

# Large Deformation Beam

## *Model Definition*

---

In this example you study the deflection of a cantilever beam undergoing very large deflections. The model is called “Straight Cantilever GNL Benchmark” and is described in detail in section 5.2 of *NAFEMS Background to Finite Element Analysis of Geometric Non-linearity Benchmarks* (Ref. 1).

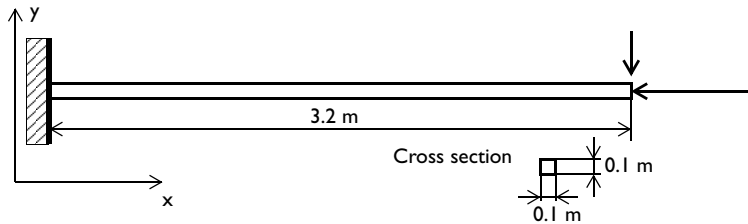


Figure 1: Cantilever beam geometry.

### **GEOMETRY**

- The length of the beam is 3.2 m.
- The cross section is a square with side lengths 0.1 m.

### **MATERIAL**

The beam is linear elastic with  $E = 2.1 \cdot 10^{11} \text{ N/m}^2$  and  $\nu = 0$ .

### **CONSTRAINTS AND LOADS**

- The left end is fixed. This boundary condition is compatible with beam theory assumptions only in the case  $\nu = 0$ .
- The right end is subjected to distributed loads with the resultants  $F_x = -3.844 \cdot 10^6 \text{ N}$  and  $F_y = -3.844 \cdot 10^3 \text{ N}$ .

## *Results and Discussion*

---

Due to the large compressive axial load and the slender geometry, this is a buckling problem. If you are to study the buckling and post-buckling behavior of a symmetric problem, it is necessary to perturb the symmetry somewhat. Here the small transversal load serves this purpose. An alternative approach would be to introduce an initial imperfection in the geometry.

Figure 2 below shows the final state (using 1:1 displacement scaling).

NCL(101)=1 Surface: von Mises stress (MPa) Surface Deformation: Displacement field (Material)

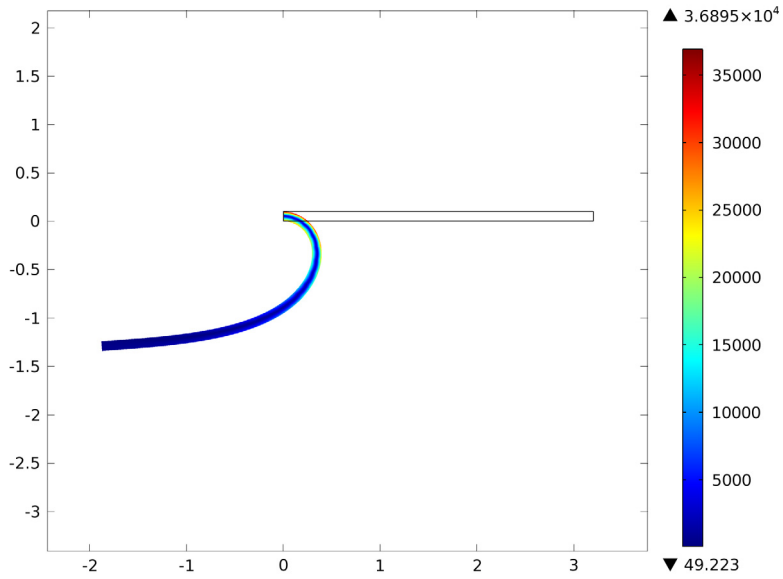


Figure 2: von Mises stress (surface) and deformation (deformation plot).

The horizontal and vertical displacements of the tip versus the compressive load normalized by its maximum value appear in the next graph.

