Chapter 9
Linear Analysis of Field Problems

The field equation,

$$D_x \frac{\partial^2 \phi}{\partial x^2} + D_y \frac{\partial^2 \phi}{\partial y^2} - A\phi + B = 0 \tag{9.1}$$

(where ϕ is the field variable) governs a wide variety of physical problems, referred to as field problems. Several engineering problems, e.g., torsion of noncircular section, ideal irrotational fluid flow, seepage, heat transfer, and electrostatic, are embedded in this equation. For each of these problems, the parameters D_x, D_y, A, and B designate a different physical property. For example, if the problem under consideration involves two-dimensional steady-state heat transfer with no heat generation within the body, then Eq. (9.1) becomes

$$D_x \frac{\partial^2 T}{\partial x^2} + D_y \frac{\partial^2 T}{\partial y^2} = 0 \tag{9.2}$$

where D_x and D_y are the thermal conductivity of the material in the x- and y-directions, respectively, and T represents temperature.

Most of the field problems can be solved using ANSYS software. However, only two types of field problems are considered within the context of this book: (i) heat transfer problems and (ii) moisture diffusion problems.

9.1 Heat Transfer Problems

In certain cases, a thermal analysis is followed by a stress analysis in order to evaluate the structural integrity of the component under the given thermal conditions. In a typical heat transfer problem, the goal is to obtain certain thermal quantities within a body under a specific set of boundary conditions. These quantities include:

The online version of this book (doi: 10.1007/978-1-4939-1007-6_9) contains supplementary material, which is available to authorized users

E. Madenci, I. Guven, *The Finite Element Method and Applications in Engineering Using ANSYS®*, DOI 10.1007/978-1-4899-7550-8_9

temperatures, thermal fluxes and gradients, and the amount of heat dissipated. There are two main types of thermal analyses:

Steady-state heat transfer: Solution is time independent.

Transient heat transfer: Subjected to specific initial conditions, the solution exhibits a time-dependent behavior. If the transient solution is obtained for a sufficiently long time period, the solution is expected to converge to the steady-state solution.

ANSYS accommodates three main heat transfer types: conduction, convection, and radiation. Convection boundary conditions require knowledge of the film coefficient and ambient temperature. There are a few different ways to impose radiation conditions within ANSYS. However, only one of them, the *Radiosity Solver* method, is explained within the context of this book.

Several boundary conditions are available, including specification of temperatures, heat fluxes, convective heat loss/gain, and radiation. As for the body loads, the user can specify heat generation rates within the domain.

In both steady-state and transient analyses, the material properties may be defined as temperature dependent. In the presence of temperature-dependent material properties, the analysis becomes nonlinear, thus requiring an iterative solution.

The most commonly used heat transfer elements are **`PLANE55`** (two-dimensional plane) and **`SOLID70`** (three-dimensional brick).

9.1.1 Steady-state Analysis

When the boundary conditions and body loads do not vary with time and there are no specified initial conditions, the solution quantities do not vary with time. In such cases, steady-state solutions are obtained. A steady-state analysis is demonstrated by considering two problems.

9.1.1.1 Analysis of a Tank/Pipe Assembly

A cylindrical tank and a small pipe form a junction, as shown in Fig. 9.1 (only 1/8th of the geometry is shown due to octant-symmetry). Inside the tank, there is fluid at a temperature of 450 °F. A steady flow of a fluid at a temperature of 100 °F is experienced inside the pipe. The film coefficient along the inner surface of the tank is 250 Btu/hr-ft^2-°F whereas the film coefficient along the inner surface of the pipe depends on the surface temperature. The geometric parameters and boundary conditions are given in Table 9.1, and the material properties are summarized in Table 9.2. Note in Tables 9.1 and 9.2 that the length units are in inches and feet. However, in order to obtain a physically correct solution the units must be consistent. Inches are used in this problem, therefore any parameter with the length unit in feet must be converted to inches. The goal is to determine the temperature distribution in the tank.

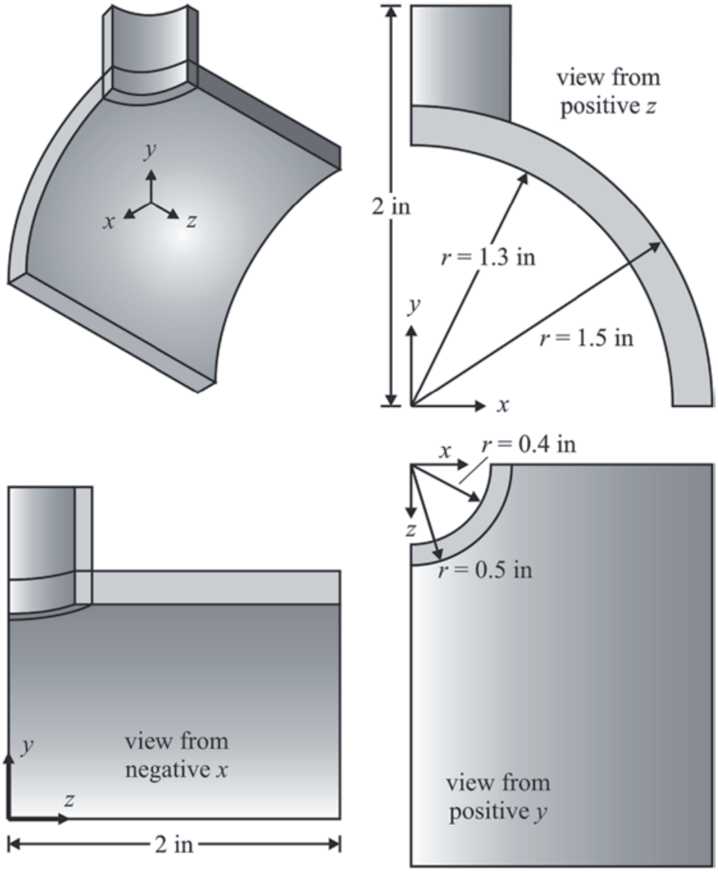


Fig. 9.1 The geometry of the tank and the pipe from different viewpoints

Table 9.1 Geometric parameters and boundary conditions used in the analysis

Parameter	Value
Inside diameter, pipe	0.8 in.
Outside diameter, pipe	1.0 in.
Inside diameter, tank	2.6 in.
Outside diameter, tank	3.0 in.
Inside bulk fluid temperature, tank	450 °F
Inside film coefficient, tank	250 Btu/hr-ft^2-°F
Inside bulk fluid temperature, pipe	100 °F

Table 9.2 Material properties used in the analysis

Temperature (°F)	Density (1b/in3)	Conductivity (Btu/hr-ft-°F)	Specific heat (Btu/lb-°F)	Film coefficient (pipe) (Btu/hr-ft^2-°F)
70	0.285	8.35	0113	426
200	0.285	8.90	0.117	405
300	0.285	9.35	0.119	352
400	0.285	9.80	0.122	275
500	0.285	10.35	0.125	221

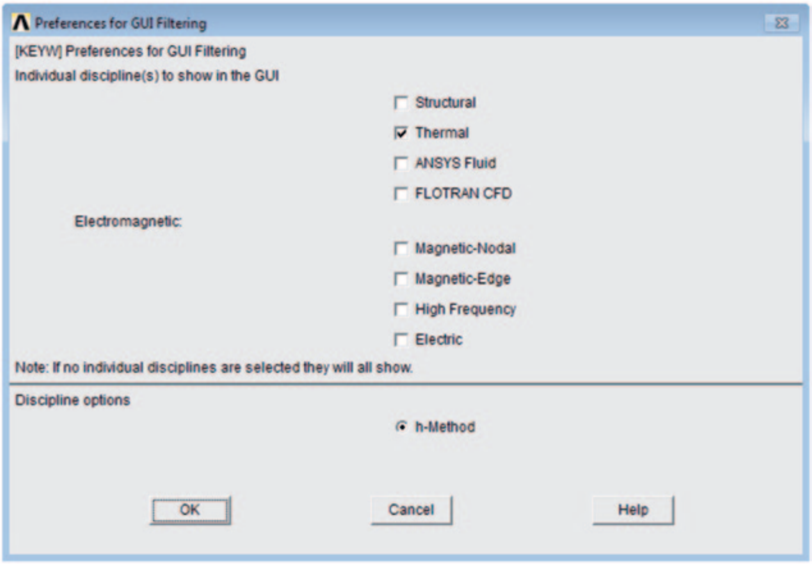


Fig. 9.2 *Preferences for GUI filtering* dialog box

Model Generation

- Declare *Preferences* for *Graphics User Interface* (GUI) filtering (Fig. 9.2) using the following menu path:

Main Menu > Preferences

  – Place a checkmark corresponding to ***Thermal***; click on ***OK***.
  – This action eliminates several menu entries that are not related to thermal analyses, thus resulting in an abridged menu.

- Define the element type (**`ET`** command) using the following menu path:

Main Menu > Preprocessor > Element Type > Add/Edit/Delete

  – *Element Types* dialog box appears; click on ***Add***.

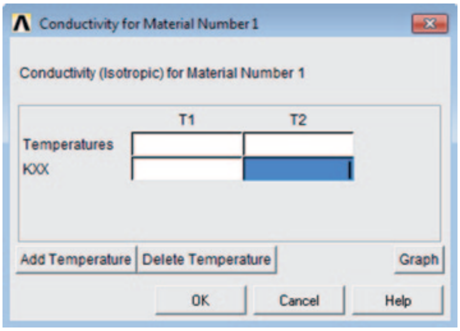


Fig. 9.3 Adding temperatures for temperature-dependent material properties

- Select ***Solid*** immediately below ***Thermal*** in the left list and ***Brick 8 Node 70*** on the right list; click on ***OK***.
- Click on ***Close***.

• Specify the temperature-independent material property, density (**`MP`** command), using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

- In the *Define Material Model Behavior* dialog box, in the right window, successively left-click on ***Thermal*** and ***Density***, which will bring up another dialog box.
- Enter ***0.285*** for ***DENS***; click on ***OK***.

• Specify temperature-dependent material properties.

- In the *Define Material Model Behavior* dialog box, in the right window, successively double-click on ***Thermal***, ***Conductivity***, and ***Isotropic***, which will bring up another dialog box.
- Click on the ***Add Temperature*** button four times (so that there are five temperature slots) (Fig. 9.3).
- Enter temperature values (i.e., ***70, …, 500***) in the top row, from left to right in ascending order (Table 9.2).
- Enter the corresponding thermal conductivity values referring to Table 9.2. However, the values given in Table 9.2 are in units of Btu/hr-ft-°F, which must be converted to Btu/hr-in-°F. This can be achieved within the dialog box, as shown in Fig. 9.4. Therefore, enter ***8.35/12, …, 10.35/12***.
- View the variation of thermal conductivity as a function of temperature by clicking on the ***Graph*** button. ***KXX*** versus ***TEMP*** appears in the *Graphics Window* (Fig. 9.5).
- Click on ***OK***.
- Specify specific heat in the same manner as thermal conductivity.

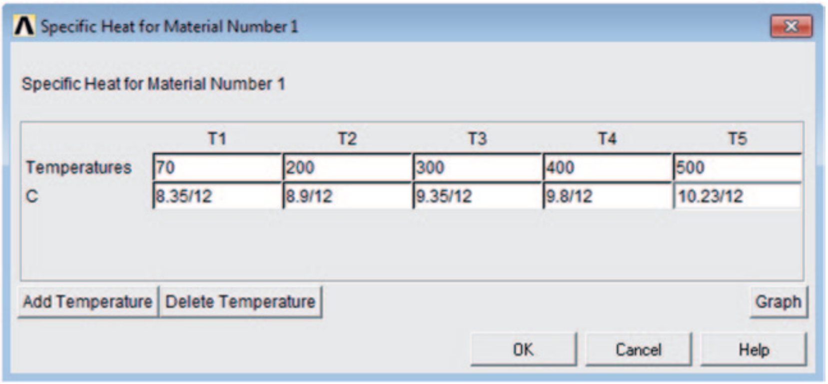


Fig. 9.4 Conductivity values at five different temperatures

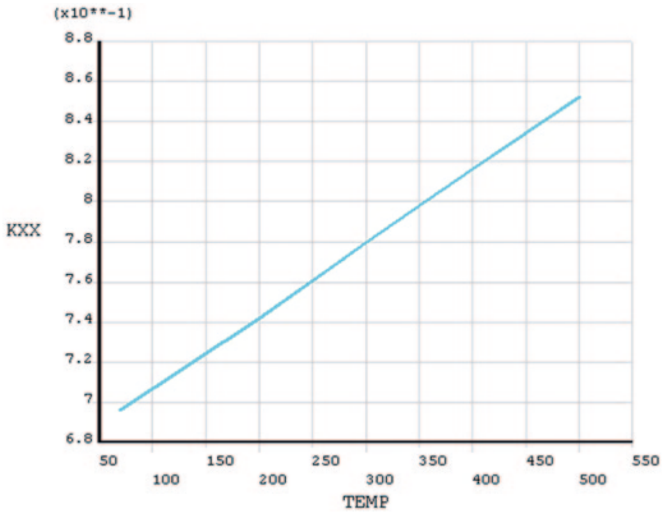


Fig. 9.5 Variation of conductivity with temperature as plotted in ANSYS

- Specify the temperature-dependent film coefficient inside the pipe**.**
- Although the tank and the pipe are of the same material, their film coefficients are different. Therefore, the film coefficient inside the pipe will be defined as a new material.
- In the *Define Material Model Behavior* dialog box, define a new material reference number using the following menu path:

Material > New Model

- A dialog box appears with the material reference number set to the next available number (in this case 2) (Fig. 9.6); click on OK.

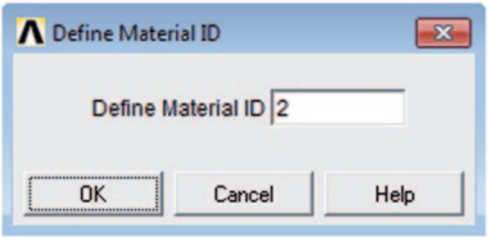


Fig. 9.6 Definition of a new material

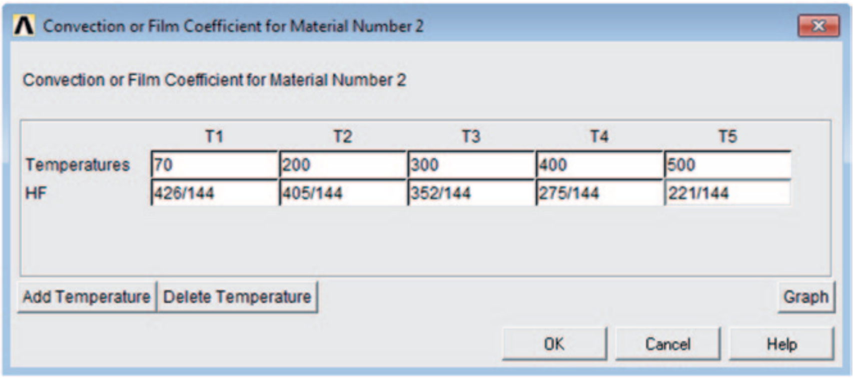


Fig. 9.7 Temperature-dependent film coefficient

- Note the new entry for Material Model Number 2 in the left list in the Define Material Model Behavior dialog box.
- In the right list, left-click on Convection or Film Coef., which brings up a new dialog box.
- Click on the Add Temperature button four times (so that there are five temperature slots); enter temperature values (i.e., 70, … , 500) in the top row, from left to right in ascending order.
- Enter the film coefficient values (division by 144 is required for unit conversion), as shown in Fig. 9.7; click on OK.
- Close the Def*ine Material Model Behavior* dialog box.
- The film coefficient for the inside surface of the tank will be specified later during the application of boundary conditions.

• Define parameters.

- In order to demonstrate the use of parameters in ANSYS the following parameters and their values are defined:

ri1 = 1.3

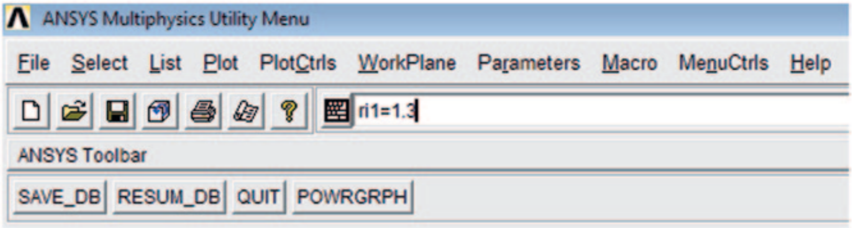


Fig. 9.8 Defining a parameter through the *Input Field*

ro1 = 1.5
z1 = 2.0
ri2 = 0.4
ro2 = 0.5
z2 = 2.0

– Parameters can be defined in two different ways:
  - Using the *Input Field*, write the user-defined parameter name followed by the equal sign (=) and the value assigned to the parameter (Fig. 9.8). Finish by the hitting ***Enter*** key on the keyboard.
  - Define the parameters using the following the menu path:

Utility Menu > Parameters > Scalar Parameters

The *Scalar Parameters* dialog box appears with an input box toward the bottom (Fig. 9.9). Write the user-defined parameter name followed by the equal sign (=) and the value assigned to the parameter; click on ***Accept***. Observe that the newly defined parameter appears in the list.

– Define all the parameters using either method.

• Create the volume for the tank—a hollow partial cylinder—(**CYLIND** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Volumes > Cylinder > By Dimensions

– Enter ***ro1*** for ***RAD1 Outer radius***, ***ri1*** for ***RAD2 Optional inner radius***, ***z1*** for the *second* text box in the ***Z1, Z2 Z-coordinates*** row, and ***90*** for ***THETA2 Ending angle (degrees)***; click on ***OK***.
– Observe the hollow partial cylinder created in the Graphics Window.
– View the tank from different angles using the following menu path:

Utility Menu > PlotCtrls > Pan Zoom Rotate

– Click on ***Obliq***.
– Figure 9.10 shows the outcome of this action as it appears in the *Graphics Window*.

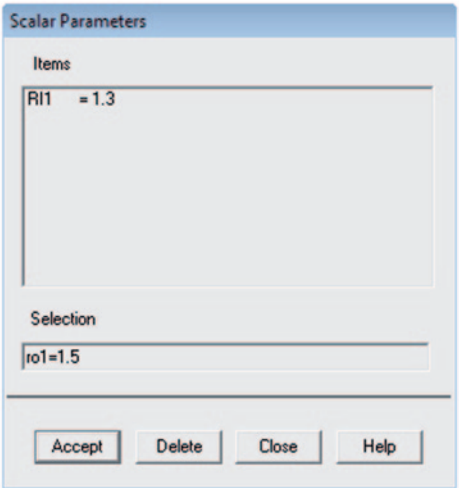


Fig. 9.9 *Scalar Parameters* dialog box

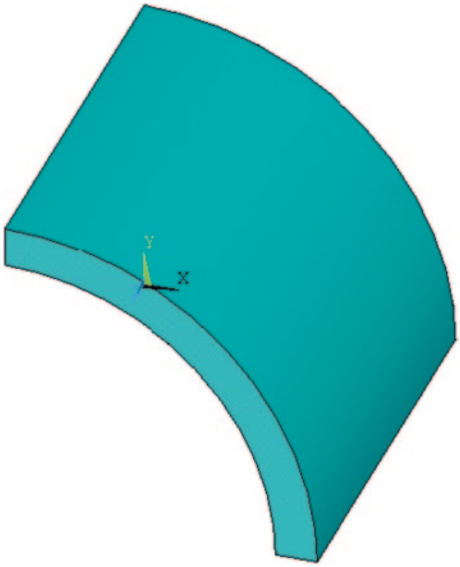


Fig. 9.10 Hollow partial cylinder as it appears in the *Graphics Window*

- Create the volume for the pipe—a second hollow partial cylinder.

  – In order to use the GUI for this action, first the *Working Plane* needs to be rotated using the following menu path:

Utility Menu > WorkPlane > Offset WP by Increments

  – *Offset WP* dialog box appears.
  – Find the *second input field* with the title: ***XY, YZ, ZX Angles***.
  – Enter ***0,− 90, 0*** in the second input field; click on ***OK*** (Fig. 9.11).

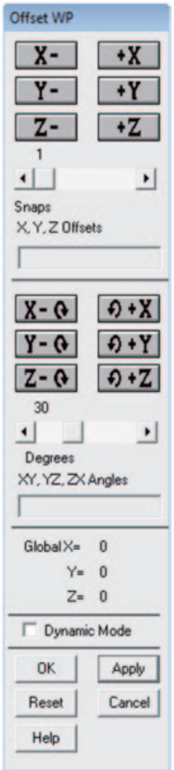


Fig. 9.11 *Offset WP* dialog box

– Create the second hollow partial cylinder using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Volumes > Cylinder > By Dimensions

– In the same manner used for the creation of the cylinder for the tank, enter ***ro2*** for ***RAD1 Outer radius***, ***ri2*** for ***RAD2 Optional inner radius***, ***z2*** for the second text box in the ***Z1, Z2 Z-coordinates*** row, and ***−90*** (*not 90*) for ***THETA2 Ending angle (degrees)***; click on ***OK***.
– Observe the second hollow partial cylinder created in the *Graphics Window* (Fig. 9.12).
– Turn the *numbering* on for the volumes using the following menu path:

Utility Menu > PlotCtrls > Numbering

– Place a *checkmark* by clicking on the square box next to ***VOLU Volume numbers***; click on ***OK***.
– In the *Pan Zoom Rotate Window*, click on ***Left*** button.

