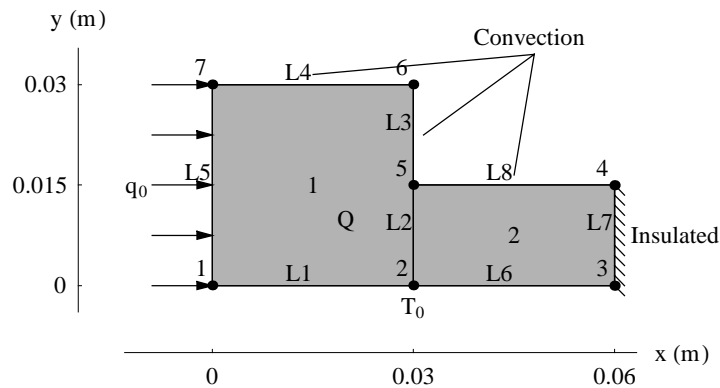


### ■ A.3.3 Steady State Heat Flow

Consider two dimensional heat flow over an L-shaped body shown in Figure A.3 using Abaqus. The thermal conductivity in both directions for the larger left area is  $45 \text{ W/m} \cdot ^\circ\text{C}$  and that for the smaller right area is  $25 \text{ W/m} \cdot ^\circ\text{C}$ . Heat is generated only in the left area at a rate of  $Q = 5 \times 10^6 \text{ W/m}^3$ . The bottom is maintained at a temperature of  $T_0 = 110^\circ\text{C}$ . Convection heat loss takes place on the top where the ambient air temperature is  $20^\circ\text{C}$  and the convection heat transfer coefficient is  $h = 55 \text{ W/m}^2 \cdot ^\circ\text{C}$ . The right side is insulated. The left side is subjected to heat flux at a uniform rate of  $q_0 = 8000 \text{ W/m}^2$ .



For this example it is convenient to use the Abaqus cae to prepare the input file.

#### Model creation using abaqus cae

##### 1. Part module

The overall geometry is created using the Part module in the abaqus cae system. A model may consist of several different parts. Interactions among these parts are specified in a later module. For this heat flow example we create a single part consisting of the L-shaped region.

Part> Create [Name: Part-1, Modeling space: 2D Planar, Type: Deformable, Base feature: Shell, Approximate size: 0.1] Continue

Add> Line> Connected lines [Enter x,y coordinates of corners of the L-shape]. Enter return after entering each pair: 0,0; 0.06,0; 0.06,0.015; 0.03,0.015; 0.03,0.03; 0,0.03; 0,0. Click on any other icon to get out of line define mode. Click on Done next to 'Sketch the section for planar shell'.

In order to assign different properties, we must partition the L-shaped region into two as follows.

Tools> Partition> Face> Shortest path between two points. Select a point (0.03,0) at the bottom edge and the inside corner of L-shape (0.03,0.015) and click on 'Create partition'. The L-shaped region should now be divided into the left square area and the right rectangular area.

## 2. Property module

Different material and section properties are created in this module. For this example two materials are defined with appropriate thermal conductivity values.

Material> Create> Material-1> Continue>Thermal> Conductivity [Enter Conductivity (k) = 45] OK

Material> Create> Material-2> Continue>Thermal> Conductivity [Enter Conductivity (k) = 25] OK

In order to use these materials later, they must be associated with appropriate sections. Two sections are defined and associated with these materials as follows.

Section> Create> Section-1> Solid> Homogeneous> Continue>[Set Material to Material-1 and plane stress/strain thickness to 1] OK

Section> Create> Section-2> Solid> Homogeneous> Continue>[Set Material to Material-2 and plane stress/strain thickness to 1] OK

The next task is to actually associate these sections to different areas of the part as follows.

Assign> Section> [Select left square area and click Done on the button next to 'Select the regions to be assigned a section']> Section-1> OK

Assign> Section> [Select right rectangular area and click Done]> Section-2> OK

## 3. Assembly module

Different parts in a model are assembled in this module. Different contact and interface conditions can be created between these parts. For this example, we only have a single part and thus our task is very simple. The assembly is just the part and is defined as follows.

