Model created in COMSOL Multiphysics 6.4

Arterial Wall Mechanics

Introduction

Arteries are blood vessels that carry freshly oxygenated blood from the heart throughout the rest of the body. They are layered structures with the intima inside, followed by the media and the adventitia. The two outer layers are predominantly responsible for the mechanical behavior of healthy arteries. Both layers are made of collagenous soft tissue that show prominent strain stiffening. Families of collagen fibers give each layer anisotropic properties. These fiber-reinforced structures enable blood vessels to sustain large elastic deformations.

The Holzapfel–Gasser–Ogden (HGO) constitutive model described in Ref. 1 captures the anisotropic, nonlinear mechanical response observed in experiments on excised arteries.

This example model demonstrates how this hyperelastic material is implemented in COMSOL Multiphysics, and the results are compared to those reported in Ref. 1.

Model Definition

The model geometry represents a sector of a carotid artery from a rabbit. Following Ref. 1, the media and adventitia are modeled as a layered cylindrical tube. Model symmetry allows the use of a 2D axisymmetric model. The main dimensions are reported in Figure 1.

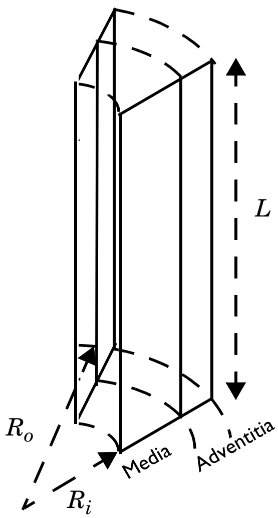


Figure 1: Carotid artery section of L = 2.5 mm in length. The inner radius R_i is 0.71 mm, the outer radius R_o is 1.1 mm, the media thickness is 0.26 mm, and the adventitia thickness is 0.13 mm.

Typical mechanical experiments measure the response of arterial sections subject to combined axial stretch and internal blood pressure. The following set of boundary conditions replicate these experiments: On the bottom surface, a symmetry boundary condition is applied, which allows the bottom end of the artery to expand freely in the radial direction. On the top surface, prescribed displacements in the axial direction account for the axial stretching. The internal pressure is applied with a pressure boundary load on the inner surface.

This model considers axial stretches between 1.5 and 1.9 and internal pressures between 0 and 160 mmHg. The mechanical response in this range is highly nonlinear, resulting in large elastic deformations, and it is described mathematically within the theory of hyperelasticity.

The HGO model is an incompressible, anisotropic hyperelastic material model defined by an isochoric strain energy density. The incompressibility condition implies adding a volumetric stress S_{vol},

$$S_{\mathrm{vol}} = -p_{\mathrm{w}} J C^{-1}$$

The auxiliary pressure p_{w} ensures the incompressibility with the weak equation

$$\text{weak} = (J-1)\text{test}(p_{\text{w}})$$

The isochoric strain energy density is defined by a function of the form

$$W_s = W_1 + W_4 + W_6 \tag{1}$$

The three terms on the right-hand side of Equation 1 depend on invariants of the elastic, isochoric right Cauchy–Green deformation tensor. Here, the first term describes the mechanical behavior of the isotropic ground substance. The strain energy density function W_1 depends on one material parameter c and the first invariant $\overline{I_1} = \text{tr}(\overline{C_{\text{el}}})$, defined in the same fashion as for an incompressible neo-Hookean material (see Ref. 3 for more details)

$$W_1 = \frac{c}{2}(\overline{I_1} - 3) \tag{2}$$

The second and third terms on the right-hand side of Equation 1 describe the mechanical contribution of the collagen fiber network. Following Ref. 4, these expressions are written as

$$W_i = \frac{k_1}{2k_2}(e^{Q_i} - 1), i = 4, 6 \tag{3}$$

$$Q_i = k_2[k_3\overline{I_1} + (1-3k_3)\overline{I_i} - 1]^2, i = 4, 6 \tag{4}$$

Herein, the fiber network is reduced to two families of fibers with equal material properties k_1, k_2, and k_3. The parameter k_1 represents the fiber stiffness (SI unit: Pa), k_2 is a dimensionless tuning parameter, and k_3 is the fiber dispersion. A family i of fibers is defined by a vector field $\mathbf{a}_{0i}$ of unit length in the reference configuration. The fibers deform under the action of the isochoric deformation gradient, so that $\overline{F_{\text{el}}} \cdot \mathbf{a}_{0i}$ describes the fiber vector in the current configuration. The length of $\overline{F_{\text{el}}} \cdot \mathbf{a}_{0i}$ is the fiber stretch, which can be used in the constitutive equations.The HGO model uses the square of the fiber stretches according to the invariants

$$\overline{I_i} = (\overline{F_{\text{el}}} \cdot \mathbf{a}_{0i}) \cdot (\overline{F_{\text{el}}} \cdot \mathbf{a}_{0i}) = \mathbf{a}_{0i} \cdot \overline{C_{\text{el}}} \cdot \mathbf{a}_{0i}, i = 4, 6 \tag{5}$$

