Model created in COMSOL Multiphysics 6.4

Blood Flow in a Stenosed Pulmonary Artery

This model is licensed under the COMSOL Software License Agreement 6.4.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

Modeling blood flow through a stenosed pulmonary artery is crucial for diagnosing and treating cardiovascular diseases. Simulations can help predict how arterial stenosis impacts blood circulation and guide clinical interventions.

In this example, a Carreau fluid is used to capture the non-Newtonian behavior of blood. The stenosis is modeled as a porous medium to capture the resistance and flow disturbances caused by the narrowing.

Although this example does not provide real data or results, the demonstrated approach offers valuable insights into blood flow dynamics, which are crucial for improving diagnostic tools and treatment strategies for pulmonary artery stenosis.

Model Definition

The modeled geometry is shown in Figure 1.

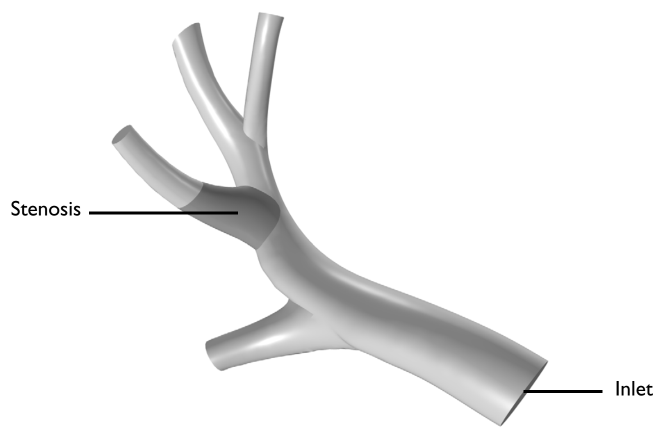


Figure 1: Modeled geometry with stenosis (dark gray)

The idea for this model comes from Ref. 1. The authors studied the hemodynamics of chronic thromboembolic pulmonary hypertension (CTEPH), comparing the effects of Newtonian and non-Newtonian fluid models on blood flow and pressure within stenosed pulmonary arteries. The authors did not provide a detailed mathematical description of the stenosed area. In this model, a **Porous Medium** is used to simulate blood flow through the stenosis. The blood is modeled as a Carreau fluid, similar to the approach used by the authors. The apparent viscosity, denoted as μ_{app}, of a Carreau fluid is defined as follows:

$$\mu_{app} = \mu_{inf} + (\mu_0 - \mu_{inf})[1 + \lambda\dot{\gamma}^2]^{\frac{n-1}{2}} \tag{1}$$

with the viscosity μ_{inf} at infinite shear rate, the relaxation time λ and the power index n and the shear rate $\dot{\gamma}$. To account for the effect of stenosis on the non-Newtonian behavior of blood, the apparent shear rate method is employed in Equation 1 using the capillary bundle approach. For the porous domain, an apparent shear rate is defined, which considers the influence of the porous material on the fluid's non-Newtonian properties.

$$\dot{\gamma}_{app} = \alpha\frac{|\mathbf{u}|}{\sqrt{\kappa\varepsilon_p}}$$

This is done by incorporating parameters such as the permeability κ and porosity ε_p of the stenosed region, the velocity magnitude $|\mathbf{u}|$, and a correction factor α that accounts for the shape of the pores (for example, tortuosity). The correction factor needs to be determined experimentally or through numerical simulations at the pore scale. An approximation can be made using the capillary bundle approach:

$$\alpha = C\left(\frac{3n+1}{4n}\right)^{\frac{n}{n-1}}$$

with the tortuosity factor C. The parameters used in this model are summarized in Table 1 The Carreau parameters for blood are taken from Ref. 1. The parameters describing the porous medium are estimated values; one approach to determining these parameters involves modeling the stenosed area at the pore level and computing the permeability based on the pressure drop (Ref. 2).

TABLE 1: MODELING PARAMETERS.

