

Peristaltic Pump

Introduction

In a peristaltic pump, rotating rollers are squeezing a flexible tube. As the pushed down rollers move along the tube, the fluid in the tube follows 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, for modeling the squeezing of the tube, and fluid dynamics, for computing the motion of the fluid. As it is the deformations of the tube that drive the fluid, the fluid part of the simulation needs to be done in a deforming geometry, defined by COMSOL Multiphysics' Moving Mesh application mode.

Model Definition

The model is set up in 2D axial symmetry (Figure 10-21). A vinyl tube of length 0.1 m, inner radius 1 cm, and outer radius 1.5 cm, contains a gas of density $\rho = 1 \text{ kg/m}^3$ and viscosity $\eta = 3\text{e-}5 \text{ Pa}\cdot\text{s}$. 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 is for simplicity modeled as a Gaussian distribution along the length of the tube. The Gaussian has a width of 1 cm and is moving with the constant velocity 0.03 m/s in the positive z direction. The engagement of the roll is represented by the force density being multiplied by a smoothed Heaviside function, kicking in at $t = 0.1 \text{ s}$ and taking 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 deformations of the tube are modeled during a full cycle of 1.5 s.

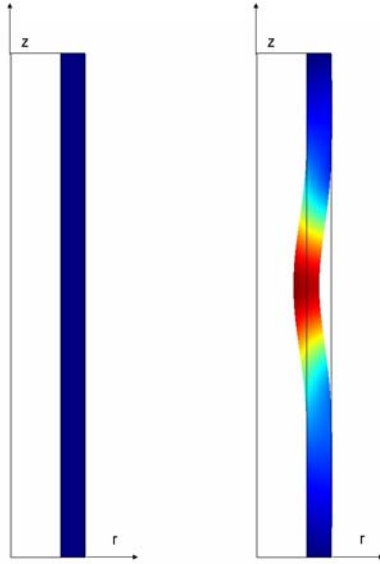


Figure 10-21: The geometry of the peristaltic pump in its initial shape (left) and as it is deforming under the pressure of the roll. The tube is rotationally symmetric with respect to the z axis. The color scale shows the deformation of the tube material.

DOMAIN EQUATIONS

The structural mechanics computations are done using a 2D axially symmetric stress-strain formulation. The material is presumed to be linear and elastic, and geometric nonlinearities are not considered. With the capabilities in the COMSOL Structural Mechanics Module, this model could be easily be extended to account for large deformations and/or use a more complicated material model.

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

$$\rho \frac{\partial \mathbf{u}}{\partial t} - \nabla \cdot \eta (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) + \rho (\mathbf{u} \cdot \nabla) \mathbf{u} + \nabla p = 0 \quad (10-8)$$

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

where ρ denotes the density [kg m^{-3}], \mathbf{u} the velocity vector [m s^{-1}], η the viscosity [N s m^{-2}], and p the pressure [Pa]. The equations are set up and solved in axial symmetry on a deformed mesh inside the tube.

The motion of the deformed mesh is modeled using Winslow smoothing. For more information, please refer to “The Moving Mesh Application Mode” on page 303 in the *COMSOL Multiphysics Modeling Guide*.

BOUNDARY CONDITIONS

For the structural mechanics computations, the time and coordinate dependent load described above is used as the boundary condition at the outer surface of the tube. Whereas the top and bottom ends of the tube are constrained along both coordinate axes, the inner surface is free. The reaction forces from the fluid on the tube are very small, 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 states that the normal component of the stress tensor is zero:

$$\mathbf{n} \cdot [-p\mathbf{I} + \eta(\nabla\mathbf{u} + (\nabla\mathbf{u})^T)] = 0 \quad (10-9)$$

At the outlet, a reference zero pressure is applied. At the tube wall, the fluid is assigned the same velocity as the tube.

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.

Results

Figure 10-22 shows three snapshots from the peristaltic pump in action, and Figure 10-24 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.

