Created in COMSOL Multiphysics 6.2

Thermal Bridges in Building Construction — 2D Square Column

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 first 2D model (Case 1).

This example studies the temperature distribution in a square column. Cold and hot temperature conditions are applied to the boundaries. Due to the symmetry of the problem, the geometry can be simplified to half of the square. The temperature field created by heat conduction is measured at 28 equidistant points in the structure to compare with the analytic data.

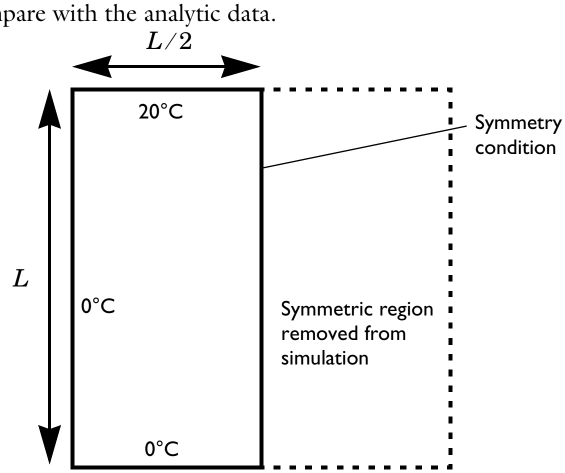

Figure 1: Geometry and boundary conditions of ISO 10211:2017 test case 1.

Model Definition

Figure 1 shows the geometry and the boundary conditions. The model only includes the half left of the square and applies a symmetry condition to the right boundary. The temperature at the remaining boundaries is maintained at the values specified in the figure.

Notes About the COMSOL Implementation

This case corresponds to a stationary analysis and the structural material has a homogeneous thermal conductivity, k. Therefore, its value does not affect the stationary temperature field. However, to properly define the model, a value must be specified for k. This implementation sets k equal to 1 W/(m·K), the density to 1 kg/m^3 and the heat capacity to 1 J/(kg·K).

The temperature is evaluated on the regular grid shown in Figure 2. Because of similarity, the value of L does not affect the results, but a length value must be specified to completely define the model. Here, L is set to 0.8 m.

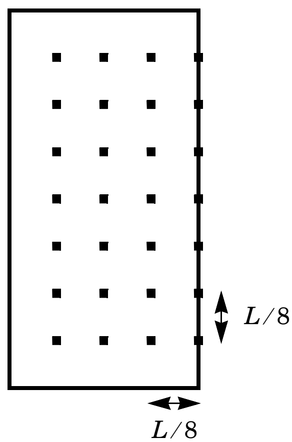

Figure 2: Regular grid where the temperature is evaluated.

Results and Discussion

Figure 3 shows the temperature gradient resulting from the temperature differences between the boundaries.

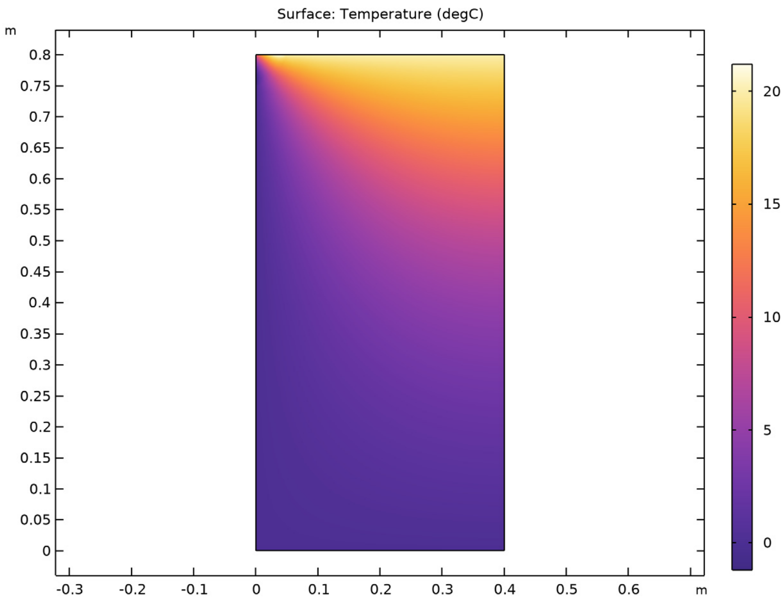

Figure 3: Temperature distribution of ISO 10211:2017 test case 1.

Table 1 compares the numerical results of COMSOL Multiphysics with the expected values provided by EN ISO 10211:2017 (Ref. 1).

