



Model created in COMSOL Multiphysics 6.4

Out-of-Plane Heat Transfer for a Thin Plate

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.

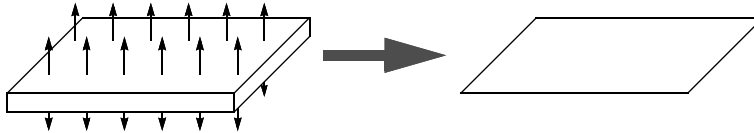


Figure 1: Model geometry reduction from 3D to 2D.

The stationary heat transfer equation for the 2D problem is

$$\begin{aligned} \nabla \cdot (-d_z k \nabla T) &= h_u (T_{\text{ext}, u}^4 - T^4) + h_d (T_{\text{ext}, d}^4 - T^4) \\ &+ \varepsilon_u \sigma (T_{\text{amb}, u}^4 - T^4) + \varepsilon_d \sigma (T_{\text{amb}, d}^4 - T^4) \end{aligned}$$

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 $10 \text{ W}/(\text{m}^2 \cdot \text{K})$ and the emissivity, $\varepsilon=\varepsilon_u=\varepsilon_d$, is equal to 0.5. The ambient and external temperatures, $T_{\text{amb}}=T_{\text{amb},u}=T_{\text{amb},d}$ and $T_{\text{ext}}=T_{\text{ext},u}=T_{\text{ext},d}$, 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.

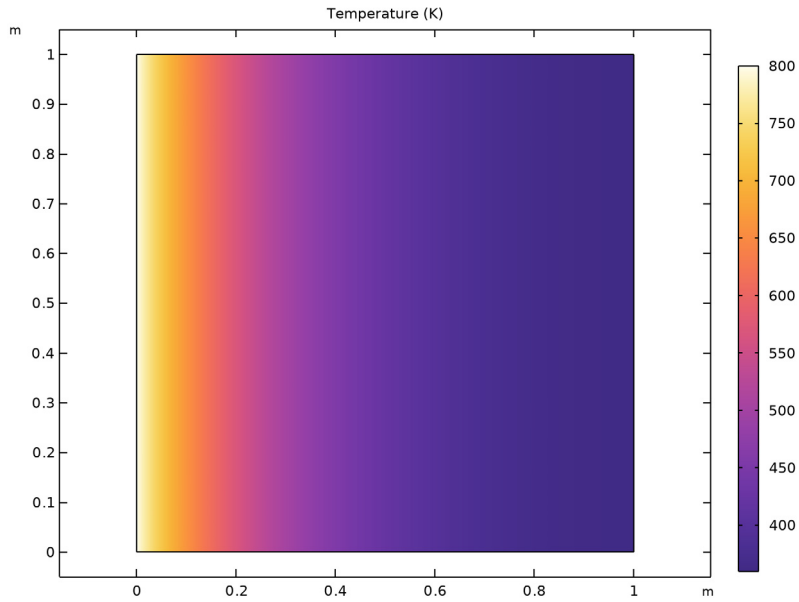


Figure 2: Stationary temperature distribution in the plate.

Figure 3 shows the temperature profile in the full 3D case.

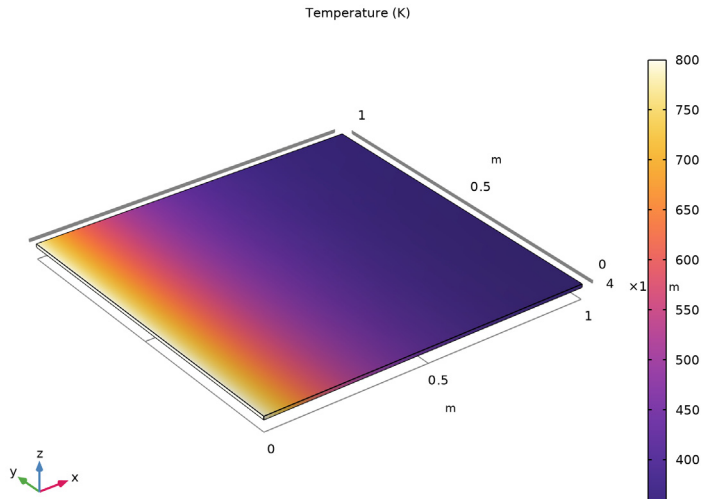


Figure 3: Temperature profile in the full 3D case.

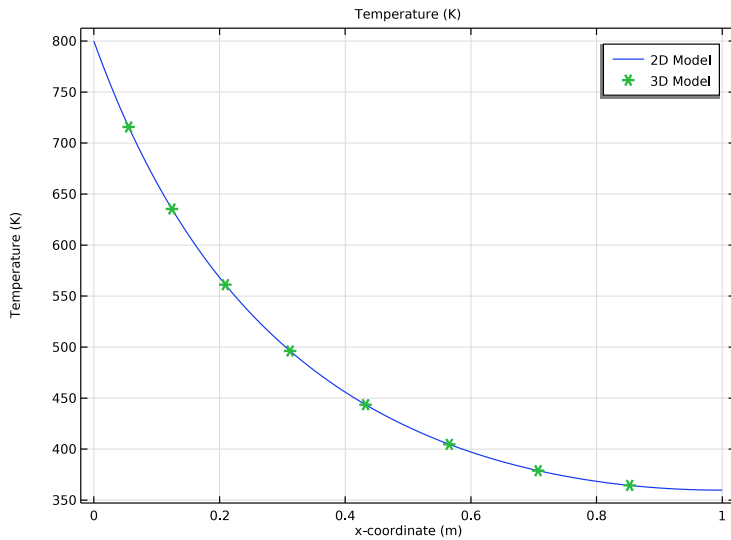


Figure 4: Comparison of the temperature (from left to right in the plate) from both the 2D out-of-plane model (blue) and the full 3D model (green stars). The curves coincide to a very high degree, making it difficult to tell them apart.


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 (sql)

- 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 **Materials** 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 the **Add to Component** button in the window toolbar.
- 5 In the **Materials** toolbar, click  **Add Material** to close the **Add Material** window.


HEAT TRANSFER IN SOLIDS (HT)

- 1 In the **Settings** window for **Heat Transfer in Solids**, locate the **Physical Model** section.
- 2 In the d_z text field, type thickness.

Out-of-Plane Heat Flux I


- 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 $T_{ext,u}$ 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 $T_{ext,d}$ text field, type T_{ext} .

Out-of-Plane Radiation I


- 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 $T_{amb,u}$ 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 $T_{amb,d}$ text field, type T_ext.

Temperature I

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


STUDY I

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

RESULTS

Temperature (ht)

The 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** checkbox for **Study I**.
- 5 Click the **Add to Component 2** button 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** checkbox for **Heat Transfer in Solids (ht)**.
- 5 Click the **Add Study** button in the window toolbar.
- 6 In the **Home** toolbar, click  **Add Study** to close the **Add Study** window.

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 **Materials** 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 the **Add to Component** button in the window toolbar.
- 5 In the **Materials** toolbar, click  **Add Material** to close the **Add Material** window.

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 **Study** toolbar, click  **Compute**.


RESULTS

Temperature (ht2)

The default plot shows the temperature distribution of the full 3D problem (compare with [Figure 3](#)).

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 **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID 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** checkbox.
- 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** checkbox.
- 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**.