Composite Thermal Barrier

Introduction

This example shows how to set up multiple sandwiched thin layers with different thermal conductivities in two different ways.

First, the composite is modeled as a 3D object. In the second approach the Thin thermally resistive layer boundary condition is used to avoid resolving the thin domains.

The technique is useful when modeling heat transfer through thermal barriers like multilayer coatings.

Model Definition

This tutorial model uses a simple geometry as shown in Figure 1. The cylinder has a radius of 2cm and a height of 4cm.

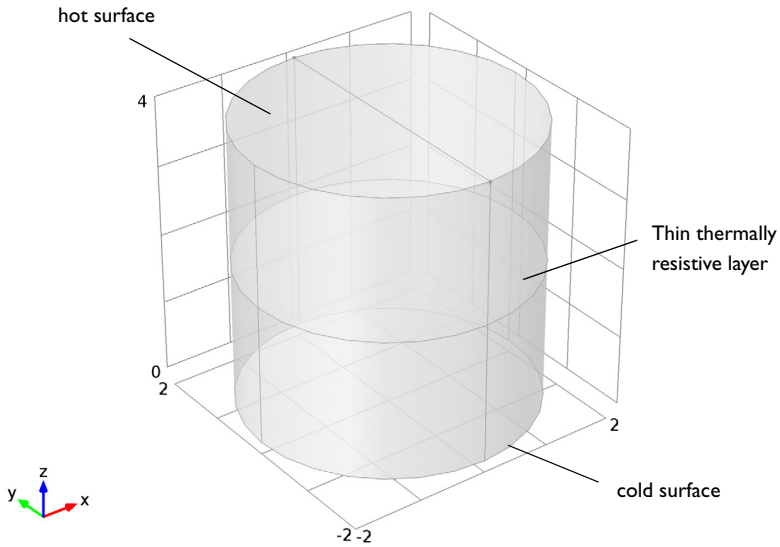

Figure 1: Geometry

The composite consists of two layers with different thermal conductivities. The first approach resolves each layer as a 3D Domain. The height of the layers is about three orders of magnitude smaller than the bulk height which often leads to difficulties in meshing.

COMSOL provides a special boundary condition which is available from the Heat Transfer module. The second approach uses the Thin thermally resistive layer boundary condition which simplifies the geometry and thus the mesh. In complex geometries the use of this boundary condition can reduce the amount of memory and time required for the simulation significantly.

The underlying equation assumes the heatflux through the layer proportional to the temperature difference between upper and lower bulk. It is based on the assumption that the bulk on each side is well stirred so that all resistance against heat transfer is within a thin layer near the wall. Due to the additivity of resistance the flux over the composite can be lumped to

$$-\mathbf{n}_d(-k_d\nabla T_d) = \frac{k_{tot}}{d_{tot}}(T_u - T_d)$$

$$-\mathbf{n}_u(-k_u\nabla T_u) = \frac{k_{tot}}{d_{tot}}(T_d - T_u)$$

where the overall thermal conductivity k_{tot} can be calculated as

$$k_{tot} = \frac{\sum_n d_n}{\sum_n \frac{d_n}{k_n}}$$

MATERIAL PROPERTIES

The cylinder is made of steel. The composite consist of two layers of different ceramics

TABLE 0-1: CERAMICS MATERIAL PROPERTIES

PROPERTY	CERAMIC 1	CERAMIC 2
Conductivity	1 [W/(m·K)]	0.5[W/(m·K)]
Density	6000 [kg/m³]	5800[kg/m³]
Heat Capacity	320[J/(kg·K)]	280[J/(kg·K)]

 ©2012 COMSOL

BOUNDARY CONDITIONS

The temperature at the bottom is fixed to 20°C whereas one half of the top boundary is held at 1220°C. All other outer boundaries are perfectly insulated.

Results and Discussion

Figure 2 shows the temperature distribution in the cylinder. The composite acts as a thermal barrier resulting in a jump of the temperature over the layer.

