4.4 Mode analysis of a one-axis precision moving table using elastic hinges

Figure 4.96 Window of Element Sizes on Picked Lines.

(5) Pick the area of the table on ANSYS Graphics window and click [F] OK button. Then the meshed drawing of the table appears on ANSYS Graphics window as shown in Figure 4.99.

Next, by performing the following steps, the thickness of 5 mm and the mesh size are determined for the drawing of the table.

**Command**

ANSYS Main Menu → Preprocessor → Modeling → Operate → Extrude → Elem Ext Opt

The window Element Extrusion Options opens (Figure 4.100).

(1) Input [A] 5 to VAL1 box. This means that the number of element divisions is 5 in the thickness direction. Then, click [B] OK button.

**Command**

ANSYS Main Menu → Preprocessor → Modeling → Areas → By XYZ Offset

The window Extrude Area by Offset opens (Figure 4.101).

(1) Pick the area of the table on ANSYS Graphics window and click [A] OK button. Then, the window Extrude Areas by XYZ Offset opens (Figure 4.102).

(2) Input [B] 0,0,0.005 to DX, DY, DZ box and click [C] OK button. Then, the drawing of the table meshed in the thickness direction appears as shown in Figure 4.103.

4.4.2.6 **Boundary Conditions**

The table is fixed at both the bottom and the region A of the table.
Chapter 4  Mode analysis

Figure 4.97  Window of Mesh Tool.

Figure 4.98  Window of Mesh Areas.
Figure 4.99  ANSYS Graphics window.

Figure 4.100  Window of Element Extrusion Options.
Figure 4.101  Window of Extrude Area by Offset.

Figure 4.102  Window of Extrude Areas by XYZ Offset.
4.4 Mode analysis of a one-axis precision moving table using elastic hinges

4.4.3 Analysis

4.4.3.1 Define the type of analysis

The following steps are performed to define the type of analysis.

Command | ANSYS Main Menu → Solution → Analysis Type → New Analysis

The window New Analysis opens (Figure 4.108).
Figure 4.104 Window of Apply U,ROT on Areas.

Figure 4.105 ANSYS Graphics window.
4.4 Mode analysis of a one-axis precision moving table using elastic hinges

Figure 4.106 Window of Apply U,ROT on Areas.

Figure 4.107 ANSYS Graphics window.
(1) Check [A] Modal and, then, click [B] OK button.

In order to define the number of modes to extract, the following steps are performed.

**COMMAND** | ANSYS Main Menu → Solution → Analysis Type → Analysis Options

The window Modal Analysis opens (Figure 4.109).

(1) Check [A] Subspace of MODOPT and input [B] 3 in the box of No. of modes to extract and click [C] OK button.

(2) Then, the window Subspace Modal Analysis as shown in Figure 4.110 opens. Input [D] 5000 in the box of FREQE and click [E] OK button.

### 4.4.3.2 EXECUTE CALCULATION

**COMMAND** | ANSYS Main Menu → Solution → Solve → Current LS

The window Solve Current Load Step opens.

(1) Click OK button and calculation starts. When the window Note appears, the calculation is finished.

(2) Click Close button and the window is closed. The window /STATUS Command is also open but this window can be closed by clicking the mark of X at the upper right side of the window.
4.4 Mode analysis of a one-axis precision moving table using elastic hinges

4.4.4 Postprocessing

4.4.4.1 Read the Calculated Results of the First Mode of Vibration

**Command** | ANSYS Main Menu → General Postproc → Read Results → First Set

4.4.4.2 Plot the Calculated Results

**Command** | ANSYS Main Menu → General Postproc → Plot Results → Deformed Shape
Figure 4.110 Window of Subspace Modal Analysis.

The window Plot Deformed Shape opens (Figure 4.111).

1. Select [A] Def+Undeformed and click [B] OK.
2. The calculated result for the first mode of vibration appears on ANSYS Graphics window as shown in Figure 4.112.

4.4.4.3 Read the Calculated Results of the Second and Third Modes of Vibration

**Command** | ANSYS Main Menu → General Postproc → Read Results → Next Set

Perform the same steps described in Section 4.4.4.2 and the results calculated for the higher modes of vibration are displayed as shown in Figures 4.113 and 4.114.
4.4 Mode analysis of a one-axis precision moving table using elastic hinges

In order to easily observe the vibration mode shape, the animation of mode shape can be used.

**Command**  
Utility Menu → PlotCtrls → Animate → Mode Shape
Chapter 4  Mode analysis

Figure 4.113  ANSYS Graphics window for the second mode.

Figure 4.114  ANSYS Graphics window for the third mode.
The window **Animate Mode Shape** opens (Figure 4.115).

![Figure 4.115 Window of Animate Mode Shape.](image)

(1) Input [A] 0.1 to **Time delay** box and click [B] **OK** button. Then the animation of the mode shape is displayed in **ANSYS Graphics** window.
5.1 Introduction

Various fluids such as air and liquid are used as an operating fluid in a blower, a compressor, and a pump. The shape of flow channel often determines the efficiency of these machines. In this chapter, the flow structures in a diffuser and the channel with a butterfly valve are examined by using FLOTRAN which is an assistant program of ANSYS. A diffuser is usually used for increasing the static pressure by reducing the fluid velocity and the diffuser can be easily found in a centrifugal pump as shown in Figure 5.1.

Figure 5.1 Typical machines for fluid.
5.2 **ANALYSIS OF FLOW STRUCTURE IN A DIFFUSER**

5.2.1 **PROBLEM DESCRIPTION**

Analyze the flow structure of an axisymmetric conical diffuser with diffuser angle $2\theta = 6^\circ$ and expansion ratio $= 4$ as shown in Figure 5.2.

![Figure 5.2](image)

**Figure 5.2** Axisymmetrical conical diffuser.

Shape of the flow channel:

1. Diffuser shape is axisymmetric and conical, diffuser angle $2\theta = 6^\circ$, expansion ratio $= 4$.
2. Diameter of entrance of the diffuser: $D_E = 0.2$ m.
3. Length of straight channel for entrance: $4.5D_E$.
5. Length of straight channel for exit: $50.0D_E$.

Operating fluid: Air (300 K)

Flow field: Turbulence

Velocity at the entrance: $20$ m/s

Reynolds number: $2.54 \times 10^5$ (assumed to set the diameter of the diffuser entrance to a representative length)

Boundary conditions:

1. Velocities in all directions are zero on all walls.
2. Pressure is equal to zero at the exit.
3. Velocity in the $y$ direction is zero on the $x$-axis.

5.2.2 **CREATE A MODEL FOR ANALYSIS**

5.2.2.1 **SELECT KIND OF ANALYSIS**

**COMMAND** | ANSYS Main Menu $\rightarrow$ Preferences
5.2 Analysis of flow structure in a diffuser

The window **Preferences for GUI Filtering** opens (Figure 5.3).


---

![Preferences for GUI Filtering](image)

**Figure 5.3** Window of Preferences for GUI Filtering.

---

### 5.2.2.2 ELEMENT TYPE SELECTION

**Command** | ANSYS Main Menu → Preprocessor → Element Type → Add/Edit/Delete

Then the window **Element Types** as shown in Figure 5.4 opens.

1. Click [A] add. Then the window **Library of Element Types** as shown in Figure 5.5 opens.

(3) Click [C] **OK** button and click [D] **Options** button in the window of Figure 5.6.
(4) The window **FLUID141 element type options** opens as shown in Figure 5.7. Select [E] **Axisym about X** in the box of **Element coordinate system** and click [F] **OK** button. Finally click [G] **Close** button in Figure 5.6.
5.2 Analysis of flow structure in a diffuser

5.2.2.3 CREATE KEYPONITS

To draw a diffuser for analysis, the method using keypoints on the window are described in this section.
Command

ANSYS Main Menu → Preprocessor → Modeling → Create → Keypoints → In Active CS

(1) The window Create Keypoints in Active Coordinate System opens (Figure 5.8).

