Heat Transfer Analysis

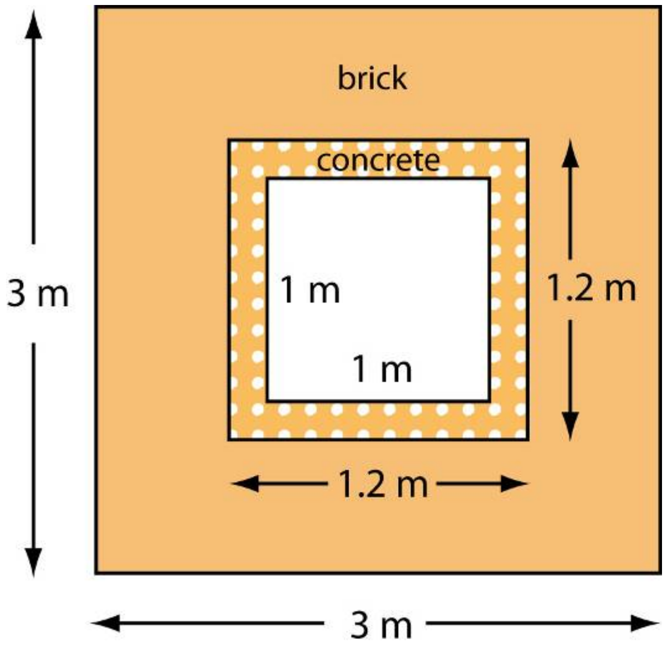
Type of solver: ABAQUS CAE/Standard

(A) Two-Dimensional Steady-State Problem – Heat Transfer through Two Walls

Problem Description:

The figure below depicts the cross-sectional view of a furnace constructed from two materials. The inner wall is made of concrete with a thermal conductivity of k_c = 0.01 W m^{-1} K^{-1}. The outer wall is made of bricks with a thermal conductivity of k_b = 0.0057 W m^{-1} K^{-1}. The temperature in the furnace is at 1273 K and the convective heat transfer coefficient is h_1 = 0.208 W m^{-2} K^{-1}. The outer brick wall comes into contact with the ambient air, which is at 293 K, and the corresponding convective heat transfer coefficient is h_2 = 0.068 W m^{-2} K^{-1}.

Formulate a 2-D FE model and solve for (i) the temperature distribution within the concrete and brick walls at steady-state conditions, and (ii) the heat flux across the walls.

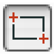


SOLUTION:

- Start ABAQUS/CAE. At the **Start Session** dialog box, click **Create Model Database**.
- From the main menu bar, select **Model→Create**. The **Edit Model Attributes** dialog box appears, name the model `2D_Walls`

A. MODULE → PART

Under the Part module, we will construct the two parts (i.e. walls): (i) Brick and (ii) Concrete

1. From the main menu bar, select **Part→Create**
2. The **Create Part** dialog box appears. Name the part `Brick` and fill in the rest of the options as in **Fig.A1**. Click **Continue** to create the part.
3. There are several ways of constructing the brick wall geometry. *One way to do this is demonstrated here:*
   (a) From the Sketcher toolbox, select the **Create Isolated Point** tool , then type in coordinates of the four key vertices (0, 0), (0.9, 0.9), (2.1, 2.1) and (3, 3). If not all plotted points are visible, press the **Auto Fit View** button  located on the toolbar.
   (b) From the Sketcher toolbox, select the **Create Lines: Rectangle** tool  and connect the inner and outer pairs of vertices to form two squares, as shown in **Fig.A2**.
   (c) Click on **Done** in the prompt area.

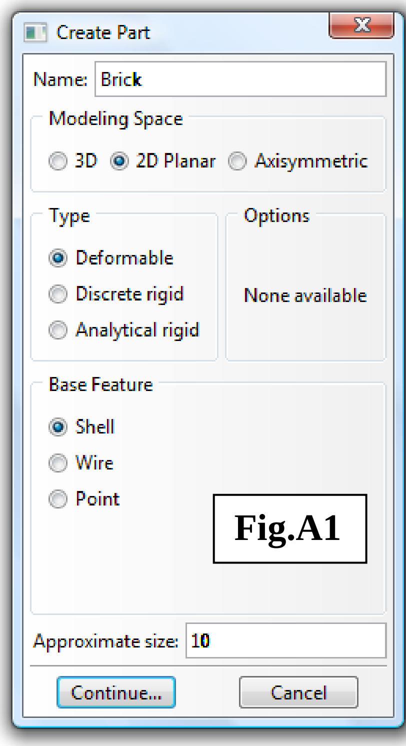


Fig.A1

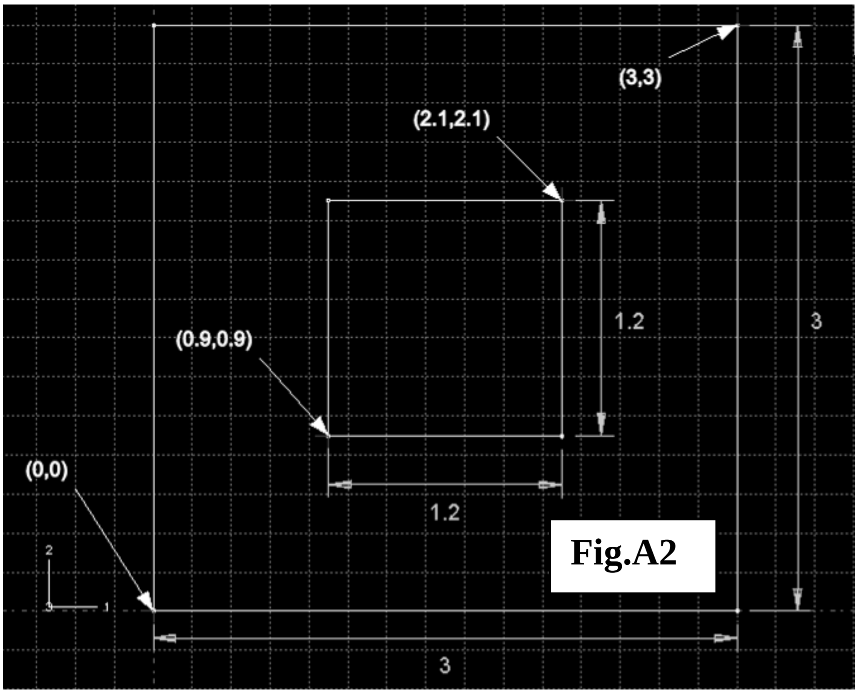


Fig.A2

4. Now construct the second part by following procedures similar to the ones outlined above. Name the new part `Concrete`. The four key vertices are (0, 0), (0.1, 0.1), (1.1, 1.1) and (1.2, 1.2).

B. MODULE → PROPERTY

(a) To define the materials:-

1. From the main menu bar, select **Material→Create**
2. The **Edit Material** dialog box appears (see **Fig.B1)**. Name it `Material-brick`. Select **Thermal→Conductivity** and enter a value of `0.0057`.
3. Click **OK**.
4. Now create `Material-concrete`. Enter a value of 0.01 as its thermal conductivity.

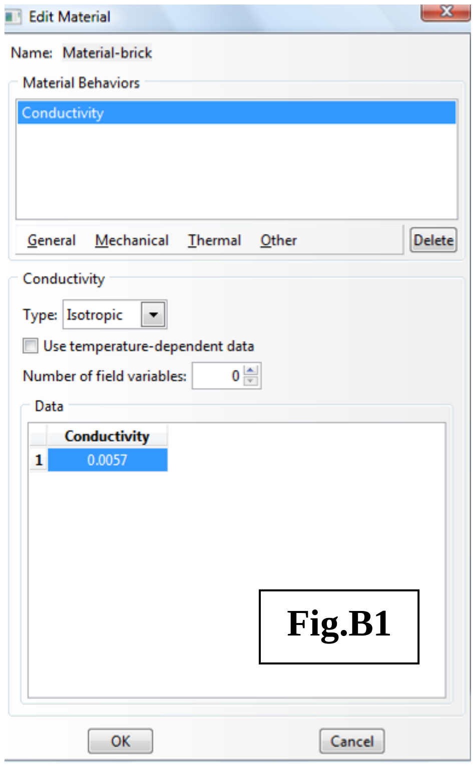


Fig.B1

(b) To define the sections:-

1. From the main menu bar, select **Section→Create**
2. The **Create Section** dialog box appears (**Fig.B2**). Name it `Section-brick`. In the **Category** list, accept **Solid** as the default selection. In the **Type** list, accept **Homogeneous** as the default selection, and click **Continue**.
3. The **section editor** appears (**Fig.B3**). Click the arrow next to the **Material** text box and choose `Material-brick`. Accept the default value for **Plane stress/strain thickness**, and click **OK.**
4. Now define `Section-concrete.`

