Lab 3a

Steady-State One-Dimensional Heat Transfer

Introduction

In this workshop you will become familiar with:

- Defining thermal material properties
- Defining sections and section assignments
- Assigning heat transfer elements to a model

Recall: The general heat transfer equation (for an isotropic material) is:

$$\frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right)+\dot{q}=\rho c_p\frac{\partial T}{\partial t} \qquad \text{Eq3a. 1}$$

where $T(x,y,z)$ is temperature distribution, $\dot{q}$ is the rate at which energy is generated per unit volume of the medium (W/m^3), $\rho c_p\frac{\partial T}{\partial t}$ is the time rate of change of the sensible (thermal) energy of the medium per unit volume , ρ and c_p are density and specific heat respectively.

If the heat transfer is one-dimensional (e.g., in the *x*-direction), under steady-state conditions and there is no energy generation, hence Eq3a.1 reduces to

$$\frac{d}{dx}\left(k\frac{dT}{dx}\right)=0 \qquad \text{Eq3b. 2}$$

The heat flux, $q''\left(\frac{W}{m^2}\right)=-k\frac{dT}{dx}=\ -k\frac{T_2-T_1}{L}=-\frac{T_2-T_1}{\frac{L}{k}}$

Consider steady-state conditions for one-dimensional conduction in a plane glass having a thermal conductivity k =1.4W/m/C and a thickness L= 0.15 m, with no internal heat generation (Figure L3a–1). Determine the heat flux, the temperature distribution, and indicate the direction of the heat flux. The units used in this model are SI (kg, m, s, N, ºC).

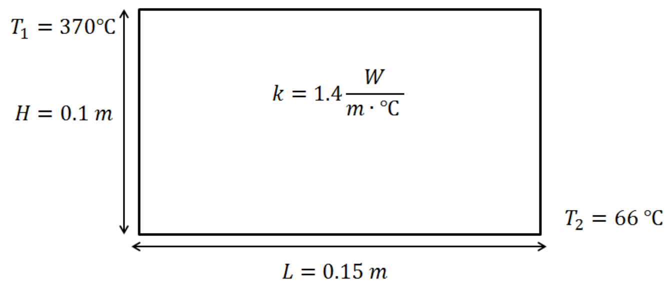


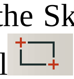
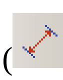
Figure L3a–1. The glass plate heat transfer model.

The Figure L3a-1 is a plane glass with properties as defined earlier. This model will help us understand that if two ends of a glass plate are maintained at different temperatures then how will heat flow between them so as to achieve thermal equilibrium. Heat flux is one such quantitative measure that indicates the heat flow through a surface. We know that without the effect of any external influence, such spontaneous heat transfer always occurs from a region of high temperature to another region of lower temperature, as described by the second law of thermodynamics.

Defining the model geometry

1. Start a new session of Abaqus/CAE
2. In the **Start Session** dialog box, underneath **Create Model Database**, click **With Standard/Explicit Model**.
3. A default model name is assigned by Abaqus/CAE (**Model-1**). In the Model Tree, click mouse button 3 on **Model-1** and select **Rename** from the menu that appears. Rename the model **`Heat Transfer`**.

To create the plate geometry:

1. Create a **two-dimensional**, deformable body with a planar shell base feature. Name the part **`Plate`**, and specify an approximate part size of **`0.5`**.
2. In the Sketcher, create an arbitrary rectangle using the **Create Lines: Rectangle** tool .
3. Dimension the left edge () by selecting the line and assign it a value of **`0.1`** m.
4. Similar to left edge, dimension the bottom edge a value of **`0.15`** m.
5. In the prompt area, click **Done** to finish the sketch.

Defining the material, and assigning the section properties

The plate is made of an isotropic, linear elastic material with conductivity = 1.4 W/m/ ºC.

1. Create the material definition; name the material as **`Glass`**.
2. From the material editor's menu bar, select **Thermal→Conductivity** and enter a thermal conductivity value of **`1.4`**.

To create and assign section properties

Next, you will create a homogeneous solid section and assign it to the plate. The section will refer to the material **Glass** that you just created.

1. Define a homogeneous solid section named **`GlassSection`**. Choose **`Glass`** as the material definition associated with the section.
2. Assign the section definition to the plate. (Tip: Use )

Creating an assembly, and defining an analysis step

The assembly for this analysis consists of a single instance of the part **`Plate`**.