Following the example in Ref. 1, the two fiber families are assumed to be oriented at an angle $\pm\beta_j$ with respect to the circumferential direction of the artery. The subscript $j = M, A$ indicates that the angle differs between the media and the adventitia. You can find a detailed background of the HGO model formulation in Ref. 1, Ref. 2, and Ref. 4.

In this example, the mechanical properties of both the media and the adventitia are assumed to be governed by these expressions. Each layer has a distinct set of material parameters c, k_1, and k_2, and k_3, and the initial fiber directions $\mathbf{a}_{0i}$ are aligned at different angles.

MATERIAL

The material parameters for the media and the adventitia are given in the following table.

Material properties (Ref. 1)	Value, media	Value, adventitia
c	3[kPa]	0.3[kPa]
k_1	2.3632[kPa]	0.5620[kPa]
k_2	0.8393	0.7112
β	29[deg]	62[deg]

Results and Discussion

The initial alignment of the fiber families in the media and the adventitia is shown in Figure 2.

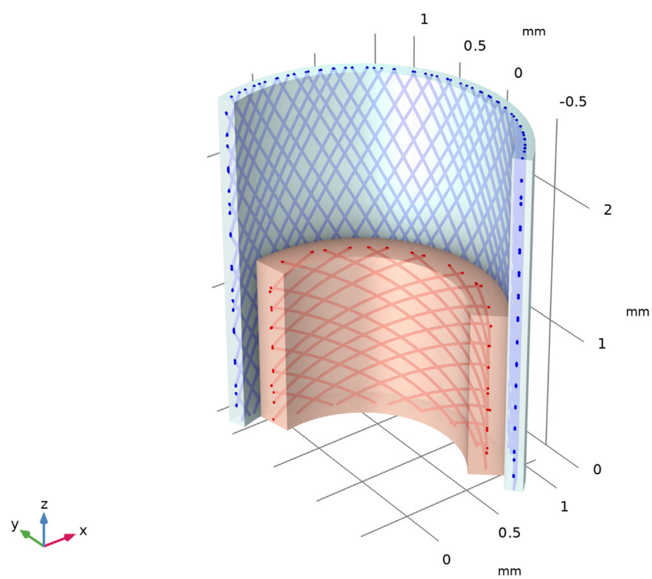


Figure 2: Fiber layout in the media (inner, red) and the adventitia (outer, blue), shown in the undeformed configuration. Note the different angles between the fiber families.

Figure 3 shows the radial stress distribution through the wall thickness at an axial stretch of 1.9 and an internal pressure of 160 mmHg.

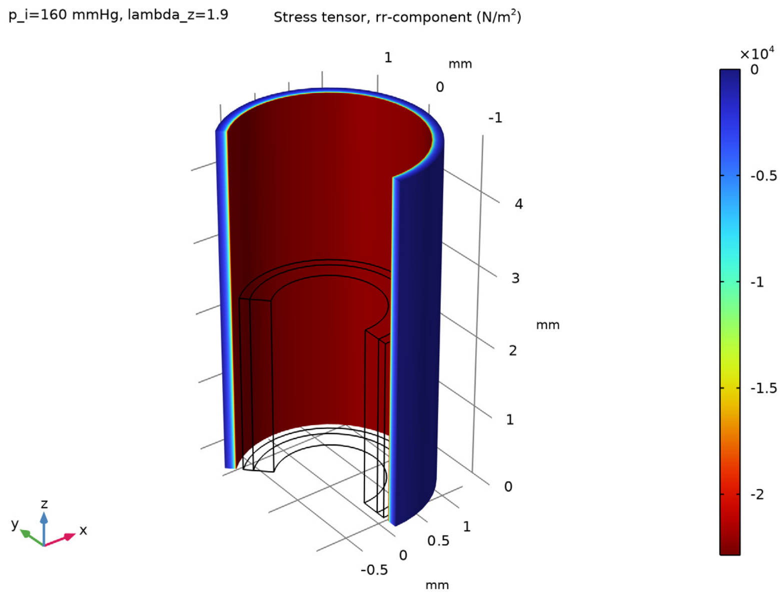


Figure 3: Radial stress distribution in the artery wall at an axial stretch of 1.9 and 160 mmHg internal pressure.

