

Fluid-Structure Interaction in a Network of Blood Vessels

Introduction

This model studies a portion of the vascular system, in particular the upper part of the aorta (Figure 1). The aorta and its ramified blood vessels are embedded in biological tissue, specifically the cardiac muscle. The flowing blood applies pressure to the artery's internal surfaces and its branches, thereby deforming the tissue. The analysis consists of two distinct but coupled procedures: first, a fluid-dynamics analysis including a calculation of the velocity field and pressure distribution in the blood (variable in time and in space); second, a mechanical analysis of the deformation of the tissue and artery. In this model, any change in the shape of the vessel walls does not influence the fluid domain, which implies that there is only a one-way fluid-structural coupling. However, in COMSOL Multiphysics it is possible to simulate a two-way coupling using the ALE (arbitrary Lagrangian-Eulerian) method.

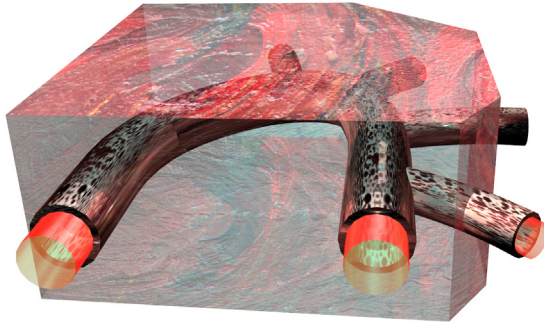


Figure 1: The model domain consists of part of the aorta, its branches, and the surrounding tissue.

Model Definition

Figure 2 shows two views of the model domain, one with and one without the cardiac muscle. The model's mechanical analysis must consider the cardiac muscle because it presents a stiffness that resists artery deformation due to the applied pressure.

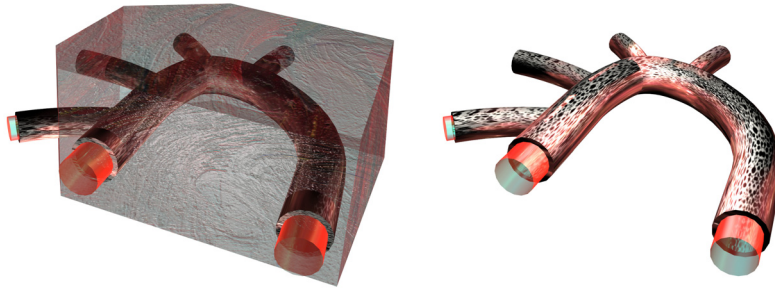


Figure 2: A view of the aorta and its ramification (branching vessels) with blood contained, shown both with (left) and without (right) the cardiac muscle.

The main characteristics of the analyses are:

- *Fluid dynamics analysis*

Here the model solves the Navier-Stokes equations in the blood domain. At each surface where the model brings a vessel to an abrupt end, it represents the load with a known pressure distribution.

- *Mechanical analysis*

This analysis is *nonlinear* due to the assumption of a large displacement and the constitutive behavior of the materials (the model describes the response of the artery and cardiac muscle with an *hyperelastic law* similar to those for rubber components). Only the domains related to the biological tissues are active in this analysis. The model represents the load with the total stress distribution it computes during the fluid-dynamics analysis.

ANALYSIS OF RUBBER-LIKE TISSUE AND ARTERY MATERIAL MODELS

The analysis of rubber-like elastomers is generally a difficult task for several reasons:

- The material can undergo very large strains (finite deformations).
- The stress-strain relationship is generally nonlinear.
- Many rubber-like materials are almost incompressible. You must revise standard displacement-based finite element formulations in order to arrive at correct results (mixed formulations).

You must pay particular attention to the definition of stress and strain measures. In a geometrically nonlinear analysis the assumptions about infinitesimal displacements are no longer valid. It is necessary to consider geometrical nonlinearity in a model when:

- Significant rigid-body rotations occur (finite rotations).
- The strains are no longer small (larger than a few percent).
- The loading of the body depends on the deformation.

All of these issues are dealt with in the hyperelastic material model built-in the Structural Mechanics Module.

