

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

- I In the Model Wizard window, click **2**D.
- 2 In the Select Physics tree, select Heat Transfer>Heat Transfer in Solids (ht).
- 3 Click Add.
- 4 Click \bigcirc Study.
- 5 In the Select Study tree, select General Studies>Time Dependent.
- 6 Click **M** Done.

GEOMETRY I

Rectangle 1 (r1)

- I 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.
- 4 In the **Height** text field, type 0.01.
- 5 Click 틤 Build Selected.

ADD MATERIAL

- I 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>Silica glass.
- 4 Click Add to Component in the window toolbar.
- 5 In the tree, select **Built-in>Copper**.

- 6 Click Add to Component in the window toolbar.
- 7 In the Home toolbar, click 🙀 Add Material to close the Add Material window.

MATERIALS

Copper (mat2)

- I 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

- I In the Model Builder window, under Component I (compl) right-click Heat Transfer in Solids (ht) 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

- I 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 I

- I 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

- I In the Model Builder window, click Initial Values I.
- 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)

- I In the Model Builder window, under Component I (compl)>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	Shell

MESH I

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

STUDY I

Step 1: Time Dependent

- I In the Model Builder window, under Study I click Step I: Time Dependent.
- 2 In the Settings window for Time Dependent, locate the Study Settings section.
- 3 Click Range.
- 4 In the Range dialog box, 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.
- **IO** In the **Home** toolbar, click **= Compute**.

RESULTS

Surface 2

- I In the Model Builder window, expand the Results>Temperature (ht) node, then click Surface 2.
- 2 In the Settings window for Surface, click to expand the Title section.

3 From the **Title type** list, choose **None**.

Temperature (ht)

I 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 **I** 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

- I 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

- I In the Home toolbar, click $\stackrel{\sim}{\sim}$ 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 check box for Heat Transfer in Solids (ht).
- 5 Click Add Study in the window toolbar.
- 6 In the Home toolbar, click $\sim\sim$ Add Study to close the Add Study window.

GEOMETRY 2

This geometry adds a domain for the layer.

Rectangle 1 (r1)

- I 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.
- 4 In the **Height** text field, type 0.01.
- 5 Click 틤 Build Selected.

Rectangle 2 (r2)

- I 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.
- 4 In the Height text field, type 1e-4.
- 5 Locate the Position section. In the y text field, type 0.01.
- 6 Click 📗 Build All Objects.

ADD MATERIAL

- I 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>Silica glass.
- 4 Click Add to Component in the window toolbar.
- 5 In the tree, select Built-in>Copper.
- 6 Click Add to Component in the window toolbar.
- 7 In the Home toolbar, click 🙀 Add Material to close the Add Material window.

MATERIALS

Silica glass (mat3)

- I In the Model Builder window, under Component 2 (comp2)>Materials click Silica glass (mat3).
- 2 Select Domain 1 only.

Copper (mat4)

- I In the Model Builder window, click Copper (mat4).
- **2** Select Domain 2 only.

HEAT TRANSFER IN SOLIDS 2 (HT2)

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

Temperature 1

- I 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

- I 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

- I In the Model Builder window, click Initial Values I.
- 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
- I In the Model Builder window, under Study 2 click Step I: Time Dependent.
- 2 In the Settings window for Time Dependent, locate the Study Settings section.
- 3 Click Range.
- 4 In the Range dialog box, 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.
- **IO** In the **Home** toolbar, click **= Compute**.

RESULTS

Look at the solution for t = 5 s.

Temperature (ht2)

- I 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.