

ANALYSIS FOR FLUID DYNAMICS

CHAPTER OUTLINE

5.1	INTRODUCTION	215
5.2	ANALYSIS OF FLOW STRUCTURE IN A DIFFUSER	216
5.3	ANALYSIS OF FLOW STRUCTURE IN A CHANNEL WITH A BUTTERFLY VALVE	242

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.

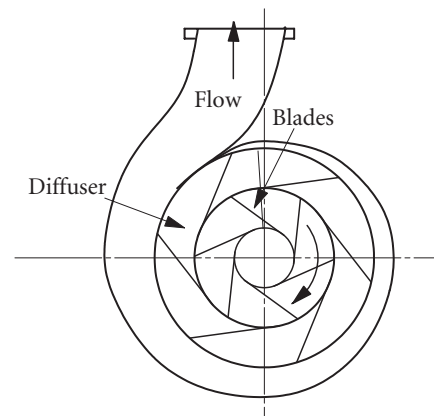


Figure 5.1 Typical machines for fluid.

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.

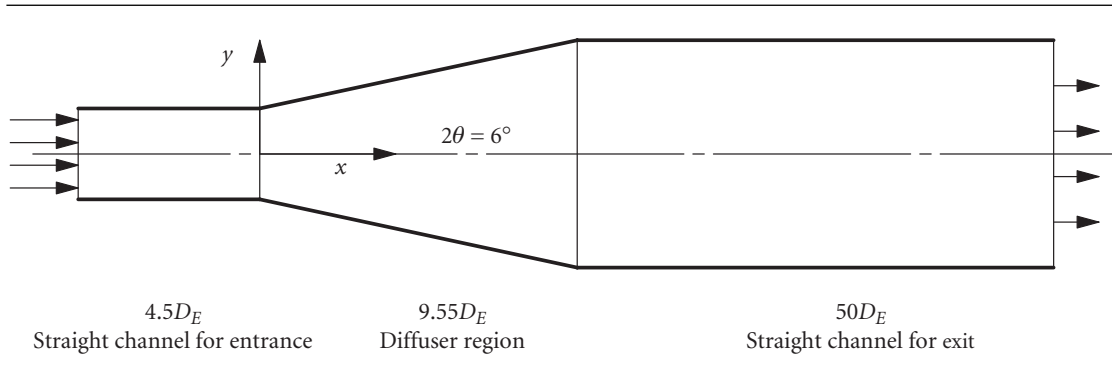


Figure 5.2 Axisymmetrical conical diffuser.

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: $4.5D_E$.
- (4) Length of diffuser region: $9.55D_E$.
- (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: 2.54×10^5 (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 → Preferences

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

- (1) Check [A] **FLOTRAN CFD** and click [B] **OK** button.