![Create Keypoints in Active Coordinate System](image)

Figure 5.8 Window of Create Keypoints in Active Coordinate System.

(2) Input [A] 0, 0 to X, Y, Z Location in active CS box, and then click [B] Apply button. Do not click OK button at this stage. If OK button is clicked, the window will be closed. In this case, open the window Create Keypoints in Active Coordinate System again and then perform step (2).

(3) In the same window, input the values of keypoints indicated in Table 5.1.

<table>
<thead>
<tr>
<th>KP No.</th>
<th>X</th>
<th>Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>0.9</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>0.9</td>
<td>0.1</td>
</tr>
<tr>
<td>4</td>
<td>0</td>
<td>0.1</td>
</tr>
<tr>
<td>5</td>
<td>2.81</td>
<td>0</td>
</tr>
<tr>
<td>6</td>
<td>2.81</td>
<td>0.2</td>
</tr>
<tr>
<td>7</td>
<td>12.81</td>
<td>0</td>
</tr>
<tr>
<td>8</td>
<td>12.81</td>
<td>0.2</td>
</tr>
</tbody>
</table>

(4) After finishing step (3), eight keypoints appear on the window as shown in Figure 5.9.
5.2 Analysis of flow structure in a diffuser

5.2.2.4 CREATE AREAS FOR DIFFUSER

Areas are created from the keypoints by performing the following steps.

**COMMAND**

ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Arbitrary → Through KPs

(1) The window **Create Area thru KPs** opens (Figure 5.10).

(2) Pick keypoints 1, 2, 3, and 4 in Figure 5.9 in order and click [A] Apply button in Figure 5.10. One area of the diffuser is created on the window.

(3) Then another two areas are made on the window by clicking keypoints listed in Table 5.2.

(4) When three areas are made, click [B] OK button in Figure 5.10.

**Figure 5.9** ANSYS Graphics window.

**Figure 5.10** Window of Create Area thru KPs.
### Chapter 5 Analysis for fluid dynamics

#### Table 5.2 Keypoint numbers for making areas

<table>
<thead>
<tr>
<th>Area No.</th>
<th>KPs</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1, 2, 3, 4</td>
</tr>
<tr>
<td>2</td>
<td>2, 5, 6, 3</td>
</tr>
<tr>
<td>3</td>
<td>5, 7, 8, 6</td>
</tr>
</tbody>
</table>

#### 5.2.2.5 CREATE MESH IN LINES AND AREAS

**Command**  
ANSYS Main Menu → Preprocessor → Meshing → Mesh Tool

The window Mesh Tool opens (Figure 5.11).

1. Click [A] Lines-Set box. Then the window **Element Size on Picked Lines** opens (Figure 5.12).
2. Pick Line 1 and Line 3 on ANSYS Graphics window (Line numbers and Keypoint numbers are indicated in Figure 5.13) and click [B] OK button. The window **Element Sizes on Picked Lines** opens (Figure 5.14).
3. Input [C] 15 to NDIV box and click [D] OK button.
4. Click [A] Lines-Set box in Figure 5.11 and pick Line 2, Line 6, and Line 9 on ANSYS Graphics window. Then click OK button.
5. Input [E] 50 to NDIV box and [F] 0.2 to SPACE in Figure 5.15. Then click [G] OK button. This means that the last dividing space between grids becomes one-fifth of the first dividing space on Line 2. When Line 2 was made according to Table 5.2, KP 2 was first picked and then KP 3. So the dividing space of grids becomes smaller toward KP 3.
6. In order to mesh all lines, input the values listed in Table 5.3.

**Command**  
ANSYS Main Menu → Preprocessor → Meshing → Mesh Tool

The window Mesh Tool opens (Figure 5.16).

1. Click Mesh on the window Mesh Tool in Figure 5.11 and the window Mesh Areas opens (Figure 5.16). Click [A] Pick All button when all areas are divided into elements as seen in Figure 5.17.
5.2 Analysis of flow structure in a diffuser

Figure 5.12 Window of Element Size on Picked Lines.

Figure 5.13 Keypoint and Line numbers for a diffuser.

**Command** Utility Menu → PlotCtrls → Pan-Zoom-Rotate

The window Pan-Zoom-Rotate opens (Figure 5.18).

1. Click [A] Box Zoom and [B] make a box on ANSYS Graphics window to zoom up the area as shown in Figure 5.17. Then the enlarged drawing surrounded by the box appears on the window as shown in Figure 5.19.
Figure 5.14  Window of Element Sizes on Picked Lines.

Figure 5.15  Window of Element Sizes on Picked Lines.
Table 5.3  Element sizes of picked lines

<table>
<thead>
<tr>
<th>Line No.</th>
<th>NDIV</th>
<th>SPACE</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>15</td>
<td>0.2</td>
</tr>
<tr>
<td>2</td>
<td>50</td>
<td>–</td>
</tr>
<tr>
<td>3</td>
<td>15</td>
<td>–</td>
</tr>
<tr>
<td>4</td>
<td>50</td>
<td>5</td>
</tr>
<tr>
<td>5</td>
<td>30</td>
<td>–</td>
</tr>
<tr>
<td>6</td>
<td>50</td>
<td>0.2</td>
</tr>
<tr>
<td>7</td>
<td>30</td>
<td>–</td>
</tr>
<tr>
<td>8</td>
<td>40</td>
<td>5</td>
</tr>
<tr>
<td>9</td>
<td>50</td>
<td>0.2</td>
</tr>
<tr>
<td>10</td>
<td>40</td>
<td>0.2</td>
</tr>
</tbody>
</table>

Figure 5.16  Window of Mesh Areas.

Figure 5.17  ANSYS Graphics window.
5.2.2.6 BOUNDARY CONDITIONS

In this section, boundary conditions listed in problem description (Section 5.2.1) are set. By performing the steps mentioned below, all lines need to appear on the window. Then set the boundary conditions.
5.2 Analysis of flow structure in a diffuser

**Command**

Utility Menu → Plot → Lines

**Command**

ANSYS Main Menu → Solution → Define Loads → Apply → Fluid/CFD → Velocity → On Lines

The window *Apply V on Lines* opens (Figure 5.20).

---

![Window of Apply V on Lines](image.png)

**Figure 5.20** Window of *Apply V on Lines*.

1. Pick Line 4 at the entrance of the diffuser and click [A] OK button. Then the window *Apply VELO load on lines* as shown in Figure 5.21 opens.

2. Input [B] 20 to VX box and [C] 0 to VY box. Then, click [D] OK button. The sign of setting boundary condition appears on ANSYS Graphics window as shown in Figure 5.22.

3. Set boundary conditions to all lines listed in Table 5.4.
Figure 5.21  Window of Apply VELO load on lines.

**COMMAND**  
ANSYS Main Menu → Solution → Define Loads → Apply → Fluid/CFD → Pressure DOF → On Lines

The window **Apply PRES on Lines** opens (Figure 5.23).

(1) Pick Line 9 on ANSYS Graphics window and click [A] OK button. Then the window **Apply PRES on lines** opens (Figure 5.24).
5.2 Analysis of flow structure in a diffuser

Figure 5.22 ANSYS Graphics window.

Table 5.4 Boundary conditions for all lines

<table>
<thead>
<tr>
<th>Line No.</th>
<th>VX</th>
<th>VY</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>–</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>3</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>4</td>
<td>20</td>
<td>0</td>
</tr>
<tr>
<td>5</td>
<td>–</td>
<td>0</td>
</tr>
<tr>
<td>6</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>7</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>8</td>
<td>–</td>
<td>0</td>
</tr>
<tr>
<td>9</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>10</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
Figure 5.23  Window of Apply PRES on lines.

Figure 5.24  Window of Apply PRES on lines.
5.2 Analysis of flow structure in a diffuser

5.2.3 Execution of the analysis

5.2.3.1 FLOTRAN SET UP

The following steps are performed to set up the analysis of FLOTRAN.

