



Shell Conduction

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.

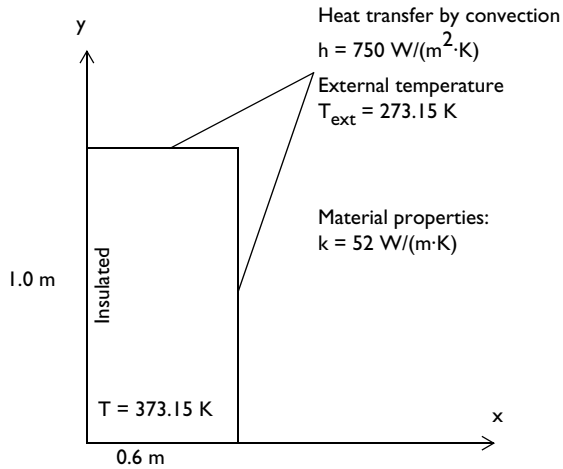


Figure 1: A 2D benchmark example for a thin conductive shell.

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

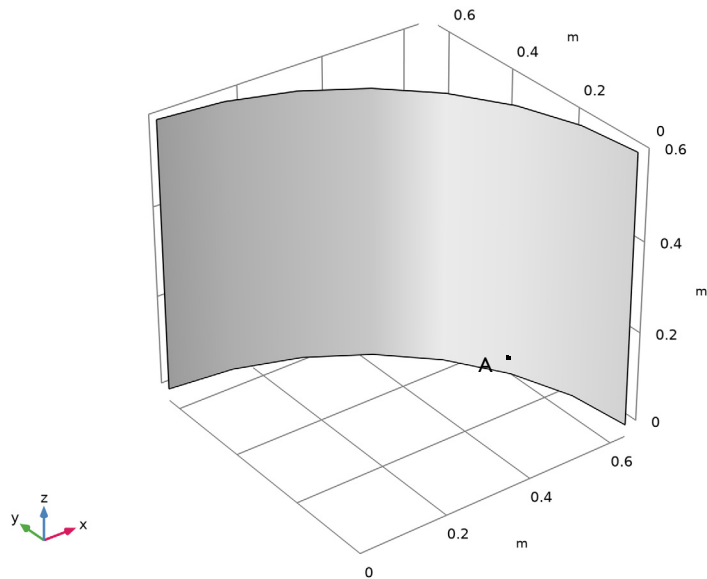


Figure 2: The 3D geometry based on the 2D model.

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.

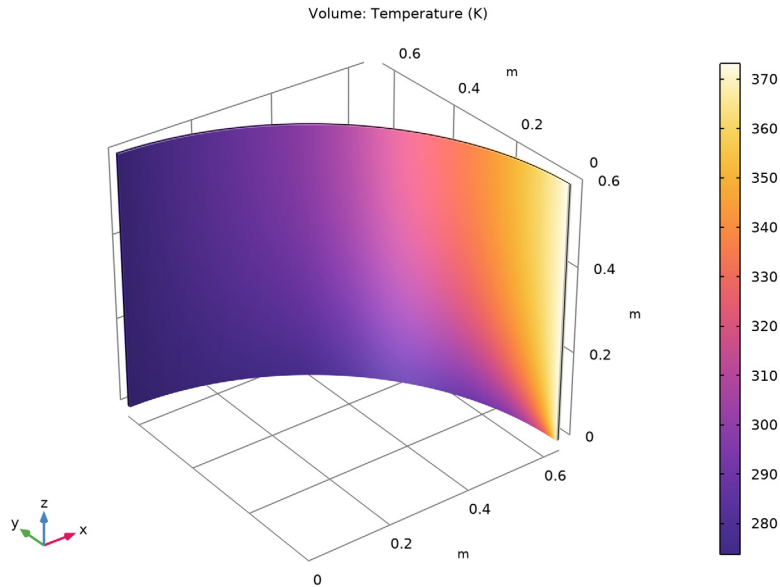


Figure 3: The resulting temperature field of the 3D model.

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 I

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters I**.
- 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/(m^2·K)	Heat transfer coefficient



GEOMETRY I

Cylinder I (cyl)


- 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 I (del)

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

- 2 On the object **cyll**, 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 $(2/\pi) \cdot \cos(\pi \cdot 18/180)$.
- 4 In the **y** text field, type $(2/\pi) \cdot \sin(\pi \cdot 18/180)$.

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} ; $k_{ii} = k_{iso}$, $k_{ij} = 0$	52	W/(m·K)	Basic
Density	ρ	8800	kg/m ³	Basic
Heat capacity at constant pressure	C_p	420	J/(kg·K)	Basic
Thickness	l_{th}	0.01 [m]	m	Shell

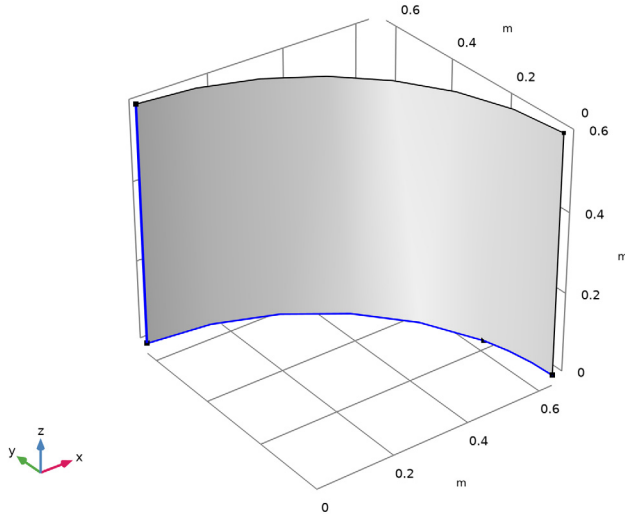
HEAT TRANSFER IN SHELLS (HTLSH)

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 I



- 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 I

Mapped I

- 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 I

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


RESULTS

Temperature, Shell (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 I

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 291.4 K.