

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 $\mu = 3 \cdot 10^{-5} \text{ Pa}\cdot\text{s}$. 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 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.

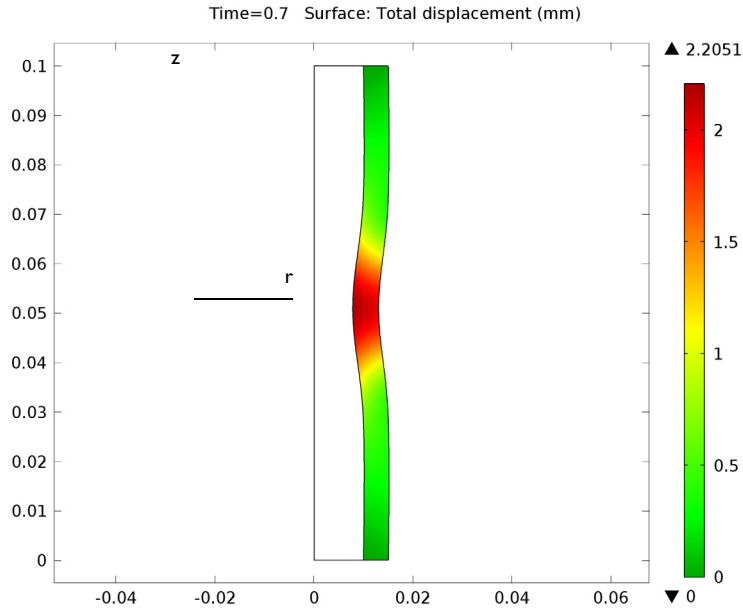


Figure 1: The geometry of the peristaltic pump as it is deforming under the pressure of the roll. The tube is rotationally symmetric with respect to the z -axis. The color shows the deformation of the tube material.

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:

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

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

where ρ denotes the density (SI unit: kg/m^3), \mathbf{u} the velocity (SI unit: m/s), μ the viscosity (SI unit: $\text{Pa}\cdot\text{s}$), 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:

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

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 \dot{V} in m^3/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: m^3), that is conveyed out of the outlet during a peristaltic cycle, which is equal to the following time integral:

$$V_{\text{pump}}(t) = \int_0^t \dot{V}_{\text{out}} dt' .$$

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

$$\frac{dV_{\text{pump}}}{dt} = \dot{V}_{\text{out}}$$

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

Results

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

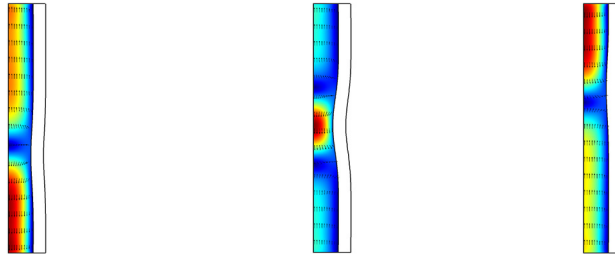


Figure 2: Snapshots of the velocity field and the shape of the inside of the tube at $t = 0.3$ s, $t = 0.7$ s, and $t = 1.2$ s. The colors represent the magnitude of the velocity, and the arrows its direction.

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.

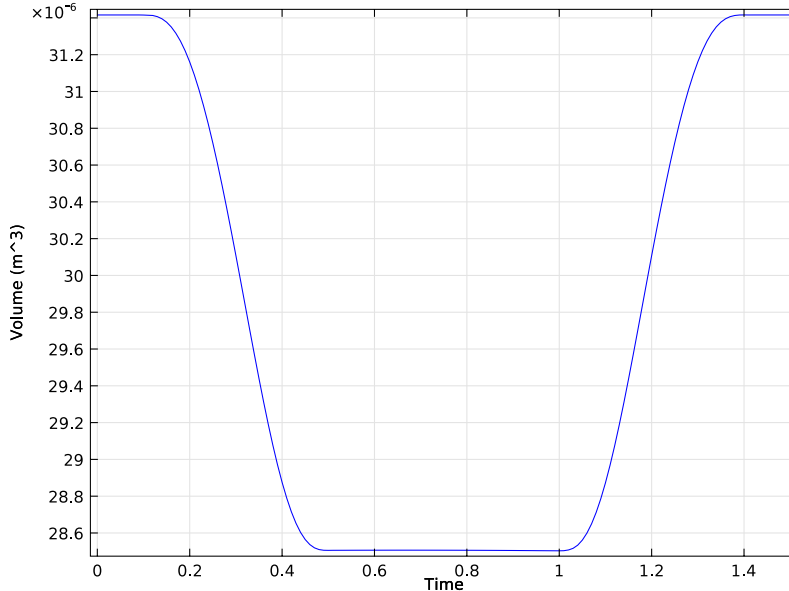


Figure 3: The inner volume (m^3) of the tube as a function of time (s).