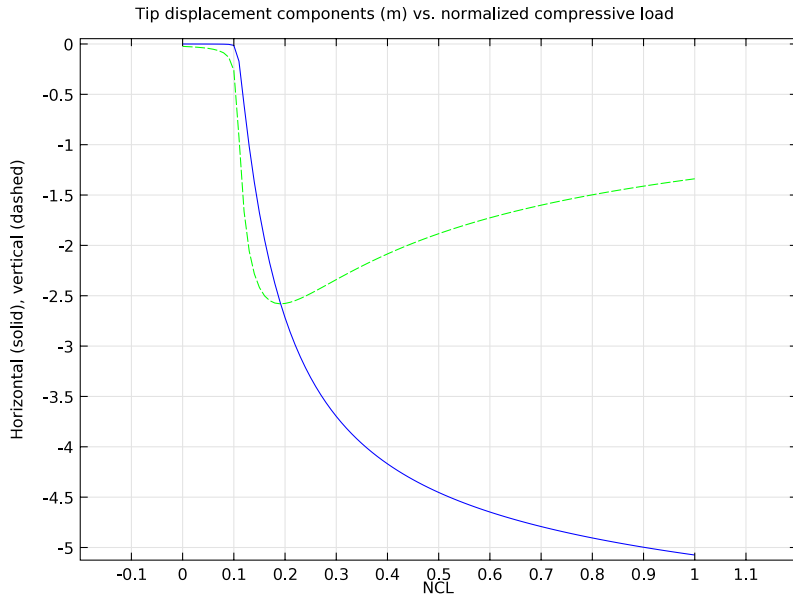


Figure 3: Horizontal (solid) and vertical (dashed) tip displacements versus normalized compressive load.

Table 1 contains a summary of some significant results. Because the reference values are given as graphs, an estimate of the error caused by reading this graph is added:

TABLE 1: COMPARISON BETWEEN MODEL RESULTS AND REFERENCE VALUES.

| QUANTITY                                 | COMSOL MULTIPHYSICS | REFERENCE        |
|--|---------------------|------------------|
| Maximum vertical displacement at the tip | -2.58               | $-2.58 \pm 0.02$ |
| Final vertical displacement at the tip   | -1.34               | $-1.36 \pm 0.02$ |
| Final horizontal displacement at the tip | -5.07               | $-5.04 \pm 0.04$ |

The results are in excellent agreement, especially considering the coarse mesh used.

The plot of the axial deflection reveals that an instability occurs at a parameter value close to 0.1, corresponding to the compressive load  $3.84 \cdot 10^5$  N. It is often seen in practice that the critical load of an imperfect structure is significantly lower than that of the ideal structure.

This problem (without the small transverse load) is usually referred to as the Euler-1 case. The theoretical critical load is

$$P_c = \frac{\pi^2 EI}{4L^2} = \frac{\pi^2 \cdot 2.1 \cdot 10^{11} \cdot (0.1^4/12)}{4 \cdot 3.2^2} = 4.22 \cdot 10^5 \text{ N} \quad (1)$$

### Reference

---

1. A.A. Becker, *Background to Finite Element Analysis of Geometric Non-linearity Benchmarks*, NAFEMS, Ref: -R0065, Glasgow, 1999.

---

**Model Library path:** Structural\_Mechanics\_Module/Verification\_Models/large\_deformation\_beam

---

### Modeling Instructions

---

#### MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click the **2D** button.
- 3 Click **Next**.
- 4 In the **Add physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
- 5 Click **Next**.
- 6 In the **Studies** tree, select **Preset Studies>Stationary**.
- 7 Click **Finish**.

#### GEOMETRY I

##### Rectangle 1

- 1 In the **Model Builder** window, right-click **Model I>Geometry I** and choose **Rectangle**.
- 2 Go to the **Settings** window for Rectangle.
- 3 Locate the **Size** section. In the **Width** edit field, type 3.2.
- 4 In the **Height** edit field, type 0.1.

##### Form Union

In the **Model Builder** window, right-click **Form Union** and choose **Build Selected**.

## GLOBAL DEFINITIONS

Define parameters for the compressive and transverse load components as well as a parameter that you will use to gradually turn up the compressive load.

### Parameters

- 1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
- 2 Go to the **Settings** window for Parameters.
- 3 Locate the **Parameters** section. In the **Parameters** table, enter the following settings:

| NAME | EXPRESSION | DESCRIPTION                 |
|------|------------|-----------------------------|
| F_Lx | -3.844[MN] | Maximum compressive load    |
| F_Ly | 1e-3*F_Lx  | Transverse load             |
| NCL  | 0          | Normalized compressive load |

By restricting the range for the parameter NCL to [0, 1], it serves as a compressive load normalized by the maximum compressive load.

