



Model created in COMSOL Multiphysics 6.4

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).

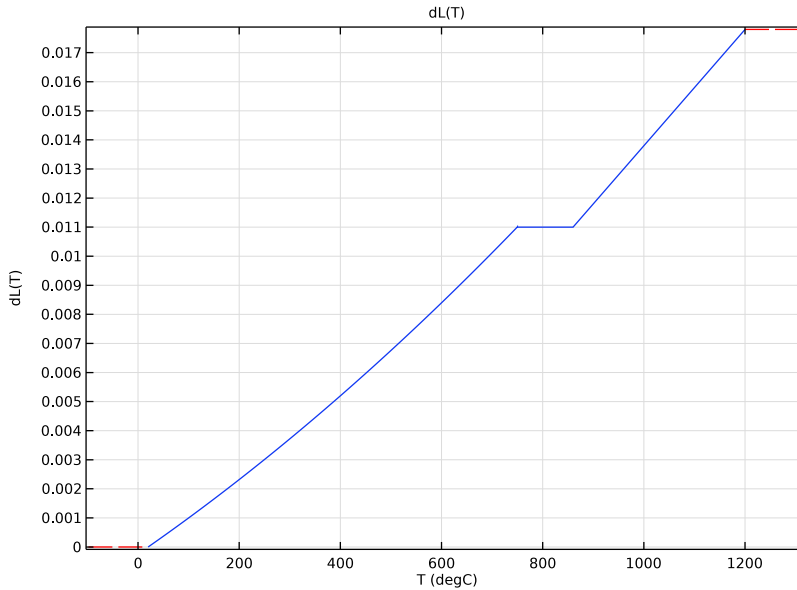


Figure 1: Temperature dependent thermal strain function.

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

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

with the given thermal strain function dL , the reference temperature $T_{ref} = 20^\circ\text{C}$ 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 I

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 I



- 1 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
T_in	100[degC]	373.15 K	Temperature

GEOMETRY I

- 1 In the **Model Builder** window, under **Component I (comp1)** 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)

- 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	l	Young's modulus and Poisson's ratio
Density	rho	7850	kg/m ³	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} ; $dL_{ii} = dL_{iso}$, $dL_{ij} = 0$	$dL(T)$	l	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, type temperature in the text field.
- 6 In the tree, select **General > Temperature (K)**.
- 7 Click **OK**.

SOLID MECHANICS (SOLID)

Roller 1


- 1 In the **Physics** toolbar, click  **Boundaries** 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 I


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 plot group from the **Result Templates** to visualize the displacement field.

RESULT TEMPLATES

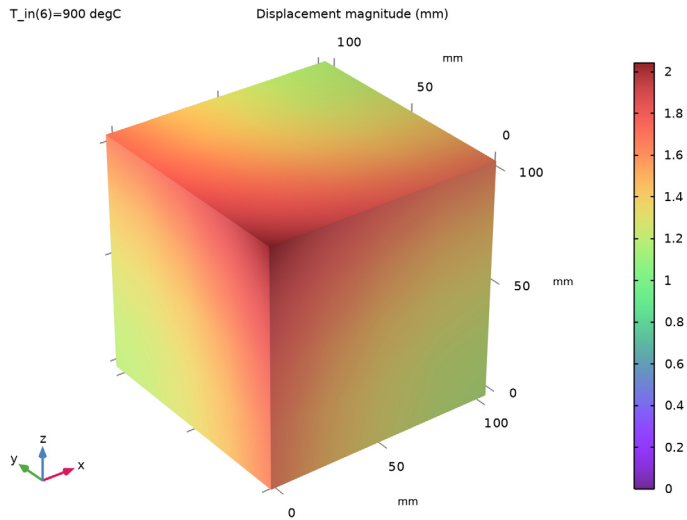
- 1 In the **Results** toolbar, click  **Result Templates** to open the **Result Templates** window.
- 2 Go to the **Result Templates** window.
- 3 In the tree, select **Study I/Solution I (sol1) > Solid Mechanics > Displacement (solid)**.
- 4 Click the **Add Result Template** button in the window toolbar.
- 5 In the **Results** toolbar, click  **Result Templates** to close the **Result Templates** window.

RESULTS

Displacement (solid)

- 1 In the **Displacement (solid)** toolbar, click  **Plot**.
- 2 Click the  **Zoom Extents** button in the **Graphics** toolbar.

3 In the **Model Builder** window, under **Results** click **Displacement (solid)**.



Surface Average 1

- 1 In the **Results** toolbar, click 8.85×10^{-12} **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 $f(x)$ **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_thermal_elongation_d1ref.txt`.
- 6 Locate the **Data Column Settings** section. In the table, click to select the cell at row number 1 and column number 1.
- 7 In the **Unit** text field, type `degC`.
- 8 In the table, click to select the cell at row number 2 and column number 1.
- 9 In the **Name** text field, type `d1_ref`.
- 10 In the **Unit** text field, type `1`.
- 11 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
<code>w</code>	<code>mm</code>	Displacement field, Z component
<code>d1_ref(T_in)</code>		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](#).