Created in COMSOL Multiphysics 6.2

Action on Structures Exposed to Fire — Thermal Stress in a Beam

Introduction

This is the 7th verification example from (Ref. 1) which is part of the European Standard EN-1991-1-2:2010-12, Eurocode 1: Actions on structures - Part 1-2: General actions - Actions on structures exposed to fire. It describes the nonlinear mechanical behavior of a beam that is exposed to a temperature gradient.

Model Definition

The modeled geometry is a beam with a cross section of 100 mm^2 and a length of 1 m (Figure 1).

Figure 1: Model setup.

The material properties are given in Ref. 2. The thermal strain function dL depends linearly on the temperature. The stress-strain relationship is a nonlinear function depending on the temperature and strain. Hence, this model is a strongly coupled multiphysics model coupling heat transfer and solid mechanics.

The beam is fixed at the ends. In order to avoid unrealistic stress concentrations when working with a 3D solid model, the beam theory fixed constraint should be interpreted as being constrained in the normal direction, but free to expand in the transverse directions. Rigid body motion is avoided by using relevant prescribed displacements at two points.

In addition the upper and lower surfaces are exposed to different temperatures on upside (`Tu`) and downside (`Td`). In the first case `Tu` = `Td` = 120°C and in the second case `Tu` = 20°C, `Td` = 220°C.

Results and Discussion

The resulting stress distribution for the second case is shown in Figure 2.

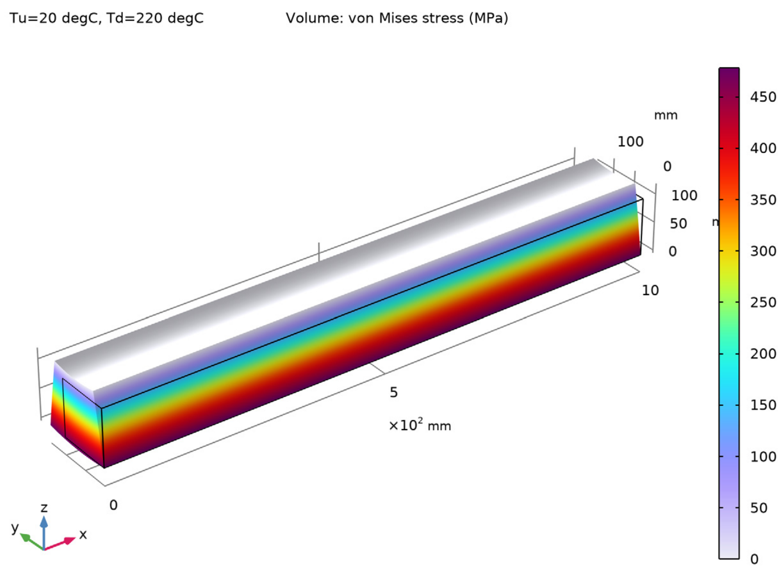


Figure 2: Stress distribution for Tu *= 20°C and* Td *= 220°C.*

To validate the results, the third principal stress is compared to the reference values (Table 1). In both cases, the error is below the maximum allowed error of 5%.

TABLE 1: RESULTS.

Case	Reference stress (N/mm^2)	Calculated stress (N/mm^2)	Error (%)
Tu=Td=120°C	-258.5	-261.9	1.3
Tu=20°C, Td=220°C	-479	-474.8	0.9

References

1. DIN EN 1991-1-2/NA, *National Annex - Nationally determined parameters - Eurocode 1: Actions on structures - Part 1-2: General actions - Actions on structures exposed to fire.*

2. DIN EN 1993-1-2 *Eurocode 3: Design of steel structures - Part 1-2: General rules - Structural fire design; German version EN 1993-1-2:2005 + AC:2009.*

Application Library path: `Heat_Transfer_Module/Verification_Examples/fire_effects_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>Thermal–Structure Interaction>Thermal Stress, Solid**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

Define parameters for the temperatures on the up- and downside of the beam.

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
Tu	120[degC]	393.15 K	Temperature, upside
Td	120[degC]	393.15 K	Temperature, downside

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Block 1 (blk1)

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `1000`.

4 In the **Depth** text field, type `100`.

5 In the **Height** text field, type `100`.

6 Click **Build All Objects**.

MATERIALS

Add a new material for steel. You will define the material properties later, after the physics is set up.

Steel

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Steel` in the **Label** text field.

SOLID MECHANICS (SOLID)

Nonlinear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Material Models>Nonlinear Elastic Material**.

2 Select Domain 1 only.

3 In the **Settings** window for **Nonlinear Elastic Material**, locate the **Nonlinear Elastic Material** section.

4 From the **Material model** list, choose **Uniaxial data**.

Roller 1

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

2 Select Boundaries 1 and 6 only.

Fixed Constraint 1

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

2 Select Point 1 only.

Prescribed Displacement 1

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

2 Select Point 3 only.

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

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

HEAT TRANSFER IN SOLIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.

Temperature 1

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

2 Select Boundary 4 only.

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

4 In the T_0 text field, type `Tu`.

Temperature 2

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

2 Select Boundary 3 only.

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

4 In the T_0 text field, type `Td`.

MULTIPHYSICS

Thermal Expansion 1 (te1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Thermal Expansion 1 (te1)**.