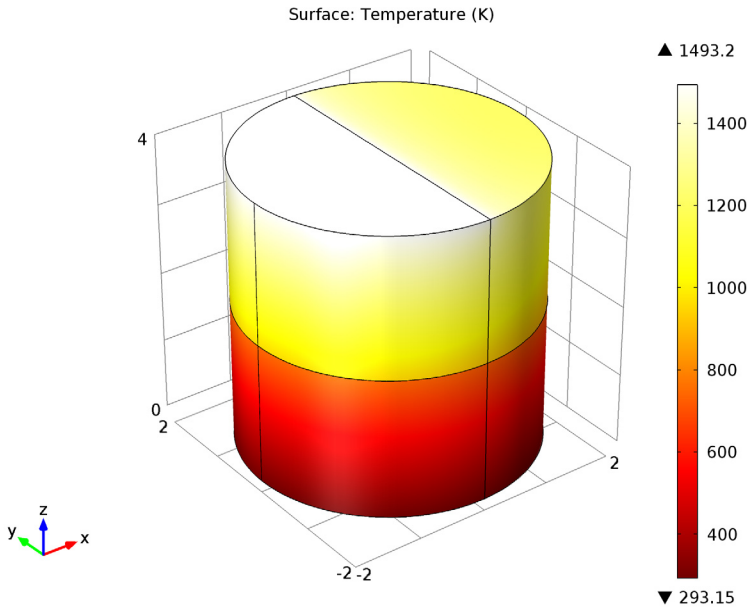

Figure 2: Temperature Distribution 3D

The interesting question is if the Thin thermally resistive layer boundary condition produces reliable results compared to resolving the thin layers in 3D. This can be done

with a comparative line graph as in Figure 3. It shows that the 2D approach produces accurate results for the bulk temperatures.

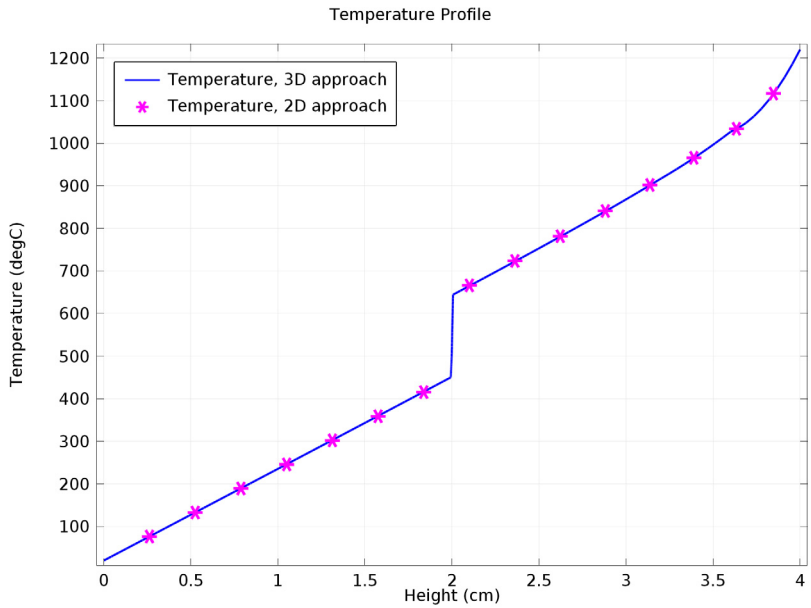

Figure 3: Temperature profile for 3D and 2D approach

Another important question for simulating is the influence of this feature on the mesh size and so on the requiered RAM.

With the default tetrahedral mesh the number of mesh elements is about 130000 elements and the meshing algorithm gives some warnings.

With the swept mesh feature you can significantly reduce the number of elements to about 12800 elements which is only 10%. In complex geometries the swept mesh algorithm is often not applicable. With the use of the thin thermally resistive layer feature even in this simple geometry the number of mesh elements reduces from 12800 to 9700 which is about 25% less . You can see the number of mesh elements used in the messages box on the bottom right window.

Notes About the COMSOL Implementation

To compare the results directly both approaches are handled within one mph-file.

 ©2012 COMSOL

Model Library path: `Heat_Transfer_Module/Tutorial_Models/composite_thermal_barrier`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

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

4 Click **Next**.

5 Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.

6 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Description
d_ceram1	50[um]	Thickness of layer 1
d_ceram2	75[um]	Thickness of layer 2
T_hot	1220[degC]	Temperature hot

GEOMETRY 1

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

2 In the **Geometry** settings window, locate the **Units** section.

3 From the **Length unit** list, choose **cm**.

Cylinder 1

1 Right-click **Model 1>Geometry 1** and choose **Cylinder**.

2 In the **Cylinder** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type 2.

4 In the **Height** edit field, type `4`.

Use a work plane to define the interface between the 2 steel cylinders and the ceramic layers.

Work Plane 1

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

2 In the **Work Plane** settings window, locate the **Work Plane** section.

3 In the **z-coordinate** edit field, type `2-(d_ceram1+d_ceram2)/2`.

Plane Geometry

Click the **Zoom Extents** button on the Graphics toolbar.

Circle 1

1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Circle**.

