

Fluid-Structure Interaction in a Network of Blood Vessels

This model is licensed under the COMSOL Software License Agreement 6.1. All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This example 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. Any change in the shape of the vessel walls does not influence the fluid domain, which implies that there is only a unidirectional fluid-structural coupling.



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 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 Navier–Stokes equations are solved 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

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

Generally, the modeling of biological tissue is an advanced subject for several reasons:

- The material can undergo very large strains (finite deformations).
- The stress-strain relationship is generally nonlinear.
- Many hyperelastic materials are almost incompressible. You must then 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 Nonlinear Structural Materials Module.

In this case, the displacements and strains are so small that it is sufficient to use a linear elastic material model. The material data is given for a neo-Hookean hyperelastic material, but in the small strain limit the interpretation of the material constants is the same for a linear elastic material.

MATERIALS

The following material properties are used:

- Blood
 - density = 1060 kg/m^3
 - dynamic viscosity = 0.005 Ns/m^2
- Artery
 - density = 960 kg/m³
 - Linear elastic behavior: the Lame' parameter μ equals $6.20 \cdot 10^6$ N/m², while the other Lame' parameter λ equals 20μ .
- Cardiac muscle
 - density = 1200 kg/m^3
 - Linear elastic behavior: the Lame' parameter μ equals $7.20 \cdot 10^6$ N/m², while the other Lame' parameter λ equals 20μ .

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, six pressure conditions are applied with the configuration shown in Figure 3.



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

The pressure conditions are:

- Section 1: 126.09 mmHg
- Section 2: 125.91 mmHg
- Section 3: 125.415 mmHg
- Section 4: 125.415 mmHg
- Section 5: 125.415 mmHg
- Section 6: 125.1 mmHg

Those pressure values are the mean values over a heart beating cycle. During a cycle the pressure varies between a minimal and a maximal value which is calculated thanks to a relative amplitude α . For the time-dependent analysis, a simple trigonometric function is used for varying the pressure distribution over time:

$$f(t) = \begin{cases} (1-\alpha)\sin(\pi t) & 0 \le t \le 0.5s \\ 1-\alpha\cos(2\pi(t-0.5)) & 0.5s \le t \le 1.5s \end{cases}$$

The first piece of function between 0 and 0.5 s has no physical significance, it is just a ramp that enables you to calculate the initial state. The second piece of function makes the pressure vary between its minimal and maximal value during a 1 s cycle.

You implement this effect in COMSOL Multiphysics using a Piecewise function.

Results and Discussion

The flow field at the time t=1 s is displayed in Figure 4 as a slice plot.



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

Figure 5 shows the total displacement at the peak load (after 1 s). The displacements are in the order of 4 μ m, which suggests that the unidirectional multiphysics coupling is a reasonable approximation.



Figure 5: Displacements in the blood vessel.

Notes About the COMSOL Implementation

In this example, 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.

Application Library path: Structural_Mechanics_Module/Fluid-Structure_Interaction/blood_vessel

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click 🔗 Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 间 3D.
- 2 In the Select Physics tree, select Fluid Flow>Fluid-Structure Interaction>Fluid-Solid Interaction, Fixed Geometry.
- 3 Click Add.
- 4 Click \bigcirc Study.
- 5 In the Select Study tree, select General Studies>Time Dependent.
- 6 Click **M** Done.

GLOBAL DEFINITIONS

Parameters 1

- I In the Model Builder window, under Global Definitions click Parameters I.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** In the table, enter the following settings:

Name	Expression	Value	Description
t	0[s]	0 s	Time continuation parameter
alpha	1/3	0.33333	Relative pressure amplitude during heart's beating

Piecewise I (pwI)

- I In the Home toolbar, click f(X) Functions and choose Global>Piecewise.
- 2 In the Settings window for Piecewise, type f in the Function name text field.
- 3 Locate the **Definition** section. In the **Argument** text field, type t.
- **4** Find the **Intervals** subsection. In the table, enter the following settings:

