COMSOL MULTIPHYSICS®

Peristaltic Pump

SOLVED WITH COMSOL MULTIPHYSICS 3.5a

© COPYRIGHT 2008. All right reserved. No part of this documentation may be photocopied or reproduced in any form without prior written consent from COMSOL AB. COMSOL, COMSOL Multiphysics, COMSOL Reaction Engineering Lab, and FEMLAB are registered trademarks of COMSOL AB. Other product or brand names are trademarks or registered trademarks of their respective holders.

COMSOL

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. Because deformations of the tube drive the fluid, you must perform the fluid portion of the simulation in a deforming geometry, one that you can define with COMSOL Multiphysics' Moving Mesh (ALE) application mode.

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 $\rho = 1$ kg/m^3 and viscosity $\eta = 3 \cdot 10^{-5}$ 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 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.

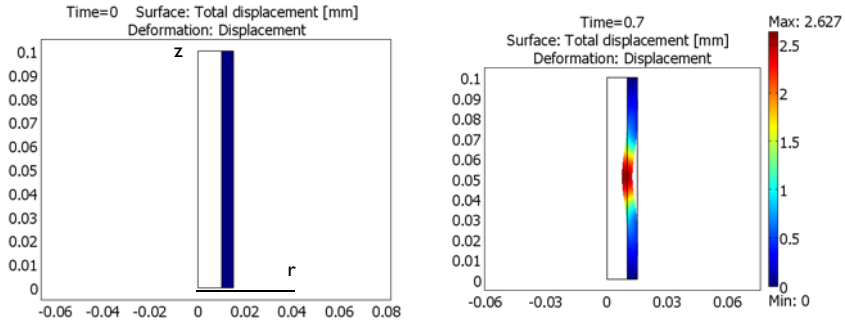

Figure 1: The geometry of the peristaltic pump in its initial shape (left) and as it is deforming under the pressure of the roll (right). The tube is rotationally symmetric with respect to the z-axis. The color shows the deformation of the tube material.

DOMAIN EQUATIONS

The structural mechanics computations in this model use a 2D axially symmetric stress-strain formulation. The model uses the assumption that the material is linear and elastic and does not take geometric nonlinearities into account. With the capabilities in the COMSOL Multiphysics Structural Mechanics Module, you can easily extend this model to include large deformations or use a more complicated material model.

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

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

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

Further, you model the motion of the deformed mesh with Winslow smoothing. For more information on this topic, please refer to "The Moving Mesh Application Mode" on page 455 in the *COMSOL Multiphysics Modeling 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, and the inner surface is free. The reaction forces from the fluid on the tube are very small compared

to the force applied by the rollers, so this model does not take them into account. This way you can solve first for the structural mechanics and then for the fluid dynamics, reducing solution time and memory consumption.

The inner surface is free, and 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:

$$\mathbf{n} \cdot [-pI + \eta(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)] = \mathbf{0} \tag{2}$$

Inside the wall of the tube, the moving mesh follows the deformations of the tube. The mesh deformations being the same as the structural deformations is also used as the moving mesh boundary condition where the fluid meets the tube wall. 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:

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

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

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

In a similar manner, you can integrate the total volume of fluid (m^3) contained in the pump as an integral over the fluid domain, A, with the equation

$$V = \int_A 2\pi r dA$$

Note that A is the deformed domain due to the deformation of the wall.

Finally, it is of interest to track how much volume of fluid, $V_{\text{pump}}(t)$ (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.

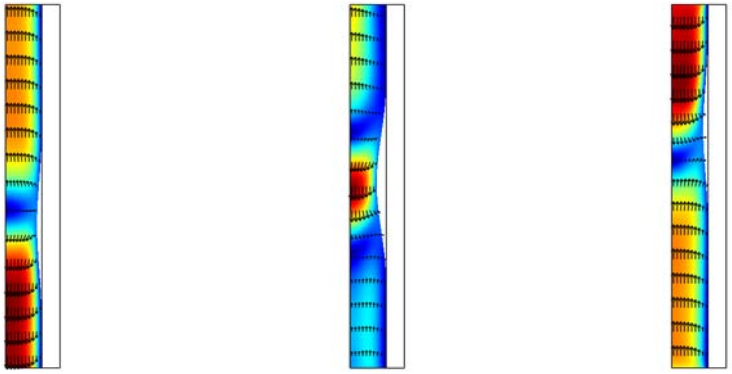

Figure 2: Snapshots of the velocity field and the shape of the inside of the tube at t = 0.3 s, t = 0.7 s, and t = 1.2 s. The colors represent the magnitude of the velocity, and the arrows its direction.

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.