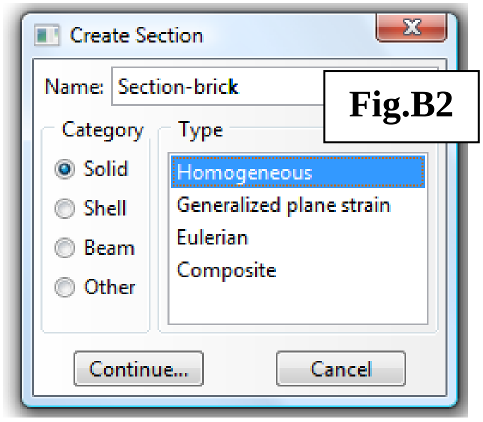


Fig.B2

(c) To assign a section to a part:-

1. From the main menu bar, select **Assign→Section**
2. Click on the Brick region and then click **Done**.
3. The **Edit Section Assignment** dialog box appears containing a list of existing sections, Click the arrow next to the **Section** text box and choose `Section-brick,`

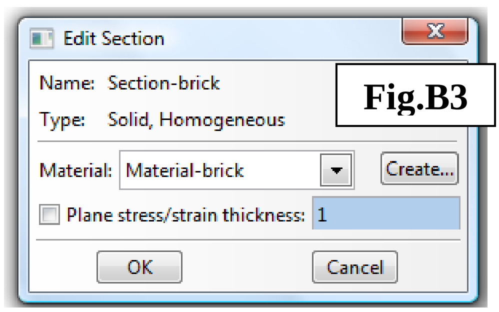


Fig.B3

and click **OK**.

Note: the colour of a part becomes *aqua* when it has been assigned a section.

5. Now assign `Section-concrete` to the concrete region.

C. MODULE → ASSEMBLY

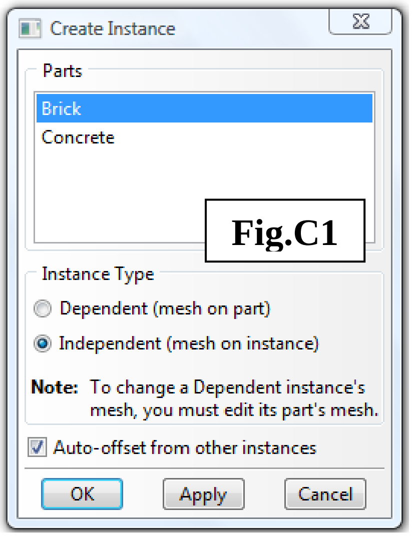


Fig.C1

1. From the main menu bar, select **Instance→Create**
2. The **Create Instance** dialog box appears (**Fig.C1**). Under **Parts**, select `Brick`. For **Instance Type**, choose **Independent (mesh on instance)**. Toggle on **Auto-offset from other instances**. Click **OK**.
3. Now create an instance for the part `Concrete`.
4. At this point, before we proceed onto assembling the instances, it would be useful to define several sets of surfaces for use in later stages of the analysis. From the main menu bar, select **Tools→Surface→Create**. The **Create Surface** dialog box appears. Name it `Brick-inside` and pick the four edges located inside the `Brick` instance, see **Fig.C2** (*Note:* you may need to press and hold the *Shift*-key to make multiple selections). Click **Done** in the prompt area. Repeat to create another set of surface called `Brick-outside`, consisting of four edges located outside the Brick instance, see **Fig.C2**.
5. Now create the following surfaces on the Concrete instance, name them: `Concrete-inside` and `Concrete-outside`, corresponding to the four inner and outer edges of the Concrete instance, as depicted in **Fig.C2**.

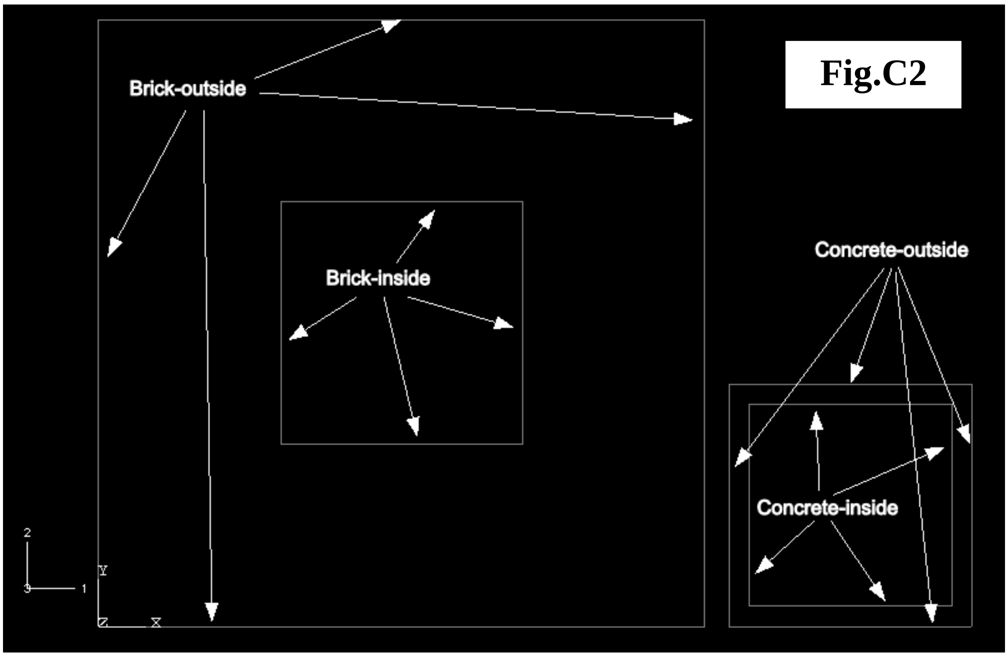


Fig.C2

6. We'll now assemble the two instances. From the main menu bar, select **Instance→Translate.** Select the `Concrete` instance and click **Done**. By picking the suitable start and end points for the translation vector, position the smaller concrete wall within the larger brick wall, so that the final assembly resembles **Fig.C3**.

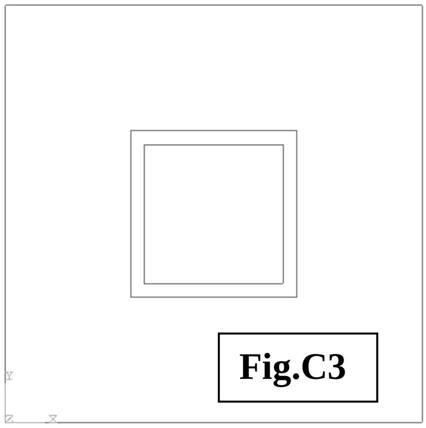


Fig.C3

D. MODULE → STEP

1. From the main menu bar, select **Step→Create**
2. The **Create Step** dialog box appears (**Fig.D1**), name it `Heating`, and select **Heat transfer** under **Procedure type**. Click **Continue**.
3. The **Edit Step** dialog box appears. Under the **Basic** tab, toggle on **Steady-state**, click **OK**.
4. From the main menu bar, select **Output→History Output Requests→Create**, accept the default name `H-Output-1`, the **Edit History Output** dialogue box appears, expand the **Thermal** button and toggle on **FTEMP**. Click **OK**.

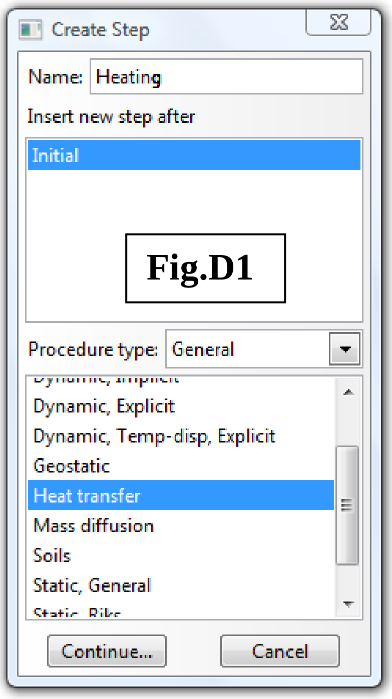


Fig.D1

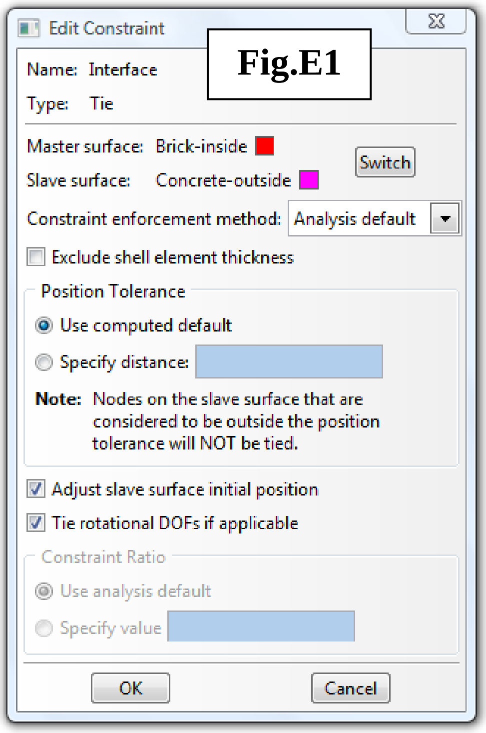
E. MODULE → INTERACTION

(a)To tie the nodes at the interfaces:-

1. From the main menu bar, select **Constraint→Create**
2. The **Create Constraint** dialog box appears, name it `Interface` and under **Type** pick **Tie**. Click **Continue**.