Start	End	Function
0	0.5	(1-alpha)*sin(pi*t)
0.5	1.5	1-alpha*cos(2*pi*(t-0.5))

5 Locate the Units section. In the Arguments text field, type s.

6 In the Function text field, type 1.

7 Click 💽 Plot.

GEOMETRY I

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

Import I (imp1)

- I In the Home toolbar, click 🗔 Import.
- 2 In the Settings window for Import, locate the Import section.
- 3 Click 📂 Browse.
- 4 Browse to the model's Application Libraries folder and double-click the file blood_vessel.mphbin.
- 5 Click ा Import.

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 (scal)

- I In the Geometry toolbar, click 💭 Transforms and choose Scale.
- 2 In the Settings window for Scale, locate the Scale Factor section.
- **3** In the **Factor** text field, type **0.01**.
- 4 Select the object impl only.

Use **Remove Details** to automatically fix small issues in the geometry. This will improve meshing and computation.

Remove Details 1 (rmd1)

- I In the Geometry toolbar, click 📉 Remove Details.
- 2 In the Settings window for Remove Details, click 📗 Build Selected.

Two faces have been collapsed.

3 Click the $\sqrt[4]{}$ Go to Default View button in the Graphics toolbar.

Form Union (fin)

- I In the Model Builder window, click Form Union (fin).
- 2 In the Settings window for Form Union/Assembly, click 틤 Build Selected.

3 Click the Transparency button in the Graphics toolbar to see the interior.



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

DEFINITIONS

Blood

- I In the Definitions toolbar, click 🐚 Explicit.
- 2 In the Settings window for Explicit, type Blood in the Label text field.
- **3** Select Domain 3 only.

Explicit 2-10

I Proceed to create nine explicit selections with the following settings:

Label	Geometric entity level	Selection
Artery	Domain	2
Muscle	Domain	1
Inlet	Boundary	38
Outlet 1	Boundary	19
Outlet 2	Boundary	9
Outlet 3	Boundary	41
Outlet 4	Boundary	70
Outlet 5	Boundary	84
Roller boundaries	Boundary	1-6,12,26,27,30,33,64, 67,83,85

The roller boundaries are the free boundaries of muscle and artery that are neither in contact with each other nor with blood.

Loaded boundaries

- I In the Definitions toolbar, click 🐂 Explicit.
- 2 In the Settings window for Explicit, type Loaded boundaries in the Label text field.
- 3 Locate the Input Entities section. From the Geometric entity level list, choose Boundary.
- 4 Select Boundaries 10, 20, 36, 42, and 68 only.
- Select the Group by continuous tangent check box. The selection should now contain 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, 77–78, 80–81.

The loaded boundaries are the inner artery boundaries that are in contact with blood.

LAMINAR FLOW (SPF)

- I In the Model Builder window, under Component I (compl) click Laminar Flow (spf).
- 2 In the Settings window for Laminar Flow, locate the Domain Selection section.
- **3** From the **Selection** list, choose **Blood**.

Inlet 1

- I In the Physics toolbar, click 🔚 Boundaries and choose Inlet.
- 2 In the Settings window for Inlet, locate the Boundary Selection section.
- **3** From the **Selection** list, choose **Inlet**.

4 Locate the Boundary Condition section. From the list, choose Pressure.

5 Locate the **Pressure Conditions** section. In the p_0 text field, type 126.09[mmHg]*f(t).

Outlet I

- I In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.
- 2 In the Settings window for Outlet, locate the Boundary Selection section.
- **3** From the **Selection** list, choose **Outlet I**.
- **4** Locate the **Pressure Conditions** section. In the p_0 text field, type 125.91[mmHg]*f(t).

