Tapered Cantilever with Two Load Cases

Introduction

This example, taken from a NAFEMS benchmark collection (Ref. 1), demonstrates how to apply and evaluate different boundary conditions acting on a cantilever beam.

Model Definition

The cantilever beam has a thickness of 0.1 m, width of 4 m and is 4 m on its long edge and 2 m on its short edge. Two cases are considered. In the first one a gravity load, mg, acts in the negative y direction with an acceleration of 9.81 m/s^2. The left end boundary is fully fixed (no displacements). In the second case gravity load is not present and instead a uniformly distributed horizontal load, F, of 10 MN/m acts along the right end. At the left end there is no displacement in the x direction. Also at a midpoint location the left end is fixed in the y direction; see Figure 1.
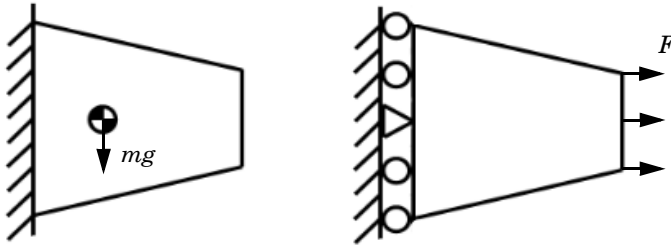

Figure 1: Schematic description of two load cases.

MATERIAL MODEL

The model uses the following material properties:

- The material is isotropic.
- The Young's modulus (elasticity modulus) is 210 GPa.
- The Poisson ratio is 0.3.
- The density is 7000 kg/m^3.

Results and Discussion

For the gravity case shear stress is evaluated. The benchmark target value of −0.200 MPa at the point (0, 2) is in good agreement with the model results. Using the default Normal mesh size, the COMSOL Multiphysics solution gives a value of −0.199 MPa; see Figure 2.
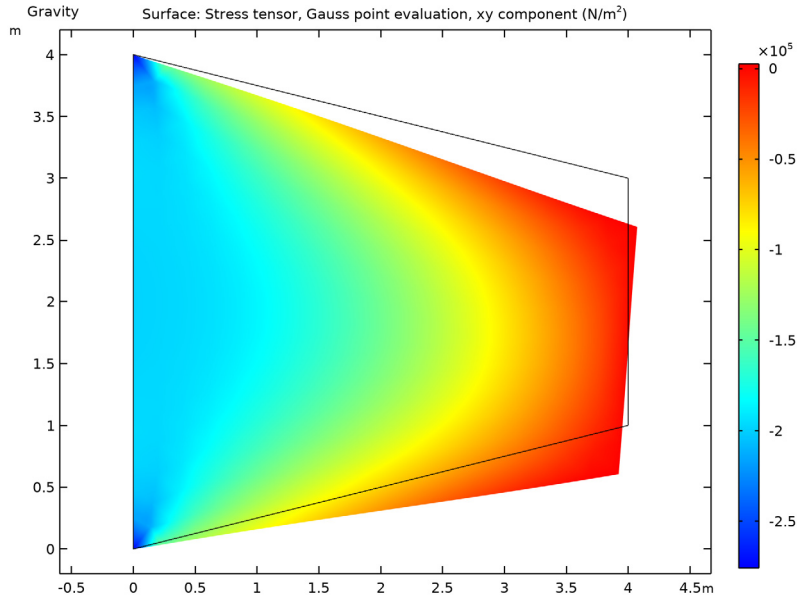

Figure 2: In plane shear stress due to gravity load.

