Created in COMSOL Multiphysics 6.1

# 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).

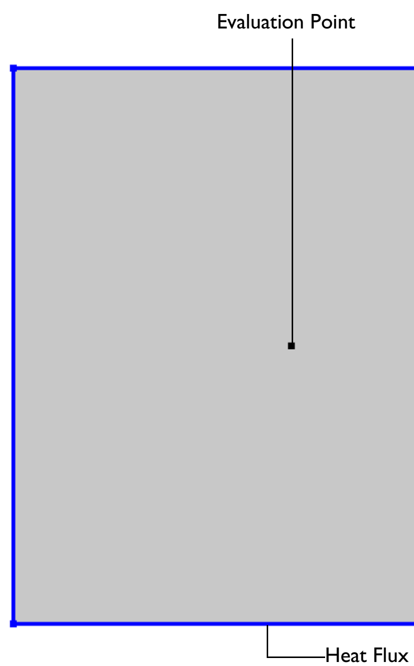


*Figure 1: Model geometry and set-up*

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 piecewise linear function of the temperature (Figure 2).

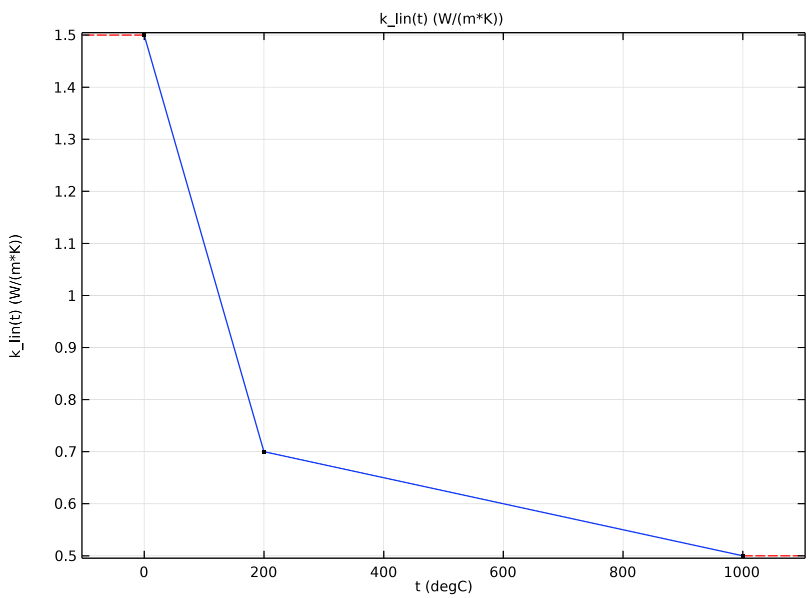


*Figure 2: Thermal conductivity function*

## *Results and Discussion*

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

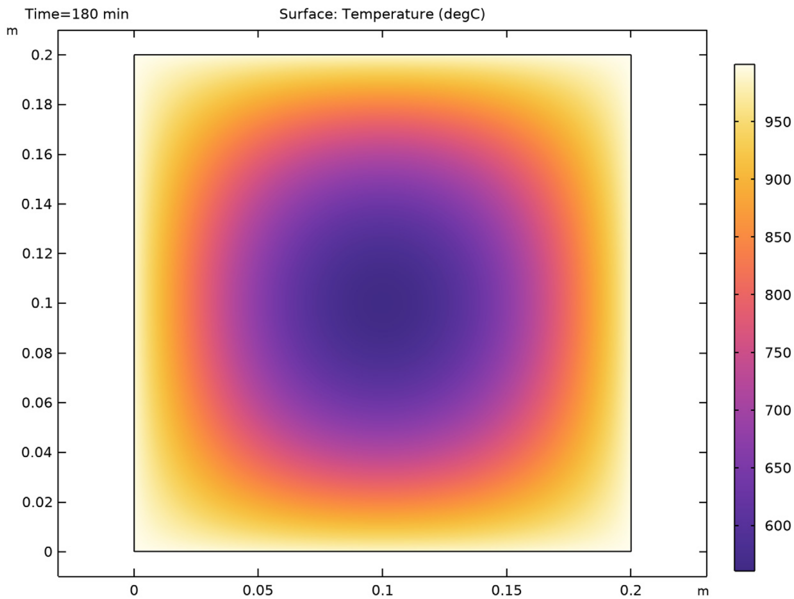


*Figure 3: Temperature distribution after 180 min.*

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

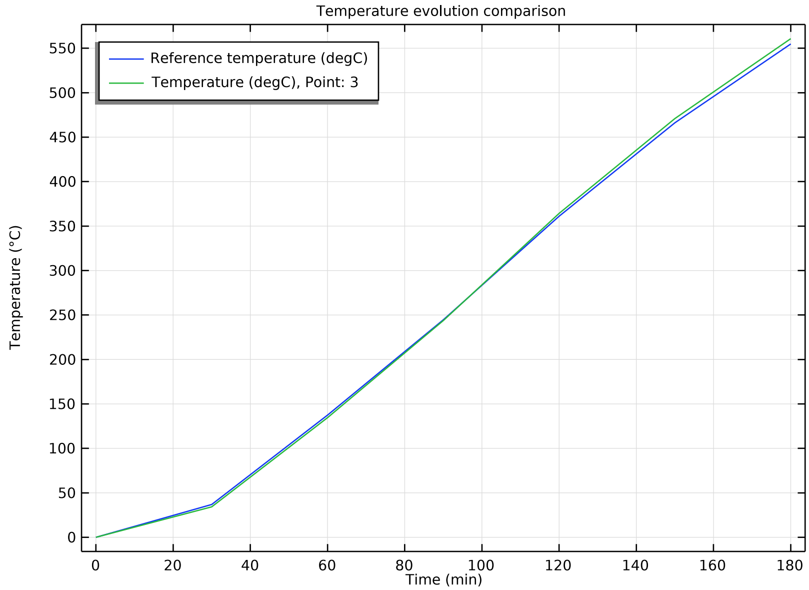


*Figure 4: Reference (blue) and calculated temperature (green).*

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 | 34.3 | 2.7 | 7.2 |
| 60 | 137.4 | 134.5 | 2.9 | 2.1 |
| 90 | 244.6 | 243.7 | 0.9 | 0.4 |
| 120 | 361.1 | 364.3 | 3.2 | 0.9 |
| 150 | 466.2 | 471.0 | 4.8 | 1.0 |
| 180 | 554.8 | 560.8 | 6.0 | 1.1 |

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 **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 **Argument** table, enter the following settings:

| Argument | Unit |
|---|---|
| t | min |

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

| Function | Unit |
|---|---|
| Tref | 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 **Argument** table, enter the following settings:

| Argument | Unit |
|---|---|
| t | degC |

6 In the **Function** table, enter the following settings:

| Function | Unit |
|---|---|
| k_lin | 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. From the **Flux type** list, choose **Convective heat flux**.

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`.

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 **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.

**3** Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

**4** In the **Title** text area, type `Temperature evolution comparison`.

**5** Locate the **Plot Settings** section.

**6** Select the **y-axis label** check box. In the associated text field, type `Temperature (°C)`.

**7** Locate the **Legend** section. From the **Position** list, choose **Upper left**.

*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.

**4** In the **Temperature** toolbar, click **Plot**.

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.