MATERIALS

The model in this example uses the following material properties:

- Blood
 - density = 1060 kg/m^3
 - dynamic viscosity = 0.005 Ns/m^2
- Artery
 - density = 960 kg/m^3
 - Neo-Hookean hyperelastic behavior: the coefficient μ equals $6.20 \cdot 10^6 \text{ N/m}^2$, while the bulk modulus equals 20μ and corresponds to a value for Poisson's ratio, ν , of 0.45. An equivalent elastic modulus equals $1.0 \cdot 10^7 \text{ N/m}^2$.
 - Cardiac muscle
 - density = 1200 kg/m^3
 - Neo-Hookean hyperelastic behavior: the coefficient μ equals $7.20 \cdot 10^6 \text{ N/m}^2$, while the bulk modulus equals 20μ and corresponds to a value for Poisson's ratio, ν , of 0.45. An equivalent elastic modulus equals $1.16 \cdot 10^6 \text{ N/m}^2$.

FLUID DYNAMICS ANALYSIS

The fluid dynamics analysis considers the solution of the 3D Navier-Stokes equations. You can do so in both a stationary case or in the time domain. To establish the

boundary conditions, the model uses six pressure conditions with the configuration in Figure 3.

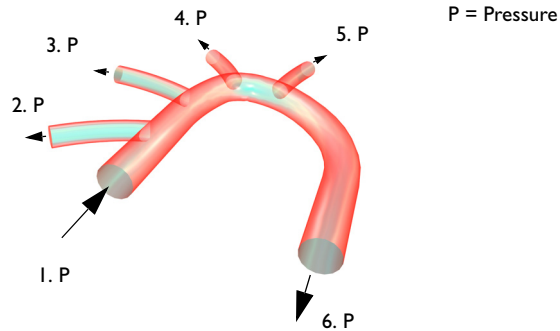


Figure 3: Boundary conditions for the fluid-flow analysis.

The pressure conditions are:

- Section 1: 11,208 Pa
- Section 2: 11,192 Pa
- Section 3: 11,148 Pa
- Section 4: 11,148 Pa
- Section 5: 11,148 Pa
- Section 6: 11,120 Pa

For the time-dependent analysis, the model uses a simple trigonometric function to vary the pressure distribution over time:

$$f(t) = \begin{cases} \sin \pi t & 0 \leq t \leq \frac{1}{2} \text{s} \\ \frac{3}{2} - \frac{1}{2} \cdot \cos\left(2\pi\left(t - \frac{1}{2}\right)\right) & \frac{1}{2} \text{s} \leq t \leq \frac{3}{2} \text{s} \end{cases} \quad (1)$$

You implement this effect in COMSOL Multiphysics using a domain variable.

Results and Discussion

First examine the steady flow field so as to compare its result to the transient case. Figure 4 shows a slice plot of the velocity field (that gives values in m/s).

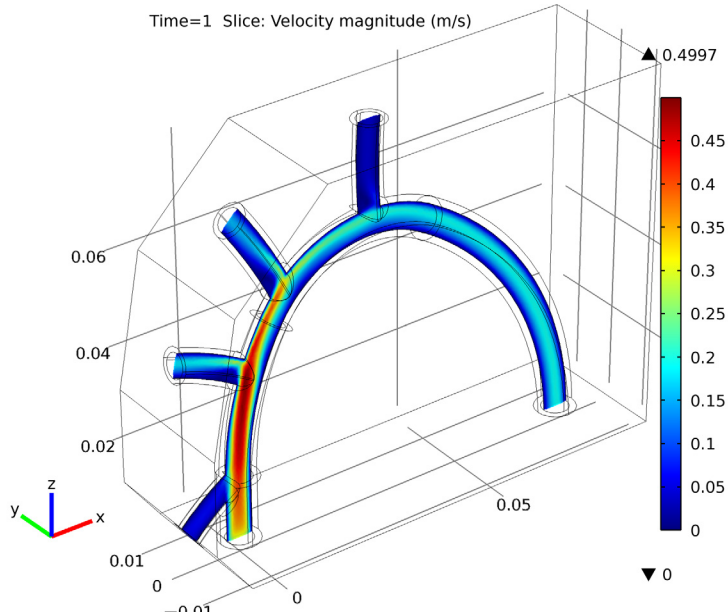


Figure 4: Velocity field color slice in the aorta and its ramification (branching).

