Created in COMSOL Multiphysics 6.1

Out-of-Plane Heat Transfer for a Thin Plate

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

Modeling heat transfer in a thin rectangular metal plate, this example demonstrates how to use the **Out-of-Plane Heat Flux** and **Out-of-Plane Radiation** features in the Heat Transfer interfaces. Because the plate thickness is only 1/100th of its length and width, it is appropriate to simulate the process using a 2D approximation where the temperature is assumed to be constant along the thickness. The plate has a fixed temperature at one end and is isolated at the other end. A surrounding liquid cools the plate by convection. In addition, the model considers surface-to-ambient radiation. To check the validity of the 2D approximation, the example finishes by setting up a full 3D model and comparing the results of the two versions.

Model Definition

The plate has a square shape of side length 1 m while its thickness is only 10 mm. Figure 1 illustrates the simplification made from 3D to 2D.

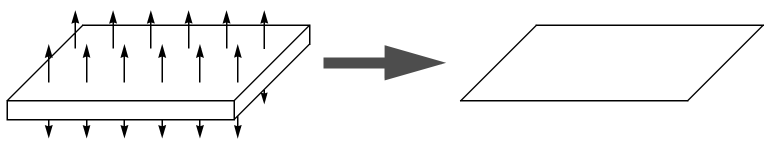

Figure 1: Model geometry reduction from 3D to 2D.

The stationary heat transfer equation for the 2D problem is

$$\begin{aligned}\nabla\cdot(-d_z k\nabla T) &= h_{\mathrm{u}}(T_{\mathrm{ext,\,u}}^4 - T^4) + h_{\mathrm{d}}(T_{\mathrm{ext,\,d}}^4 - T^4)\\ &\quad + \varepsilon_{\mathrm{u}}\sigma(T_{\mathrm{amb,\,u}}^4 - T^4) + \varepsilon_{\mathrm{d}}\sigma(T_{\mathrm{amb,\,d}}^4 - T^4)\end{aligned}$$

In this case the problem is symmetric, that is, the upside constants are equal to the downside ones. The heat transfer film coefficient, h=h_{u}=h_{d}, is equal to 10 W/(m^2·K) and the emissivity, ε=ε_{u}=ε_{d}, is equal to 0.5. The ambient and external temperatures, T_{amb}=$T_{\mathrm{amb,u}}$=$T_{\mathrm{amb,d}}$ and T_{ext}=$T_{\mathrm{ext,u}}$=$T_{\mathrm{ext,d}}$, are both set to 300 K. The model uses the default material properties for copper from the built-in material library. The variable d_z denotes the plate thickness, and σ is the Stefan-Boltzmann constant.

Finally, set the left boundary to a fixed temperature of 800 K and assume that the other boundaries are thermally insulated.

Results and Discussion

Figure 2 shows the steady-state temperature distribution.

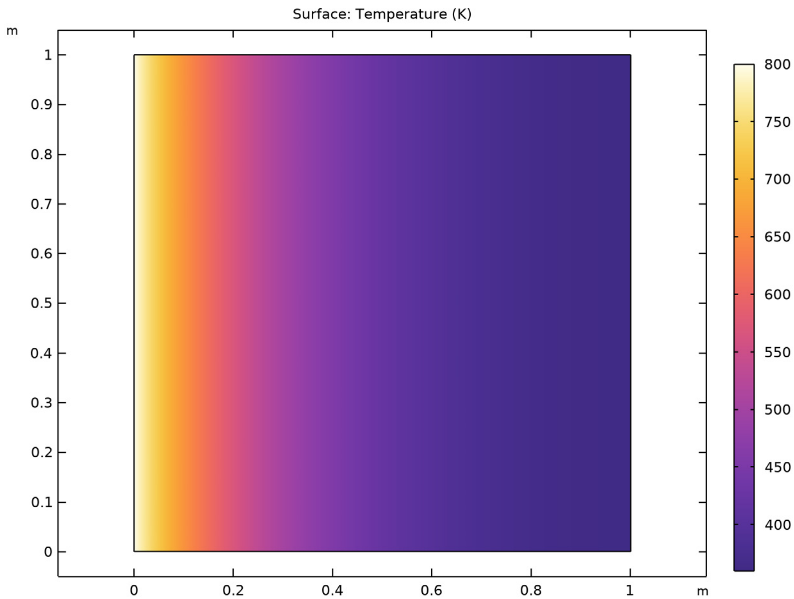

Figure 2: Stationary temperature distribution in the plate.

