Thermally Loaded Beam

Introduction

In the following tutorial, you build and solve a 3D beam model using the 3D Beam interface. This example shows how to model a thermally induced deformation of a beam. Temperature gradients are applied between the top and bottom surfaces as well as between the left and right surfaces of the beam. The deformation is compared to the value given by a theoretical solution given in Ref. 1.

Model Definition

GEOMETRY

The geometry consists of one beam. The beam cross-section area is A and the area moment of inertia I. The beam is L long, and the Young's modulus is E.

- Beam length L = 3 m.
- The beam has a square cross section with a side length of 0.04 m giving an area of $A = 1.6{\cdot}10^{-3}$ m^2 and an area moment of inertia of $I = 0.04^4/12$ m^4.

MATERIAL

- Young's modulus E = 210 GPa.
- Poisson's ratio $\nu = 0.3$.
- Coefficient of thermal expansion $\alpha = 11{\cdot}10^{-6}$/°C.

CONSTRAINTS

- At one end, the all displacements are constrained, and the rotation of the beam about its own axis is constrained.
- At the other end, transverse displacements are constrained.

THERMAL LOAD

Figure 1 shows the surface temperature at each corner of the cross section. The temperature varies linearly between each corner. The deformation caused by this temperature distribution is modeled by specifying the temperature differences across the beam in the local *y* and *z* directions.

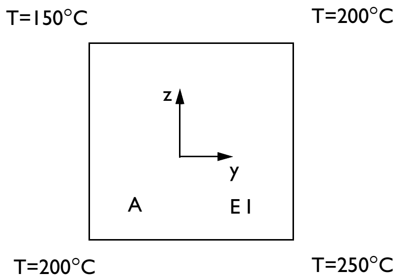

Figure 1: Geometric properties and thermal loads at corners.

Results and Discussion

Based on Ref. 1, you can compare the maximum deformation in the global *z* direction with analytical values for a simply supported 2D beam with a temperature difference between the top and the bottom surface. The maximum deformation, according to Ref. 1 is:

$$w = \frac{\alpha L^2}{8t}(T_2 - T_1)$$

where t is the depth of the beam, 0.04 m, T_2 is the temperature at the top and T_1 at the bottom.

The following table shows a comparison of the maximum global *z*-displacement, calculated with COMSOL Multiphysics, with the theoretical solution.

w	COMSOL Multiphysics (max)	Analytical
	15.5 mm	15.5 mm

Figure 2 shows the global *z*-displacement along the beam.

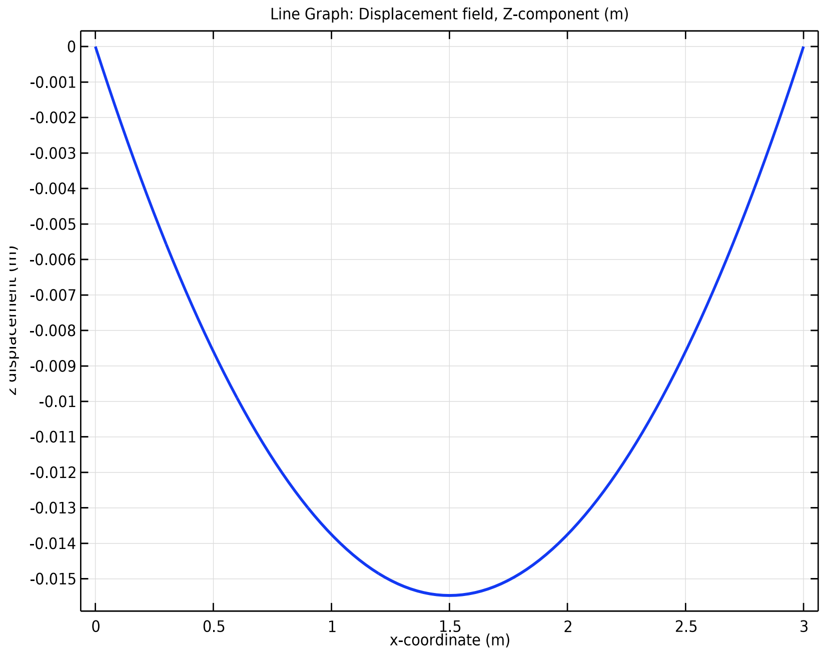

Figure 2: z-displacement along the beam.