## MATERIALS

### Material 1

- 1 In the **Model Builder** window, right-click **Model 1 > Materials** and choose **Material**.
- 2 Select Domain 1 only.
- 3 Go to the **Settings** window for Material.
- 4 Locate the **Material Contents** section. In the **Material contents** table, enter the following settings:

| PROPERTY        | NAME | VALUE  |
|-----------------|------|--------|
| Young's modulus | E    | 2.1e11 |
| Poisson's ratio | nu   | 0      |
| Density         | rho  | 7850   |

## SOLID MECHANICS

- 1 In the **Model Builder** window, click **Model 1 > Solid Mechanics**.
- 2 Go to the **Settings** window for Solid Mechanics.
- 3 Locate the **2D Approximation** section. From the **2D approximation** list, select **Plane stress**.
- 4 Locate the **Thickness** section. In the  $d$  edit field, type 0.1.

*Linear Elastic Material Model 1*

- 1 In the **Model Builder** window, expand the **Solid Mechanics** node, then click **Linear Elastic Material Model 1**.
- 2 Select Domain 1 only.
- 3 Go to the **Settings** window for Linear Elastic Material Model.
- 4 Locate the **Geometric Nonlinearity** section. Select the **Include geometric nonlinearity** check box.

*Fixed Constraint 1*

- 1 In the **Model Builder** window, right-click **Solid Mechanics** and choose **Fixed Constraint**.
- 2 Select Boundary 1 only.

*Boundary Load 1*

- 1 In the **Model Builder** window, right-click **Solid Mechanics** and choose **Boundary Load**.
- 2 Select Boundary 4 only.
- 3 Go to the **Settings** window for Boundary Load.
- 4 Locate the **Force** section. From the **Load type** list, select **Total force**.
- 5 Specify the  $\mathbf{F}_{\text{tot}}$  vector as

|          |   |
|----------|---|
| NCL*F_Lx | x |
| F_Ly     | y |

**MESH 1**

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

**STUDY 1***Step 1: Stationary*

- 1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.
- 2 Go to the **Settings** window for Stationary.
- 3 Click to expand the **Extension** section.
- 4 Select the **Continuation** check box.
- 5 Under **Continuation parameter**, click **Add**.
- 6 Go to the **Add** dialog box.
- 7 In the **Continuation parameter** list, select **NCL (Normalized compressive load)**.
- 8 Click the **OK** button.

- 9 Go to the **Settings** window for Stationary.
- 10 Locate the **Extension** section. In the **Parameter values** edit field, type range(0,0.01,1).
- 11 In the **Model Builder** window, right-click **Study 1** and choose **Show Default Solver**.
- 12 Expand the **Study 1>Solver Configurations** node.

#### *Solver 1*

- 1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solver 1** node, then click **Stationary Solver 1**.
- 2 Go to the **Settings** window for Stationary Solver.
- 3 Locate the **General** section. In the **Relative tolerance** edit field, type 1e-6.
- 4 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

## RESULTS

#### *Stress (solid)*

Modify the default surface plot to show the unscaled deformation.