To create an instance of the plate:

1. In the Model Tree, expand the branch for the **Assembly** of the **Heat Transfer** model and double-click **Instances** to create a dependent instance of the part named **Plate**.
2. In the Model Tree, expand the **Assembly** container and double-click **Sets** to create a geometry set for the left edge named **`left`**. Similarly, create another set for the right edge of the plate and name it **`right`**.

To create a Heat transfer analysis step:

In this analysis we are interested in the steady-state heat transfer of the glass plate.

1. In the **Module** list located in the context bar, select **Step** to enter the Step module.
2. From the main menu bar, select **Step→Create** to create a step.
3. In the **Create Step** dialog box that appears:
    a. Name the step **`Heat Transfer`**.
    b. From the list of available general procedures in the **Create Step** dialog box, select **Heat transfer**.
    c. Click **Continue**.
4. In the **Edit Step** dialog box that appears:
    a. In the **Description** field of the **Basic** tabbed page, enter **`heat transfer with convection`**.
    b. In the **Response**, toggle on **Steady-state**. (Click **Dismiss** on the box: "Default load variation with time has been changed to ramp linearly over step"
    c. Click the **Incrementation** and **Other** tab, and accept the default values provided for the step.
    d. Click **OK** to create the step and to exit the step editor.

Defining thermal Boundary Conditions

Next, you will assign a fixed temperature = 66 °C and 370 °C to the right and left edges of the plate, respectively.

1. In the Model Tree, double-click **BCs**. Define the right edge temperature boundary condition in **`Heat Transfer`** step so that the set **`right`** has a fixed temperature of **`66`**.
2. Similarly, define the left edge temperature boundary condition in the **`Heat Transfer`** step so that the set **`left`** has a fixed temperature of **`370`**.

Creating the mesh and defining a job

To assign an Abaqus element type:

1. In the **Module** list located in the context bar, select **Mesh** to enter the Mesh module. (make sure your select **Part** as Object)
2. From the main menu bar, select **Mesh→Element Type**.
3. In the **Element Type** dialog box that appears, select **Heat Transfer**.

   A description of the element type DC2D4 appears at the bottom of the dialog box. Abaqus/CAE will now mesh the part with DC2D4 elements.
4. Click **OK** to assign the element type and to close the dialog box.

To mesh the model:

1. Seed the part using a global element size of **`0.01`**.
2. From the main menu bar, select **Mesh→Part** to mesh the part and create mesh for the part.

Submit the analysis job

1. In the Model Tree, double-click **Jobs** to create a job named **`HeatTransfer`** for the model **`HeatTransfer`** and submit the job for analysis.

Post processing the thermal analysis results

After the analysis has run to completion, study the analysis results.

1. Click **Results** in the **Job Manager** to enter the Visualization module.

   Abaqus/CAE switches to the Visualization module, and opens the output database created by the job (**`HeatTransfer.odb`**).

To generate a contour plots:

2. In the toolbox, click  (or select **Plot→Contours→On Deformed Shape** from the main menu bar) to view a contour plot of the heat flux (HFL), as shown in Figure L3a–2.

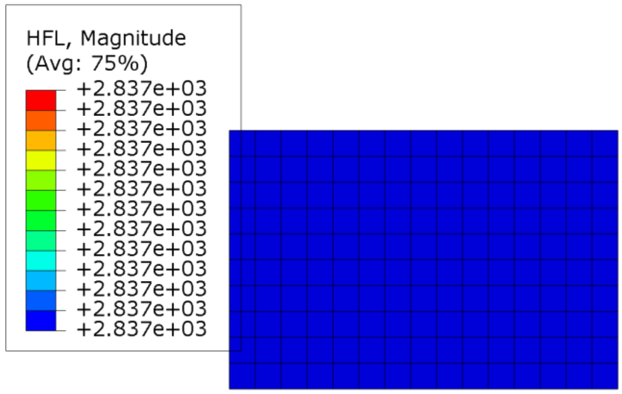
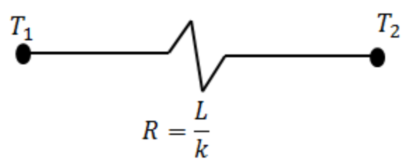


Figure L3a–2. Heat flux contour plot.