**Command**

ANSYS Main Menu → Solution → FLOTRAN Set Up → Solution Options

The window FLOTRAN Solution Options opens (Figure 5.26).

(1) Set Adiabatic, Turbulent, and Incompressible to TEMP, TURB, and COMP boxes, and, then, click [B] OK button.

**Command**

ANSYS Main Menu → Solution → FLOTRAN Set Up → Execution Ctrl

(2) Input [B] 0 to PRES box and close the window by clicking [C] OK button. The drawing on ANSYS Graphics window is displayed as shown in Figure 5.25.
Chapter 5  Analysis for fluid dynamics

The window **Steady State Control Settings** opens (Figure 5.27).

1. Input [A] 200 to EXEC box and click [B] OK button.

**Command**  |  ANSYS Main Menu → Solution → FLOTRAN Set Up → Fluid Properties

The window **Fluid Properties** opens (Figure 5.28).

1. Select [A] AIR-SI in **Density**, **Viscosity**, **Conductivity**, and **Specific heat** boxes and click [B] OK button. Then the window **CFD Fluid Properties** as shown in Figure 5.29 opens and click [C] OK button.

**Figure 5.26** Window of FLOTRAN Solution Options.
5.2 Analysis of flow structure in a diffuser

5.2.4 Execute calculation

**Command** | ANSYS Main Menu → Solution → Run FLOTRAN

When the calculation is finished, the window **Note** as shown in Figure 5.30 opens.

(1) Click [A] **Close** button.
Figure 5.28 Window of Fluid Properties.

5.2.5 POSTPROCESSING

5.2.5.1 READ THE CALCULATED RESULTS OF THE FIRST MODE OF VIBRATION

COMMAND | ANSYS Main Menu → General Postproc → Read Results → Last Set

5.2.5.2 PLOT THE CALCULATED RESULTS

COMMAND | ANSYS Main Menu → General Postproc → Plot Results → Vector Plot → Predefined
5.2 Analysis of flow structure in a diffuser

Figure 5.29 Window of CFD Fluid Properties.

Figure 5.30 Window of Note.

The window Vector Plot of Predefined Vectors as shown in Figure 5.31 opens.

2. The calculated result for velocity vectors appears on ANSYS Graphics window as shown in Figure 5.32.
Figure 5.31 Window of Vector Plot of Predefined Vectors.

Figure 5.32 ANSYS Graphics window.
5.2.5.3 PLOT THE CALCULATED RESULTS BY PATH OPERATION

In order to plot the calculated results for a selected cross section, Path Operation command is used.

**COMMAND**

ANSYS Main Menu → General Postproc → Path Operations → Define Path → By Nodes

The window By Nodes opens (Figure 5.33).

1. Check [A] Pick and pick two nodes on [B] the wall and [C] the x-axis as shown in Figure 5.34. If a wrong node on ANSYS Graphics window is selected, delete the picked node by using Unpick. Click [D] OK button.

2. The window By Nodes as shown in Figure 5.35 opens. Input the path name [E] vel01 to Name box [F] 50 to nDiv box and click [G] OK button. Then the window PATH Command appears as shown in Figure 5.36, which explains the content of the defined path. Close this window by clicking X mark at the upper right end.

![Figure 5.33 Window of By Nodes.](image_url)
Figure 5.34  ANSYS Graphics window.

Figure 5.35  Window of By Nodes.
5.2 Analysis of flow structure in a diffuser

Figure 5.36  Window of PATH Command.

The window **Map Result Items onto Path** opens (Figure 5.37).

1. Input [A] vel01 to Lab box and select [B] **DOF solution** and **Velocity VX**. Then click **OK** button.

Figure 5.37  Window of Map Result Items onto Path.

**COMMAND**  
ANSYS Main Menu → General Postproc → Path Operations → Plot Path Item → On Graph
The window **Plot of Path Items on Graph** opens (Figure 5.38).

(1) Select [A] VEL01 in Lab1-6 box and click OK button. Then the calculated result for the defined path appears on **ANSYS Graphics** window as shown in Figure 5.39.

![Figure 5.38 Window of Plot of Path Items on Graph.](image1)

![Figure 5.39 ANSYS Graphics window.](image2)

In addition, when the values of the defined path are needed, the following steps can be used.
5.2 Analysis of flow structure in a diffuser

**COMMAND**

ANSYS Main Menu → General Postproc → List Results → Path Items

The window **List of Path Items** opens (Figure 5.40).

1. Select [A] **VEL01** in **Lab1-6** box and click **OK** button. Then the list for the defined path appears as shown in Figure 5.41.

---

**Figure 5.40** Window of List Path Items.

---

**Figure 5.41** Window of PRPATH Command.
5.3 **Analysis of flow structure in a channel with a butterfly valve**

5.3.1 **Problem description**

Analyze the flow structure around a butterfly valve as shown in Figure 5.42.

**Figure 5.42** Flow channel with a butterfly valve (photo is quoted from http://www.kitz.co.jp/product/bidg-jutaku/kyutouyou/index.html).

Shape of the flow channel:

(1) Diameter of the flow channel: $D_E = 0.06$ m.
(2) Diameter of a butterfly valve: $D_E = 0.06$ m.
(3) Tilt angle of a butterfly valve: $\alpha = 30^\circ$.

Operating fluid: water
Flow field: turbulent
Velocity at the entrance: 0.01 m/s

Boundary conditions:

(1) Velocities in all directions are zero on all walls.
(2) Pressure is equal to zero at the exit.
(3) Velocity in the $y$ direction is zero on the $x$-axis.

5.3.2 **Create a model for analysis**

5.3.2.1 **Select kind of analysis**

**Command** | ANSYS Main Menu → Preferences
5.3.2.2 SELECT ELEMENT TYPE

**Command** | ANSYS Main Menu → Preprocessor → Element Type → Add/Edit/Delete

Then the window *Element Types* opens.

1. Click *add* and then the window *Library of Element Types* opens.
2. Select *FLOTTRAN CFD-2D FLOTTRAN 141*. Click *OK* button.
3. Click *Options* button in the window *Element Types*.
4. The window *FLUID141 element type options* as shown in Figure 5.43 opens. Select [A] *Cartesian* in the box of *Element coordinate system* and click *OK* button. Finally click *Close* button in the window *Element Types*.

![Figure 5.43 Window of FLUID141 element type options.](image)

5.3.2.3 CREATE KEYPONITS

To draw a flow channel with a butterfly valve for analysis, keypoints on the window are used to describe it in this section.

**Command** | ANSYS Main Menu → Preprocessor → Modeling → Create → Keypoints → In Active CS

1. The window *Create Keypoints in Active Coordinate System* opens.
2. Input the X and Y coordinate values to *X, Y, Z Location in active CS* box listed in Table 5.5.
### Table 5.5  X and Y coordinates of KPs for a flow channel

<table>
<thead>
<tr>
<th>KP No.</th>
<th>X</th>
<th>Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>0</td>
<td>-0.03</td>
</tr>
<tr>
<td>3</td>
<td>0.2</td>
<td>-0.03</td>
</tr>
<tr>
<td>4</td>
<td>0.2</td>
<td>0</td>
</tr>
<tr>
<td>5</td>
<td>0.2</td>
<td>0.03</td>
</tr>
<tr>
<td>6</td>
<td>0</td>
<td>0.03</td>
</tr>
<tr>
<td>7</td>
<td>0.05587</td>
<td>-0.02648</td>
</tr>
<tr>
<td>8</td>
<td>0.0716</td>
<td>0</td>
</tr>
<tr>
<td>9</td>
<td>0.08587</td>
<td>0.02548</td>
</tr>
<tr>
<td>10</td>
<td>0.08413</td>
<td>0.02648</td>
</tr>
<tr>
<td>11</td>
<td>0.06885</td>
<td>0</td>
</tr>
<tr>
<td>12</td>
<td>0.05413</td>
<td>-0.02548</td>
</tr>
</tbody>
</table>

(3) After finishing step (2), 12 keypoints appear on the window as shown in Figure 5.44. Four keypoints on the x-axis are made to use variable grids of which distance becomes smaller as a grid approaches the wall.
5.3 Analysis of flow structure in a channel with a butterfly valve

5.3.2.4 CREATE AREAS FOR FLOW CHANNEL

Areas are created from keypoints by performing the following steps.