For the load case, the horizontal (x direction) normal stress is evaluated. The benchmark target value of 61.3 MPa at the point (0, 2) is in good agreement with the results. Using the default mesh size, the COMSOL Multiphysics solution gives a value of 61.4 MPa; see Figure 3.
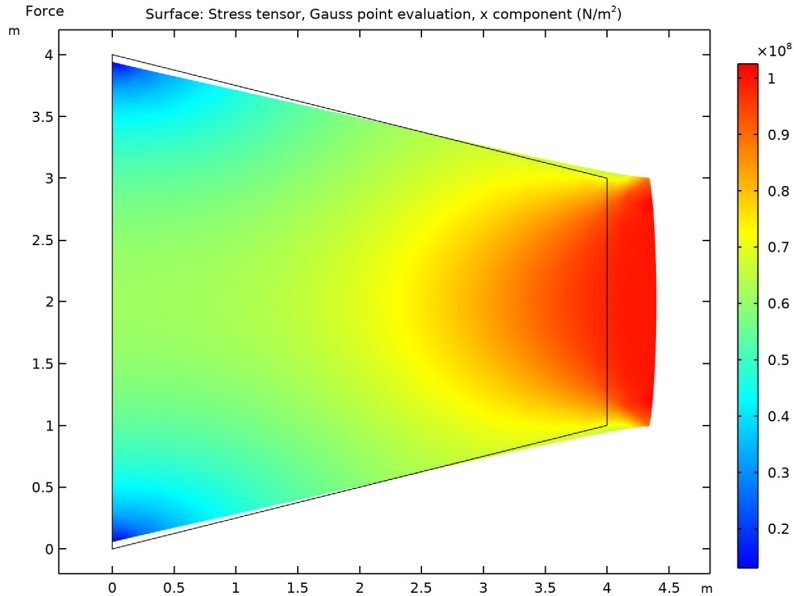
Figure 3: Counteracting stress due to applied edge load.

Notes About the COMSOL Implementation

Use predefined variables for the acceleration of gravity and density to enter the gravity load as a force/volume. COMSOL then computes the load using the thickness of the geometry.

Use the Solid Mechanics interface to perform a stress analysis. The finite element model uses the default second-order triangular Lagrange elements. To show convergence toward the benchmark value, create a finer mesh.

Use the Load Group and Constraint Group features to collect conditions that are enabled in different studies. Define a group in Global Definitions and assign its content directly from the load or constraint itself. You define load cases in the Study Extensions of a Study Type and enable active load and constraint groups; see Figure 4, where lg denotes a load

group and cg denotes a constraint group. Weight is used as a multiplication factor for the corresponding load group.

Figure 4: Definition of load cases.

Reference

1. D. Hitchings, A. Kamoulakos, and G.A.O. Davies, *Linear Statics Benchmarks Vol. 1*, NAFEMS, Glasgow, 1987.

Application Library path: COMSOL_Multiphysics/Structural_Mechanics/
tapered_cantilever

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

Polygon 1 (pol1)
1 In the **Geometry** toolbar, click **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 From the **Data source** list, choose **Vectors**.

4 In the **x** text field, type 0 4 4 0 0.

5 In the **y** text field, type 0 1 3 4 0.

6 Click ▦ **Build Selected**.

Point 1 (pt1)

1 In the **Geometry** toolbar, click ⋅ **Point**.

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

3 In the **y** text field, type 2.

4 Click ▦ **Build Selected**.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click ▦ **Build Selected**.

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	210[GPa]	Pa	Basic
Poisson's ratio	nu	0.3	1	Basic
Density	rho	7000[kg/m^3]	kg/m³	Basic

GLOBAL DEFINITIONS

Load Group Gravity

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Load and Constraint Groups>Load Group**.

2 In the **Settings** window for **Load Group**, type Load Group Gravity in the **Label** text field.

3 In the **Parameter name** text field, type lgGravity.

Load Group Force

1 In the **Model Builder** window, right-click **Load and Constraint Groups** and choose **Load Group**.

2 In the **Settings** window for **Load Group**, type Load Group Force in the **Label** text field.

3 In the **Parameter name** text field, type lgForce.

Constraint Group Gravity

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Load and Constraint Groups>Constraint Group**.

