Plastic Deformation During the Expansion of a Biomedical Stent

Introduction

Percutaneous transluminal angioplasty with stenting is a widely spread method for the treatment of atherosclerosis. During the procedure, a stent is deployed into the artery by using a balloon as an expander. Once the balloon-stent package is in place, the balloon is inflated to expand the stent. The balloon is then deflated and removed, but the stent remains expanded to act as a scaffold, keeping the blood vessel open.

Stent design is of significance for this procedure, since serious damage can be inflicted to the artery during the expansion procedure. One of the most common defect is the non-uniform deformation of the stent, where the ends expand more than the middle section, phenomenon which is also called *dogboning*. Foreshortening of the stent can also damage the artery, and it could make the positioning difficult.

The dogboning is defined according to

$$\text{dogboning} = \frac{r_{\text{distal}} - r_{\text{central}}}{r_{\text{distal}}}$$

where r_{distal} and r_{central} are the radii at the end and middle of the stent, respectively.

The foreshortening is defined as

$$\text{foreshortening} = \frac{L_0 - L_{\text{load}}}{L_0}$$

here, L_0 is the original length of the stent and L_{load} is the deformed length of the stent.

Other common parameters in stent design are the longitudinal and radial recoil. These parameters give information on the stent behavior when removing the inflated balloon.

The longitudinal recoil is defined as

$$L_{\text{recoil}} = \frac{L_{\text{load}} - L_{\text{unload}}}{L_{\text{load}}}$$

here, L_{unload} is the length of the stent once the balloon is removed, and L_{load} is the length of the stent when the balloon is fully inflated.

The radial recoil can be defined as follow

$$R_{\text{recoil}} = \frac{R_{\text{load}} - R_{\text{unload}}}{R_{\text{load}}}$$

here, R_{unload} is the radius of the stent once the balloon is removed, and R_{load} is the radius of the stent when the balloon is fully inflated.

To check the viability of a stent design, you can study the deformation process under the influence of the radial pressure that expands the stent. With this example you can both monitor the dogboning and foreshortening effects, and draw conclusions on how to change the geometry design parameters for optimum performance.

Model Definition

The model studies the Palmaz-Schatz stent model. Due to the stent's circumferential and longitudinal symmetry, it is possible to model only one twenty-forth of the geometry. Figure 1 shows the geometry used in the study, represented with the meshed domain.

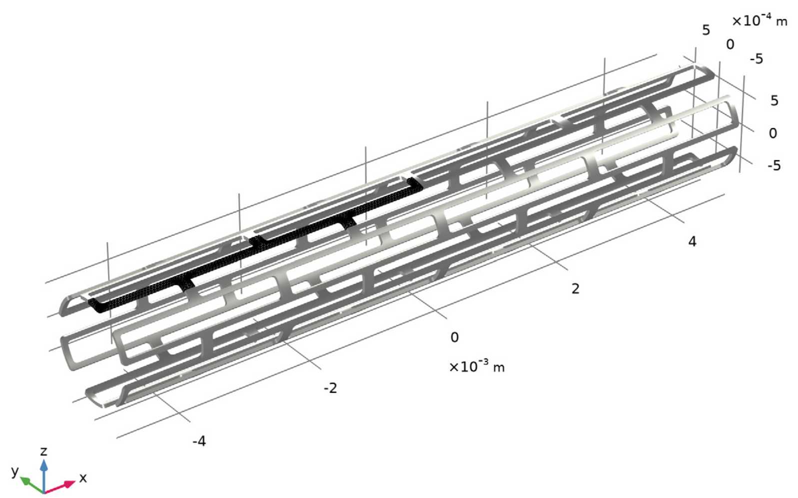

Figure 1: The reduced geometry used in the study (meshed) and the full stent geometry.

The main focus of the study consists in the stress evaluation in the stent. The angioplasty balloon is assumed to stretch with a maximum expansion radius of 2 mm.

MATERIAL

The stent is made of stainless steel. The material parameters are given in the following table.

MATERIAL PROPERTY	VALUE
Young's modulus	193[GPa]
Poisson's ratio	0.27
Initial yield stress	207[MPa]
Isotropic tangent modulus	692[MPa]

LOADS

Apply a radial outward pressure on the inner surface of the stent to represent the balloon expansion.

