Created in COMSOL Multiphysics 5.6

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 can be easily damaged. Peristaltic pumps are therefore used in medical applications, one of which is to move the blood through the body during open heart surgery. Other types of pumps would risk destroying the blood cells.

In this COMSOL Multiphysics example, a peristaltic pump is analyzed by combining structural mechanics (to model the squeezing of the tube) and fluid dynamics (to compute the fluid's motion). Thus, it is an example of a fluid-structure interaction (FSI) problem.

Model Definition

The analysis is set up in 2D axial symmetry (Figure 1). A nylon tube 0.1 m long has an inner radius of 1 cm and an outer radius of 1.5 cm; it contains fluid with the density $\rho = 1\cdot10^3\ \mathrm{kg/m^3}$ and viscosity $\mu = 5\cdot10^{-3}$ 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 operation. For the sake of simplicity, 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 $t = 0.1$ s and takes the force to its full development at $t = 0.5$ s. Likewise, the disengagement of the roll starts at $t = 1.0$ s and ends at $t = 1.4$ s. The example models the tube's deformation during a full cycle of 1.5 s.

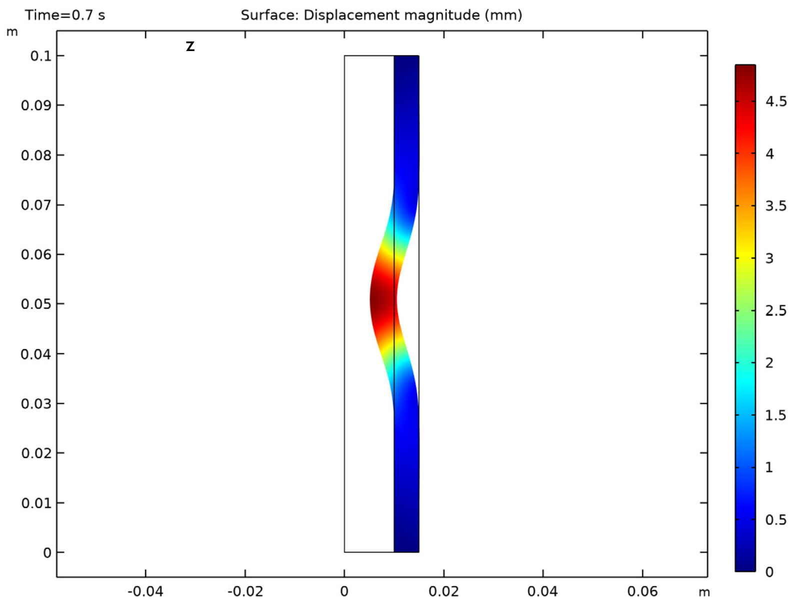

Figure 1: The geometry of the peristaltic pump as it is deforming under the pressure of the roll. The tube is rotationally symmetric with respect to the z-axis. The color shows the deformation of the tube material.

DOMAIN EQUATIONS

The structural mechanics computations use the assumption that the material is linear elastic, and they take geometric nonlinearities into account.

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

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

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

where ρ denotes the density (SI unit: kg/m^3), $\mathbf{u}$ the velocity (SI unit: m/s), μ the viscosity (SI unit: Pa·s), and p the pressure (SI unit: Pa). The equations are set up and solved 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 Hyperelastic smoothing. On the solid-fluid boundary at the tube's inner wall, the moving mesh follows the structural deformation. For more information, please refer to the chapter *Fluid-Structure Interaction* in the *Structural Mechanics Module User's Guide*.

BOUNDARY CONDITIONS

For the structural mechanics computations, the time- and coordinate-dependent load is prescribed as the boundary condition at the tube's outer surface. This is the load that drives the pump operation. 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:

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

The mesh has zero z displacement at the top and the bottom of the tube.

At the fluid-solid boundary, the structural velocity is transmitted to the fluid. As a feedback, the stresses in the fluid flow act as a loading on the inner boundary of the solid wall of the tube.