Instance> Create> Part-1> OK

## 4. Step module

This module defines the analysis type and sets desired output solution parameters.

Step> Create> [Name: Step-1, Procedure type: General]> Heat transfer> Continue> [Response: Steady state]

The desired output solution parameters are defined as follows.

Output> Field output requests> Create> [Name: F-Output-2, Step-1]> Continue> Thermal [Select NT, Nodal temperature and HFL, Heat flux vector]

#### 5. Interaction module

For structural problems contact and sliding type boundary conditions are defined in this module. For heat transfer problems, the main purpose of this module is to define the convection boundary condition.

Interaction> Create> [Name: Int-1, Step: Step-1]> Film condition> Continue. Select the lines on the top of the region by clicking. Keep the shift key pressed to be able to select multiple lines. Click on 'Done' next to the 'Select the surface'. In the resulting dialog box enter h value (55) in the Film coefficient and surrounding fluid temperature (20) in the Sink temperature fields and click OK.

#### 6. Load module

For structural problems loads and boundary conditions are created in this module. For heat transfer problems, this module is used to specify known temperature and heat flux boundary conditions.

Load> Create> [Name: Load-1, Step: Step-1, Category: Thermal]> Surface heat flux> Continue. Select the line on the left of the region by clicking. Click on 'Done' next to the 'Select surfaces for the load'. In the resulting dialog box enter Magnitude = 8000 for the specified heat flux on this side and click OK.

Load> Create> [Name: Load-2, Step: Step-1, Category: Thermal]> Body heat flux> Continue. Select the left area by clicking. Click on 'Done' next to the 'Select bodies for the load'. In the resulting dialog box enter Magnitude = 5000000 for the specified body heat flux on this area and click OK.

BC> Create> [Name: BC-1, Step: Step-1, Category: Other]> Temperature> Continue. Select the bottom lines by clicking. Click on 'Done' next to the 'Select regions for the boundary condition'. In the resulting dialog box enter Magnitude = 110 for the specified temperature on this bottom and click OK.

If no boundary condition is specified along a boundary, it automatically means no heat flow across that face which is equivalent to the insulated boundary condition. Thus there is no need to explicitly specify a zero heat flux on the right end of the solid.

#### 7. Mesh module

The next step is to actually create a finite element mesh. We must decide on an approximate size of each element, which will obviously determine the total number of elements and nodes. One can specify a global element size for the entire model. To provide further control over the mesh it is possible to specify the

target number of subdivisions of a given line. The subdivisions can be biased towards a chosen end of the line (for example to capture rapidly changing solution near a corner). For this example, looking at the physical dimensions of the model, we choose a global element size of  $0.002\text{ m}$ . Thus the length of the model will be divided roughly into 30 segments and the width into 15 resulting in a mesh of the order of  $30 \times 15 = 450$  elements which should give us reasonable results. Near the corner at key point 5 we expect rapid solution change and thus we specify 20 subdivisions for each of the three lines meeting at the corner. A bias towards the corner is created by clicking near the corner end of each line (the bias direction is indicated by an arrow by abaqus graphics). The following steps are used to accomplish these tasks.

Seed> Edge biased> Click on the three lines meeting at the interior corner (pick near the end where the mesh must be denser). Click on 'Done' near 'Enter edges to be assigned local seeds'. Enter Bias ratio = 2 (larger value implies more bias).

Seed> Instance> Enter global element size (approximate): 0.002.

Mesh> Controls> Select both areas (with shift key pressed to be able to select multiple areas) and then click on 'Done' next to 'Select the regions to be assigned mesh controls'. In the resulting dialog box choose Element shape: Quad-dominated, Technique: Free, and Algorithm: Advancing front and then click OK and Done.

Mesh> Controls> Select both areas and then click on 'Done' next to 'Select the regions to be assigned element types'. In the resulting dialog box choose Element Library: Standard, Geometric Order: Linear, and Family: Heat transfer and then click OK and Done.