Results and Discussion

The stent is expanded from an original diameter of 0.74 mm to a diameter of 2 mm in the middle section. Figure 2 shows the stress distribution at maximum balloon inflation, and Figure 3 shows the residual stress after the balloon deflation. The residual stress follows form the plastic deformation as shown in Figure 4.

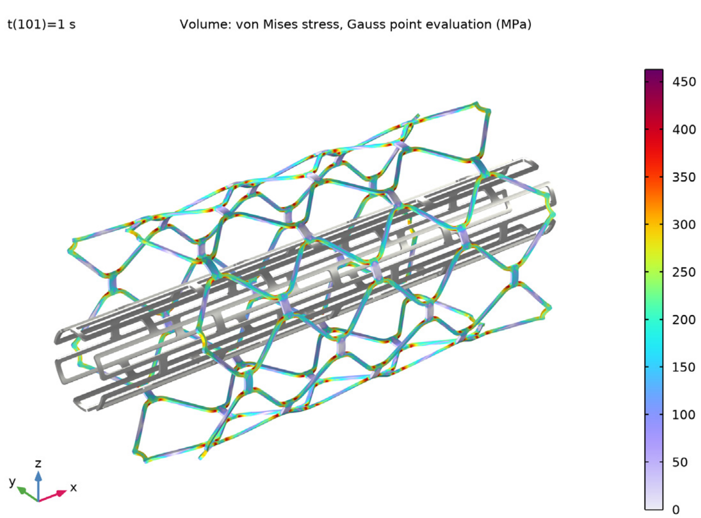

Figure 2: Maximum stress in the stent during the balloon inflation.

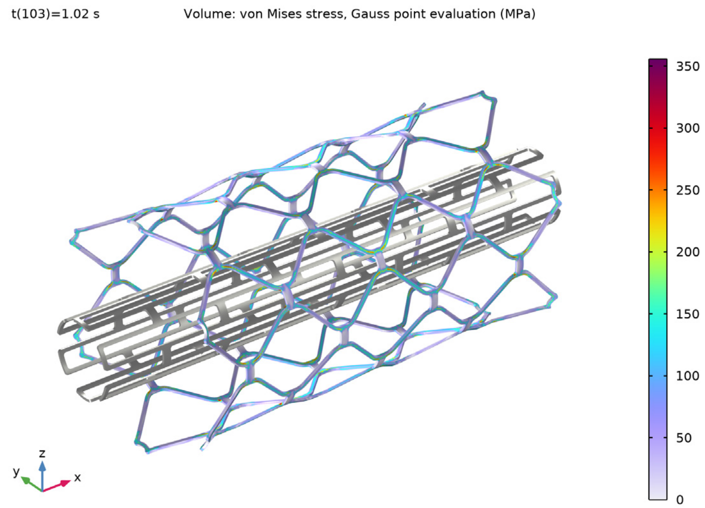

Figure 3: Residual stress in the stent after deflation of the balloon.

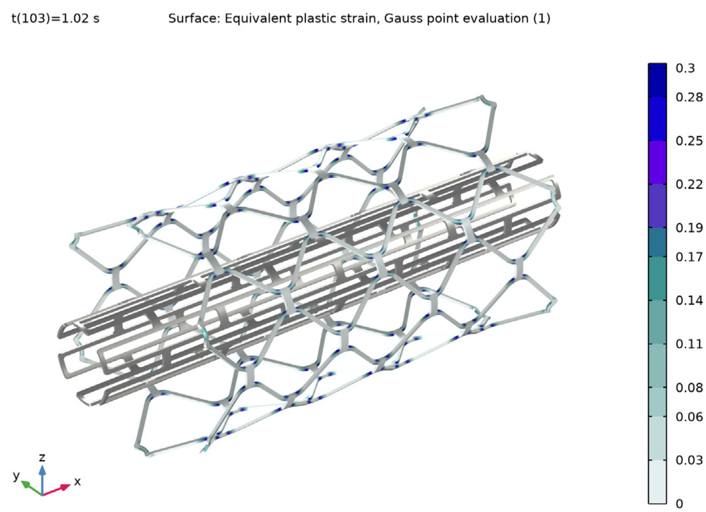

Figure 4: Equivalent plastic strain in the stent after deflation of the balloon.