[Figure 3](#) shows the temperature profile in the full 3D case.

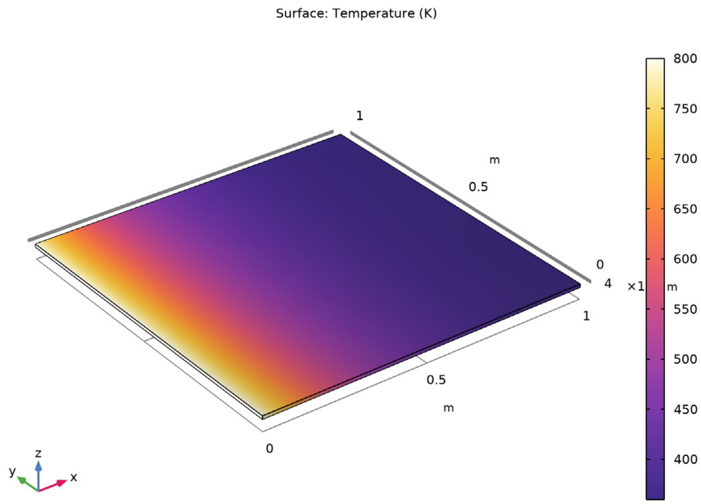

Figure 3: Temperature profile in the full 3D case.

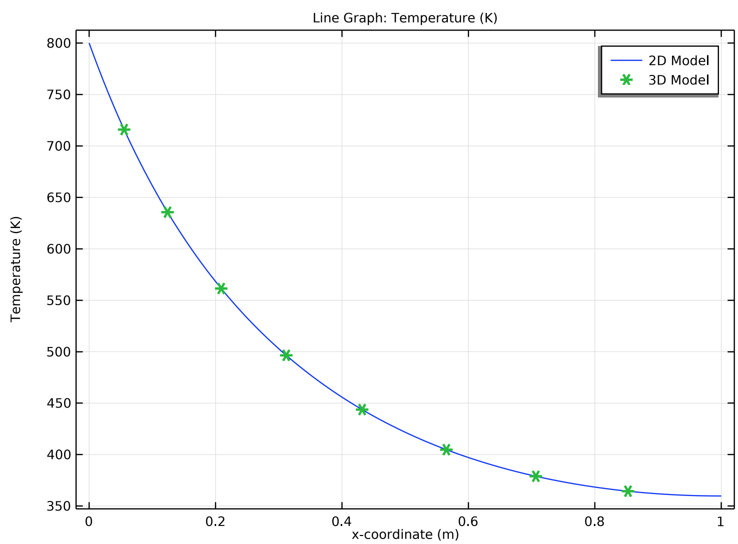

Figure 4: Comparison of the temperature (from left to right in the plate) from both the 2D out-of-plane model (blue) and the full 3D model (green stars). The curves coincide to a very high degree, making it difficult to tell them apart.

It is interesting to compare the results from the 2D approximation with the equivalent 3D model. Figure 4 shows the temperature along the bottom edge of the 2D model and the corresponding 3D result.

Application Library path: `Heat_Transfer_Module/Verification_Examples/thin_plate`

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 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `thin_plate_parameters.txt`.

GEOMETRY 1

Square 1 (sq1)

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

2 In the **Settings** window for **Square**, locate the **Size** section.

3 In the **Side length** text field, type `length`.

4 Click **Build All Objects**.

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

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in>Copper**.

4 Click **Add to Component** in the window toolbar.

5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

HEAT TRANSFER IN SOLIDS (HT)

