

Thermal Bridges in Building Construction — 3D Iron Bar Through Insulation Layer

This model is licensed under the COMSOL Software License Agreement 6.1. All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

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 3D model (Case 4).

This tutorial studies the heat conduction in a thermal bridge made up of an iron bar and an insulation layer that separates a hot internal side from a cold external side. The iron bar is embedded in the insulation layer as shown in Figure 1. After solving the model, the heat flux between the internal and external sides and the maximum temperature on the external wall are calculated, and the results are compared to the expected values.



Figure 1: Back side (left) and front side (right) views of the iron bar embedded in an insulation layer, ISO 10211:2017 test case 4. Colored regions correspond to internal and external boundaries.

Model Definition

The geometry is illustrated above in Figure 1. The square insulation layer, with a low thermal conductivity k of 0.1 W/(m·K), has a cold and a hot surface. The iron bar has a higher thermal conductivity, 50 W/(m·K). Its boundaries are mainly located in the hot environment but one of them reaches the cold side.

Cold and hot surfaces are subject to convective heat flux. The ISO 10211:2017 standard specifies the values of the thermal resistance, R, which is related to the heat transfer coefficient, h, according to

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

Notes About the COMSOL Implementation

Compared to the rest of the structure, the dimensions of the intersection between the iron bar and the insulation layer are relatively small but the temperature gradients are large. Therefore, the element size is reduced in this area to give sufficient accuracy. To save computational time, this refinement is not applied on the remaining mesh.

Results and Discussion

In Figure 2, the temperature profile shows the effects of the thermal bridge where the heat variations are most pronounced.



Figure 2: Temperature distribution of ISO 10211:2017 test case 4.

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

| MEASURED QUANTITY | EXPECTED VALUE | COMPUTED VALUE | DIFFERENCE |
|--------------------------------------|-------------------|-------------------|-------------------------|
| Highest temperature on external side | 0.805°C | 0.8016°C | 3.4·10 ⁻³ °C |
| Heat flux | 0.540 W | 0.5415 W | 0.30% |

TABLE I: COMPARISON BETWEEN EXPECTED VALUES AND COMPUTED VALUES.

The maximum permissible differences to pass this test case are $5 \cdot 10^{-3}$ °C for temperature and 1% for heat flux. The measured values are completely coherent and meet the validation criteria.

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_3d_iron_bar

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click Solution Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 间 3D.
- 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 **M** Done.

GLOBAL DEFINITIONS

Define the geometrical parameters.

Parameters 1

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

| Name | Expression | Value | Description |
|------|------------|--------|-------------------------|
| w1 | 1[m] | lm | Insulation layer width |
| d1 | 0.2[m] | 0.2 m | Insulation layer depth |
| h1 | 1[m] | l m | Insulation layer height |
| w2 | 0.1[m] | 0.1 m | Iron bar width |
| d2 | 0.6[m] | 0.6 m | Iron bar depth |
| h2 | 50[mm] | 0.05 m | Iron bar height |

GEOMETRY I

Block I (blkI)

- I In the **Geometry** toolbar, click 🗍 **Block**.
- 2 In the Settings window for Block, locate the Size and Shape section.
- 3 In the Width text field, type w1.
- 4 In the **Depth** text field, type d1.
- 5 In the **Height** text field, type h1.
- 6 Click 틤 Build Selected.

Create the iron bar at the center of the insulation layer.

Block 2 (blk2)

- I In the **Geometry** toolbar, click 🗍 **Block**.
- 2 In the Settings window for Block, locate the Size and Shape section.
- 3 In the Width text field, type w2.
- 4 In the **Depth** text field, type d2.
- **5** In the **Height** text field, type h2.
- 6 Locate the **Position** section. In the x text field, type w1/2-w2/2.
- 7 In the z text field, type h1/2-h2/2.

- 8 Click 틤 Build Selected.
- 9 Click the 🔁 Wireframe Rendering button in the Graphics toolbar to get a better view of the interior parts.

To remove the unnecessary interior boundary in the iron bar, proceed as follows.

Ignore Faces 1 (igf1)

- I In the Geometry toolbar, click 🏷 Virtual Operations and choose Ignore Faces.
- 2 On the object fin, select Boundary 11 only.

Note that you can create the selection by clicking the **Paste Selection** button and typing the indices in the dialog box that opens.

- 3 In the Geometry toolbar, click 📗 Build All.
- **4** Click the \longleftrightarrow **Zoom Extents** button in the **Graphics** toolbar.



