Created in COMSOL Multiphysics 5.6

Arterial Wall Viscoelasticity

Introduction

Arteries are blood vessels that carry freshly oxygenated blood from the heart throughout the rest of the body. The Holzapfel–Gasser–Ogden (HGO) model proposes a mechanical description of the young healthy arteries in Ref. 1 based on anisotropic hyperelastic properties, which is implemented in the Arterial Wall Mechanics example. Here we study the dynamic behavior of the artery, based on Ref. 2 and calculate the time-dependent response given a sudden axial stretching.

Model Definition

The geometry, physics interface, and material models are the same as in the example Arterial Wall Mechanics, see also Figure 1.

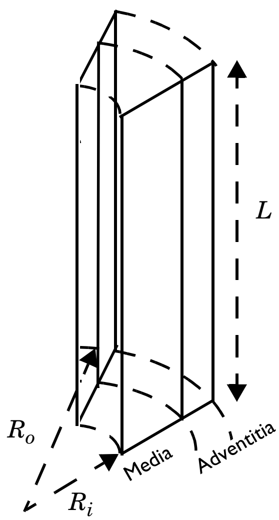


Figure 1: Carotid artery section made of a media layer and an adventitia layer.

The two modeled layers (media and adventitia) are described by the anisotropic HGO hyperelastic material model:

$$W_s = W_1 + W_4 + W_6 = W_{\mathrm{iso}}$$

$$S = S_{\mathrm{iso}} + S_{\mathrm{vol}} = \frac{\partial W_{\mathrm{iso}}}{\partial \varepsilon} - p_{\mathrm{w}} J C^{-1}$$

Only the media layer includes viscoelastic behavior. The generalized Maxwell viscoelastic model is used to represent relaxations time at different time-scales (Ref. 2):

$$S = S_{\mathrm{iso}} + S_{\mathrm{vol}} + \sum_m q_m \tag{1}$$

For each branch of the generalized Maxwell model, the viscoelastic stress follows the equation:

$$\dot{Q}_m + \frac{Q_m}{\tau_m} = \beta_m \dot{S}_{\mathrm{iso}}$$

where τ_m is the relaxation time and β_m is the energy factor per branch. Applying a variable change $q_m = \beta_m S_{\mathrm{iso}} - Q_m$ gives

$$\tau_m \dot{q}_m + q_m = \beta_m S_{\mathrm{iso}} \tag{2}$$

In this example, a generalized Maxwell viscoelastic model with five branches is used with the following values taken from Ref. 2:

Branch	**Energy factor**	**Relaxation time**
1	0.3353	0.001 s
2	0.286	0.01 s
3	0.298	0.1 s
4	0.285	1 s
5	0.348	10 s

The artery is first loaded with an internal pressure of 100 mmHg and an initial axial stretch of 1.5. After initialization, the stretch is increased to 1.7 and the viscoelastic relaxation is calculated.

Results and Discussion

The total force is computed by integrating the axial stress on the top section surface. The plot shown in Figure 2 is similar to the force relaxation presented in Ref. 2. The force relaxes almost linearly from 10^{-3} s to 10 s due to the wide range of relaxation times.

Moreover, Figure 3 shows the evolution of total viscoelastic stress and viscoelastic stress for each branch over time. The viscoelastic stress for the branches is calculated with the expression $Q_m = \beta_m S_{iso} - q_m$. The viscoelastic stress relaxes from its initial value to zero.