TABLE 1: COMPARISON BETWEEN EXPECTED VALUES AND COMPUTED VALUES AT 28 POINTS.

(X,Y) COORD. (M)	EXPECTED VALUE (°C)	COMPUTED VALUE (°C)	DIFF. (°C)	(X,Y) COORD. (M)	EXPECTED VALUE (°C)	COMPUTED VALUE (°C)	DIFF. (°C)
(0.1, 0.1)	0.3	0.34	0.04	(0.3, 0.4)	4.7	4.66	0.04
(0.2, 0.1)	0.6	0.63	0.03	(0.4, 0.4)	5.0	5.00	0.00
(0.3, 0.1)	0.8	0.82	0.02	(0.1, 0.5)	3.2	3.19	0.01
(0.4, 0.1)	0.9	0.89	0.01	(0.2, 0.5)	5.6	5.61	0.01
(0.1, 0.2)	0.7	0.74	0.04	(0.3, 0.5)	7.0	7.01	0.01
(0.2, 0.2)	1.4	1.36	0.04	(0.4, 0.5)	7.5	7.47	0.03
(0.3, 0.2)	1.8	1.77	0.03	(0.1, 0.6)	5.3	5.25	0.05
(0.4, 0.2)	1.9	1.91	0.01	(0.2, 0.6)	8.6	8.64	0.04

TABLE 1: COMPARISON BETWEEN EXPECTED VALUES AND COMPUTED VALUES AT 28 POINTS.

(X,Y) COORD. (M)	EXPECTED VALUE (°C)	COMPUTED VALUE (°C)	DIFF. (°C)	(X,Y) COORD. (M)	EXPECTED VALUE (°C)	COMPUTED VALUE (°C)	DIFF. (°C)
(0.1, 0.3)	1.3	1.26	0.04	(0.3, 0.6)	10.3	10.32	0.02
(0.2, 0.3)	2.3	2.31	0.01	(0.4, 0.6)	10.8	10.81	0.01
(0.3, 0.3)	3.0	2.99	0.01	(0.1, 0.7)	9.7	9.66	0.04
(0.4, 0.3)	3.2	3.22	0.02	(0.2, 0.7)	13.4	13.38	0.02
(0.1, 0.4)	2.0	2.01	0.01	(0.3, 0.7)	14.7	14.73	0.03
(0.2, 0.4)	3.6	3.64	0.04	(0.4, 0.7)	15.1	15.09	0.01

The maximum permissible difference, 0.1°C, to pass this case validation is 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_square_column`

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

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
L	0.8[m]	0.8 m	Square side length

GEOMETRY 1

Include only half of the square due to the symmetry in this model.

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 `L/2`.

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

5 In the **Geometry** toolbar, click **Build All**.

MATERIALS

Material 1 (mat1)

1 In the **Materials** toolbar, click **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	1	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 `0[degC]`.

Temperature 1

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

2 Select Boundaries 1 and 2 only.

3 In the **Settings** window for **Temperature**, locate the **Temperature** section.

4 In the T_0 text field, type `0[degC]`.

Temperature 2

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

2 Select Boundary 3 only.

3 In the **Settings** window for **Temperature**, locate the **Temperature** section.

4 In the T_0 text field, type `20[degC]`.

Symmetry 1

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

2 Select Boundary 4 only.

STUDY 1

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

RESULTS

Temperature (ht)

The first default plot group shows the temperature distribution. Display the plot in degrees Celsius.

Surface 1

1 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 From the **Unit** list, choose **degC**.

4 In the **Temperature (ht)** toolbar, click **Plot**.

5 Click the **Zoom Extents** button in the **Graphics** toolbar.

Figure 3 shows the computed temperature distribution.

Data 1

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

Follow the steps below to get the temperature values at the 28 points of the grid and export them to a file.

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

3 In the table, enter the following settings:

Expression	Unit	Description
T	degC	Temperature

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

5 Browse to a suitable folder, enter the filename `thermal_bridge_2d_square_column.txt`, and then click **Save**.

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

7 Click **Range** for the *x*-coordinate.

8 In the **Range** dialog box, type `L/8` in the **Start** text field.

9 In the **Step** text field, type `L/8`.

10 In the **Stop** text field, type `L/2`.

11 Click **Add**.

12 In the **Settings** window for **Data**, locate the **Output** section.

13 Click **Range** for the *y*-coordinate.

14 In the **Range** dialog box, type `L/8` in the **Start** text field.

15 In the **Step** text field, type `L/8`.

16 In the **Stop** text field, type `L-L/8`.

17 Click **Add**.

18 Click **Export**.