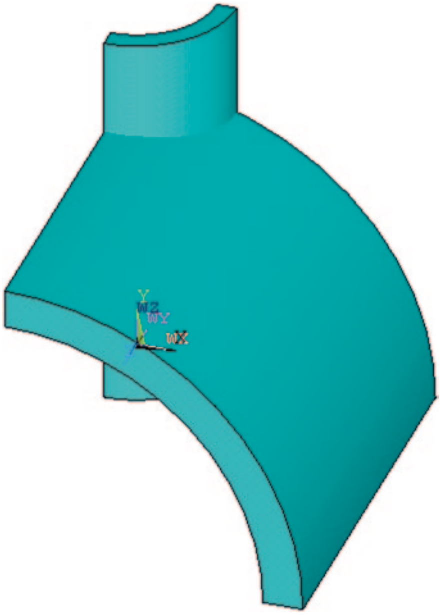


Fig. 9.12 Two hollow partial cylinders as they appear in the *Graphics Window*

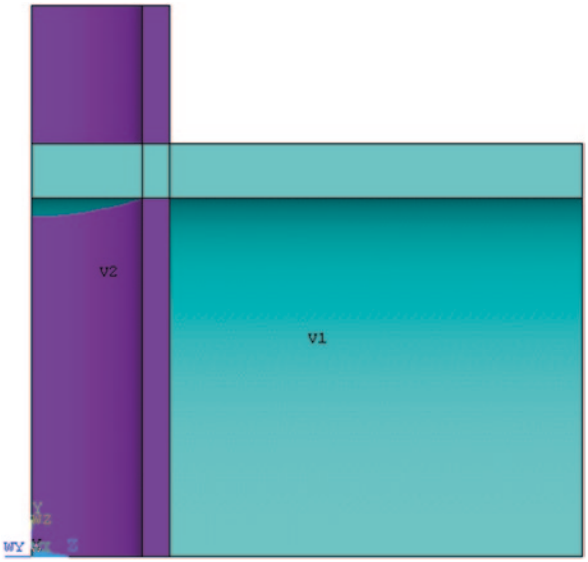


Fig. 9.13 Two hollow partial cylinders with volume numbers turned on (*both colors and numbers*)

– Figure 9.13 shows the outcome of this action as it appears in the *Graphics Window*; observe the different colors[1] for each volume and the volume numbers.

[1] Colors have not been used in the printed version of the figures. See the accompanying CD-ROM for color versions of the figures.

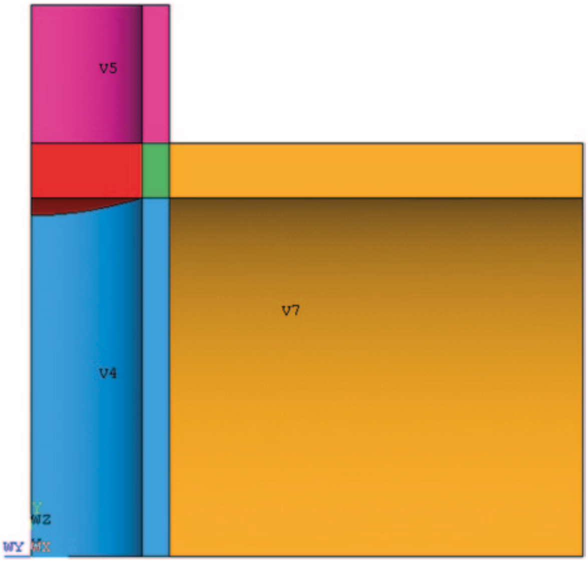


Fig. 9.14 Volumes after overlapping

– Reset the *Working Plane* to its original configuration by using the following menu path:

Utility Menu > WorkPlane > Offset WP by Increments

– *Offset WP* dialog box appears.
– Enter ***0, 90,0*** in the *second input* field; click on ***OK***.

• Use *Boolean Operator Overlap* to define the intersection volume between the two partial cylinders using the following menu path:

Main Menu > Preprocessor > Modeling > Operate > Booleans > Overlap > Volumes

– *Pick Menu* appears; pick the two volumes (or click on ***Pick All*** in the *Pick Menu*); click on ***OK***.
– In the *Pan Zoom Rotate Window*, click on ***Left*** button.
– Figure 9.14 shows the outcome of this action as it appears in the *Graphics Window*.

• Delete the excess volumes using the following menu path:

Main Menu > Preprocessor > Modeling > Delete > Volume and Below

– *Pick Menu* appears; pick the volumes to be deleted (carefully!); the volumes that need to be deleted are # **3** and # **4**.
– While picking the volumes, observe the number next to ***Volu No.*** in the *Pick Menu* change as the volumes are picked. This number corresponds to the last volume that is picked. If the wrong volume is picked, *right-click* inside the

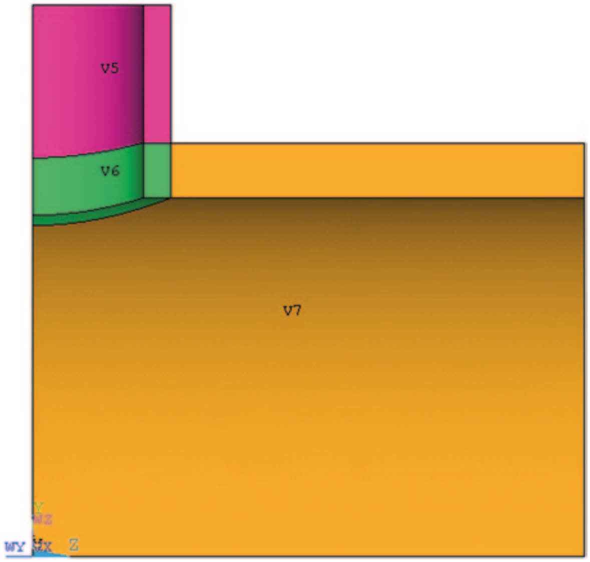


Fig. 9.15 Volumes after deletion

Graphics Window and observe that the mouse pointer changes from *upward arrow* (pick) to *downward arrow* (unpick); then unpick the wrong volume and pick the correct one.

- Once the volumes to be deleted (3 and 4) are picked, click on ***OK*** to finalize the deletion.
- Figure 9.15 shows the left view of the volumes after deletion as they appear in the *Graphics Window*.

- Set the element shape to be used as *hexahedron* (brick shaped) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesher Opts

- In the *Mesher Options* dialog box, click on the ***Mapped*** radio-button (Fig. 9.16); click on ***OK***.
- *Set Element Shape* dialog box appears (Fig. 9.17), click on ***OK***.

- In 3-D, *mapped meshing* can only be performed on volumes with **4** to **6** sides. Examination of the volumes in the current model reveals that volume 7 has 7 sides. In order to be able to perform mapped meshing on this volume, certain areas (2–5 pair) and lines (7–12, 5–10 pairs) must be *concatenated*.

- In order to be able to pick the areas and lines with ease, they should be in plain view. Rotate the model using rotation buttons on the *Pan Zoom Rotate Window* as follows: click on the negative ***X-rotation*** arrow button *twice* and negative ***Y-rotation*** arrow button *once*.

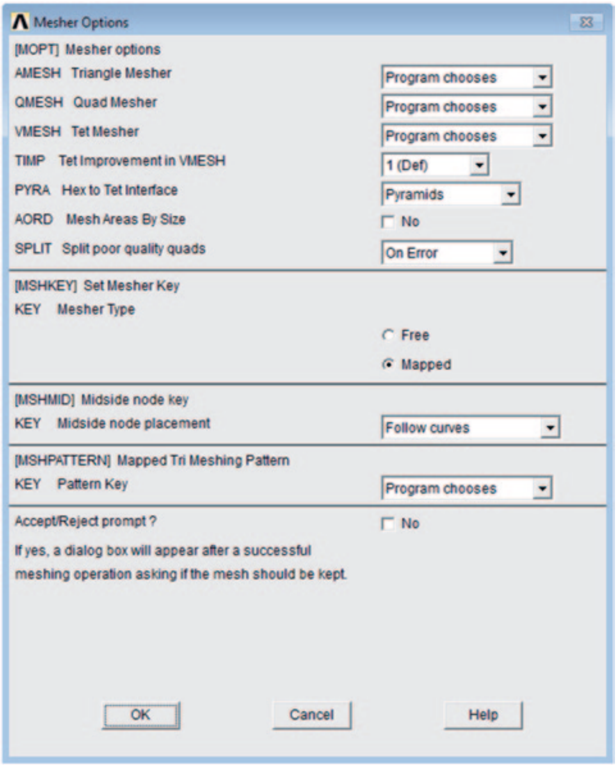


Fig. 9.16 *Mesher Options* dialog box

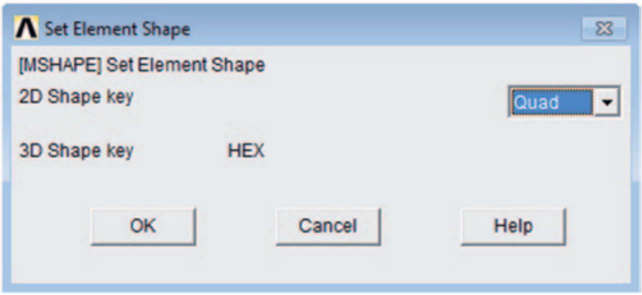


Fig. 9.17 *Set Element Shape* dialog box

– Figure 9.18 shows the outcome of this action as it appears in the *Graphics Window*.
– Turn off volume numbers and turn on area numbers using the following menu path:

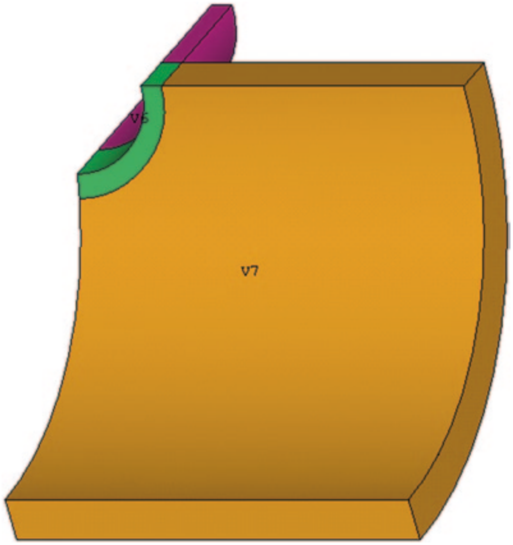


Fig. 9.18 Volumes viewed after rotations

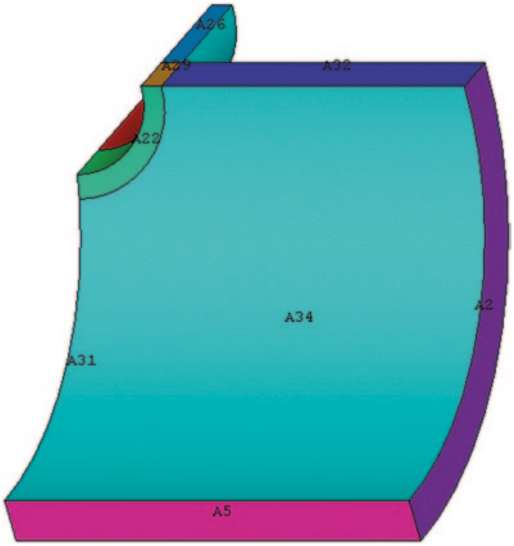


Fig. 9.19 Area plot with area numbers (*colors and numbers*) turned on

Utility Menu > PlotCtrls > Numbering

– Remove the checkmark by clicking on the square box next to ***VOLU Volume numbers*** (turning volume numbers off); place a checkmark on the square box next to ***AREA Area numbers*** (turning area numbers on); click on ***OK***.
– Plot areas using the following menu path:

Utility Menu > Plot > Areas

– Identify areas 2 and 5 (as shown in Fig. 9.19).
– *Concatenate* areas 2 and 5 using the following menu path:

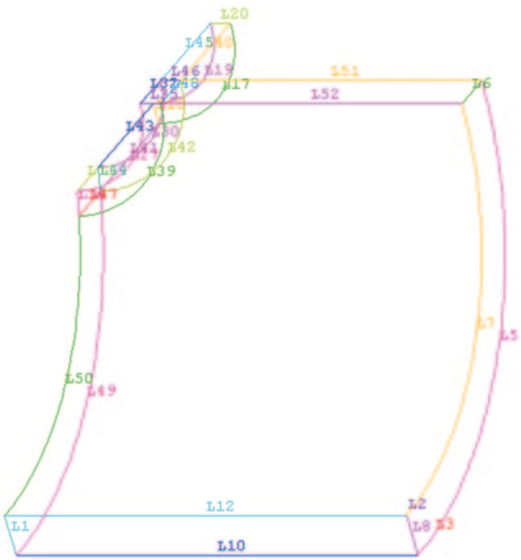


Fig. 9.20 Line plot with line numbers (*colors and numbers*) turned on

Main Menu > Preprocessor > Meshing > Concatenate > Areas

- Pick areas 2 and 5; click on ***OK***.
- Turn off area numbers and turn on line numbers.
- Plot lines.
- Identify lines 5, 7, 10, and 12 (as shown in Fig. 9.20).
- *Concatenate* lines 5 and 10 using the following menu path:

Main Menu > Preprocessor > Meshing > Concatenate > Lines

- Pick lines 5 and 10; click on ***OK***.
- Similarly, concatenate lines 7 and 12.

• Specify the number of elements on selected lines for mapped meshing. On lines 18 and 47, use **2** divisions; on line 7, use **6** divisions; on line 12, use **5** divisions; on line 39, use **11** divisions. For this action, use the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > Picked Lines

- In the text field of the *Pick Menu*, type ***18*** followed by hitting enter on the keyboard; type ***47*** and hit enter again.
- Click on ***Apply***.
- *Element Sizes on Lines* dialog box appears; type **2** in the text field corresponding to ***NDIV*** (the second text field); uncheck the first check box; click on ***Apply***.
- Repeat this procedure for lines 7, 12, and 39 with their corresponding number of divisions as **6**, **5**, and **11**, respectively. After specifying number of divisions for line 39, click on ***OK*** in the *Element Sizes on Lines* dialog box.

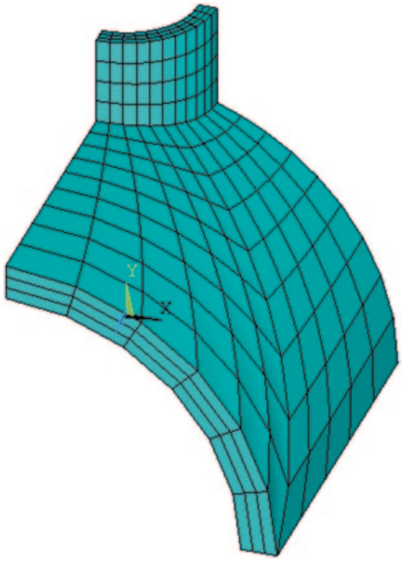


Fig. 9.21 Oblique view of the mesh

- Create the mesh using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Volumes > Mapped > 4 to 6 sided

  - In the *Pick Menu*, click on ***Pick All***.
  - Observe the mesh using the different options in the *Pan Zoom Rotate Window*.
  - Figure 9.21 shows the ***oblique*** view of the mesh.

Solution

- Apply uniform temperature to the entire model using the following menu path:

Main Menu > Solution > Define Loads > Apply > Thermal > Temperature > Uniform Temp

  - Dialog box appears; type ***450***; click on ***OK***.

- Apply boundary conditions.

  - Change the active coordinate system to the global cylindrical coordinate system using the following menu path:

Utility Menu > WorkPlane > Change Active CS to > Global Cylindrical

  - Select nodes at the inner surface of the tank for convective boundary conditions using the following menu path:

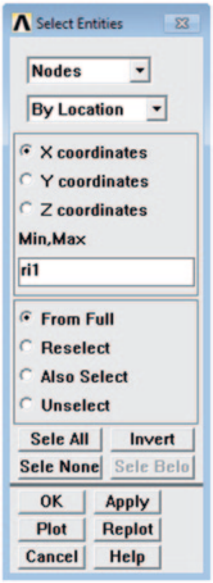


Fig. 9.22 *Select Entities* dialog box

Utility Menu > Select > Entities

- *Select Entities* dialog box appears.
- Choose ***Nodes*** from the first pull-down menu.
- Chose ***By Location*** from the second pull-down menu.
- Click on the radio-button ***X coordinates***. In cylindrical coordinates, x and y correspond to r and θ; therefore, the selection criterion will utilize the r-coordinate as long as the cylindrical coordinate system is active.
- Type ***ri1*** in the text field; click on ***OK*** (Fig. 9.22).
- Activate the *ANSYS Output Window* and examine the lines on the bottom to confirm that several nodes are selected as a result of this action (Fig. 9.23).
- Apply convective boundary conditions on the selected nodes using the following menu path:

Main Menu > Solution > Define Loads > Apply > Thermal > Convection > On Nodes

- Click on ***Pick All***.
- In the dialog box, type ***250/144*** for film coefficient (first text field) and ***450*** for bulk temperature (second text field); click on ***OK*** (Fig. 9.24).

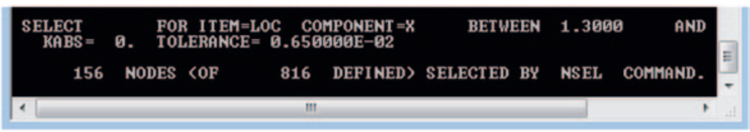


Fig. 9.23 Selection of nodes reported in the *Output Window*

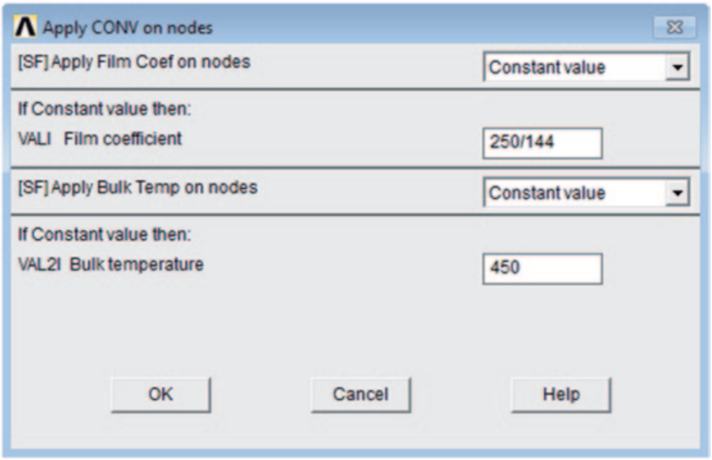


Fig. 9.24 *Apply CONV on nodes* dialog box

– Select nodes at the far edge of the tank for the temperature boundary conditions using the following menu path:

Utility Menu > Select > Entities

– *Select Entities* dialog box appears.
– Choose ***Nodes*** from the first pull-down menu.
– Chose ***By Location*** from the second pull-down menu.
– Click on the radio-button ***Z coordinates***.
– Type ***z1*** in the text field; click on ***OK***.
– Activate the *ANSYS Output Window* and examine the lines on the bottom to confirm that several nodes are selected as a result of this action.
– Apply temperature boundary conditions on the selected nodes using the following menu path:

Main Menu > Solution > Define Loads > Apply > Thermal > Temperature > On Nodes

– Click on ***Pick All***.
– In the dialog box, highlight ***TEMP*** and type ***450*** for ***VALUE Load Temp value***; click on ***OK***.
– Rotate the working plane ***-90°*** (*negative 90°*) about the *x*-axis using the following menu path:

Utility Menu > WorkPlane > Offset WP by Increments

– *Offset WP* dialog box appears.
– Type ***0, -90,0*** in the ***XY, YZ, ZX Angles*** text field (second text field); click on ***OK***.
– Define a local cylindrical coordinate system at the working plane origin using the following menu path:

Utility Menu > WorkPlane > Local Coordinate Systems > Create Local CS > At WP Origin

- Specify ***11*** for the ***KCN Ref number of new coord sys*** field (number 11 appears by default—leave it unchanged).
- Choose ***Cylindrical 1*** from the pull-down menu; click on ***OK***.
- Select nodes at the inner surface of the pipe for the convective boundary conditions using the following menu path:

Utility Menu > Select > Entities

- *Select Entities* dialog box appears.
- Choose ***Nodes*** from the first pull-down menu.
- Choose ***By Location*** from the second pull-down menu.
- Click on the radio-button ***X coordinates***.
- Type ***ri2*** in the text field; click on ***OK***.
- Apply the convective boundary condition with a temperature-dependent film coefficient on the selected nodes using the following menu path:

Main Menu > Solution > Define Loads > Apply > Thermal > Convection > On Nodes

- Click on ***Pick All***.
- In the dialog box, type ***-2*** (*negative 2*) for film coefficient and ***100*** for bulk temperature; click on ***OK***.
- Select everything using the following menu path:

Utility Menu > Select > Everything

- Obtain the solution.
  - Although steady-state (no time dependence), this problem is nonlinear due to the existence of the temperature-dependent film coefficient and thus requires an iterative solution. Therefore, time is introduced as an auxiliary variable, and related parameters are specified.
  - Specify *automatic time stepping* using the following menu path:

Main Menu > Solution > Load Step Opts > Time/Frequenc > Time and Substps

- Type ***50*** in the *second text field* corresponding to ***NSUBST*** (Fig. 9.25).
- Click on the ***ON*** radio-button for *Automatic time stepping*; click on ***OK***.
- Save using the following menu path:

Utility Menu > File > Save as

- Start the solution using the following menu path:

Main Menu > Solution > Solve > Current LS

- *Confirmation Window* appears along with *Status Report Window*.
- Review status; if OK, close the *Status Report Window* and click on ***OK*** in the *Confirmation Window*.
- Wait until ANSYS responds with ***Solution is done!***

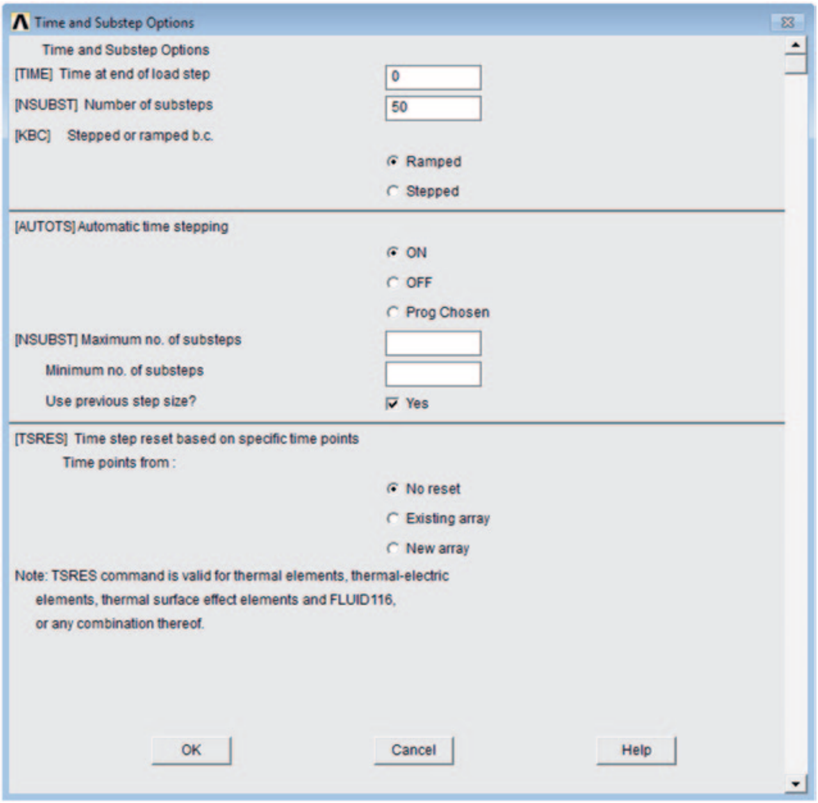


Fig. 9.25 *Time and Substep Options* dialog box

Postprocessing

- Review temperature contours using the following menu path:

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu

  – Select ***DOF Solution*** and ***Nodal Temperature***; click on ***OK***.
  – The temperature contour plot is shown in Fig. 9.26 as it appears in the *Graphics Window*.

