Created in COMSOL Multiphysics 5.6

Action on Structures Exposed to Fire — Heating Process

Introduction

This is the second 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 a heating process using a temperature dependent thermal conductivity. 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 0.2 m (Figure 1).

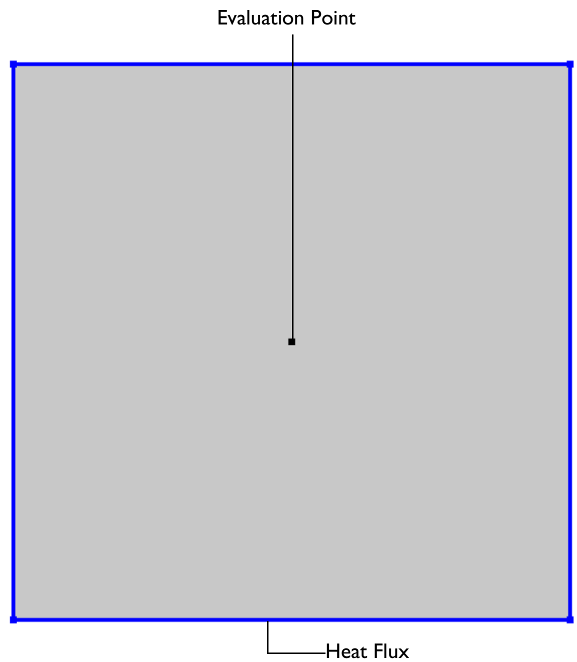

Figure 1: Model geometry and set-up

The initial temperature is 0°C. A heat flux condition is applied to all boundaries according to

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

with the heat transfer coefficient h = 10 W/(m^2·K) and T_{ext} = 1000°C. In addition, flux due to radiation is considered:

$$q_r = \varepsilon\sigma(T_{ext}^4 - T^4)$$

The surface emissivity ε is 0.8 and σ is the Stefan-Boltzmann constant.

The material properties are listed below (Table 1).

TABLE 1: MATERIAL PROPERTIES.

Property	Name	Value
Density	ρ	2400 kg/m^3
Heat Capacity	C_p	1000 J/(kg·K)

The thermal conductivity is a linear function of the temperature (Figure 2).

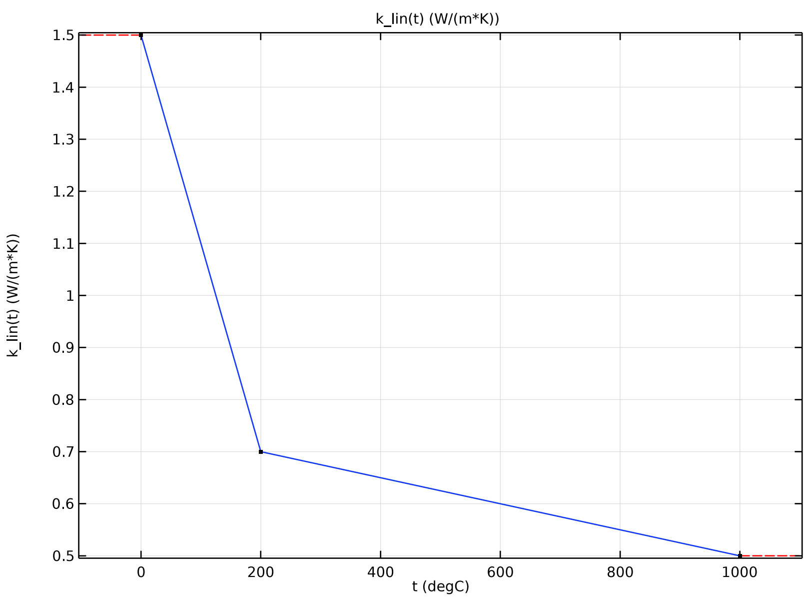

Figure 2: Thermal conductivity function

Results and Discussion

The temperature distribution after 180 min is shown in Figure 3.

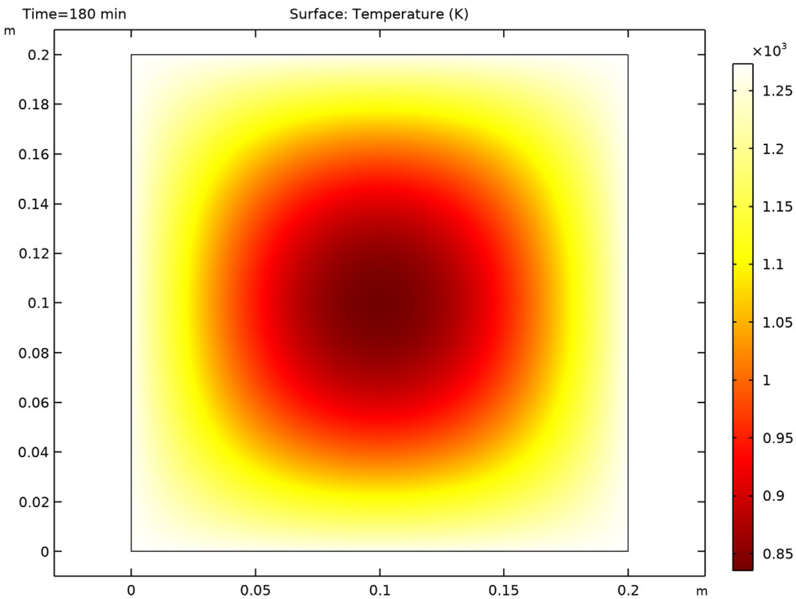

Figure 3: Temperature distribution after 180 min.

