Created in COMSOL Multiphysics 5.6

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 tissues 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 deformation.

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

This 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 2D axisymmetric model. Main dimensions are reported in Figure 1.

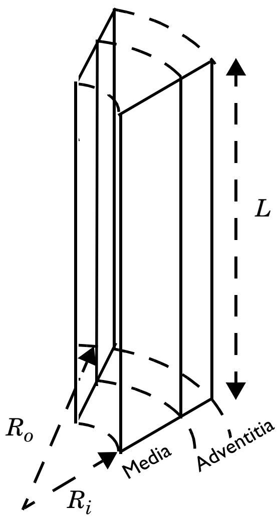


Figure 1: Carotid artery section of L = 2.5 mm in length. The inner radius R_i is 0.71 mm, the outer radius R_o is 1.1 mm, the media thickness is 0.26 mm, and the adventitia thickness is 0.13 mm.

Typical mechanical experiments measure the response of arterial sections subject to combined axial stretch and internal blood pressure. The set of boundary conditions replicate these experiments.

The symmetry boundary condition allows the bottom end of the artery to freely expand in the radial direction. On the top surface, prescribed displacements in the axial direction account for the axial stretching. 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 mathematically described 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 right Cauchy-Green tensor.

The first term describes the mechanical behavior of the elastic ground substance. The isotropic function W_1 depends on one material parameter and the first isochoric invariant $I_1(\overline{C_{el}})$, defined in the same fashion as for a neo-Hookean material (see Ref. 3 for more details on this invariant)

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

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

$$W_4 = \frac{k_1}{2k_2}(e^{k_2(I_4-1)^2} - 1) \tag{3}$$

$$W_6 = \frac{k_1}{2k_2}(e^{k_2(I_6-1)^2} - 1) \tag{4}$$

here, the fiber network is reduced to two families of fibers with material properties k_1 and k_2.

The deformation of each fiber family is measured by the invariants I_4 and I_6. You can find a detailed background for this formulation in Ref. 1 and Ref. 2.

Briefly, a family i of fibers is defined by a vector field $\mathbf{a}_{0i}$ in the undeformed direction. The fibers deform under the action of the isochoric deformation gradient, so that $\overline{F_{el}} \cdot \mathbf{a}_{0i}$ is the deformed fiber configuration. The length of $\overline{F_{el}} \cdot \mathbf{a}_{0i}$ is the fiber stretch, to be used for the constitutive equations.

The HGO model uses the square of the fiber stretches computed according to the invariants I_4 and I_6

$$I_4 = I_4(\overline{C_{el}}, \mathbf{a}_{01}) = (\overline{F_{el}} \cdot \mathbf{a}_{01}) \cdot (\overline{F_{el}} \cdot \mathbf{a}_{01}) = \mathbf{a}_{01} \cdot \overline{C_{el}} \cdot \mathbf{a}_{01} \tag{5}$$

$$I_6 = I_6(\overline{C_{el}}, \mathbf{a}_{02}) = (\overline{F_{el}} \cdot \mathbf{a}_{02}) \cdot (\overline{F_{el}} \cdot \mathbf{a}_{02}) = \mathbf{a}_{02} \cdot \overline{C_{el}} \cdot \mathbf{a}_{02} \tag{6}$$

Also, the angle β is the relative angle between $\mathbf{a}_{01}$ and $\mathbf{a}_{02}$.

The mechanical properties of both the media and adventitia are governed by these expressions. Each layer has a distinct set of material parameters c, k_1, k_2, and the initial fiber directions $\mathbf{a}_{01}$ and $\mathbf{a}_{02}$ are aligned at different angles, as shown in Figure 2.

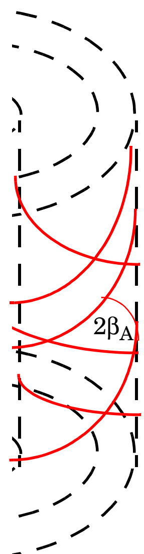


Figure 2: Angle between the two fiber families in the adventitia.