**COMMAND**

ANSYS Main Menu → Preprocessor → Modeling → Create → Areas →
Arbitrary → Through KPs

(1) The window Create Area thru KPs opens.
(2) Pick keypoints 1, 2, 3, 4, 5, 6 and 11, 12, 7, 8, 9, 10 in order, and, then, two areas
are created on the window as shown in Figure 5.45.
(3) When two areas are made, click OK button in the window Create Area thru KPs.

![Figure 5.45 ANSYS Graphics window.](image)

5.3.2.5 SUBTRACT THE VALVE AREA FROM THE CHANNEL AREA

According to the following steps, the valve area is subtracted from the channel area.

**COMMAND**

ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans →
Subtract → Areas
(1) The window **Subtract Areas** opens (Figure 5.46).
(2) Click the channel area displayed on **ANSYS Graphics** window and **OK** button in Figure 5.46. Then click the valve area and **OK** button in Figure 5.46. The drawing of the channel appears as shown in Figure 5.47.

---

**5.3.2.6 CREATE MESH IN LINES AND AREAS**

First, in order to indicate the number of keypoints and lines on **ANSYS Graphics** window, perform the following steps.

**Command**  |  Utility Menu → Plot → Lines

**Command**  |  Utility Menu → PlotCtrls → Numbering

(1) Check the boxes of **KP** and **LINE** of the window **Plot Numbering Controls** and click **OK** button.

**Command**  |  ANSYS Main Menu → Preprocessor → Meshing → Mesh Tool
5.3 Analysis of flow structure in a channel with a butterfly valve

(1) Click **Lines Set** box in the window Mesh Tool. Then the window **Element size on Picked Lines** as shown in Figure 5.48 opens.

(2) Pick lines 1, 4, 7, 10 on **ANSYS Graphics** window and click **OK** button in Figure 5.48. The window **Element Sizes on Picked Lines** as shown in Figure 5.49 opens.

(3) Input 20 to **NDIV** box and 0.2 to **SPACE** box and, then, click **Apply** button. Next, input figures to these boxes according to Table 5.6.

(4) When all figures are inputted, click **OK** button of the window **Element Size on Picked Lines**.

---

![Figure 5.48](image1.png)

**Figure 5.48** Window of **Element Size on Picked Lines**.

![Figure 5.49](image2.png)

**Figure 5.49** Window of **Element Sizes on Picked Lines**.

<table>
<thead>
<tr>
<th>Line No.</th>
<th>NDIV</th>
<th>SPACE</th>
</tr>
</thead>
<tbody>
<tr>
<td>1, 4, 7, 10</td>
<td>20</td>
<td>0.2</td>
</tr>
<tr>
<td>3, 6, 9, 12</td>
<td>20</td>
<td>5</td>
</tr>
<tr>
<td>2, 5</td>
<td>100</td>
<td>–</td>
</tr>
<tr>
<td>8, 11</td>
<td>2</td>
<td>–</td>
</tr>
</tbody>
</table>
**Command**

ANSYS Main Menu → Preprocessor → Meshing → Mesh Tool

The window Mesh Tool opens.

(1) Click **Mesh** on the window Mesh Tool and the window Mesh Areas opens. Click the channel area on ANSYS Graphics window and, then, click **OK** button of the window Mesh Areas. Meshed area appears on ANSYS Graphics window as shown in Figure 5.50.

![Figure 5.50 ANSYS Graphics window.](image)

5.3.2.7 **Boundary Conditions**

The boundary conditions can be set by the following steps.

**Command**

ANSYS Main Menu → Solution → Define Loads → Apply → Fluid/CFD → Velocity → On Lines

The window Apply V on Lines opens.

(1) Pick **Line 1** and **Line 6** at the entrance of the channel on ANSYS Graphics window and click **OK** button. Then the window Apply VELO load on lines opens (Figure 5.51).

(2) Input [A] **0.01** to VX box and [B] **0** to VY box. Then, click [C] **OK** button.
5.3 Analysis of flow structure in a channel with a butterfly valve

Figure 5.51 Window of Apply VELO load on lines.

(3) Pick Line Number of 2, 5, 7, 8, 9, 10, 11, 12 and click OK button. Input 0 to VX and VY boxes in Figure 5.51 and click OK button.

**Command**

ANSYS Main Menu → Solution → Define Loads → Apply → Fluid/CFD → Pressure DOF → On Lines

The window Apply PRES on Lines opens (Figure 5.52).

(1) Pick Line 3 and Line 4 on ANSYS Graphics window and click OK button. Then the window Apply PRES on lines opens (Figure 5.53).

(2) Input [B] 0 to PRES box and close the window by clicking [C] OK button.
Figure 5.52  Window of **Apply PRES on lines**.

Figure 5.53  Window of **Apply PRES on lines**.
5.3 Analysis of flow structure in a channel with a butterfly valve

5.3.3 Execution of the analysis

5.3.3.1 FLOTRAN Set Up

The following steps are performed to set up the analysis of FLOTRAN.

**Command**

ANSYS Main Menu → Solution → FLOTRAN Set Up → Solution Options

The window FLOTRAN Solution Options opens (Figure 5.54).

1. Set [A] Adiabatic, Turbulent, and Incompressible to TEMP, TURB, and COMP boxes, and, then, click [B] OK button.

![Figure 5.54 Window of FLOTRAN Solution Options.](image-url)
Chapter 5  Analysis for fluid dynamics

**COMMAND** | ANSYS Main Menu → Solution → FLOTRAN Set Up → Fluid Properties

The window **Fluid Properties** opens (Figure 5.55).

1. Select [A] **Liquid** in **Density**, **Viscosity**, and **Conductivity** boxes and click [B] **OK** button. Then the window **CFD Flow Properties** as shown in Figure 5.56 opens. Input [C] 1000 to **D0** box and [D] 0.001 to **V0** box. Then click [E] **OK** button.

![Figure 5.55](image_url)

**Figure 5.55** Window of **Fluid Properties**.
5.3 Analysis of flow structure in a channel with a butterfly valve

Figure 5.56 Window of CFD Flow Properties.

**COMMAND** | ANSYS Main Menu → Solution → FLOTRAN Set Up → Execution Ctrl

The window **Steady State Control Settings** opens (Figure 5.57).

1. Input [A] 200 to EXEC box and click [B] OK button.

### 5.3.4 Execute Calculation

**COMMAND** | ANSYS Main Menu → Solution → Run FLOTRAN

When the calculation is finished, the window **Note** opens.

1. Click Close button.
5.3.5 POSTPROCESSING

5.3.5.1 READ THE CALCULATED RESULTS

**Command**  | ANSYS Main Menu → General Postproc → Read Results → Last Set

---

Figure 5.57  Window of Steady State Control Settings.
5.3 Analysis of flow structure in a channel with a butterfly valve

5.3.5.2 Plot the Calculated Results

**Command**

ANSYS Main Menu → General Postproc → Plot Results → Vector Plot → Predefined

The window **Vector Plot of Predefined Vectors** opens (Figure 5.58).

2. The calculated result for velocity vectors appears on **ANSYS Graphics** window as shown in Figure 5.59.
5.3.5.3 Detailed View of the Calculated Flow Velocity

Near separation points in front of or the back of the valve surface, the flow velocity is very small and it is very difficult to distinguish the flow directions even if the area near the valve is enlarged as shown in Figure 5.60. Therefore, the size of vector arrows becomes large by the following steps.

