Created in COMSOL Multiphysics 6.2

Action on Structures Exposed to Fire — Thermal Elongation

Introduction

This is the 4th 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 verifies that the calculated elongation matches the expected values.

Model Definition

The modeled geometry is a cube with side length of 100 mm. The temperature in the block is homogeneous and prescribed. The thermal strain function dL (Figure 1) is given in (Ref. 2).

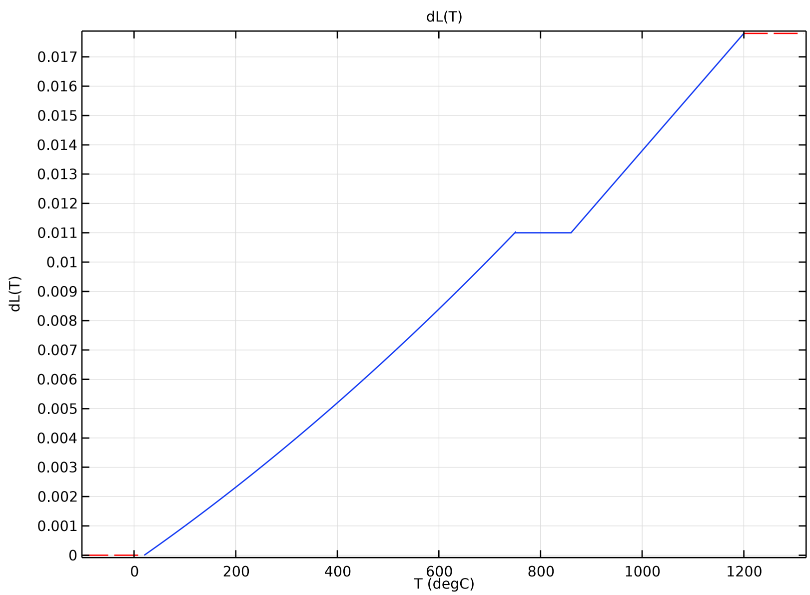


Figure 1: Temperature dependent thermal strain function.

The model is a pure structural mechanics problem. The thermal expansion is calculated according to

$$\varepsilon_{\mathrm{th}} = dL(T, T_{\mathrm{ref}})$$

with the given thermal strain function dL, the reference temperature $T_{\mathrm{ref}} = 20°C$ and the prescribed temperature T (Table 1).

Results and Discussion

The reference and calculated values are given in Table 1 and match exactly. This is expected, because the thermal strain function prescribes the deformation and the deformation is what you compute.

TABLE 1: REFERENCE AND CALCULATED ELONGATION.

TEMPERATURE (°C)	REFERENCE ELONGATION	CALCULATED ELONGATION
100	0.09984	0.09984
300	0.37184	0.37184
500	0.67584	0.67584
600	0.83984	0.83984
700	1.0118	1.0118
900	1.18000	1.18000

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_thermal_elongation`

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>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

Define a parameter for the temperature which is the input for the thermal expansion. A parametric sweep over this temperature will be performed.

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
T_in	100[degC]	373.15 K	Temperature

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 100.

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

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

6 Click **Build All Objects**.

MATERIALS

Define the material properties for steel.

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.

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	210000[N/mm^2]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	1	Young's modulus and Poisson's ratio
Density	rho	7850	kg/m³	Basic

4 Click to expand the **Material Properties** section. In the **Material properties** tree, select **Solid Mechanics>Thermal Expansion>Thermal strain (dL)**.

5 Click **Add to Material**.

Piecewise 1 (pw1)

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

2 Right-click **Component 1 (comp1)>Materials>Steel (mat1)> Thermal expansion (ThermalExpansion)** and choose **Functions>Piecewise**.

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

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

5 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

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

7 Click **Plot**. Compare with Figure 1.

Steel (mat1)

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

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

3 In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Thermal strain	dL_iso ; dLii = dL_iso, dLij = 0	dL(T)	1	3x3

The variable T is not known, yet. Add the temperature in the **Model Inputs** section to define it.