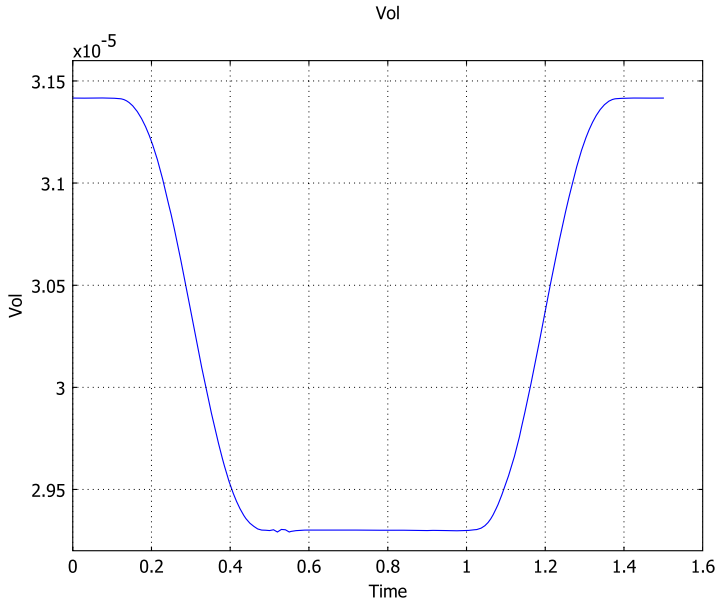

Figure 3: The inner volume (m^3) of the tube as a function of time (s).

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

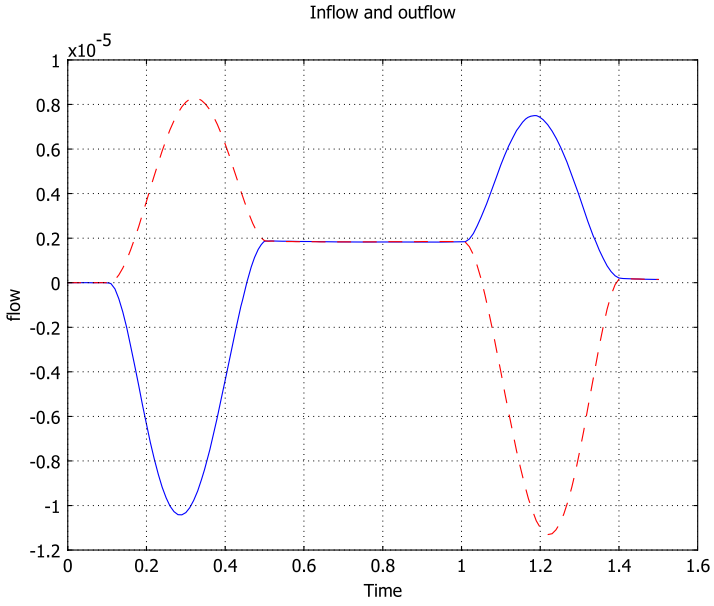

Figure 4: Inlet (blue solid line) and outlet (red dashed line) flow in m^3/s as functions of time. Positive values indicate that the gas is flowing in through the inlet and out through the outlet.

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.

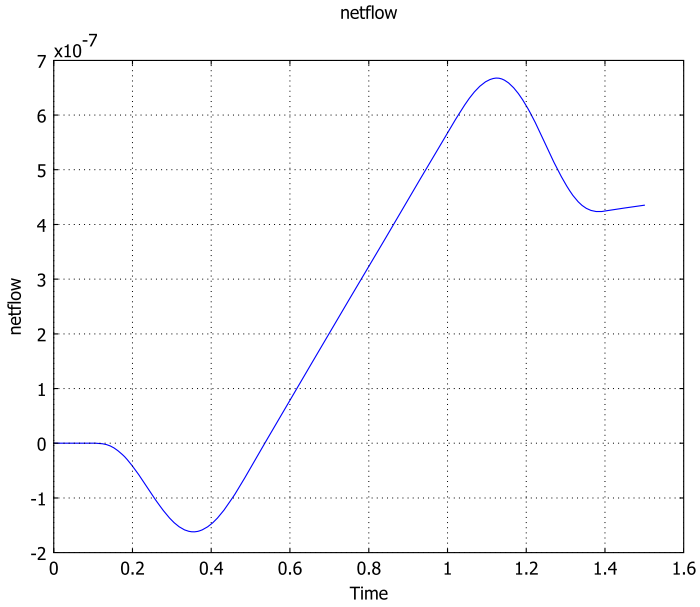

Figure 5: Accumulated flow (m^3) through the pump versus time (s).

Modeling in COMSOL Multiphysics

This model is primarily intended to demonstrate the use of the Moving Mesh (ALE) application mode, but it also shows some postprocessing features. It defines integration coupling variables to integrate the total flow rate and to compute the volume inside the tube. 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).