2 In the **Settings** window for **Constraint Group**, type Constraint Group Gravity in the **Label** text field.

3 In the **Parameter name** text field, type cgGravity.

Constraint Group Force

1 Right-click **Global Definitions** and choose **Load and Constraint Groups>Constraint Group**.

2 In the **Settings** window for **Constraint Group**, type Constraint Group Force in the **Label** text field.

3 In the **Parameter name** text field, type cgForce.

SOLID MECHANICS (SOLID)

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

2 In the **Settings** window for **Solid Mechanics**, locate the **2D Approximation** section.

3 From the list, choose **Plane stress**.

4 Locate the **Thickness** section. In the d text field, type 0.1.

First, define all constraints for both load cases and then assign them to different load and constraint groups.

Fixed Constraint 1

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

2 Select Boundaries 1 and 3 only.

3 In the **Physics** toolbar, click ⦿ **Constraint Group** and choose **Constraint Group Gravity**.

Body Load 1

1 In the **Physics** toolbar, click ⬛ **Domains** and choose **Body Load**.

2 Select Domain 1 only.

3 In the **Settings** window for **Body Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_V$ vector as

0	x
-g_const*solid.rho	y

5 In the **Physics** toolbar, click ⊞ **Load Group** and choose **Load Group Gravity**.

Roller 1

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

2 Select Boundaries 1 and 3 only.

3 In the **Physics** toolbar, click ⦿ **Constraint Group** and choose **Constraint Group Force**.

Boundary Load 1

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

2 Select Boundary 5 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 From the **Load type** list, choose **Force per unit length**.

5 Specify the $\mathbf{F}_L$ vector as

10[MN/m]	x
0	y

6 In the **Physics** toolbar, click ⊞ **Load Group** and choose **Load Group Force**.

Fixed Constraint 2

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

2 Select Point 2 only.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.

STUDY 1

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 **Define load cases** check box.

4 Click ✛ **Add**.

5 In the table, enter the following settings:

Load case	lgGravity	Weight	lgForce	Weight	cgGravity	cgForce
Gravity	√	1.0		1.0	√	

6 Click ✛ **Add**.

7 In the table, enter the following settings:

Load case	lgGravity	Weight	lgForce	Weight	cgGravity	cgForce
Force		1.0	√	1.0		√

The study extension for the load cases should look like Figure 4.

8 In the **Home** toolbar, click ▬ **Compute**.

RESULTS

Normal stress

In the **Settings** window for **2D Plot Group**, type Normal stress in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Normal stress** node, then click **Surface 1**.

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.sGpx - Stress tensor, Gauss point evaluation, x component**.

3 In the **Normal stress** toolbar, click 🔲 **Plot**.

Shear stress

1 In the **Model Builder** window, right-click **Normal stress** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type Shear stress in the **Label** text field.

3 Locate the **Data** section. From the **Load case** list, choose **Gravity**.

Surface 1

1 In the **Model Builder** window, expand the **Shear stress** node, then click **Surface 1**.

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

3 In the **Expression** text field, type solid.sGpxy.

4 In the **Shear stress** toolbar, click 🔲 **Plot**.

Point Evaluation - normal stress

1 In the **Results** toolbar, click $\frac{8.85}{e\text{-}12}$ **Point Evaluation**.

2 In the **Settings** window for **Point Evaluation**, type `Point Evaluation - normal stress` in the **Label** text field.

3 Select Point 2 only.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Stress (Gauss points)> Stress tensor, Gauss point evaluation (spatial frame) - N/m²>solid.sGpx - Stress tensor, Gauss point evaluation, x component**.

5 Click ▬ **Evaluate**.

Point Evaluation - shear stress

1 In the **Results** toolbar, click ⁸·⁸⁵ₑ₋₁₂ **Point Evaluation**.

2 In the **Settings** window for **Point Evaluation**, type `Point Evaluation - shear stress` in the **Label** text field.

3 Select Point 2 only.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Stress (Gauss points)> Stress tensor, Gauss point evaluation (spatial frame) - N/m²>solid.sGpxy - Stress tensor, Gauss point evaluation, xy component**.

5 Click ▬ **Evaluate**.