The internal pressure is plotted as a function of the inner radius of the artery for pressures from 0 to 160 mmHg and axial stretches of 1.5, 1.7, and 1.9 in Figure 4. The results are in excellent agreement with the data reproduced from Ref. 1.

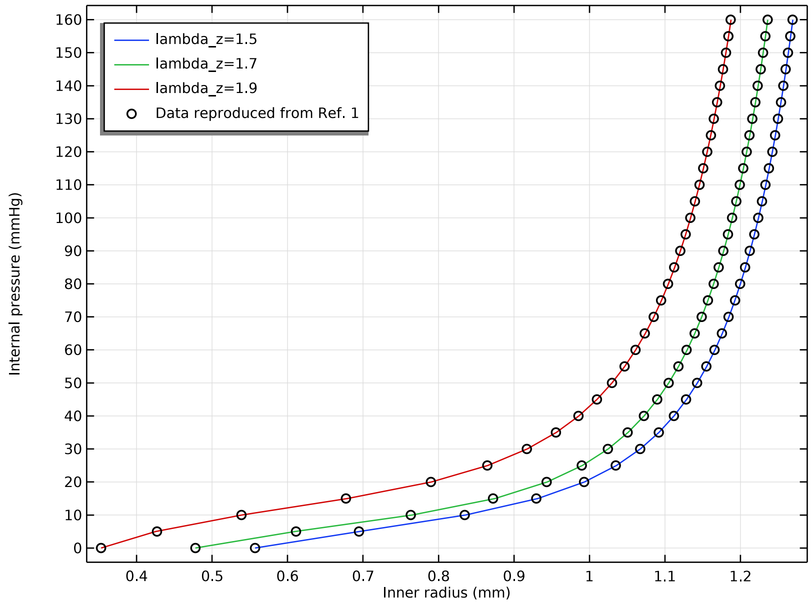


Figure 4: Plot of internal pressure vs. inner radius for three different axial stretches. Data reproduced from Ref. 1 (circles) coincide with the model results.

References

1. G. Holzapfel, T. Gasser, and R. Ogden, "A New Constitutive Framework for Arterial Wall Mechanics and a Comparative Study of Material Models," *J. Elasticity*, vol. 61, pp. 1–48, 2000.

2. G. Holzapfel, *Nonlinear Solid Mechanics: A Continuum Approach for Engineering*, John Wiley & Sons, 2000.

3. *Nonlinear Structural Materials Module User's Guide*, COMSOL Multiphysics.

4. T. Gasser, R. Ogden, and G. Holzapfel, "Hyperelastic modelling of arterial layers with distributed collagen fibre orientations," *J. R. Soc. Interface*, vol. 3, pp. 15–35, 2006.

Notes About the COMSOL Implementation

The most important aspect in this model is the implementation of the HGO material model with the **Fiber** feature.

There are two fiber families in each arterial layer. The mathematical expressions in the HGO model are the same for the media (M) and the adventitia (A), except for the different material parameters.

The initial fiber directions are identified with the use of user-defined rotated coordinate systems. Four coordinate systems are necessary, one for each fiber family. They are oriented in such a way that their second axis is aligned with the fiber directions.

The mechanical behavior of the elastic ground substance is defined in the **Hyperelastic Material** parent node (one for the media and one for the adventitia) with the use of an incompressible neo-Hookean model to define the isotropic function W_1 as in Equation 2.

The mechanical contribution of the collagen fiber network is accounted for with an anisotropic contribution to the isochoric strain energy ($W_4 + W_6$). The **Fiber** feature is used to compute such strain energy densities. Add a **Fiber** feature for each fiber family, two for the media and two for the adventitia. Only the operation for the first family of fibers in the media is discussed below. Similar operations are performed for all other fiber families.

In the settings for the **Fiber** feature, select the appropriate rotated reference system as a reference coordinate system. Then in the *Orientation* section select the orientation of the fiber ($\mathbf{a}_{01\mathrm{M}}$) to be aligned with the second axis (x_2).

The **Fiber** feature automatically computes the corresponding invariant $\overline{I}_i$ according to Equation 5 in order to compute the strain energy function W_i in Equation 3.

The option *Stiffness in tension only* is already activated for the HGO model and it sets the fiber stress and strain energy to zero if the fiber stretch is smaller than one. This means that the fibers only contribute to the stress when they are in extension.

