Lumped Composite Thermal Barrier with Shells

Introduction

This example shows how to replace two 3D finite element domains by thin structures for heat transfer modeling, and to connect them through a lumped thermal system to account for a thermal barrier.

The model is a variant of the Composite Thermal Barrier and Lumped Composite Thermal Barrier models, in which two ceramic thin layers with different thermal conductivities are sandwiched in a steel column.

Two modeling approaches are compared for the computation of the temperature distribution through the whole column. First, both the steel column and the composite (made of the ceramic layers) are modeled as 3D objects. In the second approach, to avoid resolving both the column and the ceramic layers in the geometry, the Heat Transfer in Shells interface is used to model the upper and lower parts of the steel column, and the Lumped Thermal System interface is used for the ceramic layers, and coupled to the two shells through boundary conditions.

This methodology is useful when modeling heat transfer through thermal barriers like multilayer coatings.

Model Definition

GEOMETRY

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

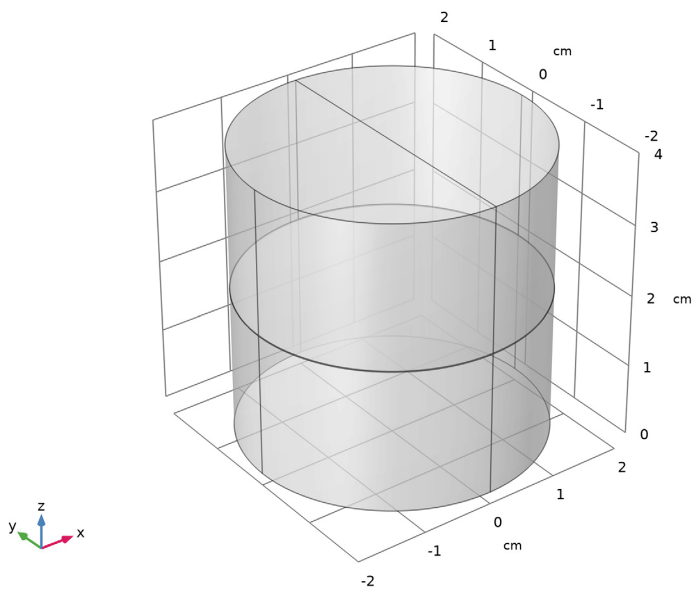

Figure 1: Geometry.

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. This often requires to build a mesh manually to accurately resolve the thin structure.

LUMPED APPROACH: THIN STRUCTURE REPRESENTATION OF THE STEEL COLUMN

The Heat Transfer in Shells interface is used to model the conductive heat transfer in the top and bottom parts of the steel column.

The top and bottom cylinders are represented by two circular shells whose thickness is defined in the **Layered Material** settings. Although the column thickness is not represented explicitly in the geometry, the Layered Material technology allows to solve the temperature distribution through the shells. The number of implicit mesh elements can be set in the **Layered Material** node as well, to fit the settings used for the swept meshes of the 3D domains.

The **Temperature, Interface** condition is applied on the bottom interface of the lower part of the column, and on the top interface of the upper part of the column, to prescribe the temperature at each extremity of the column.

LUMPED APPROACH: NETWORK REPRESENTATION OF THE THERMAL BARRIER

COMSOL Multiphysics provides the Lumped Thermal System physics interface, available from the Heat Transfer Module, and in which the **Conductive Thermal Resistor** feature allows to model conductive heat transfer without representing the underlying geometry.

The Lumped Thermal System physics interface uses a network representation of thermal systems to model heat transfer by analogy with electrical circuits. The domain and boundary conditions for heat transfer are idealized by components joined by a network of perfectly thermally conductive wires.

This 0D approach simplifies the geometry and thus the mesh. In complex geometries, this lumped approach can significantly reduce the amount of memory and time required for the simulation.

For the modeling of the thermally resistive ceramic layers, two **Conductive Thermal Resistor** components are connected in a serial circuit.

