Created in COMSOL Multiphysics 6.1

# Thermal Bridges in Building Construction — 3D Structure Between Two Floors

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 and is hence classified as a *three-dimensional steady-state high precision method*. This document presents an implementation of the first 3D model (Case 3).

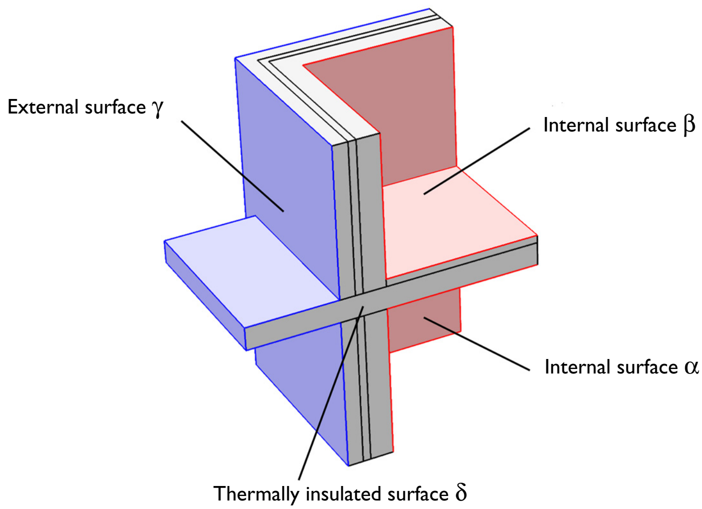



*Figure 1: Geometry and boundary conditions of ISO 10211:2017 test case 3.*

This tutorial studies the heat conduction in a building structure separating two floors from the external environment. The structure's surfaces are divided into four parts:

- the lower level, α;
- the upper level, β;
- the outside, γ;
- and the remaining thermally insulated surfaces, δ.

The values of interest for validation are the lowest temperatures at surfaces α and β, and the heat fluxes through α, β, and γ.

## Model Definition

Figure 1 illustrates the geometry. The external surface is at 0°C and the interior surface is at 20°C. Four materials with distinct thermal conductivities $k$ are used in the structure. The horizontal block separating the two floors has the highest thermal conductivity (2.5 W/(m·K)). It crosses the wall, thereby creating a thermal bridge in the structure.

The surfaces α, β, and γ 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

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

## Results and Discussion

Figure 2 shows the temperature profile. The heat losses are greater near the thermal bridge formed by the horizontal block that crosses the wall.

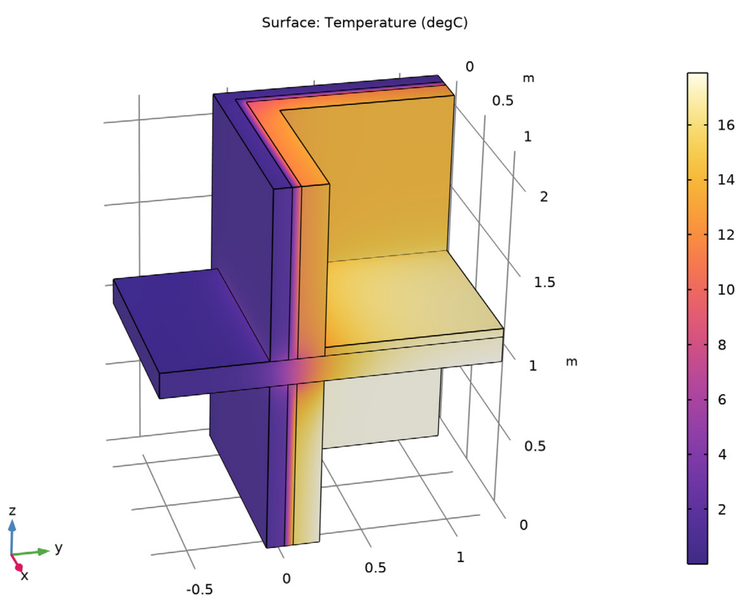



*Figure 2: Temperature distribution of ISO 10211:2017 test case 3.*

The numerical results of COMSOL Multiphysics are compared with the expected values provided by ISO 10211:2017 (Ref. 1) in Table 1.