The analytical values for the maximum total transverse displacement can be calculated by:

$$\delta = \sqrt{w^2 + v^2}$$

where v is the maximum deformation in the global y direction which is calculated in the same way as w.

A comparison of the maximum transverse displacement calculated with COMSOL Multiphysics and the analytical value is shown in the table below.

COMSOL Multiphysics	**Analytical**
21.9 mm	21.9 mm

Figure 3 shows the total displacement, the total transverse displacement and the axial displacement along the beam.

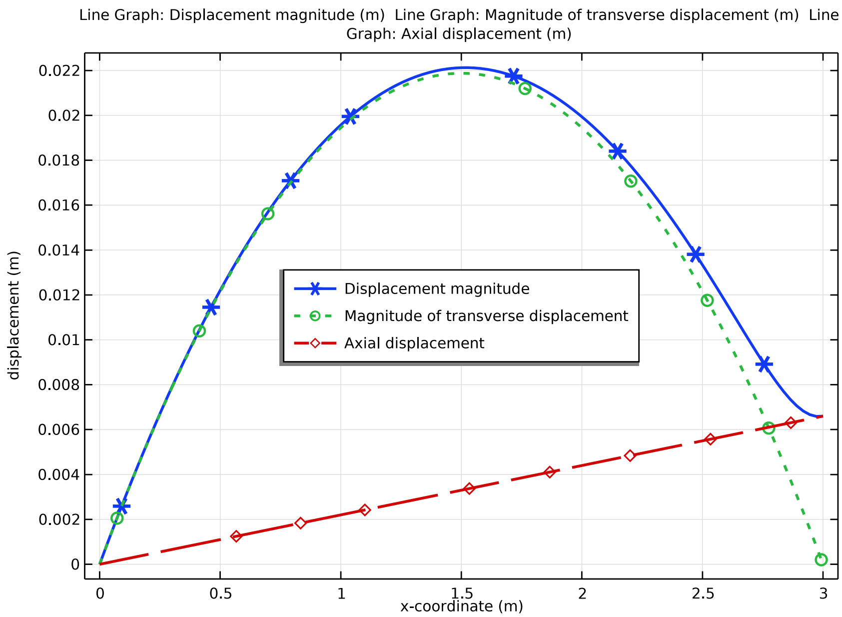

Figure 3: Camber along the beam.

Reference

1. W. Young, *Roark's Formulas for Stress & Strain*, McGraw Hill, 1989.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/thermally_loaded_beam`

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>Beam (beam)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Stationary**.
6 Click **Done**.

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
a	0.04[m]	0.04 m	Side length
deltaT	50[K]	50 K	Temperature difference
Tg	deltaT/a	1250 K/m	Temperature gradient
Lb	3[m]	3 m	Beam length

GEOMETRY 1

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Polygon**.
2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.
3 In the table, enter the following settings:

x (m)	y (m)	z (m)
0	0	0
Lb/2	0	0
Lb	0	0

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**, click to expand the **Material Properties** section.

3 In the **Material properties** tree, select **Basic Properties>Coefficient of Thermal Expansion**.

4 Click + **Add to Material**.

5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	11e-6	1/K	Basic
Young's modulus	E	210e9	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	1	Young's modulus and Poisson's ratio
Density	rho	7800	kg/m³	Basic

BEAM (BEAM)

Cross-Section Data 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Beam (beam)** click **Cross-Section Data 1**.