A final model accounts for the influence of large displacements and for the hyperelastic behavior of the biological tissues.

Figure 5 shows the total displacement at the peak load (after 1 s). The displacements are in the order of $10\ \mu\text{m}$, which suggests that the one-way multiphysics coupling is a reasonable approximation.

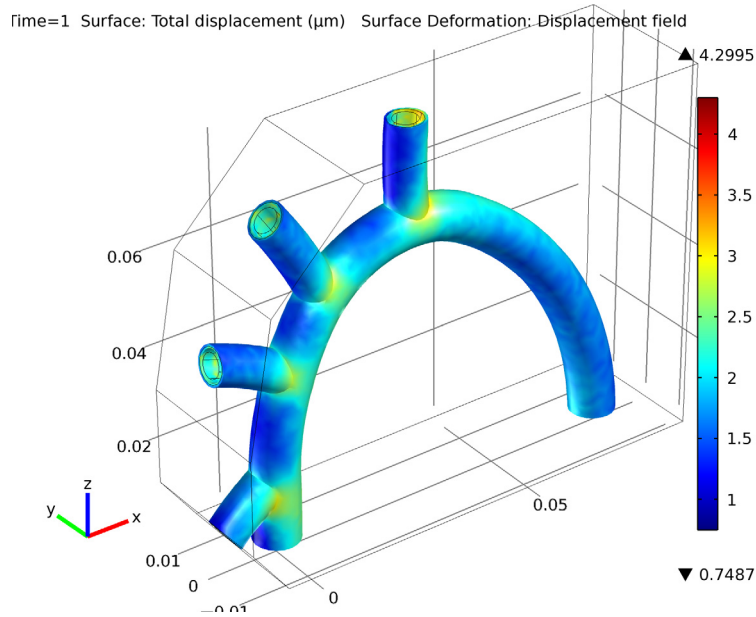


Figure 5: Displacements in the blood vessel using a hyperelastic model.

Notes About the COMSOL Implementation

In this model, and many other cases, an analysis which is time dependent for one physics can be treated as quasi-static from the structural mechanics point of view. You can handle this by running the structural analysis as a parametric sweep over a number of static load cases, where the time is used as the parameter. This method is used here.

Model Library path: Structural_Mechanics_Module/Bioengineering/
blood_vessel

Modeling Instructions

MODEL WIZARD

I Go to the **Model Wizard** window.

- 2 On the **Select Space Dimension** page, click **Next** to get a 3D geometry.
- 3 In the **Add Physics** tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.
- 4 Click **Add Selected**.
- 5 In the **Add Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
- 6 Click **Add Selected**.
- 7 Click **Next**.
- 8 In the **Studies** tree, select **Preset Studies for Selected Physics>Time Dependent**.
- 9 Click **Finish**.

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:

NAME	EXPRESSION	DESCRIPTION
p01	11208[Pa]	Pressure condition 1
p02	11192[Pa]	Pressure condition 2
p03	11148[Pa]	Pressure condition 3
p04	11148[Pa]	Pressure condition 4
p05	11148[Pa]	Pressure condition 5
p06	11120[Pa]	Pressure condition 6

DEFINITIONS

Variables 1

- 1 In the **Model Builder** window, right-click **Model 1>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:

NAME	EXPRESSION	DESCRIPTION
f	$(t \leq 0.5) * \sin(\pi * t [1/s]) + (t > 0.5) * (1.5 - 0.5 * \cos(-2 * \pi * (0.5 - t [1/s])))$	Pressure distribution time dependence

GEOMETRY I

The geometry for this model is available as an MPHBIN-file. Import the geometry as follows.

Import I

- 1 In the **Model Builder** window, right-click **Model I > Geometry I** and choose **Import**.
- 2 Go to the **Settings** window for Import.
- 3 Locate the **Import** section. Click the **Browse** button.
- 4 Browse to the model's Model Library folder and double-click the file `blood_vessel.mphbin`.
- 5 Click the **Import** button.

The length unit in the imported geometry is centimeters, while the default length unit in COMSOL Multiphysics is meters. Therefore, you need to rescale the geometry.

