Lab 3c

Transient Two-Dimensional Heat Transfer with Convection

Introduction

In this work shop we will investigate transient two-dimensional heat transfer with convection of a plate shown in Figure 3c–1.

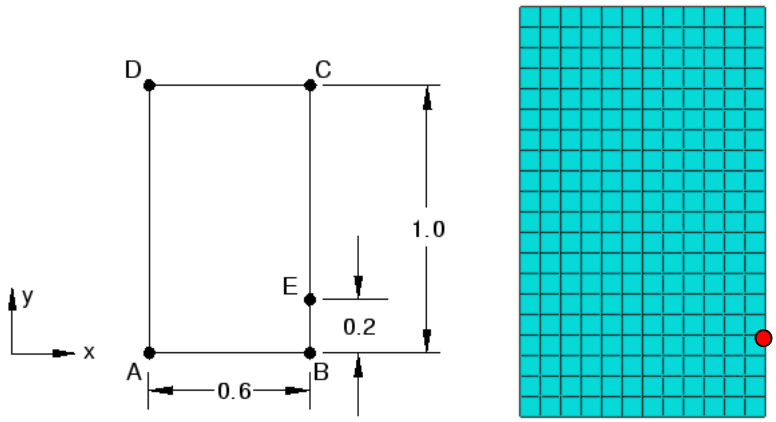

Figure L3c–1. The plate model (target node indicated).

Consider a steel plate with an initial temperature = 0 °C where the bottom edge of plate has a fixed temperature = 100 °C, and the top and right edges of the plate exposed to fluid flow with a film coefficient of 750 W/m^2/°C and a sink temperature of 0 °C. We want to investigate the temperature destitution of the plate over time, and calculate the steady-state temperature at point E.

Defining the model geometry and Create the mesh

1. Create a two-dimensional, deformable body with a planar shell base feature. Name the part `Plate`, and specify an approximate part size of `2.0`.
2. Create a rectangular with dimensions as shown in Figure 3c.1
3. In the Model Tree, expand the **plate** item underneath the **Parts** container and double-click **Mesh** in the list that appears.
4. Seed the part using a global element size of `0.05.`
5. Change the element type to DC2D4. (**Element Type** select **Heat Transfer**)
6. **Mesh→Part**

Defining the material, and assigning the section properties

1. Create a linear material (named **steel** with the following material properties:
   - Density = 7832 kg/m^3
   - Specific heat = 434 J/kg/°C
   - Conductivity = 52 W/m/°C
2. Define a homogeneous solid section named **`SteelSection`**. Choose **`Steel`** as the material definition associated with the section.
3. Assign the section to the plate

Creating an assembly

1. Create a dependent instance of the part named **Plate**.
2. Create sets and surfaces as indicated below
   - **`RefPoint`**: set (**Type**: **Node**, make sure the part is meshed before this step), select point E as shown in Figure 3c-1
   - **`bottom`**: set along bottom edge
   - **`top`**: surface along top edge
   - **`side`**: surface along right edge

Defining an analysis step

4. Create a single heat transfer step as indicated below (Figure 3c–2):
   In the **Response** tab
   - Response type = transient
   - Time period = **15000** sec

   In the **Incrementation** tab
   - Initial = minimum increment size = **30** sec
   - Maximum increment size = **500** sec
   - End step when temperature change is less than : **0.0005** °C /sec
   - Maximum allowable temperature change per increment = **30** °C
5. In the Model Tree, double-click **History Output Requests** to add a history output request for the target node. Select **Set** as the applicable domain and **RefPoint** as the set. Toggle on **Thermal** in the **Output Variables** region. Select Nodal temperature (**NT**).