Quantity	**Value**	**Description**
ε_p	0.5	Porosity
κ	10^{-7} m^2	Permeability
ρ	1120 kg/m^3	Density of blood
μ_0	0.056 Pa·s	Zero shear rate viscosity
μ_{inf}	0.00345	Infinite shear rate viscosity
λ	3.313	Relaxation time
n	0.3568	Power index

One measurement of the severity of the stenosis is the Quantitative Pulmonary Pressure Ratio (QPPR) which relates the pressures at both ends of the stenosis as follows

$$\mathrm{QPPR} = \frac{p_{\mathrm{distal}}}{p_{\mathrm{proximal}}} \tag{2}$$

For this model, QPPR is determined for the maximum systolic pressure.

Results and Discussion

The velocity distribution at the end of the simulation is depicted in Figure 2.

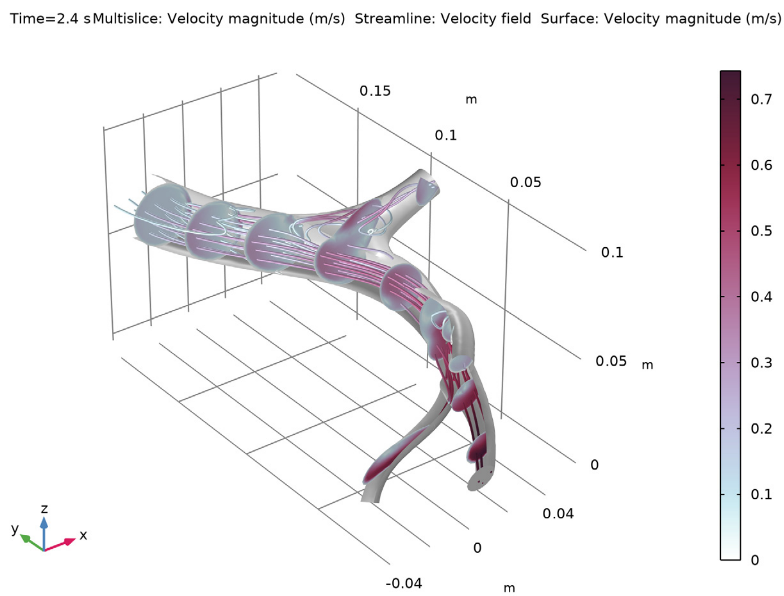


Figure 2: Velocity field after 2.4 s.

The pressure at both ends of the stenosis over the last cycle is shown in Figure 3

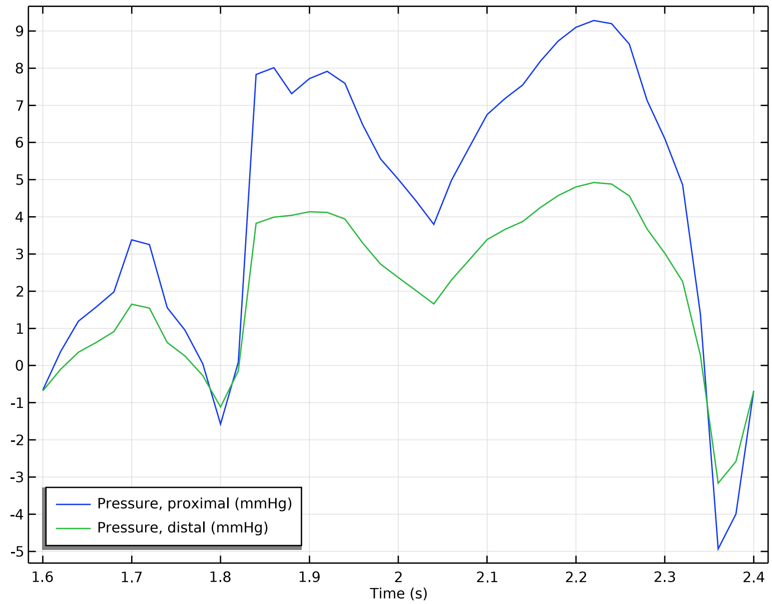


Figure 3: Pressure over the last cycle at proximal and distal ends of the stenosis.

Notes About the COMSOL Implementation

The modeling process begins by opening an mph file that includes a mesh part. This mesh part imports the artery's mesh, which is then adjusted to include the stenosis area and refined to ensure good resolution for fluid flow calculations.

The computation starts with a stationary step to get an initial velocity distribution for the subsequent time-dependent step over three blood flow cycles.

References

1. F. He, X. Wang, L. Hua, and T. Guo, "Non-Newtonian Effects of Blood Flow on Hemodynamics in Pulmonary Stenosis: Numerical Simulation," *Appl. Bionics Biomech.*, 1434832, 7 pages, 2023; doi.org/10.1155/2023/1434832.