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, $\dot{V}_{in} - \dot{V}_{out}$ and integrating over time, you generate Figure 3.

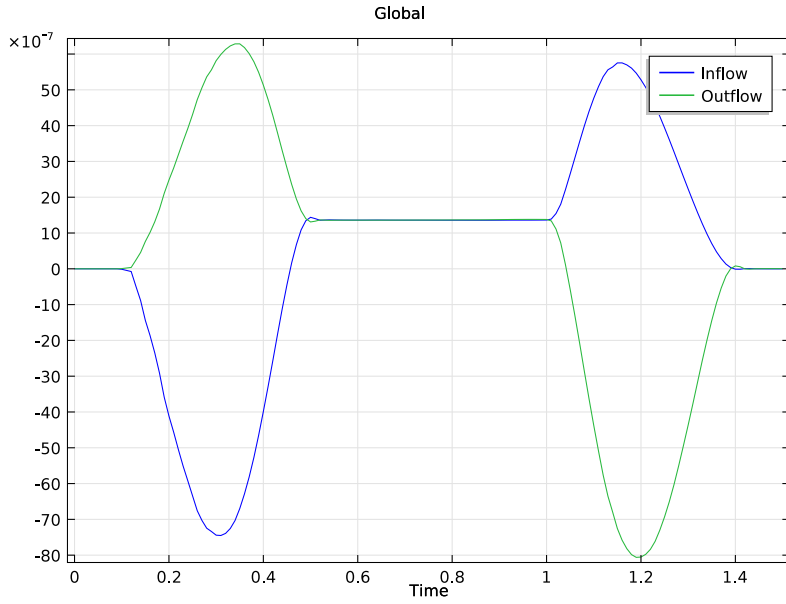


Figure 4: Inlet and outlet flow in m^3/s as functions of time. Positive values indicate that the gas is flowing in through the inlet and out through the outlet.

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.

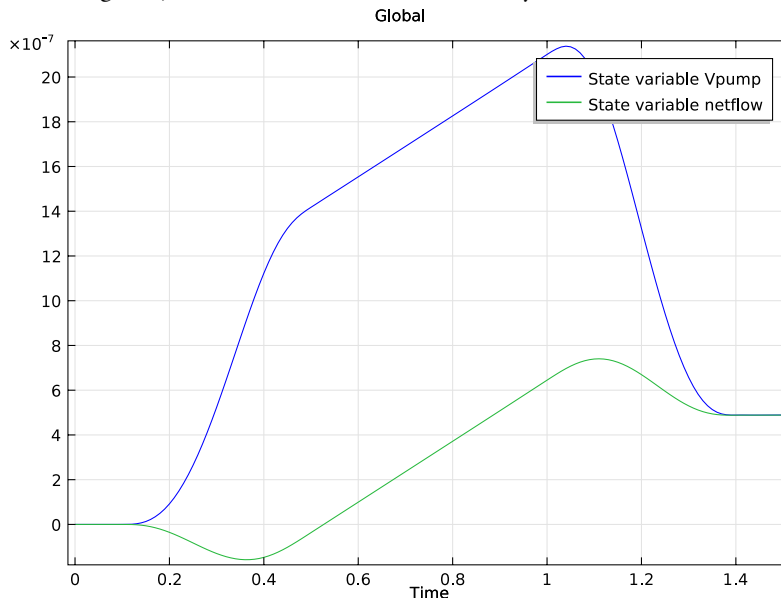


Figure 5: Accumulated flow (m^3) through the pump and volume of fluid conveyed out of the outlet versus time (s).

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 `f1c2hs` (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**.
- 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**.

GEOMETRY I

Rectangle 1

- 1 In the **Model Builder** window, right-click **Geometry I** and choose **Rectangle**.
- 2 In the **Rectangle** settings window, locate the **Size** section.
- 3 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 Right-click **Geometry I** and choose **Rectangle**.
- 2 In the **Rectangle** settings window, locate the **Size** section.
- 3 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 In the **Parameters** settings window, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Description
t_on	0.3[s]	Time when roll is engaged
t_off	1.2[s]	Time when roll is disengaged
dt	0.2[s]	Time to reach full force
z0	0.03[m]	z coordinate where roll starts
v0	0.03[m/s]	Vertical velocity of roll
width	0.01[m]	Width of Gaussian force distribution
Ttot	1.5[s]	Total time for a pump cycle
Lmax	1.e8[N/m^2]	Max load

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, under **Model 1** right-click **Definitions** and choose **Functions>Analytic**.
- 2 In the **Analytic** settings window, locate the **Function Name** section.
- 3 In the **Function name** edit field, type load.
- 4 Locate the **Parameters** section. In the **Expression** edit field, type $f1c2hs(t_off/dt - ts, 1) * f1c2hs(ts - t_on/dt, 1) * \exp(-(zs - (z0 + v0 * ts * dt) / width)^2 / 2)$.
- 5 In the **Arguments** edit field, type zs, ts .