Recall that:



$$q''(Heat\ Flux) = \frac{T_1 - T_2}{\frac{L}{k}} = \frac{370 - 66}{\frac{0.15}{1.4}} = 2837.33\ \mathrm{W/m^2}$$

Figure L3a–3. Heat flux calculation using thermal circuit method.

3. From the list of available output variables in the center of the toolbar, select output variable **NT11** (spatial displacement at nodes).

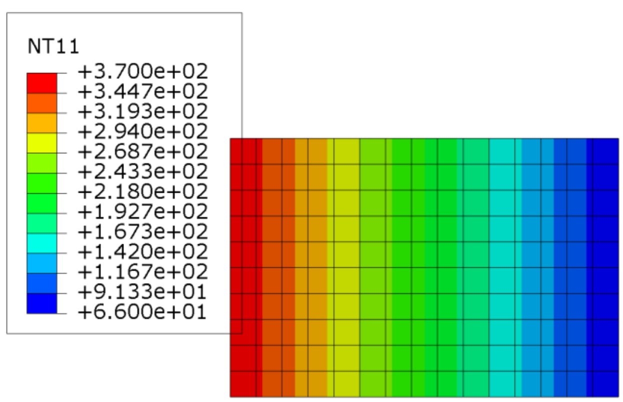
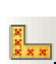


Figure L3a–4. Steady-state temperature distribution.

4. In the toolbox, click  to create a vector plot (alternatively select **Plot→Symbols→On Undeformed Shape**). The plot appears as shown in Figure L3a–5.

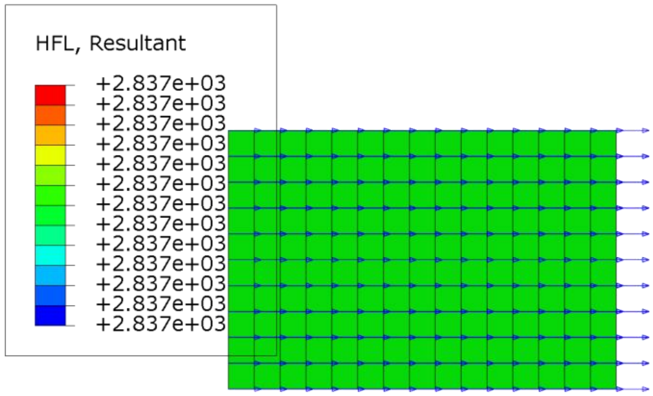


Figure L3a–5. Heat Flux vector plot.

5. Create a path plot of the temperature along the thickness. Follow the steps given below:
    a. In the Results Tree, double-click **Paths**.
       The **Create Path** dialog box appears.
    b. The **Node list** type is selected by default. Click **Continue** in the dialog box.
    c. In the **Edit Node List Path** dialog box, click **Add After**.
    d. You will be prompted to select the nodes from the viewport to be inserted into the path. Select the nodes as shown in Figure L3a–6. Click **Done** in the prompt area when the selection is complete.
    e. Click **OK** in the **Edit Node List Path** dialog box.

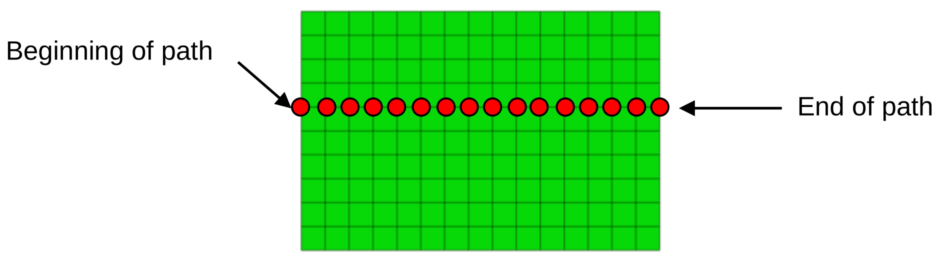


Figure L3a–6. Node path.

    f. In the Results Tree, double-click **XY Data**. Select **Path** in the **Create XY Data** dialog box and click **Continue**.
    g. Browse the settings in the **XY Data from Path** dialog box. In the **Y Values** frame, click **Step/Frame**.
    h. In the **Step/Frame** dialog box, select the last frame of the step named **HeatTransfer**. Click **OK**.
    i. In the **XY Data from Path** dialog box, make sure that **Field output variable** is set to **NT11** and click **Plot** to view the path plot.
    j. Save the **X–Y** data as **`Temperature`**.