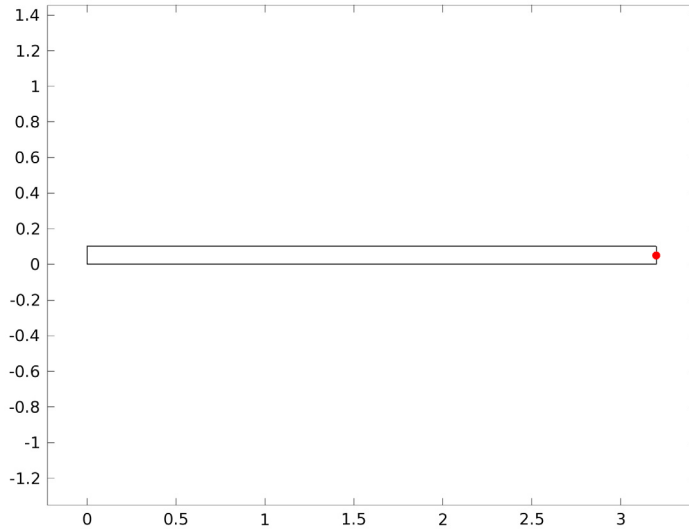
- 1 In the **Model Builder** window, expand the **Stress (solid)** node.
- 2 In the **Model Builder** window, expand the **Surface 1** node, then click **Deformation**.
- 3 Go to the **Settings** window for Deformation.
- 4 Locate the **Scale** section. Select the **Scale factor** check box.
- 5 In the associated edit field, type 1.
- 6 In the **Model Builder** window, click **Surface 1**.
- 7 Go to the **Settings** window for Surface.
- 8 Locate the **Expression** section. From the **Unit** list, select **MPa**.
- 9 Click the **Plot** button.
- 10 Click the **Zoom Extents** button on the Graphics toolbar.  
Compare the resulting plot with the one shown in [Figure 2](#).

Next, add a data set to use for plotting the displacement components at the tip and for displaying their values as functions of the normalized compressive load.

#### *Data Sets*

- 1 In the **Model Builder** window, right-click **Results>Data Sets** and choose **Cut Point 2D**.
- 2 Go to the **Settings** window for Cut Point 2D.
- 3 Locate the **Point Data** section. In the **X** edit field, type 3.2.

- 4 In the **Y** edit field, type 0.05.
- 5 Click the **Plot** button.
- 6 Click the **Zoom Extents** button on the Graphics toolbar.



#### Derived Values

- 1 In the **Model Builder** window, right-click **Results>Derived Values** and choose **Point Evaluation**.
- 2 Go to the **Settings** window for Point Evaluation.
- 3 Locate the **Data** section. From the **Data set** list, select **Cut Point 2D I**.
- 4 In the upper-right corner of the **Expression** section, click **Replace Expression**.
- 5 From the menu, choose **Solid Mechanics>Displacement field (Material)>Displacement field, X component (u)**.
- 6 Click the **Evaluate** button.  
As stated in [Table 1](#), the final horizontal displacement value is roughly -5.07 m.
- 7 In the upper-right corner of the **Expression** section, click **Replace Expression**.
- 8 From the menu, choose **Solid Mechanics>Displacement field (Material)>Displacement field, Y component (v)**.

9 Click the **Evaluate** button.

By scrolling in the **Results** window table you can verify that the maximum vertical displacement is 2.58 m (downward) and occurs at a normalized compressive load of 0.19.

### *1D Plot Group 2*

- 1 In the **Model Builder** window, right-click **Results** and choose **ID Plot Group**.
- 2 Go to the **Settings** window for 1D Plot Group.
- 3 Locate the **Data** section. From the **Data set** list, select **Cut Point 2D 1**.
- 4 Right-click **Results>ID Plot Group 2** and choose **Point Graph**.
- 5 Go to the **Settings** window for Point Graph.
- 6 In the upper-right corner of the **y-Axis Data** section, click **Replace Expression**.
- 7 From the menu, choose **Solid Mechanics>Displacement field (Material)>Displacement field, X component (u)**.
- 8 In the **Model Builder** window, right-click **ID Plot Group 2** and choose **Point Graph**.
- 9 Go to the **Settings** window for Point Graph.
- 10 In the upper-right corner of the **y-Axis Data** section, click **Replace Expression**.
- 11 From the menu, choose **Solid Mechanics>Displacement field (Material)>Displacement field, Y component (v)**.
- 12 Click to expand the **Coloring and Style** section.
- 13 Find the **Line style** subsection. From the **Line** list, select **Dashed**.
- 14 In the **Model Builder** window, click **ID Plot Group 2**.
- 15 Go to the **Settings** window for 1D Plot Group.
- 16 Locate the **Plot Settings** section. Select the **Title** check box.
- 17 In the associated edit field, type Tip displacement components (m) vs. normalized compressive load.
- 18 Select the **y-axis label** check box.
- 19 In the associated edit field, type Horizontal (solid), vertical (dashed).
- 20 Click the **Plot** button.

