

Lab 3b

Steady-State Composite Wall

Introduction

Equivalent thermal circuits may also be used for more complex systems, such as *composite walls*. Such walls may involve any number of series and parallel thermal resistances due to layers of different materials. Consider the series composite wall of Figure L3b-1. The one dimensional heat transfer rate for this system may be expressed as

$$q'' = \frac{T_{\infty,1} - T_{\infty,2}}{R_{total}}$$

where

$$R_{total} = \frac{1}{h} + \frac{L_1}{k_1} + \frac{L_2}{k_2} + \frac{L_3}{k_3} + \frac{1}{h}$$

A simple steady-state composite wall conduction problem is solved through the thickness of a heat transfer shell model. A thermal circuit can be set up to represent the wall and its environment so that the expected temperature distribution through the wall can be calculated. For the problem described by the figure below, determine T_1 , T_2 , T_3 , and T_4 , and the heat flux.

The units used in this model are SI (kg, m, s, N, °C).

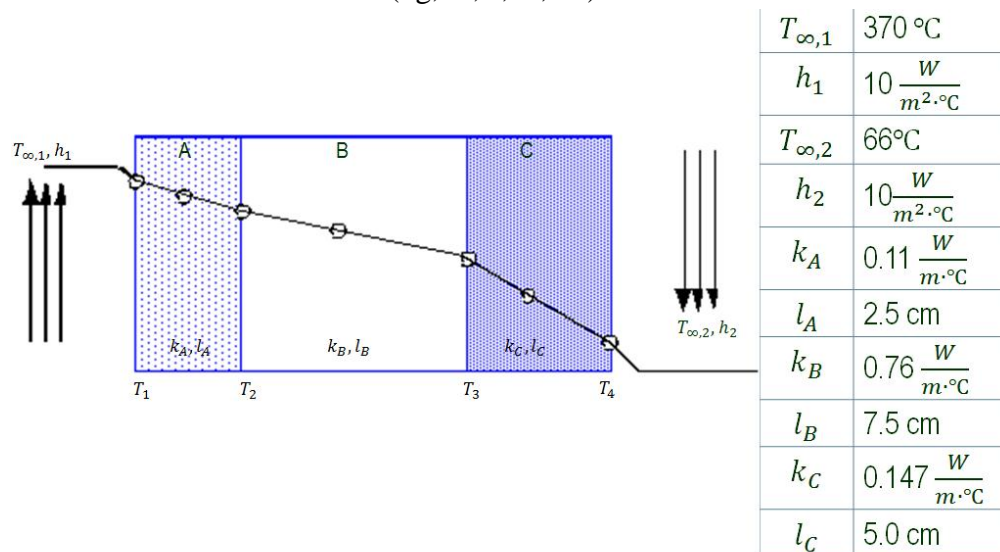



Figure L3b–1. A simple steady-state composite wall conduction problem.

Geometry and model

1. In the Model Tree, click mouse button 3 on the model **Heat Transfer** and select **Copy Model** from the menu that appears.
2. In the **Copy Model** dialog box, enter **Composite Wall** as the new model name. Click **OK**.
3. In the Model Tree, expand the container under the model **Composite Wall**.
4. In the Part module, click the **Partition Face: Sketch** icon  and sketch two vertical lines with distances 0.025 m and 0.01 m from left edge, as shown in Figure W1-2.
5. Click **Done**.