The **Conductive Thermal Resistor** feature models heat conduction in a thin shell of constant conductivity. In this example, a plane shell configuration is assumed, and the thermal resistance R (SI unit: K/W) of each layer is expressed from the thermal conductivity k (SI unit: W/(m·K)), the thickness L (SI unit: m), and the surface area A (SI unit: m^2) as follows:

$$R = \frac{L}{kA}$$

It then assumes that the heat rate P (SI unit: W) through each layer is proportional to the temperature difference ΔT (SI unit: K) across it:

$$P = -\frac{\Delta T}{R}$$

See *Theory for the Lumped Thermal System Interface* in the *Heat Transfer Module User's Guide* for more details about the underlying theory.

The complete thermal circuit modeled by the Lumped Thermal System interface is as shown in Figure 2.

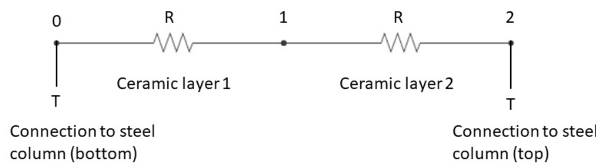

Figure 2: Thermal circuit for heat transfer in the ceramic layers.

LUMPED APPROACH: CONNECTION BETWEEN HEAT TRANSFER IN SHELLS AND LUMPED THERMAL SYSTEM

The coupling between the two physics interfaces, Heat Transfer in Shells (2D approach) and Lumped Thermal System (0D approach), is performed through the following features:

- **Lumped System Connector, Interface** feature in the Heat Transfer in Shells interface, applied on the bottom interface of the top shell, and on the top interface of the bottom shell. This feature uses the heat rate defined by each **External Terminal** feature to set a heat flux on the corresponding boundary in the distributed finite element model.
- **External Terminal** feature in the Lumped Thermal System interface, applied at each extremity of the thermal circuit. This feature prescribes the temperature T_{ext} provided by the **Lumped System Connector, Interface** feature of the Heat Transfer in Shells interface.

MATERIAL PROPERTIES

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

TABLE 1: CERAMICS MATERIAL PROPERTIES.

PROPERTY	CERAMIC 1	CERAMIC 2
Thermal conductivity	1 W/(m·K)	0.5 W/(m·K)
Density	6000 kg/m^3	5800 kg/m^3
Heat capacity at constant pressure	320 J/(kg·K)	280 J/(kg·K)

BOUNDARY CONDITIONS

The temperature at the bottom is fixed to 20°C whereas one half of the top boundary is held at 1220°C (1493 K). All other outer boundaries are perfectly insulated.

Results and Discussion

Figure 3 shows the temperature distribution in the cylinder, with the 3D and the lumped models. The composite acts as a thermal barrier resulting in a jump of the temperature over the layer.

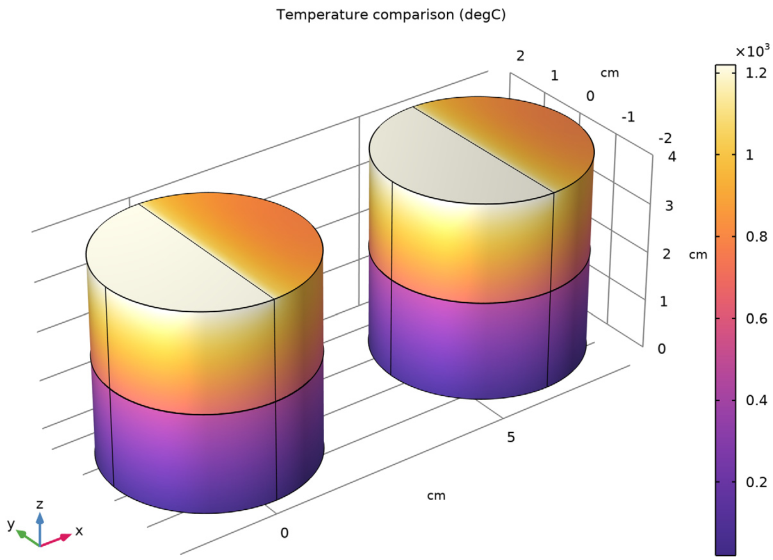

Figure 3: Temperature distribution with 3D model (left) and lumped model (right).

Of interest is if the lumped approach (using Heat Transfer in Shells and Lumped Thermal System) produces reliable results compared to resolving the whole steel column in 3D.

This can be done with a comparative line graph as in Figure 4. It shows that the lumped approach produces accurate results for the bulk temperatures.