**Command**

Utility Menu → PlotCtrls → Style → Vector Arrow Scaling

The window Vector Arrow Scaling opens (Figure 5.61).

(1) Input \( A \) 5 to VRATIO box. This means that the length of arrows becomes five times larger as long as those in Figure 5.60. Click \( B \) OK button. Then the detailed flow view near the valve is displayed as shown in Figure 5.62.

In ANSYS, contour intervals can be changed by performing the following steps.

**Command**

Utility Menu → PlotCtrls → Style → Contours → Uniform → Contours

The window Uniform Contours opens (Figure 5.63).

(1) Pick \( A \) Contour intervals – User specified. Input 0 and 0.03 to VMIN and VMAX boxes and then click \( C \) OK button. This means that the velocity arrows
5.3 Analysis of flow structure in a channel with a butterfly valve

Figure 5.60 Enlarged view near the valve in ANSYS Graphics window.

Figure 5.61 Window of Vector Arrow Scaling.
Figure 5.62 ANSYS Graphics window.

Figure 5.63 Window of Uniform Contours.
displayed on ANSYS Graphics window are colored only from 0 to 0.03 as shown in Figure 5.64.

5.3.5.4 PLOT THE CALCULATED RESULTS BY PATH OPERATION

The continuity of volume flow can be confirmed by using Path Operation command and estimating the velocity distributions at the entrance and the exit of the channel. The steps for Path Operation are described in Section 5.2.5.3.

**Command**

ANSYS Main Menu → General Postproc → Path Operations → Define Path → By Nodes

The window By Nodes opens.

(1) Check Pick and pick two end nodes of the entrance of the channel. Click OK button.

(2) The window By Nodes opens. Input the path name aa1 to Name box, the number of divisions 40 to nDiv box, and click OK button. The window PATH Command appears and then close this window by clicking X mark at the upper right end.
(3) Click **By Nodes** in **ANSYS Main Menu**. Pick two end nodes of the exit of the channel. Click **OK** button. The window **By Nodes** opens. Input the path name **aa2** to **Name** box and click **OK** button.

**COMMAND**

**ANSYS Main Menu → General Postproc → Path Operations → Map onto Path**

The window **Map Result Items onto Path** opens.

(1) Input **aa** to **Lab** box and select [B] **DOF solution** and **Velocity VX**. Then click **OK** button.

(2) Input **aa2** to **Lab** box and select [B] **DOF solution** and **Velocity VX**. Then click **OK** button.

**COMMAND**

**ANSYS Main Menu → General Postproc → Path Operations → Plot Path Item → On Graph**

The window **Plot of Path Items on Graph** opens.

(1) Select **AA** in **Lab1-6** box and click **OK** button. Then the calculated result for the defined path **AA2** appears on **ANSYS Graphics** window as shown in Figure 5.65.

---

**Figure 5.65**  **ANSYS Graphics** window.
For the path of AA1:

**COMMAND** | ANSYS Main Menu → General Postproc → Path Operations → Recall Path

(1) Select AA1 in NAME box and click OK button.

**COMMAND** | ANSYS Main Menu → General Postproc → Path Operations → Map onto Path

(1) Input aa to Lab box and select DOF solution and Velocity VX. Then click OK button.

**COMMAND** | ANSYS Main Menu → General Postproc → Path Operations → Plot Path Item → On Graph

(1) Select AA in Lab1-6 box and click OK button. Then the calculated result for the defined path AA1 is displayed on ANSYS Graphics window as shown in Figure 5.66.

![ANSYS Graphics window](image)

**Figure 5.66** ANSYS Graphics window.

The values of flow velocities of the defined paths can be listed by the following commands.
The window **List of Path Items** opens.

(1) Select **AA** in **Lab1-6** box and click **OK** button. Then the list for the defined path appears. When the path is changed, use **Recall Path** command.
Chapter 6

Application of ANSYS to Thermo Mechanics

Chapter outline

6.1 General characteristic of heat transfer problems 263
6.2 Heat transfer through two walls 265
6.3 Steady-state thermal analysis of a pipe intersection 285
6.4 Heat dissipation through ribbed surface 312

6.1 General characteristic of heat transfer problems

The transfer of heat is normally from a high-temperature object to a lower-temperature object. Heat transfer changes the internal energy of both systems involved according to the first law of thermodynamics.

Heat may be defined as energy in transit. An object does not possess “heat”; the appropriate term for the microscopic energy in an object is internal energy. The internal energy may be increased by transferring energy to the object from a higher-temperature (hotter) object – this is called heating.

A convenient definition of temperature is that it is a measure of the average translation kinetic energy associated with the disordered microscopic motion of atoms and molecules. The flow of heat is from a high-temperature region toward a lower-temperature region. The details of the relationship to molecular motion are dealt with by the kinetic theory. The temperature defined from kinetic theory is called
the kinetic temperature. Temperature is not directly proportional to internal energy since temperature measures only the kinetic energy part of the internal energy, so two objects with the same temperature do not, in general, have the same internal energy.

Internal energy is defined as the energy associated with the random, disordered motion of molecules. It is separated in scale from the macroscopic ordered energy associated with moving objects. It also refers to the invisible microscopic energy on the atomic and molecular scale. For an ideal monoatomic gas, this is just the translational kinetic energy of the linear motion of the “hard sphere” type atoms, and the behavior of the system is well described by the kinetic theory. However, for polyatomic gases there is rotational and vibrational kinetic energy as well. Then in liquids and solids there is potential energy associated with the intermolecular attractive forces.

Heat transfer by means of molecular agitation within a material without any motion of the material as a whole is called conduction. If one end of a metal rod is at a higher temperature, then energy will be transferred down the rod toward the colder end because the higher speed particles will collide with the slower ones with a net transfer of energy to the slower ones. For heat transfer between two plane surfaces, such as heat loss through the wall of a house, the rate of conduction could be estimated from,

\[
\frac{Q}{t} = \frac{\kappa A(T_{\text{hot}} - T_{\text{cold}})}{d}
\]

where the left-hand side concerns rate of conduction heat transfer; \(\kappa\) is the thermal conductivity of the barrier; \(A\) is the area through which heat transfer takes place; \(T\) is the temperature; and \(d\) is the thickness of barrier.

Another mechanism for heat transfer is convection. Heat transfer by mass motion of a fluid such as air or water when the heated fluid is caused to move away from the source of heat, carrying energy with it is called convection. Convection above a hot surface occurs because hot air expands, becomes less dense, and rises. Convection can also lead to circulation in a liquid, as in the heating of a pot of water over a flame. Heated water expands and becomes more buoyant. Cooler, more dense water near the surface descends and patterns of circulation can be formed.

Radiation is heat transfer by the emission of electromagnetic waves which carry energy away from the emitting object. For ordinary temperatures (less than red hot), the radiation is in the infrared region of the electromagnetic spectrum. The relationship governing radiation from hot objects is called the Stefan–Boltzmann law:

\[
P = e\sigma A(T^4 - T_c^4)
\]

where \(P\) is the net radiated power; \(A\) is the radiating area; \(\sigma\) is the Stefan’s constant; \(e\) is the emissivity coefficient; \(T\) is the temperature of radiator; \(T_c\) is the temperature of surroundings.
6.2 Heat transfer through two walls

6.2.1 Problem description

A furnace with dimensions of its cross-section specified in Figure 6.1 is constructed from two materials. The inner wall is made of concrete with a thermal conductivity, \( k_c = 0.01 \text{ W/m K} \). The outer wall is constructed from bricks with a thermal conductivity, \( k_b = 0.0057 \text{ W/m K} \). The temperature within the furnace is 673 K and the convection heat transfer coefficient \( k_1 = 0.208 \text{ W/m}^2\text{ K} \). The outside wall of the furnace is exposed to the surrounding air, which is at 253 K and the corresponding convection heat transfer coefficient, \( k_2 = 0.068 \text{ W/m}^2\text{ K} \).