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 model computes the static response to the applied boundary conditions. Figure 3 displays the fiber layout of both the media and the adventitia fiber families.

.

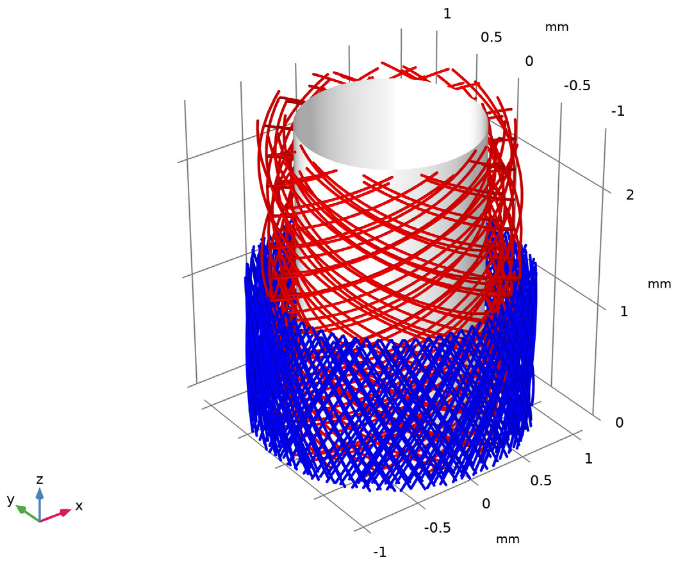


Figure 3: Fiber layout in the undeformed configuration of the media (inner, red) and the adventitia (outer, blue). Note the different angles between fiber families.

Figure 4 shows the radial stress distribution through the thickness of the wall at an axial stretch of 1.9 and an internal pressure of 160 mmHg.

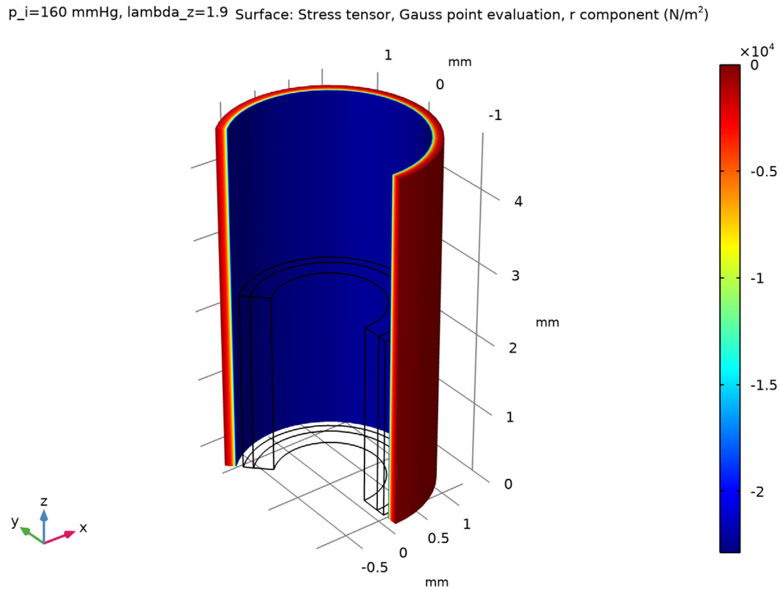


Figure 4: Radial stress distribution in the artery wall at an axial stretch of 1.9 and 160 mmHg internal pressure.

Figure 5 plots the internal pressure against the expansion of the inner radius for the entire load range. The results are in good agreements with the data reproduced from Ref. 1.

