total volume of fluid contained in the pump (Figure 3), follow the steps given below.

**Derived Values**

1. In the **Model Builder** window, right-click **Results>Derived Values** and choose **Integration>Surface Integration**.
2. Select Domain 1 only.
3. Go to the **Settings** window for Surface Integration.
4 Locate the **Expression** section. In the **Expression** edit field, type 1.
5 Select the **Description** check box.
6 In the associated edit field, type **Volume**.
7 Click to expand the **Integration Settings** section.
8 Select the **Compute volume integral** check box.
9 Click the **Evaluate** button.
10 In the **Results** window, click **Plot**.

To plot the inlet and outlet flow rates (Figure 4), accumulated flow through the pump and volume of fluid conveyed out of the outlet (Figure 5), follow the steps given below.

**1D Plot Group 5**
1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.
2 Right-click **Results>1D Plot Group 5** and choose **Global**.
3 Go to the **Settings** window for Global.
4 In the upper-right corner of the **y-Axis Data** section, click **Replace Expression**.
5 From the menu, choose **Definitions>Inflow (inflow)**.
6 In the upper-right corner of the **y-Axis Data** section, click **Add Expression**.
7 From the menu, choose **Definitions>Outflow (outflow)**.
8 Click the **Plot** button.

**1D Plot Group 6**
1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.
2 Right-click **Results>1D Plot Group 6** and choose **Global**.
3 Go to the **Settings** window for Global.
4 In the upper-right corner of the **y-Axis Data** section, click **Replace Expression**.
5 From the menu, choose **Fluid-Structure Interaction>State variable Vpump (Vpump)**.
6 In the upper-right corner of the **y-Axis Data** section, click **Add Expression**.
7 From the menu, choose **Fluid-Structure Interaction>State variable netflow (netflow)**.
8 Click the **Plot** button.

The reaction forces from the fluid on the tube are very small compared to the force applied by the rollers. Therefore, it is not actually necessary to solve the two physics together like you just did. The following instructions demonstrate how to set up the
solvers to solve first for the structural mechanics and then for the fluid dynamics, which reduces memory consumption.

**MODEL WIZARD**

1. In the Model Builder window, right-click the root node and choose Add Study.
2. Go to the Model Wizard window.
3. In the Studies tree, select Preset Studies>Time Dependent.
4. Click Finish.

**STUDY 2**

*Step 1: Time Dependent*

1. In the Model Builder window, click Study 2>Step 1: Time Dependent.
2. Go to the Settings window for Time Dependent.
3. Locate the Study Settings section. In the Times edit field, type \( \text{range}(0,0.01,1.5) \).
4. Right-click Study 2>Step 1: Time Dependent and choose Duplicate.
5. Right-click Study 2 and choose Show Default Solver.
6. In the Model Builder window, expand the Study 2>Solver Configurations node.

**Solver 2**

1. In the Model Builder window, expand the Study 2>Solver Configurations>Solver 2 node, then click Dependent Variables 1.
2. Go to the Settings window for Dependent Variables.
3. Locate the General section. From the Defined by study step list, select User defined.
4. In the Model Builder window, expand the Dependent Variables 1 node, then click mod1_p.
5. Go to the Settings window for Field.
6. Locate the General section. Clear the Solve for this field check box.
7. Repeat Steps 4 through 6 for the variables mod1_rz, mod1_ODE1, mod1_ODE2 and mod1_fsi_vWall.

To make the solver use the solution for the deformation of the tube (Step 1) for the computation of the fluid flow, do the following:

1. In the Model Builder window, click Time-Dependent Solver 1.
2. Go to the Settings window for Time-Dependent Solver.
3. Click to expand the Time Stepping section.
4 From the **Steps taken by solver** list, select **Strict**.

5 In the **Model Builder** window, click **Dependent Variables 2**.

6 Go to the **Settings** window for **Dependent Variables**.

7 Locate the **General** section. From the **Defined by study step** list, select **User defined**.

8 Locate the **Values of Variables Not Solved For** section. From the **Solution** list, select **Solver 2**.

9 From the **None** list, select **All**.

10 In the **Model Builder** window, click **Dependent Variables 2**.

11 Go to the **Settings** window for **Dependent Variables**.

12 Locate the **General** section. From the **Defined by study step** list, select **User defined**.

13 In the **Model Builder** window, click **Dependent Variables 2>mod1_u_solid**.

14 Go to the **Settings** window for **Field**.

15 Locate the **General** section. Clear the **Solve for this field** check box.

16 In the **Model Builder** window, right-click **Study 2** and choose **Compute**.

**RESULTS**

*Flow and Stress (fsi) 1*

Again, you get a default surface plot of the velocity field and the Von Mises stress distribution at \( t = 1.5 \) s. Change this plot to show the deformation of the tube at \( t = 0.7 \) s, similar to **Figure 1**.

1 Go to the **Settings** window for **2D Plot Group**.

2 Locate the **Data** section. From the **Time** list, select **0.7**.

3 In the **Model Builder** window, expand the **Flow and Stress (fsi) 1** node, then click **Surface 1**.

4 Go to the **Settings** window for **Surface**.

5 In the upper-right corner of the **Expression** section, click **Replace Expression**.

6 From the menu, choose **Fluid-Structure Interaction (Solid Mechanics)>Total displacement (fsi.disp)**.

7 Locate the **Expression** section. From the **Unit** list, select **mm**.

8 In the **Model Builder** window, right-click **Surface 2** and choose **Disable**.

9 Right-click **Arrow Surface 1** and choose **Disable**.

10 Right-click **Flow and Stress (fsi) 1** and choose **Plot**.
II Click the **Zoom Extents** button on the Graphics toolbar.