Thermal Bridges in Building Construction—2D Square Column

Introduction

The European standard EN ISO 10211:2007 for thermal bridges in building constructions provides four test models—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 the COMSOL 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 is 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.

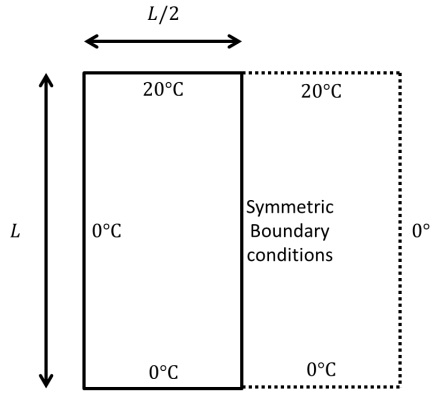

Figure 1: Geometry and boundary conditions of ISO 10211:2007 model case 1.

Model Definition

Figure 1 shows the geometry and the boundary conditions. The model only includes the left half 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 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.

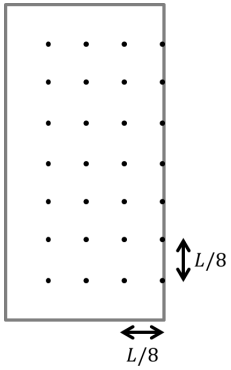

Figure 2: Regular grid where the temperature is evaluated.

 ©2012 COMSOL

Results and Discussion

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

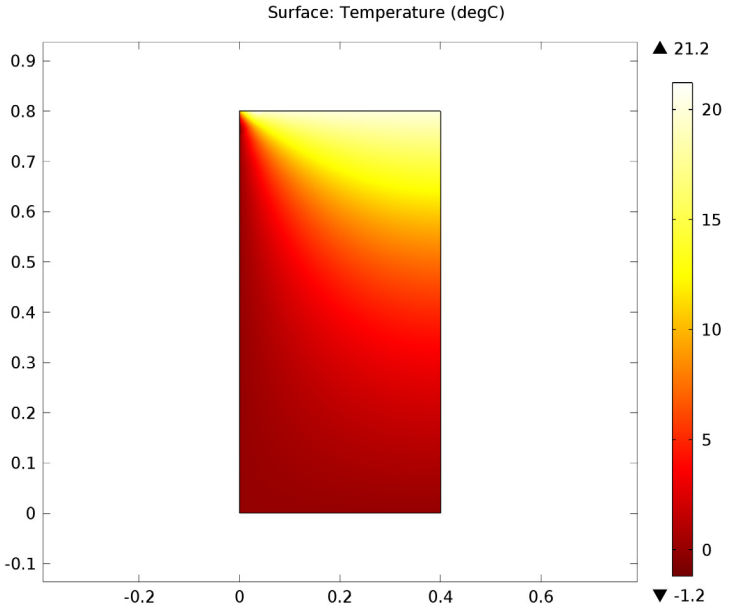

Figure 3: Temperature distribution of ISO 10211:2007 model case 1.

Table 1 compares the numerical results of COMSOL Multiphysics with the expected values provided by EN ISO 10211:2007 (Ref. 1).

TABLE 1: COMPARISON BETWEEN EXPECTED VALUES AND COMSOL VALUES AT 28 POINTS

(X,Y) COORDINATE (M)	EXPECTED VALUE (°C)	COMSOL VALUE (°C)	ERROR (°C)	(X,Y) COORDINATE (M)	EXPECTED VALUE (°C)	COMSOL VALUE (°C)	ERROR (°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 COMSOL VALUES AT 28 POINTS

(X,Y) COORDINATE (M)	EXPECTED VALUE (°C)	COMSOL VALUE (°C)	ERROR (°C)	(X,Y) COORDINATE (M)	EXPECTED VALUE (°C)	COMSOL VALUE (°C)	ERROR (°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	9.65	0.05
(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 error, 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 10211:2007)*, Appendix A, pp. 30–36, 2007.

Model Library path: `Heat_Transfer_Module/Verification_Models/thermal_bridge_2d_square_column`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click the **2D** button.

3 Click **Next**.

4 In the **Add physics** tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.

5 Click **Add Selected**.

6 Click **Next**.

7 Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.

8 Click **Finish**.

 ©2012 COMSOL

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
L	0.8[m]	Square side length

GEOMETRY 1

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

Rectangle 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type L/2.

4 In the **Height** edit field, type L.

5 Click the **Build All** button.

MATERIALS

The heat transfer equation for time-dependent problems also involves the density and the heat capacity. Because this model is stationary, you can ignore the warning about undefined properties.

Material 1

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

2 In the **Material** settings window, locate the **Material Contents** section.

3 In the table, enter the following settings:

PROPERTY	NAME	VALUE
Thermal conductivity	k	1

HEAT TRANSFER IN SOLIDS

Initial Values 1

1 In the **Model Builder** window, under **Model 1>Heat Transfer in Solids** click **Initial Values 1**.

2 In the **Initial Values** settings window, locate the **Initial Values** section.

3 In the T edit field, type `0[degC]`.

Temperature 1

1 In the **Model Builder** window, right-click **Heat Transfer in Solids** and choose the boundary condition **Temperature**.

2 Select Boundaries 1 and 2 only.

3 In the **Temperature** settings window, locate the **Temperature** section.

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

Temperature 2

1 In the **Model Builder** window, right-click **Heat Transfer in Solids** and choose the boundary condition **Temperature**.

2 Select Boundary 3 only.

3 In the **Temperature** settings window, locate the **Temperature** section.

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

Symmetry 1

1 In the **Model Builder** window, right-click **Heat Transfer in Solids** and choose the boundary condition **Symmetry**.

2 Select Boundary 4 only.

STUDY 1

1 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

The first default plot group shows the temperature distribution.

RESULTS

Temperature (ht)

Display the plot in degrees Celsius.

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

2 In the **Surface** settings window, locate the **Expression** section.

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

4 Click the **Plot** button.

Figure 3 shows the computed temperature distribution.

 ©2012 COMSOL

Export

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

1 In the **Model Builder** window, under **Results** right-click **Export** and choose **Data**.

2 In the **Data** settings window, locate the **Expressions** section.

3 In the table, enter the following settings:

EXPRESSION	UNIT	DESCRIPTION
T	degC	Temperature

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

5 Browse to a suitable folder and type the file name `thermal_bridge_2d_square_column.txt`.

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

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

8 In the **Start** edit field, type `L/8`.

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

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

11 Click the **Add** button.

12 In the **Data** settings window, locate the **Output** section.

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

14 In the **Start** edit field, type `L/8`.

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

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

17 Click the **Add** button.

18 Click the **Export** button.

 ©2012 COMSOL