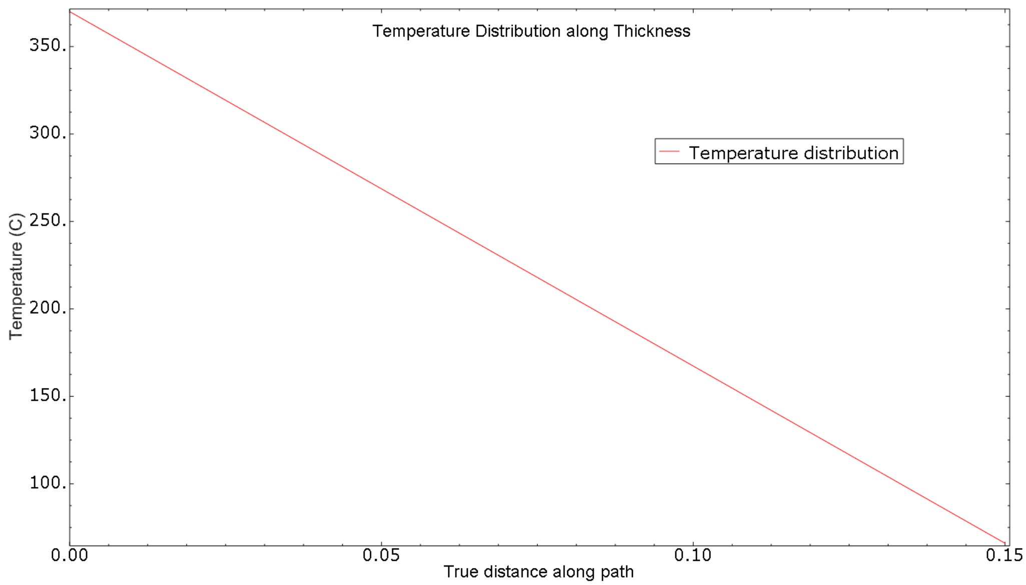
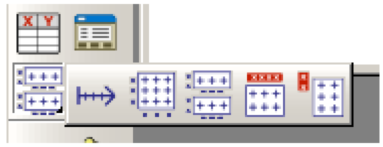
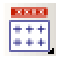


Figure L3a–7. Temperature distribution along the path.

Customize an *X–Y* plot:

1. Double-click anywhere on the chart to open the **Chart Options** dialog box.
    - In the **Grid Display** tabbed page, toggle on the major *X*- and *Y*- grid lines. Set the line color to red.
    - Change the background to white.
    - In the **Grid Area** tabbed page, select **Square** as the size. From the list of auto-alignments, choose the one that places the chart in the center of the viewport ().
2. Double-click the legend to open the **Chart Legend Options** dialog box.
    - In the **Contents** tabbed page, click  to increase the legend text font size to `14`.
    - In the **Area** tabbed page, toggle on **Inset**.
    - Toggle on **Fill** to flood the legend with a white background.
    - In the viewport, drag the legend over the chart.
3. Double-click either axis to open the **Axis Options** dialog box.
    - In the **X Axis** region of the dialog box, increase the number of *X*-axis minor tick marks per increment to **8**.
    - In the **Y Axis** region of the dialog box, in the **Scale** tabbed page, specify that the *Y*-axis should extend from `66` (the *Y*-axis minimum) to `370` (the *Y*-axis maximum).
    - Increase the number of *Y*-axis minor tick marks per increment to **3**.
    - In the **Title** tabbed page, change the *Y*-axis title to `Temperature (C)`.
    - In the **Axes** tabbed page, change the font size for both axes to 16.
4. Expand the list of plot option icons in the toolbox:



5. Examine the remaining options. Select  to open the **Plot Title Options** dialog box.
    - In the **Title** tabbed page, enter the following plot title: **`Temperature Distribution along Thickness`**; click  to change the legend text style to bold.
    - In the **Area** tabbed page, toggle on **Inset**.
    - In the viewport, drag the plot title above the chart.
6. Click  in the toolbox to open the **Curve Options** dialog box. Change the legend text for the Temperature curves to **Temperature distribution**. Change the thickness of the curve to use the second-thickest option.
7. Reposition the legend as necessary.