- Review thermal flux vectors using the following menu path:

Main Menu > General Postproc > Plot Results > Vector Plot > Predefined

  – Select ***Flux and gradient*** from the left list and ***Thermal flux TF*** from the right list; click on ***OK***.
  – Figures 9.27 and 9.28 show the flux vectors as they appear in the *Graphics Window*, viewed from top and left, respectively.

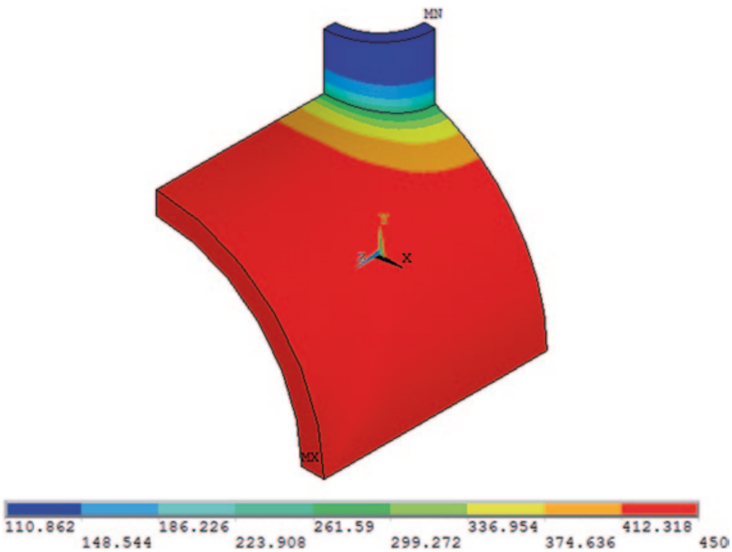


Fig. 9.26 Contour plot of temperatures

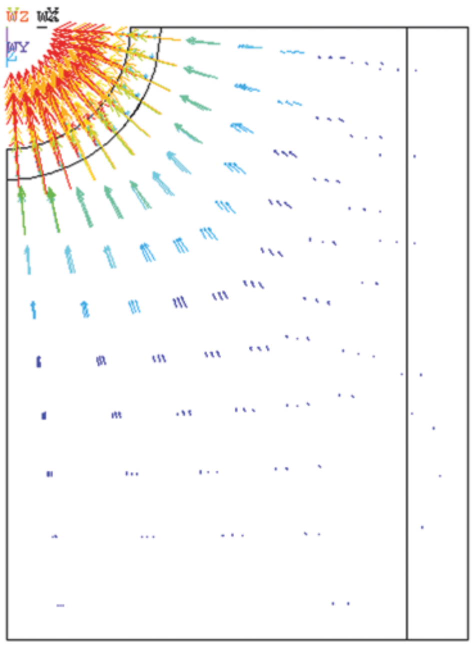


Fig. 9.27 *Top* view of thermal flux vectors

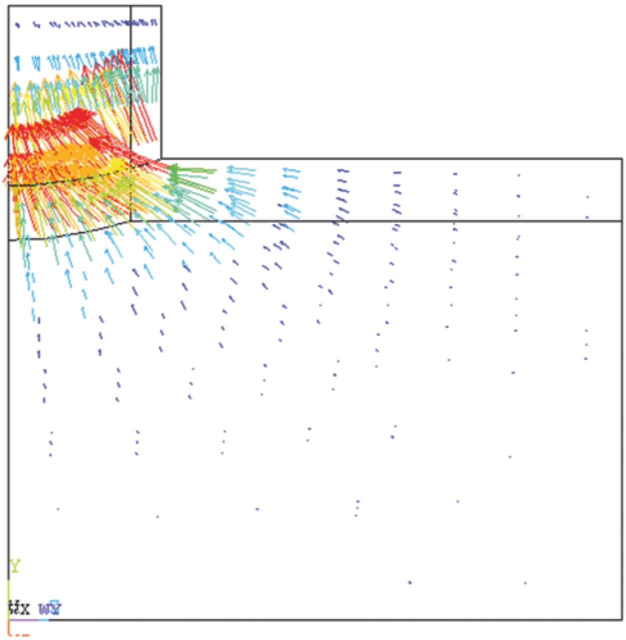


Fig. 9.28 *Left* view of thermal flux vectors

9.1.1.2 Analysis of a Window Assembly

Consider a window assembly composed of concrete, glass, and aluminum, as shown in Fig. 9.29. The inside and outside ambient temperatures are 70 and 0 °F, respectively, as indicated in Fig. 9.30. The film coefficient along the boundaries facing inside is 0.0139 Btu/hr-in^2-°F whereas the film coefficient along the boundaries facing outside is 0.0347 Btu/hr-in^2-°F. The thermal conductivities of glass, concrete, and aluminum are 0.05, 0.06, and 11 Btu/hr-in-°F, respectively. The goal is to determine the steady-state temperature distribution in the system.

Model Generation

- Define the element type (**`ET`** command) using the following menu path:

Main Menu > Preprocessor > Element Type > Add/Edit/Delete

  - *Element Types* dialog box appears; click on ***Add***.
  - Select ***Solid*** immediately below ***Thermal*** in the left list and ***Quad 4node 55*** in the right list; click on ***OK***.
  - Click on ***Close***.

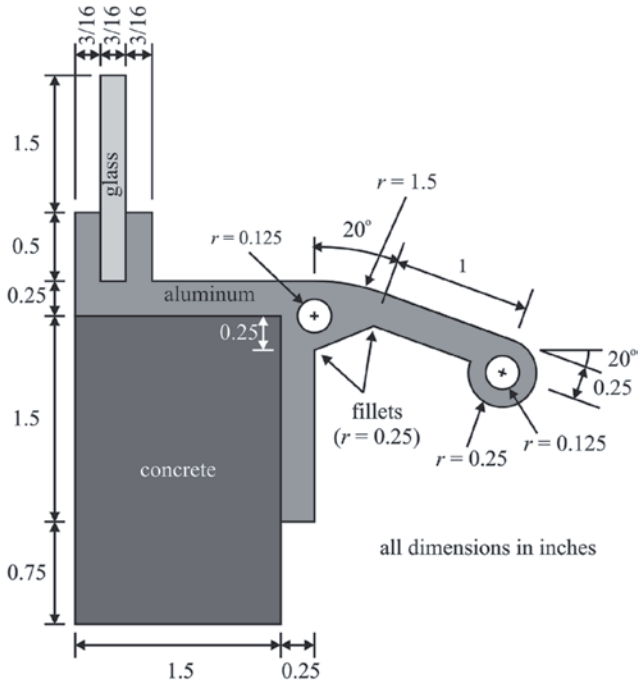


Fig. 9.29 The geometry of the window assembly

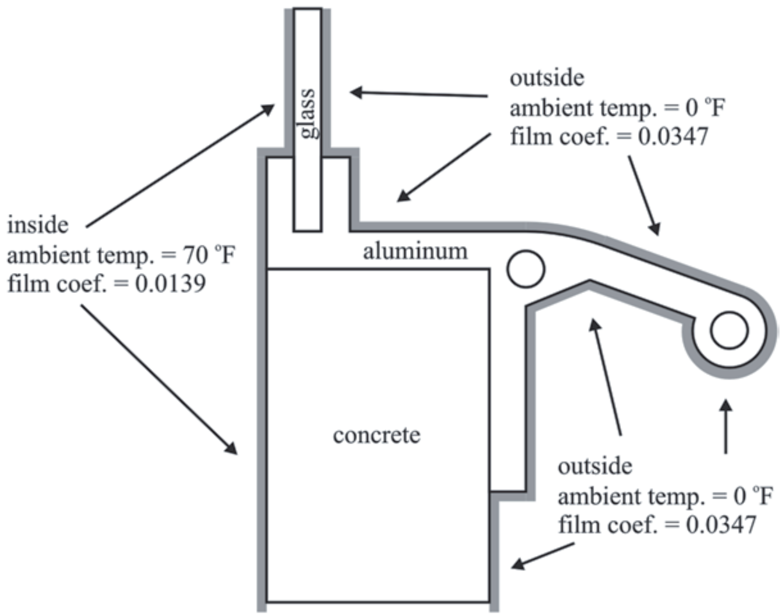


Fig. 9.30 Boundary conditions for the window assembly

Table 9.3 Coordinates defining the rectangles

Area Number	X1	X2	Y1	Y2
	(in.)			
1	0	1.5	0	2.25
2	1.5	1.75	0.75	2.25
3	0	1.75	2.25	2.5
4	0	3/16	2.5	3
5	3/16	3/8	2.5	4
6	3/8	9/16	2.5	3

- Specify thermal conductivities for aluminum (material 1), glass (material 2), and concrete (material 3) (`MP` command) using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

  – In the *Define Material Model Behavior* dialog box, in the right window, successively left-click on ***Thermal***, ***Conductivity***, and ***Isotropic***, which brings up another dialog box.
  – Enter ***11*** for ***KXX***; click on ***OK***.
  – Add new material for glass. In the *Define Material Model Behavior* dialog box, define a new material reference number using the following menu path:

Material > New Model

  – A dialog box appears with the ***material reference number*** set to the next available number (in this case 2); click on ***OK***.
  – Note the new entry for ***Material Model Number 2*** in the left list in the *Define Material Model Behavior* dialog box.
  – In the right window, successively left-click on ***Thermal***, ***Conductivity***, and ***Isotropic***, which brings up another dialog box.
  – Enter ***0.05*** for ***KXX***; click on ***OK***.
  – Add new material for concrete and specify the thermal conductivity as 0.06 following the same procedure used for glass above.
  – Close the *Define Material Model Behavior* dialog box by using the following menu path:

Material > Exit

- Create the areas (`RECTNG` command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By Dimensions

  – *Create Rectangle by Dimensions* dialog box appears; enter ***0*** and ***1.5*** for the ***X-coordinates*** (***X1***, ***X2***) and ***0*** and ***2.25*** for the ***Y-coordinates*** (***Y1***, ***Y2***); click on ***Apply***.
  – Referring to Table 9.3, create the remaining six areas. After creating the sixth area, click on ***OK*** (instead of ***Apply***).

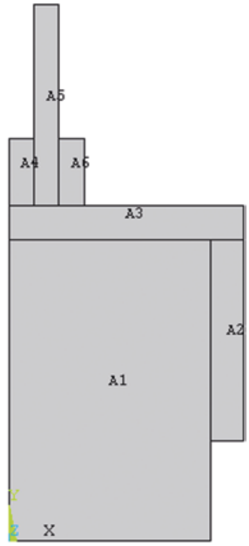


Fig. 9.31 Six areas as they appear in the *Graphics Window*

– Figure 9.31 shows the six areas as they appear in the *Graphics Window*.

- Define a local coordinate system (**LOCAL** command) using the following menu path:

Utility Menu > WorkPlane > Local Coordinate Systems > Create Local CS > At Specified Loc

– *Pick Menu* appears; type ***1.75,1*** in the text box and hit enter (as shown in Fig. 9.32); click on ***OK***.
– *Create Local CS at Specified Location* dialog box appears; select ***Cylindrical 1*** from the ***KCS Type of coordinate system*** pull-down menu; click on ***OK***.
– Observe that the local coordinate system appears in the *Graphics Window*.

- Create a keypoint in the local coordinate system (**K** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Keypoints > In Active CS

– *Create Keypoints in Active Coordinate System* dialog box appears; leave the ***NPT Keypoint number*** field blank (ANSYS issues the lowest available keypoint number to the newly created keypoints) and enter ***1.5*** and ***70*** for ***X, Y, Z Location in active CS*** (leave the ***Z*** field blank). Since the active coordinate system is *cylindrical*, ***X*** corresponds to the radius from the local coordinate system origin and ***Y*** corresponds to the angle measured from the *x*-axis in the counterclockwise direction.
– Upon clicking on ***OK***, ANSYS creates the new keypoint and plots all of the keypoints, as shown in Fig. 9.33. Note that the most recently created keypoint is 25.

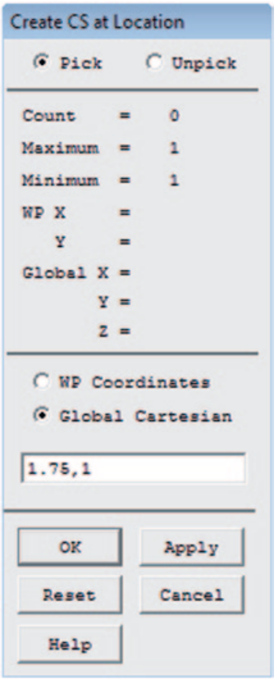


Fig. 9.32 *Pick Menu* for creating a local coordinate system at a location

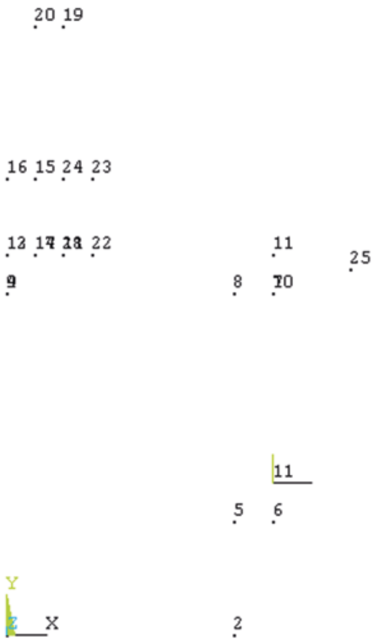


Fig. 9.33 Keypoints as they appear in the *Graphics Window*

- The slanted rectangular area and the circular portion at the end for the aluminum component are created next. For this purpose, the *Working Plane* is moved to an existing keypoint and rotated for the rectangular area. It is then moved one more time for the creation of the circular area.

  – Activate the global Cartesian coordinate system (**`CSYS`** command) using the following menu path:

Utility Menu > WorkPlane > Change Active CS to > Global Cartesian

  – Move the *Working Plane* origin to keypoint 25 (**`KWPAVE`** command) using the following menu path:

Utility Menu > WorkPlane > Offset WP to > Keypoints

  – *Pick Menu* appears; pick keypoint 25; click on ***OK***. Observe the *Working Plane* origin at keypoint 25 in the *Graphics Window*.
  – Rotate the *Working Plane* by -20° about the *z*-axis by (**`WPROTA`** command) using the following menu path:

Utility Menu > WorkPlane > Offset WP by Increments

  – *Offset WP* dialog box appears, which has two main action areas, i.e., translations in the *x*-, *y*-, and *z*-directions, and rotations about *x*-, *y*-, and *z*-axes. Each of these areas also has a *slider* that controls the rate at which translations or rotations are applied. By default, the *translation slider* is set to ***1*** and the *rotation slider* is set to ***30***. By moving the *rotation slider* to the left (or using the arrow keys), set the rate to be ***20°***. Upon clicking on the negative ***Z***-button, the *Working Plane* is rotated −20° about the *z*-axis.
  – Create the rectangular area (**`RECTNG`** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By Dimensions

  – *Create Rectangle by Dimensions* dialog box appears; enter ***0*** and ***1*** for the ***X-coordinates*** (***X1***, ***X2***) and ***–0.25*** and ***0*** for the ***Y-coordinates*** (***Y1***, ***Y2***); click on ***OK***.
  – Turn keypoint numbering on (**`/PNUM`** command) using the following menu path:

Utility Menu > PlotCtrls > Numbering

  – *Plot Numbering Controls* dialog box appears; click on the box next to ***KP Keypoint numbers***, which places a checkmark in the box. Select ***Numbers only*** from the ***[/NUM] Numbering shown with*** pull-down menu; click on ***OK***.
  – Plot keypoints (**`KPLOT`** command) using the following menu path:

Utility Menu > Plot > Keypoints > Keypoints

  – Move the *Working Plane* origin to keypoint 27 (**`KWPAVE`** command) using the following menu path:

Utility Menu > WorkPlane > Offset WP to > Keypoints

- *Pick Menu* appears; pick keypoint 27; click on ***OK***. Observe the *Working Plane* origin at keypoint 27 in the *Graphics Window*.
- Create the hollow circular area (**PCIRC** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Circle > By Dimensions

- *Circular Area by Dimensions* dialog box appears; enter ***0.25*** for ***RAD1 Outer radius*** and ***0.125*** for ***RAD2 Optional inner radius***. Click on ***OK***.
- Turn keypoint numbering off and area numbering on (**/PNUM** command) using the following menu path:

Utility Menu > PlotCtrls > Numbering

- *Plot Numbering Controls* dialog box appears; click on the box next to ***KP Keypoint numbers*** to remove the checkmark (turning keypoint numbering off) and click on the box next to ***AREA Area numbers*** to place a checkmark (turning area numbering on). Click on ***OK***.
- Plot areas (**APLOT** command) using the following menu path:

Utility Menu > Plot > Areas

- Figure 9.34 shows the areas as they appear in the *Graphics Window*.

• Overlap areas 7 and 8 (**AOVLAP** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Operate > Booleans > Overlap > Areas

- *Pick Menu* appears; pick areas 7 and 8; click on ***OK*** in the *Pick Menu*.
- Figure 9.35 shows the areas before and after overlapping.
- Delete area 9 and keypoints and lines attached to it (**ADELE** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Delete > Area and Below

- *Pick Menu* appears; pick area 9; click on ***OK*** in the *Pick Menu*.
- Observe that area 9 disappears in the *Graphics Window*.

• Reset the *Working Plane* to its default configuration (**WPSTYL** command) using the following menu path:

Utility Menu > WorkPlane > Align WP with > Global Cartesian

- *Working Plane* is aligned with global Cartesian coordinate system (both the origin and the orientation).
- Toggle the *Working Plane* display off (**WPSTYL** command) using the following menu path:

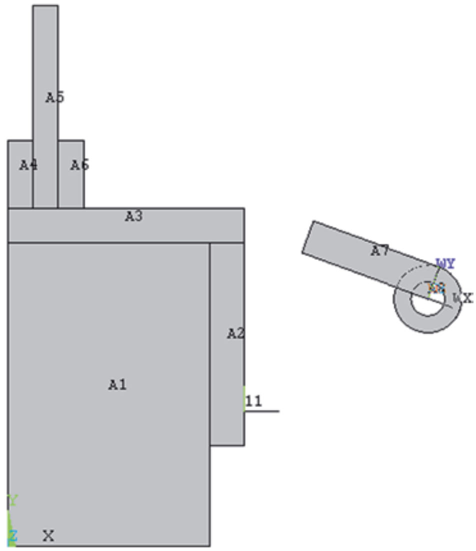


Fig. 9.34 Areas as they appear in the *Graphics Window*

Utility Menu > WorkPlane > Display Working Plane

– If a checkmark appears next to the menu entry ***Display Working Plane***, which means the display is on, click on the menu entry to toggle off the display.

- Create a keypoint (**K** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Keypoints > In Active CS

– *Create Keypoints in Active Coordinate System* dialog box appears; enter ***50*** in the ***NPT Keypoint number*** field and ***1.75*** and ***2*** for ***X, Y, Z Location in active CS*** (leave the **Z** field blank). Click on ***OK***.

- Activate local coordinate system 11 (created earlier) (**CSYS** command) using the following menu path:

Utility Menu > Change Active CS to > Specified Coord Sys

– *Change Active CS to Specified CS* dialog box appears; enter ***11*** for ***KCN Coordinate system number***; click on ***OK***.

- Turn keypoint numbering on and area numbering off. Plot keypoints.
- Create a line between keypoints 11 and 29 (**L** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Lines > Lines > In Active Coord

– *Pick Menu* appears; pick keypoints 11 and 29. Keypoint 29 is coincident with keypoint 25; therefore, when attempting to pick keypoint 29, ANSYS issues

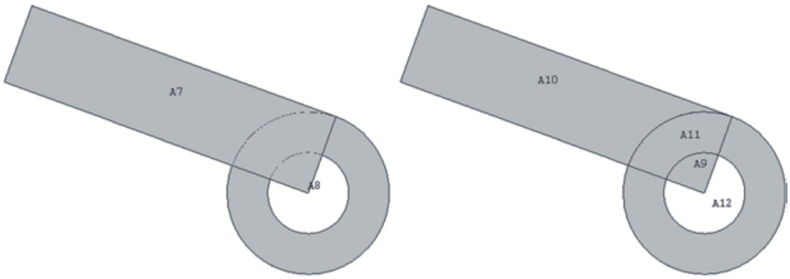


Fig. 9.35 Areas before (*left*) and after (*right*) overlapping

a warning to the user that there are more than one keypoints at the picked location and that the currently picked keypoint is keypoint 25. Click on ***Next*** in this *Multiple_Entities Warning Window*, which picks the next higher numbered keypoint (keypoint 29 in this case) at the coincident location. Click on ***OK*** in the *Multiple_Entities Warning Window*; click on ***OK*** in the *Pick Menu*.

– The line that is just created is a curved line. This is because it is created in the active coordinate system, which is a local cylindrical coordinate system.
– Reset the active coordinate system to be global Cartesian (**`CSYS`** command) using the following menu path:

Utility Menu > WorkPlane > Change Active CS to > Global Cartesian

• Create an area using keypoints 50, 26, 29, and 11 (**`A`** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Arbitrary > Through KPs

– *Pick Menu* appears; pick the keypoints listed above. Note that when picking keypoint 29, since it is coincident with keypoint 25, ANSYS warns the user that there are two keypoints at the picked location. Similar to the previous case, click on ***Next*** to pick keypoint 29. When finished with picking all four keypoints, click on ***OK*** in the *Pick Menu* and observe that area 7 appears in the *Graphics Window*.
– Since a curved line was created between keypoints 11 and 29 previously, and both of these keypoints are used in the definition of area 7, the boundary of the area between keypoints 11 and 29 is a curved one. If the line were not created, then this boundary would have been a straight line.

