Created in COMSOL Multiphysics 6.3

Thermal Bridges in Building Construction — 2D Composite Structure

Introduction

The European standard EN ISO 10211:2017 for thermal bridges in building constructions provides four test cases — two 2D and two 3D — for validating a numerical method (Ref. 1). If the values obtained by a method conform to the results of all these four cases, the method is classified as a *three-dimensional steady-state high precision method*.

COMSOL Multiphysics successfully passes all the test cases described by the standard. This document presents an implementation of the second 2D model (Case 2).

The example studies the temperature distribution and heat flux through a two-dimensional cross-section of an insulating wall. Table 1 and Table 2 compare the numerical results with the target solution provided by the standard.

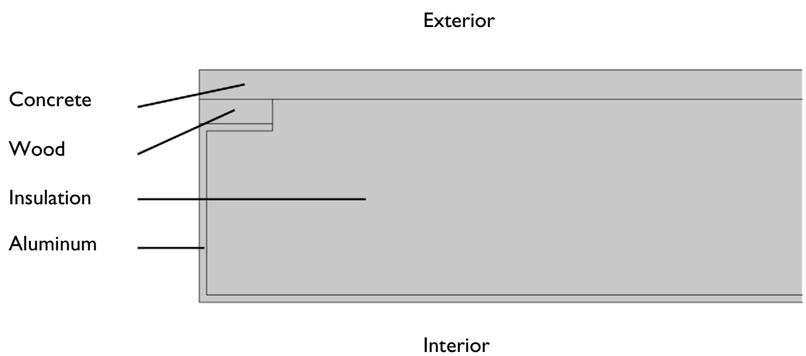


Figure 1: Geometry and material distribution of ISO 10211:2017 test case 2.

Model Definition

Figure 1 shows the geometry and material distribution. Four materials with distinct thermal conductivities are used in the structure: a concrete layer on the top side, an aluminum layer on the left and bottom sides, an insulation material layer occupying the largest space in the structure, and a wooden batten between the aluminum and concrete layers.

The left and right boundaries are thermally insulated. The top and bottom boundaries, corresponding to cold exterior and hot interior sides, are subject to convective heat flux. The exterior temperature is 0°C while interior temperature is 20°C. The ISO 10211:2017 standard specifies the values of the thermal resistance, R, which is related to the heat transfer coefficient, h, according to

$$h = \frac{1}{R}$$

Results and Discussion

The temperature profile in Figure 2 shows the effects of the highly conductive aluminum layer on the left side.

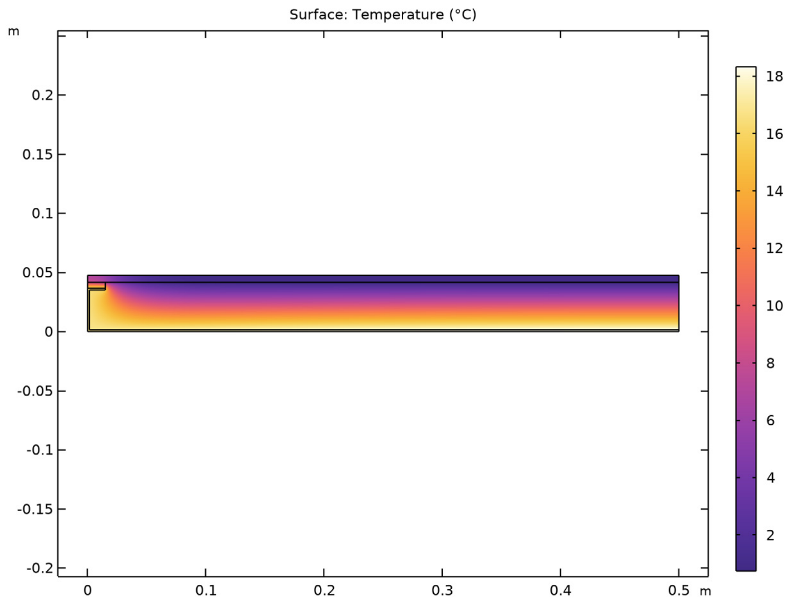


Figure 2: Temperature distribution of ISO 10211:2017 test case 2.

The numerical results of the COMSOL Multiphysics simulation are compared with the expected values provided by EN ISO 10211:2017 (Ref. 1). Table 1 shows the comparison for the total heat flux and Table 2 compares temperature at nine particular points.