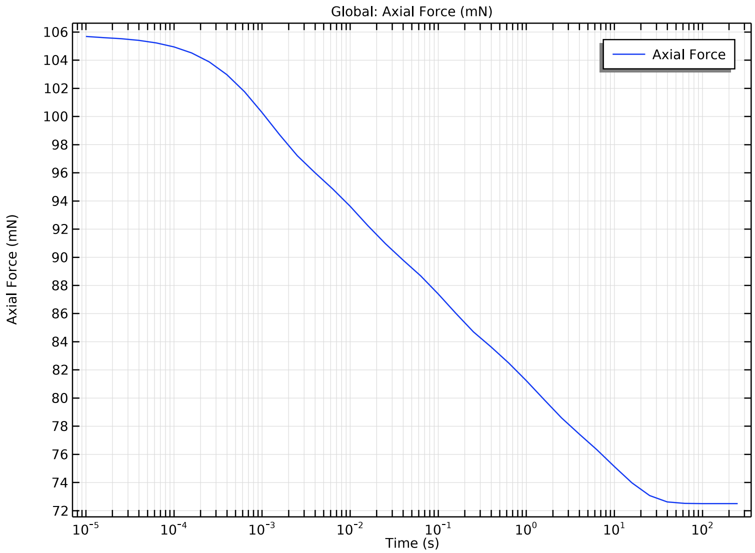


Figure 2: Relaxation of the axial force after stretching.

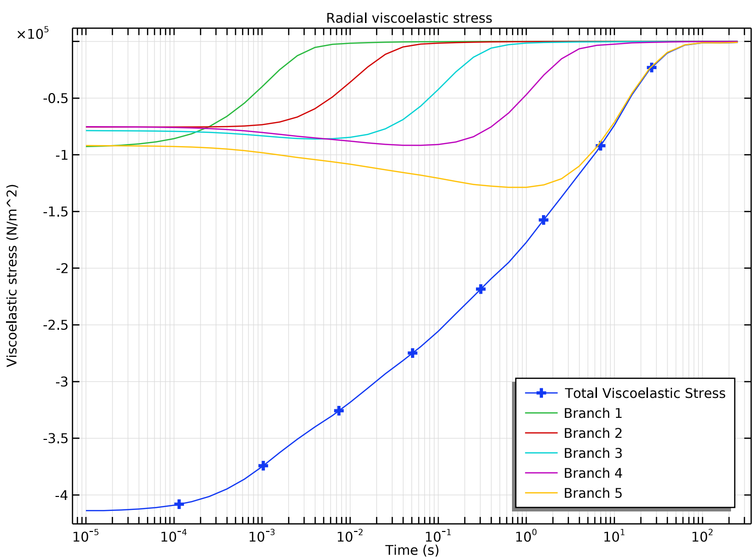


Figure 3: Variation of viscoelastic stress. Both the total stress and the stress in each branch is shown.

Notes About the COMSOL Implementation

A stationary study step is needed to prestress the artery with initial pressure and stretch. As this initial state is assumed to be a steady-state, the static stiffness in **Viscoelasticity** node must be set to **Long-term**. This ensures that the viscoelastic model has no effects in the stationary step.

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.A. Holzapfel, T.C. Gasser, M. Stadler, "A Structural Model for the Viscoelastic Behavior of Arterial Walls: Continuum Formulation and Finite Element Analysis", *European Journal of Mechanics A/Solid*, vol.21, pp. 441–463, 2002

Application Library path: `Nonlinear_Structural_Materials_Module/Viscoelasticity/arterial_wall_viscoelasticity`

Modeling Instructions

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

From the Application Libraries root, browse to the folder `Nonlinear_Structural_Materials_Module/Hyperelasticity` and double-click the file `arterial_wall_mechanics.mph`.

GLOBAL DEFINITIONS

Parameters 1

Set the parameters used for loads and constraints in the time dependent study. Parameter `t` is the time, and is needed to define the prescribed displacement in the stationary study step.

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
p_i	100[mmHg]	13332 Pa	Internal pressure
lambda_z	1.7	1.7	Axial stretch
lambda_z0	1.5	1.5	Initial axial stretch
t	0[s]	0 s	Time

Create a step function to apply the stretch in the time-dependent study.

Step 1 (step1)

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

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

3 Locate the **Parameters** section. In the **Location** text field, type `5e-6`.

4 In the **From** text field, type `lambda_z0`.

5 In the **To** text field, type `lambda_z`.

6 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `1e-5`.

7 Click **Plot**.

COMPONENT 1 (COMP1)

In the **Model Builder** window, expand the **Component 1 (comp1)** node.

SOLID MECHANICS (SOLID)

Hyperelastic Material 1

Add a generalized Maxwell viscoelasticity model according to Ref. 1.