**Figure 6.1** Cross-section of the furnace.

Determine the temperature distribution within the concrete and brick walls under steady-state conditions. Also determine the heat fluxes through each wall.

This is a two-dimensional (2D) problem and will be modeled using graphical user interface (GUI) facilities.

6.2.2 Construction of the model

From ANSYS Main Menu select Preferences. This frame is shown in Figure 6.2.
Depending on the nature of analysis to be attempted an appropriate analysis type should be selected. In the problem considered here [A] Thermal was selected as shown in Figure 6.2.

From ANSYS Main Menu select Preprocessor → Element Type → Add/Edit/Delete. The frame shown in Figure 6.3 appears.
Clicking [A] **Add** button activates a new set of options which are shown in Figure 6.4.

---

**Figure 6.4** Library of Element Types.

Figure 6.4 shows that for the problem considered the following were selected: [A] **Thermal Mass → Solid** and [B] **4node 55**. This element is referred to as Type 1 PLANE55.

From **ANSYS Main Menu** select **Preprocessor → Material Props → Material Models**. Figure 6.5 shows the resulting frame.

---

**Figure 6.5** Define Material Model Behavior.

From the right-hand column select [A] **Thermal → Conductivity → Isotropic**. As a result, the frame shown in Figure 6.6 appears. Thermal conductivity [A] $K_{XX} = 0.01 \text{ W/m K}$, was selected as shown in Figure 6.6.
Chapter 6 Application of ANSYS to thermo mechanics

Figure 6.6 Conductivity for Material Number 1.

Clicking [B] OK button ends input into Material Number 1. In the frame shown in Figure 6.7 select from the top menu [A] Material → New Model. Database for Material Number 2 is created.

Figure 6.7 Define Material Model Behavior.

As in the case of Material Number 1 select [B] Thermal → Conductivity → Isotropic. The frame shown in Figure 6.8 appears. Enter [A] \( K_{XX} = 0.0057 \) W/m K and click [B] OK button as shown in Figure 6.8.

In order to have created primitives numbered from ANSYS Utility Menu select PlotCtrls → Numbering and check the box area numbers.

From ANSYS Main Menu select Preprocessor → Modelling → Create → Areas → Rectangle → By Dimensions. Figure 6.9 shows the resulting frame.
6.2 Heat transfer through two walls

Figure 6.8 Conductivity for Material Number 2.

Figure 6.9 Create Rectangle by Dimensions.

Inputs [A] \( X1 = -15 \); [B] \( X2 = 15 \); [C] \( Y1 = -15 \); and [D] \( Y2 = 15 \) to create outer wall perimeter are shown in Figure 6.9. Next, the perimeter of inner wall is created in the same way. Figure 6.10 shows the frame with appropriate entries.

Figure 6.10 Create Rectangle by Dimensions.
In order to generate the brick wall area of the chimney, subtract the two areas which have been created. From ANSYS Main Menu select Preprocessor → Modelling → Operate → Booleans → Subtract → Areas. Figure 6.11 shows the resulting frame.

![Subtract Areas](image)

Figure 6.11  Subtract Areas.

Figure 6.12  Brick wall outline.

First select the larger area (outer brick wall) and click [A] OK button in the frame of Figure 6.11. Next, select the smaller area (inner concrete wall) and click [A] OK button. The smaller area is subtracted from the larger area and the outer brick wall is produced. It is shown in Figure 6.12.

Using a similar approach, the inner concrete wall is constructed. From ANSYS Main Menu select Preprocessor → Modelling → Create → Areas → Rectangle → By Dimensions. Figure 6.13 shows the resulting frame with inputs: [A] X1 = −6; [B] X2 = 6; [C] Y1 = −6, and [D] Y2 = 6. Pressing [E] OK button creates the rectangular area A1.

Again, from ANSYS Main Menu select Preprocessor → Modelling → Create → Areas → Rectangle → By Dimensions. Frame with inputs: [A] X1 = −5; [B] X2 = 5; [C] Y1 = −5, and [D] Y2 = 5 is shown in Figure 6.14.

Clicking [E] OK button creates the rectangular area A2. As before, to create the concrete area of the furnace, subtract area A2 from area A1. From ANSYS Main Menu
select **Preprocessor → Modelling → Operate → Booleans → Subtract → Areas**. The frame shown in Figure 6.11 appears. Select area A1 first and click [A] OK button. Next select area A2 and click [A] OK button. As a result, the inner concrete wall is created. This is shown in Figure 6.15.
From ANSYS Main Menu select Preprocessor $\rightarrow$ Meshing $\rightarrow$ Size Cntrls $\rightarrow$ ManualSize $\rightarrow$ Global $\rightarrow$ Size. As a result of this selection, the frame shown in Figure 6.16 appears.

Figure 6.16  Global Element Sizes.

Figure 6.17  Glue Areas.

Figure 6.18  Area Attributes.
6.2 Heat transfer through two walls

Input for the element edge length [A] \( \text{SIZE} = 0.5 \) and click [B] OK button.

Because the outer brick wall and the inner concrete wall were created as separate entities, therefore, it is necessary to “glue” them together so that they share lines along their common boundaries. From ANSYS Main Menu select Preprocessor → Modelling → Operate → Boolean → Glue → Areas. The frame shown in Figure 6.17 appears.

Select [A] Pick All option in the frame of Figure 6.17 to glue the outer and inner wall areas. Before meshing is done, it is necessary to specify material numbers for the concrete and the brick walls.

From ANSYS Main Menu select Preprocessor → Meshing → Mesh Attributes → Picked Areas. The frame shown in Figure 6.18 is created.

Select first the concrete wall area and click [A] OK button in the frame of Figure 6.18. A new frame is produced as shown in Figure 6.19.

Material Number 1 is assigned to the concrete inner wall as shown in Figure 6.19.

Next, assign Material Number 2 to the brick outer wall following the procedure outlined above that is recall frame of Figure 6.19 and select brick outer wall. Figure 6.20 shows the frame with appropriate entry.

Now meshing of both areas can be carried out. From ANSYS Main Menu select Preprocessor → Meshing → Mesh → Areas → Free. The frame shown in Figure 6.21 appears.

Select [A] Pick All option shown in Figure 6.21 to mesh both areas.

In order to see both areas meshed, from Utility Menu select PlotCtrls → Numbering. In the appearing frame, shown in Figure 6.22, select [A] Material numbers and click [B] OK button.

As a result of that, both walls with mesh are displayed (see Figure 6.23).
Chapter 6  Application of ANSYS to thermo mechanics

Figure 6.20  Area Attributes.

Figure 6.21  Mesh Areas.
6.2 Heat transfer through two walls

Figure 6.22  Plot Numbering Controls.

Figure 6.23  Outer and inner walls of the furnace meshed.
6.2.3 Solution

Before a solution can be run boundary conditions have to be applied. From ANSYS Main Menu select Solution → Define Loads → Apply → Thermal → Convection → On Lines. This selection produces the frame shown in Figure 6.24.

![Figure 6.24 Apply CONV on Lines.](image)

First pick the convective lines (facing inside the furnace) of the concrete wall and press [A] OK button. The frame shown in Figure 6.25 is created.

As seen in Figure 6.25, the following selections were made: [A] Film coefficient $= 0.208 \text{ W/m}^2 \text{ K}$ and [B] Bulk temperature $= 673 \text{ K}$, as specified for the concrete wall in the problem formulation.

Again from ANSYS Main Menu select Solution → Define Loads → Apply → Thermal → Convection → On Lines. The frame shown in Figure 6.24 appears. This time pick the exterior lines of the brick wall and press [A] OK button. The frame shown in Figure 6.26 appears.