The circular hole within the aluminum component near the top-right corner of the concrete block is created next. For this purpose, the *Working Plane* origin is moved to keypoint 10. After creating the circle, areas around the circle are overlapped and the ones defining the hole are deleted.

– Plot keypoints (**`KPLOT`** command) using the following menu path:

Utility Menu > Plot > Keypoints > Keypoints

- Move the *Working Plane* origin to keypoint 10 (**KWPAVE** command) using the following menu path:

Utility Menu > WorkPlane > Offset WP to > Keypoints

- *Pick Menu* appears; pick keypoint 10 (keypoint 10 is coincident with keypoint 7 and they are located between keypoints 11 and 50) and click on ***OK***. Observe the *Working Plane* origin at keypoint 10 in the *Graphics Window*.
- Create a circular area (**PCIRC** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Circle > By Dimensions

- *Circular Area by Dimensions* dialog box appears; enter ***0.125*** for ***RAD1 Outer radius*** and remove any leftover numbers from the ***RAD2 Optional inner radius*** field. Click on ***OK***.
- Reset the *Working Plane* to its default configuration (**WPSTYL** command) using the following menu path:

Utility Menu > WorkPlane > Align WP with > Global Cartesian

- *Working Plane* is aligned with global Cartesian coordinate system (both the origin and the orientation).
- Toggle the *Working Plane* display off (**WPSTYL** command) using the following menu path:

Utility Menu > WorkPlane > Display Working Plane

- If a checkmark appears next to the menu entry ***Display Working Plane***, which means the display is on, click on the menu entry to toggle off the display.
- Turn area numbering on (**/PNUM** command) and keypoint numbering off using the following menu path:

Utility Menu > PlotCtrls > Numbering

- *Plot Numbering Controls* dialog box appears; click on the box next to ***AREA Area numbers*** to place a checkmark (turning area numbering on). Remove the checkmark for keypoints. Click on ***OK***.
- Plot areas (**APLOT** command) using the following menu path:

Utility Menu > Plot > Areas

- Overlap areas 2, 3, 7, and 8 (**AOVLAP** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Operate > Booleans > Overlap > Areas

- *Pick Menu* appears; pick areas 2, 3, 7, and 8 and click on ***OK*** in the *Pick Menu*.
- Figures 9.36 and 9.37 show the areas before and after overlapping, respectively.

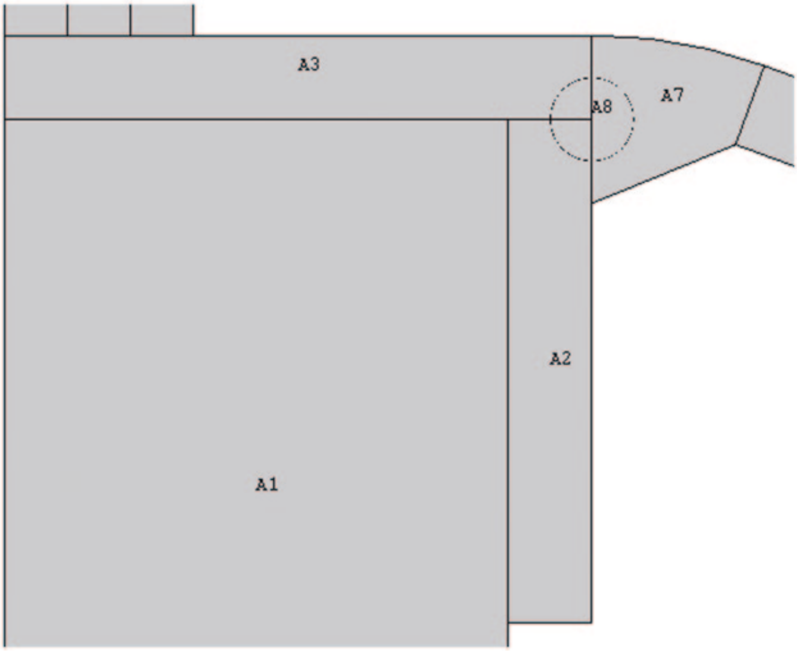


Fig. 9.36 Areas before overlapping

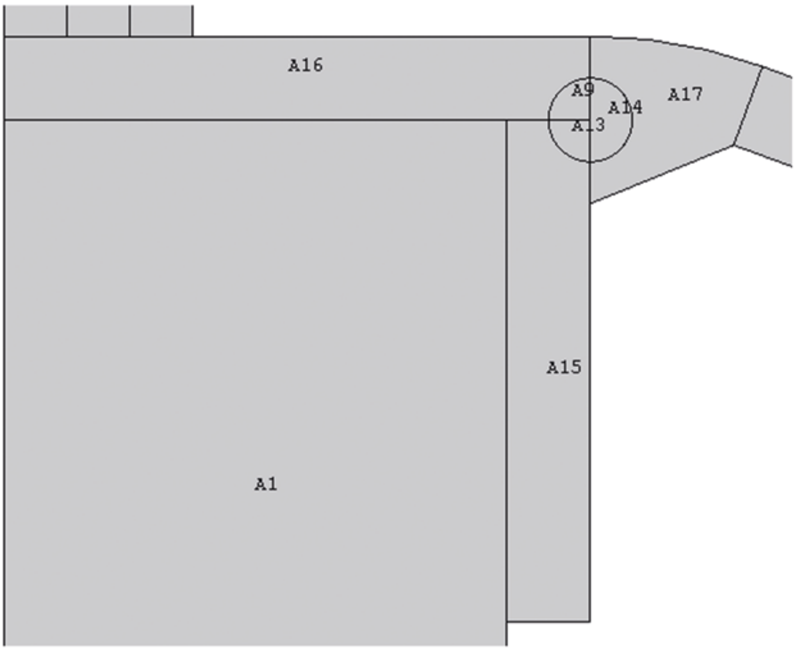


Fig. 9.37 Areas after overlapping

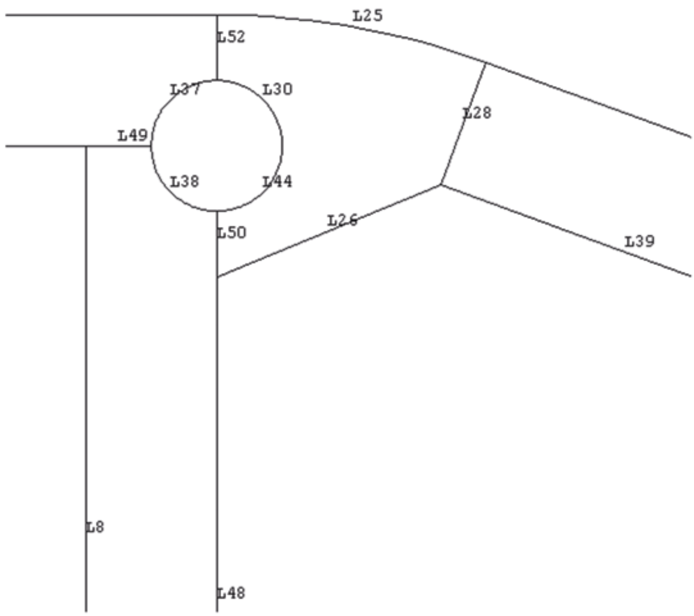


Fig. 9.38 Lines around the hole

- Delete areas 9, 13, and 14 and the keypoints and lines attached to them (**`ADELE`**command) using the following menu path:

Main Menu > Preprocessor > Modeling > Delete > Area and Below

- *Pick Menu* appears; pick areas 9, 13, and 14; click on ***OK*** in the *Pick Menu*.
- Observe that the areas disappear in the *Graphics Window*.

• Two fillets are created next.

- Turn line numbering on (**`/PNUM`** command) using the following menu path:

Utility Menu > PlotCtrls > Numbering

- *Plot Numbering Controls* dialog box appears; click on the box next to ***LINE Line numbers*** to place a checkmark (turning line numbering on). Make sure no other numbering (keypoint and/or area) is on. Click on ***OK***.
- Plot lines (**`LPLOT`** command) using the following menu path:

Utility Menu > Plot > Lines

- Zoom-in to the region around the hole inside the aluminum component near the top-right corner of the concrete, as shown in Fig. 9.38.

- Create line fillets between lines 48 and 26 and between lines 26 and 39, both of them having a radius of 0.25 in (**LFILLT** command), using the following menu path:

Main MenuPreprocessor > Modeling > Create > Lines > Line Fillet

- *Pick Menu* appears, prompting the user to pick two intersecting lines. Pick lines ***48*** and ***26***; click on ***OK*** in the *Pick Menu*.
- *Line Fillet* dialog box appears; enter ***0.25*** for ***RAD Fillet radius*** and ***55*** for ***PCENT Number to assign to generated keypoint at fillet center***. Click on ***Apply*** to create the next fillet.
- *Pick Menu* reappears. Repeat the same procedure by picking lines 26 and 39, using the same radius (***0.25***), and using ***60*** for ***PCENT***. When finished, click on ***OK*** in the *Line Fillet* dialog box.
- Turn keypoint numbering on (**/PNUM** command) using the following menu path:

Utility Menu > PlotCtrls > Numbering

- *Plot Numbering Controls* dialog box appears; click on the box next to ***KP Keypoint numbers*** to place a checkmark. Make sure no other numbering (line and/or area) is on. Click on ***OK***.
- Plot keypoints (**KPLOT** command) using the following menu path:

Utility Menu > Plot > Keypoints > Keypoints

- Create areas (for fillets) through keypoints (**A** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Arbitrary > Through KPs

- *Pick Menu* appears; pick keypoints 7, 10, and 50; click on ***Apply*** in the *Pick Menu*.
- *Pick Menu* remains active; pick keypoints 40, 41, and 26; click on ***OK*** in the *Pick Menu*.
- Turn area numbering on (**/PNUM** command) using the following menu path:

Utility Menu > PlotCtrls > Numbering

- *Plot Numbering Controls* dialog box appears; click on the box next to ***AREA Area numbers*** to place a checkmark (turning area numbering on). Make sure no other numbering (keypoint and/or line) is on. Click on ***OK***.
- Plot areas (**APLOT** command) using the following menu path:

Utility Menu > Plot > Areas

- Areas appear in the *Graphics Window,* as shown in Fig. 9.39.
- Add areas 2, 3, 10, 15, and 17 (**AADD** command) using the following menu path:

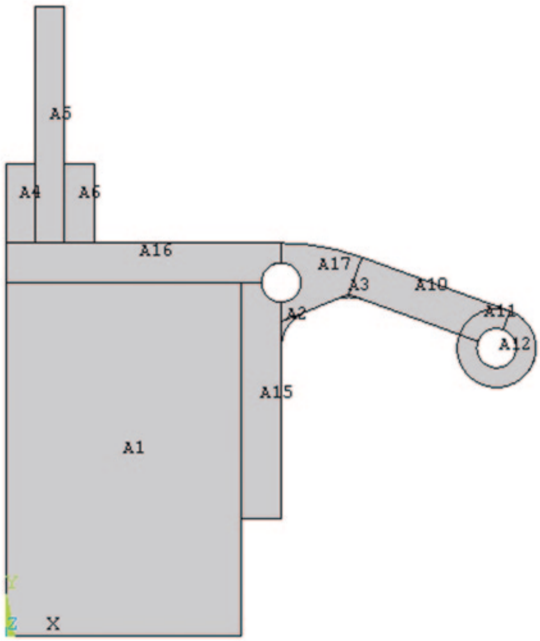


Fig. 9.39 Areas after fillet creation

Main Menu > Preprocessor > Modeling > Booleans > Add > Areas

- *Pick Menu* appears; pick areas 2, 3, 10, 15, and 17; click on ***OK*** in the *Pick Menu*.
- Glue all areas (**AGLUE** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Booleans > Glue > Areas

- *Pick Menu* appears; click on ***Pick All*** in the *Pick Menu*.
- Figure 9.40 shows the areas after adding and gluing operations as they appear in the *Graphics Window*.

• Specify size controls for meshing, compress entity numbers, and create the mesh.

- Specify global element size (**ESIZE** command) usi8.5ng the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Global > Size

- *Global Element Sizes* dialog box appears; enter ***1/16*** for ***SIZE Element edge length***; click on ***OK***.
- Turn keypoint numbering on (**/PNUM** command) using the following menu path:

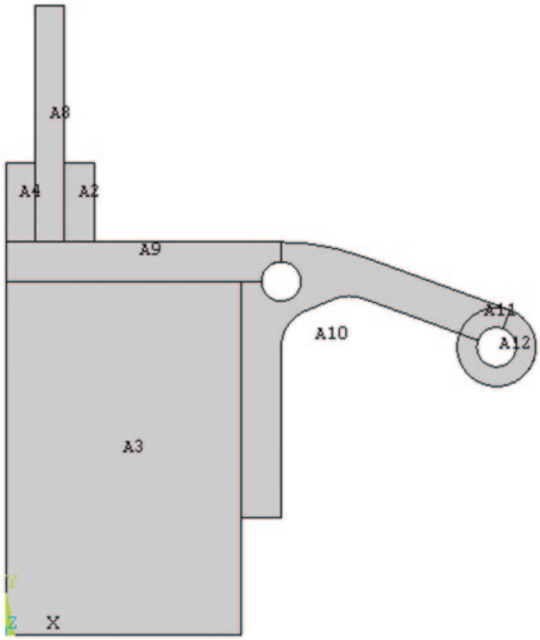


Fig. 9.40 Areas after adding and gluing operations

Utility Menu > PlotCtrls > Numbering

- *Plot Numbering Controls* dialog box appears; click on the box next to ***KP Keypoint numbers*** to place a checkmark. Make sure no other numbering (line and/or area) is on. Click on ***OK***.
- Plot keypoints (**`KPLOT`** command) using the following menu path:

Utility Menu > Plot > Keypoints > Keypoints

- Specify element size near keypoints 1 and 2 as 0.25 (**`KESIZE`** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Keypoints > Picked KPs

- *Pick Menu* appears; pick keypoints 1 and 2 (these are the bottom corners of the concrete block—keypoint 1 is at the global origin); click on ***OK*** in the *Pick Menu*.
- *Element Size at Picked Keypoints* dialog box appears; enter ***0.25*** for ***SIZE Element edge length***; click on ***OK***.
- Compress entity numbers (**`NUMCMP`** command) using the following menu path:

Main Menu > Preprocessor > Numbering Ctrls > Compress Numbers

- *Compress Numbers* dialog box appears; select ***All*** from the ***Label Item to be compressed*** pull-down menu; click on ***OK***.

– Turn area numbering on (**/PNUM** command) using the following menu path:

Utility Menu > PlotCtrls > Numbering

– *Plot Numbering Controls* dialog box appears; click on the box next to ***AREA Area numbers*** to place a checkmark (turning area numbering on). Make sure no other numbering (keypoint and/or line) is on. Click on ***OK***.
– Plot areas (**APLOT** command) using the following menu path:

Utility Menu > Plot > Areas

– Areas appear in the *Graphics Window*, as shown in Fig. 9.41.
– Mesh is generated next.
– Change default material attribute to 3 (**MAT** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh Attributes > Default Attribs

– *Meshing Attributes* dialog box appears; select ***3*** from the ***[MAT] Material number*** pull-down menu; click on ***OK***. Mesh generated after this point will have material reference number 3, until it is changed to another number.
– Create mesh for area 2 (concrete) (**AMESH** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Areas > Free

– *Pick Menu* appears; pick area 2; click on ***OK***.
– Plot areas (**APLOT** command) using the following menu path:

Utility Menu > Plot > Areas

– Change default material attribute to 2 (**MAT** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh Attributes > Default Attribs

– *Meshing Attributes* dialog box appears; select ***2*** from the ***[MAT] Material number*** pull-down menu; click on ***OK***.
– Create mesh for area 4 (glass) (**AMESH** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Areas > Free

– *Pick Menu* appears; pick area 4; click on ***OK***.
– Plot areas (**APLOT** command) using the following menu path:

Utility Menu > Plot > Areas

– Change default material attribute to 1 (**MAT** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh Attributes > Default Attribs

– *Meshing Attributes* dialog box appears; select ***1*** from the ***[MAT] Material number*** pull-down menu; click on ***OK***.

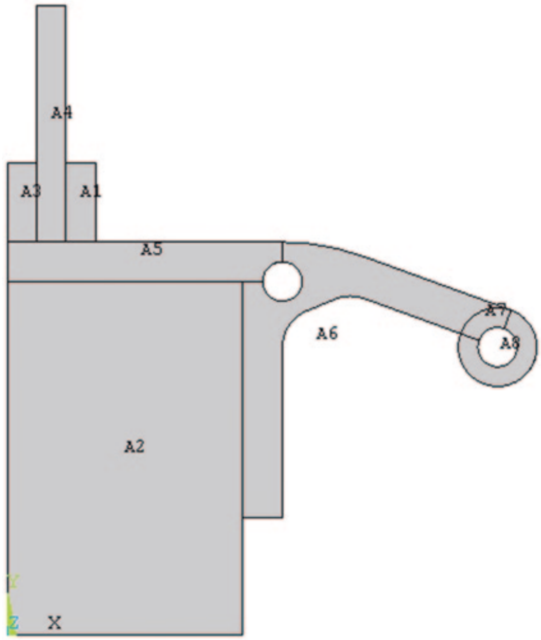


Fig. 9.41 Areas after entity number compression

– Create mesh for the remaining areas (aluminum) (**AMESH** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Areas > Free

– *Pick Menu* appears; pick the remaining areas (1, 3, 5–8); click on ***OK***.
– A warning message may appear; if so, close it.
– Turn the element numbering on so that the elements are plotted with different colors based on their material attribute (**/PNUM** command) using the following menu path:

Utility Menu > PlotCtrls > Numbering

– *Plot Numbering Controls* dialog box appears; select ***Material numbers*** from the ***Elem/Attrib numbering*** pull-down menu; select ***Colors only*** from the ***[/NUM] Numbering shown with*** pull-down menu. Click on ***OK***.
– Plot elements (**EPLOT** command) using the following menu path:

Utility Menu > Plot > Elements

– Elements appear in the *Graphics Window*, as shown in Fig. 9.42.

Solution

• Apply convective boundary conditions.

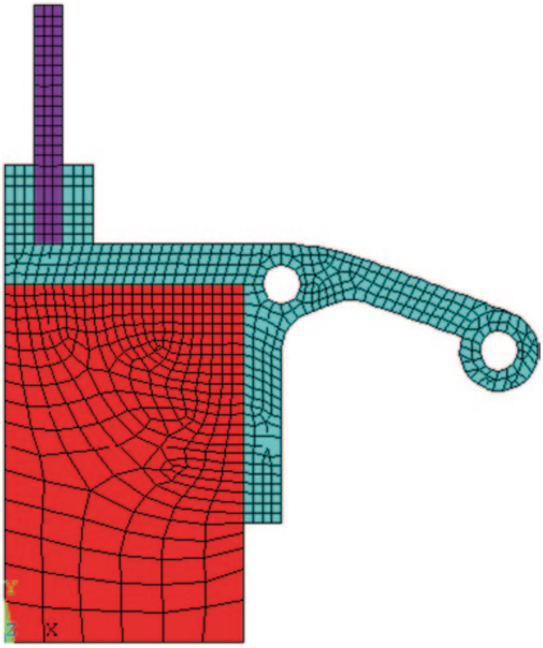


Fig. 9.42 Elements after meshing plotted with different colors based on their material number attributes

- Turn line numbering on (**/PNUM** command) using the following menu path:

Utility Menu > PlotCtrls > Numbering

- *Plot Numbering Controls* dialog box appears; click on the box next to ***LINE Line numbers*** to place a checkmark. Select ***No numbering*** from the ***Elem/Attrib numbering*** pull-down menu, and select ***Numbers only*** from the ***[/NUM] Numbering shown with*** pull-down menu. Click on ***OK***.
- Plot lines (**LPLOT** command) using the following menu path:

Utility Menu > Plot > Lines

- Apply convective boundary conditions on the lines along the side facing outside (as shown in Fig. 9.30) (**SFL** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Thermal > Convection > On Lines

- *Pick Menu* appears; pick the lines shown in Fig. 9.43; click on ***OK*** in the *Pick Menu*.
- *Apply CONV on lines* dialog box appears; type ***0.0347*** for ***VALI Film coefficient*** and ***0*** for ***VAL2I Bulk temperature***; click on ***OK***.
- Apply convective boundary conditions on the lines along the side facing inside (as shown in Fig. 9.30) (**SFL** command) using the following menu path:

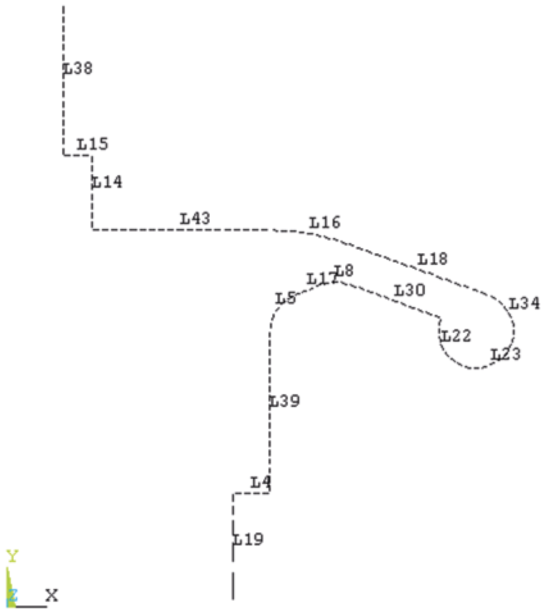


Fig. 9.43 Lines facing outside

Main Menu > Solution > Define Loads > Apply > Thermal > Convection > On Lines

- *Pick Menu* appears; pick the lines shown in Fig. 9.44; click on ***OK*** in the *Pick Menu*.
- *Apply CONV on lines* dialog box appears; type ***0.0139*** for ***VALI Film coefficient*** and ***70*** for ***VAL2I Bulk temperature***; click on ***OK***.

- Obtain the solution (**`SOLVE`** command) using the following menu path:

Main Menu > Solution > Solve > Current LS

- *Confirmation Window* appears along with *Status Report Window*.
- Review status; if OK, close the *Status Report Window*; click on ***OK*** in the *Confirmation Window*.
- Wait until ANSYS responds with ***Solution is done!***

Postprocessing

- Turn colors on (**`/PNUM`** command) using the following menu path:

Utility Menu > PlotCtrls > Numbering

- *Plot Numbering Controls* dialog box appears; click on the box next to ***LINE Line numbers*** to remove the checkmark. Select ***Colors only*** from the ***[/NUM] Numbering shown with*** pull-down menu; click on ***OK***.