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

Viscoelasticity 1

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

2 Select Domain 1 only.

3 In the **Settings** window for **Viscoelasticity**, locate the **Viscoelasticity Model** section.

4 Click + **Add** four times.

5 In the table, enter the following settings:

Branch	Energy factor (1)	Relaxation time (s)
1	0.353	0.001
2	0.286	0.01
3	0.298	0.1
4	0.285	1
5	0.348	10

Prescribed Displacement 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** right-click **Prescribed Displacement 1** and choose **Duplicate**.

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

3 In the u_{0z} text field, type `(lambda(t[1/s])-1)*L`.

DEFINITIONS

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

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

4 Select Boundaries 3 and 6 only.

MESH 2

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

Mapped 1

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

Distribution 1

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

2 Select Boundaries 2 and 3 only.

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

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type `20`.

6 In the **Element ratio** text field, type `10`.

7 Select the **Reverse direction** check box.

Distribution 2

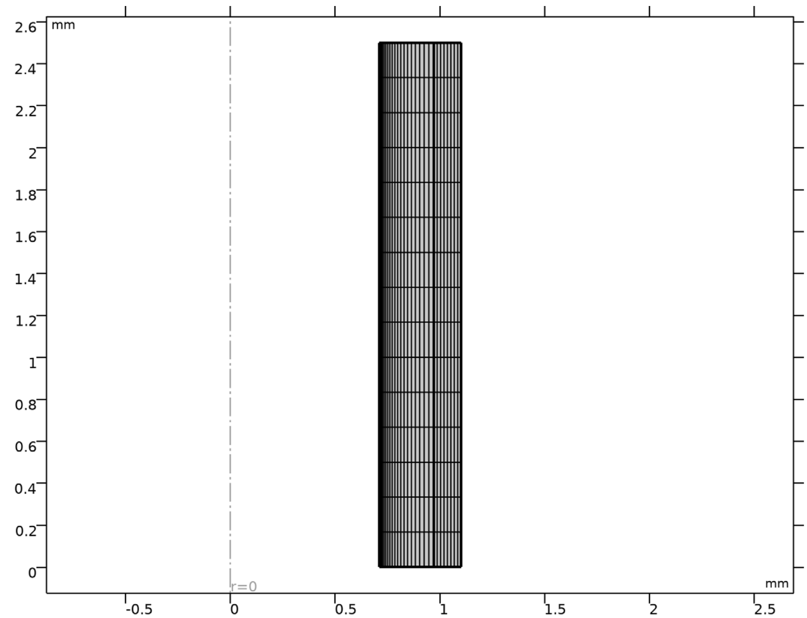
1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

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 8.

5 In the **Model Builder** window, right-click **Mesh 2** and choose **Build All**.



Add a new study. The first stationary step is used to prestress the artery with a stretch and an internal pressure. Select **Long-term stiffness** in the **Viscoelasticity** node, so the viscoelastic effect is disabled. The time-dependent step is used to compute the dynamic response to an additional stretch.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check boxes for **Curvilinear Coordinates (cc)**, **Curvilinear Coordinates 2 (cc2)**, **Curvilinear Coordinates 3 (cc3)**, and **Curvilinear Coordinates 4 (cc4)**.

5 Click **Add Study** in the window toolbar.

6 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Stationary

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

2 Select the **Auxiliary sweep** check box.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
p_i (Internal pressure)	`range(0,10,100)`	mmHg

Time Dependent

1 In the **Study** toolbar, click **Study Steps** and choose **Time Dependent> Time Dependent**.

2 In this study clear the **Solve** check box for all **Curvilinear Coordinates** interfaces.

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

4 In the **Output times** text field, type `range(0,0.5e-6,9.5e-6) 10^{range(-5,0.2, 2.4)}`.

5 From the **Tolerance** list, choose **User controlled**.

6 In the **Relative tolerance** text field, type `0.001`.

7 Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

8 From the **Method** list, choose **Solution**.

9 From the **Study** list, choose **Study 2, Stationary**.

10 From the **Selection** list, choose **Last**.

Solution 2 (sol2)

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

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Time-Dependent Solver 1**.

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**.