TABLE 1: COMPARISON BETWEEN EXPECTED VALUES AND COMPUTED VALUES (TOTAL HEAT FLUX).

EXPECTED VALUE	COMPUTED VALUE	DIFFERENCE
9.5 W/m	9.49 W/m	0.08%

TABLE 2: COMPARISON BETWEEN EXPECTED VALUES AND COMPUTED VALUES (TEMPERATURE).

(X,Y) COORD. (MM)	EXPECTED VALUE (°C)	COMPUTED VALUE (°C)	DIFF. (°C)	(X,Y) COORD. (MM)	EXPECTED VALUE (°C)	COMPUTED VALUE (°C)	DIFF. (°C)
(0, 47.5)	7.1	7.07	0.03	(15, 41.5)	6.3	6.27	0.03
(0, 41.5)	7.9	7.90	0.00	(15, 36.5)	16.3	16.33	0.03
(0, 36.5)	16.4	16.41	0.01	(500, 47.5)	0.8	0.76	0.04
(0, 0)	16.8	16.77	0.03	(500, 41.5)	0.8	0.83	0.03
				(500, 0)	18.3	18.33	0.03

The maximum permissible differences to pass this case validation, 0.1°C for temperature and 0.1% for total heat flux, are respected.

Reference

1. European Committee for Standardization, *EN ISO 10211, Thermal bridges in building construction – Heat flows and surface temperatures – Detailed calculations (ISO 10211:2017)*, Appendix A, pp. 54–60, 2017.

Application Library path: `Heat_Transfer_Module/Buildings_and_Constructions/thermal_bridge_2d_composite_structure`

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 > Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

Define the parameters necessary to build the geometry.

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
W	0.5[m]	0.5 m	Structure width
w	15[mm]	0.015 m	Short width for wood and aluminum domains
h1	6[mm]	0.006 m	Concrete domain height
h2	5[mm]	0.005 m	Wood domain height
h3	41.5[mm]	0.0415 m	Insulation domain height
h4	h3-h2	0.0365 m	Aluminum domain height
t4	1.5[mm]	0.0015 m	Aluminum domain thickness

GEOMETRY 1

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type W.
4 In the **Height** text field, type h3+h1.
5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	h3

6 Click **Build Selected**.

This two-layered rectangle corresponds to the insulation and concrete layers. Continue with the aluminum layer, which consists of three rectangles.

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `W`.

4 In the **Height** text field, type `t4`.

5 Click **Build Selected**.

Rectangle 3 (r3)

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

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `t4`.

4 In the **Height** text field, type `h4`.

5 Click **Build Selected**.

Rectangle 4 (r4)

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

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `W`.

4 In the **Height** text field, type `t4`.

5 Locate the **Position** section. In the **y** text field, type `h4-t4`.

6 Click **Build Selected**.

Union 1 (uni1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 In the **Settings** window for **Union**, locate the **Union** section.

3 Clear the **Keep interior boundaries** checkbox.

4 In the **Model Builder** window, click **Union 1 (uni1)**.

5 Click the **Clear Selection** button for **Input objects**.

6 Select the objects **r2**, **r3**, and **r4** only.

7 Click **Build Selected**.

Add a rectangle for the wooden batten.

Rectangle 5 (r5)

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

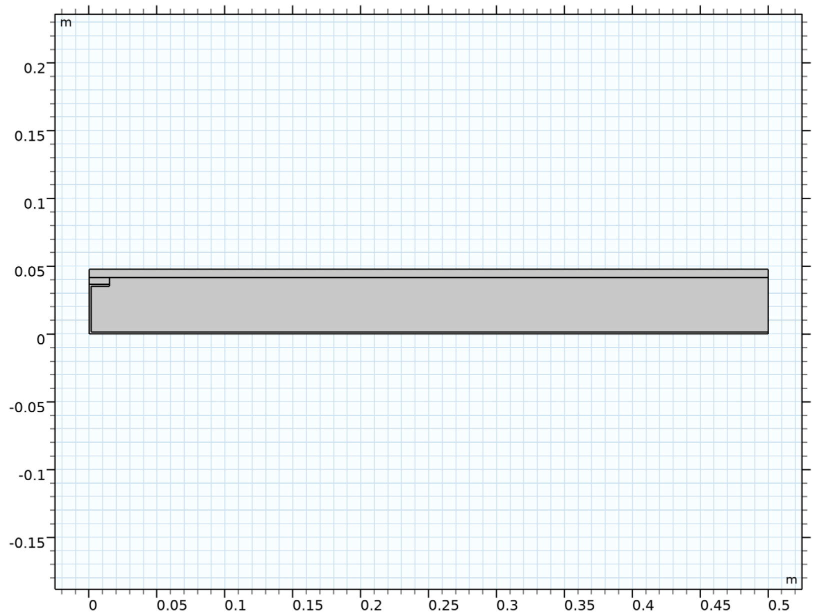
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `W`.