In Figure 5, you can see the evolution of the dogboning and foreshortening effects with respect to the pressure during the inflation of the balloon. The longitudinal recoil is about –0.9%, the distal radial recoil is about 0.4%, and the central radial recoil is about 0.7%.

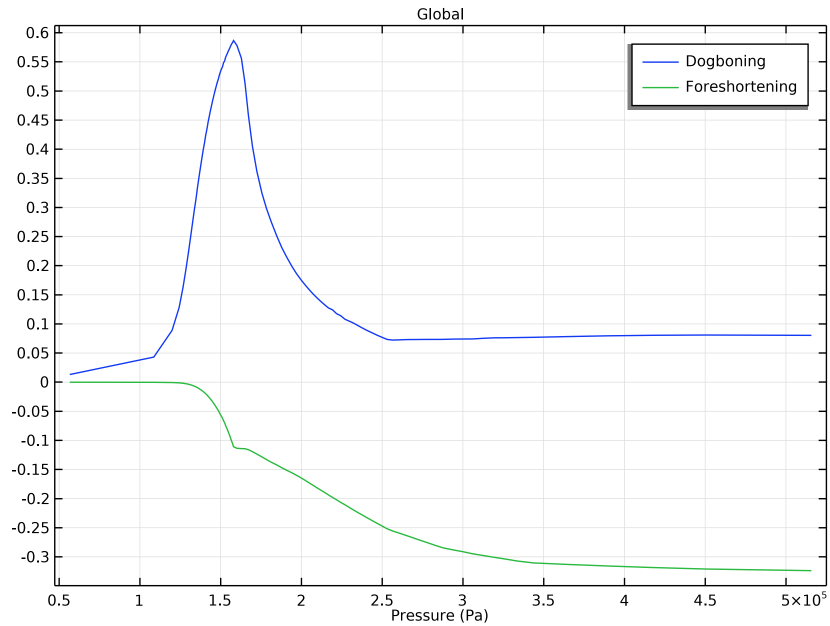

Figure 5: Stent dogboning (blue) and foreshortening (green) versus pressure inside the angioplasty balloon.

Notes About the COMSOL Implementation

The maximum radius of the angioplasty balloon is represented with a step function, the pressure is applied as long as the inner radius of the stent is smaller than the maximum balloon radius. Above this limit the pressure is set to zero.

For a highly nonlinear problem like this, the choice of the continuation parameter can improve the convergence during the computation of the solution. A displacement control parameter is usually better than a load parameter. In this example, the average displacement of the stent's inner radius is prescribed, and a **Global Equation** is used to compute the corresponding applied pressure load.

Application Library path: `Nonlinear_Structural_Materials_Module/Plasticity/biomedical_stent`

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 **3D**.
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**.

GEOMETRY 1

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.
2 In the **Settings** window for **Import**, locate the **Import** section.
3 From the **Source** list, choose **COMSOL Multiphysics file**.
4 Click **Browse**.
5 Browse to the model's Application Libraries folder and double-click the file `biomedical_stent.mphbin`.
6 Click **Import**.
7 Click the **Zoom Extents** button in the **Graphics** toolbar.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

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

Plasticity 1

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

2 In the **Settings** window for **Plasticity**, locate the **Plasticity Model** section.

3 From the **Formulation** list, choose **Large strains**.

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
Young's modulus	E	193[GPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.27	1	Young's modulus and Poisson's ratio
Density	rho	7050	kg/m³	Basic
Initial yield stress	sigmags	207[MPa]	Pa	Elastoplastic material model
Isotropic tangent modulus	Et	692[MPa]	Pa	Elastoplastic material model

Choose the steel material type to improve the visualization of the stent during postprocessing.

4 Click to expand the **Appearance** section. From the **Material type** list, choose **Steel**.

DEFINITIONS

Step 1 (step1)

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

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

3 In the **Location** text field, type 2[mm].

4 In the **From** text field, type 1.

5 In the **To** text field, type 0.

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

Variables 1

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

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
r	sqrt(y^2+z^2)	m	Radial distance from x-axis

Average 1 (aveop1)

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 **Edge**.