TABLE 1: COMPARISON BETWEEN EXPECTED VALUES AND COMPUTED VALUES.

| MEASURED QUANTITY | EXPECTED VALUES | COMPUTED VALUES | DIFFERENCE |
|---|---|---|---|
| Minimum temperature on α | 11.32°C | 11.32°C | <0.01°C |
| Minimum temperature on β | 11.11°C | 11.10°C | 0.01°C |
| Heat flux through α | 46.09 W | 46.19 W | 0.22% |
| Heat flux through β | 13.89 W | 13.92 W | 0.25% |
| Heat flux through γ | 59.98 W | 60.12 W | 0.23% |

The maximum permissible differences to pass this test case are 0.1°C for temperature and 1% for heat flux. The measured values are completely consistent and meet the validation criteria. Note that they may change slightly depending on geometry representation and operating system.

As shown in Figure 3 and Figure 4, the minimum temperature of the surfaces α and β are located at their respective corners.

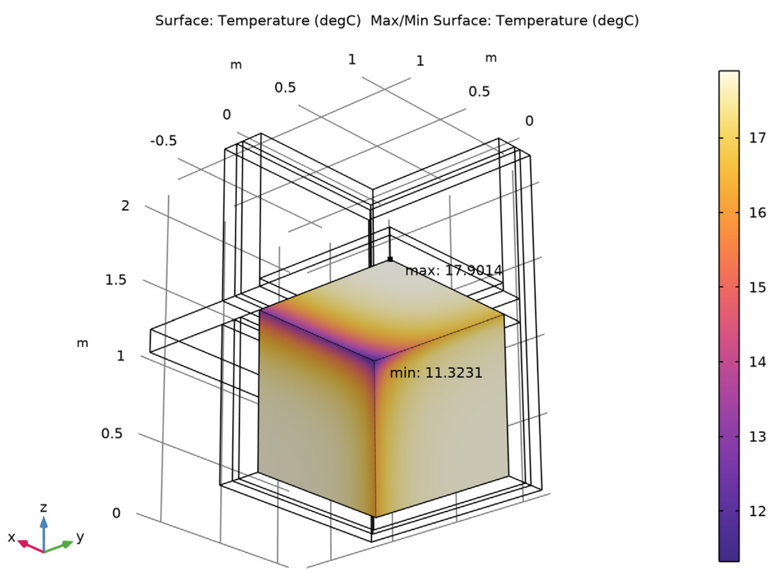



*Figure 3: Minimum and maximum temperatures on surface α, ISO 10211:2017 test case 3.*

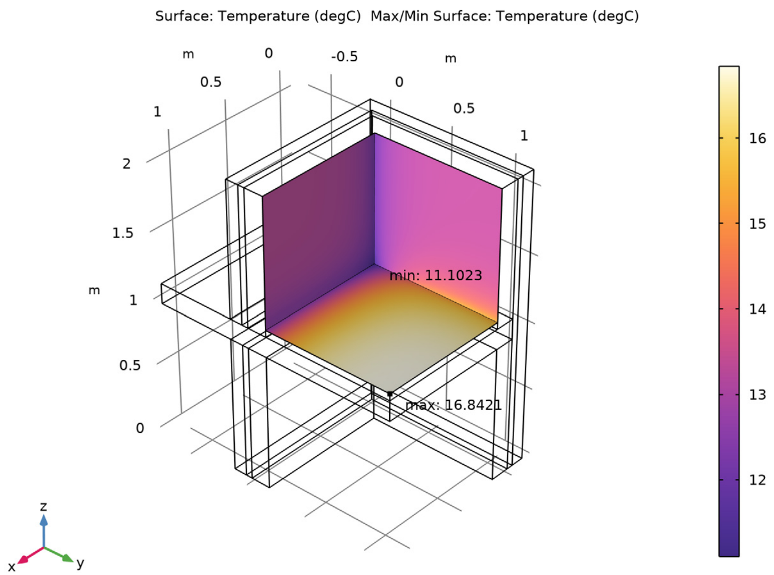



*Figure 4: Minimum and maximum temperatures on surface* β, *ISO 10211:2017 test case 3.*

## *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_two_floors`

## *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 **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 **Done**.

## GLOBAL DEFINITIONS

Define the geometrical parameters.