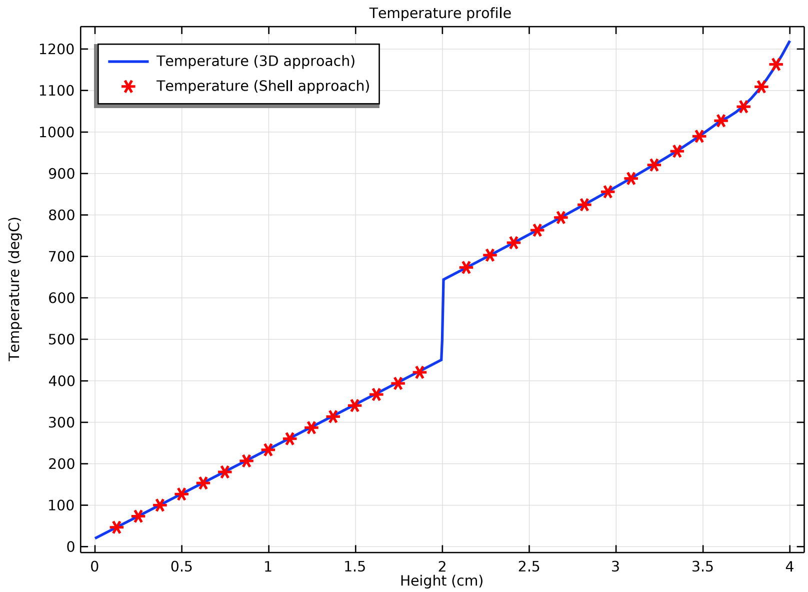

Figure 4: Temperature profile for 3D and lumped approaches.

Another important question for simulating is the influence on the mesh size and on the required RAM.

With the default tetrahedral mesh of the 3D model, the number of mesh elements is about 130,000 elements and the meshing algorithm gives some warnings.

With the swept mesh feature you can significantly reduce the number of elements to about 2800 elements (prisms), which is only 2% of the initial number of elements. Note that in complex geometries the swept mesh algorithm is often not applicable.

Using the lumped approach, the number of mesh elements reduces to about 1300 elements (triangles). You can see the number of mesh elements used in the **Messages** window below the **Graphics** window.

Notes About the COMSOL Implementation

To compare the results directly, both approaches are handled in a single MPH-file.

Application Library path: Heat_Transfer_Module/Tutorials,_Thin_Structure/lumped_composite_thermal_barrier_shells

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 **3D**.

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 In the table, enter the following settings:

Name	Expression	Value	Description
d_ceram1	50[um]	5E-5 m	Thickness of layer 1
d_ceram2	75[um]	7.5E-5 m	Thickness of layer 2
T_hot	1220[degC]	1493.2 K	Hot temperature

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

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

Cylinder 1 (cyl1)

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

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

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

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

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

Now, create thin cylinders to define the ceramic layers between the two steel domains.

Cylinder 2 (cyl2)

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

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

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

4 In the **Height** text field, type `d_ceram1`.

5 Locate the **Position** section. In the **z** text field, type `2-(d_ceram1+d_ceram2)/2`.

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

Cylinder 3 (cyl3)

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

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

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

4 In the **Height** text field, type `d_ceram2`.

5 Locate the **Position** section. In the **z** text field, type `2-(d_ceram1+d_ceram2)/2+ d_ceram1`.

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

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

x (cm)	y (cm)	z (cm)
0	-2	4
0	2	4

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

MATERIALS

Material Link 1 (matlnk1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **More Materials>Material Link**.

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

3 Click **Add Material from Library**.

ADD MATERIAL TO MATERIAL LINK 1 (MATLNK1)

1 Go to the **Add Material to Material Link 1 (matlnk1)** window.

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

3 Right-click and choose **Add to Material Link 1 (matlnk1)**.

MATERIALS

Material Link 2 (matlnk2)

1 Right-click **Materials** and choose **More Materials>Material Link**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `2` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Material Link**, locate the **Link Settings** section.

7 Click **Blank Material**.

8 In the **Model Builder** window, click **Material Link 2 (matlnk2)**.

9 Click **Go to Material**.

GLOBAL DEFINITIONS

Ceramic 1

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Material 2 (mat2)**.