Note that the function arguments are made dimensionless by $zs = z / width$ 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 In the **Integration** settings window, locate the **Source Selection** section.

- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundary 2 only.

Integration 2

- 1 Right-click **Definitions** and choose **Model Couplings>Integration**.
- 2 In the **Integration** settings window, locate the **Source Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundary 3 only.

Variables 1

- 1 Right-click **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
inflow	$\text{intop1}(2*\pi*r*w_fluid)$	Inflow
outflow	$\text{intop2}(2*\pi*r*w_fluid)$	Outflow

MATERIALS

Material Browser

- 1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Open Material Browser**.
- 2 In the **Material Browser** window, locate the **Materials** section.
- 3 In the tree, select **Built-In>Nylon**.
- 4 Right-click and choose **Add Material to Model** from the menu.

Nylon

- 1 In the **Model Builder** window, under **Model 1>Materials** click **Nylon**.
- 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
Poisson's ratio	nu	0.33

Material 2

- 1 In the **Model Builder** window, 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	1
Dynamic viscosity	mu	1e-3

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**.

Linear Elastic Material 1

- 1 In the **Model Builder** window, expand the **Fluid-Structure Interaction** node, then click **Linear Elastic Material 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 In the **Boundary Load** settings window, locate the **Force** section.
- 4 In the \mathbf{F}_A table, enter the following settings:

-Lmax*load(z/width, t/dt)	r
0	z

Fixed Constraint 1

- 1 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 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 Right-click **Fluid-Structure Interaction** and choose **Prescribed Mesh Displacement**.
- 2 Select Boundaries 2 and 3 only.
- 3 In the **Prescribed Mesh Displacement** settings window, locate the **Prescribed Mesh Displacement** section.
- 4 Clear the **Prescribed r displacement** check box.

Prescribed Mesh Displacement 3

- 1 Right-click **Fluid-Structure Interaction** and choose **Prescribed Mesh Displacement**.
- 2 Select Boundary 1 only.
- 3 In the **Prescribed Mesh Displacement** settings window, locate the **Prescribed Mesh Displacement** section.
- 4 Clear the **Prescribed z displacement** check box.

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

- 5 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

Global Equations 1

- 1 Right-click **Fluid-Structure Interaction** and choose **Global>Global Equations**.
- 2 In the **Global Equations** settings window, locate the **Global Equations** section.
- 3 In the table, enter the following settings:

Name	$f(u,ut,utt,t)$
netflow	$netflowt - (outflow + inflow) / 2$

Global Equations 2

- 1 Right-click **Fluid-Structure Interaction** and choose **Global>Global Equations**.
- 2 In the **Global Equations** settings window, locate the **Global Equations** section.
- 3 In the table, enter the following settings:

Name	$f(u,ut,utt,t)$
Vpump	$Vpumpt - outflow$

STUDY 1*Step 1: Time Dependent*

- 1 In the **Model Builder** window, expand the **Study 1** 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, 0.01, 1.5).
- 4 Select the **Relative tolerance** check box.
- 5 In the associated edit field, type 0.001.

Solver 1

- 1 In the **Model Builder** window, right-click **Study 1** and choose **Show Default Solver**.
- 2 In the **Time-Dependent Solver** settings window, click to expand the **Absolute Tolerance** section.
- 3 Clear the **Update scaled absolute tolerance** check box.
- 4 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 $t = 1.5$ s. To plot the total displacement at $t = 0.7$ s ([Figure 1](#)), follow these steps:

- 1 In the **Model Builder** window, under **Results>Flow and Stress (fsi)** click **Surface 1**.
- 2 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Fluid-Structure Interaction (Solid Mechanics)>Displacement>Total displacement (fsi.disp)**.
- 3 Locate the **Expression** section. From the **Unit** list, choose **mm**.
- 4 In the **Model Builder** window, under **Results>Flow and Stress (fsi)** right-click **Surface 2** and choose **Disable**.
- 5 Right-click **Results>Flow and Stress (fsi)>Arrow Surface 1** and choose **Disable**.
- 6 In the **Model Builder** window, click **Flow and Stress (fsi)**.
- 7 In the **2D Plot Group** settings window, locate the **Data** section.
- 8 From the **Time** list, choose **0.7**.
- 9 Click the **Plot** button.
- 10 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:

