



Model created in COMSOL Multiphysics 6.4

# Action on Structures Exposed to Fire — Cooling Process

## Introduction

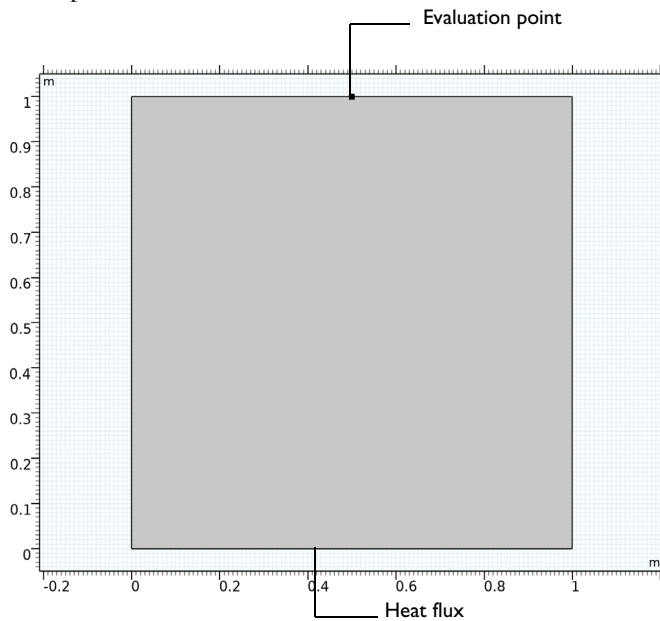
---

This is the first 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. A transient cooling process is modeled. You verify that the numerical results obtained with COMSOL Multiphysics are within the validity ranges specified in the norm.

## Model Definition

---

The modeled geometry is a square with a side length of 1 m. [Figure 1](#) shows the geometry and setup.



*Figure 1: Model geometry and setup.*

The material properties are listed in [Table 1](#).

TABLE 1: MATERIAL PROPERTIES.

PROPERTY	NAME	VALUE
Thermal conductivity	k	1 W/(m·K)

TABLE I: MATERIAL PROPERTIES.

PROPERTY	NAME	VALUE
Density	$\rho$	1000 kg/m <sup>3</sup>
Heat Capacity	$C_p$	1 J/(kg·K)

The initial temperature is set to 1000°C and is cooled down using a heat flux condition on the bottom boundary according to

$$q_0 = h(T_{\text{ext}} - T)$$

with the heat transfer coefficient  $h = 1 \text{ W}/(\text{m}^2\cdot\text{K})$  and the external temperature  $T_{\text{ext}} = 0^\circ\text{C}$ . All other boundaries are adiabatic. The temperature evolution over 30 min is computed and the results are compared to the reference values given by Ref. 1. To fulfill the norm, the maximum deviation from the reference values must not exceed a relative error of 1% and an absolute error of 5 K.

### Results and Discussion

The temperature distribution after 30 min is shown in Figure 2.