Model Library path: `COMSOL_Multiphysics/Multiphysics/peristaltic_pump`

1 Start COMSOL Multiphysics.

2 In the **Model Navigator,** set the **Space dimension** to **Axial symmetry (2D)** and click the **Multiphysics** button.

3 Select **COMSOL Multiphysics>Structural Mechanics>Axial Symmetry, Stress-Strain>Transient analysis** in the list of application modes and click **Add**.

4 Select **COMSOL Multiphysics>Deformed Mesh>Moving Mesh (ALE)>Transient analysis** and click **Add**.

5 Click the **Application Mode Properties** button and set the **Smoothing method** to **Winslow**. Click **OK**.

6 Select **COMSOL Multiphysics>Fluid Dynamics>Incompressible Navier-Stokes>Transient analysis** and click **Add**.

7 Click **OK** to close the **Model Navigator**.

GEOMETRY MODELING

1 Specify two rectangles according to the table below.

NAME	WIDTH	HEIGHT	BASE	R	Z
R1	0.01	0.1	Corner	0	0
R2	5e-3	0.1	Corner	0.01	0

2 Click the **Zoom Extents** button on the Main toolbar.

OPTIONS AND SETTINGS

1 Open the **Constants** dialog box from the **Options** menu.

2 Enter constants according to the following table (the descriptions are optional); when done, click **OK**.

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

NAME	EXPRESSION	DESCRIPTION
Ttot	1.5[s]	Total time for a pump cycle
Lmax	4e6[N/m^2]	Max load

3 From the **Options** menu, choose **Functions**.

4 Define a new function for the Gaussian pulse according to following table; when done, click **OK**.

NAME	ARGUMENTS	EXPRESSION
gauss	x, sigma	1/(sigma*sqrt(2*pi))*exp(-x^2/(2*sigma^2))

5 Go to **Physics>Global Equations** and define an ordinary differential equation (see "Computation of Volumetric Flow Rates and Total Volume of Pumped Fluid" on page 3) with the following settings; when done, click **OK**.

NAME (U)	EQUATION	INIT(U)	INIT (UT)
netflow	netflowt-(inflow+outflow)/(2*Ttot)	0	0

6 Select **Integration Coupling Variables>Boundary Variables** from the **Options** menu and specify the following boundary integration variables for the inflow and outflow, each on a separate row in the table; when done, click **OK**.

SOURCE BOUNDARY	NAME	EXPRESSION	INTEGRATION ORDER	GLOBAL DESTINATION
2	inflow	2*pi*r*v	4	Selected
3	outflow	2*pi*r*v	4	Selected

7 Select **Integration Coupling Variables>Subdomain Variables** from the **Options** menu and specify the following subdomain integration variable for the volume; when done, click **OK**.

SOURCE SUBDOMAIN	NAME	EXPRESSION	INTEGRATION ORDER	GLOBAL DESTINATION
1	Vol	2*pi*r	4	Selected

PHYSICS SETTINGS—STRESS-STRAIN

Subdomain Settings

1 In the **Multiphysics** menu, select **Axial Symmetry, Stress-Strain**.

2 From the **Physics** menu, open the **Subdomain Settings** dialog box.

3 Select Subdomain 1 and clear the **Active in this domain** check box.

4 Select Subdomain 2 and click **Load**. Select **Nylon** from the **Basic Material Properties** folder in the **Materials** list. Click **OK**.

5 Click **OK** to close the **Subdomain Settings** dialog box.

Boundary Conditions

Open the **Boundary Settings** dialog box from the **Physics** menu.

6 Apply the following boundary settings.

SETTINGS	BOUNDARIES 5, 6	BOUNDARY 7
Constraint R_R	0	
Constraint R_Z	0	
Load F_R		See below
Load F_Z		0

7 For the F_R load on Boundary 7, enter the following expression:

```
-Lmax*flc2hs((t-t_on)[1/s],dt[1/s])*gauss((z-z0-v0*t)[1/m],
width[1/m])*flc2hs((t_off-t)[1/s],dt[1/s])
```

The purpose of the unit conversions `[1/s]` and `[1/m]` is to make the input to the `flc2hs` and `gauss` functions dimensionless.

8 Click **OK**.

MESH GENERATION

1 Open the **Free Mesh Parameters** dialog box from the **Mesh** menu.

2 On the **Global** page, click the **Custom mesh size** button and set the **Maximum element size** to `2e-3`.

3 Click **Remesh**, then click **OK**.

COMPUTING THE SOLUTION

1 In the **Solver Manager**, go to the **Solve For** tab and select **Axial Symmetry, Stress-Strain**. Only this application mode and its solution variables **uor** and **w** should be selected. Click **OK** to close the dialog box.