   Note: since we assume there is no thermal resistance across the brick-concrete wall interface, the **Tie** constraint will equate temperatures at the matching nodes.

3. In the prompt area, choose the master type as **Surface**, click on the Surfaces... button at the lower right hand corner of the prompt area. Select `Brick-inside` and click **Continue**. Click the **Surface** button in the prompt area and select `Concrete-outside` as the slave surface. Click **Continue**.



4. The **Edit Constraint** dialog box appears (**Fig.E1**), accept the default settings and click **OK**.

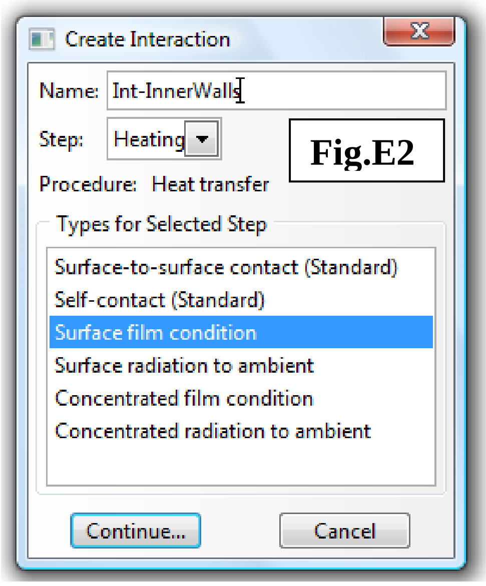
(b) To assign convective heat transfer conditions:-

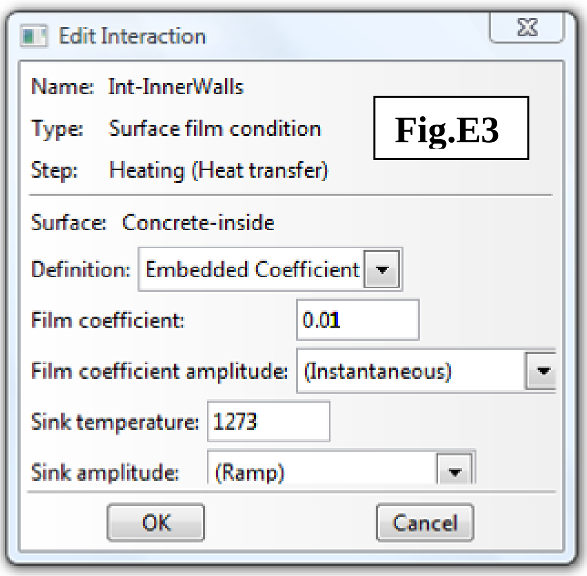
1. From the main menu bar, select **Interaction→Create**
2. The **Create Interaction** dialog box appears (**Fig.E2**), name it `Int-InnerWalls`. Under **Step**, choose `Heating`. For **Types for Selected Step**, choose **Surface film condition**, click **Continue**. In the **Region Selection** dialog box, select the surface defined earlier as `Concrete-inside` and click **Continue**.

   Note: if the **Region Selection** dialog box does not appear, click on the Surfaces... button at the bottom right hand corner of the prompt area.

3. The **Edit Interaction** dialog box appears (**Fig.E3**), enter `0.208` (W m^{-2} K^{-1}) as the **Film coefficient**, and `1273` (K) as the **Sink temperature**.

4. Now create surface film condition for the brick walls that are in contact with the ambient air, name it `Int-OuterWalls`. Apply it to the surface called `Brick-outside.` Enter `0.068` (W m^{-2} K^{-1}) as the **Film coefficient**, and `293` (K) as the **Sink temperature**.





F. MODULE → MESH

(a) To seed the part instance:-

1. From the main menu bar, select **Seed→Instance**
2. Left click on the `Brick` region, click **Done** in prompt area. The Global Seeds dialog box appears, enter `0.1` for **Approximate global size**, accept the rest of the settings and click **OK**.
3. By following the above steps, now apply an **Approximate global seed size** of `0.02` to the `Concrete` region.

(b) To assign mesh controls:-

1. From the main menu bar, select **Mesh→Controls**
2. Select both the `Brick` and `Concrete` regions, you can do this by dragging a box across them. Click **Done** (on the prompt area).
3. The **Mesh Controls** dialog box appears, follow the settings depicted in **Fig.F1**. Ensure that **Medial axis** algorithm is chosen.

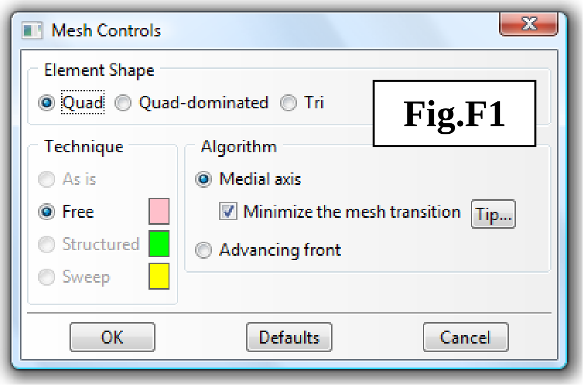


Fig.F1

(c) To assign element type:-

1. From the main menu bar, select **Mesh→Element Type**
2. Select both regions. Click **Done**.
3. The **Element Type** dialog box appears (**Fig.F2**), under the **Family** list, ensure that **Heat transfer** is selected. The element type to be assigned is **DC2D4**.

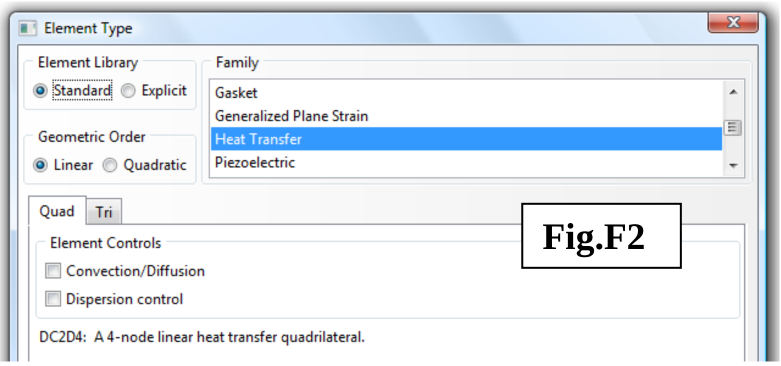


Fig.F2

(d) To mesh the part instance:-

1. From the main menu bar, select **Mesh→Instance**
2. Select both regions. Click **Done**. The generated mesh should resemble **Fig.F3**.

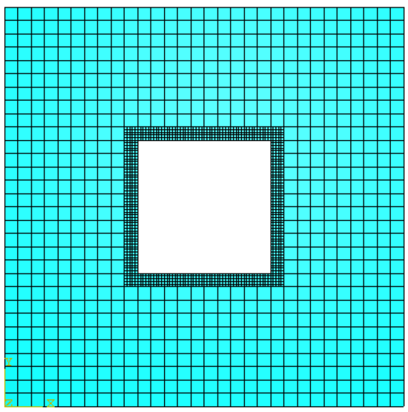


Fig.F3

G. MODULE → JOB

(a) To create a new job:-

1. From the main menu bar, select **Job→Create**
2. The **Create Job** dialog box appears, enter `Job-2D-Thermal`. Click **Continue**.
3. The **Edit Job** dialog box appears, accept the default settings and click **OK**.

(b) To submit the job:-

1. From the main menu bar, select **Job→Manager**
2. The **Job Manager** dialog box appears (**Fig.G1**), select `Job-2D-Thermal` and click on the **Submit** button. To see the progress of the analysis, and to monitor error and warning messages, click the **Monitor** button to bring up the **Monitor** dialog box (**Fig.G2**).