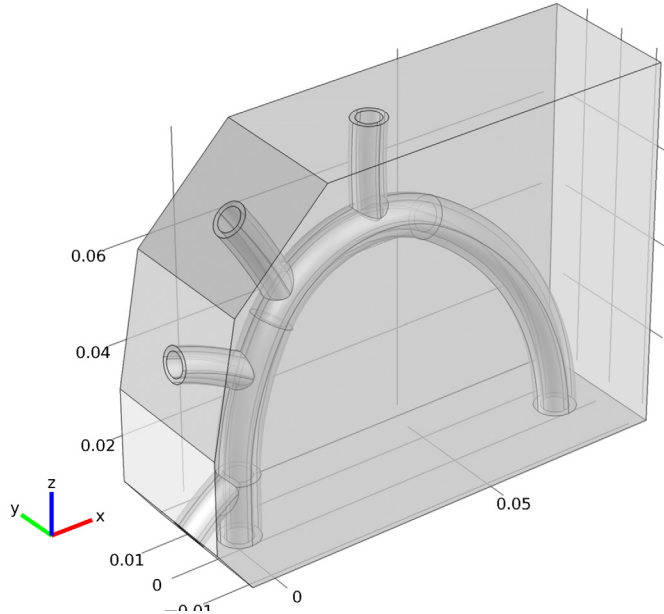
Scale I

- 1 In the **Model Builder** window, right-click **Geometry I** and choose **Scale**.
- 2 Go to the **Settings** window for Scale.
- 3 Locate the **Scale Factor** section. In the **Factor** edit field, type 0.01.
- 4 Select the object **impI** only.
- 5 Click the **Build Selected** button.
- 6 Click the **Go to Default 3D View** button on the Graphics toolbar.

Form Union

- 1 In the **Model Builder** window, right-click **Form Union** and choose **Build Selected**.

- 2 Click the **Transparency** button on the Graphics toolbar to see the interior.



DEFINITIONS

Next, define a number of selections as sets of geometric entities for use in setting up the model.

Selection 1

- 1 In the **Model Builder** window, right-click **Model 1 > Definitions** and choose **Selection**.
- 2 Select Domain 3 only.
- 3 Right-click **Selection 1** and choose **Rename**.
- 4 Go to the **Rename Selection** dialog box and type **blood** in the **New name** edit field.
- 5 Click **OK**.

Selection 2

- 1 Right-click **Definitions** and choose **Selection**.
- 2 Select Domain 2 only.
- 3 Right-click **Selection 2** and choose **Rename**.
- 4 Go to the **Rename Selection** dialog box and type **artery** in the **New name** edit field.
- 5 Click **OK**.

Selection 3

- 1 Right-click **Definitions** and choose **Selection**.
- 2 Select Domain 1 only.
- 3 Right-click **Selection 3** and choose **Rename**.
- 4 Go to the **Rename Selection** dialog box and type `muscle` in the **New name** edit field.
- 5 Click **OK**.

Selection 4

- 1 Right-click **Definitions** and choose **Selection**.
- 2 Go to the **Settings** window for Selection.
- 3 Locate the **Geometric Scope** section. From the **Geometric entity level** list, select **Boundary**.
Select Boundary 38 only as follows:
 - 4 Click **Paste Selection**, then go to the **Paste Selection** dialog box.
 - 5 In the **Selection** edit field, type `38`, then click **OK** to close the dialog box.
 - 6 Right-click **Selection 4** and choose **Rename**.
 - 7 Go to the **Rename Selection** dialog box and type `inlet` in the **New name** edit field.
 - 8 Click **OK**.

Selection 5

- 1 Right-click **Definitions** and choose **Selection**.
- 2 Go to the **Settings** window for Selection.
- 3 Locate the **Geometric Scope** section. From the **Geometric entity level** list, select **Boundary**.
- 4 Select Boundary 19 only.
- 5 Right-click **Selection 5** and choose **Rename**.
- 6 Go to the **Rename Selection** dialog box and type `outlet1` in the **New name** edit field.
- 7 Click **OK**.

Selection 6

- 1 Right-click **Definitions** and choose **Selection**.
- 2 Go to the **Settings** window for Selection.
- 3 Locate the **Geometric Scope** section. From the **Geometric entity level** list, select **Boundary**.
- 4 Select Boundary 9 only.

