

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

This model is licensed under the COMSOL Software License Agreement 6.1. All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

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 mm² 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 $Tu = Td = 120^{\circ}C$ and in the second case $Tu = 20^{\circ}C$, $Td = 220^{\circ}C$.

Results and Discussion

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



Figure 2: Stress distribution for $Tu = 20^{\circ}C$ and $Td = 220^{\circ}C$.

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%.

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

TABLE I: RE	SULTS.
-------------	--------

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

- I In the Model Wizard window, click 间 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 **M** Done.

GLOBAL DEFINITIONS

Parameters 1

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

- I In the Model Builder window, under Global Definitions click Parameters I.
- 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 I

- I In the Model Builder window, under Component I (compl) click Geometry I.
- 2 In the Settings window for Geometry, locate the Units section.
- 3 From the Length unit list, choose mm.

Block I (blk1)

- I 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

- I In the Model Builder window, under Component I (compl) 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 I

- I In the Model Builder window, under Component I (compl) 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 I

- I In the Physics toolbar, click 📄 Boundaries and choose Roller.
- **2** Select Boundaries 1 and 6 only.

Fixed Constraint I

I In the Physics toolbar, click 🗁 Points and choose Fixed Constraint.

2 Select Point 1 only.

Prescribed Displacement I

- I 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 Select the Prescribed in z direction check box.

HEAT TRANSFER IN SOLIDS (HT)

In the Model Builder window, under Component I (compl) click Heat Transfer in Solids (ht).

Temperature 1

- I 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

- I 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 (tel)

- I In the Model Builder window, under Component I (compl)>Multiphysics click Thermal Expansion I (tel).
- **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)

- I In the Model Builder window, under Component I (compl)>Materials click Steel (matl).
- 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_a x(T, eax)	N/m²	Nonlinear elastic material
Poisson's ratio	nu	0.3	I	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	Ср	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 (matl) 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).
- IO Click OK.

Interpolation 1 (int1)

I 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 **[I** 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 I	К
Column 2	1

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

Piecewise 1 (pw1)

- I 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)

I 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)

- I 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 I

Create a swept mesh.

Mapped I

- I In the Mesh toolbar, click \triangle Boundary and choose Mapped.
- **2** Select Boundary 1 only.

Distribution I

- I Right-click Mapped I and choose Distribution.
- **2** Select Edges 1 and 4 only.

Swept 1

I In the Mesh toolbar, click 🦓 Swept.

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



STUDY I

Parametric Sweep

- I 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

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

- 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 **Comextents** button in the **Graphics** toolbar.

Surface Average 1

- I In the Results toolbar, click ^{8,85}_{e-12} 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 I (compl)> Solid Mechanics>Stress>Principal stresses>solid.sp3 Third principal stress N/m².
- 4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
solid.sp3	N/mm^2	Third principal stress

5 Click **=** Evaluate.

TABLE

I Go to the **Table** window.

Compare with Table 1.