### *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 |
|---|---|---|---|
| wall_t1 | 5[cm] | 0.05 m | Insulation layer thickness |
| wall_t2 | 30[cm] | 0.3 m | Wall thickness |
| rect1 | 1.2[m] | 1.2 m | Wall first rectangle basis |
| rect2 | 1.3[m] | 1.3 m | Wall second rectangle basis |
| rect_shift | -10[cm] | -0.1 m | Shift for the rectangles |
| wall_h | 2.15[m] | 2.15 m | Wall height |
| blk_w | 1.15[m] | 1.15 m | Rectangle horizontal block width |
| blk_d | 1.9[m] | 1.9 m | Rectangle horizontal block depth |
| blk_h | 0.15[m] | 0.15 m | Rectangle horizontal block height |
| blk_shiftx | 5[cm] | 0.05 m | Rectangle horizontal block x-shift |
| blk_shifty | -0.7[m] | -0.7 m | Rectangle horizontal block y-shift |
| blk_shiftz | 1[m] | 1 m | Rectangle horizontal block z-shift |
| sq_l | 1[m] | 1 m | Square horizontal block side length |

| Name | Expression | Value | Description |
|---|---|---|---|
| sq_h | 5[cm] | 0.05 m | Square horizontal block height |
| sq_shift | 0.2[m] | 0.2 m | Square horizontal block shift |

## GEOMETRY 1

To build the walls separating the external and internal surfaces, create the cross section geometry and extrude it.

### *Work Plane 1 (wp1)*

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

### *Work Plane 1 (wp1)>Plane Geometry*

1 In the **Model Builder** window, click **Plane Geometry**.

The first two rectangles below correspond to the insulation layer of the wall.

### *Work Plane 1 (wp1)>Rectangle 1 (r1)*

1 In the **Work Plane** toolbar, click **Rectangle**.

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

3 In the **Width** text field, type `rect1`.

4 In the **Height** text field, type `wall_t1`.

5 Click **Build Selected**.

### *Work Plane 1 (wp1)>Rectangle 2 (r2)*

1 In the **Work Plane** toolbar, click **Rectangle**.

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

3 In the **Width** text field, type `wall_t1`.

4 In the **Height** text field, type `rect1`.

5 Click **Build Selected**.

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

### *Work Plane 1 (wp1)>Union 1 (uni1)*

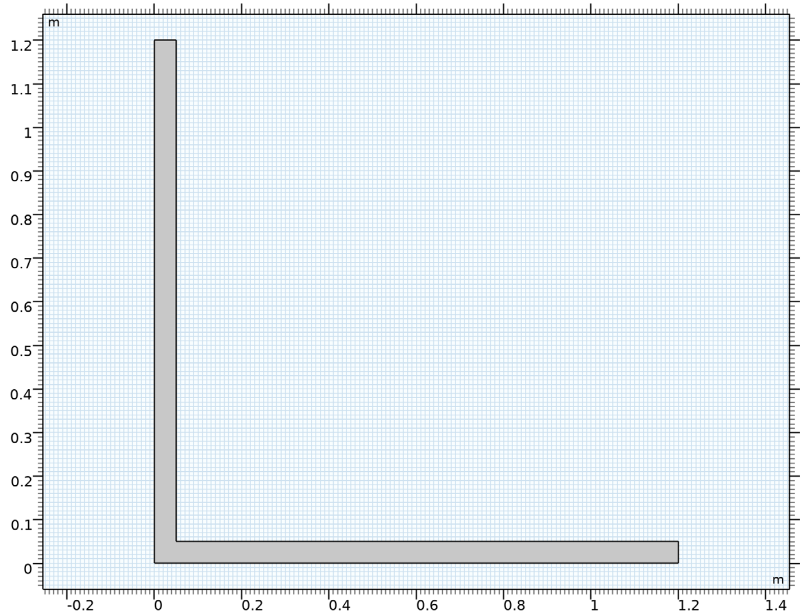
1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

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

4 Clear the **Keep interior boundaries** check box.

**5** Click **Build Selected**.




*Work Plane 1 (wp1)>Plane Geometry*

Build the remaining layers of the walls in a similar manner.