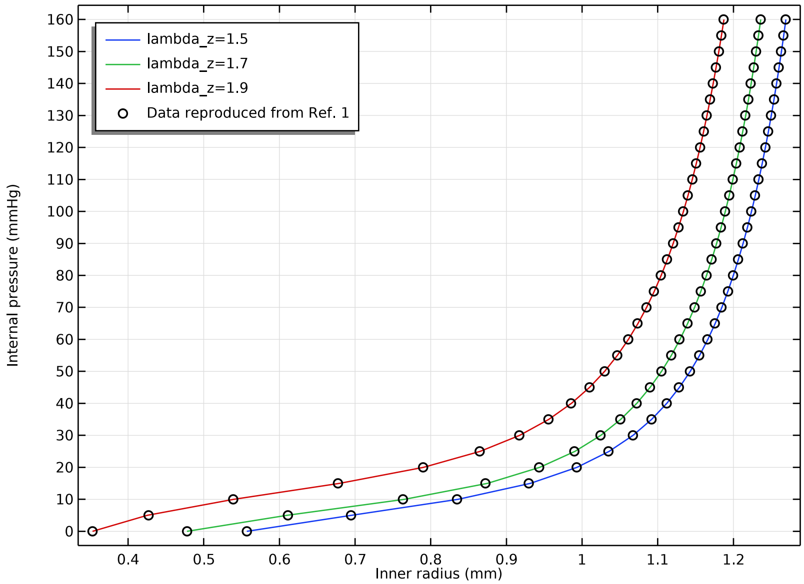


Figure 5: Plot of internal pressure vs. inner radius for three different axial stretches. Data reproduced from Ref. 1 *(circles) coincides well with the model results.*

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.

Notes About the COMSOL Implementation

The most important aspect in this model is the implementation of the HGO material model with a user-defined strain energy density function. The initial fiber directions are user-defined variables.

There are two fiber families in each arterial layer. The mathematical expressions in the HGO model are the same for the media as for the adventitia, except for the different material parameters. Only the expressions for the media are discussed below, denoted by the index `M`. Replace this index with the index `A` to obtain the expressions for the adventitia.

The vector field $\mathbf{a}_{01\mathrm{M}}$ of the first fiber family in the media is represented by the components `a01M1` (radial), `a01M2` (azimuthal), and `a01M3` (axial). The second fiber family $\mathbf{a}_{02\mathrm{M}}$ in the media has the components `a02M1`, `a02M2`, and `a02M3`.

Each fiber family has an associated invariant computed according to Equation 5. The internal variables for the isochoric right Cauchy-Green deformation tensor $\overline{C_{el}}$ in the local coordinate system are `solid.CIelIJ` (Ref. 3), where `I` and `J` are indexes running from `1` to `3`. In the **Cylindrical System**, `1` represents the radial direction, `2` the azimuthal and `3` the axial direction.

In the settings for the hyperelastic material, select the cylindrical system as a reference coordinate system. The invariant associated with the first fiber family is named `I4CIel`, and according to Equation 5 it is computed as

```
a01M1*solid.CIel11*a01M1+2*a01M1*solid.CIel12*a01M2+
2*a01M1*solid.CIel13*a01M3+a01M2*solid.CIel22*a01M2+
2*a01M2*solid.CIel23*a01M3+a01M3*solid.CIel33*a01M3.
```

The invariant `I6CIel` is defined similarly, but it uses the components of the second fiber family, `a02M1`, `a02M2`, and `a02M3`.

Once the fiber directions and related invariants are defined, implement the strain energy density functions. Equation 2 defines an isotropic isochoric function. The corresponding invariant $I_1(\overline{C_{el}})$ is defined by the variable `solid.I1CIel`, thus the expression in the media reads `cM/2*(solid.I1CIel-3)`.

The second and third terms on the right-hand side of Equation 1 are the anisotropic strain energy functions for the fiber families $\mathbf{a}_{01\mathrm{M}}$ and $\mathbf{a}_{02\mathrm{M}}$. With the definition of the fiber invariants in the media for the fiber family $\mathbf{a}_{01\mathrm{M}}$, Equation 3 becomes

```
k1M/(2*k2M)*(exp(k2M*(I4CIel-1)^2)-1)*(I4CIel>1).
```

The last factor, `(I4CIel>1)`, evaluates to zero if the fiber stretch is smaller than one. This means that the fibers only contribute to tensile stress. To implement Equation 4, simply replace `I4CIel` with `I6CIel`, which corresponds to the second fiber family $\mathbf{a}_{02\mathrm{M}}$.