2. P. Owasit and S. Sriyab, "Mathematical modeling of non-Newtonian fluid in arterial blood flow through various stenoses," *Adv. Differ. Equ.*, 340, 2021; doi.org/10.1186/s13662-021-03492-9.

Application Library path: `Polymer_Flow_Module/Tutorials/pulmonary_artery_stenosis`

Modeling Instructions

From the **Main Toolbar** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Fluid Flow** > **Porous Media and Subsurface Flow** > **Free and Porous Media Flow, Brinkman (fp)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies** > **Stationary**.

6 Click **Done**.

COMPONENT 1 (COMP1)

Start by importing an mphbin file of the artery.

MESH-BASED GEOMETRY 1

In the **Mesh** toolbar, click **Add Mesh** and choose **Add Mesh-Based Geometry**.

Import 1

1 In the **Settings** window for **Import**, locate the **Import** section.

2 Click **Browse**.

3 Browse to the model's Application Libraries folder and double-click the file `pulmonary_artery_stenosis_mesh.mphbin`.

4 Click **Import**.

5 Clear the **Import selections** checkbox.

Intersect with Plane 1

The stenosed area is introduced by modifying the imported mesh.

1 In the **Mesh** toolbar, click **Booleans and Partitions** and choose **Intersect with Plane**.

2 In the **Settings** window for **Intersect with Plane**, locate the **Geometric Entity Selection** section.

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

4 Select Boundaries 8 and 9 only.

5 Locate the **Plane Definition** section. From the **Plane type** list, choose **Face parallel**.

6 Select Boundary 7 only.

7 In the **Offset in normal direction** text field, type `3.8[cm]`.

8 Select the **Reverse normal direction** checkbox.

9 Click **Build Selected**.

Create Domains 1

1 In the **Mesh** toolbar, click **Entities** and choose **Create Domains**.

2 In the **Settings** window for **Create Domains**, click **Build All**.

FREE AND POROUS MEDIA FLOW, BRINKMAN (FP)

Now, continue with the setup of the physics.

Fluid Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Free and Porous Media Flow, Brinkman (fp)** click **Fluid Properties 1**.

2 In the **Settings** window for **Fluid Properties**, locate the **Fluid Properties** section.

3 Find the **Constitutive relation** subsection. From the list, choose **Inelastic non-Newtonian**.

4 From the **Inelastic model** list, choose **Carreau**.

Porous Medium 1

1 In the **Physics** toolbar, click **Domains** and choose **Porous Medium**.

2 Select Domain 5 only.

Fluid 1

1 In the **Model Builder** window, click **Fluid 1**.

2 In the **Settings** window for **Fluid**, locate the **Fluid Properties** section.

3 Find the **Constitutive relation** subsection. From the list, choose **Inelastic non-Newtonian**.

4 From the **Inelastic model** list, choose **Carreau**.

5 Find the **Apparent shear rate** subsection. From the **Correction factor** list, choose **Capillary bundle approach**.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	1120	kg/m³	Basic
Zero shear rate viscosity	mu0	0.0560	Pa·s	Carreau model
Infinite shear rate viscosity	mu_inf	0.00345	Pa·s	Carreau model
Relaxation time	lam_car	3.313	s	Carreau model
Power index	n_car	0.3568	1	Carreau model
Porosity	epsilon	0.5	1	Basic
Permeability	kappa_iso ; kappaii = kappa_iso, kappaij = 0	1e-7	m²	Basic

FREE AND POROUS MEDIA FLOW, BRINKMAN (FP)

Porosity and permeability are highly influenced by the degree of stenosis. The values used here are estimates (see Table 1).

Inlet 1

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

2 Select Boundary 1 only.

GLOBAL DEFINITIONS

The inlet velocity is a time-dependent function that models the blood flow cycle and will be defined in the following steps.

Interpolation 1 (int1)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global** > **Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `pulmonary_artery_stenosis_interpolation.txt`.

5 Locate the **Interpolation and Extrapolation** section. From the **Interpolation** list, choose **Piecewise cubic**.

6 Click **Plot**.

To make the interpolation function periodic, create an analytic function that references the interpolation function and enforces periodicity.