The reference and computed temperatures are compared in Figure 4. The numerical values match the norm values very well.

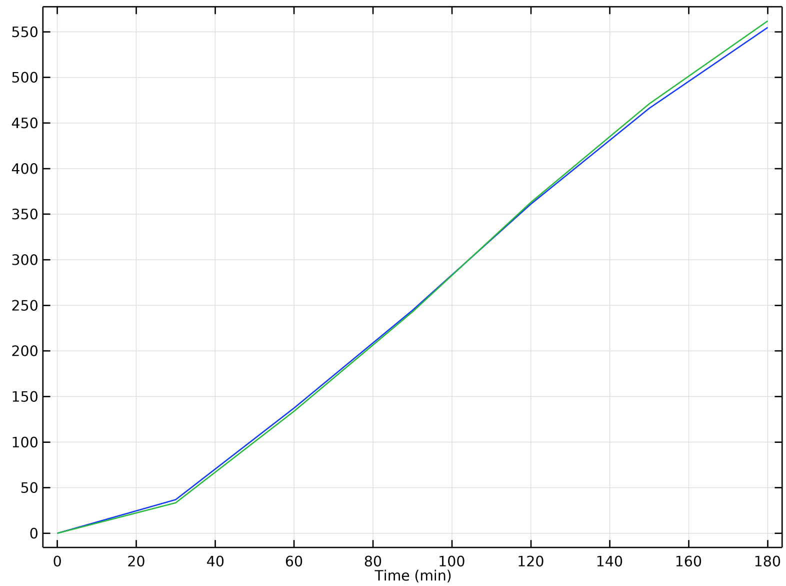

Figure 4: Reference (blue) and calculated temperature (green).

The exact values, and 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	Relative error
30	36.9	33.4	3.5	9.4
60	137.4	133.9	3.5	2.5
90	244.6	242.9	1.7	0.7
120	361.1	363.0	1.9	0.5
150	466.2	471.1	4.9	1.1
180	554.8	562.0	7.2	1.3

To fulfill the norm, the maximum deviation from the reference values must not exceed 5 K for $t \leq 60$ min and 3% for $t > 60$ min.

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_heating`

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

Start with creating an interpolation function for the norm values. It will be used later for comparison with the numerical results.

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_heating_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 `min`.

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

Create another interpolation function for the thermal conductivity.

Thermal conductivity

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Interpolation**.

2 In the **Settings** window for **Interpolation**, type `Thermal conductivity` in the **Label** text field.

3 Locate the **Definition** section. In the **Function name** text field, type `k_lin`.

4 In the table, enter the following settings:

t	f(t)
0	1.5
200	0.7
1000	0.5

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

6 In the **Function** text field, type `W/(m*K)`.

7 Click **Plot**.

GEOMETRY 1

Square 1 (sq1)

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

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

3 In the **Side length** text field, type `0.2`.

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 `0.1`.

4 In the **y** text field, type `.1`.

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	`k_lin(T)`	W/(m·K)	Basic
Density	rho	2400	kg/m³	Basic
Heat capacity at constant pressure	Cp	1000	J/(kg·K)	Basic

Note, that for the thermal conductivity, you use the interpolation function defined before with the expression `k_lin(T)`.

DEFINITIONS

Ambient Properties 1 (ampr1)

1 In the **Physics** toolbar, click **Shared Properties** and choose **Ambient Properties**.

2 In the **Settings** window for **Ambient Properties**, locate the **Ambient Conditions** section.

3 In the T_{amb} text field, type `1000[degC]`.

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 `Tref(0)`.

Heat Flux 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 In the **Settings** window for **Heat Flux**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

4 Locate the **Heat Flux** section. Click the **Convective heat flux** button.

5 In the h text field, type `10`.

6 From the T_{ext} list, choose **Ambient temperature (ampr1)**.

Surface-to-Ambient Radiation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Surface-to-Ambient Radiation**.

2 In the **Settings** window for **Surface-to-Ambient Radiation**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

4 Locate the **Surface-to-Ambient Radiation** section. From the T_{amb} list, choose **Ambient temperature (ampr1)**.

5 From the ε list, choose **User defined**. In the associated text field, type `0.8`.

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 30 60 90 120 150 180`.

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

RESULTS

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)`	`degC`	`Reference temperature`

4 In the **Label** text field, type `Reference temperature`.

5 Click **Evaluate**.

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	degC	Temperature

5 In the **Label** text field, type `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.

Compare with Figure 4.

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**, locate the **Data** section.

3 From the **Time selection** list, choose **Manual**.

4 In the **Time indices (1-7)** text field, type `2 3 4 5 6 7`.

5 Select Point 3 only.

6 In the **Label** text field, type `Absolute and relative error`.

7 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

8 Click **= Evaluate**.

TABLE

1 Go to the **Table** window.

The absolute and relative errors are within the allowed range. Compare with Table 2.