Created in COMSOL Multiphysics 6.0

Action on Structures Exposed to Fire — Heat Transfer in Multiple Layers

Introduction

This is the third 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 computes heat transfer through layers with different material properties.

Model Definition

The modeled geometry is depicted in Figure 1. The total side length is 0.1 m.

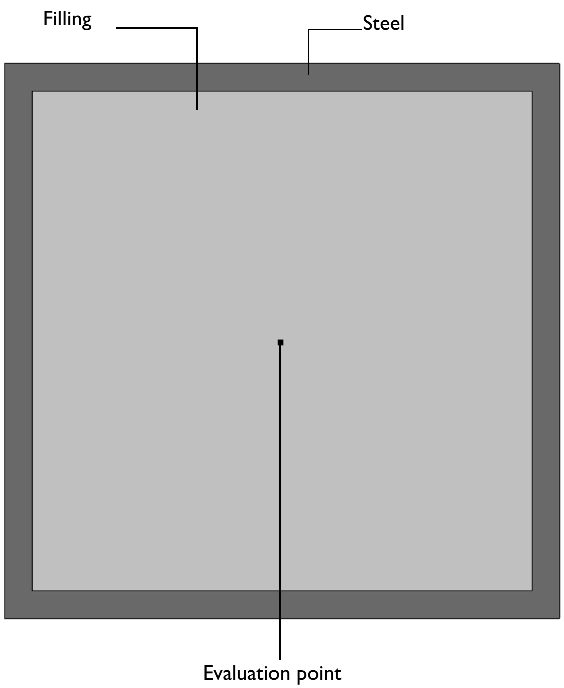

Figure 1: Model geometry and setup. The size of the outer layer in this image is larger for better visibility.

The thickness of the steel layer is 0.5 mm.

The material properties for the filling are listed in Table 1

TABLE 1: MATERIAL PROPERTIES.

Property	Value
Thermal conductivity	0.05 W/(m·K)
Density	50 kg/m^3
Heat capacity	1000 J/(kg·K)

The material properties for steel are given in Ref. 2. The heat capacity is a highly nonlinear function of the temperature (Figure 2).

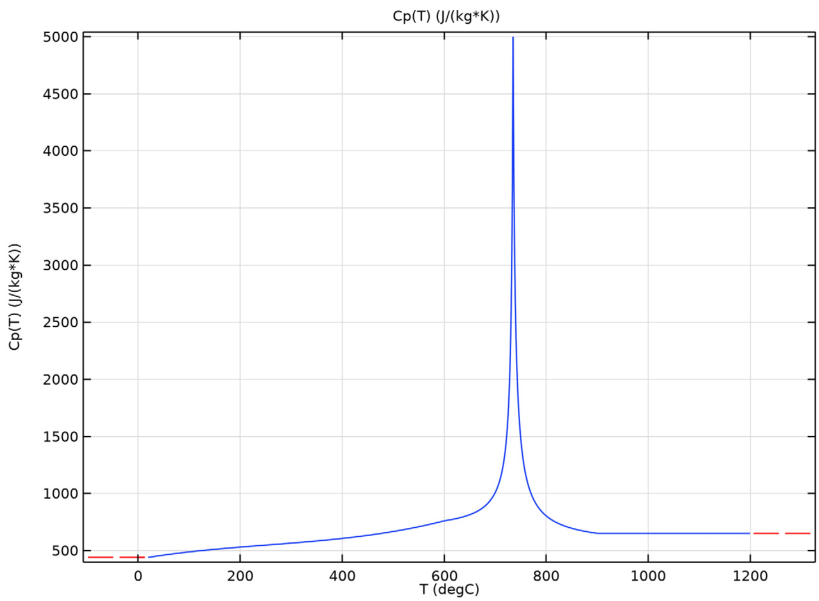

Figure 2: Temperature-dependent heat capacity.

The thermal conductivity depends linearly on the temperature according to

$$k(T(°\mathrm{C})) = \begin{cases} 54 - 0.0333T, & 20 \le T < 800 \\ 27.3 & T \ge 800 \end{cases}$$

The model computes the heat transfer over 180 min starting from an initial temperature of 0°C. All outer boundaries are cooled by convective and radiative heat flux

$$q = h(T_{ext} - T) + \varepsilon\sigma(T_{ext}^4 - T^4)$$

with the heat transfer coefficient h = 10 W/(m²·K) and T_{ext} = 0°C.