5 In the **Model Builder** window, click **Study 2**.

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

7 Clear the **Generate default plots** check box.

8 In the **Study** toolbar, click **=** **Compute**.

RESULTS

Study 2/Solution 2 (sol2)

1 In the **Model Builder** window, expand the **Results>Datasets** node, then click **Study 2/Solution 2 (sol2)**.

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

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

Integrate the axial stress on the top surfaces to calculate the reaction force and reproduce Figure 2.

Force Relaxation

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

2 In the **Settings** window for **1D Plot Group**, type `Force Relaxation` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

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

5 In the **Times (s)** text field, type `10^{range(-5,0.2,2.4)}`.

6 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

7 In the **Force Relaxation** toolbar, click **Plot**.

Global 1

1 Right-click **Force Relaxation** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	**Unit**	**Description**
`intop1(solid.sz)`	`mN`	`Axial Force`

4 In the **Force Relaxation** toolbar, click **Plot**.

Add a new plot group to plot viscoelastic stress and reproduce Figure 3.

Viscoelastic Stress

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

2 In the **Settings** window for **1D Plot Group**, type `Viscoelastic Stress` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

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

5 In the **Times (s)** text field, type `10^{range(-5,0.2,2.4)}`.

6 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

7 In the **Title** text area, type `Radial viscoelastic stress`.

8 Locate the **Plot Settings** section. Select the **y-axis label** check box.

9 In the associated text field, type `Viscoelastic stress (N/m^2)`.

10 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Point Graph 1

1 Right-click **Viscoelastic Stress** 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)>Solid Mechanics>Stress>Viscoelastic stress tensor (material and geometry frames) - N/m²>solid.SqRR - Viscoelastic stress tensor, RR component**.

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

5 From the **Legends** list, choose **Manual**.

6 In the table, enter the following settings:

Legends
`Total Viscoelastic Stress`

Point Graph 2

1 Right-click **Point Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `solid.hmm1.vis1.betavm1*solid.Sliso11-solid.hmm1.vis1.qm1_11`.

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

Legends
Branch 1

5 Duplicate the point graph four times and change properties according to the table:

Point graph	Expression of y-Axis Data	Legends
Point Graph 3	solid.hmm1.vis1.betavm2*solid.Sliso11-solid.hmm1.vis1.qm2_11	Branch 2
Point Graph 4	solid.hmm1.vis1.betavm3*solid.Sliso11-solid.hmm1.vis1.qm3_11	Branch 3
Point Graph 5	solid.hmm1.vis1.betavm4*solid.Sliso11-solid.hmm1.vis1.qm4_11	Branch 4
Point Graph 6	solid.hmm1.vis1.betavm5*solid.Sliso11-solid.hmm1.vis1.qm5_11	Branch 5

Point Graph 1

1 In the **Model Builder** window, click **Point Graph 1**.

2 In the **Settings** window for **Point Graph**, click to expand the **Coloring and Style** section.

3 Find the **Line markers** subsection. From the **Marker** list, choose **Plus sign**.

Viscoelastic Stress

1 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

2 In the **Model Builder** window, click **Viscoelastic Stress**.

3 In the **Viscoelastic Stress** toolbar, click **Plot**.

You can group plot groups to improve the clarity of the Results tree.

Radial Stress

In the **Model Builder** window, right-click **Radial Stress** and choose **Group**.

Stationary Results

In the **Settings** window for **Group**, type `Stationary Results` in the **Label** text field.

Pressure vs. Radius

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

Stationary Results

In the **Model Builder** window, right-click **Stationary Results** and choose **Move Down**.

Force Relaxation

In the **Model Builder** window, right-click **Force Relaxation** and choose **Group**.

Viscoelasticity Results

In the **Settings** window for **Group**, type `Viscoelasticity Results` in the **Label** text field.

Viscoelastic Stress

In the **Model Builder** window, right-click **Viscoelastic Stress** and choose **Viscoelasticity Results**.

Disable the new prescribed displacement and the viscoelasticity nodes in study 1 in order to keep it in its original state.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Prescribed Displacement 2**.

5 Click **Disable**.

6 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Hyperelastic Material 1>Viscoelasticity 1**.

7 Click **Disable**.