4 Select Edge 28 only.

5 Locate the **Advanced** section. From the **Frame** list, choose **Material (X, Y, Z)**.

Piecewise 1 (pw1)

1 In the **Definitions** toolbar, click **Piecewise**.

2 In the **Settings** window for **Piecewise**, type r0 in the **Function name** text field.

3 Locate the **Definition** section. In the **Argument** text field, type t.

4 Find the **Intervals** subsection. In the table, enter the following settings:

Start	End	Function
0	1	(2e-3-7.1e-4)*t+7.1e-4
1	2	(2e-3-7.1e-4)*(1-t)+2e-3

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

6 In the **Function** text field, type m.

7 Click **Plot**.

SOLID MECHANICS (SOLID)

Symmetry 1

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

2 Select Boundaries 5, 12, 18, 24, 30, and 31 only.

Boundary Load 1

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

2 Select Boundary 4 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*step1(r)`.

6 Click the **Show More Options** button in the **Model Builder** toolbar.

7 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation-Based Contributions**.

8 Click **OK**.

Global Equations 1

1 In the **Physics** toolbar, click **Global** and choose **Global Equations**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
p	aveop1(r)-r0(t)	0	0	Pressure

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `pressure` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Pressure (Pa)**.

8 Click **OK**.

9 In the **Settings** window for **Global Equations**, locate the **Units** section.

10 Click **Select Source Term Quantity**.

11 In the **Physical Quantity** dialog box, type `length` in the text field.

12 Click **Filter**.

13 In the tree, select **General>Length (m)**.

14 Click **OK**.

MESH 1

Free Triangular 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

2 Select Boundary 3 only.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section.

5 Select the **Maximum element size** check box. In the associated text field, type `4.5e-5`.

6 Select the **Minimum element size** check box. In the associated text field, type `4e-6`.

7 Select the **Maximum element growth rate** check box. In the associated text field, type `1.4`.

8 Select the **Curvature factor** check box. In the associated text field, type `0.3`.

Swept 1

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

Distribution 1

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

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

3 In the **Number of elements** text field, type `2`.

4 Click **Build All**.

DEFINITIONS

Create variables for the results processing.

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 **Point**.

4 Select Point 57 only.

5 In the **Operator name** text field, type `central`.

6 Locate the **Advanced** section. From the **Frame** list, choose **Material (X, Y, Z)**.

Integration 2 (intop2)

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 **Point**.

4 Select Point 3 only.

5 In the **Operator name** text field, type `distal`.

6 Locate the **Advanced** section. From the **Frame** list, choose **Material (X, Y, Z)**.

Variables 1

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

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
dogboning	(distal(r)-central(r))/distal(r)		Dogboning
length	2*abs(distal(x)-central(x))	m	Length of the deformed stent
L0	2*abs(distal(X)-central(X))	m	Length of the undeformed stent
foreshortening	(length-L0)/length		Foreshortening

GLOBAL DEFINITIONS

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 In the table, enter the following settings:

Name	Expression	Value	Description
t	0[s]	0 s	Time

STUDY 1

Step 1: Stationary

Set up an auxiliary continuation sweep for the `t` parameter.

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
t (Time)	range(0,1e-2,1.5)	s

Solution 1 (sol1)

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

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)>Dependent Variables 1** node, then click **Pressure (comp1.ODE1)**.

4 In the **Settings** window for **State**, locate the **Scaling** section.

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

6 In the **Scale** text field, type 1e6.

Add a stop condition to prevent the computed pressure from becoming negative.

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

8 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1>Parametric 1** and choose **Stop Condition**.

9 In the **Settings** window for **Stop Condition**, locate the **Stop Expressions** section.

10 Click **Add**.

11 In the table, enter the following settings:

Stop expression	Stop if	Active	Description
comp1.p<0	True (>=1)	√	Stop expression 1

Specify that the solution is to be stored just before the stop condition is reached.

12 Locate the **Output at Stop** section. From the **Add solution** list, choose **Step before stop**.

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

14 In the **Home** toolbar, click **Add Predefined Plot**.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.

2 In the tree, select **Study 1/Solution 1 (sol1)>Solid Mechanics>Equivalent Plastic Strain (solid)**.

3 Click **Add Plot** in the window toolbar.

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

RESULTS

Stress (solid)

Use mirror 3D and sector 3D datasets to display the solution on the full geometry.

Mirror 3D 1

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

2 Right-click **Results>Datasets** and choose **More 3D Datasets>Mirror 3D**.

Mirror 3D 2

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, locate the **Data** section.

3 From the **Dataset** list, choose **Mirror 3D 1**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.

Sector 3D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Sector 3D**.