2 In the **Settings** window for **Material**, type `Ceramic 1` in the **Label** text field.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	1	W/(m·K)	Basic
Density	rho	6000	kg/m³	Basic
Heat capacity at constant pressure	Cp	320	J/(kg·K)	Basic

MATERIALS

Material Link 3 (matlnk3)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **More Materials>Material Link**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `3` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Material Link**, locate the **Link Settings** section.

7 Click **Blank Material**.

8 In the **Model Builder** window, click **Material Link 3 (matlnk3)**.

9 Click **Go to Material**.

GLOBAL DEFINITIONS

Ceramic 2

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Material 3 (mat3)**.

2 In the **Settings** window for **Material**, type `Ceramic 2` in the **Label** text field.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	0.5	W/(m·K)	Basic
Density	rho	5800	kg/m³	Basic
Heat capacity at constant pressure	Cp	280	J/(kg·K)	Basic

HEAT TRANSFER IN SOLIDS (HT)

Temperature 1

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

2 Select Boundary 3 only.

Temperature 2

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

2 Select Boundary 13 only.

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

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

MESH 1

First, mesh the top surface with a free triangular mesh and extrude it in layers through the cylindrical geometry. With a **Distribution** node, specify how many mesh layers are to be created within the domain. Resolve the composite layers with two elements in thickness.

Free Triangular 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

2 Select Boundaries 13 and 18 only.

3 In the **Settings** window for **Free Triangular**, click **Build Selected**.

Swept 1

In the **Mesh** toolbar, click **Swept**.

Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.

2 Select Domains 2 and 3 only.

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

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

5 Click **Build All**.

ADD COMPONENT

Right-click **Distribution 1** and choose **Add Component>3D**.

GEOMETRY 2

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

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

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1)>Circle 1 (c1)

1 In the **Work Plane** toolbar, click **Circle**.

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

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

Work Plane 2 (wp2)

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

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

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

Work Plane 2 (wp2)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 2 (wp2)>Circle 1 (c1)

1 In the **Work Plane** toolbar, click **Circle**.

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

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

Polygon 1 (pol1)

1 In the **Model Builder** window, right-click **Geometry 2** and choose **More Primitives>Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

x (cm)	y (cm)	z (cm)
0	-2	2+(d_ceram1+d_ceram2)/2
0	2	2+(d_ceram1+d_ceram2)/2

4 Click **Build All Objects**.

MATERIALS

Layered Material Link 1 (llmat1)

1 In the **Model Builder** window, under **Component 2 (comp2)** right-click **Materials** and choose **Layers>Layered Material Link**.

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

GLOBAL DEFINITIONS

Layered Material 1 (lmat1)

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Layered Material 1 (lmat1)**.

2 In the **Settings** window for **Layered Material**, locate the **Layer Definition** section.

3 In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
Layer 1	Steel AISI 4340 (mat1)	0.0	2[cm]-(d_ceram1+d_ceram2)/2	2

MATERIALS

Layered Material Link 1 (llmat1)

1 In the **Model Builder** window, under **Component 2 (comp2)>Materials** click **Layered Material Link 1 (llmat1)**.

2 In the **Settings** window for **Layered Material Link**, click **Section_bar** in the upper-right corner of the **Layered Material Settings** section. From the menu, choose **Layer Cross-Section Preview**.

3 Locate the **Orientation and Position** section. From the **Position** list, choose **Bottom side on boundary**.

GLOBAL DEFINITIONS

Layered Material 1 (lmat1)

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Layered Material 1 (lmat1)**.

2 In the **Settings** window for **Layered Material**, locate the **Layer Definition** section.

3 In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
`Layer 1`	Steel AISI 4340 (mat1)	`0.0`	`2[cm]-(d_ceram1+d_ceram2)/2`	`5`

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>Thin Structures>Heat Transfer in Shells (htlsh)**.
4. Click **Add to Component 2** in the window toolbar.
5. In the tree, select **Heat Transfer>Lumped Thermal System (lts)**.
6. Click **Add to Component 2** in the window toolbar.
7. In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

LUMPED THERMAL SYSTEM (LTS)

Ceramic 1

1. Right-click **Component 2 (comp2)>Lumped Thermal System (lts)** and choose **Conductive Thermal Resistor**.
2. In the **Settings** window for **Conductive Thermal Resistor**, type `Ceramic 1` in the **Label** text field.
3. Locate the **Component Parameters** section. From the **Specify** list, choose **Thermal and geometric properties**.
4. From the **Material** list, choose **Ceramic 1 (mat2)**.
5. In the A text field, type `pi*(2[cm])^2`.
6. In the L text field, type `d_ceram1`.

Ceramic 2

1. In the **Physics** toolbar, click **Global** and choose **Conductive Thermal Resistor**.
2. In the **Settings** window for **Conductive Thermal Resistor**, type `Ceramic 2` in the **Label** text field.

3 Locate the **Node Connections** section. In the table, enter the following settings:

Label	Node names
p1	1
p2	2

4 Locate the **Component Parameters** section. From the **Specify** list, choose **Thermal and geometric properties**.

5 From the **Material** list, choose **Ceramic 2 (mat3)**.

6 In the A text field, type `pi*(2[cm])^2`.

7 In the L text field, type `d_ceram2`.

External Terminal 1 (term1)

1 In the **Physics** toolbar, click **Global** and choose **External Terminal**.

2 In the **Settings** window for **External Terminal**, locate the **Node Connections** section.

3 In the **Node name** text field, type `0`.

External Terminal 2 (term2)

1 In the **Physics** toolbar, click **Global** and choose **External Terminal**.

2 In the **Settings** window for **External Terminal**, locate the **Node Connections** section.

3 In the **Node name** text field, type `2`.

HEAT TRANSFER IN SHELLS (HTLSH)

Solid 1

1 In the **Model Builder** window, under **Component 2 (comp2)>Heat Transfer in Shells (htlsh)** click **Solid 1**.

2 In the **Settings** window for **Solid**, locate the **Layer Model** section.

3 From the **Layer type** list, choose **General**.

Lumped System Connector, Interface 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Lumped System Connector, Interface**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Lumped System Connector, Interface**, locate the **Interface Selection** section.

4 From the **Apply to** list, choose **Top interface**.

Temperature, Interface 1

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

2 Select Boundary 1 only.

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

4 From the **Apply to** list, choose **Bottom interface**.

Lumped System Connector, Interface 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Lumped System Connector, Interface**.

2 Select Boundaries 2 and 3 only.

3 In the **Settings** window for **Lumped System Connector, Interface**, locate the **Interface Selection** section.

4 From the **Apply to** list, choose **Bottom interface**.

5 Locate the **Terminal Inputs** section. From the P_{ext} list, choose **External Terminal 2 (term2) (lts/term2)**.

Temperature, Interface 2

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

2 Select Boundary 2 only.

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

4 From the **Apply to** list, choose **Top interface**.

5 Locate the **Temperature** section. In the T_0 text field, type `T_hot`.

MESH 2

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

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Finer**.

STUDY 1

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check boxes for **Heat Transfer in Shells (htlsh)** and **Lumped Thermal System (lts)**.

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

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

STUDY 2

Step 1: Stationary

1 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

2 In the table, clear the **Solve for** check box for **Heat Transfer in Solids (ht)**.

STUDY 1

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

STUDY 2

Click **Compute**.

RESULTS

Isothermal Contours (ht), Temperature (ht)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Temperature (ht)** and **Isothermal Contours (ht)**.

2 Right-click and choose **Group**.

3D Approach

In the **Settings** window for **Group**, type `3D Approach` in the **Label** text field.

Surface

1 In the **Model Builder** window, expand the **Results>3D Approach>Temperature (ht)** node, then click **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

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

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

Isosurface

1 In the **Model Builder** window, expand the **Results>3D Approach>Isothermal Contours (ht)** node, then click **Isosurface**.

2 In the **Settings** window for **Isosurface**, locate the **Expression** section.

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

4 In the **Isothermal Contours (ht)** toolbar, click **Plot**.

Temperature (htlsh)

In the **Model Builder** window, under **Results** right-click **Temperature (htlsh)** and choose **Group**.

Shell Approach

In the **Settings** window for **Group**, type `Shell Approach` in the **Label** text field.

Surface

1 In the **Model Builder** window, expand the **Results>Shell Approach>Temperature (htlsh)** node, then click **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

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

Isothermal Contours (htlsh)

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Isothermal Contours (htlsh)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Layered Material 1**.

Isosurface 1

1 Right-click **Isothermal Contours (htlsh)** and choose **Isosurface**.

2 In the **Settings** window for **Isosurface**, locate the **Expression** section.

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

4 Locate the **Levels** section. In the **Total levels** text field, type `10`.

5 Locate the **Coloring and Style** section. Click **Change Color Table**.

6 In the **Color Table** dialog box, select **Thermal>HeatCameraLight** in the tree.

7 Click **OK**.

Temperature Comparison

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

Follow the instructions below to reproduce the plot of Figure 3.

2 In the **Settings** window for **3D Plot Group**, type `Temperature Comparison` in the **Label** text field.

T (Solid)

1 Right-click **Temperature Comparison** and choose **Surface**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (1) (sol1)**.

4 In the **Label** text field, type `T (Solid)`.

5 Locate the **Expression** section. From the **Unit** list, choose **degC**.

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

7 In the **Title** text area, type `Temperature comparison (degC)`.

8 Locate the **Coloring and Style** section. Click **Change Color Table**.

9 In the **Color Table** dialog box, select **Thermal>HeatCameraLight** in the tree.

10 Click **OK**.

Temperature Comparison

1 In the **Model Builder** window, click **Temperature Comparison**.

2 In the **Settings** window for **3D Plot Group**, click to expand the **Plot Array** section.

3 Select the **Enable** check box.

4 In the **Relative padding** text field, type `0.5`.

T2 (Shells)

1 Right-click **Temperature Comparison** and choose **Surface**.

2 In the **Settings** window for **Surface**, type `T2 (Shells)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Layered Material 1**.

4 Locate the **Expression** section. In the **Expression** text field, type `T2`.

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

6 Locate the **Title** section. From the **Title type** list, choose **None**.

7 Click to expand the **Inherit Style** section. From the **Plot** list, choose **T (Solid)**.

8 In the **Temperature Comparison** toolbar, click **Plot**.

Follow the instructions below to reproduce the plot of Figure 4.

Temperature Profile

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Temperature Profile` in the **Label** text field.

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

4 In the **Title** text area, type `Temperature profile`.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** check box. In the associated text field, type `Height (cm)`.

7 Select the **Flip the x- and y-axes** check box.

8 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Line Graph 1

1 Right-click **Temperature Profile** and choose **Line Graph**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `15, 17, 19, 21` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

7 In the **Expression** text field, type `z`.

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

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

10 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

11 Click to expand 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
`Temperature (3D approach)`

Temperature Profile

In the **Model Builder** window, click **Temperature Profile**.

Through Thickness 1

1 In the **Temperature Profile** toolbar, click **More Plots** and choose **Through Thickness**.

2 In the **Settings** window for **Through Thickness**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 2 (4) (sol2)**.

4 Locate the **Selection** section. Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `3` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Through Thickness**, locate the **x-Axis Data** section.

8. In the **Expression** text field, type `T2`.
9. From the **Unit** list, choose **degC**.
10. Click to expand the **Title** section. Locate the **y-Axis Data** section. From the **Parameter** list, choose **Expression**.
11. In the **Expression** text field, type `llmat1.th`.
12. Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.
13. From the **Color** list, choose **Red**.
14. Find the **Line markers** subsection. From the **Marker** list, choose **Asterisk**.
15. From the **Positioning** list, choose **Interpolated**.
16. In the **Number** text field, type `15`.
17. Click to expand the **Legends** section. Select the **Show legends** check box.
18. From the **Legends** list, choose **Manual**.
19. In the table, enter the following settings:

Legends
`Temperature (Shell approach)`

20. Click to expand the **Quality** section.

Through Thickness 2

1. Right-click **Through Thickness 1** and choose **Duplicate**.
2. In the **Settings** window for **Through Thickness**, locate the **Selection** section.
3. Click to select the **Activate Selection** toggle button.
4. In the list, select **3**.
5. Click **Clear Selection**.
6. Click **Paste Selection**.
7. In the **Paste Selection** dialog box, type `4` in the **Selection** text field.
8. Click **OK**.
9. In the **Settings** window for **Through Thickness**, locate the **y-Axis Data** section.
10. In the **Expression** text field, type `llmat1.th + 2[cm]`.
11. Locate the **Legends** section. Clear the **Show legends** check box.
12. In the **Temperature Profile** toolbar, click **Plot**.