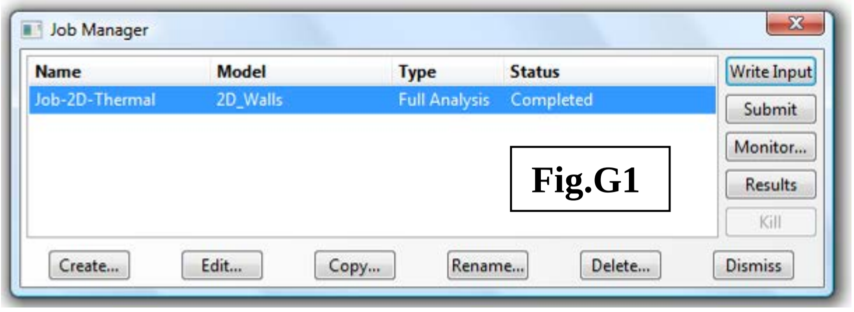


Fig.G1

(c) To analyse the results:-

When the job is **Completed**, click on the **Results** button on the **Job Manager** dialog box (**Fig.G1**).

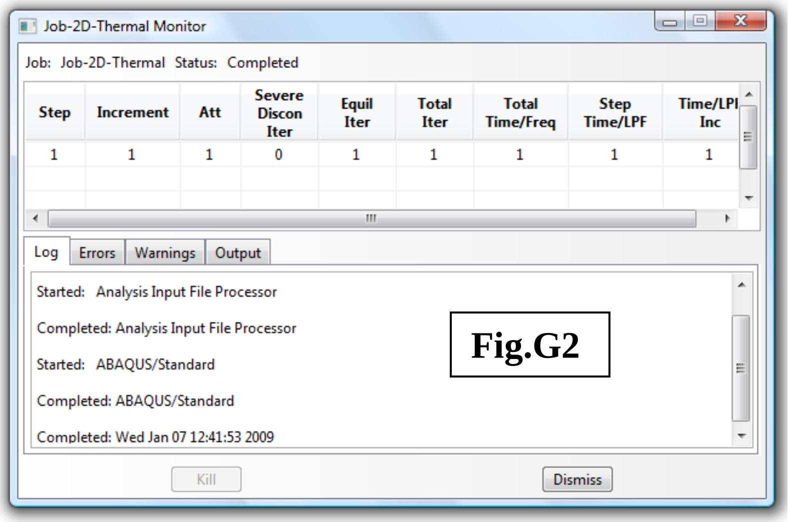


Fig.G2

Note: If the job fails to complete, go back to the **Monitor** dialog box (**Fig.G2**) and examine the messages under **Errors** and **Warnings** tabs, which often will provide clues on how to troubleshoot the analysis.

H. MODULE → VISUALIZATION

1. From the main menu bar, select **Results→Field Output**

2. The **Field Output** dialog box appears, under **Primary Variable**, select **NT11** and click **OK** to produce the nodal temperature distribution plot (**Fig.H1**). *Note:* The temperature is in Kelvin.

3. Click on **Contour Options** on the prompt area, and you'll be presented with various ways of customising the output.

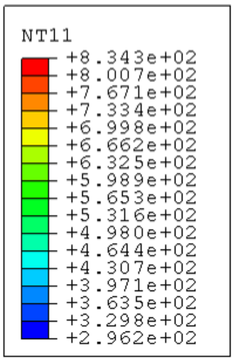
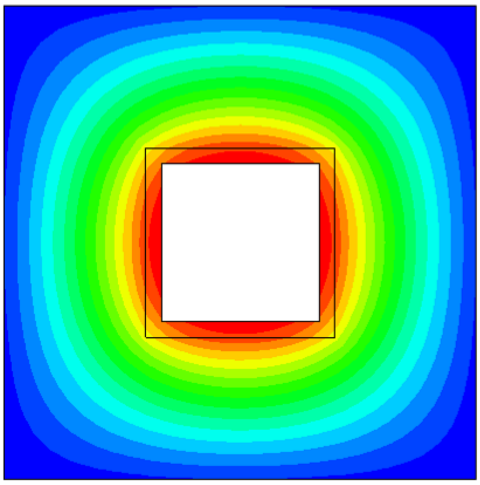


Fig.H1



4. To display the heat flux distribution within the walls, from the main menu bar, select **Results→Field Output**

5. The **Field Output** dialog box appears, under **Primary Variable**, select **HFL** and click **OK.** The heat fluxes shown here are in W m^{-2} (**Fig.H2**), consistent with the SI unit we employed while setting up the model.

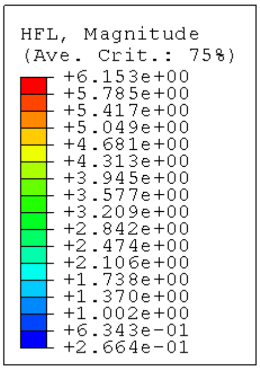
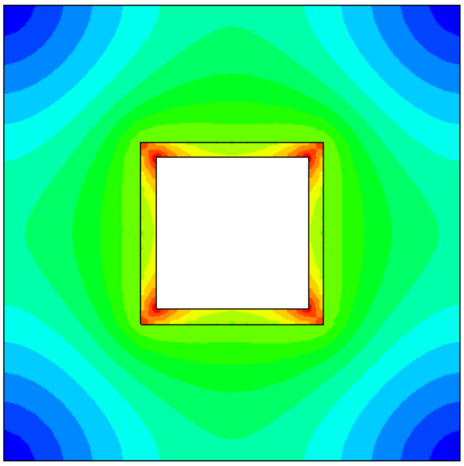


Fig.H2



6. To generate a resultant vector plot of the heat fluxes (**Fig.H3**), from the main menu bar, select **Plot→Symbols**. Click on **Symbol Options** on the prompt area to customise the vector plot.

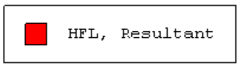
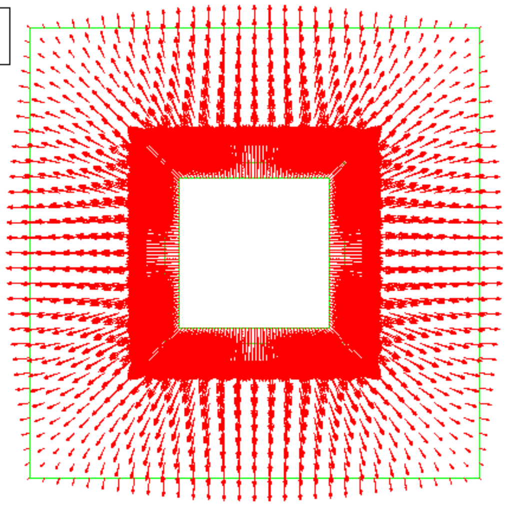


Fig.H3



TASKS

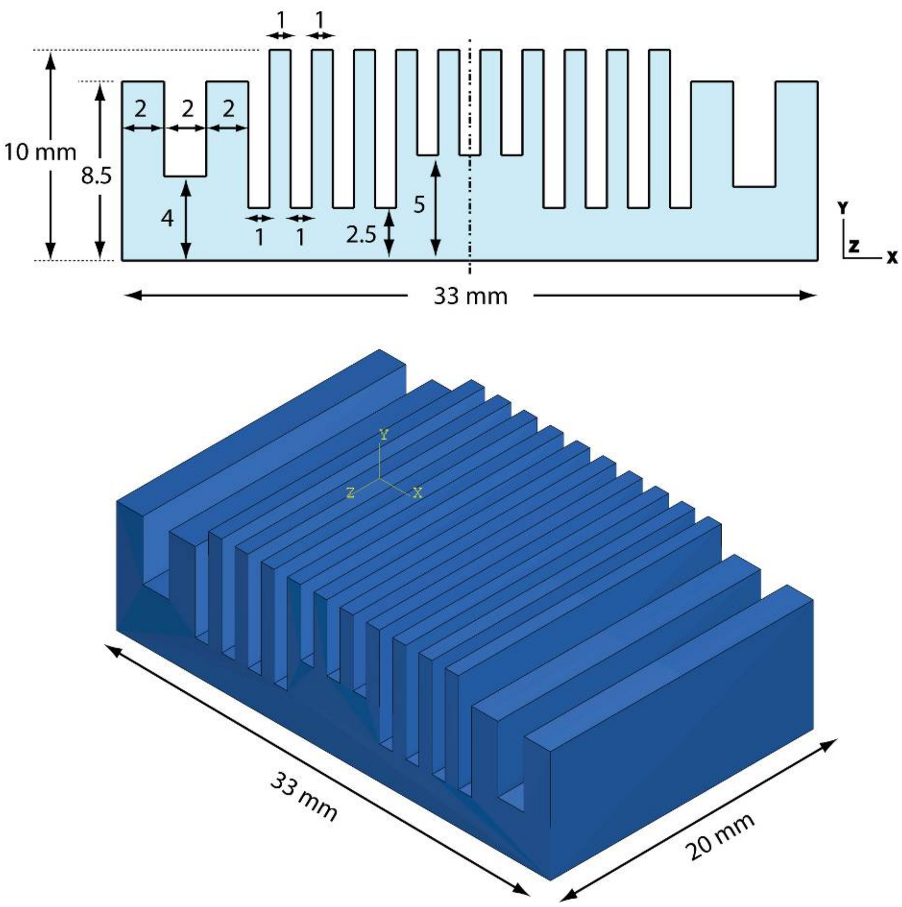
1. Due to symmetry, it is possible to model a quarter or just one-eighth of the system by applying suitable boundary conditions. Demonstrate how this could be done in ABAQUS.
2. When it comes to meshing, there are many possibilities in terms of the choice of mesh size and density, element type, shape and order, and meshing technique or algorithm. Explore how some of the above can affect the accuracy of your model predictions.
3. Compare the variation of thermal gradients across the different sections of the walls (e.g. along the horizontal, vertical, diagonal directions etc.).