This model also shows how to use the Curvilinear Coordinates interface to plot the configuration of the fiber families. The interface takes care of the mapping from the global coordinate system in 2D axisymmetric component to the cylindrical coordinates 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**.

Use Curvilinear Coordinates to visualize the fiber layout. Add it four times, once for each fiber family.

1 In the **Select Physics** tree, select **Mathematics>Curvilinear Coordinates (cc)**.

2 Click **Add** four times.

3 Click **Study**.

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

5 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 In the **Number of arguments** text field, type `1`.

7 Click **Import**.

8 Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file
hgo_pr_1_5	1
hgo_pr_1_7	2
hgo_pr_1_9	3

9 Locate the **Units** section. In the **Arguments** text field, type `kPa`.

10 In the **Function** text field, type `mm`.

GEOMETRY 1

Construct the model geometry by first drawing two circular sections on a work plane. Then, form a difference between them and extrude it.

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`.

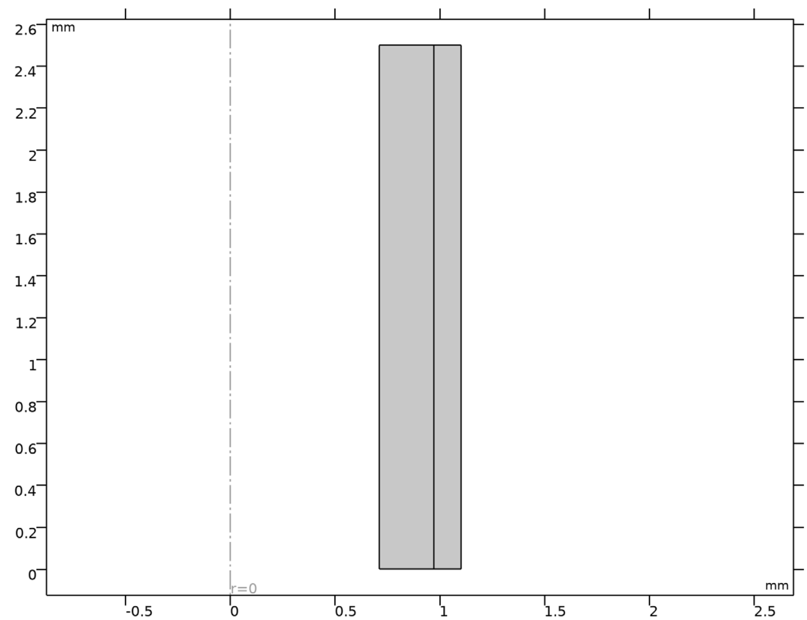
6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
`Layer 1`	`HA`

7 Clear the **Layers on bottom** check box.

8 Select the **Layers to the right** check box.

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



DEFINITIONS

Load all model variables from files. These define the initial directions of all fiber families. The files also contain the expressions for the user defined strain energy density functions.

Fiber Directions, Media

1 In the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 1 only.

5 Locate the **Variables** section. Click **Load from File**.

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

Fiber Directions, Adventitia

1 In the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 2 only.

5 Locate the **Variables** section. Click **Load from File**.

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

Strain Energy Density, Media

1 In the **Home** toolbar, click **Variables** and choose **Local Variables**.

2 In the **Settings** window for **Variables**, type `Strain Energy Density, Media` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 1 only.

5 Locate the **Variables** section. Click **Load from File**.

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

Strain Energy Density, Adventitia

1 In the **Home** toolbar, click **Variables** and choose **Local Variables**.

2 In the **Settings** window for **Variables**, type `Strain Energy Density, Adventitia` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 2 only.