- 5 Right-click **Selection 6** and choose **Rename**.
- 6 Go to the **Rename Selection** dialog box and type outlet2 in the **New name** edit field.
- 7 Click **OK**.

Selection 7

- 1 Right-click **Definitions** and choose **Selection**.
- 2 Go to the **Settings** window for Selection.
- 3 Locate the **Geometric Scope** section. From the **Geometric entity level** list, select **Boundary**.
- 4 Select Boundary 41 only.
- 5 Right-click **Selection 7** and choose **Rename**.
- 6 Go to the **Rename Selection** dialog box and type outlet3 in the **New name** edit field.
- 7 Click **OK**.

Selection 8

- 1 Right-click **Definitions** and choose **Selection**.
- 2 Go to the **Settings** window for Selection.
- 3 Locate the **Geometric Scope** section. From the **Geometric entity level** list, select **Boundary**.
- 4 Select Boundary 70 only.
- 5 Right-click **Selection 8** and choose **Rename**.
- 6 Go to the **Rename Selection** dialog box and type outlet4 in the **New name** edit field.
- 7 Click **OK**.

Selection 9

- 1 Right-click **Definitions** and choose **Selection**.
- 2 Go to the **Settings** window for Selection.
- 3 Locate the **Geometric Scope** section. From the **Geometric entity level** list, select **Boundary**.
- 4 Select Boundary 86 only.
- 5 Right-click **Selection 9** and choose **Rename**.
- 6 Go to the **Rename Selection** dialog box and type outlet5 in the **New name** edit field.
- 7 Click **OK**.

Selection 10

- 1 Right-click **Definitions** and choose **Selection**.

- 2 Go to the **Settings** window for Selection.
 - 3 Locate the **Geometric Scope** section. From the **Geometric entity level** list, select **Boundary**.
 - 4 Select Boundaries 1–6, 12, 26, 27, 30, 33, 64, 67, 85, and 87 only.
 - 5 Right-click **Selection 10** and choose **Rename**.
 - 6 Go to the **Rename Selection** dialog box and type roller boundaries in the **New name** edit field.
 - 7 Click **OK**.
- The roller boundaries are the free boundaries of muscle and artery that are neither in contact with each other nor with blood.

Selection 11

- 1 Right-click **Definitions** and choose **Selection**.
 - 2 Go to the **Settings** window for Selection.
 - 3 Locate the **Geometric Scope** section. From the **Geometric entity level** list, select **Boundary**.
 - 4 Select the **All boundaries** check box.
 - 5 Select Boundaries 10, 11, 16, 17, 20, 21, 23, 24, 36, 37, 39, 40, 42, 43, 45, 46, 50–53, 58, 59, 61, 62, 68, 69, 71, 72, 75, 76, 79, 80, 82, and 83 only.
 - 6 Right-click **Selection 11** and choose **Rename**.
 - 7 Go to the **Rename Selection** dialog box and type loaded boundaries in the **New name** edit field.
 - 8 Click **OK**.
- The loaded boundaries are the inner boundaries of the artery that are in contact with blood.

Selection 12

- 1 Right-click **Definitions** and choose **Selection**.
- 2 Select Domain 2 only.
- 3 Go to the **Settings** window for Selection.
- 4 Locate the **Geometric Scope** section. From the **Selection output** list, select **Adjacent boundaries**.
- 5 Select the **Interior boundaries** check box.
- 6 Right-click **Selection 12** and choose **Rename**.

- 7 Go to the **Rename Selection** dialog box and type `artery walls` in the **New name** edit field.
- 8 Click **OK**.

LAMINAR FLOW

- 1 In the **Model Builder** window, click **Model 1>Laminar Flow**.
- 2 Go to the **Settings** window for Laminar Flow.
- 3 Locate the **Domains** section. From the **Selection** list, select **blood**.
- 4 Locate the **Physical Model** section. From the **Compressibility** list, select **Incompressible flow**.

Inlet 1

- 1 Right-click **Model 1>Laminar Flow** and choose **Inlet**.
- 2 Go to the **Settings** window for Inlet.
- 3 Locate the **Boundaries** section. From the **Selection** list, select **inlet**.
- 4 Locate the **Boundary Condition** section. From the **Boundary condition** list, select **Pressure, no viscous stress**.
- 5 In the p_0 edit field, type $p01*f$.