Fig. 9.44 Lines facing inside

- Review temperature contours (**PLNSOL** command) using the following menu path:

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu

  - *Contour Nodal Solution Data* dialog box appears. Successively select ***DOF Solution*** and ***Nodal Temperatue***; click on ***OK***.
  - The temperature contour plot is shown in Fig. 9.45 as it appears in the *Graphics Window*.

- Review thermal flux and gradient vectors using the following menu path:

Main Menu > General Postproc > Plot Results > Vector Plot > Predefined

  - *Vector Plot of Predefined Vectors* dialog box appears. Select ***Flux & gradient*** from the left list and ***Thermal flux TF*** from the right list; click on ***OK***.
  - Figure 9.46 shows the flux vectors as they appear in the *Graphics Window*.
  - Repeat the same procedure for the thermal gradient by selecting ***Thermal grad TG*** from the right list. Figure 9.47 shows the thermal gradient vectors as they appear in the *Graphics Window*.

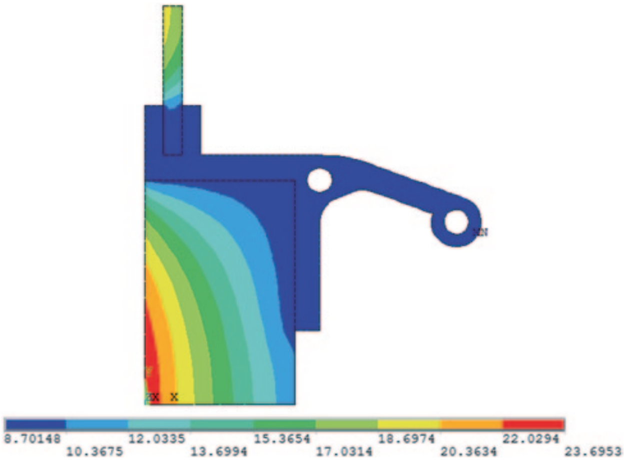
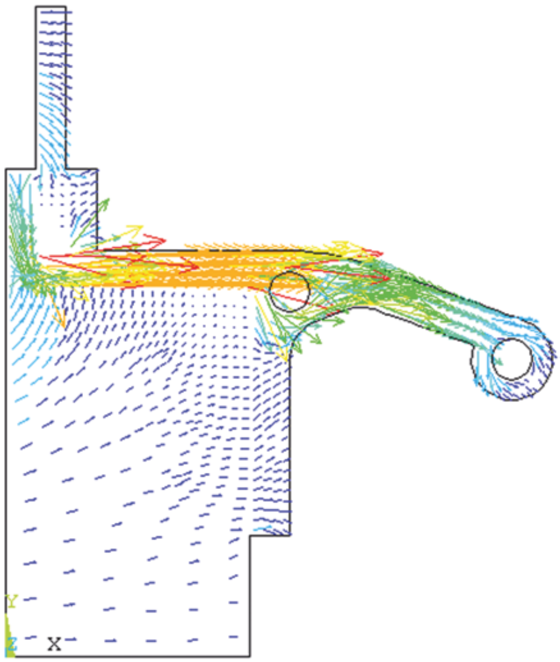


Fig. 9.45 Contour plot of temperatures

Fig. 9.46 Vector plot of thermal flux



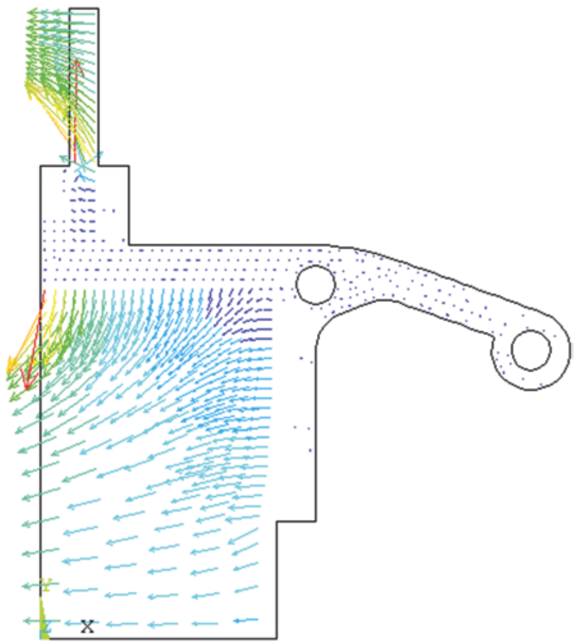


Fig. 9.47 Vector plot of thermal gradient

9.1.2 Transient Analysis

A transient analysis is utilized to simulate the heat transfer phenomenon in the presence of time-dependent boundary conditions, body loads, and/or initial conditions. In a transient analysis, in addition to the initial and boundary conditions and the body loads, the user must specify time-related quantities, such as time step size, number of load steps, number of substeps, and the final time. Depending on the values of these quantities, solutions to the same problem may differ considerably. As a rule of thumb, the solution is expected to be more accurate as the time step size decreases (increased number of substeps). However, this may increase the cost of analysis significantly. Therefore, the user has to build a certain level of knowledge through numerical experimentation on the time-dependent parameters.

9.1.2.1 Transient Thermomechanical Analysis of an Electronic Package

A common cause of failure in electronic devices is the thermal stresses at elevated temperatures caused by a coefficient of thermal expansion mismatch. In most thermomechanical analyses, the thermal load is assumed to be uniform. However, a nonuniform temperature distribution may be required for specific cases. In order to obtain a transient thermal stress field, first, a transient heat conduction (diffusion) solution is obtained, followed by a stress analysis utilizing the nonuniform thermal field at a specific time as body loading. ANSYS provides a convenient method for achieving this task. In order to demonstrate this capability, a problem involving

Table 9.4 Properties of the constituent materials in the electronic package

	Substrate	Die-Attach	Silicon	Copper
Material reference no.	1	2	3	4
E (GPa)	22	7.4	163	129
ν	0.39	0.4	0.278	0.344
α (10^{-6}/°C)	18	52	2.6	14.3
κ (W/m·°C)	2	100	150	396
c (J/kg·°C)	840	535	703	384
ρ (kg/m^3)	220	6450	2330	8940

Table 9.5 Coordinates defining the areas and the corresponding material reference numbers

Area No.	X1	X2	Y1	Y2	Mat. Ref. No.
	(mm)				
1	0	5	0	2	1
2	5	7.5	0	2	1
3	7.5	10	0	2	1
4	0	5	2	2.1	2
5	0	5	2.1	3.1	3
6	0	5	3.1	5.1	4

an electronic device is considered. The device contains a silicon die (chip), epoxy die-attach, substrate, and a copper heat spreader, as shown in Fig. 9.48. Thermal and mechanical properties of the constituent materials are given in Table 9.4. The solid model is generated utilizing rectangles with the coordinates given in Table 9.5, along with their associations with material reference numbers.

In this transient thermal analysis, the surrounding air is at a temperature of $T_\infty = 25\,°\mathrm{C}$ (ambient temperature). All of the surfaces, except the symmetry line, are subjected to convective heat loss with a heat transfer coefficient of $h = 5$ W/(m^2·°C). There is no heat transfer through the symmetry line (insulation). In ANSYSthermal analyses, when boundary conditions are not specified along a boundary, insulation is imposed automatically. Heat is generated at the bottom face of the die, which is expressed in terms of a constant heat flux of $q = 1000$ W/m. The initial temperature of the device is assumed to be uniform at $T_0 = 25\,°\mathrm{C}$. Heat transfer is simulated for a period of 5 min (300 s), after which the device reaches a steady state.

Once the time-dependent temperature field is obtained, the stress field is calculated based on the temperatures at desired time points.

Transient Thermal Analysis

Model Generation

- Specify the jobname (**`/FILNAM`** command) using the following menu path:

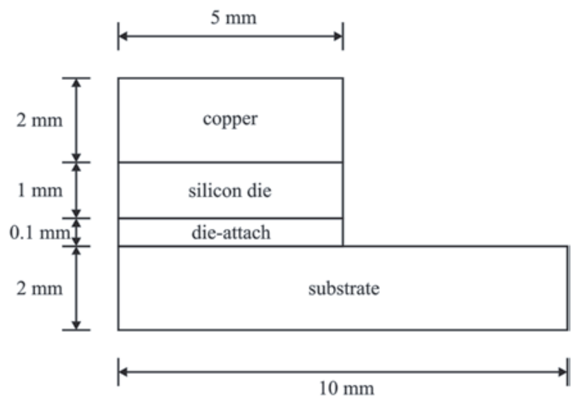


Fig. 9.48 Schematic of the electronic package

Utility Menu > File > Change Jobname

– *Change Jobname* dialog box appears. Type ***TH***; click on ***OK***.

- Define the element type (**`ET`** command) using the following menu path:

Main Menu > Preprocessor > Element Type > Add/Edit/Delete

– *Element Types* dialog box appears; click on ***Add***.
– Select ***Solid*** immediately below ***Thermal*** in the left list and ***Quad 4 Node 55*** in the right list; click on ***OK***.
– Click on ***Close***.

- Specify material properties (**`MP`** command) using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

– *Define Material Model Behavior* dialog box appears. Structural properties are first specified, followed by thermal properties.
– In the right window, successively left-click on ***Structural***, ***Linear***, ***Elastic***, and, finally, ***Isotropic***, which brings up another dialog box
– Referring to Table 9.4, enter ***22E9*** for ***EX*** and ***0.39*** for ***PRXY***; click on ***OK***
– In the right list, successively double-click on ***Structural***, ***Thermal Expansion***, ***Secant Coefficient***, and, finally, ***Isotropic***, which brings up another dialog box.
– Enter ***18E–6*** for ***ALPX***; click on ***OK***.
– Specify density by successively double-clicking on ***Structural*** and ***Density***, which brings up another dialog box. Type ***220***, click on ***OK***.
– In order to specify thermal properties, first successively double-click on ***Thermal***, ***Conductivity***, and ***Isotropic***; then enter ***2*** in the newly appeared dialog box; click on ***OK***. Thermal conductivity is specified.
– Specify specific heat by successively double-clicking on ***Thermal*** and ***Specific Heat***; enter ***840*** in the newly appeared dialog box; click on ***OK***.
– Add new material model using the following menu path:

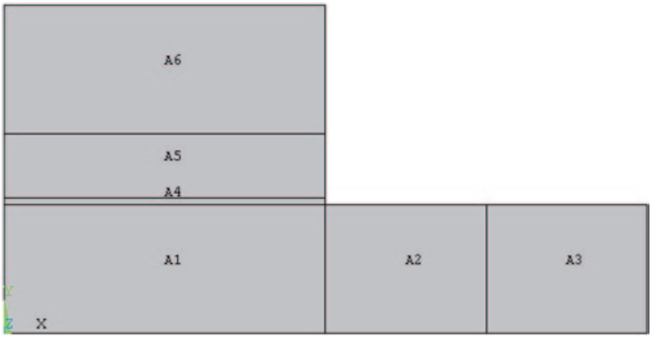


Fig. 9.49 Solid model of the electronic device with area numbers turned on

Material > New Model

- Repeat the procedure for the remaining materials (2 through 4).
- When finished, close the *Define Material Model Behavior* dialog box by using the following menu path:

Material > Exit

- Create rectangles as identified in Fig. 9.49 (**`RECTNG`** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By Dimensions

- *Create Rectangle by Dimensions* dialog box appears. Referring to Table 9.5, enter ***0*** and ***5E–3*** for ***X1*** and ***X2*** and ***0*** and ***2E–3*** for ***Y1*** and ***Y2***; click on ***Apply***.
- Repeat the procedure for the remaining areas (2 through 6). When creating Area 6, click on ***OK*** after entering the coordinates.

- Glue the areas (**`AGLUE`** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Operate > Booleans > Glue > Areas

- *Pick Menu* appears; click on ***Pick All*** button.

- Mesh the areas (**`AMESH`** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Areas > Mapped > 3 or 4 sided

- *Pick Menu* appears; click on ***Pick All***.
- At this point, all the elements have *Material Reference Number* 1. Attributes can be changed after the elements are created. For this purpose, the areas are selected and then the elements that are attached to the selected areas are selected. Finally, elements are modified to have the correct attributes. The

correspondence between the areas and material numbers is given in Table 9.5. Select areas (**ASEL**command) using the following menu path:

Utility Menu > Select > Entities

- *Select Entities* dialog box appears; select ***Areas*** from the first pull-down menu; click on ***OK***.
- *Pick Menu* appears; pick area 4 as defined in Table 9.5; click on ***OK***.
- Now, select the elements that are attached to the selected areas (**ESLA** command) using the following menu path:

Utility Menu > Select > Entities

- *Select Entities* dialog box appears; select ***Elements*** from the first pull-down menu; select ***Attached to*** from the second pull-down menu. Click on the ***Areas*** radio-button; click on ***OK***.
- Modify the attributes of the selected set of elements (**EMODIF** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Move/Modify > Elements > Modify Attrib

- *Pick Menu* appears; click on ***Pick All***, which brings up the *Modify Elem Attributes* dialog box.
- Select ***Material MAT*** from the pull-down menu; enter ***2*** in the ***I1 New attribute number*** field; click on ***OK***.
- Repeat this procedure for area 5 (material reference number 3) and area 6 (material reference number 4).
- When finished, select everything (**ALLSEL** command) using the following menu path:

Utility Menu > Select > Everything

- Save the database (**SAVE** command) using the following menu path:

Utility Menu > Save as Jobname.db

- The database is saved in fi.le ***TH.db***.

Solution

- Declare the new analysis to be a transient analysis (**ANTYPE** command) using the following menu path:

Main Menu > Solution > Analysis Type > New Analysis

- *New Analysis* dialog box appears; click on the ***Transient*** radio-button; click on ***OK***.
- *Transient Analysis* dialog box appears; click on ***OK***.

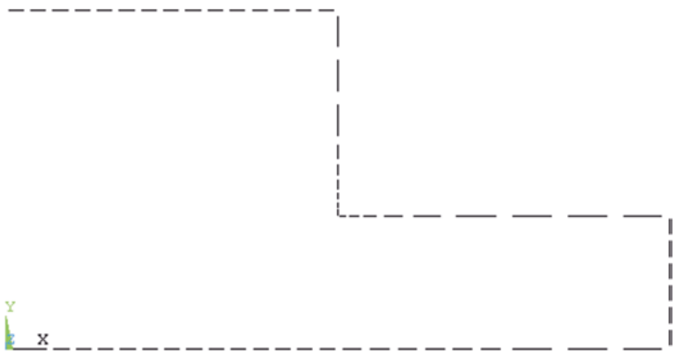


Fig. 9.50 Lines to be selected for convective boundary conditions

- Apply convective boundary conditions on lines (`SFL` command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Thermal > Convection > On Lines

  – Pick *Menu* appears; pick the exterior lines *except* the ones along $x=0$ (lines to be picked are shown in Fig. 9.50); click on ***OK*** in the *Pick Menu*.
  – Enter ***5*** for ***VALI Film coefficient*** and ***25*** for ***VAL2I Bulk temperature***; click on ***OK***.

- Apply the heat flux condition (`SFL` command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Thermal > Heat Flux > On Lines

  – *Pick Menu* appears; pick the line between the die and the die-attach; click on ***OK*** in the *Pick Menu*.
  – *Apply HFLUX on lines* dialog box appears; enter ***1000*** for ***VALI Heat flux***; click on ***OK***.

- Apply the initial condition on the nodes (`IC` command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Initial Condit'n > Define

  – *Pick Menu* appears; click on ***Pick All*** in *Pick Menu*.
  – *Define Initial Conditions* dialog box appears; select ***TEMP*** from the pull-down menu; enter ***25*** for ***VALUE Initial value of DOF***; click on ***OK***.

- Specify solution controls using the following menu path:

Main Menu > Solution > Analysis Type > Sol'n Controls

- *Solution Controls* dialog box appears, which has five tabs. In the ***Basic*** tab, within the ***Time Control*** field, enter ***300*** for ***Time at end of loadstep*** and select ***Off*** from the ***Automatic time stepping*** pull-down menu.
- Enter ***100*** for ***Number of substeps***.
- In the ***Write Items to Results File*** field, select ***Write every substep*** from the ***Frequency*** pull-down menu; click on ***OK***.

• Obtain the solution (**`SOLVE`** command) using the following menu path:

Main Menu > Solution > Solve > Current LS

- *Confirmation Window* appears along with *Status Report Window*.
- Review status; if OK, close the *Status Report Window*; click on ***OK*** in the Confirmation Window.
- Wait until ANSYS responds with ***Solution is done!***

Postprocessing

• Results are stored in the ***TH.rth*** file in the *Working Directory*. Review the temperature distribution at different time steps using the following menu path:

Main Menu > General Postproc > Read Results > By Load Step

- *Read Results by Load Step Number* dialog box appears; enter ***10*** for ***SBSTEP Substep number***; click on ***OK***.
- Plot temperature contours (**`PLNSOL`** command) at the selected substep using the following menu path:

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu

- *Contour Nodal Solution Data* dialog box appears. Select ***DOF Solution*** and ***Nodal Temperature***; click on ***OK***. The contour plot appears, as shown in Fig. 9.51.
- Repeat this *procedure* for substeps 50 and 100. Contour plot corresponding to substep 100 (time = 300 s) is shown in Fig. 9.52.

• Stress fields corresponding to substeps 10, 50, and 100 are obtained in the next subsection.

Thermomechanical Analysis

• Clear the database (**`/CLEAR`** command) using the following menu path:

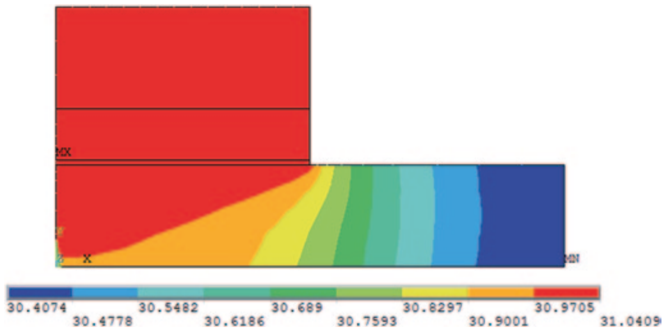


Fig. 9.51 Contour plot of temperature at substep 10 (time = 30 s)

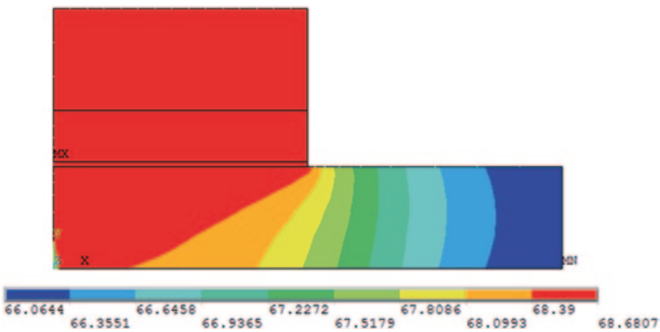


Fig. 9.52 Contour plot of temperature at substep 100 (time = 300 s)

Utility Menu > File > Clear & Start New

- *Clear Database and Start New* dialog box appears; click on ***OK***.
- *Verify* dialog box appears; click on ***Yes***.

- Resume from the ***TH.db*** database file (**RESUME** command) using the following menu path:

Utility Menu > File > Resume from

- *Resume Database* dialog box appears. Browse for ***TH.db***; click on ***OK***.

Preprocessor

- In order to perform structural analysis, define a new element type, **PLANE182**, using the following menu path:

Main Menu > Preprocessor > Element Type > Add/Edit/Delete

- *Element Types* dialog box appears; click on ***Add***.
- Select ***Solid*** immediately below ***Structural*** in left list and ***Quad 4Node 182*** in right list; click on ***OK***.
- Click on ***Options***.
- *PLANE182 element type options* dialog box appears; select the ***Plane strain*** item from the pull-down menu corresponding to ***Element behavior K3***.
- Click on ***OK***; click on ***Close***.

• Modify the element attribute corresponding to the element type from **`PLANE55`** to **`PLANE182`** for the entire model using the following menu path:

Main Menu > Preprocessor > Modeling > Move/Modify > Elements > Modify Attrib

- *Pick Menu* appears; click on the ***Pick All*** button.
- *Modify Elem Attributes* dialog box appears; pick ***Elem type ELEM*** from the ***STLOC Attribute to change*** pull-down menu and enter ***2*** in the ***I1 New attribute number*** text field; click on ***OK***.

Solution

• Declare the new analysis to be static (**`ANTYPE`** command) using the following menu path:

Main Menu > Solution > Analysis Type > New Analysis

- *New Analysis* dialog box appears; click on the ***Static*** radio-button; click on ***OK***.

• Apply displacement constraints (**`D`** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; pick the nodes along $x=0$ (y-axis); click on ***OK***.
- *Apply U, ROT on Nodes* dialog box appears; highlight ***UX***; and click on ***Apply***.
- *Pick Menu* reappears; pick the node at $(x, y)=(0, 0)$; click on ***OK***.
- *Apply U, ROT on Nodes* dialog box reappears; highlight ***UX*** and ***UY***; click on ***OK***.

• Specify the stress-free temperature (**`TREF`** command) using the following menu path:

Main Menu > Solution > Define Loads > Settings > Reference Temp

- *Reference Temperature* dialog box appears; enter ***25*** for ***Reference temperature***; click on ***OK***.

- Read nonuniform temperature field as body load (**`LDREAD`** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Temperature > From Therm Analy

  - *Apply TEMP from Thermal Analysis* dialog box appears; enter ***1*** and ***10*** for ***Load step and substep no***; click on ***Browse.***
  - *Fname Name of results file* dialog box appears; browse for and select the ***TH.rth*** file; click on ***Open***.
  - Click on ***OK*** in the *Apply TEMP from Thermal Analysis* dialog box.