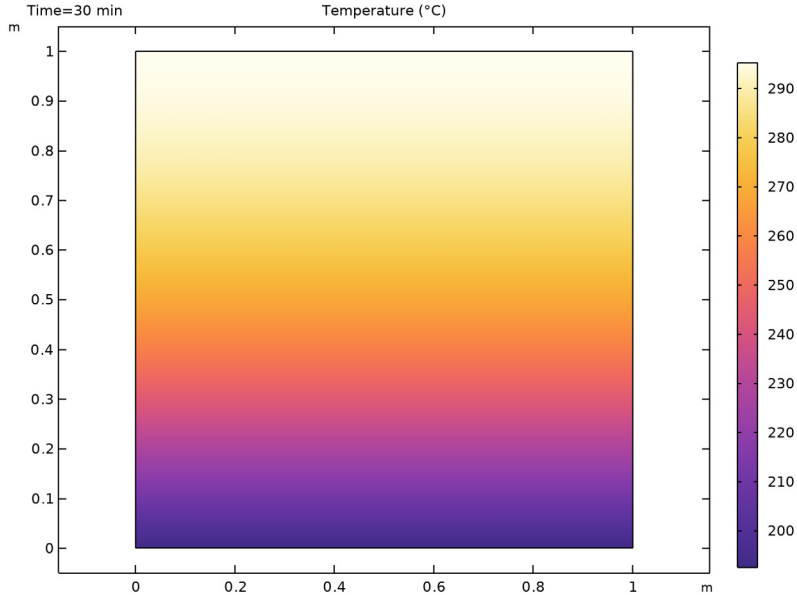
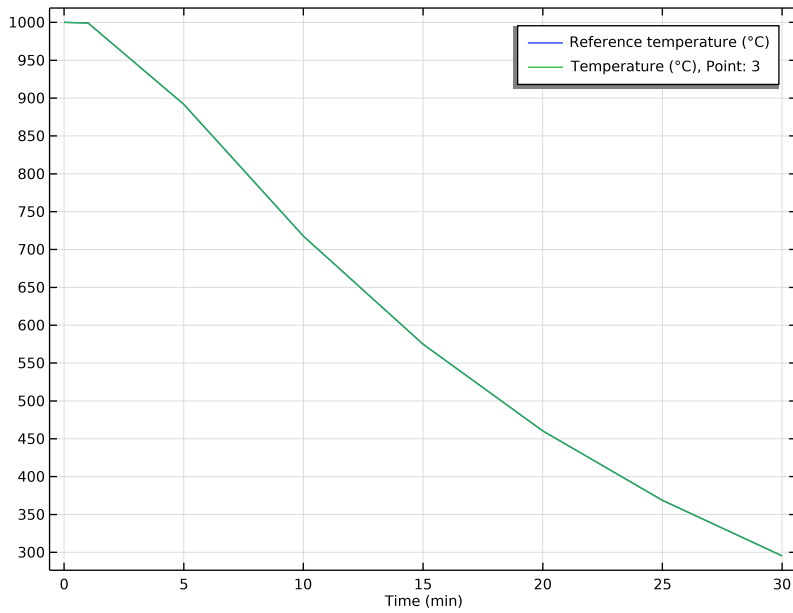


Figure 2: Temperature distribution after 30 min.

The reference and computed temperatures are compared in [Figure 3](#). The numerical values match the norm data very well.



*Figure 3: Reference (blue) and calculated temperature (green) over time.*

The reference and calculated temperatures together with the absolute and relative errors for each time are listed in [Table 2](#).

TABLE 2: RESULTS.

Time (min)	Reference temperature(°C)	Calculated temperature(°C)	Absolute error (K)	Relative Error (%)
0	1000	1000	0	0
1	999.3	999.3	<0.05	<0.01
5	891.8	891.8	<0.05	<0.01
10	717.7	717.7	<0.05	<0.01
15	574.9	574.9	<0.05	<0.01
20	460.4	460.4	<0.05	<0.01
25	368.7	368.7	<0.05	<0.01
30	295.3	295.3	<0.05	0.01

## Reference

---

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.*

---

**Application Library path:** Heat\_Transfer\_Module/Verification\_Examples/  
fire\_effects\_cooling


---

## 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  **2D**.
- 2 In the **Select Physics** tree, select **Heat Transfer** > **Heat Transfer in Solids (ht)**.
- 3 Click **Add**.
- 4 Click  **Study**.
- 5 In the **Select Study** tree, select **General Studies** > **Time Dependent**.
- 6 Click  **Done**.

### GEOMETRY I

#### Square 1 (sq1)

In the **Geometry** toolbar, click  **Square**.

#### Point 1 (pt1)

- 1 In the **Geometry** toolbar, click  **Point**.
- 2 In the **Settings** window for **Point**, locate the **Point** section.
- 3 In the **x** text field, type 0.5.
- 4 In the **y** text field, type 1.
- 5 Click  **Build All Objects**.

## 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**, locate the **Material Contents** section.
- 3 In the table, enter the following settings:


Property	Variable	Value	Unit	Property group
Thermal conductivity	$k_{iso}$ ; $k_{ii} = k_{iso}$ , $k_{ij} = 0$	1	W/(m·K)	Basic
Density	$\rho$	1000	kg/m <sup>3</sup>	Basic
Heat capacity at constant pressure	$C_p$	1	J/(kg·K)	Basic

## HEAT TRANSFER IN SOLIDS (HT)

### Initial Values 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)** > **Heat Transfer in Solids (ht)** click **Initial Values 1**.
- 2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
- 3 In the  $T$  text field, type 1000[degC].

### Heat Flux 1



- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Heat Flux**.
- 2 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.
- 3 From the **Flux type** list, choose **Convective heat flux**.
- 4 In the  $h$  text field, type 1.
- 5 In the  $T_{ext}$  text field, type 0[degC].
- 6 Select Boundary 2 only.