COMPUTATION OF VOLUMETRIC FLOW RATES AND TOTAL VOLUME OF PUMPED FLUID

The model's dependent variables are the displacements of the tube wall together with the fluid 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 need to perform some additional calculations. To obtain the volumetric flow rate at any instant t, compute a boundary integral over the pipe's inlet and outlet boundary:

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

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

where $\mathbf{n}$ is the outward-pointing unit normal of the boundary, $\mathbf{u}$ is 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 the flow.

It is of interest to track how much fluid is conveyed through the outlet during a peristaltic cycle, This can be calculated as 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 will integrate this equation.

Results

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

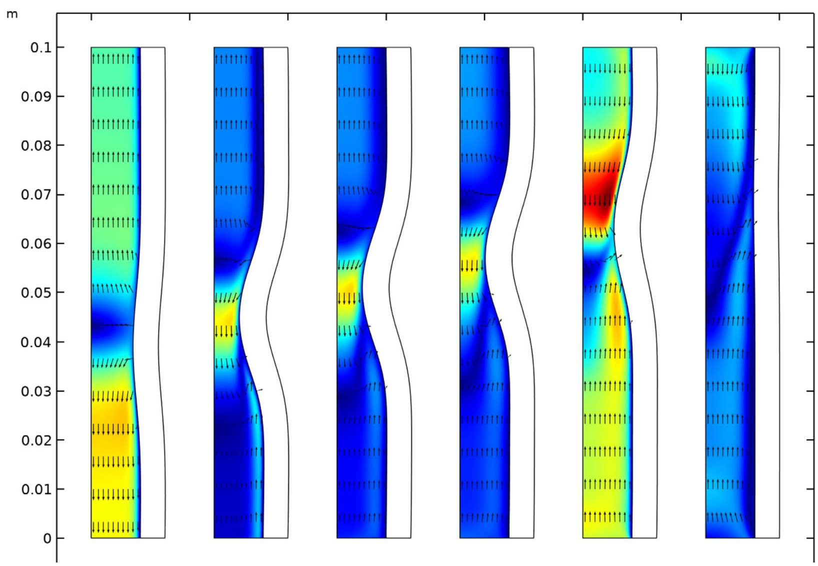

Figure 2: Snapshots of the velocity field and the shape of the inside of the tube at t = 0.3 s, t = 0.5 s, t = 0.7 s, t = 0.9 s, t = 1.1 s and t = 1.3 s. The colors represent the magnitude of the velocity, and the arrows its direction.

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 it is increasing its pressure on the tube. As less and less space is left for the fluid, it is streaming out of the tube, through both the inlet and the outlet. At t = 0.5 s, the roll has been fully engaged for a while. As it is moving upward along the tube, so does the fluid, 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 = 1.3 s, the engagement process is reversed, and the roll is disengaging. As a result, the fluid is streaming into the tube from both ends.

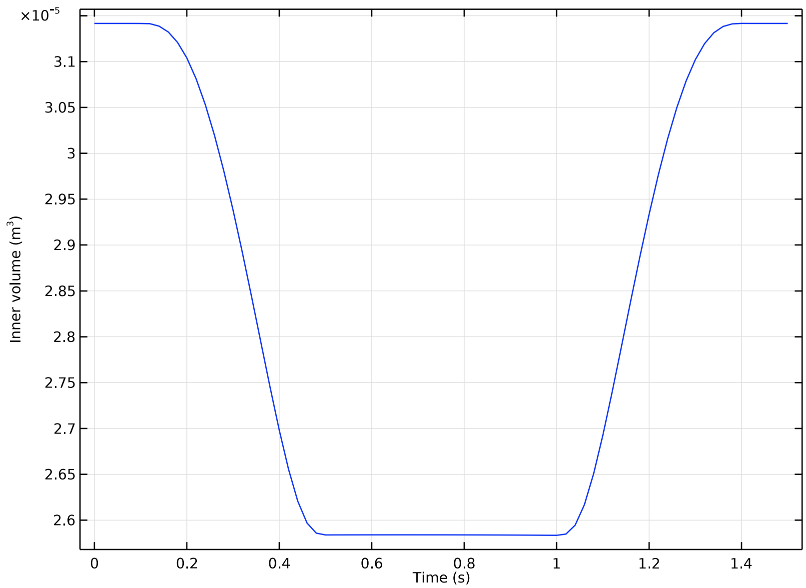

Figure 3: The inner volume (m^3) of the tube as a function of time (s).