Results and Discussion

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

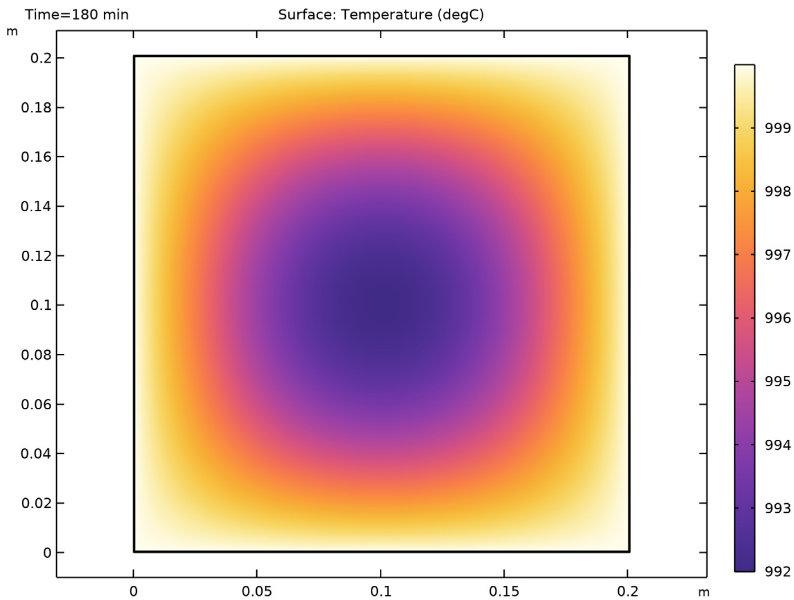

Figure 3: Temperature distribution after 180 min.

The reference and computed temperatures are compared in Figure 4.

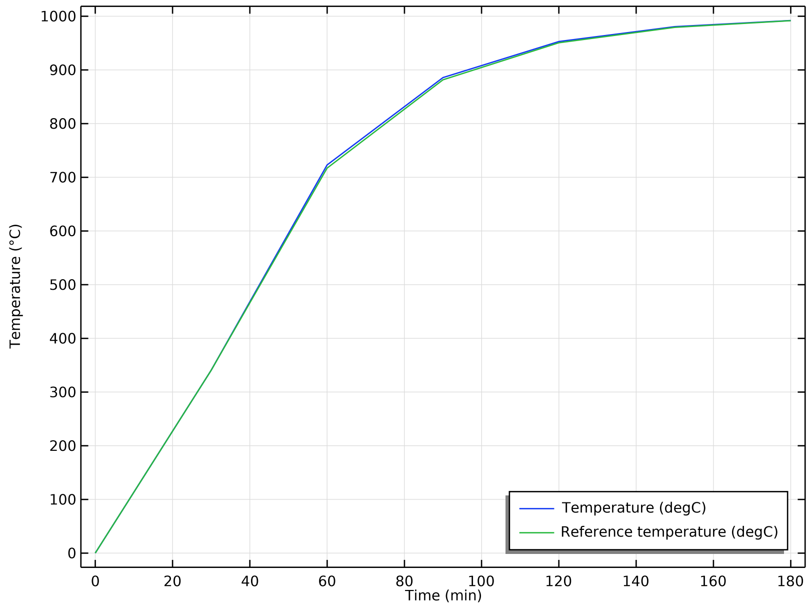

Figure 4: Reference (blue) and calculated temperature (green) over time.

Table 2 lists the reference and calculated temperatures as well as the absolute and relative errors.

TABLE 2: RESULTS.

Time (s)	Reference temperature(°C)	Calculated temperature(°C)	Absolute error (K)	Relative error (%)
30	340.5	340.6	0.1	0.03
60	717.1	723.0	5.9	0.8
90	881.6	885.7	4.1	0.5
120	950.6	952.9	2.3	0.2
150	979.3	980.6	1.3	0.1
180	991.7	992.0	0.3	0.03

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. The norm is not fulfilled by means of absolute error in the range of temperature where the heat capacity is strongly nonlinear. The reference values are defined as the average temperature calculated by different software packages, whereas one value was out of range itself and hence the temperatures are too low in this area.

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_multiple_layers`

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**.

GLOBAL DEFINITIONS

Create an interpolation function for the reference temperatures to compare the simulation results with these data.

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_multiple_layers_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 **Function** table, enter the following settings:

Function	Unit
Tref	degC

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

Argument	Unit
t	min

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 `0.201`.