2 In the **Circle** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `2`.

Extrude 1

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

2 In the **Extrude** settings window, locate the **Distances from Plane** section.

3 In the table, enter the following settings:

Distances (cm)
`d_ceram1`
`d_ceram1+d_ceram2`

Work Plane 2

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

2 In the **Work Plane** settings window, locate the **Work Plane** section.

3 In the **z-coordinate** edit field, type `4`.

Bézier Polygon 1

1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 2** right-click **Plane Geometry** and choose **Bézier Polygon**.

2 In the **Bézier Polygon** settings window, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click the **Add Linear** button.

 ©2012 COMSOL

4 Find the **Control points** subsection. In row **1**, set **yw** to `-2`.

5 In row **2**, set **yw** to `2`.

6 In the **Model Builder** window, right-click **Geometry 1** and choose **Build All**.

MATERIALS

Material Browser

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** window, locate the **Materials** section.

3 In the tree, select **Built-In>Steel AISI 4340**.

4 Right-click and choose **Add Material to Model** from the menu.

Material 2

1 In the **Model Builder** window, right-click **Materials** and choose **Material**.

2 Right-click **Material 2** and choose **Rename**.

3 Go to the **Rename Material** dialog box and type `Ceramic 1` in the **New name** edit field.

4 Click **OK**.

5 Select Domain 2 only.

6 In the **Material** settings window, locate the **Material Contents** section.

7 In the table, enter the following settings:

Property	Name	Value
Thermal conductivity	k	1
Density	rho	6000
Heat capacity at constant pressure	Cp	320

Material 3

1 In the **Model Builder** window, right-click **Materials** and choose **Material**.

2 Right-click **Material 3** and choose **Rename**.

3 Go to the **Rename Material** dialog box and type `Ceramic 2` in the **New name** edit field.

4 Click **OK**.

5 Select Domain 3 only.

6 In the **Material** settings window, locate the **Material Contents** section.

7 In the table, enter the following settings:

Property	Name	Value
Thermal conductivity	k	0.5
Density	rho	5800
Heat capacity at constant pressure	Cp	280

HEAT TRANSFER IN SOLIDS

Temperature 1

1 In the **Model Builder** window, under **Model 1** right-click **Heat Transfer in Solids** and choose **Temperature**.

2 Select Boundary 3 only.

Temperature 2

1 In the **Model Builder** window, right-click **Heat Transfer in Solids** and choose **Temperature**.

2 In the **Temperature** settings window, locate the **Temperature** section.

3 In the T_0 edit field, type `T_hot`.

4 Select Boundary 13 only.

MESH 1

First mesh the top surface with a free triangular mesh and then extrude it in layers through the cylindrical geometry. With a **Distribution** node you can specify how many mesh layers to create within the domain. Here, resolve the composite layers with two elements in thickness.

Free Triangular 1

1 In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

2 Select Boundaries 13 and 18 only.

Swept 1

In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

Distribution 1

1 In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Swept 1** and choose **Distribution**.

2 Select Domains 2 and 3 only.

 ©2012 COMSOL

3 In the **Distribution** settings window, locate the **Distribution** section.

4 In the **Number of elements** edit field, type 2.

5 Click the **Build All** button.

STUDY 1

In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

Temperature

The following plots are produced per default: The Temperature on the surface as in Figure 2 and an isothermal contour plot.

1 In the **Model Builder** window, expand the **Temperature** node, then click **Surface**.

2 In the **Surface** settings window, locate the **Expression** section.

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

4 Click the **Plot** button.

Next, create a temperature profile along the height of the cylinder. You will later compare the graph with the results of the 2D approach.

1D Plot Group 3

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

2 Right-click **1D Plot Group 3** and choose **Line Graph**.

3 Select Edges 15, 17, 19, and 21 only.

4 In the **Line Graph** settings window, locate the **y-Axis Data** section.

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

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** edit field, type z.

8 Locate the **Coloring and Style** section. Find the **Line style** subsection. In the **Width** edit field, type 2.

9 Click to expand the **Legends** section. Select the **Show legends** check box.

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

11 In the table, enter the following settings:

Legends
Temperature, 3D approach

12 In the **Model Builder** window, click **1D Plot Group 3**.

13 In the **1D Plot Group** settings window, locate the **Plot Settings** section.

14 Select the **x-axis label** check box.

15 In the associated edit field, type `Height (cm)`.

16 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

17 In the **Title** text area, type `Temperature Profile`.

18 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

19 Right-click **1D Plot Group 3** and choose **Rename**.

20 Go to the **Rename 1D Plot Group** dialog box and type `Temperature Profile` in the **New name** edit field.

21 Click **OK**.

ROOT

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

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

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

4 Click **Next**.

Add a second study solving for the second model only. This way you make sure to keep all the results from the previous simulation which then can be compared to the results of the second one.

