Created in COMSOL Multiphysics 6.0

Shell Conduction

This model is licensed under the COMSOL Software License Agreement 6.0.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

The following example illustrates how to build and solve a model using the Heat Transfer in 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.

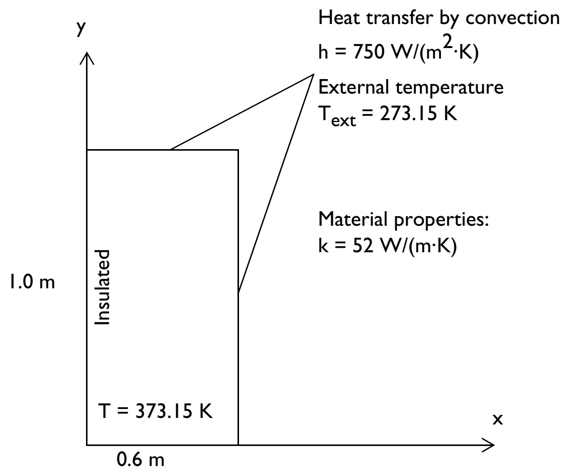


Figure 1: A 2D benchmark example for a thin conductive shell.

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

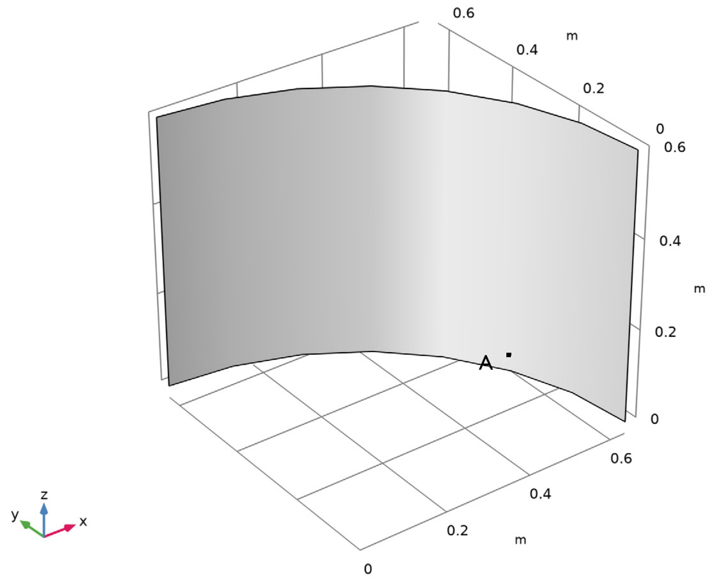


Figure 2: The 3D geometry based on the 2D model.

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.

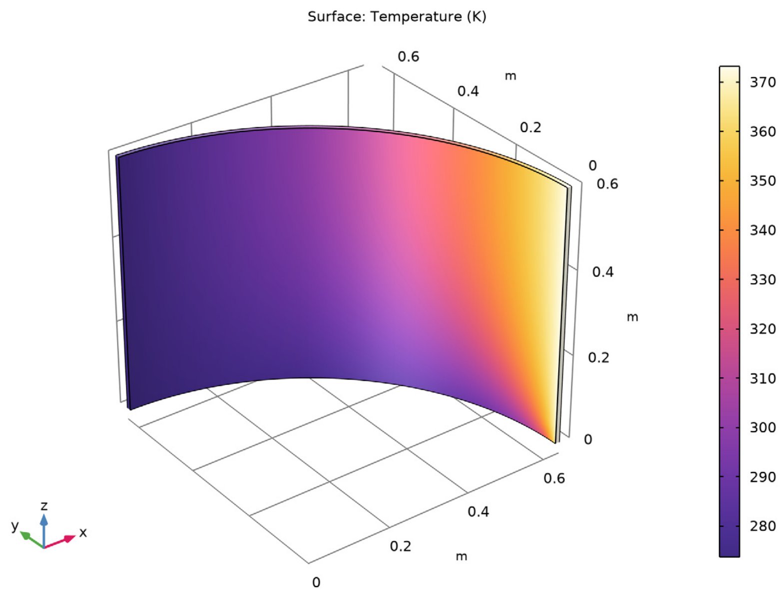


Figure 3: The resulting temperature field of the 3D model.

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

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
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

Material 1 (mat1)

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

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
Thickness	lth	0.01[m]	m	Shell

HEAT TRANSFER IN SHELLS (HTLSH)

Temperature 1

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

2 Select Edge 5 only.

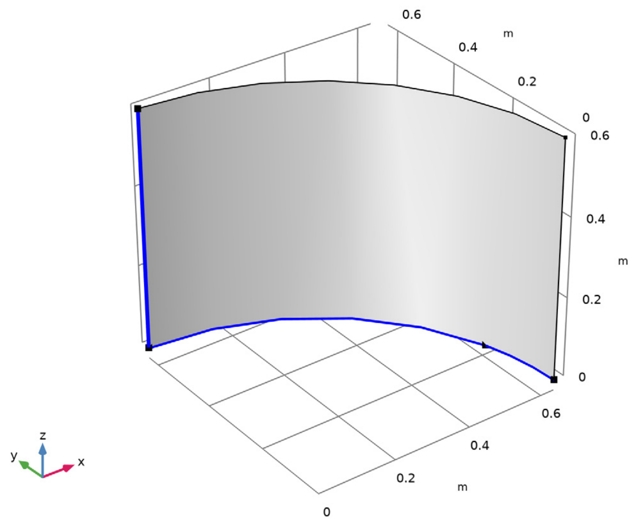
3 In the **Settings** window for **Temperature**, locate the **Temperature** section.

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 From the **Flux type** list, choose **Convective heat flux**.

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. Compare with Figure 3.

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

Point Evaluation 1

1 In the **Results** toolbar, click 8.85 e-12 **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 291.4 K.