(B) Three-Dimensional Transient Problem – Heat Dissipation through Ribbed Surfaces

Problem Description:

Ribbed surfaces or fins are commonly used in engineering applications to dissipate heat. The figure below shows the 2-D cross-section and 3-D geometry of an aluminium heat sink designed for cooling a microprocessor. The thermal conductivity of aluminium is $k = 170\ \mathrm{W\ m^{-1}\ K^{-1}}$. The initial temperature of the heat sink is 293 K. When the microprocessor is operating, the bottom surface of the heat sink is exposed to a constant heat flux of $q = 1000\ \mathrm{W\ m^{-2}}$. Forced air flow from a cooling fan over the developed surface maintains the surrounding surface at 323 K. The convective heat transfer coefficient between the fin and the ambient surrounding is at $h = 80\ \mathrm{W\ m^{-2}\ K^{-1}}$.

Formulate a transient 3-D FE model to predict (i) the time needed for the heat sink to achieve steady-state conditions, and (ii) the temperature distribution within the developed surfaces.



SOLUTION:

- Start ABAQUS/CAE. At the **Start Session** dialog box, click **Create Model Database**.
- From the main menu bar, select **Model→Create**. The **Edit Model Attributes** dialog box appears, name the model `3D_Fin`

A. MODULE → PART

1. From the main menu bar, select **Part→Create**
2. Name the part `Fin` and follow the settings depicted in **Fig.A1**. The approximate size is set at `0.1` (metre).
3. Sketch the 2-D profile (**Fig.A2**) according to the dimensions given in the ***Problem Description***.

   Note: Remember to construct the model in SI units.

   Tips: (a) To ease sketching, click on the **Sketcher Options** tool  located in the Sketcher toolbox and change the **Grid spacing** to `0.001` and the **Minor Intervals** to `1`.
   (b) You can exploit the symmetry by using the “Mirror” tool, located under **Edit→Transform→Mirror**.
4. When done sketching, click **Done** in the prompt area. The **Edit Base Extrusion** dialog box appears, enter the base extrusion depth as `0.02`

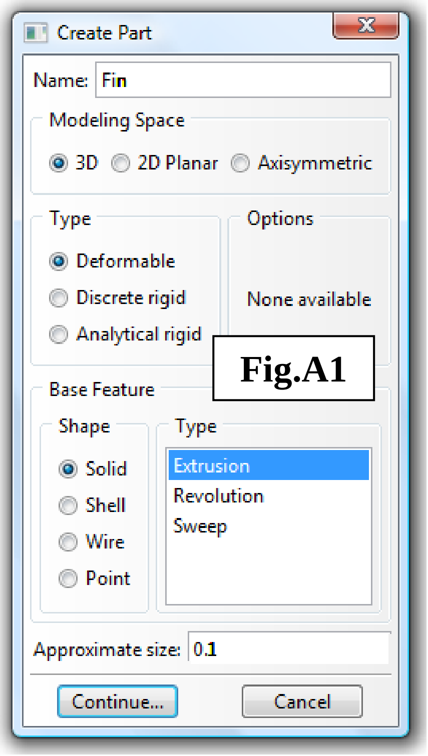


Fig.A1

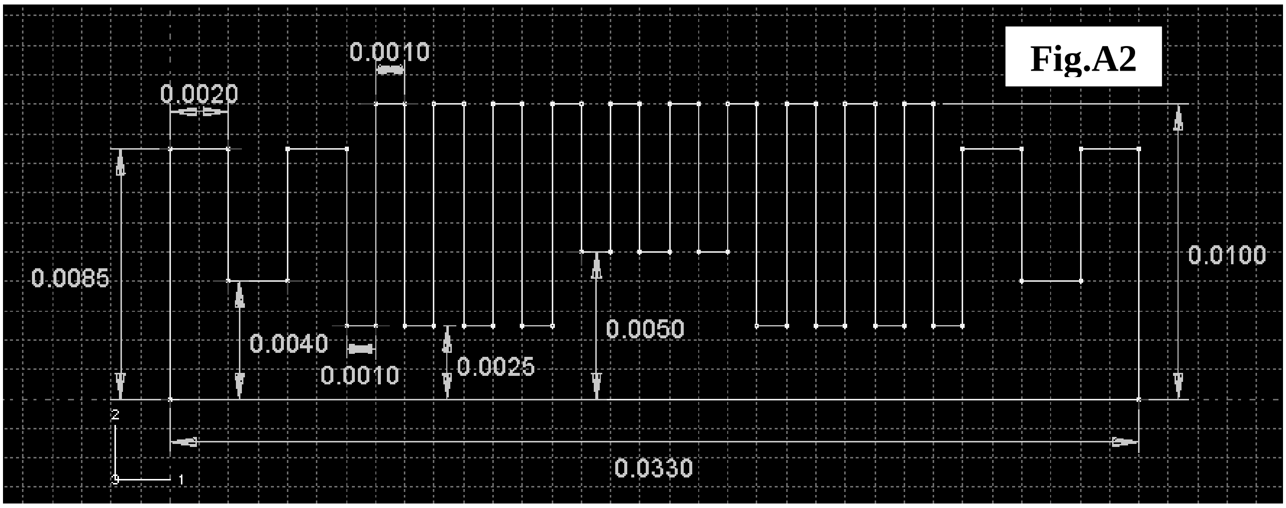


Fig.A2

B. MODULE → PROPERTY

1. From the main menu bar, select **Material→Create**

2. Name the material `Aluminium`.

3. Create the following material properties (**Fig.B1**):-

   General→Density `2700` kg m^{-3}

   Thermal→Conductivity `170` W m^{-1} K^{-1}

   Thermal→Specific Heat `950` J kg^{-1} K^{-1}

   Note: Since this will be transient heat transfer analysis, we need to include both density and specific heat properties.