2 In the **Settings** window for **Cross-Section Data**, locate the **Cross-Section Definition** section.

3 From the **Section type** list, choose **Rectangle**.

4 In the h_y text field, type a.

5 In the h_z text field, type a.

Section Orientation 1

1 In the **Model Builder** window, click **Section Orientation 1**.

2 In the **Settings** window for **Section Orientation**, locate the **Section Orientation** section.

3 From the **Orientation method** list, choose **Orientation vector**.

4 Specify the *V* vector as

0	X
1	Y
0	Z

Prescribed Displacement/Rotation 1

1 In the **Physics** toolbar, click **Points** and choose **Prescribed Displacement/Rotation**.

2 Select Point 1 only.

3 In the **Settings** window for **Prescribed Displacement/Rotation**, locate the **Prescribed Displacement** section.

4 From the **Displacement in x direction** list, choose **Prescribed**.

5 From the **Displacement in y direction** list, choose **Prescribed**.

6 From the **Displacement in z direction** list, choose **Prescribed**.

7 Locate the **Prescribed Rotation** section. From the list, choose **Rotation**.

8 Select the **Free rotation around y direction** check box.

9 Select the **Free rotation around z direction** check box.

Prescribed Displacement/Rotation 2

1 In the **Physics** toolbar, click **Points** and choose **Prescribed Displacement/Rotation**.

2 Select Point 3 only.

3 In the **Settings** window for **Prescribed Displacement/Rotation**, locate the **Prescribed Displacement** section.

4 From the **Displacement in y direction** list, choose **Prescribed**.

5 From the **Displacement in z direction** list, choose **Prescribed**.

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

Thermal Expansion 1

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

2 In the **Settings** window for **Thermal Expansion**, locate the **Model Input** section.

3 Click **Go to Source** for **Volume reference temperature**.

GLOBAL DEFINITIONS

Default Model Inputs

1 In the **Model Builder** window, under **Global Definitions** click **Default Model Inputs**.

2 In the **Settings** window for **Default Model Inputs**, locate the **Browse Model Inputs** section.

3 Find the **Expression for remaining selection** subsection. In the **Volume reference temperature** text field, type `0`.

BEAM (BEAM)

Thermal Expansion 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Beam (beam)>Linear Elastic Material 1** click **Thermal Expansion 1**.

2 In the **Settings** window for **Thermal Expansion**, locate the **Model Input** section.

3 From the T list, choose **User defined**. In the associated text field, type `200`.

4 Locate the **Thermal Bending** section. In the T_{gy} text field, type `Tg`.

5 In the T_{gz} text field, type `-Tg`.

STUDY 1

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

RESULTS

Displacements

In the **Settings** window for **3D Plot Group**, type `Displacements` in the **Label** text field.

Line 1

1 In the **Model Builder** window, expand the **Displacements** node, then click **Line 1**.

2 In the **Settings** window for **Line**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Beam>Displacement>beam.disp - Displacement magnitude - m**.

3 In the **Displacements** toolbar, click **Plot**.

Transverse Displacement

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

2 In the **Settings** window for **1D Plot Group**, type `Transverse Displacement` in the **Label** text field.

3 Locate the **Plot Settings** section.

4 Select the **y-axis label** check box. In the associated text field, type `z displacement (m)`.

Transverse displacement (z direction)

1 Right-click **Transverse Displacement** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, type `Transverse displacement (z direction)` in the **Label** text field.

3 Click in the **Graphics** window and then press Ctrl+A to select both edges.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `w`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `x`.

7 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

8 In the **Transverse Displacement** toolbar, click **Plot**.

Displacement

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

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

3 Locate the **Plot Settings** section.

4 Select the **y-axis label** check box. In the associated text field, type `displacement (m)`.

5 Locate the **Legend** section. From the **Position** list, choose **Center**.

Displacement magnitude

1 Right-click **Displacement** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, type `Displacement magnitude` in the **Label** text field.

3 Click in the **Graphics** window and then press Ctrl+A to select both edges.

4 Locate the **y-Axis Data** section. Select the **Description** check box.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `x`.

7 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.

8 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

9 From the **Positioning** list, choose **Interpolated**.

10 From the **Width** list, choose **2**.

11 Click to expand the **Legends** section. Find the **Include** subsection. Select the **Description** check box.

12 Clear the **Solution** check box.

13 Select the **Show legends** check box.

14 Right-click **Displacement magnitude** and choose **Duplicate**.

Magnitude of transverse displacement

1 In the **Model Builder** window, under **Results>Displacement** click **Displacement magnitude 1**.

2 In the **Settings** window for **Line Graph**, type `Magnitude of transverse displacement` in the **Label** text field.

3 Locate the **Selection** section. Click to select the **Activate Selection** toggle button.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `sqrt(v^2+w^2)`.

5 In the **Description** text field, type `Magnitude of transverse displacement`.

6 Right-click **Magnitude of transverse displacement** and choose **Duplicate**.

Axial displacement

1 In the **Model Builder** window, under **Results>Displacement** click **Magnitude of transverse displacement 1**.

2 In the **Settings** window for **Line Graph**, type `Axial displacement` in the **Label** text field.

3 Locate the **y-Axis Data** section. In the **Expression** text field, type `u`.

4 In the **Description** text field, type `Axial displacement`.

5 In the **Displacement** toolbar, click **Plot**.