Outlet 2-5

Proceed to add four outlet boundary nodes with the following settings:

Boundary Selection	p0
Outlet 2	125.415[mmHg]*f(t)
Outlet 3	125.415[mmHg]*f(t)
Outlet 4	125.415[mmHg]*f(t)
Outlet 2	125.1[mmHg]*f(t)

SOLID MECHANICS (SOLID)

- I In the Model Builder window, under Component I (comp I) click Solid Mechanics (solid).
- **2** Select Domains 1 and 2 only.

Linear Elastic Material I

- I In the Model Builder window, under Component I (comp1)>Solid Mechanics (solid) click Linear Elastic Material I.
- **2** In the **Settings** window for **Linear Elastic Material**, locate the **Linear Elastic Material** section.
- 3 From the Specify list, choose Lamé parameters.

Roller I

- I In the Physics toolbar, click 📄 Boundaries and choose Roller.
- 2 In the Settings window for Roller, locate the Boundary Selection section.
- **3** From the Selection list, choose Roller boundaries.

MULTIPHYSICS

Fluid-Structure Interaction 1 (fsi1)

I In the Model Builder window, under Component I (compl)>Multiphysics click Fluid-Structure Interaction I (fsil).

- 2 In the Settings window for Fluid-Structure Interaction, locate the Fixed Geometry section.
- **3** From the **Fixed geometry coupling type** list, choose **Fluid loading on structure** to ensure a unidirectional coupling, from the fluid to the solid.

MATERIALS

Blood

- I In the Model Builder window, under Component I (compl) right-click Materials and choose Blank Material.
- 2 In the Settings window for Material, type Blood in the Label text field.
- **3** Locate the Geometric Entity Selection section. From the Selection list, choose Blood.
- **4** Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	1060	kg/m³	Basic
Dynamic viscosity	mu	0.005	Pa·s	Basic

Artery

- I Right-click Materials and choose Blank Material.
- 2 In the Settings window for Material, type Artery in the Label text field.
- 3 Locate the Geometric Entity Selection section. From the Selection list, choose Artery.
- **4** Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Lamé parameter λ	lambLame	20*muLame-2* muLame/3	N/m²	Lamé parameters
Lamé parameter μ	muLame	6.20e6	N/m²	Lamé parameters
Density	rho	960	kg/m³	Basic

Muscle

- I Right-click Materials and choose Blank Material.
- 2 In the Settings window for Material, type Muscle in the Label text field.
- 3 Locate the Geometric Entity Selection section. From the Selection list, choose Muscle.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Lamé parameter λ	lambLame	20*muLame-2* muLame/3	N/m²	Lamé parameters
Lamé parameter μ	muLame	7.20e6	N/m²	Lamé parameters
Density	rho	1200	kg/m³	Basic

MESH I

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, locate the Physics-Controlled Mesh section.
- **3** From the **Element size** list, choose **Fine**.

Size

- I Right-click Component I (compl)>Mesh I and choose Edit Physics-Induced Sequence.
- 2 In the Settings window for Size, locate the Element Size section.
- **3** Click the **Custom** button.
- **4** Locate the **Element Size Parameters** section. In the **Minimum element size** text field, type 0.0003.

5 Click 📗 Build All.



STUDY I

The structural problem is quasistatic, so you can use the time 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 the first transient study.

Step 1: Time Dependent

- I In the Model Builder window, under Study I click Step I: Time Dependent.
- 2 In the Settings window for Time Dependent, locate the Study Settings section.
- 3 In the **Output times** text field, type range(0,0.05,1.5).
- **4** Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Solid Mechanics (solid)**.

Stationary

- I In the Study toolbar, click 🔀 Study Steps and choose Stationary>Stationary.
- 2 In the Settings window for Stationary, locate the Physics and Variables Selection section.
- 3 In the table, clear the Solve for check box for Laminar Flow (spf).

- 4 Click to expand the Study Extensions section. Select the Auxiliary sweep check box.
- 5 Click + Add.
- 6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
t (Time continuation parameter)	range(0,0.05,1.5)	S

- 7 Click to expand the Values of Dependent Variables section. Find the Values of variables not solved for subsection. From the Settings list, choose User controlled.
- 8 From the Method list, choose Solution.
- 9 From the Study list, choose Study I, Time Dependent.