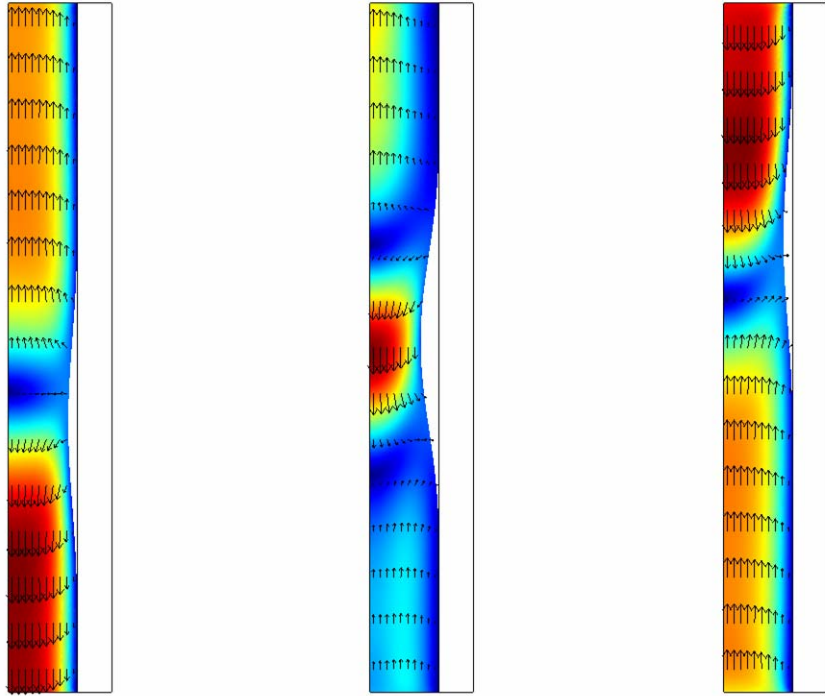


Figure 10-22: Snapshots of the velocity field and the shape of the inside of the tube at $t = 0.3$ s, $t = 0.7$ s, and $t = 1.2$ s. The colors represent the magnitude of the velocity, and the arrows its direction.

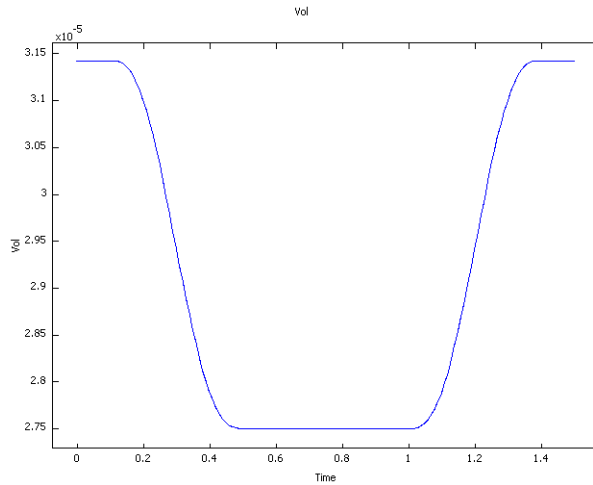


Figure 10-23: The inner volume (m³) of the tube as a function of time (s).

In Figure 10-24, the inlet and outlet flows are shown. The overall behavior indicated in the velocity snapshots is confirmed. As the roll is engaged, gas is squeezed out through both the inlet and the outlet. When the fully engaged roll is moving along the tube, gas is steadily flowing in through the inlet and out through the outlet. As the roll is disengaging, the gas is being sucked back into the tube. In a real peristaltic pump, the peaks associated with the volume changes are usually removed or kept to a minimum, by another roll engaging at the same time as the first roll is disengaging.

This way, there are hardly any volume changes, and the fluid is flowing forward all the time.