Mesh> Instance> Click on 'Yes' button next to 'OK to mesh part instance?'.

The resulting finite model is shown in Figure A.13. The model generation is now complete. Additional information, such as boundary conditions and load symbols, can be displayed on the model by selecting View> Assembly display options. The graphics can be saved to a file by using File> Print menu and setting the Destination to 'File', giving a file name and specifying the desired format.

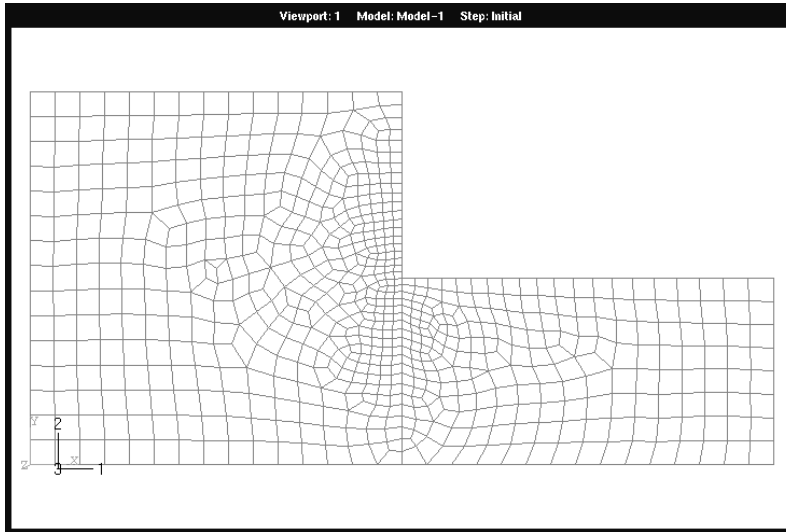


Figure A.13. Heat flow model in Abaqus

#### 8. Job module

We are now ready to actually create the finite element analysis input file as follows.

Job> Create> [Name: LShapeHeat, Model: Model-1]> Continue. In the resulting dialog box accept the default options [Job Type: Full analysis, Run Mode: Background, Submit Time: Immediately]

This step will create an input file named LShapeHeat.inp.

### Solution

The actual analysis can be performed in the batch mode using the input file by issuing the command 'abaqus job=LShapeHeat' at the shell prompt. Alternatively we can use the menu 'Job> Submit' from within the job module of abaqus cae. With this option the progress of the solution can be monitored using 'Job> Monitor' option. Once the analysis is complete the results will be saved in a database LShapeHeat.odb and summary of the execution steps and requested output will be written to a text file LShapeHeat.dat.

### Postprocessing using abaqus viewer

The postprocessing of the results is done using the viewer module. If the abaqus cae environment is already open one can simply switch to the viewer module and open the results database. Otherwise the viewer can be launched separately by issuing the 'abaqus viewer database=LShapeHeat' command at the shell prompt.

The results can now be viewed as numerical lists or plotted in various forms. For example Figure A.14 shows a contour plot of nodal temperatures and vector plot of element heat flux obtained using the following steps.

Results> Field output> [Select Name: NT11, Nodal temperatures at nodes]

Plot> Contours

Results> Field output> [Select Name: HFL, Heat flux vector at integration points, Invariant: Magnitude]

Plot> Contours

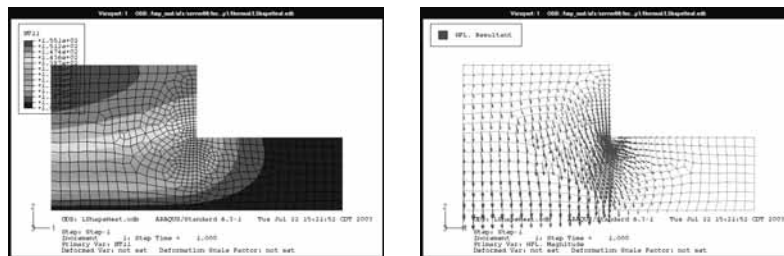


Figure A.14. Temperature contours and heat flux vectors