2 In the **Settings** window for **Sector 3D**, locate the **Data** section.

3 From the **Dataset** list, choose **Mirror 3D 2**.

4 Locate the **Axis Data** section. In row **Point 2**, set **x** to 1 and **z** to 0.

5 Locate the **Symmetry** section. In the **Number of sectors** text field, type 6.

Stress (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

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

3 From the **Dataset** list, choose **Sector 3D 1**.

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

Volume 1

1 In the **Model Builder** window, expand the **Stress (solid)** node, then click **Volume 1**.

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

3 From the **Unit** list, choose **MPa**.

Use a **Surface Plot** with a **Material Appearance** subnode to visualize the stent in its original state.

Surface 1

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

2 In the **Settings** window for **Surface**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

Material Appearance 1

1 Right-click **Surface 1** and choose **Material Appearance**.

2 In the **Stress (solid)** toolbar, click **Plot**.

Stress (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

2 In the **Settings** window for **3D Plot Group**, click **Plot Previous** twice to plot the maximum stress.

3 In the **Stress (solid)** toolbar, click **Plot**.

Equivalent Plastic Strain (solid)

1 In the **Model Builder** window, click **Equivalent Plastic Strain (solid)**.

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

3 From the **Dataset** list, choose **Sector 3D 1**.

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

Surface 2

1 In the **Model Builder** window, expand the **Equivalent Plastic Strain (solid)** node.

2 Right-click **Results>Equivalent Plastic Strain (solid)>Surface 1** and choose **Duplicate**.

Deformation 1

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

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

3 Select the **Scale factor** check box. In the associated text field, type 1.

Surface 2

1 In the **Model Builder** window, under **Results>Equivalent Plastic Strain (solid)** click **Surface 2**.

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

3 From the **Title type** list, choose **None**.

Material Appearance 1

1 Right-click **Surface 2** and choose **Material Appearance**.

2 In the **Equivalent Plastic Strain (solid)** toolbar, click **Plot**.

Dogboning and Foreshortening

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

2 In the **Settings** window for **1D Plot Group**, type `Dogboning and Foreshortening` in the **Label** text field.

3 Locate the **Data** section. From the **Time selection** list, choose **From list**.

4 In the parameter values list, select all solution steps between 0 and 1.

Global 1

1 Right-click **Dogboning and Foreshortening** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>dogboning - Dogboning**.

3 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>foreshortening - Foreshortening**.

4 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>p - Pressure - Pa**.

5 In the **Dogboning and Foreshortening** toolbar, click **Plot**.

Evaluate the longitudinal recoil, the distal radial recoil, and the central radial recoil using the **Evaluation Group**.

Click on the check box in the Results node to enable automatic reevaluation of evaluation groups when the model is resolved.

6 In the **Model Builder** window, click **Results**.

7 In the **Settings** window for **Results**, locate the **Update of Results** section.

8 Select the **Reevaluate all evaluation groups after solving** check box.

Recoil Evaluation

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, type `Recoil Evaluation` in the **Label** text field.

3 Locate the **Data** section. From the **Time selection** list, choose **From list**.

4 In the **Parameter values (t (s))** list, select **1**.

Global Evaluation 1

1 Right-click **Recoil Evaluation** and choose **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
`(length-with(103,length))/length`	1	`Longitudinal recoil`
`(distal(r)-with(103,distal(r)))/distal(r)`	1	`Distal radial recoil`
`(central(r)-with(103,central(r)))/central(r)`	1	`Central radial recoil`

4 In the **Recoil Evaluation** toolbar, click **Evaluate**.

The steps below illustrate how to display the geometry as in Figure 1.

Full Geometry and Mesh

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

2 In the **Settings** window for **3D Plot Group**, type `Full Geometry and Mesh` in the **Label** text field.

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

Surface 1

1 Right-click **Full Geometry and Mesh** and choose **Surface**.

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

3 From the **Dataset** list, choose **Sector 3D 1**.

Material Appearance 1

Right-click **Surface 1** and choose **Material Appearance**.

Mesh 1

1 In the **Model Builder** window, right-click **Full Geometry and Mesh** and choose **Mesh**.

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

3 From the **Element color** list, choose **None**.

4 In the **Full Geometry and Mesh** toolbar, click **Plot**.