For the outer brick wall, the following selections were made (see the frame in Figure 6.26): [A] Film coefficient $= 0.068 \text{ W/m}^2 \text{ K}$ and [B] Bulk temperature $= 253 \text{ K}$ as specified for the brick wall in the problem formulation.
6.2 Heat transfer through two walls

Apply CONV on lines

Figure 6.25 Apply CONV on lines (the inner wall).

Apply CONV on lines

Figure 6.26 Apply CONV on lines (the outer wall).
Finally, to see the applied convective boundary conditions from Utility Menu select **PlotCtrls \(\rightarrow\) Symbols.** The frame shown in Figure 6.27 appears.

In the frame shown in Figure 6.27 select \([A]\) **Show pres and convect as = Arrows** and click \([B]\) OK button.

---

**Figure 6.27** Symbols.
From **Utility Menu** select **Plot → Lines** to produce an image shown in Figure 6.28.

![Figure 6.28](image)

**Figure 6.28**  Applied convective boundary conditions.

To solve the problem select from **ANSYS Main Menu, Solution → Solve → Current LS**. Two frames appear. One gives summary of solution options. After checking correctness of the options, it should be closed using the menu at the top of the frame. The other frame is shown in Figure 6.29. Clicking [A] **OK** button initiates solution process.

![Solve Current Load Step](image)

**Figure 6.29**  Solve Current Load Step.
6.2.4 **Postprocessing**

The end of a successful solution process is denoted by the message “solution is done.” The postprocessing phase can be started. First it is necessary to obtain information about temperatures and heat fluxes across the furnace's walls.

From **ANSYS Main Menu** select **General Postproc → Plot Results → Contour Plot → Nodal Solu**. The frame shown in Figure 6.30 appears.

![Contour Nodal Solution Data](image)

**Figure 6.30** Contour Nodal Solution Data.

Selections made are shown in Figure 6.30. Clicking [A] **OK** button results in the graph shown in Figure 6.31.

In order to observe the heat flow across the walls the following command should be issued: **General Postproc → Plot Results → Vector Plot → Predefined**. This produces the frame shown in Figure 6.32.

Clicking [A] **OK** button produces a graph shown in Figure 6.33.

In order to observe temperature variations across the walls, it is necessary to define the path along which the variations are going to be determined. From **Utility Menu** select **Plot → Areas**. Next, from **ANSYS Main Menu** select **General Postproc → Path Operations → Define Path → On Working Plane**. The resulting frame is shown in Figure 6.34.

By activating [A] **Arbitrary path** button and clicking [B] **OK**, another frame is produced and is shown in Figure 6.35.
6.2 Heat transfer through two walls

Figure 6.31 Temperature distribution in the furnace as a contour plot.

Figure 6.32 Vector Plot of Predefined Vectors.
Figure 6.33  Heat flow across the wall plotted as vectors.

Figure 6.34  On Working Plane (definition of the path).
6.2 Heat transfer through two walls

Two points should be selected by clicking on the inner line of the concrete wall and moving in Y direction at the right angle by clicking on the outer line of the brick wall. As a result of clicking [A] OK button frame shown in Figure 6.36 appears.

In the box [A] Define Path Name, write AB and click [B] OK button.

From ANSYS Main Menu select General Postproc → Path Operations → Map onto Path. The frame shown in Figure 6.37 appears.

In Figure 6.37, the following selections are made: [A] Flux & gradient and [B] Thermal grad TGX. By repeating steps described above, recall the frame shown in Figure 6.37. This time select the following: [A] Flux & gradient and [B] Thermal grad TGY. Finally, recall the frame shown in Figure 6.37 and select: [A] Flux & gradient and [B] Thermal grad TGSUM as shown in Figure 6.38 and click [C] OK button.

From ANSYS Main Menu select General Postproc → Path Operations → Plot Path Item → On Graph. The frame shown in Figure 6.39 appears.

The selections made [A] are highlighted in Figure 6.39. Pressing [B] OK button results in a graph shown in Figure 6.40.
Figure 6.36  On Working Plane (path name: AB).

Figure 6.37  Map Result Items onto Path (AB path).
6.3 Steady-state thermal analysis of a pipe intersection

6.3.1 Description of the problem

A cylindrical tank is penetrated radially by a small pipe at a point on its axis remote from the ends of the tank, as shown in Figure 6.41.
The inside of the tank is exposed to a fluid with temperature of 232°C. The pipe experiences a steady flow of fluid with temperature of 38°C, and the two flow regimes are isolated from each other by means of a thin tube. The convection (film) coefficient in the pipe varies with the metal temperature and is thus expressed as a
material property. The objective is to determine the temperature distribution at the pipe–tank junction.

The following data describing the problem are given:

- Inside diameter of the pipe = 8 mm
- Outside diameter of the pipe = 10 mm
- Inside diameter of the tank = 26 mm
- Outside diameter of the tank = 30 mm
- Inside bulk fluid temperature, tank = 232°C
- Inside convection coefficient, tank = 4.92 W/m²°C
- Inside bulk fluid temperature, pipe = 38°C
- Inside convection coefficient (pipe) varies from about 19.68 to 39.36 W/m²°C, depending on temperature.

Table 6.1 provides information about variation of the thermal parameters with temperature.

<table>
<thead>
<tr>
<th>Temperature [°C]</th>
<th>21</th>
<th>93</th>
<th>149</th>
<th>204</th>
<th>260</th>
</tr>
</thead>
<tbody>
<tr>
<td>Convection coefficient [W/m²°C]</td>
<td>41.918</td>
<td>39.852</td>
<td>34.637</td>
<td>27.06</td>
<td>21.746</td>
</tr>
<tr>
<td>Density [kg/m³]</td>
<td>7889</td>
<td>7889</td>
<td>7889</td>
<td>7889</td>
<td>7889</td>
</tr>
<tr>
<td>Conductivity [J/s m°C]</td>
<td>0.2505</td>
<td>0.267</td>
<td>0.2805</td>
<td>0.294</td>
<td>0.3069</td>
</tr>
</tbody>
</table>

The assumption is made that the quarter symmetry is applicable and that, at the terminus of the model (longitudinal and circumferential cuts in the tank), there is sufficient attenuation of the pipe effects such that these edges can be held at 232°C.

The solid model is constructed by intersecting the tank with the pipe and then removing the internal part of the pipe using Boolean operation.

Boundary temperatures along with the convection coefficients and bulk fluid temperatures are dealt with in the solution phase, after which a static solution is executed.

Temperature contours and thermal flux displays are obtained in postprocessing.

Details of steps taken to create the model of pipe intersecting with tank are outlined below.
6.3.2 Preparation for Model Building

From ANSYS Main Menu select Preferences. This frame is shown in Figure 6.42.

![Preferences for GUI Filtering](image)

Figure 6.42 Preferences: Thermal.

Depending on the nature of analysis to be attempted an appropriate analysis type should be selected. In the problem considered here [A] Thermal was selected as shown in Figure 6.42.

From ANSYS Main Menu select Preprocessor and then Element Type and Add/Edit/Delete. The frame shown in Figure 6.43 appears.

Clicking [A] Add button activates a new set of options which are shown in Figure 6.44.

Figure 6.44 indicates that for the problem considered here the following was selected: [A] Thermal Mass → Solid and [B] 20node 90.

From ANSYS Main Menu select Material Props and then Material Models. Figure 6.45 shows the resulting frame.

From the options listed on the right hand select [A] Thermal as shown in Figure 6.45.

Next select [B] Conductivity, Isotropic. The frame shown in Figure 6.46 appears.
6.3 Steady-state thermal analysis of a pipe intersection

Then, using conductivity versus temperature values, listed in Table 6.1, appropriate figures should be typed in as shown in Figure 6.46.

By selecting [C] Specific Heat option on the right-hand column (see Figure 6.45), the frame shown in Figure 6.47 is produced.

Appropriate values of specific heat versus temperature, taken from Table 6.1, are typed as shown in Figure 6.47.