- In real applications in the electronics industry, the copper heat spreader and the die are not in perfect contact; a thermal grease compound is dispensed in between these two components. Therefore, the copper heat spreader will be excluded from the analysis here in order to represent the real situation better. This will be achieved by selecting the nodes associated the substrate, die-attach, and die only (excluding those associated with the heat spreader). Select elements with material numbers 1, 2, and 3 (**`ESEL`** command) using the following menu path:

Utility Menu > Select > Entities

  - *Select Entities* dialog box appears; select ***Elements*** from the first pull-down menu and ***By Attributes*** from the second pull-down menu. Click on the ***Material num*** radio-button and enter ***1,3,1*** in the ***Min, Max,Inc*** text field. Click on ***Apply***.
  - In the *Select Entities* dialog box, select ***Nodes*** from the pull-down menu and ***Attached to*** from the second pull-down menu. Click on the ***Elements*** radio-button; click on ***OK***.
  - Now, only the nodes associated with the substrate, die-attach, and die are selected (active) and only they will be included in the solution process.

- Obtain the solution (**`SOLVE`** command) using the following menu path:

Main Menu > Solution > Solve > Current LS

  - *Confirmation Window* appears along with *Status Report Window*.
  - Review status; if OK, close the *Status Report Window*; click on ***OK*** in the Confirmation Window.
  - Wait until ANSYS responds with ***Solution is done!***

Postprocessing

- Review the deformed shape resulting from the nonuniform temperature loading (**`PLDISP`** command) using the following menu path:

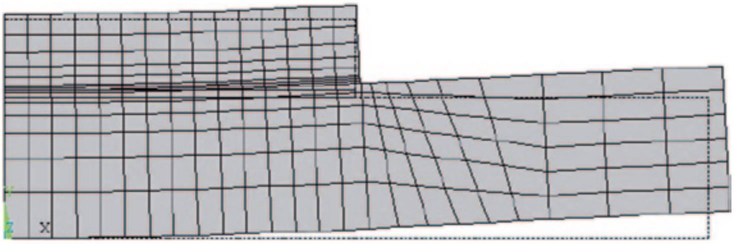


Fig. 9.53 Deformed shape of the electronic package due to thermal loading at load step 1, substep 10

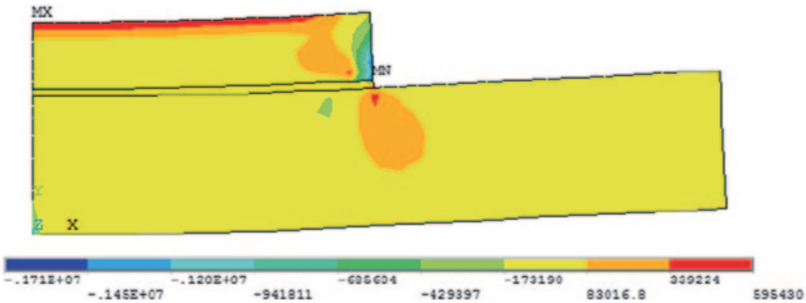


Fig. 9.54 Contour plot of the peeling stress due to thermal loading at load step 1, substep 10

Main Menu > General Postproc > Plot Results > Deformed Shape

- *Plot Deformed Shape* dialog box appears; click on the ***Def+undef edge*** radio-button; click on ***OK***.
- The deformed shape appears, as shown in Fig. 9.53.

- Review normal and shear stress contours (**`PLNSOL`** command) using the following menu path:

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu

- *Contour Nodal Solution Data* dialog box appears; select ***Stress*** and ***Y- Component of stress***; click on ***OK***.
- The contour plot of normal stress in the *y*-direction (peeling stress) appears, as shown in Fig. 9.54.
- Repeating this procedure for shear stresses, the contour plot appears, as shown in Fig. 9.55.
- The thermomechanical solution may be repeated for any substep of interest. Figs. 9.56 and 9.57 show the contour plots of normal stress in the y-direction and shear stress for substep 100 (last substep, time = 300 s), respectively.

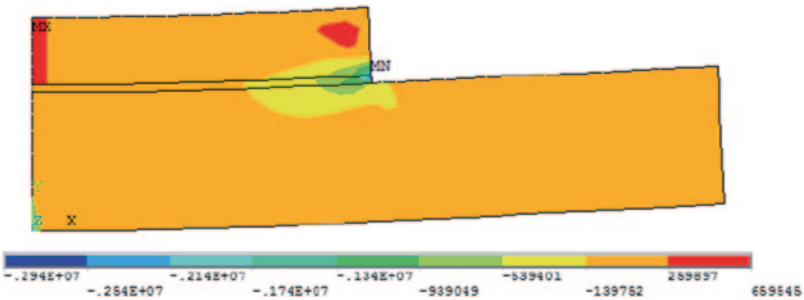


Fig. 9.55 Contour plot of the shearing stress due to thermal loading at load step 1, substep 10

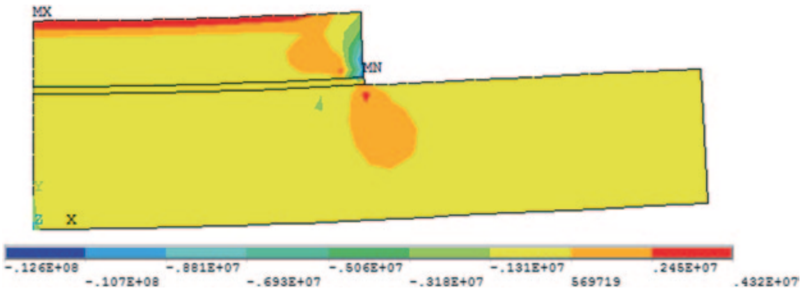


Fig. 9.56 Contour plot of the peeling stress due to thermal loading at load step 1, substep 100

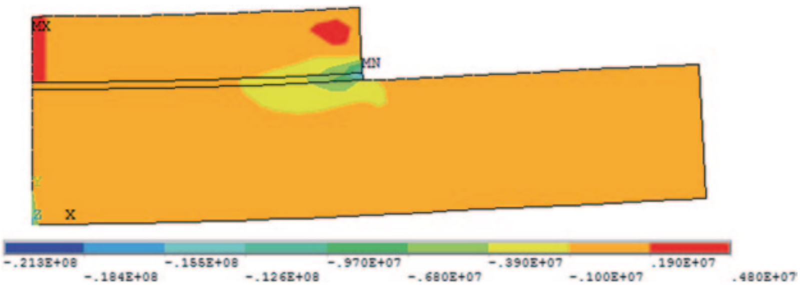


Fig. 9.57 Contour plot of the shearing stress due to thermal loading at load step 1, substep 100

9.1.2.2 Transient Thermomechanical Analysis of a Welded Joint

Consider the welded steel joint shown in Fig. 9.58 (due to symmetry with respect to the plane of the weld pool, only half of the geometry is shown). The plate is 12.5 cm long, 2 cm high, and 10 cm wide. The weld pool is assumed to be 0.5 cm long. In this heat transfer analysis, the surrounding air is at a temperature of $T_\circ = 229.82\,°\text{K}$ (ambient temperature). The temperature-dependent film coefficients for the top, bottom, and side surfaces are given in Table 9.6, as well as the temperature-dependent thermal conductivity The density and spe-

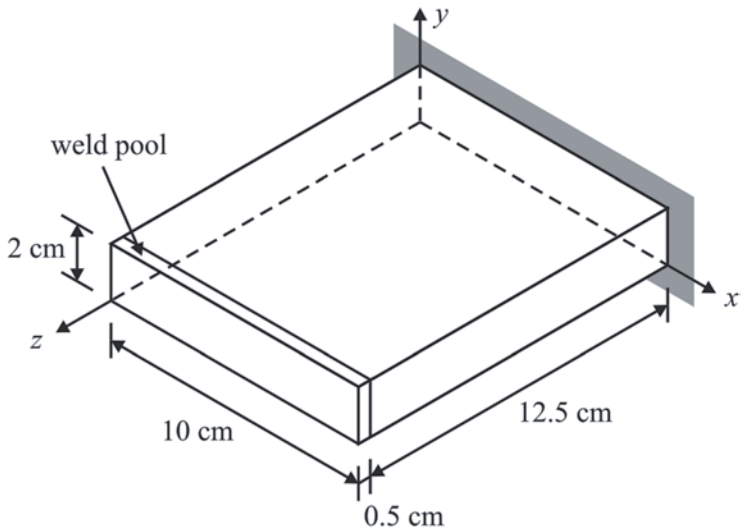


Fig. 9.58 Schematic of the welded joint

cific heat are assumed to be constant at $\rho = 7850\ \text{kg/m}^3$ and $c = 500\ \text{J/(kg}\cdot{}^\circ\text{K)}$, respectively. The temperature is fixed at 299.82 °K along the surface coinciding with the *x*-*y* plane, and there is no heat transfer along the opposite surface (symmetry plane). The initial temperature for the weld pool is 1852.94 °K, and for the remaining volume, 299.82 °K. In this thermomechanical analysis, the plate is fixed in all directions along the side coinciding with the *x*-*y* plane, and the symmetry plane is constrained in the *z*-direction only. The elastic modulus, Poisson's ratio, and coefficient of thermal expansion are $E = 200$ GPa, $\nu = 0.3$, and $\alpha = 60\times 10^{-6}\ \text{ppm/}^\text{o}\text{K}$, respectively. The goal is to obtain a time-dependent thermal solution followed by a thermomechanical solution, which provides displacement and stress fields at different times.

Transient Thermal Analysis

Model Generation

- Specify the jobname (**`/FILNAM`** command) using the following menu path:

Utility Menu > File > Change Jobname

  – *Change Jobname* dialog box appears; type ***WELD***; click on ***OK***.

- Define the element type (**`ET`** command) using the following menu path:

Main Menu > Preprocessor > Element Type > Add/Edit/Delete

  – *Element Types* dialog box appears; click on ***Add***.

Table 9.6 Temperature-dependent properties

Temperature (°K)	Thermal conductivity [W/(m· °K)]	Film coefficient [W/(m^2 · °K)]		
		Top surface	Bottom surface	Side surfaces
433.15	30.158	9.08	4.99	8.18
593.15	30.385	11.69	6.05	9.89
753.15	30.611	13.57	6.79	10.96
913.15	30.838	14.96	7.28	11.86
1073.15	31.064	16.19	7.69	12.51
1233.15	31.290	17.25	8.09	13.16
1393.15	31.517	18.15	8.42	13.66
1553.15	31.743	18.97	8.67	14.15
1713.15	31.969	19.79	8.99	14.55
1873.15	25.338	20.52	9.24	14.96

- Select ***Solid*** immediately below ***Thermal*** in the left list and ***Brick 8node 70*** in the right list; click on ***OK***.
- Click on ***Close***.

• Specify material properties (**`MP`** command) using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

- *Define Material Model Behavior* dialog box appears. Structural properties are specified first, followed by thermal properties.
- In the right window, successively left-click on ***Structural***, ***Linear***, ***Elastic***, and, finally, ***Isotropic***, which brings up another dialog box.
- Enter ***200E9*** for ***EX*** and ***0.3*** for ***PRXY***; click on ***OK***
- In the right list, successively left-click on ***Structural***, ***Thermal Expansion***, ***Secant Coefficient***, and, finally, ***Isotropic***, which brings up another dialog box.
- Enter ***3.6E–5*** for ***ALPX***; click on ***OK***.
- Specify density by successively clicking on ***Structural*** and ***Density***, which brings up another dialog box. Type ***7850***; click on ***OK***.
- Specify specific heat by successively double-clicking on ***Thermal*** and ***Specific Heat***; enter ***500*** in the newly appeared dialog box; click on ***OK***.
- Specify temperature-dependent thermal conductivity by successively double-clicking on ***Thermal***, ***Conductivity***, and ***Isotropic***.
- *Conductivity for Material Number 1* dialog box appears; click on the ***Add Temperature*** button 9 times (so that there are 10 temperature columns). Referring to Table 9.6, enter temperature values and corresponding thermal conductivity values. When finished, click on ***OK***.

Table 9.7 Coordinates defining the volumes.

Volume number	XI	X2	Y1	Y2	Z1	Z2
	(cm)					
1	0	10	0	2	0	3
2	0	10	0	2	3	8
3	0	10	0	2	8	12
4	0	10	0	2	12	12.5

– Temperature-dependent film coefficient values for top, bottom, and side surfaces are input as having different material reference numbers. Add new material model using the following menu path:

Material > New Model

– In the right list, successively left-click on ***Thermal*** and ***Convection or Film Coef.***, which brings up the *Convection of Film Coefficient for Material 2* dialog box. Click on the ***Add Temperature*** button 9 times (so that there are 10 temperature columns). Referring to Table 9.6, enter temperature values and corresponding film coefficient values at the top surface. When finished, click on ***OK***.
– Repeat the procedure for film coefficient values at the bottom (material 3) and side (material 4) surfaces.
– When finished, close the *Define Material Model Behavior* dialog box by using the following menu path:

Material > Exit

• Create rectangular blocks (**`BLOCK`** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Volumes > Block > By Dimensions

– *Create Block by Dimensions* dialog box appears. Referring to Table 9.7, enter ***0*** and ***10E–2*** for ***X1*** and ***X2***, ***0*** and ***2E–2*** for ***Y1*** and ***Y2***, and ***0*** and ***3E–2*** for ***Z1*** and ***Z2***; click on ***Apply***.
– Repeat the procedure for the remaining volumes (2 through 4). When creating volume 4, click on ***OK*** after entering the coordinates (instead of ***Apply***).

• Glue the volumes (**`VGLUE`** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Operate > Booleans > Glue > Volumes

– *Pick Menu* appears; click on ***Pick All*** button.

• Mesh the volumes (**`VMESH`** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Volumes > Mapped > 4 to 6 sided

- *Pick Menu* appears; click on ***Pick All***.

- Save the database (**SAVE** command) using the following menu path:

Utility Menu > Save as Jobname.db

- The database is saved in file ***WELD.db***.

Solution

- Declare the new analysis to be a transient analysis (**ANTYPE** command) using the following menu path:

Main Menu > Solution > Analysis TypeNew Analysis

- *New Analysis* dialog box appears; click on the ***Transient*** radio-button; click on ***OK***.
- *Transient Analysis* dialog box appears; click on ***OK***.

- Specify solution controls using the following menu path:

Main Menu > Solution > Analysis Type > Sol'n Controls

- *Solution Controls* dialog box appears, which has five tabs. In the ***Basic*** tab, within the ***Time Control*** field, enter ***3600*** for ***Time at end of loadstep***.
- Click on the ***Time increment*** radio-button and enter ***36***, ***3.6***, and ***500*** for ***Time step size***, ***Minimum time step***, and ***Maximum time step***, respectively.
- In the ***Write Items to Results File*** field, select ***Write every substep*** from the ***Frequency*** pull-down menu; click on ***OK***.

- Initial conditions are specified next. For this purpose, volumes are selected first, followed by selection of the nodes attached to the selected volumes.

  - Select volumes (**VSEL** command) using the following menu path:

Utility Menu > Select > Select Entities

- *Select Entities* dialog box appears; choose ***Volumes*** from the first pull-down menu; click on ***OK***.
- *Pick Menu* appears; pick the volumes indicated in Fig. 9.59; click on ***OK*** in the *Pick Menu*.
- Select nodes attached to the selected volumes (**NSLV** command) using the following menu path:

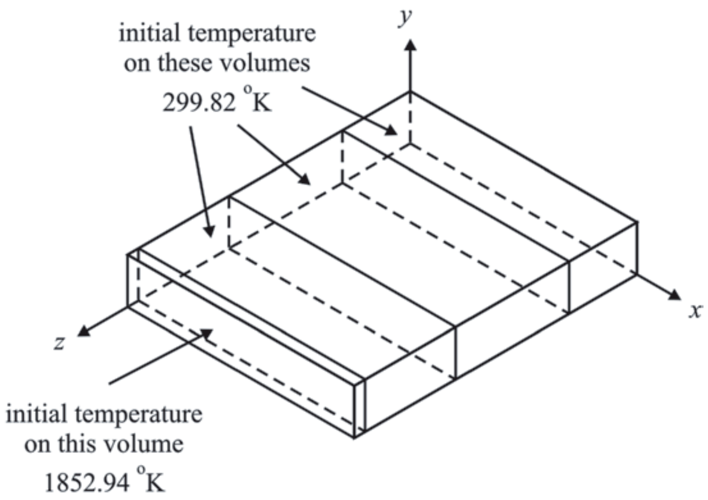


Fig. 9.59 Volumes to be selected for initial temperatures

Utility Menu > Select > Select Entities

- *Select Entities* dialog box appears; choose ***Nodes*** from the first pull-down menu and ***Attached to*** from the second pull-down menu. Click on the ***Volumes, all*** radio-button; click on ***OK***.
- Apply initial conditions on the selected nodes (`IC` command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Initial Condit'n > Define

- *Pick Menu* appears; click on ***Pick All***.
- *Define Initial Conditions* dialog box appears; select ***TEMP*** from the pull-down menu and enter ***299.8167*** for ***VALUE Initial value of DOF***; click on ***OK***.
- Select everything (`ALLSEL` command) using the following menu path:

Utility Menu > Select > Everything

- Repeat the same procedure to apply initial conditions on nodes attached to the remaining volume corresponding to the welded region. This time, apply a temperature of ***1852.594***°K.
- Select everything (`ALLSEL` command) using the following menu path:

Utility Menu > Select > Everything

• Apply temperature boundary conditions on the selected nodes (`D` command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Thermal > Temperature > On Nodes

- *Pick Menu* appears; pick the nodes along the $z = 0$ surface and click on ***OK*** (in order to pick the nodes conveniently, plot the nodes and obtain a top view).
- *Apply TEMP on Nodes* dialog box appears; highlight ***TEMP*** and type ***299.8167*** for ***VALUE Load Temp value***; click on ***OK***.

• Convective boundary conditions are applied next. Temperature-dependent film coefficients were specified previously for the top (material 2), bottom (material 3), and side (material 4) surfaces.

- Apply convective boundary conditions on the nodes along the side surfaces (`SF` command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Thermal > Convection > On Nodes

- *Pick Menu* appears; pick the nodes along the $x = 0$ and $x = 10 \times 10^{-2}$ surfaces (side surfaces); click on ***OK*** in the *Pick Menu*. For this case, ***Front View*** is the most convenient viewpoint.
- *Apply CONV on nodes* dialog box appears; enter ***−4*** for film coefficient (first text field) and ***299.8167*** for bulk temperature (second text field); click on ***Apply***.
- *Pick Menu* reappears; pick the nodes along the $y = 2 \times 10^{-2}$ surface (top surface); click on ***OK*** in the *Pick Menu*. ***Front View*** is the most convenient viewpoint for this case, also.
- *Apply CONV on nodes* dialog box appears; enter ***−2*** for film coefficient and ***299.8167*** for bulk temperature; click on ***Apply***.
- *Pick Menu* reappears; pick the nodes along the $y = 0$ surface (bottom surface); click on ***OK*** in the *Pick Menu*. ***Right View*** and ***Left View*** are the most convenient viewpoints for this case.
- *Apply CONV on nodes* dialog box appears; enter ***−3*** for film coefficient and ***299.8167*** for bulk temperature; click on ***OK***.

• Obtain the solution (`SOLVE` command) using the following menu path:

Main Menu > Solution > Solve > Current LS

- *Confirmation Window* appears along with *Status Report Window*.
- Review status; if OK, close the *Status Report Window* and click on ***OK*** in the *Confirmation Window*.
- Wait until ANSYS responds with ***Solution is done!***

Postprocessing

- Results are stored in the ***WELD.rth*** file in the *Working Directory*. Review the temperature distribution at different time points using the following menu path:

Main Menu > General Postproc > Read Results > By Pick

  - *Results File: WELD.rth* dialog box appears, in which available solution sets are tabulated. Highlight set ***5***; click on ***Read*** and click on ***Close***.
  - Plot temperature contours (**PLNSOL** command) at the selected substep using the following menu path:

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu

  - *Contour Nodal Solution Data* dialog box appears. Select ***DOF Solution*** and ***Nodal Temperature***; click on ***OK***. The contour plot appears, as shown in Fig. 9.60.
  - Repeat this procedure for substep 22 (last substep). Corresponding contour plot is shown in Fig. 9.61.

- Stress fields corresponding to substeps 10, 50, and 100 are obtained in the next subsection.

Thermomechanical Analysis

- Clear the database (**/CLEAR** command) using the following menu path:

Utility Menu > File > Clear & Start New

  - *Clear Database and Start New* dialog box appears; click on ***OK***.
  - *Verify* dialog box appears; click on ***Yes***.

- Resume from the ***WELD.db*** database file (**RESUME** command) using the following menu path:

Utility Menu > File > Resume from

  - *Resume Database* dialog box appears. Browse for ***WELD.db***; click on ***OK***.

Preprocessor

- In order to perform structural analysis, define a new element type, **SOLID185**, using the following menu path:

Main Menu > Preprocessor > Element Type > Add/Edit/Delete

  - *Element Types* dialog box appears; click on ***Add***.

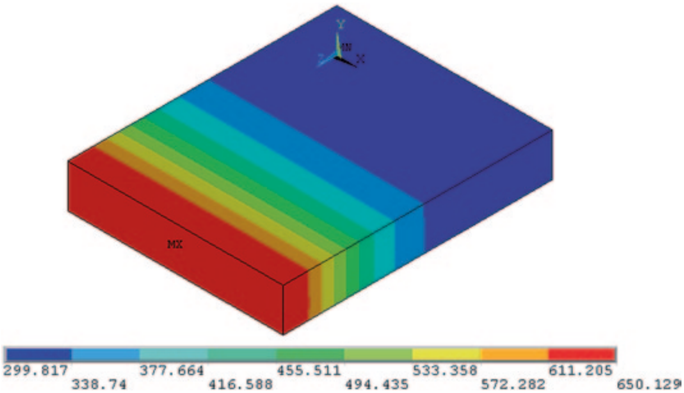


Fig. 9.60 Contour plot of temperature at substep 5 (time = 69.409 s)

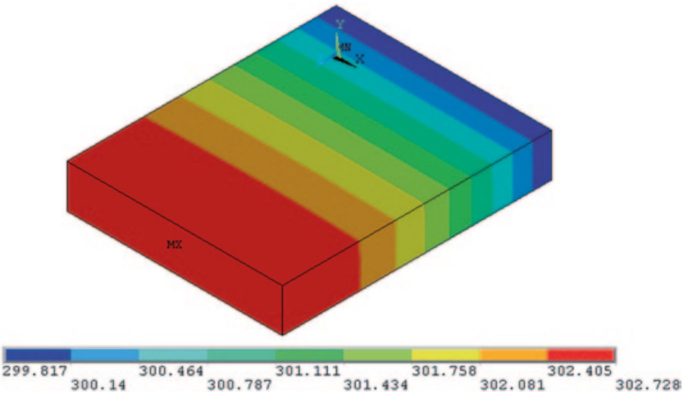


Fig. 9.61 Contour plot of temperature at substep 22 (time = 3600 s)

- Select ***Solid*** immediately below ***Structural*** in left list and ***Brick 8Node 185*** in right list; click on ***OK***.
- Click on ***Close***.