Analytic 1 (an1)

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

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

3 Locate the **Definition** section. In the **Expression** text field, type `int1(t)`.

4 In the **Arguments** text field, type `t`.

5 Click to expand the **Periodic Extension** section. Select the **Make periodic** checkbox.

6 In the **Upper limit** text field, type `0.8`.

7 Locate the **Units** section. In the **Function** text field, type `m/s`.

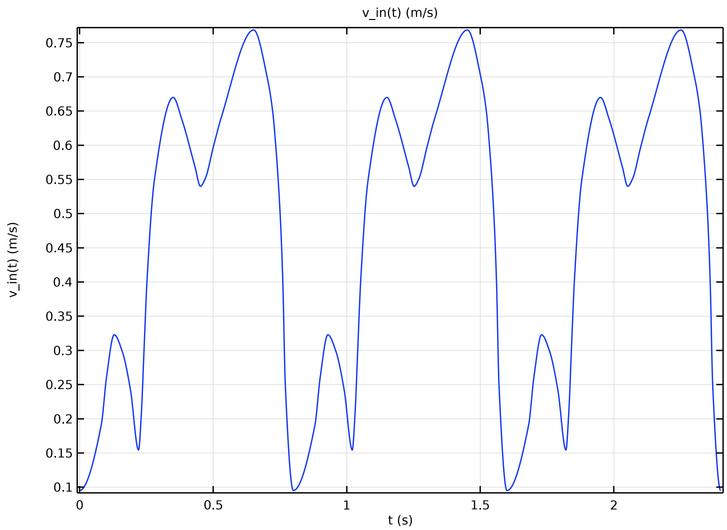
8 In the table, enter the following settings:

Argument	Unit
t	s

9 Locate the **Plot Parameters** section. In the table, enter the following settings:

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	t	0	2.4	0	s

10 Click **Plot**.



Next, call the function for the inlet velocity, which is time dependent and should be invoked with the argument `t` using the expression `v_in(t)`. Since the computation first calculates a stationary solution to establish an initial velocity distribution for the subsequent time-dependent study step, the variable `t` for time is not defined at that stage. This would cause the stationary study to produce an error with the `v_in(t)` expression. To prevent this error, use the `try_catch` operator in the form `v_in(try_catch(t,0))`. This means that the expression will first attempt to evaluate `v_in` for `t`, and if `t` is undefined, it will default to evaluating `v_in` for `0`.

FREE AND POROUS MEDIA FLOW, BRINKMAN (FP)

Inlet 1

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Free and Porous Media Flow, Brinkman (fp)** click **Inlet 1**.
2. In the **Settings** window for **Inlet**, locate the **Velocity** section.
3. In the U_0 text field, type `v_in(try_catch(t,0))`.

Outlet 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.
2. Select Boundaries 4, 7, 10, and 19 only.

Now, build the physics-controlled mesh.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 In the table, clear the **Use** checkbox for **Geometric Analysis, Detail Size**The geometric details might not be accurately captured this way, but they could just be artifacts from the imported mesh rather than real details. So there is no strong justification for including them in the mesh, and excluding them reduces computational effort.

4 Click **Build All**.

STUDY 1

Step 2: Time Dependent

1 In the **Study** toolbar, click **Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type `range(0,0.02,2.4)`.

4 In the **Model Builder** window, click **Study 1**.

5 In the **Settings** window for **Study**, locate the **Study Settings** section.

6 Clear the **Generate default plots** checkbox.

7 In the **Study** toolbar, click **Compute**.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Free and Porous Media Flow, Brinkman** > **Velocity (fp)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Multislice 1

1 In the **Model Builder** window, expand the **Results** > **Velocity (fp)** node, then click **Multislice 1**.

2 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.

3 Find the **x-planes** subsection. In the **Planes** text field, type `0`.

4 Find the **y-planes** subsection. In the **Planes** text field, type `8`.

5 Find the **z-planes** subsection. In the **Planes** text field, type `0`.

6 Locate the **Coloring and Style** section. From the **Color table** list, choose **Passiflora**.

Streamline 1

1 In the **Model Builder** window, right-click **Velocity (fp)** and choose **Streamline**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Streamline**, locate the **Selection** section.

4 Click **Clear Selection**.

5 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Uniform density**.