IO From the Selection list, choose Automatic (all solutions).

Solution 1 (soll)

- I In the Study toolbar, click **The Show Default Solver**.
- 2 In the Model Builder window, expand the Solution I (soll) node, then click Time-Dependent Solver I.
- **3** In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
- **4** From the **Steps taken by solver** list, choose **Intermediate**. This way the solver computes at least once between each output time step in order to reduce possible interpolation errors in the fluid-load evaluation.
- **5** In the **Study** toolbar, click **= Compute**.

RESULTS

Velocity (spf)

I Click the Transparency button in the Graphics toolbar to restore the original transparency state.

By default, you get a slice plot of the velocity and a contour plot of the fluid pressure on the wall surface. The plot in Figure 4 corresponds to the first default plot.

- 2 In the Model Builder window, click Velocity (spf).
- 3 In the Settings window for 3D Plot Group, locate the Data section.
- 4 From the Parameter value (t (s)) list, choose I.

Slice

- I In the Model Builder window, click Slice.
- 2 In the Settings window for Slice, locate the Plane Data section.
- 3 From the Plane list, choose zx-planes.
- 4 In the **Planes** text field, type 1.
- 5 In the Velocity (spf) toolbar, click 💿 Plot.
- 6 Click the View button in the Graphics toolbar.

Pressure (spf)

The default unit for pressure plots is Pascal. As the mmHg unit is not available in the selection list, type it directly in the text field.

- I In the Model Builder window, under Results click Pressure (spf).
- 2 In the Settings window for 3D Plot Group, click to expand the Title section.

Surface

- I In the Model Builder window, expand the Pressure (spf) node, then click Surface.
- 2 In the Settings window for Surface, locate the Expression section.
- 3 In the **Unit** field, type mmHg.
- **4** In the **Pressure (spf)** toolbar, click **I** Plot.

Stress (solid)

- I In the Model Builder window, under Results click Stress (solid).
- 2 In the Settings window for 3D Plot Group, click to expand the Selection section.
- **3** From the Geometric entity level list, choose Domain.
- 4 From the Selection list, choose Artery.
- 5 In the Model Builder window, expand the Stress (solid) node.

Deformation

- I In the Model Builder window, expand the Results>Stress (solid)>Volume I node, then click Deformation.
- 2 In the Settings window for Deformation, locate the Scale section.
- 3 Select the Scale factor check box. In the associated text field, type 300.

To reproduce the plot shown in Figure 5, begin by adding **Displacement (solid)** from the predefined plots.

4 In the Home toolbar, click **Markov Add Predefined Plot**.

ADD PREDEFINED PLOT

- I Go to the Add Predefined Plot window.
- 2 In the tree, select Study I/Solution I (soll)>Solid Mechanics>Displacement (solid).
- 3 Click Add Plot in the window toolbar.
- **4** In the **Home** toolbar, click **Markov** Add Predefined Plot.

RESULTS

Displacement (solid)

- I In the Model Builder window, under Results click Displacement (solid).
- 2 In the Settings window for 3D Plot Group, locate the Data section.
- **3** From the **Parameter value (t (s))** list, choose **I**.
- 4 Locate the Selection section. From the Geometric entity level list, choose Domain.
- **5** From the **Selection** list, choose **Artery**.

Volume 1

- I In the Model Builder window, expand the Displacement (solid) node, then click Volume I.
- 2 In the Settings window for Volume, locate the Expression section.
- **3** From the **Unit** list, choose **µm**.

Deformation

- I In the Model Builder window, expand the Volume I node, then click Deformation.
- 2 In the Settings window for Deformation, locate the Scale section.
- 3 Select the Scale factor check box. In the associated text field, type 300.
- 4 In the Displacement (solid) toolbar, click **O** Plot.
- 5 Click the **v** Go to Default View button in the Graphics toolbar.