*Work Plane 1 (wp1)>Rectangle 3 (r3)*

**1** In the **Work Plane** toolbar, click **Rectangle**.

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

**3** In the **Width** text field, type `rect2`.

**4** In the **Height** text field, type `wall_t2`.

**5** Locate the **Position** section. In the **xw** text field, type `rect_shift`.

**6** In the **yw** text field, type `rect_shift`.

**7** Click **Build Selected**.

*Work Plane 1 (wp1)>Rectangle 4 (r4)*

**1** In the **Work Plane** toolbar, click **Rectangle**.

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

**3** In the **Width** text field, type `wall_t2`.

**4** In the **Height** text field, type `rect2`.

5 Locate the **Position** section. In the **xw** text field, type `rect_shift`.

6 In the **yw** text field, type `rect_shift`.

7 Click **Build Selected**.

*Work Plane 1 (wp1)>Union 2 (uni2)*

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

2 Select the objects **r3** and **r4** only.

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

4 Clear the **Keep interior boundaries** check box.

5 Click **Build Selected**.

*Extrude 1 (ext1)*

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

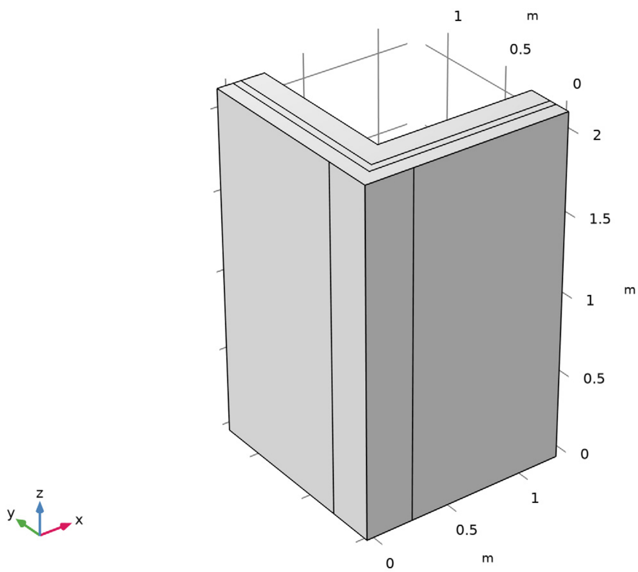
2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

| **Distances (m)** |
|---|
| `wall_h` |

4 Click **Build Selected**.

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




*Block 1 (blk1)*

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

This block separates the two floors of the structure.

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

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

4 In the **Depth** text field, type `blk_d`.

5 In the **Height** text field, type `blk_h`.

6 Locate the **Position** section. In the **x** text field, type `blk_shiftx`.

7 In the **y** text field, type `blk_shifty`.

8 In the **z** text field, type `blk_shiftz`.

9 Click **Build Selected**.

To avoid unnecessary edges, remove the intersection of the block with the walls. To do so, use the boolean operation **Difference** to subtract a copy of the block from the walls. Begin by creating the copy.

*Block 2 (blk2)*

Right-click **Block 1 (blk1)** and choose **Duplicate**.

*Difference 1 (dif1)*

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

2 Select the object **ext1** only.

The object labeled `ext1` is made up of the walls previously obtained by extrusion.

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

4 Find the **Objects to subtract** subsection. Click to select the **Activate Selection** toggle button.

5 Select the object **blk1** only.

6 Click **Build Selected**.

*Block 3 (blk3)*

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

This block corresponds to the floor of the inside upper level.

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

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

4 In the **Depth** text field, type `sq_l`.

5 In the **Height** text field, type `sq_h`.

6 Locate the **Position** section. In the **x** text field, type `sq_shift`.

7 In the **y** text field, type `sq_shift`.

8 In the **z** text field, type `blk_shiftz+blk_h`.

9 Click **Build Selected**.

*Ignore Edges 1 (ige1)*

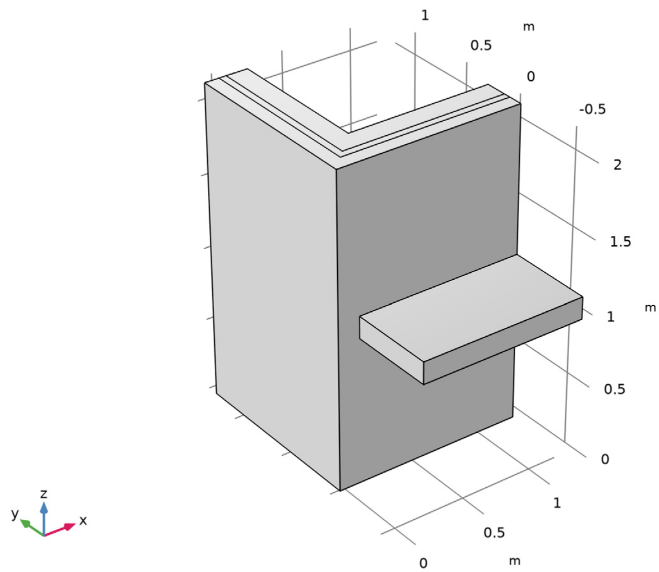
1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Edges**.