4. Create a new section, name it `Section-Fin`, use the settings as shown in **Fig.B2** and **Fig.B3.**

5. Assign the section to the `Fin` part.

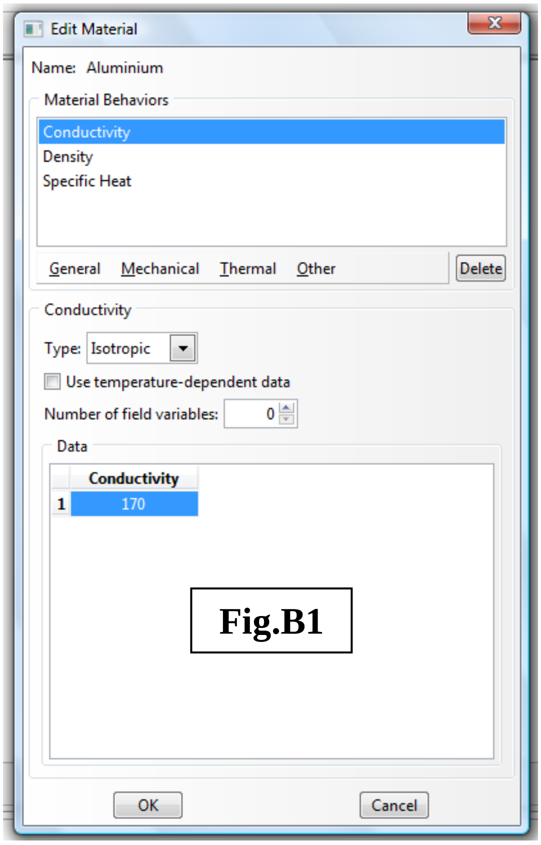


Fig.B1

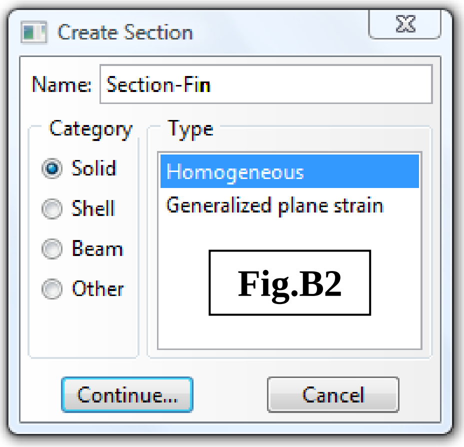


Fig.B2

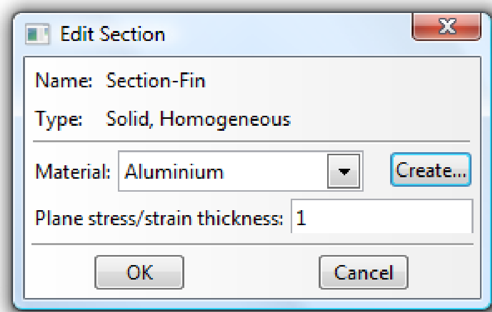


Fig.B3

C. MODULE → ASSEMBLY

1. From the main menu bar, select **Instance→Create**

2. Create an instance of the `Fin` part. Under **Instance Type**, make sure to select **Independent (mesh on instance)**.

D. MODULE → STEP

(a) To create the transient analysis step

1. From the main menu bar, select **Step→Create**
2. Name it `Transient-heating`. The **Procedure type** is **General→Heat Transfer**
3. In the **Edit step** dialog box (**Fig.D1**), under the **Basic** tab, ensure that the **Response** is **Transient,** and set the **Time period** as `5000` (seconds).

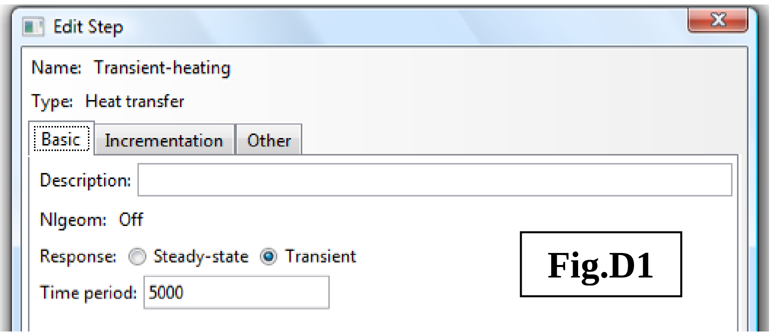


Fig.D1

4. Click on the **Incrementation** tab (**Fig.D2**), increase the **Maximum number of increments** to `200`. Change the **Initial Increment size** to `0.1` and **Maximum Increment size** to `100`.

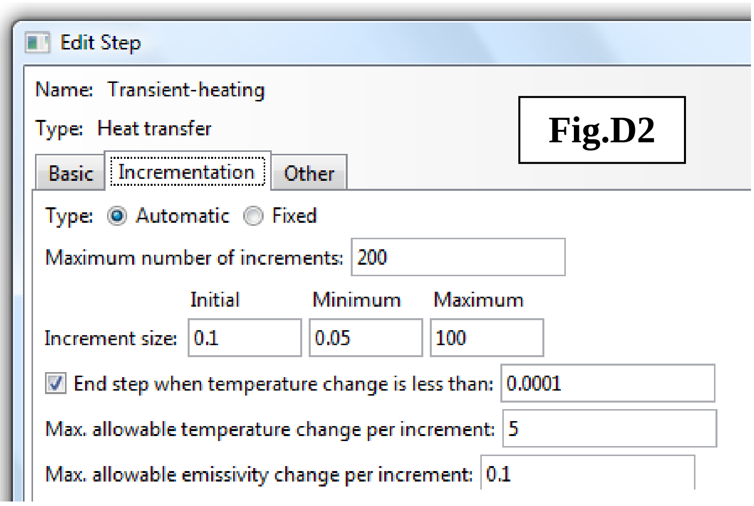


Fig.D2

5. Toggle on **End step when temperature change is less than** and enter `0.0001`, so that iteration will stop once thermal equilibrium is reached.
6. Set the **Max. allowable temperature change per increment** to `5` (Kelvin).
7. Accept the default settings under the **Other** tab.

(b) To edit the field output

1. From the main menu bar, select **Output→Field Output Requests→Edit→**`F-Output-1`
2. Under **Output Variables**, toggle on **Thermal** and select **NT** and **HFL**.

(c) To edit the history output

1. First we create a node set to record the temperature history. From the main menu bar, select **Tools→Set→Create**. The **Create Set** dialog box appears, name it `Set-Node1` and pick the node depicted in **Fig.D3**.

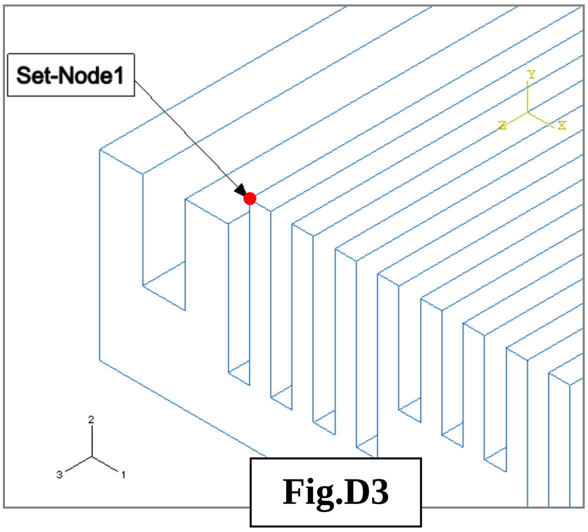


Fig.D3

2. From the main menu bar, select **Output→History Output Request→Create→**`H-Output-1`

3. The **Edit History Output Request** dialogue box appears (**Fig.D4**). Change the **Domain** to **Set** and choose `Set-Node1`. Save output at **Every** `1` increments. Under **Output Variables**, choose **Thermal→NT, Nodal temperature**

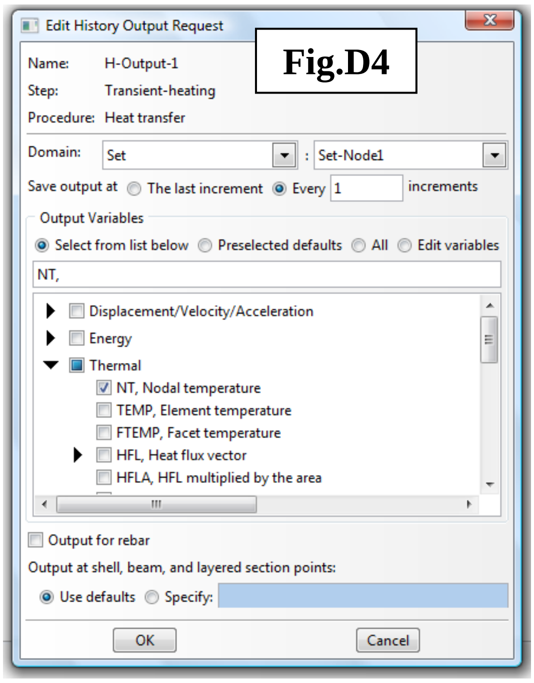


Fig.D4

(d) Create a DOF monitor

1. A Degree of Freedom (DOF) monitor is useful to follow the progress of a transient analysis. Here, we'll set up `Set-Node1` to monitor the temperature evolution.