Outlet 1

- 1 In the **Model Builder** window, right-click **Laminar Flow** and choose **Outlet**.
- 2 Go to the **Settings** window for Outlet.
- 3 Locate the **Boundaries** section. From the **Selection** list, select **outlet1**.
- 4 Locate the **Boundary Condition** section. In the p_0 edit field, type $p02*f$.

Outlet 2

- 1 In the **Model Builder** window, right-click **Laminar Flow** and choose **Outlet**.
- 2 Go to the **Settings** window for Outlet.
- 3 Locate the **Boundaries** section. From the **Selection** list, select **outlet2**.
- 4 Locate the **Boundary Condition** section. In the p_0 edit field, type $p03*f$.

Outlet 3

- 1 In the **Model Builder** window, right-click **Laminar Flow** and choose **Outlet**.
- 2 Go to the **Settings** window for Outlet.
- 3 Locate the **Boundaries** section. From the **Selection** list, select **outlet3**.
- 4 Locate the **Boundary Condition** section. In the p_0 edit field, type $p04*f$.

Outlet 4

- 1 In the **Model Builder** window, right-click **Laminar Flow** and choose **Outlet**.
- 2 Go to the **Settings** window for Outlet.
- 3 Locate the **Boundaries** section. From the **Selection** list, select **outlet4**.
- 4 Locate the **Boundary Condition** section. In the p_0 edit field, type $p05*f$.

Outlet 5

- 1 In the **Model Builder** window, right-click **Laminar Flow** and choose **Outlet**.
- 2 Go to the **Settings** window for Outlet.
- 3 Locate the **Boundaries** section. From the **Selection** list, select **outlet5**.
- 4 Locate the **Boundary Condition** section. In the p_0 edit field, type $p06*f$.

SOLID MECHANICS

- 1 In the **Model Builder** window, click **Model 1>Solid Mechanics**.
- 2 Select Domains 1 and 2 only.
- 3 Go to the **Settings** window for Solid Mechanics.
- 4 Click to expand the **Discretization** section.
- 5 From the **Displacement field** list, select **Linear**.

This matches the default order of the temperature field elements.

Hyperelastic Material Model 1

- 1 Right-click **Model 1>Solid Mechanics** and choose **Hyperelastic Material Model**.
- 2 Go to the **Settings** window for Hyperelastic Material Model.
- 3 Locate the **Domains** section. From the **Selection** list, select **artery**.

Hyperelastic Material Model 2

- 1 In the **Model Builder** window, right-click **Solid Mechanics** and choose **Hyperelastic Material Model**.
- 2 Go to the **Settings** window for Hyperelastic Material Model.
- 3 Locate the **Domains** section. From the **Selection** list, select **muscle**.

Roller 1

- 1 In the **Model Builder** window, right-click **Solid Mechanics** and choose **Roller**.
- 2 Go to the **Settings** window for Roller.
- 3 Locate the **Boundaries** section. From the **Selection** list, select **roller boundaries**.

Boundary Load 1

- 1 In the **Model Builder** window, right-click **Solid Mechanics** and choose **Boundary Load**.
- 2 Go to the **Settings** window for Boundary Load.
- 3 Locate the **Boundaries** section. From the **Selection** list, select **loaded boundaries**.
- 4 Locate the **Force** section. Specify the \mathbf{F}_A vector as

-spf.T_stressx	x
-spf.T_stressy	y
-spf.T_stressz	z

MATERIALS*Material 1*

- 1 In the **Model Builder** window, right-click **Model 1 > Materials** and choose **Material**.
- 2 Go to the **Settings** window for Material.
- 3 Locate the **Geometric Scope** section. From the **Selection** list, select **blood**.
- 4 Locate the **Material Contents** section. In the **Material Contents** table, enter the following settings:

PROPERTY	NAME	VALUE
Density	rho	1060
Dynamic viscosity	mu	0.005

- 5 Right-click **Material 1** and choose **Rename**.
- 6 Go to the **Rename Material** dialog box and type blood in the **New name** edit field.
- 7 Click **OK**.