Figure 4 shows the inlet and outlet flows, and it confirms the overall behavior indicated in the velocity snapshots. 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, $\dot{V}_{\text{in}} - \dot{V}_{\text{out}}$ and integrating over time, you generate Figure 3.

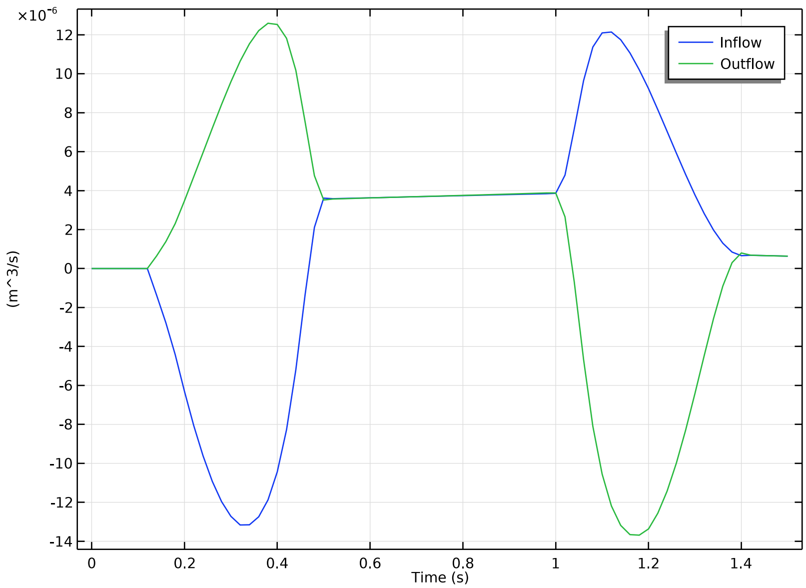

Figure 4: Inlet and outlet flow in m^3/s *as functions of time. Positive values indicate that the fluid is flowing in through the inlet and out through the outlet.*

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.

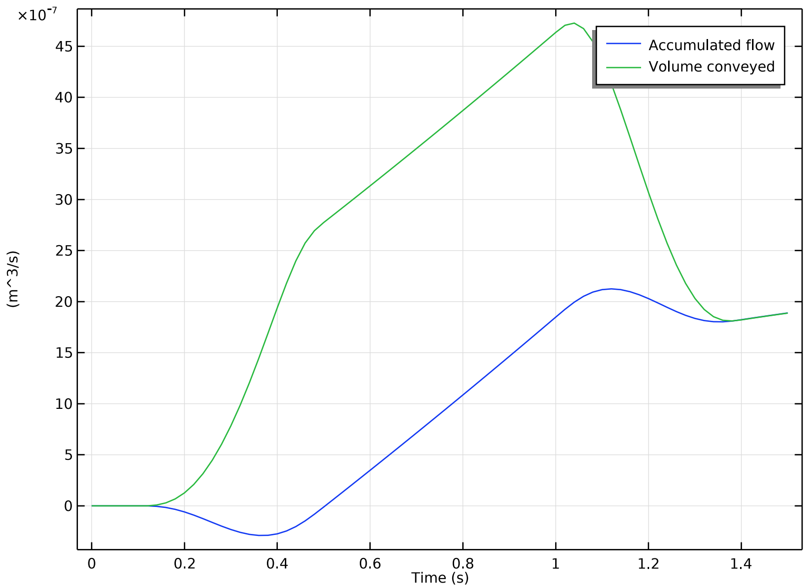

Figure 5: Accumulated flow (m^3) through the pump and volume of fluid conveyed out of the outlet versus time (s).

