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.

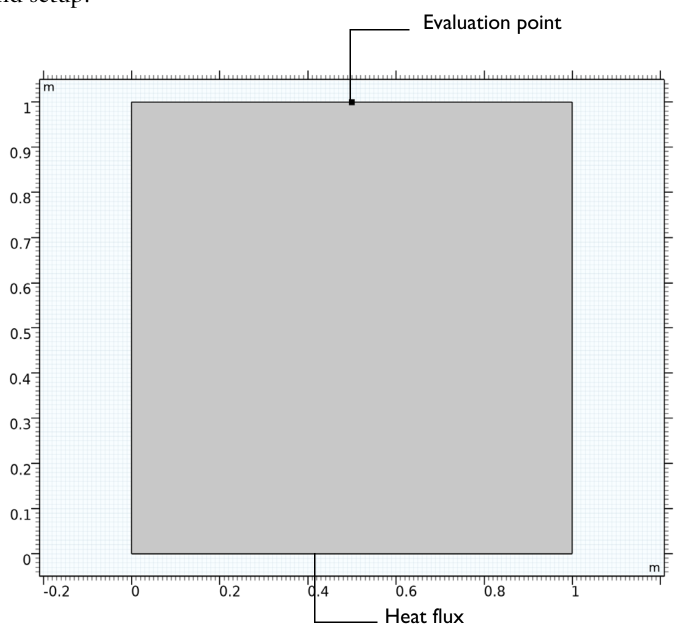

Figure 1: Model geometry and setup.

The material properties are listed in Table 1.

TABLE 1: MATERIAL PROPERTIES.

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

TABLE 1: MATERIAL PROPERTIES.

PROPERTY	NAME	VALUE
Density	r	1000 kg/m³
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

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

with the heat transfer coefficient h = 1 W/(m²·K) and the external temperature T_{ext} = 0°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 derivation 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.

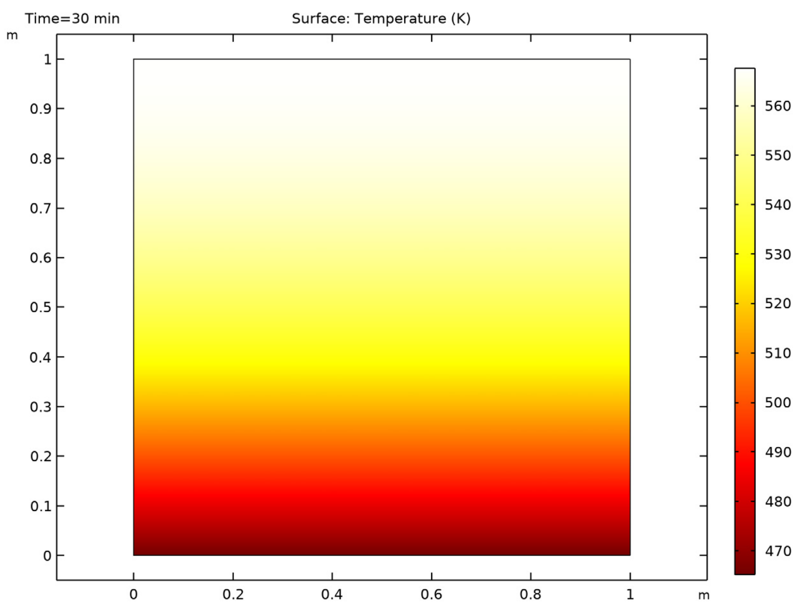

Figure 2: Temperature distribution after 30 min.

The reference and computed temperatures are compared in Figure 3. The numerical values match the norm data very well.

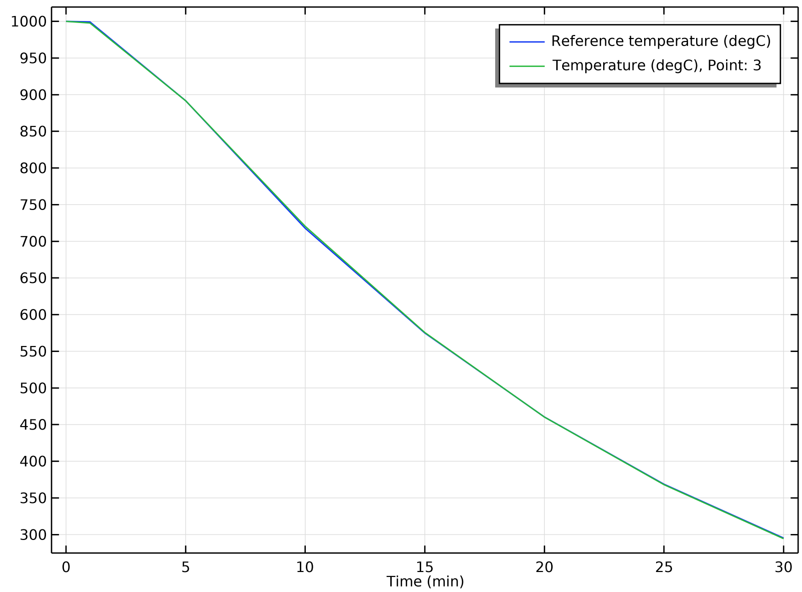

Figure 3: Reference (blue) and calculated temperature (green) over time.

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	997.7	1.6	0.2
5	891.8	891.7	0.1	0.01
10	717.7	720.4	2.7	0.4
15	574.9	575.6	0.7	0.1
20	460.4	460.2	0.2	0.04
25	368.7	368.0	0.7	0.2
30	295.3	294.4	0.9	0.3

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 1

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 ; kii = k_iso, kij = 0	1	W/(m·K)	Basic
Density	rho	1000	kg/m³	Basic
Heat capacity at constant pressure	Cp	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 Click the **Convective heat flux** button.

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 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_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 **Arguments** text field, type `s`.

10 In the **Function** text field, type `degC`.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** 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`.

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

RESULTS

Temperature (ht)

A temperature and isothermal contour plot are created automatically. Compare the temperature plot with Figure 2.

Global Evaluation: Reference temperature

1 In the **Results** toolbar, click 8.5 **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)`	`degC`	`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 8.85 e-12 **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`	`degC`	`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 1 - Global Evaluation: Reference temperature (Tref(t))**.

TABLE

1 Go to the **Table** window.

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

RESULTS

Temperature

1 In the **Model Builder** window, under **Results** click **1D Plot Group 3**.

2 In the **Settings** window for **1D 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** check box.

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 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
`abs(T-Tref(t))`	`K`	`Absolute error`
`abs(T-Tref(t))/(Tref(t)-273.15[K])`	`%`	`Relative error`

5 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

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