4 Locate the **Model Inputs** section. Click **Select Quantity**.

5 In the **Physical Quantity** dialog box, type `temperature` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Temperature (K)**.

8 Click **OK**.

SOLID MECHANICS (SOLID)

Roller 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Roller**.

2 Select Boundaries 3, 5, and 6 only.

The roller condition ensures that the structure expands in all directions uniformly.

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 **Thermal Expansion Properties** section.

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

Define a parameter for the input temperature.

4 Locate the **Model Input** section. From the T list, choose **User defined**. In the associated text field, type `T_in`.

MESH 1

Swept 1

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

Size

1 In the **Model Builder** window, click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extremely coarse**.

4 Click **Build All**.

A very coarse mesh is sufficient. Even just one element would be enough, because the deformation is prescribed and you verify that the calculated deformation gives the same value. This is a basic test to validate that the tested functionality works correctly.

STUDY 1

Step 1: Stationary

Set up a parametric sweep over the input temperature.

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 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
T_in (Temperature)	100 300 500 600 700 900	degC

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

A 3D stress plot is created automatically. Add a predefined plot group to visualize the displacement field.

ADD PREDEFINED PLOT

1 In the **Home** toolbar, click **Windows** and choose **Add Predefined Plot**.

2 Go to the **Add Predefined Plot** window.

3 In the tree, select **Study 1/Solution 1 (sol1)>Solid Mechanics>Displacement (solid)**.

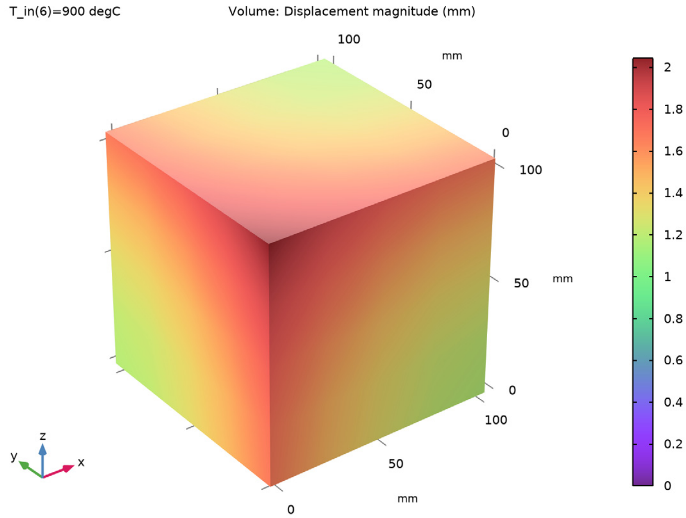
4 Click **Add Plot** in the window toolbar.

RESULTS

Displacement (solid)

1 In the **Displacement (solid)** toolbar, click **Plot**.

2 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 4 only.

3 In the **Settings** window for **Surface Average**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit	Description
`w`	`mm`	`Displacement field, Z component`

5 Click **Evaluate**.

TABLE 1

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

To compare these results with the reference values, import the data as interpolation function.

GLOBAL DEFINITIONS

Interpolation 1 (int1)

1 In the **Home** toolbar, click **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_thermal_elongation_dlref.txt`.

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

Function name	Position in file
dl_ref	1

7 Locate the **Units** section. In the **Argument** table, enter the following settings:

Argument	Unit
Column 1	degC

8 Locate the **Definition** section. Click **Import**.

It is not necessary to compute the whole study again. To make the data available for postprocessing, just update the solution.

STUDY 1

In the **Study** toolbar, click **Update Solution**.

RESULTS

Surface Average 1

1 In the **Model Builder** window, under **Results>Derived Values** click **Surface Average 1**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
w	mm	Displacement field, Z component
dl_ref(T_in)		Interpolation 1

4 In the **Results** toolbar, click **Evaluate** and choose **Clear and Evaluate All**.

TABLE 1

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

The computed and reference values match exactly. Compare with Table 1.