Activate the option *Contribute to total stress* to add the fiber strain energy directly to the total energy. This option allows to consider the fibers and the ground substance as a single anisotropic material and the resulting stress will account for all contributions.

This model also shows how to use the fiber direction variables to plot the orientation of the fiber families (see Figure 2). The COMSOL implementation takes care of the mapping from the global coordinate system in a 2D axisymmetric model to the cylindrical coordinate system used in the revolved result plots.

Application Library path: `Nonlinear_Structural_Materials_Module/Hyperelasticity/arterial_wall_mechanics`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **2D Axisymmetric**.
2 In the **Select Physics** tree, select **Structural Mechanics** > **Solid Mechanics (solid)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies** > **Stationary**.
6 Click **Done**.

GLOBAL DEFINITIONS

Load all model parameters from a file containing parameters for the geometry, the material properties, and the boundary conditions.

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `arterial_wall_mechanics_parameters.txt`.

Now add an interpolation function for importing the pressure versus radius data reproduced from Ref. 1. Use it for comparison.

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 From the **Data source** list, choose **File**.
4 Click **Browse**.
5 Browse to the model's Application Libraries folder and double-click the file `arterial_wall_mechanics_pressure_radius.txt`.

   This file contains the data adapted from Ref. 1.

6 Click **Import**.

7 Locate the **Data Column Settings** section. In the table, enter the following settings:

Columns	Type	Settings
Inner radius (mm), axial stretch 1.5	Function values	Function name=int1a
Inner radius (mm), axial stretch 1.7	Function values	Function name=int1b

8 In the table, click to select the cell at row number 1 and column number 1.

9 In the **Unit** text field, type `kPa`.

10 In the table, click to select the cell at row number 2 and column number 1.

11 In the **Name** text field, type `hgo_pr_1_5`.

12 In the **Unit** text field, type `mm`.

13 In the table, click to select the cell at row number 3 and column number 1.

14 In the **Name** text field, type `hgo_pr_1_7`.

15 In the **Unit** text field, type `mm`.

16 In the table, click to select the cell at row number 4 and column number 1.

17 In the **Name** text field, type `hgo_pr_1_9`.

18 In the **Unit** text field, type `mm`.

GEOMETRY 1

Construct the model geometry by drawing a circumferential cross-section of the artery. Use **Layers** to divide it into two domains corresponding to the Media and the Adventitia.

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

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `Ro-Ri`.

4 In the **Height** text field, type `L`.

5 Locate the **Position** section. In the **r** text field, type `Ri`.

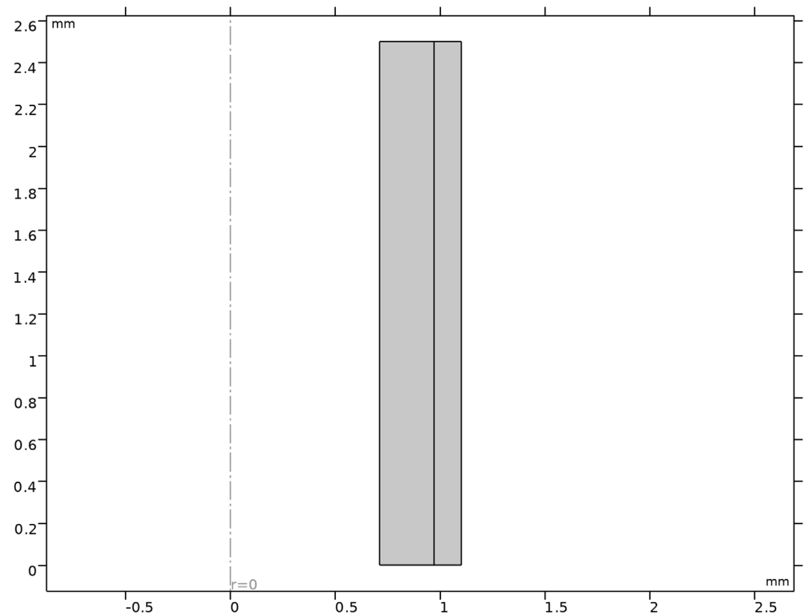
6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
`Layer 1`	HA

7 Clear the **Layers on bottom** checkbox.

8 Select the **Layers to the right** checkbox.

9 In the **Geometry** toolbar, click **Build All**.



DEFINITIONS

Define coordinate systems. These are used to define the initial directions of all fiber families.

Rotated System Media Fiber Family 1

1 In the **Definitions** toolbar, click **Coordinate Systems** and choose **Rotated System**.

2 In the **Settings** window for **Rotated System**, type `Rotated System Media Fiber Family 1` in the **Label** text field.