To compare the simulation results with the reference values, create an interpolation function for the norm data which are given in a file.

## GLOBAL DEFINITIONS

### Reference temperature

- 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_cooling_Tref.txt`.
- 6 Click  **Import**.
- 7 In the **Label** text field, type `Reference temperature`.
- 8 Locate the **Definition** section. In the **Function name** text field, type `Tref`.
- 9 Locate the **Units** section. In the **Argument** table, enter the following settings:


Argument	Unit
<code>t</code>	<code>s</code>

- 10 In the **Function** table, enter the following settings:

Function	Unit
<code>Tref</code>	<code>degC</code>

## STUDY I

### *Step 1: Time Dependent*


- 1 In the **Model Builder** window, under **Study I** click **Step 1: Time Dependent**.
- 2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
- 3 From the **Time unit** list, choose **min**.
- 4 In the **Output times** text field, type `0 1 5 10 15 20 25 30`.  
The default solver is accurate enough to validate the benchmark. Tightening the tolerance improves the results, especially in terms of energy balance which you can check with the quantity `ht.energyBalance`.
- 5 From the **Tolerance** list, choose **User controlled**.
- 6 In the **Relative tolerance** text field, type `1e-5`.
- 7 In the **Study** toolbar, click  **Compute**.

## RESULTS

Change the unit of the temperature results to degrees Celsius.

### *Preferred Units 1*

- 1 In the **Results** toolbar, click  **Configurations** and choose **Preferred Units**.

- 2 In the **Settings** window for **Preferred Units**, locate the **Units** section.
- 3 Click  **Add Physical Quantity**.
- 4 In the **Physical Quantity** dialog, select **General > Temperature (K)** in the tree.
- 5 Click **OK**.
- 6 In the **Settings** window for **Preferred Units**, locate the **Units** section.
- 7 In the table, enter the following settings:



Quantity	Unit	Preferred unit
Temperature	K	°C

- 8 Click  **Apply**.


*Temperature (ht)*

Default plot shows temperature distribution. Compare it with [Figure 2](#).


*Surface I*

- 1 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Surface I**.
- 2 In the **Temperature (ht)** toolbar, click  **Plot**.
- 3 Click the  **Zoom Extents** button in the **Graphics** toolbar.


*Global Evaluation: Reference temperature*

- 1 In the **Results** toolbar, click  **Global Evaluation**.
- 2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.
- 3 In the table, enter the following settings:

Expression	Unit	Description
Tref(t)	°C	Reference temperature

- 4 In the **Label** text field, type Global Evaluation: Reference temperature.
- 5 Click  **Evaluate**.

*Point Evaluation: Temperature*

- 1 In the **Results** toolbar, click  **Point Evaluation**.
- 2 Select Point 3 only.
- 3 In the **Settings** window for **Point Evaluation**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit	Description
T	°C	Temperature

5 In the **Label** text field, type Point Evaluation: Temperature.

Instead of creating a new table, evaluate the results in the same table as before.

6 Right-click on the **Point Evaluation: Temperature** node.

7 Go to **Evaluate** and click **Table I - Global Evaluation: Reference temperature (Tref(t))**.

#### TABLE I

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

2 Click the **Table Graph** button in the window toolbar.

#### RESULTS

##### Temperature

1 In the **Model Builder** window, under **Results** click **ID Plot Group 2**.

2 In the **Settings** window for **ID Plot Group**, type Temperature in the **Label** text field.

##### Table Graph 1

1 In the **Model Builder** window, click **Table Graph 1**.

2 In the **Settings** window for **Table Graph**, click to expand the **Legends** section.

3 Select the **Show legends** checkbox.

The reference and computed values match very well (compare with [Figure 3](#)).

Finally, evaluate the absolute and relative errors.

##### Absolute and Relative Error

1 In the **Results** toolbar, click  $\frac{8.85}{e-12}$  **Point Evaluation**.

2 In the **Settings** window for **Point Evaluation**, type Absolute and Relative Error in the **Label** text field.

3 Select Point 3 only.

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

Expression	Unit	Description
$\text{abs}(T - T_{\text{ref}}(t))$	K	Absolute error
$\text{abs}(T - T_{\text{ref}}(t)) / (T_{\text{ref}}(t) - 273.15[\text{K}])$	%	Relative error

5 Click  **Evaluate**.

**TABLE 2**

1 Go to the **Table 2** window.

The absolute and relative errors are within the allowed range of 5 K or 1% respectively.  
Compare with [Table 2](#).