4 In the **Height** text field, type `h2`.

5 Locate the **Position** section. In the **y** text field, type `h4`.

6 Click **Build All Objects**.



MATERIALS

Concrete

1 In the **Materials** toolbar, click **Blank Material**.

2 In the **Settings** window for **Material**, type `Concrete` in the **Label** text field.

3 Select Domain 3 only.

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	`1.15[W/(m*K)]`	W/(m·K)	Basic
Density	rho	`2300[kg/m^3]`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`880[J/(kg*K)]`	J/(kg·K)	Basic

Wood

1 In the **Materials** toolbar, click **Blank Material**.

2 In the **Settings** window for **Material**, type `Wood` in the **Label** text field.

3 Select Domain 2 only.

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	`0.12[W/(m*K)]`	W/(m·K)	Basic
Density	rho	`500[kg/m^3]`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`2500[J/(kg*K)]`	J/(kg·K)	Basic

Insulation

1 In the **Materials** toolbar, click **Blank Material**.

2 In the **Settings** window for **Material**, type `Insulation` in the **Label** text field.

3 Select Domain 4 only.

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	`0.029[W/(m*K)]`	W/(m·K)	Basic
Density	rho	`150[kg/m^3]`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`1000[J/(kg*K)]`	J/(kg·K)	Basic

Aluminum

1 In the **Materials** toolbar, click **Blank Material**.

2 In the **Settings** window for **Material**, type `Aluminum` in the **Label** text field.

3 Select Domain 1 only.

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	230[W/(m*K)]	W/(m·K)	Basic
Density	rho	2700[kg/m^3]	kg/m³	Basic
Heat capacity at constant pressure	Cp	900[J/(kg*K)]	J/(kg·K)	Basic

HEAT TRANSFER IN SOLIDS (HT)

Heat Flux 1

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

2 Select Boundaries 2 and 10 only.

3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

4 From the **Flux type** list, choose **Convective heat flux**.

5 In the h text field, type `1[W/(m^2*K)]/0.11`.

6 In the T_{ext} text field, type `20[degC]`.

Heat Flux 2

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

2 Select Boundary 9 only.

3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

4 From the **Flux type** list, choose **Convective heat flux**.

5 In the h text field, type `1[W/(m^2*K)]/0.06`.

6 In the T_{ext} text field, type `0[degC]`.

STUDY 1

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**.

The default plot shows the temperature distribution (Figure 2).

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Heat Transfer in Solids** > **Isothermal Contours (ht)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

The second plot shows isothermal contours in the structure.

RESULTS

Follow the steps below to compute the heat flux given in Table 1.

Line Integration 1

1 In the **Results** toolbar, click **More Derived Values** and choose **Integration** > **Line Integration**.

2 Select Boundaries 2 and 10 only.

3 In the **Settings** window for **Line Integration**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Heat Transfer in Solids** > **Boundary fluxes** > **ht.q0 - Inward heat flux - W/m²**.

4 Click **Evaluate**.

Finally, export the temperature values at the points given in Table 2 to compare the results with the expected values.

Data 1

1 In the **Results** toolbar, click **Data** and choose **Data**.

2 In the **Settings** window for **Data**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`T`	`°C`	`Temperature`

4 Locate the **Output** section. Click **Browse**.

5 Browse to a suitable folder, enter the filename `thermal_bridge_2d_composite_structure_result.txt`, and then click **Save**.

6 From the **Points to evaluate in** list, choose **Grid**.

7 In the **X** text field, type `0,w,W`.

8 In the **Y** text field, type `0,h4,h4+h2,h4+h2+h1`.

9 Click **Export**.