3 Locate the **Rotation** section. From the **Input method** list, choose **General rotation**.

4 Find the **Euler angles** subsection. In the β text field, type `betaM`.

5 Locate the **Origin** section. From the **Frame** list, choose **Material (R, PHI, Z)**.

Rotated System Media Fiber Family 2

1 Right-click **Rotated System Media Fiber Family 1** and choose **Duplicate**.

2 In the **Settings** window for **Rotated System**, type `Rotated System Media Fiber Family 2` in the **Label** text field.

3 Locate the **Rotation** section. Find the **Euler angles** subsection. In the β text field, type `-betaM`.

Rotated System Adventitia Fiber Family 1

1 Right-click **Rotated System Media Fiber Family 2** and choose **Duplicate**.

2 In the **Settings** window for **Rotated System**, type `Rotated System Adventitia Fiber Family 1` in the **Label** text field.

3 Locate the **Rotation** section. Find the **Euler angles** subsection. In the β text field, type `betaA`.

Rotated System Adventitia Fiber Family 2

1 Right-click **Rotated System Adventitia Fiber Family 1** and choose **Duplicate**.

2 In the **Settings** window for **Rotated System**, type `Rotated System Adventitia Fiber Family 2` in the **Label** text field.

3 Locate the **Rotation** section. Find the **Euler angles** subsection. In the β text field, type `-betaA`.

MATERIALS

Create the materials for the media and the adventitia.

Material (Media)

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 **Geometric Entity Selection** section.

3 In the list box, select **2**.

4 Click **Remove from Selection**.

5 Select Domain 1 only.

6 In the **Label** text field, type `Material (Media)`.

Material (Adventitia)

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Material (Adventitia)` in the **Label** text field.

3 Select Domain 2 only.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **Structural Transient Behavior** section.

3 From the list, choose **Quasistatic**.

Hyperelastic Material (Media)

1 In the **Physics** toolbar, click **Domains** and choose **Hyperelastic Material**.

2 In the **Settings** window for **Hyperelastic Material**, type `Hyperelastic Material (Media)` in the **Label** text field.

3 Select Domain 1 only.

Hyperelastic Material (Adventita)

1 In the **Physics** toolbar, click **Domains** and choose **Hyperelastic Material**.

2 In the **Settings** window for **Hyperelastic Material**, type `Hyperelastic Material (Adventita)` in the **Label** text field.

3 Select Domain 2 only.

4 Locate the **Hyperelastic Material** section. From the **Compressibility** list, choose **Incompressible**.

Hyperelastic Material (Media)

1 In the **Model Builder** window, click **Hyperelastic Material (Media)**.

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

3 From the **Compressibility** list, choose **Incompressible**.

Fiber Family 1

1 In the **Physics** toolbar, click **Attributes** and choose **Fiber**.

2 In the **Settings** window for **Fiber**, type `Fiber Family 1` in the **Label** text field.

3 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Rotated System Media Fiber Family 1 (sys2)**.

4 Locate the **Distribution and Orientation** section. From the **a** list, choose **Second axis**.

5 Locate the **Fiber Model** section. Select the **Contribute to total stress** checkbox.

Fiber Family 2

1 Right-click **Fiber Family 1** and choose **Duplicate**.

2 In the **Settings** window for **Fiber**, type `Fiber Family 2` in the **Label** text field.

3 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Rotated System Media Fiber Family 2 (sys3)**.

Hyperelastic Material (Adventita)

In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** click **Hyperelastic Material (Adventita)**.

Fiber Family 1

1 In the **Physics** toolbar, click **Attributes** and choose **Fiber**.

2 In the **Settings** window for **Fiber**, type `Fiber Family 1` in the **Label** text field.

3 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Rotated System Adventitia Fiber Family 1 (sys4)**.

4 Locate the **Fiber Model** section. Select the **Contribute to total stress** checkbox.

5 Locate the **Distribution and Orientation** section. From the **a** list, choose **Second axis**.

Fiber Family 2

1 Right-click **Fiber Family 1** and choose **Duplicate**.

2 In the **Settings** window for **Fiber**, type `Fiber Family 2` in the **Label** text field.

3 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Rotated System Adventitia Fiber Family 2 (sys5)**.

Symmetry Plane 1

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

2 Select Boundaries 2 and 5 only.

Prescribed Displacement 1

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

2 Select Boundaries 3 and 6 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 From the **Displacement in z direction** list, choose **Prescribed**.