• Modify the element attribute corresponding to element type from **SOLID70** to **SOLID185** for the entire model using the following menu path:

Main Menu > Preprocessor > Modeling > Move/Modify > Elements > Modify Attrib

- *Pick Menu* appears; click on the ***Pick All*** button.
- *Modify Elem Attributes* dialog box appears. Pick ***Elem type ELEM*** from the ***STLOC Attribute to change*** pull-down menu and enter ***2*** in the ***I1 New attribute number*** text field; click on ***OK***.

Solution

- Declare the new analysis to be static (`ANTYPE` command) using the following menu path:

Main Menu > Solution > Analysis Type > New Analysis

 – *New Analysis* dialog box appears; click on the ***Static*** radio-button; click on ***OK***.

- Apply displacement constraints (`D` command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

 – *Pick Menu* appears; pick the nodes along $z=0$; click on ***OK***.
 – *Apply U, ROT on Nodes* dialog box appears; highlight ***All DOF***; click on ***Apply***.
 – *Pick Menu* reappears; pick the nodes along $z=12.5\times10^{-2}$; click on ***OK***.
 – *Apply U, ROT on Nodes* dialog box reappears; highlight ***UZ*** and remove the highlight from ***All DOF***; click on ***OK***.

- Specify the stress-free temperature (`TREF` command) using the following menu path:

Main Menu > Solution > Define Loads > Settings > Reference Temp

 – *Reference Temperature* dialog box appears; enter ***299.8167*** for ***Reference temperature***; click on ***OK***.

- Read the nonuniform temperature field as body load (`LDREAD` command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Temperature > From Therm Analy

 – *Apply TEMP from Thermal Analysis* dialog box appears; enter ***1*** and ***5*** for ***Load step and substep no.***; click on ***Browse***.
 – *Fname Name of results file* dialog box appears; search for and select ***WELD.rth*** file; click on ***OK***.
 – Click on ***OK*** in the *Apply TEMP from Thermal Analysis* dialog box.

- Obtain the solution (`SOLVE` command) using the following menu path:

Main Menu > Solution > Solve > Current LS

 – *Confirmation Window* appears along with *Status Report Window*
 – Review status; if OK, close the *Status Report Window*; click on ***OK*** in the *Confirmation Window*.
 – Wait until ANSYS responds with ***Solution is done!***

Postprocessing

- Review contours for normal stresses in the z-direction (**`PLNSOL`** command) using the following menu path:

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu

  – *Contour Nodal Solution Data* dialog box appears; select ***Stress*** and ***Z-Component of stress***; click on ***OK***.
  – The contour plot of normal stresses in the z-direction appears, as shown in Fig. 9.62.
  – The thermomechanical solution may be repeated for any substep of interest. Figure 9.63 shows the contour plots of normal stress in the z-direction for substep 22 (last substep).

9.1.3 Radiation Analysis

Radiation is the transfer of thermal energy between two surfaces through electromagnetic waves. No medium is required for radiation heat transfer to take place. The transfer of thermal energy between two surfaces depends on the difference between the fourth powers of absolute temperatures along the surfaces. When radiation conditions are present, the problem is certain to be nonlinear. A radiation analysis requires specific knowledge of concepts that are not covered in this book. Therefore, it is highly recommended that the user have a good understanding of the radiation phenomenon before using ANSYS for radiation analyses.

Radiation analyses in ANSYS require the Stefan-Boltzmann constant, as well as the emissivity, view factor, and space temperature values for the surfaces involved. A radiation analysis is demonstrated by solving a simple problem using the *Radiosity Solver* method.

Two partial hollow circular regions are radiating to each other, as shown in Fig. 9.64. The emissivity of the outer surface of the inner region and inner surface of the outer region are 0.9 and 0.7, respectively, while the inner surface of the inner region and outer surface of the outer region are maintained at temperatures of 1500 °F and 100 °F, respectively. The space temperature is 70 °F. Both regions have a thermal conductivity of 0.1. The goal is to obtain the steady-state temperature and heat flux variations.

Model Generation

- Define the element type (**`ET`** command) using the following menu path:

Main Menu > Preprocessor > Element Type > Add/Edit/Delete

  – *Element Types* dialog box appears; click on ***Add***.
  – Select ***Solid*** immediately below ***Thermal*** in the left list and ***Quad 4 Node 55*** in the right list; click on ***OK***.

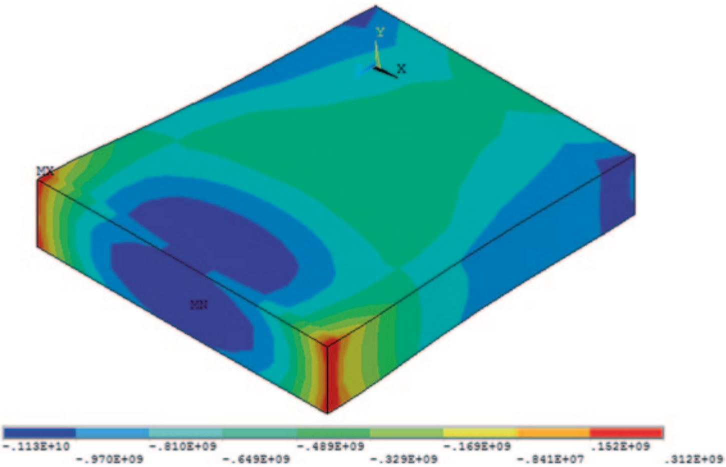


Fig. 9.62 Contour plot of the stresses in the z-direction due to thermal loading at load step 1, substep 5

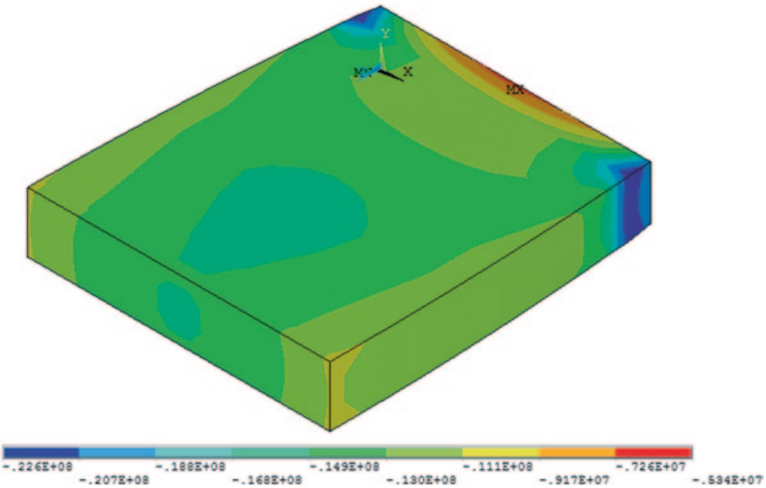


Fig. 9.63 Contour plot of the stresses in the z-direction due to thermal loading at load step 1, substep 22

– Exit from the *Element Types* dialog box by clicking on ***Close***.

• Specify material properties (`MP` command) using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

– *Define Material Model Behavior* dialog box appears. In the right window, successively left-click on ***Thermal***, ***Conductivity***, and ***Isotropic***, which brings up another dialog box.

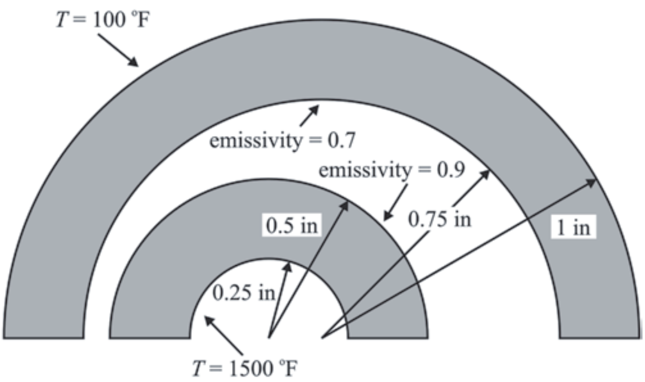


Fig. 9.64 Schematic of the conical fin cross section

- Enter ***0.1*** for ***KXX***; click on ***OK***.
- Close the *Define Material Model Behavior* dialog box by using the following menu path:

Material > Exit

- Create partial hollow circular areas (**`PCIRC`** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Circle > By Dimensions

- *Circular Area by Dimensions* dialog box appears. Referring to Fig. 9.64, enter ***0.5*** and ***0.25*** for ***RAD1*** and ***RAD2***, and ***0*** and ***180*** for ***THETA1*** and ***THETA2***; click on ***OK***.
- Move the center of the *Working Plane* by 0.2 in the *x*-direction using the following menu path:

Utility Menu > WorkPlane > Offset WP by Increments

- *Offset WP* menu appears; click on the ***+X*** button four times and observe the *Working Plane* triad move in the *Graphics Window*. Also observe, toward the bottom of the *Offset WP* menu, that ***0.2*** appears next to ***Global X=***.
- Exit from the *Offset WP* menu by clicking on ***OK***.
- Create the remaining partial hollow circular area (**`PCIRC`** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Circle > By Dimensions

- *Circular Area by Dimensions* dialog box appears. Referring to Fig. 9.64, enter ***1*** and ***0.75*** for ***RAD1*** and ***RAD2*** and ***0*** and ***180*** for ***THETA1*** and ***THETA2***; click on ***OK***.
- The areas appear in the *Graphics Window*, as shown in Fig. 9.65.

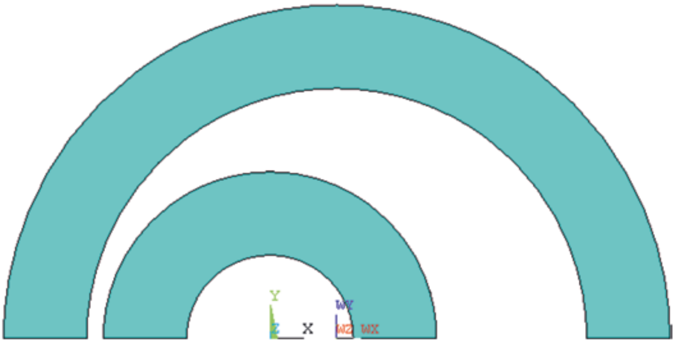


Fig. 9.65 Area plot for the conical fin

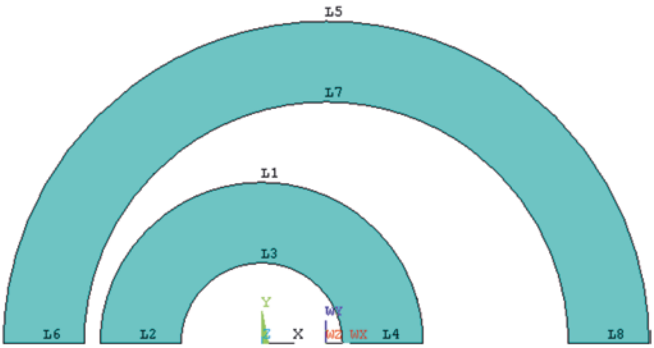


Fig. 9.66 Area plot with line numbers turned on

- Turn line numbering on using the following menu path:

Utility Menu > PlotCtrls > Numbering

- – *Plot Numbering Controls* dialog box appears; click on the square box next to ***LINE Line numbers*** so that a checkmark appears. Click on ***OK***. Line numbers appear, as shown in Fig. 9.66.

- Specify the number of elements along selected lines (**`LESIZE`** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > Picked Lines

- – *Pick Menu* appears; pick lines 1, 3, 5, and 7; click on ***OK***.
- – *Element Sizes on Picked Lines* dialog box appears; enter ***40*** in the text field for ***NDIV*** and remove the checkmark next to ***KYNDIV***. Click on ***Apply***.
- – *Pick Menu* reappears; pick lines 2, 4, 6, and 8; click on ***OK***.
- – In the *Element Sizes on Picked Lines* dialog box, enter ***10*** in the text field for ***NDIV***; click on ***OK***.

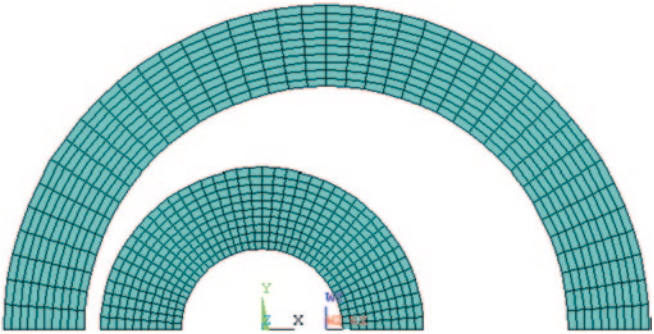


Fig. 9.67 Mesh representing the conical fin

- Mesh the areas (**`AMESH`** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Areas > Mapped > 3 or 4 sided

- – *Pick Menu* appears; click on ***Pick All***.
- – The mesh appears in the *Graphics Window*, as shown in Fig. 9.67.

Solution

- Apply radiation conditions on lines (**`SFL`** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Thermal > Radiation > On Lines

- – *Pick Menu* appears; pick line 1 (refer to Fig. 9.66); click on ***OK*** in the *Pick Menu*.
- – *Apply RDSF on Lines* dialog box appears; enter ***0.9*** for ***VALUE Emissivity*** and ***1*** for ***VALUE2 Enclosure number***. Click on ***Apply***.
- – *Pick Menu* reappears; pick line 7; click on ***OK*** in the *Pick Menu*.
- – In the *Apply RDSF on Lines* dialog box, enter ***0.7*** for ***VALUE Emissivity*** and ***1*** for ***VALUE2 Enclosure number***. Click on ***OK***.

- Specify the temperatures on lines (**`DL`** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Thermal > Temperature > On Lines

- – *Pick Menu* appears; pick line 3; click on ***OK*** in the *Pick Menu*.
- – *Apply TEMP on Lines* dialog box appears; highlight ***TEMP*** from the list and enter ***1500*** for ***VALUE Load TEMP value***; click on ***Apply***.
- – *Pick Menu* reappears; pick line 5; and click on ***OK*** in the *Pick Menu*.
- – In the *Apply TEMP on Lines* dialog box enter ***100*** for ***VALUE Load TEMP value***; click on ***OK***.

- Define *Radiosity Solver* options (**STEF**, **TOFFST**, **RADOPT**, **SPCTEMP** commands) using the following menu path:

Main Menu > Solution > Radiation Opts > Solution Opt

  - *Radiation Solution Options* dialog box appears; enter ***460*** for ***[TOFFST] Temperature difference, 0.5*** for ***Radiation flux relax. factor, 0.01*** for ***Convergence tolerance***, and ***70*** for ***Value*** (underneath ***Space option***). Click on ***OK***.

- Specify the time-related parameters (**TIME**, **DELTIM** commands) using the following menu path:

Main Menu > Solution > Load Step Opts > Time/Frequenc > Time—Time Step

  - *Time and Time Step Options* dialog box appears. Enter ***1*** for ***[TIME] Time at end of load step***, ***0.5*** for ***[DELTIM] Time step size, 0.1*** for ***[DELTIM] Minimum time step size***, and ***1*** for ***Maximum time step size***. Click on ***OK***.

- Specify the maximum number of equilibrium iterations (**NEQIT** command) using the following menu path:

Main Menu > Solution > Load Step Opts > Nonlinear > Equilibrium Iter

  - *Equilibrium Iterations* dialog box appears. Enter ***1000*** for ***[NEQIT] No. of equilibrium iter***; click on ***OK***.

- Obtain the solution (**SOLVE** command) using the following menu path:

Main Menu > Solution > Solve > Current LS

  - *Confirmation Window* appears along with *Status Report Window*.
  - Review status; if OK, close the *Status Report Window*; click on ***OK*** in the *Confirmation Window*.

- Wait until ANSYS responds with ***Solution is done!***

Postprocessing

- Review the temperature distribution (**PLNSOL** command) using the following menu path:

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu

  - *Contour Nodal Solution Data* dialog box appears; select ***DOF Solution*** and ***Nodal Temperature***; click on ***OK***.
  - Temperature contours appear in the *Graphics Window*, as shown in Fig. 9.68.

- Review flux vectors (**PLVECT** command) using the following menu path:

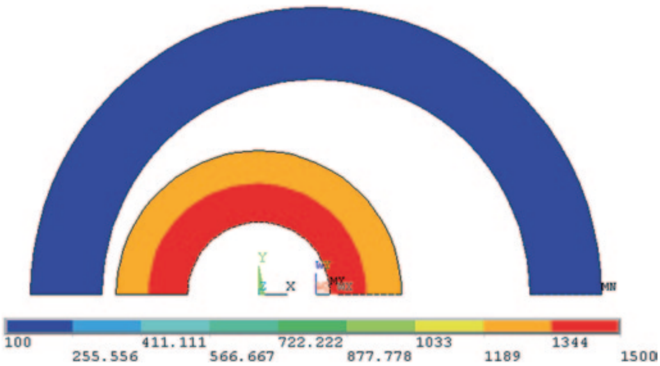


Fig. 9.68 Contour plot of temperature

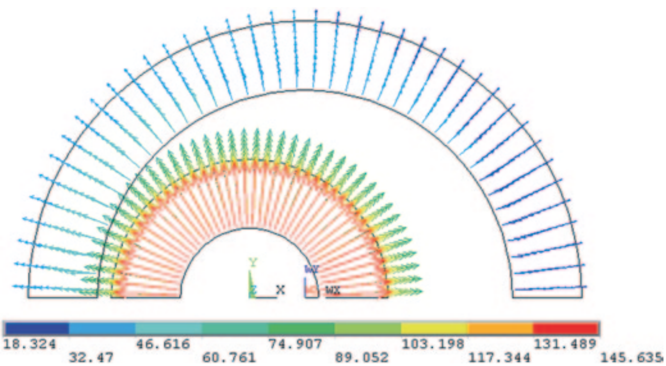


Fig. 9.69 Vector plot of heat flux

Main Menu > General Postproc > Plot Results > Vector Plot > Predefined

– *Vector Plot of Predefined Vectors* dialog box appears; select ***Flux & gradient*** from the left list and ***Thermal flux TF*** from the right list; click on ***OK***.
– Flux vectors appear in the *Graphics Window*, as shown in Fig. 9.69.

9.2 Moisture Diffusion

The moisture diffusion phenomenon is an important issue because polymeric materials are being used more extensively in engineering applications. The moisture diffusion into a water permeable medium is governed by

$$\frac{\partial C}{\partial t} = D\left(\frac{\partial^2 C}{\partial x^2} + \frac{\partial^2 C}{\partial y^2} + \frac{\partial^2 C}{\partial z^2}\right) \tag{9.3}$$

Table 9.8 Correspondence table for thermal/moisture analogy

Property	Thermal	Moisture
Primary variable	Temperature, T	Wetness, ω
Density	ρ (kg/m^3)	1
Conductivity	κ (W/m· °C)	$D{\cdot}C_{sat}$ (kg/s· m)
Specific heat	c (J/kg· °C)	C_{sat} (kg/m^3)

where C is the moisture concentration, D is the moisture diffusivity, and t designates time. This equation is analogous to the heat diffusion (transient heat transfer) equation given by

$$\frac{\partial T}{\partial t} = \alpha\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}\right) \tag{9.4}$$

where T is the temperature and α is the thermal diffusivity written in terms of thermal conductivity κ, specific heat c, and density ρ as

$$\alpha = \frac{\kappa}{\rho c} \tag{9.5}$$

Although Eq. (9.3) and (9.4) are analogous, the finite element formulation for heat diffusion cannot be directly used for solving moisture diffusion problems involving multiple dissimilar materials. This is because the moisture concentration, C, unlike the temperature, T, is not continuous along material interfaces. However, when C is normalized with respect to the saturated moisture concentration, C_{sat}, this incompatibility is removed and the finite element formulation for heat diffusion can now be used for solving moisture diffusion problems. The normalized moisture concentration is called the *wetness* parameter and is written as

$$w = \frac{C}{C_{sat}} \tag{9.6}$$

As is clear from Eq. (9.4), thermal conductivity, κ, specific heat, c, and density, ρ, need to be known to solve a heat diffusion problem. After the introduction of the wetness parameter, in order to utilize the finite element formulation for heat diffusion for solving moisture diffusion problems, the user must use the correspondence table given in Table 9.8.

Once the solution is obtained, the weight of the moisture absorbed by an element, $W^{(e)}$, is computed by the product of average moisture concentration and the volume of the element in the form

$$W^{(e)} = \left(\frac{1}{N}\sum_{i=1}^{N} \omega_i \cdot C_{sat}\right) \cdot V^{(e)} \tag{9.7}$$

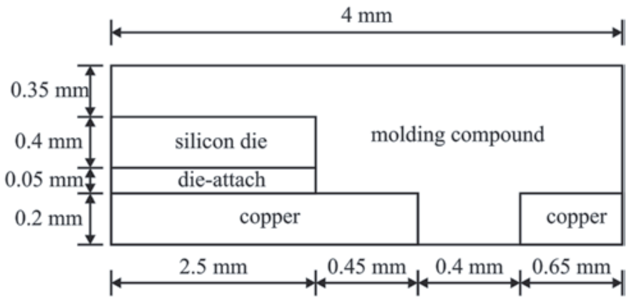


Fig. 9.70 Schematic of the electronic device undergoing moisture diffusion

Table 9.9 Moisture diffusion related material properties of the constituent materials in the electronic package

	Molding compound	Die-Attach
D (mm^2/sec) absorption	7.43×10^{-7}	1.25×10^{-5}
D (mm^2/sec) desorption	0.18	0.35
C_{sat} (mg/mm^3)	7.06×10^{-3}	6.20×10^{-3}

in which N is the number of nodes per element and $V^{(e)}$ represents the volume of the element. The wetness at the ith node is indicated by ω_i. The total weight of the absorbed moisture can then be found by simply summing the weight of the moisture absorbed by each element.

When simulating absorption conditions, the exposed surfaces of the moisture-absorbing materials are subjected to "*wet*" conditions, i.e., $T = 1$. Similarly, in desorption simulations, those surfaces are subjected to "*dry*" conditions, $T = 0$.