5 Locate the **Variables** section. Click **Load from File**.

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

SOLID MECHANICS (SOLID)

Hyperelastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Material Models>Hyperelastic Material**.

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

3 From the **Selection** list, choose **All domains**.

4 Locate the **Hyperelastic Material** section. From the **Material model** list, choose **User defined**.

5 From the **Compressibility** list, choose **Incompressible material**.

The HGO model is incompressible, select **Incompressible material** to apply a mixed formulation.

6 In the W_{siso} text field, type `W1+W4+W6`.

This is the isochoric part of the strain energy density function. The functions `W1`, `W4` and `W6` are defined with different properties in the media and adventitia.

7 From the ρ list, choose **User defined**. In the associated text field, type `1100`.

8 In the **Model Builder** window, click **Solid Mechanics (solid)**.

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

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

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 Select the **Prescribed in z direction** check box.

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

Use lambda_z as a continuation parameter in the solver to vary axial stretch.

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.

Use p_i as a continuation parameter in the solver settings to vary the internal pressure.

CURVILINEAR COORDINATES (CC)

Now set up the curvilinear coordinates for all fiber families by adding a user defined vector field in the cylindrical system for the components of the fiber directions.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Curvilinear Coordinates (cc)**.

2 Select Domain 1 only.

User Defined 1

In the **Physics** toolbar, click **Domains** and choose **User Defined**.

Set the components of the vector field to the cylindrical components of the first fiber family.

1 In the **Settings** window for **User Defined**, locate the **User Defined** section.

2 Specify the **u** vector as

a01M1	R
a01M2	PHI
a01M3	Z

3 Set up the other curvilinear coordinates similarly according to the table:

	Curvilinear Coordinates (cc)	**Curvilinear Coordinates 2 (cc2)**	**Curvilinear Coordinates 3 (cc3)**	**Curvilinear Coordinates 4 (cc4)**
Domain selection	1	1	2	2
u vector, R component	a01M1	a02M1	a01A1	a02A1
u vector, PHI component	a01M2	a02M2	a01A2	a02A2
u vector, Z component	a01M3	a02M3	a01A3	a02A3

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

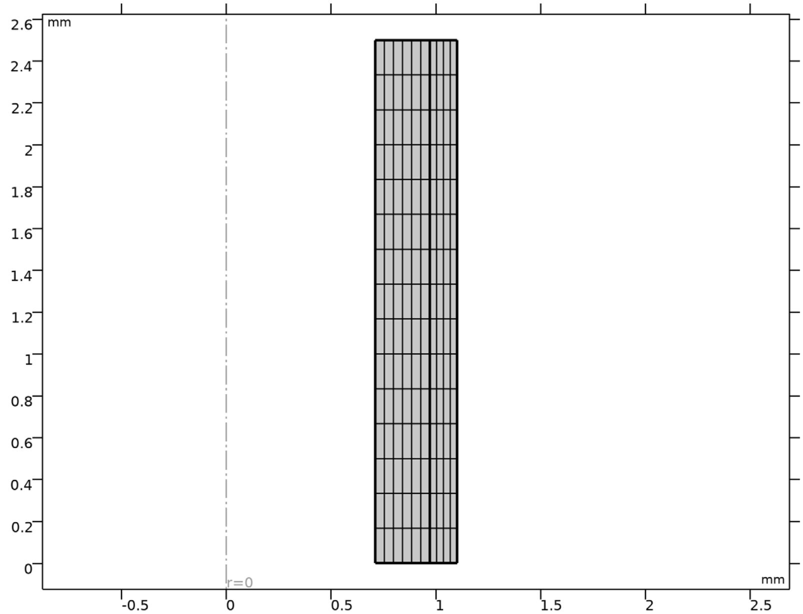
1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

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**.



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** check box.

   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** check box.
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 with 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 convergence when the solution is very nonlinear in the swept 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. Those datasets create the cylindrical geometry from the 2D axisymmetric plane that was used for the computation.