DEFINITIONS

Create a set of selections for use when setting up the physics. First, select the boundaries inside the region $y \ge 0.1$.

Internal

I In the **Definitions** toolbar, click here is a second sec

- 2 In the Settings window for Box, type Internal in the Label text field.
- 3 Locate the Geometric Entity Level section. From the Level list, choose Boundary.
- 4 Locate the Box Limits section. In the y minimum text field, type 0.1.
- 5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Next, select all the boundaries inside the region $y \le 0.1$.

External

- I In the **Definitions** toolbar, click **here Box**.
- 2 In the Settings window for Box, type External in the Label text field.
- 3 Locate the Geometric Entity Level section. From the Level list, choose Boundary.
- **4** Locate the **Box Limits** section. In the **y maximum** text field, type **0.1**.
- **5** Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.
- 6 Click the 🔁 Wireframe Rendering button in the Graphics toolbar.

MATERIALS

Insulation

- I In the Materials toolbar, click 🚦 Blank Material.
- 2 In the Settings window for Material, type Insulation 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 | 0.1 | W/(m·K) | Basic |
| Density | rho | 500 | kg/m³ | Basic |
| Heat capacity at constant pressure | Ср | 1700 | J/(kg·K) | Basic |

Iron

- I In the Materials toolbar, click 🚦 Blank Material.
- 2 In the Settings window for Material, type Iron 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 | 50 | W/(m·K) | Basic |
| Density | rho | 7800 | kg/m³ | Basic |
| Heat capacity at constant pressure | Ср | 460 | J/(kg·K) | Basic |

HEAT TRANSFER IN SOLIDS (HT)

Heat Flux 1

- I In the Model Builder window, under Component I (compl) right-click Heat Transfer in Solids (ht) and choose Heat Flux.
- 2 In the Settings window for Heat Flux, locate the Boundary Selection section.
- 3 From the Selection list, choose Internal.
- 4 Locate the Heat Flux section. From the Flux type list, choose Convective heat flux.
- **5** In the *h* text field, type 1/0.10.
- **6** In the T_{ext} text field, type 1[degC].

Heat Flux 2

- I 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 External.
- 4 Locate the Heat Flux section. From the Flux type list, choose Convective heat flux.
- **5** In the *h* text field, type 1/0.10.
- **6** In the T_{ext} text field, type O[degC].

MESH I

Because the largest temperature variations are expected at the exterior boundary of the iron bar, refine the mesh in this region. Use the default settings in the remaining domains.

Free Tetrahedral I

In the Mesh toolbar, click \land Free Tetrahedral.

Size I

I In the Mesh toolbar, click Size Attribute and choose Extremely Fine.

- 2 In the Settings window for Size, locate the Geometric Entity Selection section.
- **3** From the **Geometric entity level** list, choose **Domain**.
- **4** Select Domain 2 only.
- 5 Click 📗 Build All.

STUDY I

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

RESULTS

Surface

The first default plot group shows the temperature distribution; compare with Figure 2.

- I In the Model Builder window, expand the Results>Temperature (ht) node, then click Surface.
- 2 In the Settings window for Surface, locate the Expression section.
- 3 From the Unit list, choose degC.
- **4** Click the 4 **Zoom Extents** button in the **Graphics** toolbar.

Follow the steps below to calculate the maximum temperature on the external surface and the heat flux between the internal and external sides. Compare the values with the expected results listed in Table 1.

Surface Maximum I

- I In the Results toolbar, click ^{8,85}_{e-12} More Derived Values and choose Maximum> Surface Maximum.
- 2 In the Settings window for Surface Maximum, locate the Selection section.
- 3 From the Selection list, choose External.
- 4 Locate the **Expressions** section. In the table, enter the following settings:

| Expression | Unit | Description |
|------------|------|-------------|
| Т | degC | Temperature |

5 Click **= Evaluate**.

TABLE

I Go to the **Table** window.

The displayed value should be close to 0.805°C.

RESULTS

Surface Integration 1

- I In the Results toolbar, click ^{8,85}_{e-12} More Derived Values and choose Integration> Surface Integration.
- 2 In the Settings window for Surface Integration, locate the Selection section.
- 3 From the Selection list, choose External.
- 4 Click Replace Expression in the upper-right corner of the Expressions section. From the menu, choose Component I (compl)>Heat Transfer in Solids>Boundary fluxes>ht.q0 Inward heat flux W/m².
- 5 Click **= Evaluate**.

TABLE

I Go to the Table window.

The measured flux should be close to 0.540 W.