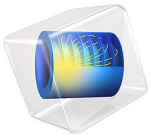


Model created in COMSOL Multiphysics 6.4

Silica Glass Block Coated with a Copper Layer

Introduction

This application demonstrates how to use the Thin Layer feature of the Heat Transfer interface.

Model Definition

This example constructs a 2D time-dependent model of a silica glass block that is coated with a thin copper layer. Figure 1 shows the model geometry and boundary conditions.

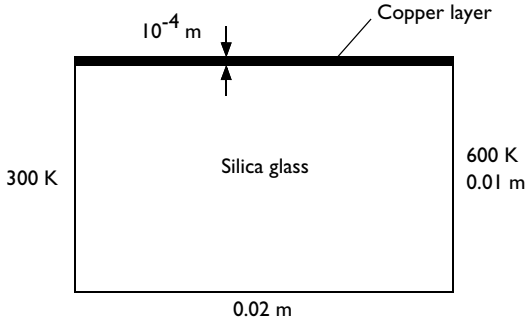


Figure 1: Model geometry for a silica block with a copper layer.

The model sets the initial temperature to 300 K. The following table shows the thermal properties for silica glass and copper:

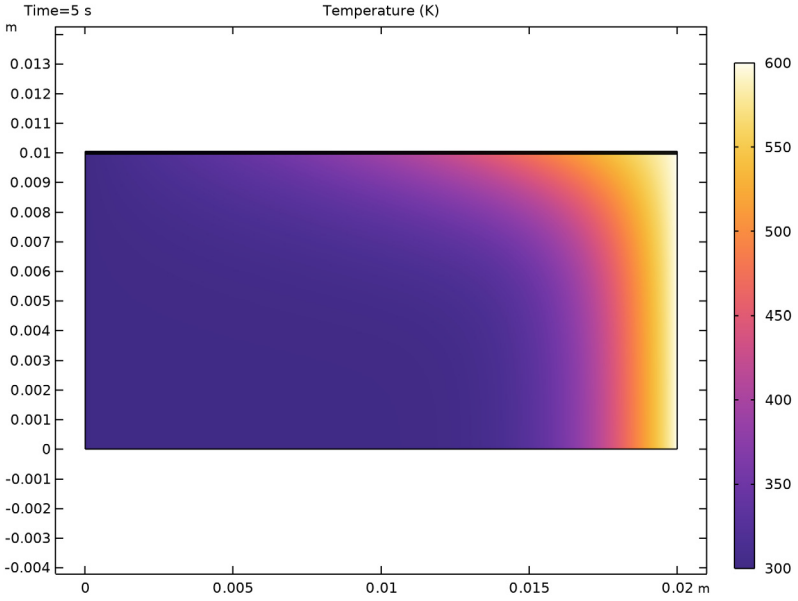
QUANTITY	SILICA GLASS	COPPER	DESCRIPTION
ρ	2203 kg/m^3	8960 kg/m^3	Density
C_p	703 J/(kg·K)	385 J/(kg·K)	Heat capacity at constant pressure
k	1.38 W/(m·K)	400 W/(m·K)	Thermal conductivity

The thermal conductivity of copper is much higher than that for silica glass. Given this fact and that the copper layer is thin, it is possible to model the layer with the Thin Layer feature. Using this feature you do not need to resolve the thin layer with an extremely fine mesh, which would require a significantly longer computation time.

In a second model version, you compare the results using the Thin Layer feature with a setup where the copper layer has been meshed instead. This model produces the same results, but requires a denser mesh and longer computation time.

Results

Figure 2 shows the temperature field after 5 s, 10 s, and 60 s. The results show that the temperature rise is faster in the copper layer than in the silica glass. After 60 s the temperature field has almost reached steady state, and the temperature field varies linearly between the two vertical boundaries.