Material 2

- 1 Right-click **Materials** and choose **Material**.
- 2 Go to the **Settings** window for Material.
- 3 Locate the **Geometric Scope** section. From the **Selection** list, select **artery**.
- 4 Locate the **Material Contents** section. In the **Material Contents** table, enter the following settings:

PROPERTY	NAME	VALUE
Lamé constant	lambda	$20 \cdot \mu - 2 \cdot \mu / 3$

PROPERTY	NAME	VALUE
Lamé constant	mu	6.20e6
Density	rho	960

- 5 Right-click **Material 2** and choose **Rename**.
- 6 Go to the **Rename Material** dialog box and type artery in the **New name** edit field.
- 7 Click **OK**.

Material 3

- 1 Right-click **Materials** and choose **Material**.
- 2 Go to the **Settings** window for Material.
- 3 Locate the **Geometric Scope** section. From the **Selection** list, select **muscle**.
- 4 Locate the **Material Contents** section. In the **Material Contents** table, enter the following settings:

PROPERTY	NAME	VALUE
Lamé constant	lambda	$20 \cdot \mu - 2 \cdot \mu / 3$
Lamé constant	mu	7.20e6
Density	rho	1200

- 5 Right-click **Material 3** and choose **Rename**.
- 6 Go to the **Rename Material** dialog box and type muscle in the **New name** edit field.
- 7 Click **OK**.

MESH 1

Free Tetrahedral 1

In the **Model Builder** window, right-click **Model 1 > Mesh 1** and choose **Free Tetrahedral**.

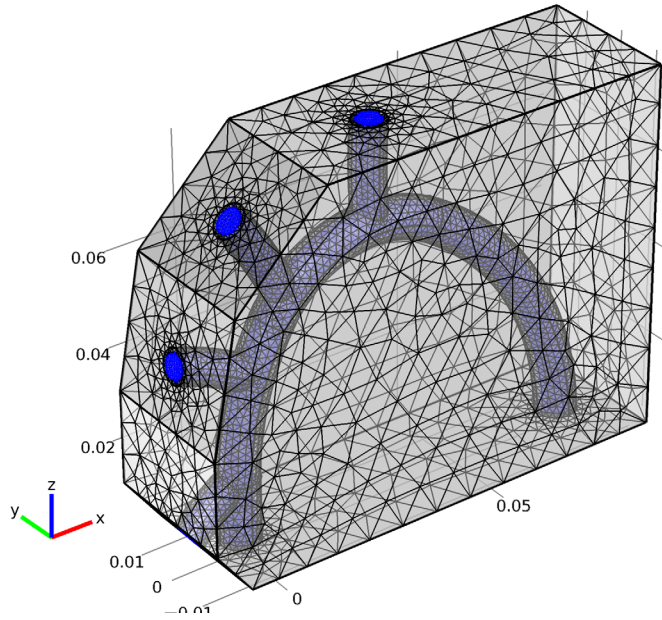
Size 1

- 1 In the **Model Builder** window, right-click **Free Tetrahedral 1** and choose **Size**.
- 2 Go to the **Settings** window for Size.
- 3 Locate the **Geometric Scope** section. From the **Geometric entity level** list, select **Domain**.
- 4 From the **Selection** list, select **blood**.
- 5 Locate the **Element Size** section. Click the **Custom** button.
- 6 Click to expand the **Element Size Parameters** section.

- 7 Select the **Maximum element size** check box.
- 8 In the associated edit field, type $1e-3$.

Size

- 1 In the **Model Builder** window, click **Size**.
- 2 Go to the **Settings** window for Size.
- 3 Locate the **Element Size** section. From the **Predefined** list, select **Fine**.
- 4 Click the **Build All** button.



STUDY 1

The structural problem is quasi-static, so you can use the time just as a parameter for the parametric solver, together with a stationary solver. Thus the whole study can be divided into two steps. First run the transient study for the fluid-mechanics part of the problem and then use the stationary solver to solve the structural part using the solution from first transient study.

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, 0.05, 1).
- 4 Locate the **Physics Selection** section. In the **Physics interfaces** list, select **Solid mechanics (solid)**.
- 5 Clear the **Use in this study** check box.
- 6 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

By default, you get a surface and a slice plot of the velocity field. Modify the slice plot in the second plot group to reproduce the plot in Figure 4.