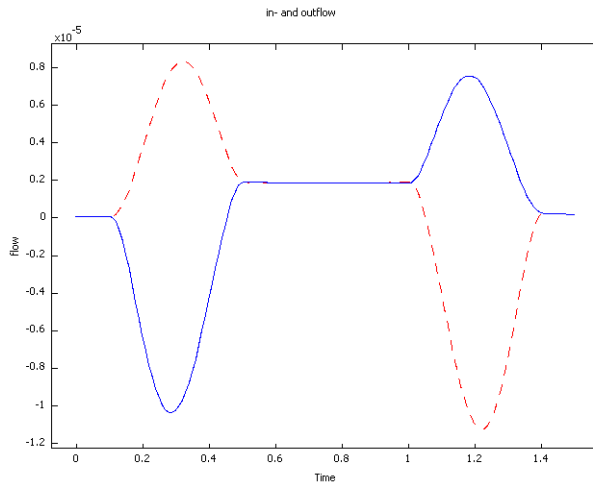


Figure 10-24: Inlet (blue solid line) and outlet (red dashed line) flow in m^3/s as functions of time. Positive values indicate that the gas is flowing in through the inlet and out through the outlet.

Figure 10-25 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.

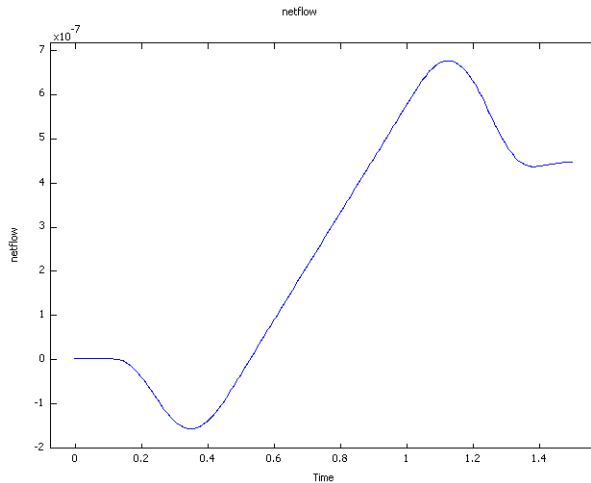


Figure 10-25: Accumulated flow (m^3) through the pump versus time (s).

Modeling in COMSOL

This model is primarily intended to demonstrate the use of the Moving Mesh (ALE) application mode, but it also shows some postprocessing features. Integration coupling variables are defined for the flow computations and for evaluating the inside volume of the tube. An ordinary differential equation is used for calculating the accumulated flow. The smooth step function in this model is `f1c2hs`, which is a C^2 -continuous step with a given step width.

Model Library path: `COMSOL_Multiphysics/Multiphysics/peristaltic_pump`

Modeling Using the Graphical User Interface

- 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>Structural Mechanics>Axial Symmetry, Stress-Strain>Transient analysis** in the list of application modes and click **Add**.
- 4 Select **COMSOL>Deformed Mesh>Moving Mesh (ALE)>Transient analysis** and click **Add**.
- 5 Click the **Application Mode Properties** button and set the **Smoothing method** to **Winslow**.
- 6 Select **COMSOL>Fluid Dynamics>Incompressible Navier-Stokes>Transient analysis** and click **Add**.
- 7 Click **OK** to close the **Model Navigator**.

GEOMETRY MODELING

- 8 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

- 9 Click the **Zoom Extents** button in the Main toolbar.

OPTIONS AND SETTINGS

- 1 Open the **Constants** dialog box from the **Options** menu and enter constants according to the following table. The descriptions are optional.

NAME	EXPRESSION	DESCRIPTION
t_on	0.3	Time when roll is engaged (s)
t_off	1.2	Time when roll is disengaged (s)
dt	0.2	Time to reach full force (s)
z0	0.03	z coordinate where roll starts (m)
v0	0.03	Vertical velocity of roll (m/s)
width	0.01	Width of Gaussian force distribution (m)
Ttot	1.5	Total time for a pump cycle (s)
Lmax	4e6	Max load (N/m ²)

- 2 In the **Options** menu, go to **Functions** and define a new function according to following table.