In the first steps of the geometry sequence, six unused vertical edges were created on the walls. They are responsible for unnecessary constraints on the mesh and they generate extra boundaries by splitting some faces. For these reasons, follow the instructions below to remove them.

2 On the object **fin**, select Edges 6, 17, 33, 38, 60, and 63 only.

3 To reach the edges, click the **Wireframe Rendering** button in the **Graphics** toolbar. Note that you can make the selection by clicking the **Paste Selection** button and typing the indices in the dialog box that opens.

**4** In the **Settings** window for **Ignore Edges**, click **Build Selected**.




## MATERIALS

### *Interior Wall*

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

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

**3** Select Domains 4 and 5 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 | 0.7 | W/(m·K) | Basic |
| Density | rho | 1700 | kg/m³ | Basic |
| Heat capacity at constant pressure | Cp | 800 | J/(kg·K) | Basic |

### *Isolation*

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

**2** In the **Settings** window for **Material**, type `Isolation` 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 | 0.04 | W/(m·K) | Basic |
| Density | rho | 200 | kg/m³ | Basic |
| Heat capacity at constant pressure | Cp | 1000 | J/(kg·K) | Basic |

*Exterior Wall*

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

**2** In the **Settings** window for **Material**, type `Exterior Wall` 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 | 1 | W/(m·K) | Basic |
| Density | rho | 2000 | kg/m³ | Basic |
| Heat capacity at constant pressure | Cp | 1000 | J/(kg·K) | Basic |

*Horizontal Structure*

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

**2** In the **Settings** window for **Material**, type `Horizontal Structure` 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 | 2.5 | W/(m·K) | Basic |
| Density | rho | 5000 | kg/m³ | Basic |
| Heat capacity at constant pressure | Cp | 600 | J/(kg·K) | Basic |

### *Floor*

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

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

3 Select Domain 6 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 | W/(m·K) | Basic |
| Density | rho | 1000 | kg/m³ | Basic |
| Heat capacity at constant pressure | Cp | 800 | J/(kg·K) | Basic |

## HEAT TRANSFER IN SOLIDS (HT)

### *Heat Flux 1*

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Heat Transfer in Solids (ht)** and choose **Heat Flux**.

2 Select Boundaries 33–35 only.

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

4 Click **Create Selection**.

5 In the **Create Selection** dialog box, type `Alpha` in the **Selection name** text field.

6 Click **OK**.

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

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

9 In the $h$ text field, type `1/0.2`.

10 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 Boundaries 39–41 only.

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

4 Click **Create Selection**.

5 In the **Create Selection** dialog box, type `Beta` in the **Selection name** text field.

6 Click **OK**.

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

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

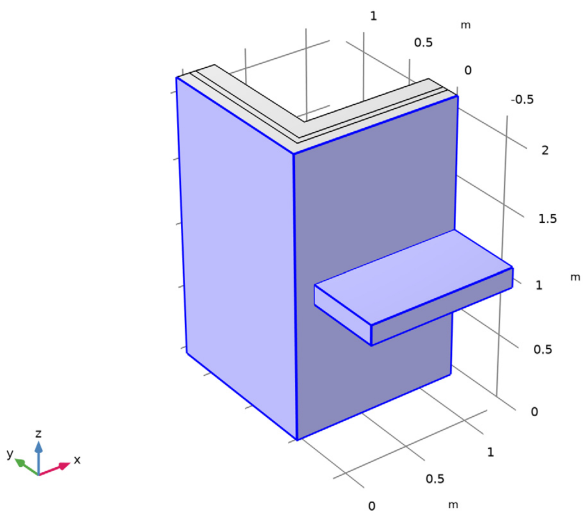
**9** In the $h$ text field, type `1/0.2`.