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

2 In the **Settings** window for **Heat Transfer in Solids**, locate the **Physical Model** section.

3 In the d_z text field, type `thickness`.

Out-of-Plane Heat Flux 1

1 In the **Physics** toolbar, click **Domains** and choose **Out-of-Plane Heat Flux**.

2 Select Domain 1 only.

3 In the **Settings** window for **Out-of-Plane Heat Flux**, locate the **Upside Inward Heat Flux** section.

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

5 In the h_{u} text field, type `h_c`.

6 In the $T_{\mathrm{ext,u}}$ text field, type `T_ext`.

7 Locate the **Downside Inward Heat Flux** section. From the **Flux type** list, choose **Convective heat flux**.

8 In the h_{d} text field, type `h_c`.

9 In the $T_{\mathrm{ext,d}}$ text field, type `T_ext`.

Out-of-Plane Radiation 1

1 In the **Physics** toolbar, click **Domains** and choose **Out-of-Plane Radiation**.

2 Select Domain 1 only.

3 In the **Settings** window for **Out-of-Plane Radiation**, locate the **Upside Parameters** section.

4 From the ε_{u} list, choose **User defined**. In the associated text field, type `0.5`.

5 In the $T_{\mathrm{amb,u}}$ text field, type `T_ext`.

6 Locate the **Downside Parameters** section. From the ε_{d} list, choose **User defined**. In the associated text field, type `0.5`.

7 In the $T_{\mathrm{amb,d}}$ text field, type `T_ext`.

Temperature 1

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

2 Select Boundary 1 only.

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

4 In the T_0 text field, type `T_hot`.

STUDY 1

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

RESULTS

Temperature (ht)

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

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

Now add a second model for the full 3D problem.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component>3D**.

ADD PHYSICS

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1**.

5 Click **Add to Component 2** in the window toolbar.

6 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Heat Transfer in Solids (ht)**.

5 Click **Add Study** in the window toolbar.

6 In the **Model Builder** window, click the root node.

7 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

GEOMETRY 2

In the **Model Builder** window, under **Component 2 (comp2)** click **Geometry 2**.

Block 1 (blk1)

1 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 `length`.

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

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

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in>Copper**.

4 Click **Add to Component** in the window toolbar.

5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

HEAT TRANSFER IN SOLIDS 2 (HT2)

In the **Model Builder** window, under **Component 2 (comp2)** click **Heat Transfer in Solids 2 (ht2)**.

Heat Flux 1

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

2 Select Boundaries 3 and 4 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 `h_c`.

6 In the T_{ext} text field, type `T_ext`.

Surface-to-Ambient Radiation 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Surface-to-Ambient Radiation**.
2. Select Boundaries 3 and 4 only.
3. In the **Settings** window for **Surface-to-Ambient Radiation**, locate the **Surface-to-Ambient Radiation** section.
4. From the ε list, choose **User defined**. In the associated text field, type `0.5`.
5. In the T_{amb} text field, type `T_ext`.

Temperature 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.
2. Select Boundary 1 only.
3. In the **Settings** window for **Temperature**, locate the **Temperature** section.
4. In the T_0 text field, type `T_hot`.

STUDY 2

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

RESULTS

Temperature (ht2)

The default plots show the temperature distribution of the full 3D problem (compare with Figure 3) and the isothermal contours.

To compare the temperature profiles along the bottom edge for the 2D approximation and 3D model, as done in Figure 4, follow the steps given below.

Models Comparison

1. In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Models Comparison` in the **Label** text field.

Line Graph 1

1. Right-click **Models Comparison** and choose **Line Graph**.
2. Select Boundary 2 only.
3. In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Geometry>Coordinate>x - x-coordinate**.
4. Click to expand the **Legends** section. Select the **Show legends** check box.

5 From the **Legends** list, choose **Manual**.

6 In the table, enter the following settings:

Legends
`2D Model`

7 In the **Models Comparison** toolbar, click **Plot**.

Models Comparison

In the **Model Builder** window, click **Models Comparison**.

Line Graph 2

1 In the **Models Comparison** toolbar, click **Line Graph**.

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

3 From the **Dataset** list, choose **Study 2/Solution 2 (3) (sol2)**.

4 Select Edge 3 only.

5 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 2 (comp2)>Heat Transfer in Solids 2>Temperature>T2 - Temperature - K**.

6 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 2 (comp2)>Geometry>Coordinate>x - x-coordinate**.

7 Click to expand the **Title** section. From the **Title type** list, choose **None**.

8 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

9 Find the **Line markers** subsection. From the **Marker** list, choose **Asterisk**.

10 From the **Positioning** list, choose **Interpolated**.

11 Locate the **Legends** section. Select the **Show legends** check box.

12 From the **Legends** list, choose **Manual**.

13 In the table, enter the following settings:

Legends
`3D Model`

14 In the **Models Comparison** toolbar, click **Plot**.