Notes About the COMSOL Implementation

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

Application Library path: `Structural_Mechanics_Module/Fluid-Structure_Interaction/peristaltic_pump`

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 Axisymmetric**.

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**.

GEOMETRY 1

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 `0.01`.

4 In the **Height** text field, type `0.1`.

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 `5e-3`.

4 In the **Height** text field, type `0.1`.

5 Locate the **Position** section. In the **r** text field, type `0.01`.

6 Click **Build All Objects**.

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
t_on	0.3[s]	0.3 s	Time when roll is engaged
t_off	1.2[s]	1.2 s	Time when roll is disengaged
dt	0.2[s]	0.2 s	Time to reach full force
z0	0.03[m]	0.03 m	z coordinate where roll starts
v0	0.03[m/s]	0.03 m/s	Vertical velocity of roll
width	0.01[m]	0.01 m	Width of Gaussian force distribution
Ttot	1.5[s]	1.5 s	Total time for a pump cycle
Lmax	1.5e8[N/m^2]	1.5E8 N/m²	Max load

DEFINITIONS

Follow the steps given below to define the force density of the load applied to the outer wall of the tube.

Analytic 1 (an1)

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

2 In the **Settings** window for **Analytic**, type `load` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `flc2hs(t_off/dt-ts,1)*flc2hs(ts-t_on/dt,1)*exp(-(zs-(z0+v0*ts*dt)/width)^2/2)`.

4 In the **Arguments** text 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 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 2 only.

5 Locate the **Advanced** section. Clear the **Compute integral in revolved geometry** check box.

Integration 2 (intop2)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 3 only.

5 Locate the **Advanced** section. Clear the **Compute integral in revolved geometry** check box.

Variables 1

1 In the **Definitions** toolbar, click **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
inflow	intop1(2*pi*r*w_fluid)	m³/s	Inflow
outflow	intop2(2*pi*r*w_fluid)	m³/s	Outflow

Deforming Domain 1

1 In the **Model Builder** window, click **Deforming Domain 1**.

2 Select Domain 1 only.

LAMINAR FLOW (SPF)

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

2 Select Domain 1 only.

Open Boundary 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Open Boundary**.

2 Select Boundaries 2 and 3 only.

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

3 Click the **Show More Options** button in the **Model Builder** toolbar.

4 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation-Based Contributions**.

5 Click **OK**.

Global Equations 1

1 In the **Physics** toolbar, click **Global** and choose **Global Equations**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
netflow	netflowt-(outflow+inflow)/2	0	0	Accumulated flow

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `volume` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Volume (m^3)**.

8 Click **OK**.

9 In the **Settings** window for **Global Equations**, locate the **Units** section.

10 Click **Select Source Term Quantity**.

11 In the **Physical Quantity** dialog box, type `volumepertime` in the text field.

12 Click **Filter**.

13 In the tree, select **General>Volume per time (m^3/s)**.

14 Click **OK**.

Global Equations 2

1 In the **Physics** toolbar, click **Global** and choose **Global Equations**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
Vpump	Vpumpt-outflow	0	0	Volume conveyed

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `volume` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Volume (m^3)**.

8 Click **OK**.

9 In the **Settings** window for **Global Equations**, locate the **Units** section.

10 Click **Select Source Term Quantity**.

11 In the **Physical Quantity** dialog box, type `volumepertime` in the text field.

12 Click **Filter**.

13 In the tree, select **General>Volume per time (m^3/s)**.

14 Click **OK**.

SOLID MECHANICS (SOLID)

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

2 Select Domain 2 only.

Linear Elastic Material 1

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

Damping 1

1 In the **Physics** toolbar, click **Attributes** and choose **Damping**.

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

3 In the α_{dM} text field, type `1e-2`.

4 In the β_{dK} text field, type `1e-3`.

Fixed Constraint 1

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

2 Select Boundaries 5 and 6 only.

Boundary Load 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundary 7 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

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

DEFINITIONS

Symmetry/Roller 1

1 In the **Definitions** toolbar, click **Moving Mesh** and choose **Symmetry/Roller**.

2 Select Boundaries 2 and 3 only.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

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

3 In the tree, select **Built-in>Nylon**.

4 Click **Add to Component** in the window toolbar.

5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Nylon (mat1)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Manual**.