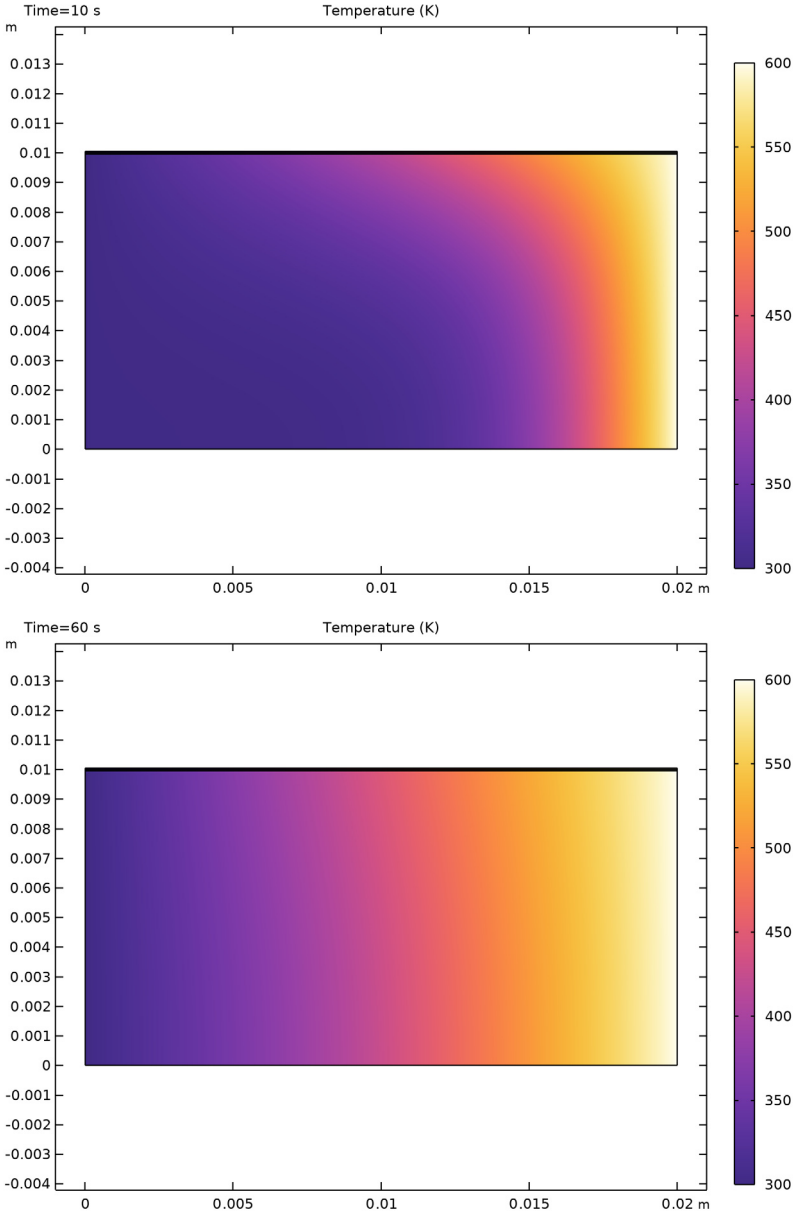


Figure 2: Temperature field after 5, 10, and 60 seconds.

Application Library path: `Heat_Transfer_Module/Tutorials,_Thin_Structure/copper_layer`

Modeling Instructions — Thin Layer

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** > **Time Dependent**.

6 Click **Done**.

GEOMETRY 1

Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `0.02[m]`.

4 In the **Height** text field, type `0.01[m]`.

Form Union (fin)

1 In the **Geometry** toolbar, click **Build All**.

2 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** > **Silica glass**.

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

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

6 Click the **Add to Component** button in the window toolbar.

7 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Copper (mat2)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Geometric entity level** list, choose **Boundary**.

3 Select Boundary 3 only.

HEAT TRANSFER IN SOLIDS (HT)

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 `300[K]`.

Temperature 2

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

2 Select Boundary 4 only.

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

4 In the T_0 text field, type `600[K]`.

Thin Layer 1

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

2 Select Boundary 3 only.

3 In the **Settings** window for **Thin Layer**, locate the **Layer Model** section.

4 From the **Layer type** list, choose **Thermally thin approximation**.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the T text field, type `300[K]`.

MATERIALS

Copper (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click **Copper (mat2)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Thickness	lth	`1e-4[m]`	m	Shell

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 Click **Range**.

4 In the **Range** dialog, type `5` in the **Step** text field.

5 In the **Stop** text field, type `60`.

6 Click **Replace**.

The default solver is accurate enough to get good results in terms of temperature. Tightening the tolerance improves the results in terms of energy balance, which you can check with the quantity `ht.energyBalance`.

7 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

8 From the **Tolerance** list, choose **User controlled**.

9 In the **Relative tolerance** text field, type `1e-5`.

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

RESULTS

Layered Shell

1 In the **Model Builder** window, expand the **Results** > **Temperature (ht)** node, then click **Layered Shell**.

2 In the **Settings** window for **Surface**, click to expand the **Title** section.

Temperature (ht)

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

The first default plot shows the temperature at the end of the simulated time interval, that is, at $t = 60$ s. Compare with the last plot of the series in Figure 2.

Reproduce the corresponding plots for $t = 5$ s and $t = 10$ s:

2 In the **Model Builder** window, click **Temperature (ht)**.

3 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

4 From the **Time (s)** list, choose **5**.

5 In the **Temperature (ht)** toolbar, click **Plot**.

6 From the **Time (s)** list, choose **10**.

7 In the **Temperature (ht)** toolbar, click **Plot**.

Meshed Copper Layer

Now, set up the second model version.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component** > **2D**.

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 1**.

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** > **Time Dependent**.

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

This geometry adds a domain for the layer.

Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `0.02[m]`.

4 In the **Height** text field, type `0.01[m]`.

Rectangle 2 (r2)

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

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

3 In the **Width** text field, type `0.02[m]`.

4 In the **Height** text field, type `1e-4[m]`.

5 Locate the **Position** section. In the **y** text field, type `0.01[m]`.

Form Union (fin)

1 In the **Geometry** toolbar, click **Build All**.

2 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** > **Silica glass**.

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

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

6 Click the **Add to Component** button in the window toolbar.

7 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Silica glass (mat3)

Select Domain 1 only.

Copper (mat4)

1 In the **Model Builder** window, click **Copper (mat4)**.

2 Select Domain 2 only.

HEAT TRANSFER IN SOLIDS 2 (HT2)

Temperature 1

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

2 Select Boundaries 1 and 3 only.

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

4 In the T_0 text field, type `300[K]`.

Temperature 2

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

2 Select Boundaries 6 and 7 only.

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

4 In the T_0 text field, type `600[K]`.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the $T2$ text field, type `300[K]`.

MESH 2

In the **Model Builder** window, under **Component 2 (comp2)** right-click **Mesh 2** and choose **Build All**.

STUDY 2

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 Click **Range**.

4 In the **Range** dialog, type `5` in the **Step** text field.

5 In the **Stop** text field, type `60`.

6 Click **Replace**.

Tighten the tolerance to improve the results in terms of energy balance.

7 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

8 From the **Tolerance** list, choose **User controlled**.

9 In the **Relative tolerance** text field, type `1e-5`.

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

RESULTS

Look at the solution for $t = 5$ s.

Temperature (ht2)

1 In the **Model Builder** window, under **Results** click **Temperature (ht2)**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Time (s)** list, choose **5**.

4 In the **Temperature (ht2)** toolbar, click **Plot**.

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