2 In the **Solver Parameters** dialog box, select **Time dependent** from the **Solver** list. Set **Times** to `range(0,0.01,1.5)`, **Relative tolerance** to `0.001` and **Absolute tolerance** to `1e-5`.

3 On the **Time Stepping** tab, set **Time steps taken by solver** to **Strict**.

4 Select **BDF** in the **Method** list.

5 Click **OK,** then click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The default plot shows the von Mises effective stress distribution.

1. In the **Plot Parameters** dialog box, click the **Deform** tab.
2. Select the **Deformed shape plot** check box and set the **Scale factor** to 1.
3. Use the **Solution at time** list on the **General** tab and click **Apply** to study the stress distribution and the deformation of the tube at a few points in time.

PHYSICS SETTINGS—INCOMPRESSIBLE NAVIER-STOKES

Subdomain Settings

1. In the **Multiphysics** menu, select **Incompressible Navier-Stokes**.
2. Open the **Subdomain Settings** dialog box. Select Subdomain 2 and clear the **Active in this domain** check box.
3. In Subdomain 1, set ρ to `1` (kg/m^3) and η to `3e-5` (Pa·s).
4. Click **OK** to close the **Subdomain Settings** dialog box.

Boundary Conditions

1. Open the **Boundary Settings** dialog box from the **Physics** menu.
2. Apply the following boundary conditions; when done, click **OK**.

SETTINGS	BOUNDARY 1	BOUNDARY 2	BOUNDARY 3	BOUNDARY4
Type	Symmetry	Open	Open	Inlet
Condition	Axial symmetry	Normal stress	Normal stress	Velocity
u_0				`uaxi_t_axi`
v_0				`wt`
f_0		0	0	

PHYSICS SETTINGS—MOVING MESH

Subdomain Settings

1. In the **Multiphysics** menu, select **Moving Mesh (ALE)**.
2. Open the **Subdomain Settings** dialog box. Keep the default **Free displacement** in Subdomain 1.
3. In Subdomain 2, select **Prescribed displacement**. Enter `uaxi_axi` for **dr** and `w` for **dz**. These are the structural r and z deformations. The reason you cannot use physics-induced displacements here is that `uaxi_axi` is not a solution variable in the Stress-Strain application mode.

4 Click **OK** to close the **Subdomain Settings** dialog box.

Boundary Conditions

1 Open the **Boundary Settings** dialog box from the **Physics** menu.

2 Apply the following boundary settings; when done, click **OK**.

SETTINGS	BOUNDARY 1	BOUNDARIES 2, 3	BOUNDARY 4
Type	Mesh displacement	Mesh displacement	Mesh displacement
dr	0		`uaxi_axi`
dz		0	`w`

COMPUTING THE SOLUTION

1 In the **Solver Manager**, go to the **Initial Value** tab and click the **Store Solution** button. By default, all solutions are selected in the **Store Solution** dialog box. Click **OK** to confirm the selection.

2 In the **Values of variables not solved for and linearization point** area, click the **Stored solution** option button and select **All** in the **Solution at time** list. This makes the solver use the solution for the deformation of the tube when computing the fluid flow.

3 On the **Solve For** page, select all variables except **uor** and **w**. Click **OK**.

4 Open the **Solver Parameters** dialog box, go to the **General** tab. Set the **Absolute tolerance** to `1e-3` and the **Relative tolerance** to `0.01`.

5 On the **Time Stepping** tab, set the **Error estimation strategy** to **Exclude algebraic**.

6 Click **OK**, then click **Solve**.

POSTPROCESSING AND VISUALIZATION

The plot still shows the deformations from the stress-strain computation. To see the velocity field as in Figure 2 on page 4, do the following:

1 In the **Plot Parameters** dialog box, go to the **General** page and clear the **Deformed shape** check box.

2 On the **Surface** page, select **Incompressible Navier Stokes (ns)>Velocity field** from the **Predefined quantities** list.

3 On the **Arrow** tab, select **Incompressible Navier Stokes (ns)>Velocity field** from the **Predefined quantities** list. Click the **Color** button and choose a black color; then click **OK**. Select the **Arrow plot** check box and click **Apply**.

4 Once again you can select different times to look at on the **General** page. You see the deformations even though you are not making a deformation plot because the mesh is deformed.

5 To animate the velocity field as a function of time, click the **Animate** tab and then click **Start Animation**. It might take some time for your computer to render the animation.

6 After closing the **Plot Parameters** dialog box, go to **Postprocessing>Domain Plot Parameters**.

7 Use the **Expression** field on the **Point** page to plot, for instance, the expressions `inflow`, `outflow`, `netflow`, or `Vol`. Because these variables are defined with a global destination and do not depend on the location, the coordinates do not matter. To show more than one curve at the same time, select the **Keep current plot** check box on the **General** page.