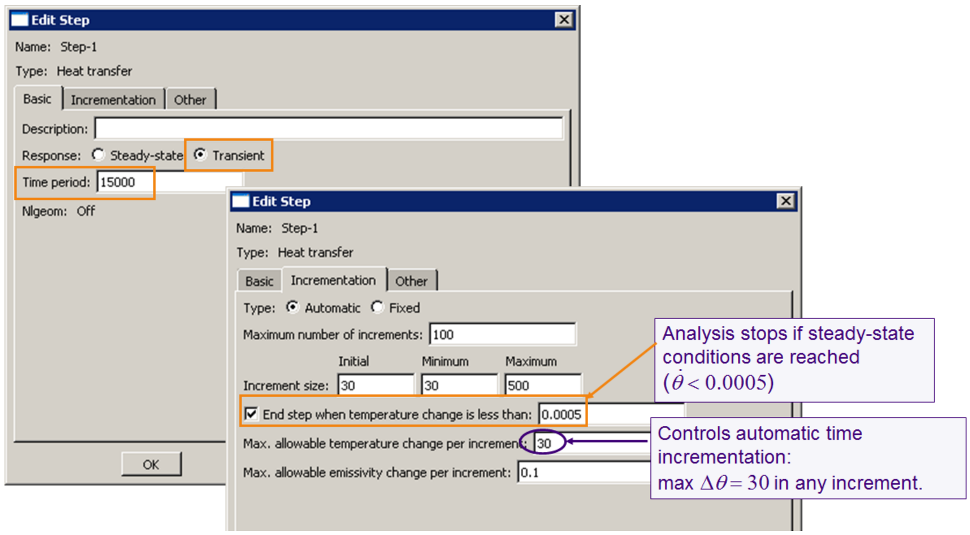

Figure L3c–2. Define a transient step.

Defining thermal Boundary Conditions

6. Assign a fixed temperature = **100** °C to the bottom edge of the plate.
7. Assign initial temperature = **0** °C to the whole model. (Tip: In the Model Tree, double-click **Predefined Fields**.)
8. Create two film interactions (one on the right edge and one on the top edge of the plate). Specify a film coefficient of **750** W/m^2/°C and a sink temperature of **0** °C for each.

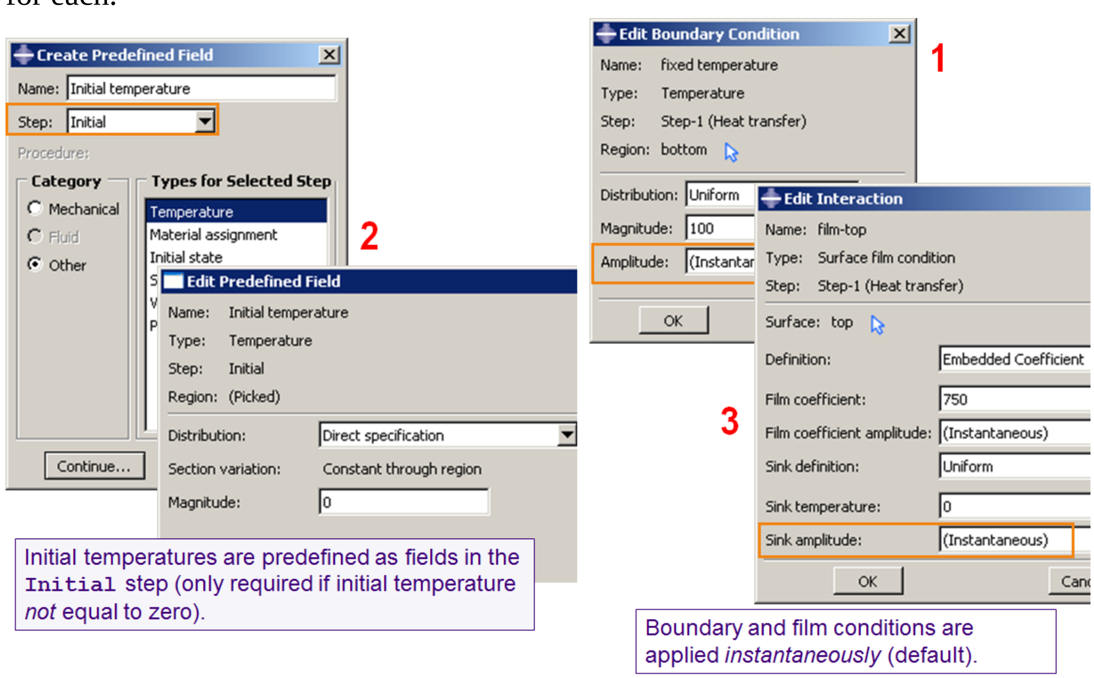

Figure L3c–3. Define boundary condition (Convection) for top and side edge.

Submit the analysis job

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