The next material property to be defined is density. According to Table 6.1, density is constant for all temperatures used. Therefore, selecting [D] Density...
from the right-hand column (see Figure 6.45), results in the frame shown in Figure 6.48.

Density of 7888.8 kg/m$^3$ is typed in the box shown in Figure 6.48.

All the above properties were used to characterize Material Number 1. Convection or film coefficient is another important parameter characterizing the system being analyzed. However, it is not a property belonging to Material Number 1 (material of the tank and pipe) but to a thin film formed by the liquid on solid surfaces. It is a different entity and, therefore, is called Material Number 2.
6.3 Steady-state thermal analysis of a pipe intersection

Selecting [E] Convection or Film Coef. (see Figure 6.45) results in the frame shown in Figure 6.49. Appropriate values of film coefficient for various temperatures, taken from Table 6.1, are introduced as shown in Figure 6.49.

6.3.3 Construction of the model

The entire model of the pipe intersecting with the tank is constructed using one of the three-dimensional (3D) primitive shapes, that is cylindrical. Only one-quarter of
the tank–pipe assembly will be sufficient to use in the analysis. From ANSYS Main Menu select Preprocessor → Modelling → Create → Volumes → Cylinder → By Dimensions. Figure 6.50 shows the resulting frame.

In Figure 6.50, as shown, the following inputs are made: [A] RAD1 = 1.5 cm; [B] RAD2 = 1.3 cm; [C] Z1 = 0; [D] Z2 = 2 cm; [E] THETA1 = 0; [F] THETA2 = 90.

As the pipe axis is at right angle to the cylinder axis, therefore it is necessary to rotate the working plane (WP) to the pipe axis by 90°. This is done by selecting from Utility Menu WorkPlane → Offset WP by Increments. The resulting frame is shown in Figure 6.51.
6.3 Steady-state thermal analysis of a pipe intersection

In Figure 6.51, the input is shown as $[A] \ XY = 0; \ YZ = -90$ and the $ZX$ is left unchanged from default value. Next, from ANSYS Main Menu select Preprocessor $\rightarrow$ Modelling $\rightarrow$ Create $\rightarrow$ Volumes $\rightarrow$ Cylinder $\rightarrow$ By Dimensions. Figure 6.52 shows the resulting frame.
In Figure 6.52, as shown, the following inputs are made: [A] RAD1 = 0.5 cm; [B] RAD2 = 0.4 cm; [C] Z1 = 0; [D] Z2 = 2 cm; [E] THETA1 = 0; [F] THETA2 = −90.

After that the WP should be set to the default setting by inputting in Figure 6.51 YZ = 90 this time. As the cylinder and the pipe are separate entities, it is necessary to overlap them in order to make the one component. From ANSYS Main Menu select Preprocessor → Modelling → Create → Operate → Booleans → Overlap → Volumes. The frame shown in Figure 6.53 is created.

Pick both elements, that is cylinder and pipe, and press [A] OK button to execute the selection. Next activate volume numbering which will be of help when carrying out further operations on volumes. This is done by selecting from Utility Menu PlotCtrls → Numbering and checking, in the resulting frame, VOLU option.

Finally, 3D view of the model should be set by selecting from Utility Menu the following: PlotCtrls → View Settings. The resulting frame is shown in Figure 6.54.

The following inputs should be made (see Figure 6.54): [A] XV = −3; [B] YV = −1; [C] ZV = 1 in order to plot the model as shown in Figure 6.55. However, this is not the only possible view of the model and any other preference may be chosen.
6.3 Steady-state thermal analysis of a pipe intersection

Certain volumes of the models, shown in Figure 6.55, are redundant and should be deleted. From ANSYS Main Menu select Preprocessor → Modelling → Delete → Volume and Below. Figure 6.56 shows the resulting frame.

Volumes V4 and V3 (a corner of the cylinder) should be picked and [A] OK button pressed to implement the selection. After the delete operation, the model looks like that shown in Figure 6.57.
Chapter 6  Application of ANSYS to thermo mechanics

**Figure 6.56** Delete Volume and Below.

**Figure 6.57** Quarter symmetry model of the tank–pipe intersection.
Finally, volumes V5, V6, and V7 should be added in order to create a single volume required for further analysis. From **ANSYS Main Menu** select **Preprocessor → Modelling → Operate → Booleans → Add → Volumes**. The resulting frame asks for picking volumes to be added. Pick all three volumes, that is V5, V6, and V7, and click **OK** button to implement the operation. Figure 6.58 shows the model of the pipe intersecting the cylinder as one volume V1.

![Quarter symmetry model of the tank–pipe intersection represented by a single volume V1.](image)

**Figure 6.58** Quarter symmetry model of the tank–pipe intersection represented by a single volume V1.

Meshing of the model usually begins with setting size of elements to be used. From **ANSYS Main Menu** select **Meshing → Size Cntrls → SmartSize → Basic**. A frame shown in Figure 6.59 appears.

![Basic SmartSize Settings](image)

**Figure 6.59** Basic SmartSize Settings.
For the case considered, [A] Size Level – 1 (fine) was selected as shown in Figure 6.59. Clicking [B] OK button implements the selection. Next, from ANSYS Main Menu select Mesh → Volumes → Free. The frame shown in Figure 6.60 appears. Select the volume to be meshed and click [A] OK button.

The resulting network of elements is shown in Figure 6.61.

Figure 6.60  Mesh Volumes frame.

Figure 6.61  Meshed quarter symmetry model of the tank–pipe intersection.

6.3.4 Solution

The meshing operation ends the model construction and the Preprocessor stage. The solution stage can now be started. From ANSYS Main Menu select Solution → Analysis Type → New Analysis. Figure 6.62 shows the resulting frame.

Activate [A] Steady-State button. Next, select Solution → Analysis Type → Analysis Options. In the resulting frame, shown in Figure 6.63, select [A] Program chosen option.
6.3 Steady-state thermal analysis of a pipe intersection

In order to set starting temperature of $232^\circ C$ at all nodes select Solution $\rightarrow$ Define Loads $\rightarrow$ Apply $\rightarrow$ Thermal $\rightarrow$ Temperature $\rightarrow$ Uniform Temp. Figure 6.64 shows the resulting frame. Input [A] Uniform temperature $= 232^\circ C$ as shown in Figure 6.64.

From Utility Menu select WorkPlane $\rightarrow$ Change Active CS to $\rightarrow$ Specified Coord Sys. As a result of that the frame shown in Figure 6.65 appears.
Chapter 6  Application of ANSYS to thermo mechanics

In order to re-establish a cylindrical coordinate system with Z as the axis of rotation select [A] Coordinate system number = 1 and press [B] OK button to implement the selection.

Nodes on inner surface of the tank ought to be selected to apply surface loads to them. The surface load relevant in this case is convection load acting on all nodes located on inner surface of the tank. From Utility Menu select Select → Entities. The frame shown in Figure 6.66 appears.

From the first pull down menu select [A] Nodes, from the second pull down menu select [B] By Location. Also, activate [C] X coordinates button and enter [D] Min,Max = 1.3 (inside radius of the tank). All the four required steps are shown in Figure 6.66. When the subset of nodes on inner surface of the tank is selected then the convection load at all nodes has to be applied. From ANSYS Main Menu select Solution → Define Load → Apply → Thermal → Convection → On nodes. The resulting frame is shown in Figure 6.67.

Press [A] Pick All in order to call up another frame shown in Figure 6.68.

Inputs into the frame of Figure 6.68 are shown as: [A] Film coefficient = 4.92 and [B] Bulk temperature = 232. Both quantities are taken from Table 6.1.

From Utility Menu select Select → Entities in order to select a subset of nodes located at the far edge of the tank. The frame shown in Figure 6.69 appears.

From the first pull down menu select [A] Nodes, from the second pull down menu select [B] By Location. Also, activate [C] Z coordinates button and [D] enter Min,Max = 2 (the length of the tank in Z-direction). All the four required steps are