5 In the u_{0z} text field, type `(lambda_z-1)*L`.

Boundary Load 1

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

2 Select Boundary 1 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 From the **Load type** list, choose **Pressure**.

5 In the p text field, type `p_i`.

MESH 1

Mapped 1

In the **Mesh** toolbar, click **Mapped**.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 2 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `6`.

Distribution 2

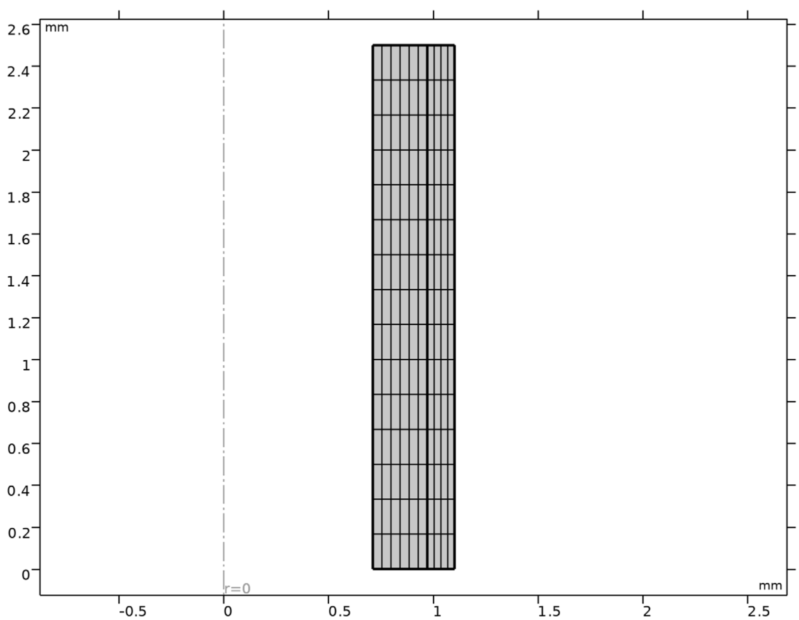
1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 5 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `4`.

5 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.



MATERIALS

Material (Media) (mat1)

Before solving, all necessary materials properties should be defined using the model parameters loaded.

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click **Material (Media) (mat1)**.

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
Lamé parameter μ	muLame	cM	N/m²	Lamé parameters
Density	rho	rhoM	kg/m³	Basic
Fiber stiffness	k1HGO	k1M	Pa	Holzapfel-Gasser-Ogden
Model parameter	k2HGO	k2M	1	Holzapfel-Gasser-Ogden
Fiber dispersion	k3HGO	k3M	1	Holzapfel-Gasser-Ogden

Material (Adventitia) (mat2)

1 In the **Model Builder** window, click **Material (Adventitia) (mat2)**.

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
Lamé parameter μ	muLame	cA	N/m²	Lamé parameters
Density	rho	rhoA	kg/m³	Basic
Fiber stiffness	k1HGO	k1A	Pa	Holzapfel-Gasser-Ogden
Model parameter	k2HGO	k2A	1	Holzapfel-Gasser-Ogden
Fiber dispersion	k3HGO	k3A	1	Holzapfel-Gasser-Ogden

STUDY 1

Now set up a study to compute the static response of the artery segment subject to combined axial stretch and internal pressure.

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

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

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

You will not need the default plots in this model.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** checkbox.

4 Click + **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
lambda_z (Axial stretch)	`1.5 1.7 1.9`	

The parameter `lambda_z` controls the axial stretch.

6 Click + **Add**.

7 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
p_i (Internal pressure)	`range(0,5,160)`	`mmHg`

Use `p_i` to vary the internal pressure from 0 to 160 mmHg in steps of 5 mmHg.

8 From the **Sweep type** list, choose **All combinations**.

9 From the **Reuse solution from previous step** list, choose **Auto**.

Using the **Auto** option for **Reuse solution for previous step** is suitable for this kind of multiparameter sweep with continuation.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

Using constant prediction for the continuation sweep improves the convergence when the solution is very nonlinear in the sweep parameter.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** node, then click **Parametric 1**.

4 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.

5 From the **Predictor** list, choose **Constant**.

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

RESULTS

1 In the **Model Builder** window, expand the **Results** node.

Before you examine the results, create **Revolution 2D** datasets. These datasets create the 3D cylindrical geometry from the 2D axisymmetric plane that was used for the computation.