Study 1/Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Results>Datasets** node, then click **Study 1/Solution 1 (sol1)**.

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

3 From the **Frame** list, choose **Material (R, PHI, Z)**.

Sector Revolution

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

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

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

4 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. Use these in one of the plots below.

Inner Wall

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

2 In the **Settings** window for **Solution**, type `Inner Wall` 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 **Boundary**.

4 Select Boundary 1 only.

Inner Wall, Revolution

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

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

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

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 (Gauss points)>Stress tensor, Gauss point evaluation (spatial frame) - N/m²>solid.sGpr - Stress tensor, Gauss point evaluation, r component**.

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** check box.

4 In the associated text field, type `1`.

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

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

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

Pressure vs. Radius

1 In the **Home** toolbar, click **Add Plot Group** and choose **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. Select the **x-axis label** check box.

5 In the associated text field, type `Inner radius (mm)`.

6 Select the **y-axis label** check box.

7 In the associated text field, type `Internal pressure (mmHg)`.

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**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `p_i`.

5 In the **Unit** field, type `mmHg`.

6 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Geometry>Coordinate (spatial frame)>r - r-coordinate**.

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

8 Click to expand the **Legends** section. Select the **Show legends** check box.

9 Find the **Include** subsection. Clear the **Point** check box.

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
p_i	mmHg	Internal pressure

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** text field, type `hgo_pr_1_5(p_i)`.

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

8 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

9 From the **Color** list, choose **From theme**.

10 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

11 From the **Positioning** list, choose **In data points**.

12 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

13 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 **x-Axis Data** section.

3 In the **Expression** text field, type `hgo_pr_1_7(p_i)`.

This is the inner radius as a function of internal pressure at an axial stretch of 1.7.

4 Locate the **Legends** section. Clear the **Show legends** check box.

Global 3

1 Right-click **Global 2** and choose **Duplicate**.

2 In the **Settings** window for **Global**, locate the **x-Axis Data** section.

3 In the **Expression** text field, type `hgo_pr_1_9(p_i)`.

This is the inner radius as a function of internal pressure at an axial stretch of 1.9.

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

Fiber Directions

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

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

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

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Streamline 1

1 In the **Fiber Directions** toolbar, click **Streamline**.

2 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Curvilinear Coordinates>cc.vR,...,cc.vZ - Vector field (material and geometry frames)**.

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

4 In the **Separating distance** text field, type `0.05`.

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

6 In the **Tube radius expression** text field, type `0.1`.

This is the vector field of the first Curvilinear Coordinate physics and corresponds to the first fiber family in the media.

Streamline 2

1 Right-click **Streamline 1** and choose **Duplicate**.

2 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Curvilinear Coordinates 2>cc2.vR,...,cc2.vZ - Vector field (material and geometry frames)**.

This field is the second fiber family in the media.

Streamline 3

1 Right-click **Streamline 2** and choose **Duplicate**.

This field is the first fiber family in the adventitia.

2 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Curvilinear Coordinates 3>cc3.vR,...,cc3.vZ - Vector field (material and geometry frames)**.

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

4 Locate the **Streamline Positioning** section. In the **Separating distance** text field, type `0.03`.

Filter 1

1 Right-click **Streamline 3** and choose **Filter**.

This boolean expression restricts the plotting of fibers to the half of the geometry.

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`.

Streamline 4

1 In the **Model Builder** window, under **Results>Fiber Directions** right-click **Streamline 3** and choose **Duplicate**.

2 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Curvilinear Coordinates 4>cc4.vR,...,cc4.vZ - Vector field (material and geometry frames)**.

This is the second fiber family in the adventitia.

Fiber Directions

In the **Model Builder** window, click **Fiber Directions**.

Surface 1

1 In the **Fiber Directions** toolbar, click **Surface**.

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

3 From the **Dataset** list, choose **Inner Wall, 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 **White**.

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