5 Find the **Selected physics** subsection. In the table, enter the following settings:

Physics	Solve for
Heat Transfer in Solids (ht)	×

6 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Stationary**.

7 Click **Finish**.

GEOMETRY 2

1 In the **Model Builder** window, under **Model 2** click **Geometry 2**.

2 In the **Geometry** settings window, locate the **Units** section.

 ©2012 COMSOL

3 From the **Length unit** list, choose **cm**.

Cylinder 1

1 Right-click **Model 2>Geometry 2** and choose **Cylinder**.

2 In the **Cylinder** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type 2.

4 In the **Height** edit field, type 4.

Work Plane 1

1 In the **Model Builder** window, right-click **Geometry 2** and choose **Work Plane**.

2 In the **Work Plane** settings window, locate the **Work Plane** section.

3 In the **z-coordinate** edit field, type 2.

Circle 1

1 In the **Model Builder** window, under **Model 2>Geometry 2>Work Plane 1** right-click **Plane Geometry** and choose **Circle**.

2 In the **Circle** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type 2.

Work Plane 2

1 In the **Model Builder** window, right-click **Geometry 2** and choose **Work Plane**.

2 In the **Work Plane** settings window, locate the **Work Plane** section.

3 In the **z-coordinate** edit field, type 4.

Bézier Polygon 1

1 In the **Model Builder** window, under **Model 2>Geometry 2>Work Plane 2** right-click **Plane Geometry** and choose **Bézier Polygon**.

2 In the **Bézier Polygon** settings window, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click the **Add Linear** button.

4 Find the **Control points** subsection. In row **1**, set **yw** to -2.

5 In row **2**, set **yw** to 2.

6 In the **Model Builder** window, right-click **Geometry 2** and choose **Build All**.

7 Click the **Transparency** button on the Graphics toolbar.

MATERIALS

Material Browser

1 In the **Model Builder** window, under **Model 2** right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** window, locate the **Materials** section.

3 In the tree, select **Recent Materials>Steel AISI 4340**.

4 Right-click and choose **Add Material to "geom2"** from the menu.

HEAT TRANSFER IN SOLIDS 2

Thin Thermally Resistive Layer 1

1 In the **Model Builder** window, under **Model 2** right-click **Heat Transfer in Solids 2** and choose **Thin Thermally Resistive Layer**.

2 In the **Thin Thermally Resistive Layer** settings window, locate the **Thin Thermally Resistive Layer** section.

3 Select the **Multiple layers** check box.

4 From the **Number of layers** list, choose **2**.

5 Select Boundary 6 only.

6 From the **Solid material 1** list, choose **Ceramic 1**.

7 In the d_{s1} edit field, type `d_ceram1`.

8 From the **Solid material 2** list, choose **Ceramic 2**.

9 In the d_{s2} edit field, type `d_ceram2`.

Temperature 1

1 In the **Model Builder** window, right-click **Heat Transfer in Solids 2** and choose **Temperature**.

2 Select Boundary 3 only.

Temperature 2

1 Right-click **Heat Transfer in Solids 2** and choose **Temperature**.

2 Select Boundary 7 only.

3 In the **Temperature** settings window, locate the **Temperature** section.

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

 ©2012 COMSOL

MESH 2

Free Triangular 1

1 In the **Model Builder** window, under **Model 2** right-click **Mesh 2** and choose **More Operations>Free Triangular**.

2 Select Boundaries 7 and 10 only.

Swept 1

1 In the **Model Builder** window, right-click **Mesh 2** and choose **Swept**.

2 Right-click **Mesh 2** and choose **Build All**.

STUDY 2

In the **Model Builder** window, right-click **Study 2** and choose **Compute**.

RESULTS

Temperature 1

1 In the **Model Builder** window, expand the **Temperature 1** node, then click **Surface**.

2 In the **Surface** settings window, locate the **Expression** section.

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

4 Click the **Transparency** button on the Graphics toolbar.

Temperature Profile

1 In the **Model Builder** window, under **Results** right-click **Temperature Profile** and choose **Line Graph**.

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

3 From the **Data set** list, choose **Solution 3**.

4 Select Edges 9 and 11 only.

5 Locate the **y-Axis Data** section. In the **Expression** edit field, type `T2`.

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

7 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

8 In the **Expression** edit field, type `z`.

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

10 From the **Color** list, choose **Magenta**.

11 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

12 In the **Number** edit field, type `15`.

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

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

15 In the table, enter the following settings:

Legends
`Temperature, 2D approach`

16 Click the **Plot** button.

The plot should look like that in Figure 3.

 ©2012 COMSOL