4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	0.0005

5 Click **Build All Objects**.

6 Select the **Layers to the left** check box.

7 Select the **Layers to the right** check box.

8 Select the **Layers on top** check box.

9 Click **Build All Objects**.

MATERIALS

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.

Heat Capacity

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Materials>Steel (mat1)** node.

2 Right-click **Component 1 (comp1)>Materials>Steel (mat1)>Basic (def)** and choose **Functions>Piecewise**.

3 In the **Settings** window for **Piecewise**, type `Heat Capacity` in the **Label** text field.

4 In the **Function name** text field, type `Cp`.

5 Locate the **Definition** section. In the **Argument** text field, type `T`.

6 Find the **Intervals** subsection. In the table, enter the following settings:

Start	End	Function
20	600	425+7.73e-1*T-1.69e-3*T^2+2.22e-6*T^3
600	735	666+13002/(738-T)
735	900	545+17820/(T-731)
900	1200	650

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

8 In the **Function** text field, type `J/(kg*K)`.

Plot the function and compare with Figure 2.

9 Click **Plot**.

Thermal conductivity

1 Right-click **Basic (def)** and choose **Functions>Piecewise**.

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

3 In the **Function name** text field, type `k`.

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	800	`54-3.33e-2*T`
800	1200	`27.3`

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

7 In the **Function** text field, type `W/m/K`.

Steel (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Steel (mat1)**.

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(T)`	W/(m·K)	Basic
Density	rho	7850	kg/m³	Basic
Heat capacity at constant pressure	Cp	`Cp(T)`	J/(kg·K)	Basic

Filling

1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.

2 Select Domain 5 only.

3 In the **Settings** window for **Material**, type `Filling` in the **Label** text field.

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

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	0.05	W/(m·K)	Basic
Density	rho	50	kg/m³	Basic
Heat capacity at constant pressure	Cp	1000	J/(kg·K)	Basic

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)**.

7 Locate the **Boundary Selection** section. Click **Copy Selection**.

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 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, Press `Shift+V` on you keyboard.

5 Make sure all outer boundaries are in the **Boundary Selection** list.

6 click **OK**.

7 In the **Settings** window for **Surface-to-Ambient Radiation**, locate the **Surface-to-Ambient Radiation** section.

8 From the T_{amb} list, choose **Ambient temperature (ampr1)**.

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

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Extra fine**.

4 Locate the **Sequence Type** section. From the list, choose **User-controlled mesh**.

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Triangular 1**.

2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 5 only.

Mapped 1

In the **Mesh** toolbar, click **Mapped**.

Size 1

1 Right-click **Mapped 1** and choose **Size**.

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

5 In the associated text field, type `0.0002`.

This way, you ensure to resolve the thin outer layer properly.

6 Click **Build All**.

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`.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 From the **Maximum step constraint** list, choose **Constant**.

5 In the **Maximum step** text field, type 1.

This setting prevents the solver from taking too large time steps when the overall convergence is good. Otherwise, it could happen that the solver overestimates the time step size, which is required to resolve the transient behavior properly.

6 Click **Compute**.

The solver takes about 30 seconds to compute the solution.

RESULTS

Create a **Cut Point** dataset that is used to evaluate the temperature and compare it to the values in the norm.

Cut Point 2D 1

1 In the **Results** toolbar, click **Cut Point 2D**.

2 In the **Settings** window for **Cut Point 2D**, locate the **Point Data** section.

3 In the **X** text field, type 0.1.

4 In the **Y** text field, type 0.1.

Point Evaluation 1

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 **Dataset** list, choose **Cut Point 2D 1**.

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

Expression	Unit	Description
T	degC	Temperature
Tref(t)	degC	Reference temperature

5 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

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

RESULTS

Table Graph 1

Compare with Table 2.

1 In the **Model Builder** window, under **Results>1D Plot Group 3** 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 From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
`Temperature (degC)`
`Reference temperature (degC)`

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 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

4 Locate the **Plot Settings** section. Select the **y-axis label** check box.

5 In the associated text field, type `Temperature (°C)`.

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

Compare with Figure 4.

Finally, evaluate the absolute and relative errors.

Point Evaluation 2

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 From the **Dataset** list, choose **Cut Point 2D 1**.

6 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`

7 Click Evaluate.

8 Click next to Evaluate, then choose **New Table**.

Compare with Table 2.