



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](#).

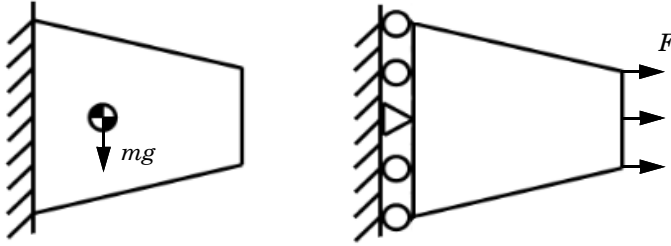


Figure 1: Schematic description of two load cases.

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](#).

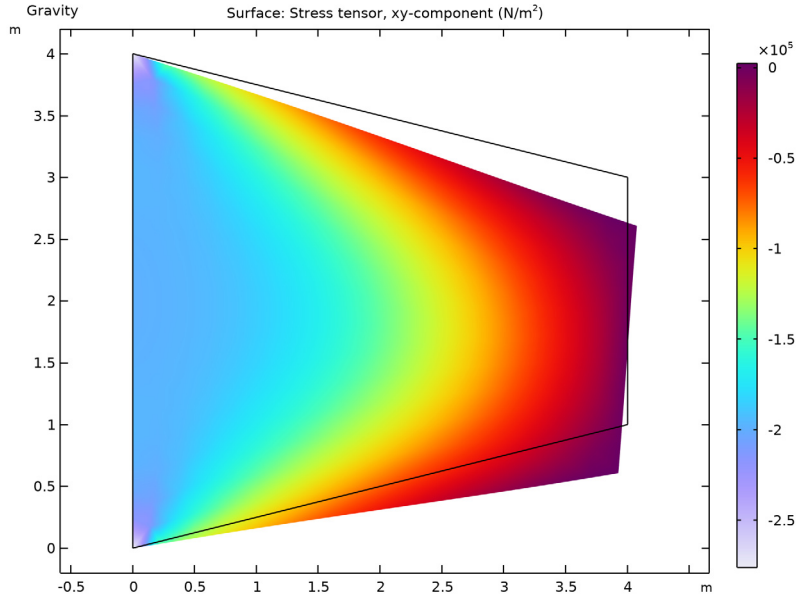


Figure 2: In plane shear stress due to gravity load.

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](#).

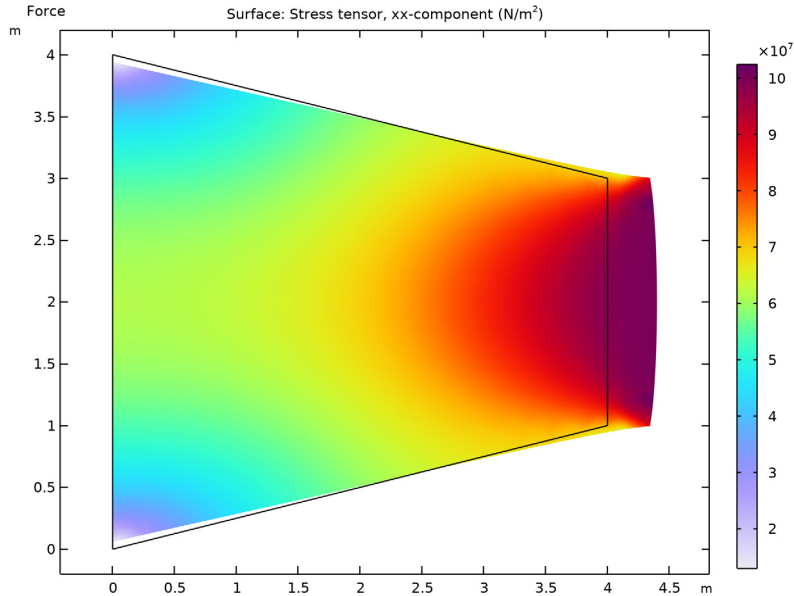


Figure 3: Counteracting stress due to applied edge load.

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.

<input checked="" type="checkbox"/> Define load cases							
» Load case	lgGravity	Weight	lgForce	Weight	cgGravity	cgForce	
Gravity	✓	1.0	✗	1.0	✓	✗	
Force	✗	1.0	✓	1.0	✗	✓	

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	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	1	Young's modulus and Poisson's ratio
Density	rho	7000 [kg/m^3]	kg/m ³	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 1gG.

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 1gF.

Constraint Group Gravity

- 1 Right-click **Load and Constraint Groups** and choose **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 **Load and Constraint Groups** and choose **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 1**.

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
$-g_const \cdot solid.rho$	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	IgG	Weight	IgF	Weight	cgGravity	cgForce
Gravity	√	1.0		1.0	√	

6 Click  **Add**.

7 In the table, enter the following settings:

Load case	IgG	Weight	IgF	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>Stress tensor (spatial frame) - N/m²>solid.sGp_{xx} - Stress tensor, xx-component**.
- 3 In the **Normal stress** toolbar, click  **Plot**.


Normal stress

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



Shear stress

- 1 In the **Model Builder** window, under **Results** click **Normal stress 1**.
- 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.sGp_{xy}**.
- 4 In the **Shear stress** toolbar, click  **Plot**.

Point Evaluation - normal stress

- 1 In the **Results** toolbar, click  **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>Stress tensor (spatial frame) - N/m²>solid.sGp_{xx} - Stress tensor, xx-component**.
- 5 Click  **Evaluate**.

Point Evaluation - shear stress

- 1 In the **Results** toolbar, click  **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>Stress tensor (spatial frame) - N/m²>solid.sGp_{xy} - Stress tensor, xy-component**.
- 5 Click  **Evaluate**.