2. From the main menu bar, select **Output→DOF Monitor**

3. Fill out the options as in **Fig.D5**. Note that DOF 11 corresponds to temperature in ABAQUS/CAE.

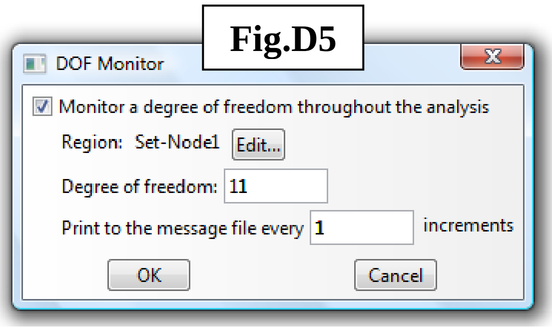


Fig.D5

E. MODULE → INTERACTION

1. From the main menu bar, select **Interaction→Create**

2. Name it `Int-Convection`. Under **Types for Selected Step**, choose **Surface film condition**, see **Fig.E1**.

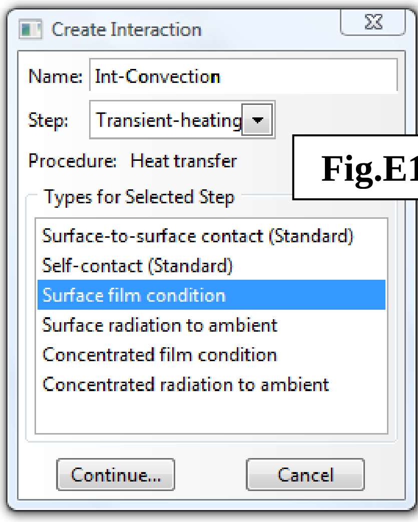
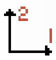


Fig.E1

3. The next task is to select the surfaces to apply the film conditions. However, since there are so many surfaces involved, it will be more convenient to do it as follows:-

   (a) From the main menu bar, select **View→Toolbars→Views**

   (b) Click the **Apply Front View** button: 

   (c) Now drag a box across the screen to pick all surfaces above the base surface (as indicated by dotted lines in **Fig.E2**). *Important*: Ensure that all surfaces are selected except the base.

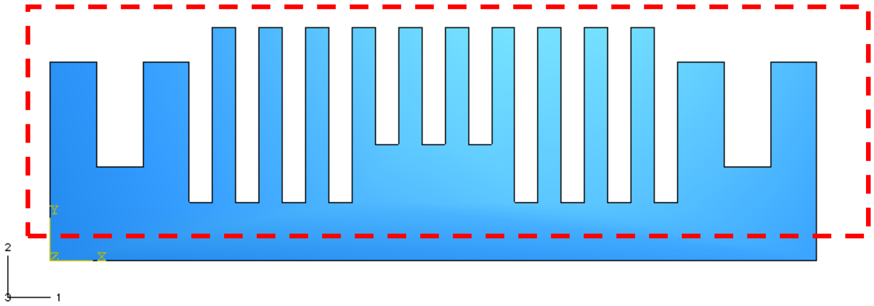


Fig.E2

   (d) The **Edit Interaction** dialog box appears (**Fig.E3**), fill in the **Film coefficient** as `80` (K) and set the **Sink temperature** as `323` (W m^{-2} K^{-1}).

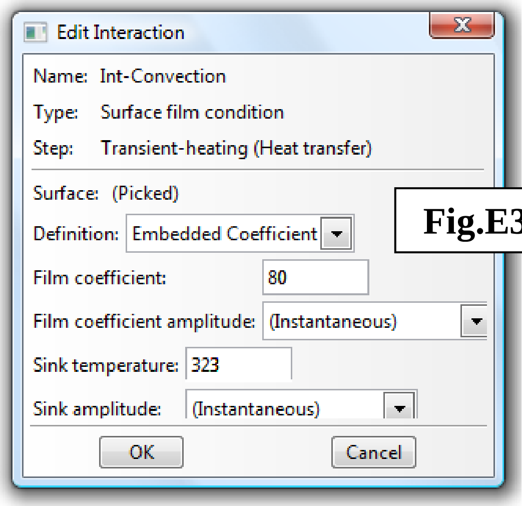


Fig.E3

F. MODULE → LOAD

(a) To create load (i.e. heat flux at the base of heat sink)

1. From the main menu bar, select **Load→Create**
2. Name it `Apply_Heatflux`. Under **Types for Selected Step**, choose **Surface heat flux**, see **Fig.F1**.
3. When prompted to choose the surface for the surface heat flux, it would be necessary to rotate the view  so that the bottom surface of the heat sink can be selected (**Fig.F2**).

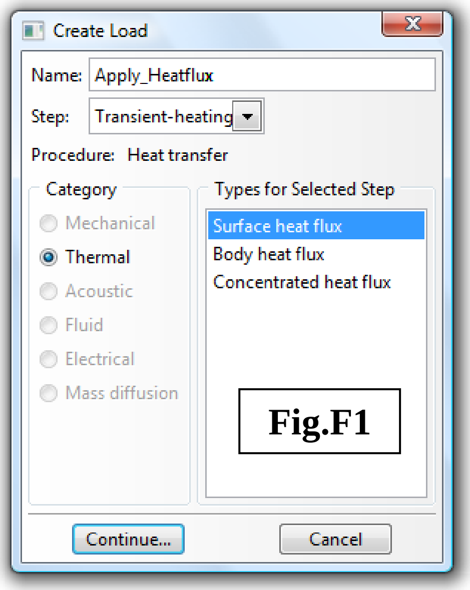


Fig.F1

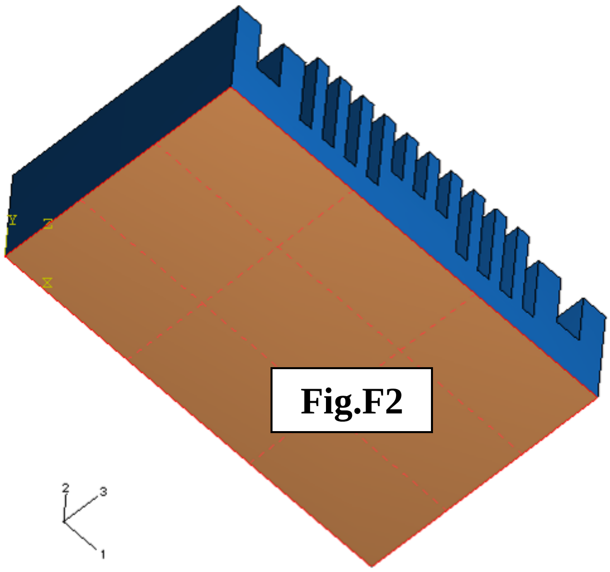


Fig.F2

4. Fill out the **Edit Load** dialog box as in **Fig.F3**.

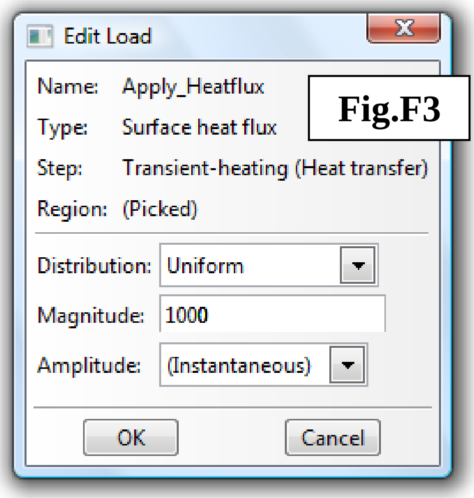


Fig.F3

(b) To create field (i.e. initial temperature)

1. From the main menu bar, select **Predefined Field→Create**
2. Name it `Initial-Temperature`. Under **Step**, choose **Initial**. Under **Category**, choose **Other→Temperature**, see **Fig.F4**.

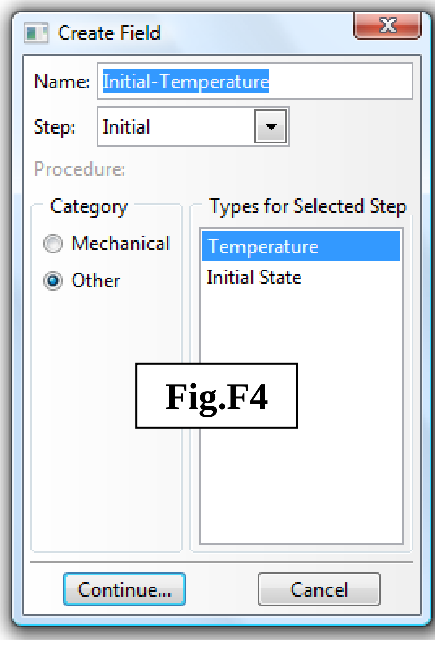


Fig.F4

3. When prompted to select region for the field, drag a box across the whole assembly to select all surfaces. Fill out the **Edit Filed** dialogue box as in **Fig.F5**.

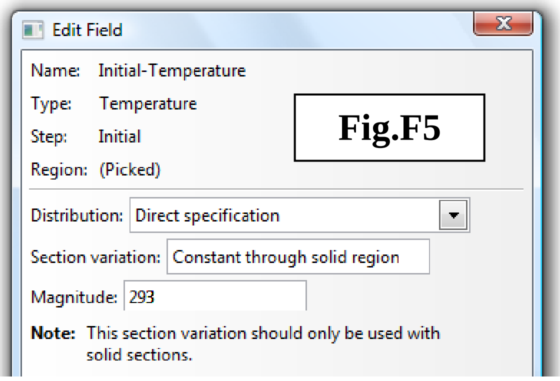


Fig.F5

G. MODULE → MESH

(a) To seed the part instance:-

1. From the main menu bar, select **Seed→Instance**
2. Apply `0.0005` for the **Approximate global size**.

(b) To assign mesh controls:-

1. From the main menu bar, select **Mesh→Controls**
2. The **Mesh Controls** dialog box appears, follow the settings depicted in **Fig.G1**.

