Model created in COMSOL Multiphysics 6.4

Modeling Stress Dependent Elasticity

Introduction

In this example, you learn how to implement a material, in which the elastic modulus varies under compressive and tensile stress.

Model Definition

The geometry is a 10 cm by 1 cm cantilever, fixed at the left end. A pure bending moment (–5/3 Nm) is applied at the other end, which results in a linear stress distribution with peak values of 0.1 MPa.

The material model is a linear elastic, where Young's modulus is a function of the sign of the mean stress (that is, the pressure). Under compression the material has an elastic modulus of 700 MPa, while under tension the elastic modulus drops to 300 MPa. The transition between the two values is assumed to be continuous between compressive and tensile stress, with a transition zone of 10^3 Pa.

Note: In this example, Poisson's ratio is assumed to be constant. This actually violates thermodynamic laws, and an accurate implementation would require a variable Poisson's ratio too.

Results and Discussion

Figure 1 shows the von Mises stress in the cantilever at maximum loading. Note that the stress distribution is not symmetric as it would be with a constant elastic modulus. The von Mises stress is higher in the compression region, where the material has a higher stiffness.

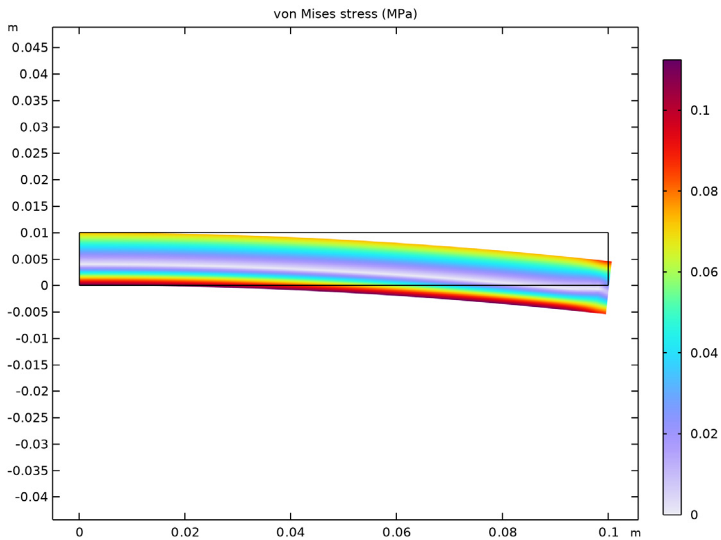

Figure 1: Von Mises stress distribution.

Figure 2 shows the value of Young's modulus. Note that the compressive region (in red) is thinner than the tensile one (in blue).

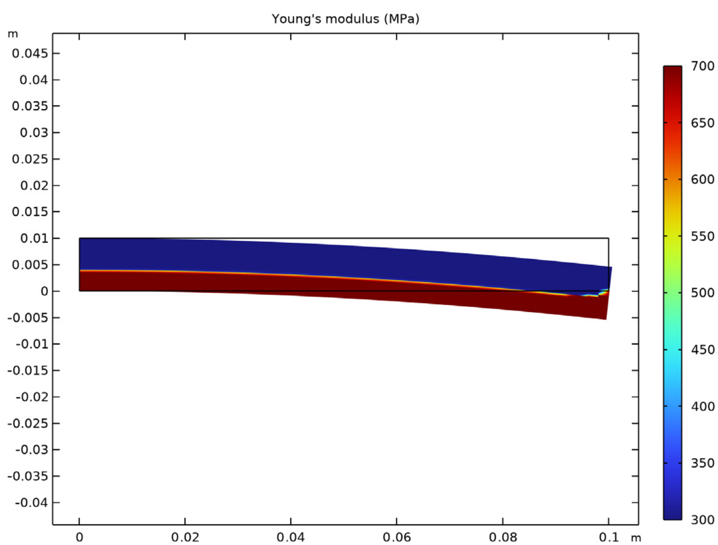

Figure 2: Distribution of mean stress dependent Young's modulus.

Figure 3 shows the stress in the x direction across the section at $x = 5$ cm. The slope is steeper in the compressive stress region than in the tensile one. The neutral line between tensile and compressive stress regions has shifted from the middle of the cantilever.

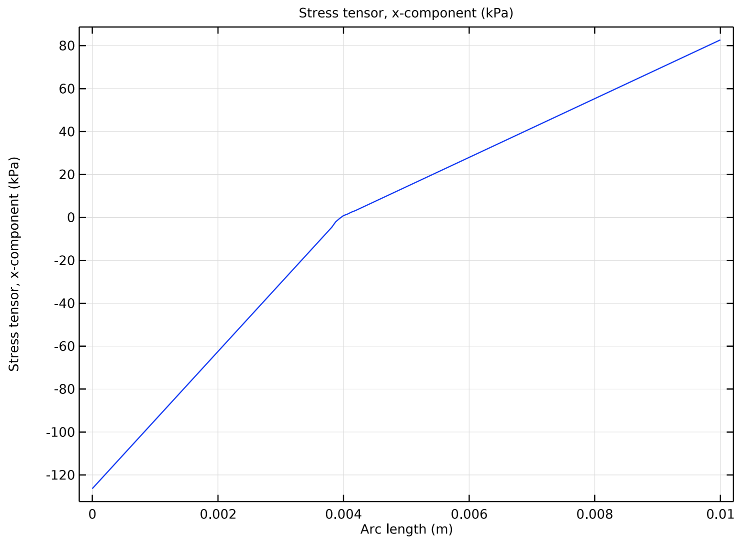

Figure 3: Stress in x direction across the cantilever height (x = 5 cm).