Sector Revolution

1 In the **Model Builder** window, expand the **Results** > **Datasets** node.

2 Right-click **Results** > **Datasets** and choose **Revolution 2D**.

3 In the **Settings** window for **Revolution 2D**, type `Sector Revolution` in the **Label** text field.

4 Click to expand the **Revolution Layers** section. In the **Start angle** text field, type `-90`.

5 In the **Revolution angle** text field, type `225`.

Full Revolution

1 In the **Results** toolbar, click **More Datasets** and choose **Revolution 2D**.

2 In the **Settings** window for **Revolution 2D**, type `Full Revolution` in the **Label** text field.

Now duplicate the datasets and add a selection for the media. Use these in one of the plots below.

Media

1 In the **Model Builder** window, under **Results** > **Datasets** right-click **Study 1/ Solution 1 (sol1)** and choose **Duplicate**.

2 In the **Settings** window for **Solution**, type `Media` in the **Label** text field.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

4 Select Domain 1 only.

Media Sector Revolution

1 In the **Results** toolbar, click **More Datasets** and choose **Revolution 2D**.

2 In the **Settings** window for **Revolution 2D**, type `Media Sector Revolution` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Media (sol1)**.

4 Locate the **Revolution Layers** section. In the **Start angle** text field, type `-40`.

5 In the **Revolution angle** text field, type `140`.

Now duplicate the datasets and add a selection for the adventitia. Use these in one of the plots below.

Adventitia

1 Right-click **Study 1/Solution 1 (sol1)** and choose **Duplicate**.

2 In the **Settings** window for **Solution**, type `Adventitia` in the **Label** text field.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

4 Select Domain 2 only.

Adventitia Sector Revolution

1 In the **Results** toolbar, click **More Datasets** and choose **Revolution 2D**.

2 In the **Settings** window for **Revolution 2D**, type `Adventitia Sector Revolution` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Adventitia (sol1)**.

4 Locate the **Revolution Layers** section. In the **Start angle** text field, type `-50`.

5 In the **Revolution angle** text field, type `160`.

Create a 3D plot group for the radial stress distribution.

Radial Stress

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Radial Stress` in the **Label** text field.

Surface 1

1 In the **Radial Stress** toolbar, click **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Stress** > **Stress tensor (spatial frame) - N/m²** > **solid.sGprr - Stress tensor, rr-component**.

3 Locate the **Coloring and Style** section. From the **Color table transformation** list, choose **Reverse**.

Deformation 1

1 In the **Radial Stress** toolbar, click **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** checkbox. In the associated text field, type `1`.

4 In the **Radial Stress** toolbar, click **Plot**.

5 Click the **Go to Default View** button in the **Graphics** toolbar.

Create a 1D plot group to compare the pressure versus radius relationship with the data from Ref. 1.

Pressure vs. Radius

1 In the **Results** toolbar, click **1D Plot Group**.

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

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** checkbox. In the associated text field, type `Inner radius (mm)`.

6 Select the **y-axis label** checkbox. In the associated text field, type `Internal pressure (mmHg)`.

7 Select the **Flip the x- and y-axes** checkbox.

8 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Point Graph 1

1 Right-click **Pressure vs. Radius** and choose **Point Graph**.

2 Select Point 1 only.

3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Geometry** > **Coordinate (spatial frame)** > **r - r-coordinate**.

4 In the **Pressure vs. Radius** toolbar, click **Plot**.

5 Click to expand the **Legends** section. Select the **Show legends** checkbox.

6 Find the **Include** subsection. Clear the **Point** checkbox.

Global 1

1 In the **Model Builder** window, right-click **Pressure vs. Radius** and choose **Global**.

2. In the **Settings** window for **Global**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.
4. From the **Parameter selection (lambda_z)** list, choose **Last**.
5. Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
`hgo_pr_1_5(p_i)`	`mm`	`Interpolation 1`

   This is the interpolation function with data from Ref. 1. It returns the inner radius as a function of the internal pressure at an axial stretch of 1.5, 1.7, and 1.9.

6. Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.
7. From the **Color** list, choose **From theme**.
8. Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.
9. Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.
10. In the table, enter the following settings:

Legends
`Data reproduced from Ref. 1`

Global 2

1. Right-click **Global 1** and choose **Duplicate**.
2. In the **Settings** window for **Global**, locate the **y-Axis Data** section.
3. In the table, enter the following settings:

Expression	**Unit**	**Description**
`hgo_pr_1_7(p_i)`	`mm`	`Interpolation 1`
`hgo_pr_1_9(p_i)`	`mm`	`Interpolation 1`

4. Locate the **Legends** section. Clear the **Show legends** checkbox.
5. In the **Pressure vs. Radius** toolbar, click **Plot**.

Create a 3D plot group to display the fiber path.