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

3 From the **Input type** list, choose **Thermal strain**.

Now, define the missing material properties. After the physics is set up, the software notices which material properties are necessary to solve the model.

MATERIALS

Steel (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Steel (mat1)**.

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
Uniaxial stress function	sax	`sigma_ax(T, eax)`	N/m²	Nonlinear elastic material
Poisson's ratio	nu	`0.3`	1	Young's modulus and Poisson's ratio
Density	rho	`7850`	kg/m³	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	`k(T)`	W/(m·K)	Basic
Heat capacity at constant pressure	Cp	`Cp(T)`	J/(kg·K)	Basic
Thermal strain	dL_iso ; dLii = dL_iso, dLij = 0	`dL(T)`	1	Thermal expansion

The uniaxial nonlinear elastic data are given in Ref. 2. Here, load the data as interpolation function into the material properties. The function depends on the temperature and the strain.

4 In the **Model Builder** window, expand the **Steel (mat1)** node, then click **Nonlinear elastic material (NonlinearElasticMaterial)**.

5 In the **Settings** window for **Nonlinear Elastic Material**, locate the **Model Inputs** section.

6 Click **Select Quantity**.

7 In the **Physical Quantity** dialog box, type `eax` in the text field.

8 Click **Filter**.

9 In the tree, select **Solid Mechanics>Elastic uniaxial strain (1)**.

10 Click **OK**.

Interpolation 1 (int1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

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

4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `fire_effects_beam_stress_strain.txt`.

6 Click **Import**.

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

Function name	Position in file
sigma_ax	1

8 Locate the **Units** section. In the **Function** table, enter the following settings:

Function	Unit
sigma_ax	N/m^2

9 In the **Argument** table, enter the following settings:

Argument	Unit
Column 1	K
Column 2	1

Define the functions for `k(T)`, `Cp(T)`, and `dL(T)` in the next steps.

Piecewise 1 (pw1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Piecewise**.

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

3 Locate the **Definition** section. In the **Argument** text field, type `T`.

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

Start	End	Function
20	800	54-3.33e-2*T
800	1200	27.3

5 Locate the **Units** section. In the **Arguments** text field, type `degC`.

6 In the **Function** text field, type `W/(m*K)`.

Piecewise 2 (pw2)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Piecewise**.

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

3 Locate the **Definition** section. In the **Argument** text field, type `T`.

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

Start	End	Function
20	600	425+7.73e-1*T-1.69e-3*T^2+2.22e-6*T^3
600	735	666+13002/(738-T)
735	900	545+17820/(T-731)
900	1200	650

5 Locate the **Units** section. In the **Arguments** text field, type `degC`.

6 In the **Function** text field, type `J/(kg*K)`.

Piecewise 1 (pw1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Piecewise**.

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

3 Locate the **Definition** section. In the **Argument** text field, type `T`.

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

Start	End	Function
20	750	1.2e-5*T+0.4e-8*T^2-2.416e-4
750	860	1.1e-2
860	1200	2e-5*T-6.2e-3

5 Locate the **Units** section. In the **Arguments** text field, type `degC`.

MESH 1

Create a swept mesh.

Mapped 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.

2 Select Boundary 1 only.

Distribution 1

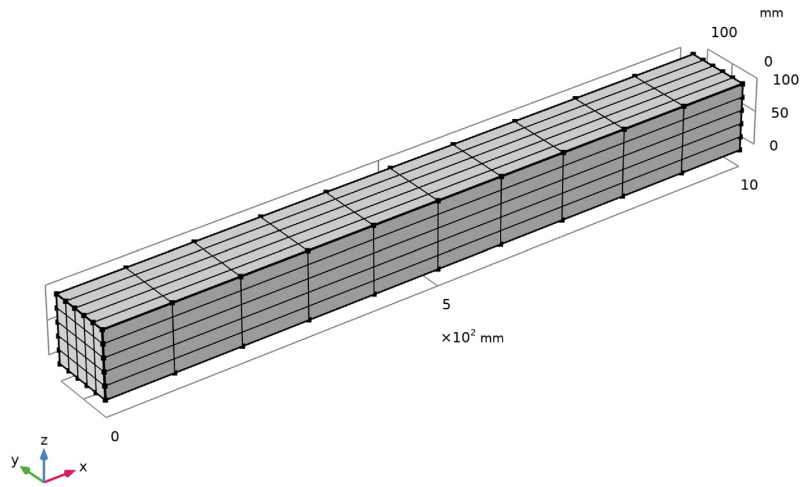
1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 1 and 4 only.

Swept 1

1 In the **Mesh** toolbar, click **Swept**.

2 In the **Settings** window for **Swept**, click **Build All**.



STUDY 1

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Tu (Temperature, upside)	120 20	degC
Td (Temperature, downside)	120 220	degC

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

RESULTS

Volume 1

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

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

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

4 In the **Stress (solid)** toolbar, click **Plot**.

5 Click the **Zoom Extents** button in the **Graphics** toolbar.

Surface Average 1

1 In the **Results** toolbar, click **More Derived Values** and choose **Average>Surface Average**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Surface Average**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Stress>Principal stresses>solid.sp3Gp - Third principal stress - N/m²**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
solid.sp3Gp	N/mm^2	Third principal stress

5 Click **Evaluate**.

TABLE 1

1 Go to the **Table 1** window.

Compare with Table 1.