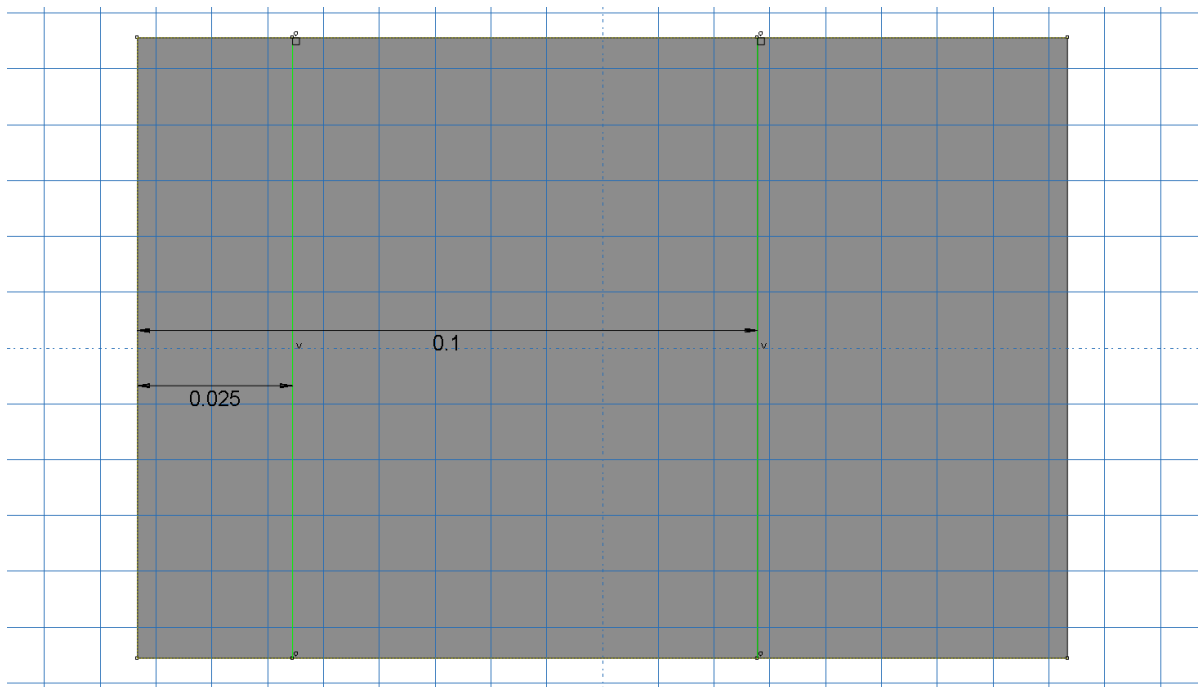




Figure L3b-2. Partition the part.

Material definition, and assigning the section properties

Define the material properties of the material-A, material-B, and material-C.

1. Create the material definition; name the material **Material-A**.
2. From the material editor's menu bar, select **Thermal> Conductivity** and enter the thermal conductivity value of **0.11**.
3. Similarly, create **Material-B** (enter a thermal conductivity value of **0.76**) and **Material-C** (enter a thermal conductivity value of **0.147**).
4. Click  in the toolbox to create a solid homogeneous section named **Section-A**. Associate the section with the material.

5. Similarly, define two other homogeneous solid sections named **Section-B** and **Section-C**
6. From the main menu bar, select **Section> Assignment Manager** and delete **GlassSection**.
7. Click on section Assignment Manager  in the toolbox to assign sections to each partition based on Figure L3b-1.

Creating surfaces

1. In the Model Tree, expand the **Assembly** container and double-click **Surfaces** to create a geometry set for the left edge named **left**. Similarly, create another surface for the right edge of the plate and name it **right**.

Defining thermal loads

1. Define a film condition on the left and right surfaces that are exposed to air.
 - a. In the Model Tree, double-click **Interactions**.
Abaqus/CAE switches to the Interaction module, and the **Create Interaction** dialog box appears.
 - b. Name the interaction **convection_right**, select **Heat Transfer** as the step and **Surface film condition** as the type for the selected step, and click **Continue**.
 - c. When prompted to select the surface, click **Surfaces** in the prompt area. From the list in the **Region Selection** dialog box that appears, select the surface **right**. Toggle on **Highlight selections in viewport** to view the selection. Click **Continue**.
 - d. In the interaction editor that appears, enter a value of **10** for the film coefficient and **66** for the sink temperature, and select **Ramp** as the sink amplitude (see Figure L3b-3). Click **OK**.

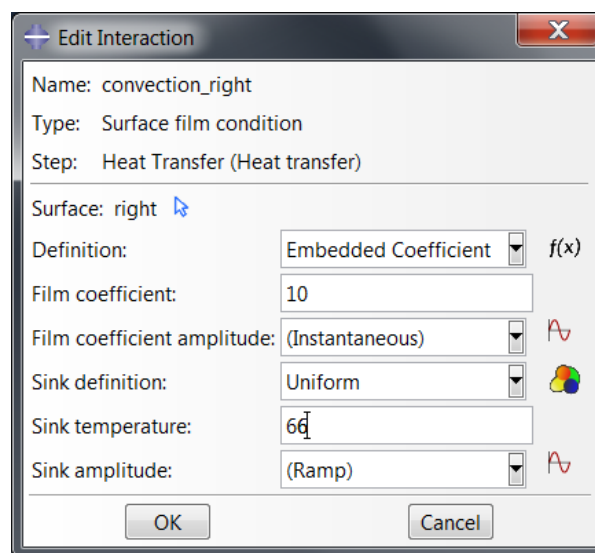


Figure L3b-3. Interaction editor associated with convection

2. Similarly, define an interaction **convection_left**, for surface **left** with a value of **10** for the film coefficient and **370** for the sink temperature

Creating the mesh and defining a job

The element type as defined earlier remains of the type Heat Transfer. But since we made changes to the part geometry, the mesh associated with the geometry was deleted.

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 **CompositeWall** for the model **CompositeWall** and submit the job for analysis.