Once the solution is complete, the analyst can use Eq. (9.6) to obtain the moisture concentration, C, in each material.

The moisture diffusion simulation using ANSYS is demonstrated by considering a typical example from the electronics industry: an analysis of the moisture diffusion of an electronic package.

Consider an electronic package consisting of a silicon die, die-attach, copper, and molding compound, as shown in Fig. 9.70. The package is preconditioned at 85 % relative humidity and a temperature of 85 °C for one week (168 h) before being placed in a reflow oven at 225 °C for 5 min. The moisture diffusivity during absorption and desorption and the saturated moisture concentration values of the molding compound and the die-attach are tabulated in Table 9.9. After applying the substitutions required for a thermal-moisture analogy, the values given in Table 9.10 are obtained. These are the material properties to be used in the ANSYS analysis. As noted in Table 9.11, the silicon die and copper do not absorb moisture and so appropriate material properties are used for these components. The geometry of the problem possesses half-symmetry; thus only half the package is simulated, with insulation along the symmetry axis.

Table 9.10 Moisture properties converted to heat diffusion properties

a. Absorption					b. Desorption	
	Molding compound	Die-Attach	Silicon die	Copper	Molding compound	Die-Attach
Material ref. no.	1	2	3	4	5	6
κ (kg/hr·m)	1.88×10^{-5}	2.79×10^{-4}	1×10^{-11}	1×10^{-11}	4.24×10^{-3}	9.32×10^{-3}
c (kg/m^3)	7060	6200	1	1	7060	6200
ρ	1	1	1	1	1	1

Table 9.11 Coordinates defining the areas

Area number	X1	X2	Y1	Y2
	(mm)			
1	0	2.50	0	1.00
2	0	2.95	0	1.00
3	0	3.35	0	1.00
4	0	4.00	0	1.00
5	0	4.00	0	0.20
6	0	4.00	0	0.25
7	0	4.00	0	0.65

The solution has two stages. First, absorption is simulated. During absorption, all the external surfaces of the moisture-absorbing materials are subjected to $T = 1$. After the absorption simulation is complete, desorption is simulated, with boundary conditions $T = 0$ along the same surfaces.

Model Generation

- Define the element type (**`ET`** command) using the following menu path:

Main Menu > Preprocessor > Element Type > Add/Edit/Delete

  - *Element Types* dialog box appears; click on ***Add***.
  - Select ***Solid*** immediately below ***Thermal*** in the left list and ***Quad 4 Node 55*** in the right list; click on ***OK***.
  - Click on ***Close***.

- Specify material properties (**`MP`** command) using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

  - *Define Material Model Behavior* dialog box appears. Specify conductivity by successively clicking on ***Thermal***, ***Conductivity***, and ***Isotropic***; enter ***1.88E–5*** in the newly appeared dialog box; click on ***OK***.

- Specify density by successively clicking on ***Thermal*** and ***Specific Heat***, which brings up another dialog box. Type ***7060***, click on ***OK***.
- Specify specific heat by successively clicking on ***Thermal*** and ***Density***; enter ***1*** in the newly appeared dialog box; click on ***OK***.
- Add new material model using the following menu path:

Material > New Model

- Repeat the procedure for the remaining materials (2 through 6) by referring to Table 9.10.
- When finished, close the *Define Material Model Behavior* dialog box by using the following menu path:

Material > Exit

- Create rectangular areas (**`RECTNG`** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By Dimensions

- *Create Rectangle by Dimensions* dialog box appears. Referring to Table 9.11, enter ***0*** and ***2.5E–3*** for ***X1*** and ***X2*** and ***0*** and ***1E–3*** for ***Y1*** and ***Y2***; click on ***Apply***. Note that the units used in the analysis are in meters while the values given in Table 9.11 are in millimeters.
- Repeat the procedure for the remaining areas (2 through 7). When creating Area 7, click on ***OK*** after entering the coordinates.

- Overlap the areas (**`AOVLAP`** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Operate > Booleans > Overlap > Areas

Pick Menu appears; click on ***Pick All*** button.

- Compress entity numbers (**`NUMCMP`** command) using the following menu path:

Main Menu > Preprocessor > Numbering Ctrls > Compress Numbers

- *Compress Numbers* dialog box appears; select ***All*** from the pull-down menu; click on ***OK***.

- Specify the global element size (**`ESIZE`** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Global > Size

- *Global Element Sizes* dialog box appears; enter ***4E–5*** for ***SIZE Element edge length***; click on ***OK***.

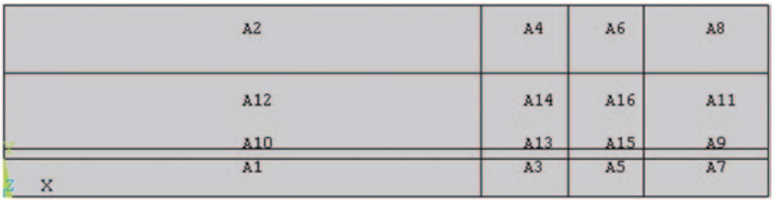


Fig. 9.71 Area plot with area numbers turned on

Table 9.12 Areas corresponding to specific materials

Material	Area numbers
Molding compound	2, 4, 5, 6, 8, 9, 11, 13, 14, 15, 16
Die-attach	10
Silicon die	12
Copper	1, 3, 7

- Mesh the areas (**AMESH** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Areas > Mapped > 3 or 4 sided

- *Pick Menu* appears; click on ***Pick All***.
- At this point, all the elements have *Material Reference Number* 1. Attributes can be changed after the elements are created. For this purpose, areas are selected and then the elements that are attached to the selected areas are selected. Finally, elements are modified to have the correct attributes. The correspondence between the areas and material numbers are given in Table 9.12. Figure 9.71 shows the area plot with area numbers printed. Plot areas (**APLOT** command) using the following menu path:

Utility Menu > Plot > Areas

- Turn area numbers on (**/PNUM** command) using the following menu path:

Utility Menu > PlotCtrls > Numbering

- *Plot Numbering Controls* dialog box appears; place a checkmark next to ***AREA Area numbers*** by clicking on the box; select ***Numbers only*** from the pull-down menu next to ***[/NUM] Numbering shown with***. Click on ***OK***. Areas appear with their numbers printed, as shown in Fig. 9.71.
- Select areas (**ASEL** command) using the following menu path:

Utility Menu > Select > Entities

- *Select Entities* dialog box appears; select ***Areas*** from the first pull-down menu; click on ***OK***.

– *Pick Menu* appears; pick area 10; click on ***OK***.
– Now, select the elements that are attached to the selected areas (**ESLA** command) using the following menu path:

Utility Menu > Select > Entities

– *Select Entities* dialog box appears; select ***Elements*** from the first pull-down menu; select ***Attached to*** from the second pull-down menu. Click on the ***Areas*** radio-button; click on ***OK***.
– Modify the attributes of the selected set of elements (**EMODIF** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Move/Modify > ElementsModify Attrib

– *Pick Menu* appears; click on ***Pick All***, which brings up the *Modify Elem Attributes* dialog box.
– Select ***Material MAT*** from the pull-down menu and enter ***2*** in the ***I1 New attribute number*** field; click on ***OK***.
– Repeat this procedure for area 12 (material reference number 3), and areas 1, 3, and 7 (material reference number 4).
– When finished, select everything (**ALLSEL** command) using the following menu path:

Utility Menu > Select > Everything

• Plot the elements (**EPLOT** command) using the following menu path:

Utility Menu > Plot > Elements

– Plot the elements with different colors based on their material reference numbers (**/PNUM** command) using the following menu path:

Utility Menu > PlotCtrls > Numbering

– *Plot Numbering Controls* dialog box appears; remove the checkmark next to ***AREA Area numbers*** by clicking on the box. Select ***Material Numbers*** for ***Elem/ Attrib numbering*** pull-down menu; select ***Colors only*** from the pull-down menu next to ***[/NUM] Numbering shown with***; click on ***OK***. When elements are plotted, they appear in different colors based on their material reference numbers, as shown in Fig. 9.72.

Solution

• Declare the new analysis to be a transient analysis (**ANTYPE** command) using the following menu path:

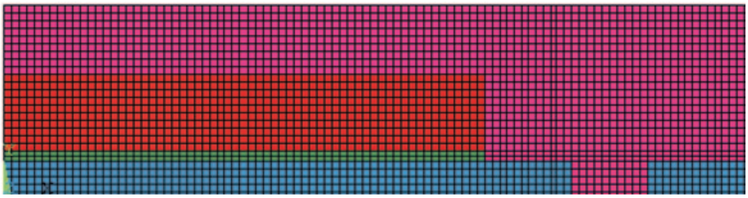


Fig. 9.72 *Mesh* representing the electronic device

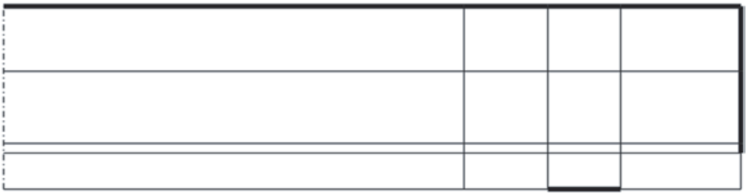


Fig. 9.73 *Bold lines* represent the surfaces of moisture diffusion

Main Menu > Solution > Analysis Type > New Analysis

- *New Analysis* dialog box appears; click on the ***Transient*** radio-button; click on ***OK***.
- *Transient Analysis* dialog box appears; click on ***OK***.

- Apply the initial condition on the nodes (**`IC`** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Initial Condit'n > Define

- *Pick Menu* appears; click on ***Pick All*** in *Pick Menu*.
- *Define Initial Conditions* dialog box appears; select ***TEMP*** from the pull-down menu; enter ***0*** for ***VALUE Initial value of DOF***; click on ***OK***.

- Apply temperature (wetness) boundary conditions on the lines (**`SFL`** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Thermal > Temperature > On Lines

- *Pick Menu* appears; pick the exterior lines that are shown in bold in Fig. 9.73; click on ***OK*** in the *Pick Menu*.
- *Apply TEMP on Lines* dialog box appears; select TEMP from the list and enter ***1*** for ***VALUE Load TEMP value***; click on ***OK***.

- Specify solution controls using the following menu path:

Main Menu > Solution > Analysis Type > Sol'n Controls

- *Solution Controls* dialog box appears, which has five tabs. In the ***Basic*** tab, within the ***Time Control*** field, enter ***168*** for ***Time at end of loadstep***.
- Click on the ***Time increment*** radio-button; enter ***1*** for ***Time step size***.
- In the ***Write Items to Results File*** field, select ***Write every substep*** from the ***Frequency*** pull-down menu; click on ***OK***.

- Obtain the solution (**`SOLVE`** command) using the following menu path:

Main Menu > Solution > Solve > Current LS

- *Confirmation Window* appears along with *Status Report Window*.
- Review status; if OK, close the *Status Report Window* and click on ***OK*** in the *Confirmation Window*.
- Wait until ANSYS responds with ***Solution is done!***

The solution for pre-conditioning is complete. Next, the solution for the reflow process is obtained. The moisture diffusivity values for the molding compound and the die-attach are different for pre-conditioning (absorption) and reflow (desorption). For this purpose, the material attributes of the molding compound and die-attach need to be modified to be 5 and 6, respectively.

- Select elements corresponding to the molding compound (**`ESEL`** command) using the following menu path:

Utility Menu > Select Entities

- *Select Entities* dialog box appears; select ***Elements*** from the first pull-down menu; select ***By Attributes*** from the second pull-down menu. Click on the ***Material num*** radio-button and type ***1*** in the test field; click on ***OK***.

- Modify the attributes of the selected set of elements (**`EMODIF`** command) using the following menu path:

Main Menu > Solution > Other > Change Mat Props > Change Mat Num

- *Change Material Number* dialog box appears; type ***5*** in the ***Mat New material number*** text field and type ***ALL*** in the ***ELEM Element no. to be modified*** text field; click on ***OK***.

- Repeat the same procedure for die-attach elements; first select elements with material attribute number 2; then modify their material attribute number to be 6.
- When finished, select everything (**`ALLSEL`** command) using the following menu path:

Utility Menu > Select > Everything

- Apply temperature (wetness) boundary conditions on lines (**`SFL`** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Thermal > Temperature > On Lines

  - *Pick Menu* appears; pick the exterior lines shown in Fig. 9.73; click on ***OK*** in the *Pick Menu*.
  - *Apply TEMP on Lines* dialog box appears; select TEMP from the list and enter ***0*** for ***VALUE Load TEMP value***; click on ***OK***.

- Specify solution controls using the following menu path:

Main Menu > Solution > Analysis Type > Sol'n Controls

  - *Solution Controls* dialog box appears, which has five tabs. In the ***Basic*** tab, within the ***Time Control*** field, enter ***168 + 5/60*** for ***Time at end of loadstep***.
  - Click on the ***Time increment*** radio-button; enter ***5/60/20*** for ***Time step size***.
  - In the ***Write Items to Results File*** field, select ***Write every substep*** from the ***Frequency*** pull-down menu; click on ***OK***.

- Obtain the solution (**`SOLVE`** command) using the following menu path:

Main Menu > Solution > Solve > Current LS

  - *Confirmation Window* appears along with *Status Report Window*.
  - Review status; if OK, close the *Status Report Window* and click on ***OK*** in *Confirmation Window*.
  - Wait until ANSYS responds with ***Solution is done!***

Postprocessing

- Read the results for the last time point using the following menu path:

Main Menu > General Postproc > Read Results > Last Set

  - Plot temperature (wetness) contours (**`PLNSOL`** command) at the last time point using the following menu path:

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu

  - *Contour Nodal Solution Data* dialog box appears. Select ***DOF Solution*** and ***Nodal Temperature***; click on ***OK***. The contour plot appears, as shown in Fig. 9.74.
  - Plot flux vector contours (**`PLVECT`** command) at the last time point using the following menu path:

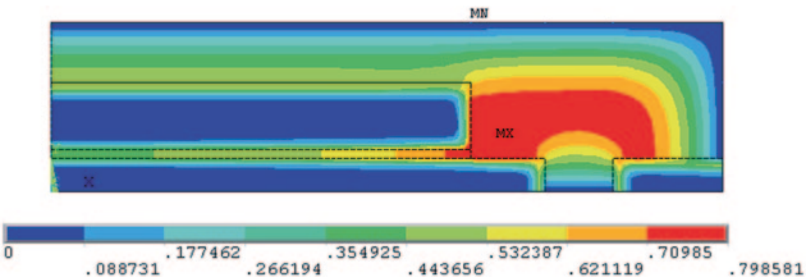


Fig. 9.74 Contour plot of wetness

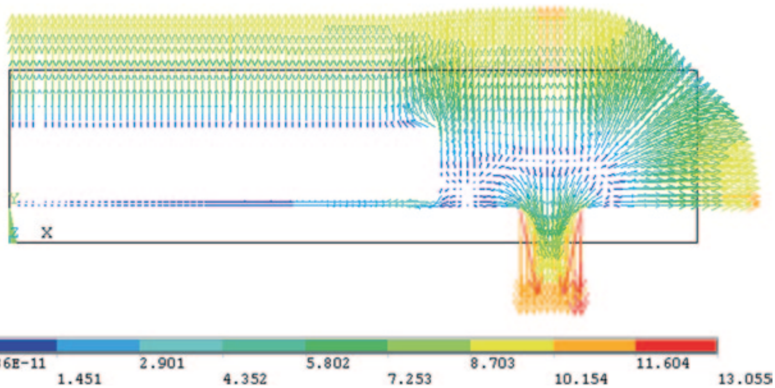


Fig. 9.75 Vector plot of flux

Main Menu > General Postproc > Plot Results > Vector Plot > Predefined

– *Vector Plot of Predefined Vectors* dialog box appears; click on ***OK***. The vector plot appears, as shown in Fig. 9.75.

- Obtain an animation of the temperature (wetness) over time using the following menu path:

Utility Menu > PlotCtrls > Animate > Over Time

- *Animate Over Time* dialog box appears; enter ***100*** for ***Number of animation frames***, select ***Load Step Range*** radio-button and enter ***1*** and ***2*** for ***Range Minimum, Maximum***. Click on ***OK***; wait until the *Animation Controls Window* appears.
- Obtain the moisture content as a function of time. This portion of the postprocessing is performed using the ANSYS Parametric Design Language (APDL) as discussed in substantial detail in Chap. 7. It is assumed that the results are stored in a file named ***WET.RTH***. The input file (written using APDL) is given below. There are two `DO` loops: one over the substeps and the other over the elements. During each of the loops over the elements, each element's area, saturated

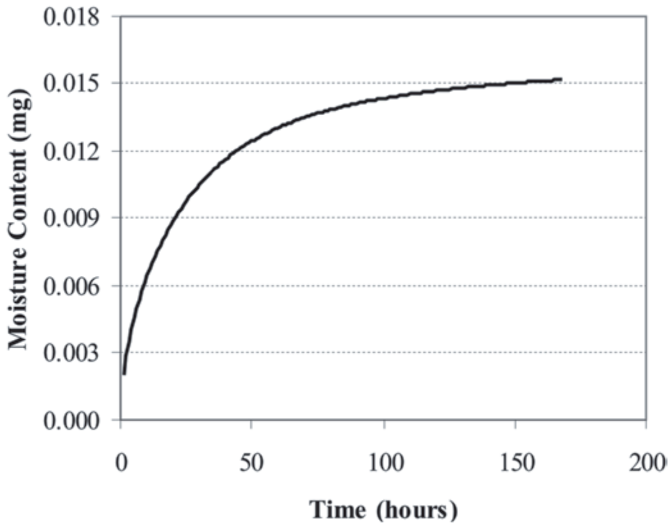


Fig. 9.76 Moisture content vs. time during absorption

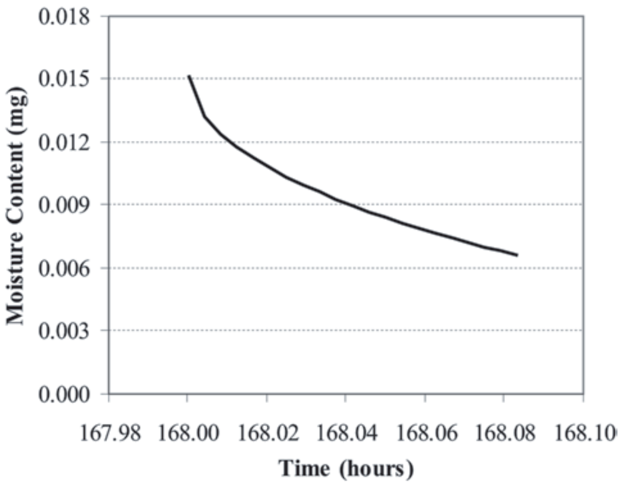


Fig. 9.77 Moisture content vs. time during desorption

moisture concentration, and wetness parameter are used to calculate the element's moisture content in milligrams. Since this is a two-dimensional idealization, the package is assumed to have a unit thickness in the third direction. The results (time and total moisture content in the package) are written to two different text files, i.e., ***ABS.OUT*** for absorption and ***DES.OUT*** for desorption. Figs. 9.76 and 9.77 show the variation of total moisture content in the package against time during absorption and desorption, respectively.

```
/POST1                          ! ENTER POSTPROCESSOR
FILE,'WET','RTH','.'            ! DECLARE RESULTS FILE
CSAT5=7.06E-3                   ! C_sat FOR MOLDING COMPOUND
CSAT6=6.20E-3                   ! C_sat FOR DIE-ATTACH
CSAT3=0                         ! C_sat FOR DIE
CSAT4=0                         ! C_sat FOR COPPER
*GET,NUMEL,ELEM,0,COUNT         ! OBTAIN NUMBER OF ELEMENTS
SET,FIRST                       ! READ 1ST RESULTS SET
                                ! (SUBSTEP 1)
*DO,J,1,188                     ! LOOP OVER SUBSTEPS (TOTAL
                                ! 188)
ETABLE,,TEMP                    ! STORE WETNESS IN ELEMENT
                                ! TABLE
*GET,MINEL,ELEM,0,NUM,MIN       ! OBTAIN MINIMUM ELEMENT
                                ! NUMBER
*GET,TIM,ACTIVE,0,SET,TIME      ! OBTAIN CURRENT TIME
SUM=0                           ! INITIALIZE TOTAL MOISTURE
                                ! MASS
*DO,I,1,NUMEL                   ! LOOP OVER ELEMENTS
*GET,EA,ELEM,MINEL,AREA         ! OBTAIN ELEMENT AREA
*GET,EM,ELEM,MINEL,ATTR,MAT     ! OBTAIN ELEMENT MATERIAL
                                ! NUMBER
*GET,EW,ETAB,1,ELEM,MINEL       ! OBTAIN ELEMENT WETNESS
EC=1E6*EA*EW*CSAT%EM%           ! CALCULATE MOISTURE MASS (mg)
SUM=SUM+EC                      ! UPDATE TOTAL MOISTURE MASS
MINEL=ELNEXT(MINEL)             ! MOVE ON TO NEXT ELEMENT
*ENDDO                          ! END LOOP OVER ELEMENTS
*IF,J,LE,168,THEN               ! IF TRUE - ABSORPTION
/OUTPUT,ABS,OUT,,APPEND         ! REDIRECT OUTPUT TO FILE
                                ! ABS.OUT

*VWRITE,TIM,SUM                 ! WRITE TIME AND MOISTURE MASS



                                ! TO FILE
(E16.8,5X,E16.8)                ! FORMAT FOR *VWRITE COMMAND
/OUTPUT                         ! REDIRECT OUTPUT TO OUTPUT
                                ! WINDOW
*ELSEIF,J,GT,168,THEN           ! IF TRUE - DESORPTION
/OUTPUT,DES,OUT,,APPEND         ! REDIRECT OUTPUT TO FILE
                                ! DES.OUT
*VWRITE,TIM,SUM                 ! WRITE TIME AND MOISTURE MASS
                                ! TO FILE
(E16.8,5X,E16.8)                ! FORMAT FOR *VWRITE COMMAND
/OUTPUT                         ! REDIRECT OUTPUT TO OUTPUT
                                ! WINDOW
*ENDIF                          !
*IF,J,LT,188,THEN               !
SET,,,,,,,J+1                   ! READ NEXT RESULTS SET
*ENDIF                          !
*ENDDO                          ! END LOOP OVER SUBSTEPS
```