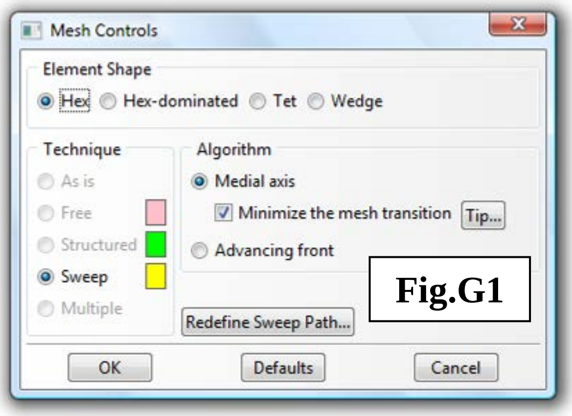


Fig.G1

(c) To assign element type:-

1. From the main menu bar, select **Mesh→Element Type**
2. The **Element Type** dialog box appears (**Fig.G2**), under the **Family** list, choose **Heat transfer**. The type of element assigned is **DC3D8**.

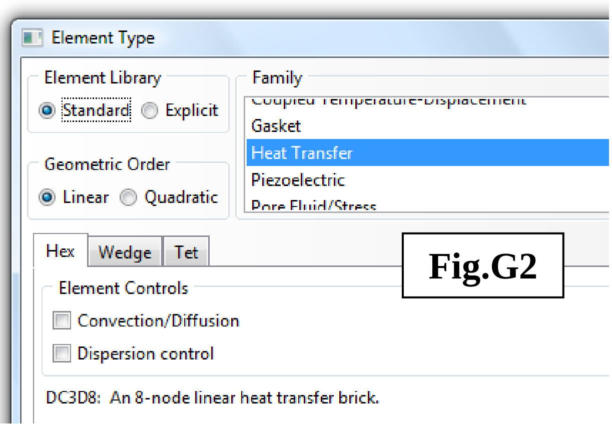


Fig.G2

(d) To mesh the part instance:-

1. From the main menu bar, select **Mesh→Instance**
2. The generated mesh should resemble **Fig.G3**.

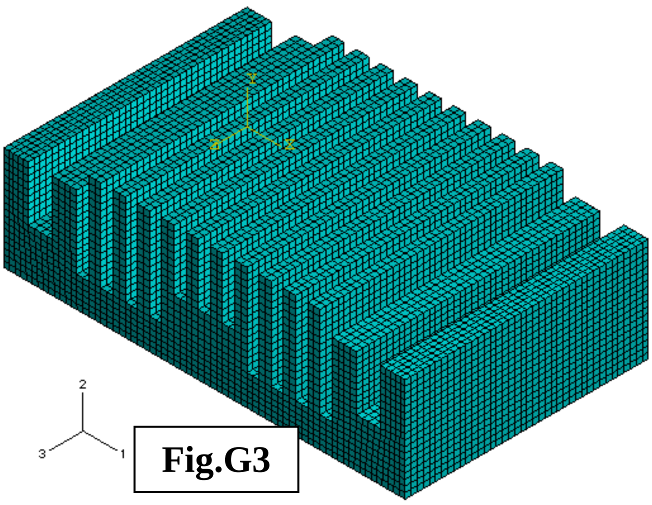


Fig.G3

H. MODULE → JOB

1. From the main menu bar, select **Job→Create**
2. Enter `Job-3D-Fin` as the job name, ensure that the **Source Model** chosen is `3D_Fin`.
3. Submit the job and monitor the progress. Since this is a transient analysis, longer computation time is expected (may take 10 to 20 minutes depending on your system).
4. When the job is completed, from the **Job Manager** dialogue box, click on **Results**.

I. MODULE → VISUALIZATION

1. From the main menu bar, select **Results→History Output**. Plot the nodal temperature of `Set-Node1` as a function of time (**Fig.I1**). It can be seen that it takes about 500 s to reach steady-state conditions.

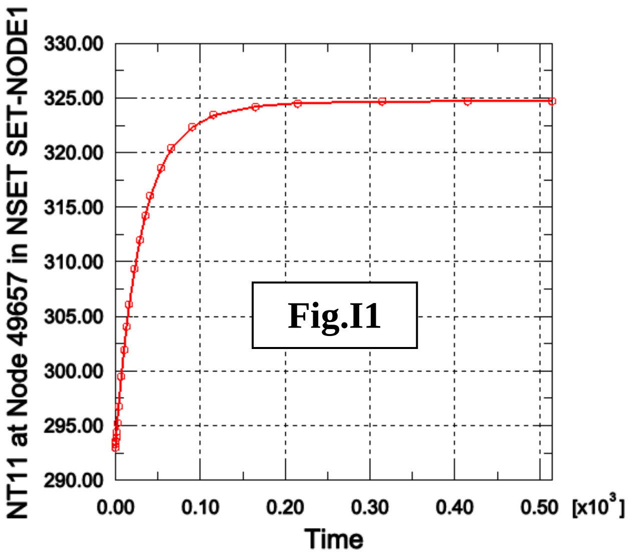


Fig.I1

2. To display the nodal temperature distribution, from the main menu bar, select **Results→Field Output** and select **NT11. Fig.I2** shows the temperature field at steady-state.

   Note: Using the control buttons  in the context bar, you can step though the frames to examine the temporal evolution of the thermal field.

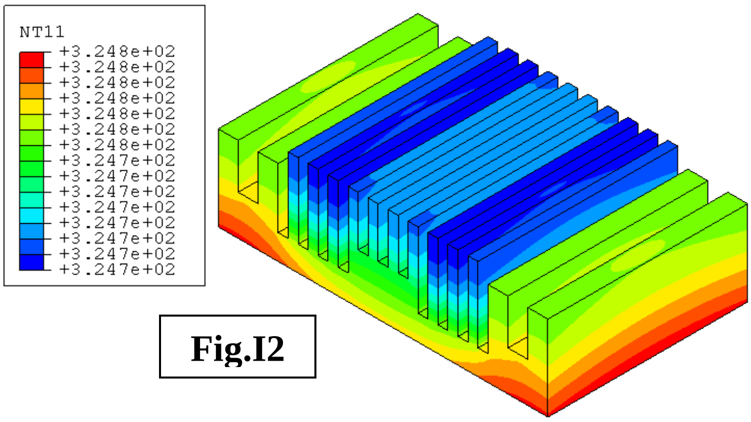


Fig.I2

3. To display the heat flux distribution, from the **Field Output** select **HFL. Fig.I3** shows the temperature field at the steady-state condition.

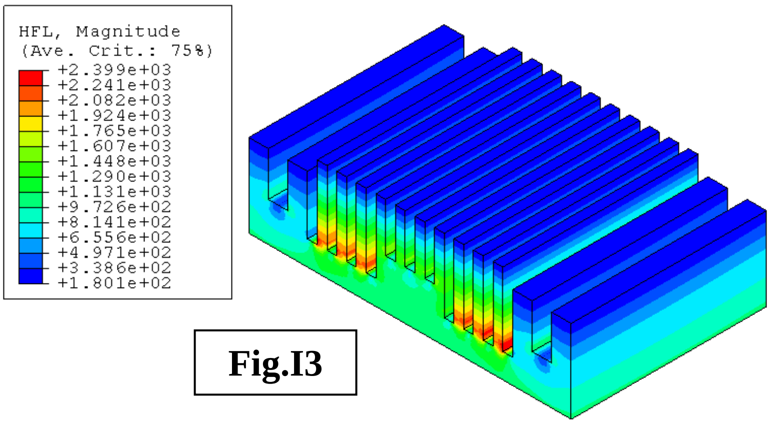


Fig.I3

J. TASKS

1. Instead of using a transient model, solve the above problem using a steady-state model.
2. Compute the temperature gradients across different sections of the heat sink. Investigate how sensitive the solutions are toward the choice of mesh size and/or element type.
3. How could one modify the current heat sink design to reduce the time for it to reach steady-state conditions? Demonstrate through a comparative FE analysis.
4. In practice, it's most likely that the heat flux at the base of the heat sink will vary as a function of time, say by increasing linearly from 0 to 1000 W m^{-2} over 200 sec. How can you model such a changing boundary condition in ABAQUS?