3D Plot Group 2

- 1 Click the **Transparency** button on the Graphics toolbar to restore the original transparency state.
- 2 In the **Model Builder** window, expand the **Results>3D Plot Group 2** node, then click **Slice 1**.
- 3 Go to the **Settings** window for Slice.
- 4 Locate the **Plane Data** section. From the **Plane** list, select **zx-planes**.
- 5 In the **Planes** edit field, type 1.
- 6 Click the **Plot** button.
- 7 Click the **Go to Default 3D View** button on the Graphics toolbar.

Now turn to the structural problem.

GLOBAL DEFINITIONS

Parameters

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

NAME	EXPRESSION	DESCRIPTION
t	0	Time as parameter

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 for Selected Physics>Stationary**.

4 Click **Finish**.

STUDY 2

Step 1: Stationary

- 1 In the **Model Builder** window, click **Study 2>Step 1: Stationary**.
- 2 Go to the **Settings** window for Stationary.
- 3 Locate the **Physics Selection** section.
- 4 In the **Physics interfaces** list, select **Laminar flow (spf)**.
- 5 Clear the **Use in this study** check box.

Parametric Sweep

- 1 In the **Model Builder** window, right-click **Study 2** and choose **Parametric Sweep**.
- 2 Go to the **Settings** window for Parametric Sweep.
- 3 Locate the **Study Settings** section. Under **Parameter names**, click **Add**.
- 4 Go to the **Add** dialog box.
- 5 In the **Parameter names** list, select **Time as parameter (t)**.
- 6 Click the **OK** button.
- 7 Go to the **Settings** window for Parametric Sweep.
- 8 Locate the **Study Settings** section. In the **Parameter values** edit field, type range (0,0.05,1).

Solver 2

- 1 In the **Model Builder** window, right-click **Study 2** and choose **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solver 2>Dependent Variables 1** node, then click **Dependent Variables 1**.
- 3 Go to the **Settings** window for Dependent Variables.
- 4 Locate the **Values of Variables Not Solved For** section. From the **Method** list, select **Solution**.
- 5 From the **Solution** list, select **Solver 1**.
- 6 From the **Time** list, select **All**.
- 7 In the **Model Builder** window, click **Dependent Variables 1>mod1_u2**.
- 8 Go to the **Settings** window for Field.
- 9 Locate the **Scaling** section. From the **Method** list, select **Manual**.
- 10 In the **Scale** edit field, type $1e-5$.

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

RESULTS

To reproduce the plot shown in Figure 5, begin by defining a selection for the solution data set to make interior boundaries visible in the plot.

Data Sets

- 1** In the **Model Builder** window, expand the **Results>Data Sets** node.
- 2** Right-click **Solution 2** and choose **Add Selection**.
- 3** Go to the **Settings** window for Selection.
- 4** Locate the **Geometric Scope** section. From the **Geometric entity level** list, select **Boundary**.
- 5** From the **Selection** list, select **artery walls**.

3D Plot Group 3

- 1** In the **Model Builder** window, expand the **3D Plot Group 3** node, then 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 **Solid Mechanics>Total displacement (solid.disp)**.
- 5** Locate the **Expression** section. From the **Unit** list, select μm .
- 6** Click the **Plot** button.
- 7** Right-click **Surface 1** and choose **Deformation**.
- 8** Go to the **Settings** window for Deformation.
- 9** In the upper-right corner of the **Expression** section, click **Replace Expression**.
- 10** From the menu, choose **Solid Mechanics>Displacement field (u2, v2, w2)**.
- 11** Locate the **Scale** section. Select the **Scale factor** check box.
- 12** In the associated edit field, type 300.
- 13** In the **Model Builder** window, click **3D Plot Group 3**.
- 14** Go to the **Settings** window for 3D Plot Group.
- 15** Locate the **Plot Settings** section. Select the **Title** check box.
- 16** In the associated edit field, type Time=1 Surface: Total displacement (μm)
Surface Deformation: Displacement field.
- 17** Click the **Plot** button.
- 18** Click the **Go to Default 3D View** button on the Graphics toolbar.