**10** In the $T_{ext}$ text field, type `15[degC]`.

*Heat Flux 3*

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

**2** Select Boundaries 1, 2, and 11–14 only.




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

**4** Click **Create Selection**.

**5** In the **Create Selection** dialog box, type `Gamma` in the **Selection name** text field.

**6** Click **OK**.

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

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

**9** In the $h$ text field, type `1/0.05`.

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

## STUDY 1

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

## RESULTS

### *Temperature (ht)*

The current plot group shows the temperature distribution; compare with .

### *Surface*

1. In the **Model Builder** window, expand the **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. In the **Temperature (ht)** toolbar, click **Plot**.

   Follow the steps below to find the minimum temperatures on α and β as well as the heat fluxes through α, β, and γ.

### *Surface Minimum 1*

1. In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Minimum> Surface Minimum**.
2. In the **Settings** window for **Surface Minimum**, locate the **Selection** section.
3. From the **Selection** list, choose **Alpha**.
4. Locate the **Expressions** section. In the table, enter the following settings:

| Expression | Unit | Description |
| --- | --- | --- |
| T | degC | Temperature |

5. Click **Evaluate**.

   The displayed value should be close to 11.3°C.

### *Surface Minimum 2*

1. In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Minimum> Surface Minimum**.
2. In the **Settings** window for **Surface Minimum**, locate the **Selection** section.
3. From the **Selection** list, choose **Beta**.
4. Locate the **Expressions** section. In the table, enter the following settings:

| Expression | Unit | Description |
| --- | --- | --- |
| T | degC | Temperature |

5. Click **Evaluate**.

   The displayed value should be close to 11.1°C.

*Surface Integration 1*

1 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 **Alpha**.

4 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**.

5 Click **Evaluate**.

The displayed value should be close to 46.2 W.

*Surface Integration 2*

1 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 **Beta**.

4 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**.

5 Click **Evaluate**.

The displayed value should be close to 13.9 W.

*Surface Integration 3*

1 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 **Gamma**.

4 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**.

5 Click **Evaluate**.

**TABLE**

1 Go to the **Table** window.

The displayed value should be close to 60.1 W.

## RESULTS

*Temperature (ht)*

To plot the location of the minimum temperature on α, follow the instructions below.

The resulting dataset is restricted to the surfaces α.

*Minimum Temperature on Alpha*

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Results** and choose **3D Plot Group**.

3 In the **Settings** window for **3D Plot Group**, type `Minimum Temperature on Alpha` in the **Label** text field.

*Surface 1*

1 In the **Minimum Temperature on Alpha** toolbar, click **Surface**.

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

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

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Thermal>HeatCameraLight** in the tree.

6 Click **OK**.

*Selection 1*

1 In the **Minimum Temperature on Alpha** toolbar, click **Selection**.

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

3 From the **Selection** list, choose **Alpha**.

*Minimum Temperature on Alpha*

In the **Model Builder** window, under **Results** click **Minimum Temperature on Alpha**.

*Max/Min Surface 1*

1 In the **Minimum Temperature on Alpha** toolbar, click **More Plots** and choose **Max/Min Surface**.

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

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

*Selection 1*

1 In the **Minimum Temperature on Alpha** toolbar, click **Selection**.

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

3 From the **Selection** list, choose **Alpha**.

**4** In the **Minimum Temperature on Alpha** toolbar, click **Plot**.

As shown in Figure 3, the minimum temperature is at the corner of α.

Now plot the location of the minimum temperature on β.

*Minimum Temperature on Beta*

**1** Right-click **Selection 1** and choose **Duplicate**.

**2** In the **Settings** window for **3D Plot Group**, type `Minimum Temperature on Beta` in the **Label** text field.

**3** In the **Model Builder** window, expand the **Minimum Temperature on Beta** node.

*Selection 1*

**1** In the **Model Builder** window, expand the **Results>Minimum Temperature on Beta>Surface 1** node, then click **Selection 1**.

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

**3** From the **Selection** list, choose **Beta**.

*Selection 1*

**1** In the **Model Builder** window, expand the **Results>Minimum Temperature on Beta>Max/Min Surface 1** node, then click **Selection 1**.

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

**3** From the **Selection** list, choose **Beta**.

**4** In the **Minimum Temperature on Beta** toolbar, click **Plot**.