Notes About the COMSOL Implementation

A stress dependent elastic modulus introduces a circular dependency as the stress tensor itself is a function of Young's modulus and the strain tensor. A stress dependent Young's modulus would lead to Hooke's law being described as:

$$\sigma = E(\sigma) \cdot \varepsilon$$

To solve the problem with the circular variable dependency, you need to introduce a new variable onto which the stress value is mapped. Use a weak contribution and an auxiliary dependent variable to map the computed pressure value to the new variable p. Finally, define the Young's modulus as a function of this new variable p.

In this example, an easier solution would have been to reformulate the problem by instead making the Young's modulus a function of the volumetric strain, so that

$$\sigma = E(\varepsilon) \cdot \varepsilon$$

Such a reformulation, which is not always possible to find, avoids the circular dependency and a standard nonlinear problem is obtained.

To access the weak contribution feature, you need to activate the **Advanced Physics Options**.

Application Library path: `Structural_Mechanics_Module/Material_Models/stress_dependent_elasticity`

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**.
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

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 `10[cm]`.
4 In the **Height** text field, type `1[cm]`.
5 Click **Build Selected**.

SOLID MECHANICS (SOLID)

Roller 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Roller**.
2 Select Boundary 1 only.

Fixed Constraint 1

1 In the **Physics** toolbar, click **Points** and choose **Fixed Constraint**.

2 Select Point 1 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 **Resultant**.

5 Specify the **M** vector as

`-(5/3)[N*m]`	z

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

7 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics** > **Equation Contributions**.

8 Click **OK**.

Weak Contribution 1

1 In the **Physics** toolbar, click **Domains** and choose **Weak Contribution**.

2 Select Domain 1 only.

3 In the **Settings** window for **Weak Contribution**, locate the **Weak Contribution** section.

4 In the **Weak expression** text field, type `test(p)*(p-solid.pm)`.

Auxiliary Dependent Variable 1

1 In the **Physics** toolbar, click **Attributes** and choose **Auxiliary Dependent Variable**.

2 In the **Settings** window for **Auxiliary Dependent Variable**, locate the **Auxiliary Dependent Variable** section.

3 In the **Field variable name** text field, type `p`.

There is no need to enforce continuity for the mapped pressure, so using a Gauss-point shape function is a good choice.

4 Locate the **Discretization** section. From the **Shape function type** list, choose **Gauss-point data**.

5 Find the **Base geometry** subsection. From the **Element order** list, choose **4**.

MATERIALS

You will now define the material properties. As the study is stationary, you can assume the density to be zero.

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
Poisson's ratio	nu	0.3	1	Young's modulus and Poisson's ratio
Density	rho	0	kg/m³	Basic

You will now create a step function. Later you will use this function to define the Young's modulus with the input argument being the pressure.

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

Piecewise 1 (pw1)

1 In the **Model Builder** window, expand the **Material 1 (mat1)** node.

2 Right-click **Component 1 (comp1)** > **Materials** > **Material 1 (mat1)** > **Young's modulus and Poisson's ratio (Enu)** and choose **Functions** > **Piecewise**.

3 In the **Settings** window for **Piecewise**, type `E0` in the **Function name** text field.

4 Locate the **Definition** section. From the **Smoothing** list, choose **Continuous second derivative**.

5 From the **Transition zone** list, choose **Absolute size**.

6 In the **Size of transition zone** text field, type `1e3[Pa]`.

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

Start	End	Function
-3000[Pa]	0	300[MPa]
0	3000[Pa]	700[MPa]

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** > **Material 1 (mat1)** click **Young's modulus and Poisson's ratio (Enu)**.

2 In the **Settings** window for **Young's Modulus and Poisson's Ratio**, locate the **Output Properties** section.

3 In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Young's modulus	E	E0(p)	Pa	1x1

MESH 1

Mapped 1

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

Distribution 1

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

2 Select Boundary 1 only.

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

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

Distribution 2

1 In the **Model Builder** window, 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 50.

5 Click **Build All**.

STUDY 1

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)** > **Stationary Solver 1** node, then click **Fully Coupled 1**.

4 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

5 From the **Nonlinear method** list, choose **Constant (Newton)**.

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

RESULTS

Surface 1

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

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

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

The default plot shows the von Mises stress distribution, as shown in Figure 1.

Next, add a plot from **Result Templates** to visualize the applied moment resulting in a linearly varying traction.

RESULT TEMPLATES

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

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

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Solid Mechanics** > **Applied Loads (solid)** > **Boundary Loads (solid)**.

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

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

RESULTS

Boundary Loads (solid)

Now create a new plot group for the Young's modulus, to reproduce Figure 2.

Young's Modulus

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Young's Modulus` in the **Label** text field.

Surface 1

1 In the **Young's Modulus** 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** > **Material properties** > **solid.E - Young's modulus - Pa**.

3 Locate the **Expression** section. From the **Unit** list, choose **MPa**.

Change also the quality settings to get a better visualization.

4 Click to expand the **Quality** section. From the **Evaluation settings** list, choose **Manual**.

5 From the **Resolution** list, choose **No refinement**.

6 From the **Smoothing** list, choose **None**.

Deformation 1

1 In the **Young's Modulus** toolbar, click **Deformation**.

2 Click **Plot**.

Finally, display the stress variation through the thickness of the beam, Figure 3.

Cut Line 2D 1

1 In the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Line Data** section.

3 In row **Point 1**, set **X** to `5[cm]`.

4 In row **Point 2**, set **X** to `5[cm]` and **Y** to `1[cm]`.

Stress Through Thickness

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Line 2D 1**.

Line Graph 1

1 Right-click **Stress Through Thickness** and choose **Line Graph**.

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

3 In the **Expression** text field, type `solid.sx`.

4 From the **Unit** list, choose **kPa**.

5 In the **Stress Through Thickness** toolbar, click **Plot**.