- 11** In the **Model Builder** window, under **Results>Flow and Stress (fsi)** right-click **Surface 1** and choose **Disable**.
- 12** Right-click **Results>Flow and Stress (fsi)>Surface 2** and choose **Enable**.
- 13** Right-click **Results>Flow and Stress (fsi)>Arrow Surface 1** and choose **Enable**.
- 14** In the **Model Builder** window, click **Flow and Stress (fsi)**.
- 15** In the **2D Plot Group** settings window, locate the **Data** section.
- 16** From the **Time** list, choose **0.3**.
- 17** Click the **Plot** button.
- 18** In the **Model Builder** window, click **Flow and Stress (fsi)**.
- 19** In the **2D Plot Group** settings window, locate the **Data** section.
- 20** From the **Time** list, choose **0.7**.
- 21** Click the **Plot** button.
- 22** In the **Model Builder** window, click **Flow and Stress (fsi)**.
- 23** In the **2D Plot Group** settings window, locate the **Data** section.
- 24** From the **Time** list, choose **1.2**.
- 25** Click the **Plot** button.

Export

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

- 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** Click the **Generate Frame** button.
- 5** Right-click **Results>Export>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, under **Results** right-click **Derived Values** and choose **Integration>Surface Integration**.
- 2** Select Domain 1 only.

- 3 In the **Surface Integration** settings window, locate the **Expression** section.
- 4 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. Select the **Compute volume integral** check box.
- 8 Click the **Evaluate** button.
- 9 In the **Results** window, click **Graph 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.

ID Plot Group 5

- 1 In the **Model Builder** window, right-click **Results** and choose **ID Plot Group**.
- 2 Right-click **ID Plot Group 5** 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>Inflow (inflow)**.
- 4 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Definitions>Outflow (outflow)**.
- 5 Click the **Plot** button.

ID Plot Group 6

- 1 In the **Model Builder** window, right-click **Results** and choose **ID Plot Group**.
- 2 Right-click **ID Plot Group 6** 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 **Fluid-Structure Interaction>State variable Vpump (Vpump)**.
- 4 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Fluid-Structure Interaction>State variable netflow (netflow)**.
- 5 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.

- 1 In the **Model Builder** window, right-click the root node and choose **Add Study**.

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Find the **Studies** subsection. In the tree, select **Preset Studies>Time Dependent**.
- 3 Click **Finish**.

STUDY 2

Step 1: Time Dependent

- 1 In the **Model Builder** window, under **Study 2** 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, 0.01, 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 In the **Dependent Variables** settings window, locate the **General** section.
- 3 From the **Defined by study step** list, choose **User defined**.
- 4 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solver 2>Dependent Variables 1** node, then click **mod1.p**.
- 5 In the **Field** settings window, locate the **General** section.
- 6 Clear the **Solve for this field** check box.
- 7 Repeat Steps 4 through 6 for the variables `mod1_u_fluid`, `mod1_rz`, `mod1_ODE1`, `mod1_ODE2`, and `mod1_fsi_wall`.

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:

- 8 In the **Model Builder** window, under **Study 2>Solver Configurations>Solver 2** click **Time-Dependent Solver 1**.
- 9 In the **Time-Dependent Solver** settings window, click to expand the **Time Stepping** section.
- 10 From the **Steps taken by solver** list, choose **Strict**.

- 11 In the **Model Builder** window, under **Study 2>Solver Configurations>Solver 2** click **Dependent Variables 2**.
- 12 In the **Dependent Variables** settings window, locate the **General** section.
- 13 From the **Defined by study step** list, choose **User defined**.
- 14 Locate the **Values of Variables Not Solved For** section. From the **Solution** list, choose **Solver 2**.
- 15 From the **Selection** list, choose **All**.
- 16 In the **Model Builder** window, under **Study 2>Solver Configurations>Solver 2** click **Dependent Variables 2**.
- 17 In the **Dependent Variables** settings window, locate the **General** section.
- 18 From the **Defined by study step** list, choose **User defined**.
- 19 In the **Model Builder** window, under **Study 2>Solver Configurations>Solver 2>Dependent Variables 2** click **mod1.u_solid**.
- 20 In the **Field** settings window, locate the **General** section.
- 21 Clear the **Solve for this field** check box.
- 22 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, just like [Figure 1](#).

- 1 In the **2D Plot Group** settings window, locate the **Data** section.
- 2 From the **Time** list, choose **0.7**.
- 3 In the **Model Builder** window, expand the **Flow and Stress (fsi) 1** node, then click **Surface 1**.
- 4 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Fluid-Structure Interaction (Solid Mechanics)>Displacement>Total displacement (fsi.disp)**.
- 5 Locate the **Expression** section. From the **Unit** list, choose **mm**.
- 6 In the **Model Builder** window, under **Results>Flow and Stress (fsi) 1** right-click **Surface 2** and choose **Disable**.
- 7 Right-click **Results>Flow and Stress (fsi) 1>Arrow Surface 1** and choose **Disable**.

- 8 Right-click **Flow and Stress (fsi) I** and choose **Plot**.
- 9 Click the **Zoom Extents** button on the Graphics toolbar.