3 Select Domain 2 only.

Material 2 (mat2)

1 In the **Model Builder** window, 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/m^3	Basic
Dynamic viscosity	mu	5e-3	Pa·s	Basic

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(0,0.02,1.5)`.

Get the initial values, which will also generate the default plot to be shown while solving.

4 Right-click **Study 1>Step 1: Time Dependent** and choose **Get Initial Value for Step**.

STUDY 1

Step 1: Time Dependent

1. In the **Model Builder** window, expand the **Results>Datasets** node, then click **Study 1> Step 1: Time Dependent**.
2. In the **Settings** window for **Time Dependent**, click to expand the **Results While Solving** section.
3. Select the **Plot** check box.
4. In the **Home** toolbar, click **Compute**.

RESULTS

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

Displacement (solid)

1. In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.
2. In the **Settings** window for **2D Plot Group**, type `Displacement (solid)` in the **Label** text field.
3. Locate the **Data** section. From the **Time (s)** list, choose **0.7**.

Surface 1

1. Right-click **Displacement (solid)** and choose **Surface**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.
3. In the **Expression** text field, type `solid.disp`.
4. From the **Unit** list, choose **mm**.
5. In the **Displacement (solid)** toolbar, click **Plot**.

Velocity (spf)

Animate the velocity field as a function of time, as shown in Figure 2.

1. In the **Model Builder** window, click **Velocity (spf)**.

Animation 1

In the **Velocity (spf)** toolbar, click **Animation** and choose **Player**.

Velocity (spf)

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

Surface Integration 1

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

2 Right-click **Derived Values** and choose **Integration>Surface Integration**.

3 Select Domain 1 only.

4 In the **Settings** window for **Surface Integration**, locate the **Expressions** section.

5 In the table, enter the following settings:

Expression	Unit	Description
1	m^3	Inner volume

6 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

2 Click the right end of the **Display Table 1 - Surface Integration 1** split button in the window toolbar.

3 From the menu, choose **Table Graph**.

RESULTS

Table Graph 1

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 9

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

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **y-axis label** check box.

4 In the associated text field, type `(m^3/s)`.

Global 1

1 Right-click **1D Plot Group 9** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
inflow	m^3/s	Inflow
outflow	m^3/s	Outflow

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

5 In the **1D Plot Group 9** toolbar, click **Plot**.

1D Plot Group 10

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

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **y-axis label** check box.

4 In the associated text field, type `(m^3/s)`.

Global 1

1 Right-click **1D Plot Group 10** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`netflow`	`m^3`	`Accumulated flow`
`Vpump`	`m^3`	`Volume conveyed`

4 Locate the **Title** section. From the **Title type** list, choose **None**.

5 In the **1D Plot Group 10** toolbar, click **Plot**.

Flow and Stress, 3D

1 In the **Model Builder** window, under **Results** click **Velocity, 3D (spf)**.

2 In the **Settings** window for **3D Plot Group**, type `Flow and Stress, 3D` in the **Label** text field.

3 Locate the **Data** section. From the **Time (s)** list, choose **0.7**.

Surface 2

1 Right-click **Flow and Stress, 3D** and choose **Surface**.

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

3 In the **Expression** text field, type `solid.mises`.

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

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **Traffic**.

6 In the **Flow and Stress, 3D** toolbar, click **Plot**.

7 Click the **Zoom Extents** button in the **Graphics** toolbar.

The resulting plot should be similar to the one shown in the following figure: