Created in COMSOL Multiphysics 5.4

Shell Conduction

Introduction

The following example illustrates how to build and solve a model using the Heat Transfer in Thin Shells interface. This example is a 2D NAFEMS benchmark (Ref. 1), which was transformed to 3D.

Model Definition

Figure 1 describes the 2D benchmark example.

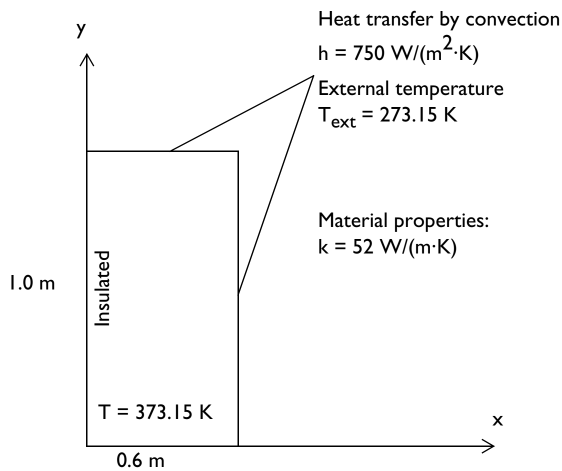

Figure 1: A 2D benchmark example for a thin conductive shell.

The 3D model bends this plate so that it becomes a quarter of a cylinder (Figure 2).

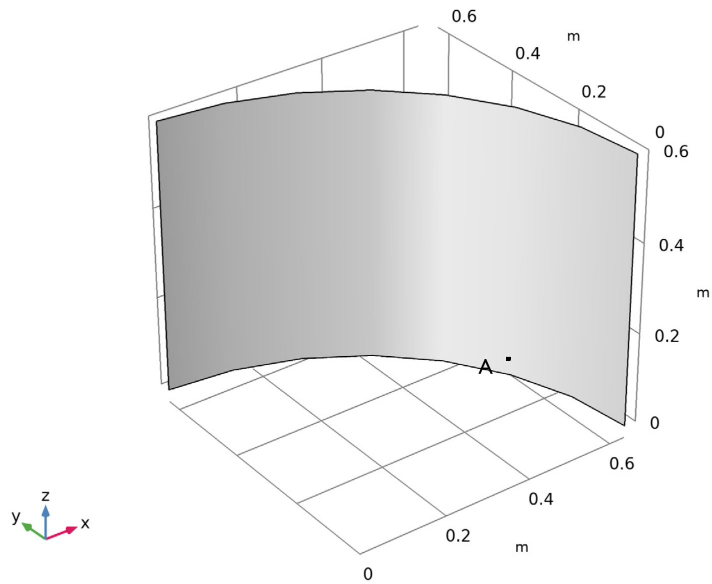

Figure 2: The 3D geometry based on the 2D model.

Results

The temperature at point *A* in Figure 2 (291.40 K) is in agreement with that from the NAFEMS benchmark (Ref. 1). Figure 3 shows the temperature distribution.

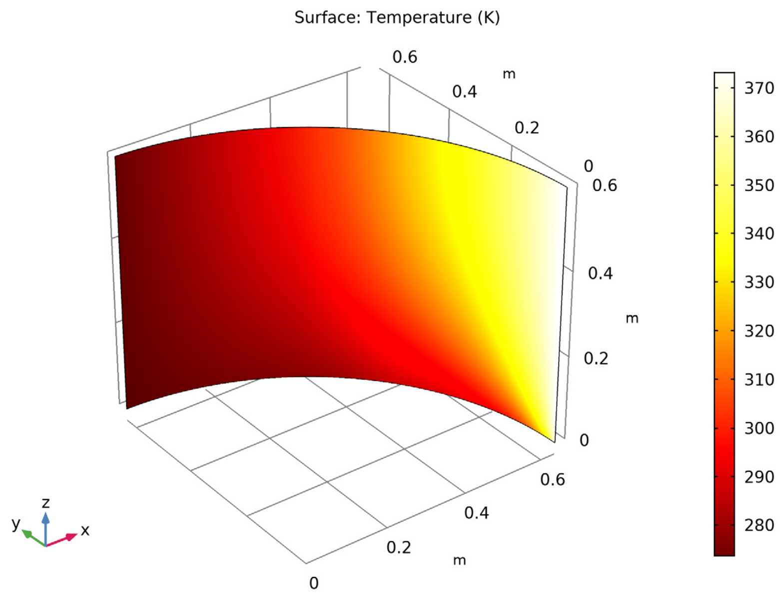

Figure 3: The resulting temperature field of the 3D model.

Reference

1. J.A. Casey and G.B Simpson, “Two-dimensional Steady State,” *Benchmark Tests for Thermal Analysis*, NAFEMS, Test 10, p. 2.9, 1986.

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

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>Thin Structures>Heat Transfer in Shells (htlsh)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

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
T_edge	373.15[K]	373.15 K	Edge temperature
T_ext	273.15[K]	273.15 K	External temperature
ht	750[W/(m^2*K)]	750 W/(m²·K)	Heat transfer coefficient

GEOMETRY 1

Cylinder 1 (cyl1)

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

2 In the **Settings** window for **Cylinder**, locate the **Object Type** section.

3 From the **Type** list, choose **Surface**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type `2/pi`.

5 In the **Height** text field, type `0.6`.

6 Click **Build Selected**.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 On the object **cyl1**, select Boundaries 1–3 only.

3 In the **Settings** window for **Delete Entities**, click **Build Selected**.

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

Point 1 (pt1)

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

2 In the **Settings** window for **Point**, locate the **Point** section.

3 In the **x** text field, type `(2/pi)*cos(pi*18/180)`.

4 In the **y** text field, type `(2/pi)*sin(pi*18/180)`.

This step embeds the point where you compare the calculated solution with the benchmark.

5 Click **Build Selected**.

MATERIALS

Single Layer Material 1 (slmat1)

1 In the **Materials** toolbar, click **Blank Material**.

2 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Layers>Single Layer Material**.

3 In the **Settings** window for **Single Layer Material**, locate the **Layer Definition** section.

4 From the **Material** list, choose **Material 1 (mat1)**.

5 Locate the **Boundary Selection** section. From the **Selection** list, choose **All boundaries**.

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Material 1 (mat1)**.

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
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	52	W/(m·K)	Basic
Density	rho	8800	kg/m³	Basic
Heat capacity at constant pressure	Cp	420	J/(kg·K)	Basic

HEAT TRANSFER IN SHELLS (HTLSH)

Temperature 1

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

2 Select Edge 5 only.

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

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

Heat Flux 1

1 In the **Physics** toolbar, click **Edges** and choose **Heat Flux**.

2 Select Edges 1, 2, and 4 only.

3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

4 Click the **Convective heat flux** button.

5 In the h text field, type `ht`.

6 In the T_{ext} text field, type `T_ext`.

MESH 1

Mapped 1

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

2 Select Boundary 1 only.

3 In the **Settings** window for **Mapped**, click **Build All**.

STUDY 1

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

RESULTS

Temperature (htlsh)

The default plot is the surface plot of the temperature and the arrow plot of the conductive heat flux; compare with Figure 3.

Derived Values

Follow the steps below to obtain the temperature at the benchmark verification point.

Point Evaluation 1

1 In the **Results** toolbar, click **Point Evaluation**.

2 Select Point 3 only.

3 In the **Settings** window for **Point Evaluation**, click **Evaluate**.

The result shown in the **Table** window below the **Graphics** window should be approximately 290.4 K.