Fiber Direction

1. In the **Results** toolbar, click **3D Plot Group**.
2. In the **Settings** window for **3D Plot Group**, type `Fiber Direction` in the **Label** text field.
3. Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Adventitia

1 Right-click **Fiber Direction** and choose **Volume**.

2 In the **Settings** window for **Volume**, type `Adventitia` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Adventitia Sector Revolution**.

4 Locate the **Expression** section. In the **Expression** text field, type `1`.

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **Custom**.

7 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.

8 Click **Define custom colors**.

9 Set the RGB values to 224, 255, and 255, respectively.

10 Click **Add to custom colors**.

11 Click **Show color palette only** or **OK** on the cross-platform desktop.

Transparency 1

1 Right-click **Adventitia** and choose **Transparency**.

2 In the **Settings** window for **Transparency**, locate the **Transparency** section.

3 Find the **Transparency** subsection. Set the **Transparency** value to **0.2**.

Media

1 In the **Model Builder** window, right-click **Adventitia** and choose **Duplicate**.

2 In the **Settings** window for **Volume**, type `Media` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Media Sector Revolution**.

4 Locate the **Coloring and Style** section. Click **Define custom colors**.

5 Set the RGB values to 252, 199, and 178, respectively.

6 Click **Add to custom colors**.

7 Click **Show color palette only** or **OK** on the cross-platform desktop.

Filter 1

1 Right-click **Media** and choose **Filter**.

2 In the **Settings** window for **Filter**, locate the **Element Selection** section.

3 In the **Logical expression for inclusion** text field, type `Z<L/2 && R<(Ri+HM*0.99)`.

Fiber Family Adventitia 1

1 In the **Model Builder** window, right-click **Fiber Direction** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, type `Fiber Family Adventitia 1` in the **Label** text field.

3 Locate the **Expression** section. In the **R-component** text field, type `solid.hmm2.fib1.a0R`.

4 In the **PHI-component** text field, type `solid.hmm2.fib1.a0PHI`.

5 In the **Z-component** text field, type `solid.hmm2.fib1.a0Z`.

6 Locate the **Data** section. From the **Dataset** list, choose **Adventitia Sector Revolution**.

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

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

9 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Color** list, choose **Blue**.

10 Find the **Line style** subsection. From the **Type** list, choose **Tube**.

Fiber Family Adventitia 2

1 Right-click **Fiber Family Adventitia 1** and choose **Duplicate**.

2 In the **Settings** window for **Streamline**, type `Fiber Family Adventitia 2` in the **Label** text field.

3 Locate the **Expression** section. In the **R-component** text field, type `solid.hmm2.fib2.a0R`.

4 In the **PHI-component** text field, type `solid.hmm2.fib2.a0PHI`.

5 In the **Z-component** text field, type `solid.hmm2.fib2.a0Z`.

6 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Fiber Family Adventitia 1**.

Fiber Family Media 1

1 In the **Model Builder** window, right-click **Fiber Family Adventitia 1** and choose **Duplicate**.

2 In the **Settings** window for **Streamline**, type `Fiber Family Media 1` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Media Sector Revolution**.

4 Locate the **Expression** section. In the **R-component** text field, type `solid.hmm1.fib1.a0R`.

5 In the **PHI-component** text field, type `solid.hmm1.fib1.a0PHI`.

6 In the **Z-component** text field, type `solid.hmm1.fib1.a0Z`.

7 Locate the **Streamline Positioning** section. In the **Density level** text field, type `8`.

8 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Color** list, choose **Red**.

Filter 1

1 Right-click **Fiber Family Media 1** and choose **Filter**.

2 In the **Settings** window for **Filter**, locate the **Element Selection** section.

3 In the **Logical expression for inclusion** text field, type `Z<L/2 && R<(Ri+HM*0.99)`.

Fiber Family Media 2

1 In the **Model Builder** window, right-click **Fiber Family Media 1** and choose **Duplicate**.

2 In the **Settings** window for **Streamline**, type `Fiber Family Media 2` in the **Label** text field.

3 Locate the **Expression** section. In the **R-component** text field, type `solid.hmm1.fib2.a0R`.

4 In the **PHI-component** text field, type `solid.hmm1.fib2.a0PHI`.

5 In the **Z-component** text field, type `solid.hmm1.fib2.a0Z`.

6 Locate the **Inherit Style** section. From the **Plot** list, choose **Fiber Family Media 1**.

7 In the **Fiber Direction** toolbar, click **Plot**.

8 In the **Graphics** window toolbar, click ▾ next to **Scene Light**, then choose **Ambient Occlusion**.