6 In the **Density level** text field, type `10`.

7 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Tube**.

8 Select the **Radius scale factor** checkbox. In the associated text field, type `5e-4`.

9 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Multislice 1**.

Color Expression 1

Right-click **Streamline 1** and choose **Color Expression**.

Surface 1

1 In the **Model Builder** window, right-click **Velocity (fp)** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 From the **Coloring** list, choose **Uniform**.

4 From the **Color** list, choose **Gray**.

Selection 1

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

2 Select Boundaries 3, 6, 9, 10, 13, 14, and 17 only.

3 In the **Velocity (fp)** toolbar, click **Plot**.

Velocity (fp)

1 In the **Model Builder** window, under **Results** click **Velocity (fp)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 Clear the **Plot dataset edges** checkbox.

4 In the **Velocity (fp)** toolbar, click **Plot**.

RESULT TEMPLATES

Visualize the pressure and the location of the maximum pressure as in Figure 3.

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Free and Porous Media Flow, Brinkman** > **Pressure (fp)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Surface

1 In the **Model Builder** window, expand the **Pressure (fp)** node, then click **Surface**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 In the **Number of bands** text field, type 40.

Pressure (fp)

In the **Model Builder** window, click **Pressure (fp)**.

Max/Min Volume 1

1 In the **Pressure (fp)** toolbar, click **More Plots** and choose **Max/Min Volume**.

2 In the **Settings** window for **Max/Min Volume**, locate the **Expression** section.

3 In the **Expression** text field, type p.

4 Locate the **Display** section. From the **Display** list, choose **Max**.

5 In the **Pressure (fp)** toolbar, click **Plot**.

MAXIMUM AND MINIMUM VALUES

Go to the **Maximum and Minimum Values** window.

Next, evaluate the pressure at both ends of the stenosis for the last cycle.

RESULTS

Surface Average 1

1 In the **Results** toolbar, click **More Derived Values** and choose **Average** > **Surface Average**.

2 Select Boundary 18 only.

3 In the **Settings** window for **Surface Average**, locate the **Data** section.

4 From the **Time selection** list, choose **Interpolated**.

5 In the **Times (s)** text field, type `range(1.6,0.02,2.4)`.

6 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
p	mmHg	Pressure, proximal

7 Click **Evaluate**.

Surface Average 2

1 Right-click **Surface Average 1** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Boundary 15 only.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
p	mmHg	Pressure, distal

6 Click next to **Evaluate**, then choose **Table 1 - Surface Average 1**.

TABLE 1

1 Go to the **Table 1** window.

2 Click the **Table Graph** button in the window toolbar.

RESULTS

Table Graph 1

1 In the **Settings** window for **Table Graph**, click to expand the **Legends** section.

2 Select the **Show legends** checkbox.

Pressure at Stenosis

1 In the **Model Builder** window, under **Results** click **1D Plot Group 3**.

2 In the **Settings** window for **1D Plot Group**, type `Pressure at Stenosis` in the **Label** text field.

3 Locate the **Legend** section. From the **Position** list, choose **Lower left**.

4 In the **Pressure at Stenosis** toolbar, click **Plot**.

One important quantity for assessing the severity of a stenosis is the Quantitative Pulmonary Pressure Ratio (Equation 2). To calculate this, we apply averaging operators at the boundaries of the stenosis and update the model to include these values in the solution, without requiring a recomputation.

DEFINITIONS

Average: Proximal

1. In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.
2. In the **Settings** window for **Average**, locate the **Source Selection** section.
3. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundary 18 only.
5. In the **Label** text field, type `Average: Proximal`.

Average: Distal

1. In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.
2. In the **Settings** window for **Average**, type `Average: Distal` in the **Label** text field.
3. Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundary 15 only.

STUDY 1

In the **Study** toolbar, click **Update Solution**.

RESULTS

Global Evaluation 1

1. In the **Results** toolbar, click **Global Evaluation**.
2. In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.
3. In the table, enter the following settings:

Expression	Unit	Description
aveop2(p)/aveop1(p)	1	QPPR

4. Locate the **Data** section. From the **Time selection** list, choose **From list**.
5. In the **Times (s)** list box, select **1.86**, which corresponds to the maximum systolic pressure.

6 Click **Evaluate**.

The value for QPPR is about 0.5.