NAME	ARGUMENTS	EXPRESSION
gauss	x, sigma	$1/(\text{sigma} \cdot \sqrt{2 \cdot \pi}) \cdot \exp(-x^2 / (2 \cdot \text{sigma}^2))$

- 3 Go to **Physics>ODE Settings** and define an ordinary differential equation with the following settings:

NAME (U)	EQUATION	INIT(U)	INIT (UT)
netflow	$\text{netflow}t - (\text{inflow} + \text{outflow}) / (2 * T_{\text{tot}})$	0	0

- 4 Select **Integration Coupling Variables>Boundary Variables** from the **Options** menu and specify the following boundary integration variables:

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

- 5 Select **Integration Coupling Variables>Subdomain Variables** from the **Options** menu and specify the following subdomain integration variables:

SOURCE SUBDOMAIN	NAME	EXPRESSION	INTEGRATION ORDER	GLOBAL DESTINATION
1	Vol	$2 * \pi * r$	4	checked

PHYSICS SETTINGS—STRESS-STRAIN

Subdomain Settings

- In the **Multiphysics** menu, select **Axial Symmetry, Stress-Strain**.
- Open the **Subdomain Settings** dialog box. Select subdomain 1 and clear the **Active in this domain** check box.
- Select subdomain 2 and click **Load**. Select **Nylon** from the list of materials and click **OK**.
- Click **OK** to close the **Subdomain Settings** dialog box.

Boundary Conditions

Open the **Boundary Settings** dialog box from the **Physics** menu and apply the following boundary settings.

BOUNDARY	4	5, 6	7
Constraint R_R		0	
Constraint R_Z		0	
Load F_R			$-L_{\text{max}} * \text{flc}2\text{hs}(t - t_{\text{on}}, dt) * \text{gauss}(z - z_0 - v_0 * t, \text{width}) * \text{flc}2\text{hs}(t_{\text{off}} - t, dt)$
Load F_Z			

MESH GENERATION

- 1 Open the **Mesh Parameters** dialog box from the **Mesh** menu.
- 2 On the **General** page, set **Maximum element size** to $2e-3$.
- 3 Click **Remesh** and then **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 $0:0.01:1.5$, **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 Click **OK** and then **Solve**.

POSTPROCESSING AND VISUALIZATION

The default plot shows the von Mises stress distribution.

- 1 In the **Plot Parameters** dialog box, go to 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 the **Density** to 1 (kg/m^3) and the **Dynamic viscosity** to $3e-5$ ($\text{Pa}\cdot\text{s}$).
- 4 Click **OK** to close the **Subdomain Settings** dialog box.

Boundary Conditions

Open the **Boundary Settings** dialog box from the **Physics** menu and apply the following boundary settings.

BOUNDARY	1	2	3	4
Type	Axial Symmetry	Neutral	Outflow/Pressure	Inflow/Outflow velocity
u_0				uaxi_t_axi
v_0				w
P_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 u_{axi_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 u_{axi_axi} is not a solution variable in the Stress-Strain application mode.
- 4 Click **OK** to close the **Subdomain Settings** dialog box.

Boundary Conditions

Open the **Boundary Settings** dialog box from the **Physics** menu and apply the following boundary settings.

BOUNDARY	1	2-3	4
Type	Mesh displacement	Mesh displacement	Mesh displacement
dr	0		u_{axi_axi}
dz		0	w

COMPUTING THE SOLUTION

- 1 In the **Solver Manager**, go to the **Initial** 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 Set **Value of variables not solved for and linearization point** to **Stored solution** and **Solution at time** to **All**. This makes the software use the solution for the deformation of the tube when computing the fluid flow.
- 3 On the **Solve For** tab, select all variables but **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** and then **Solve**.

POSTPROCESSING AND VISUALIZATION

The plot still shows the deformations from the stress-strain computation. To see the velocity field, do the following:

- 1 In the **Plot Parameters** dialog box, go to the **General** tab and clear the **Deformed shape** check box.
- 2 On the **Surface** tab, select **Velocity field (ns)** among the **Predefined quantities**.
- 3 On the **Arrow** tab, select **Velocity field (ns)** among the **Predefined quantities**. Click the **Color** button and choose a black color.
- 4 Once more, you can select different times to look at on the **General** tab. You will see the deformations even though you are not making a deformation plot, because the mesh is deformed.
- 5 If you want to animate the velocity field as a function of time, go to the **Animate** tab and click **Start Animation**. It may 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** tab 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. If you want to show more than one curve at the same time, use the **Keep current plot** check box on the **General** tab.