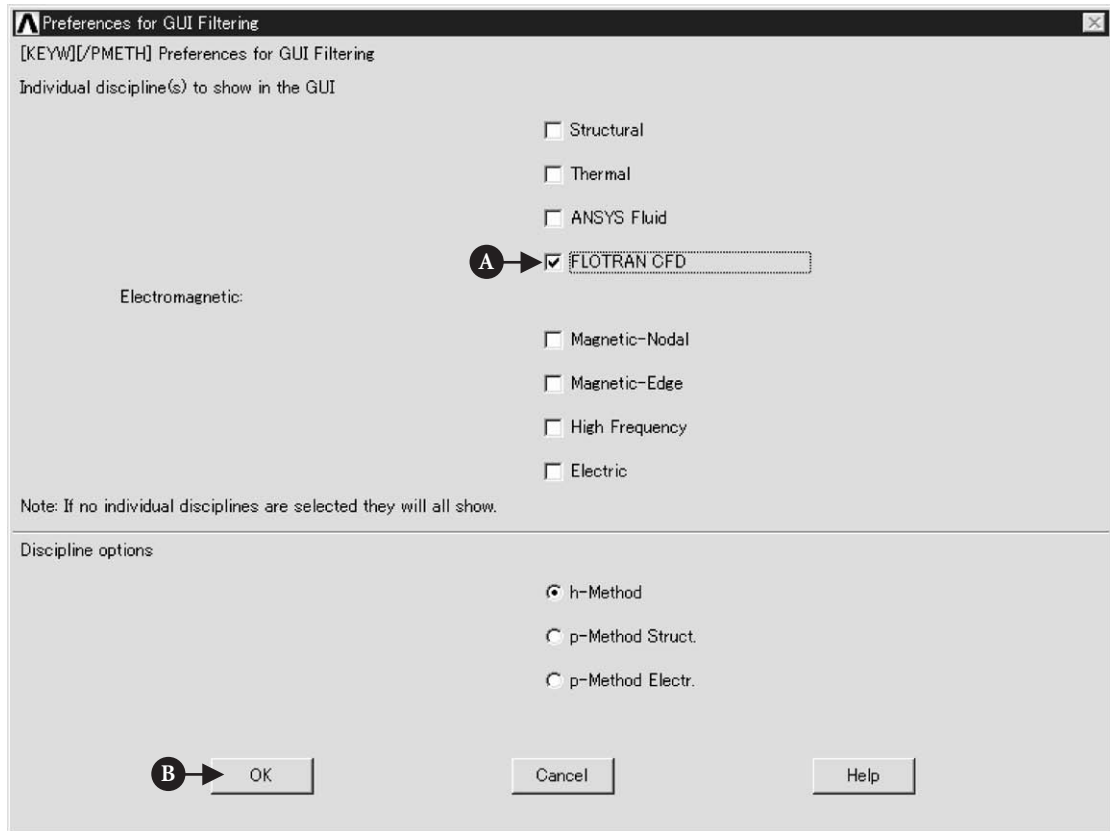


Figure 5.3 Window of **Preferences for GUI Filtering**.

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.
- (2) Select [B] **FLOTRAN CFD-2D FLOTRAN 141**.



Figure 5.4 Window of **Element Types**.

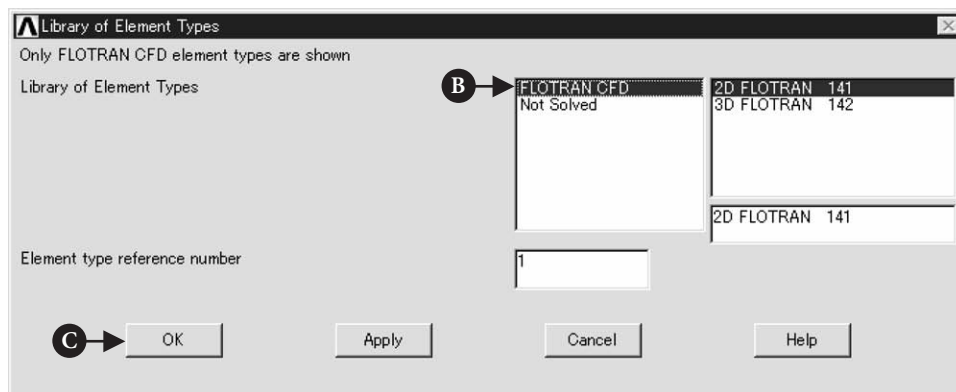


Figure 5.5 Window of **Library of Element Types**.

- (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] **Axisymm about X** in the box of **Element coordinate system** and click [F] **OK** button. Finally click [G] **Close** button in Figure 5.6.

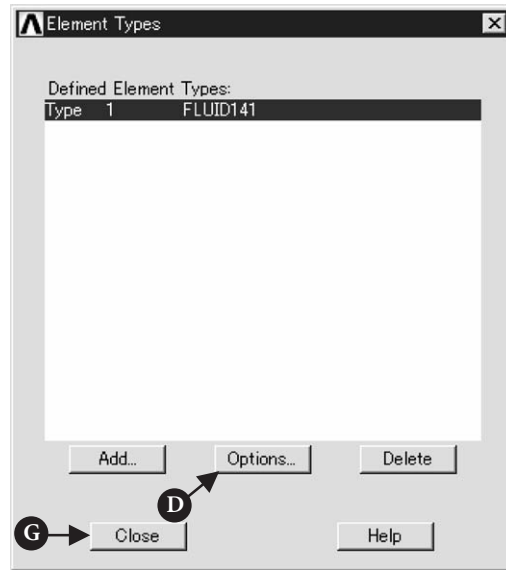


Figure 5.6 Window of **Element Types**.

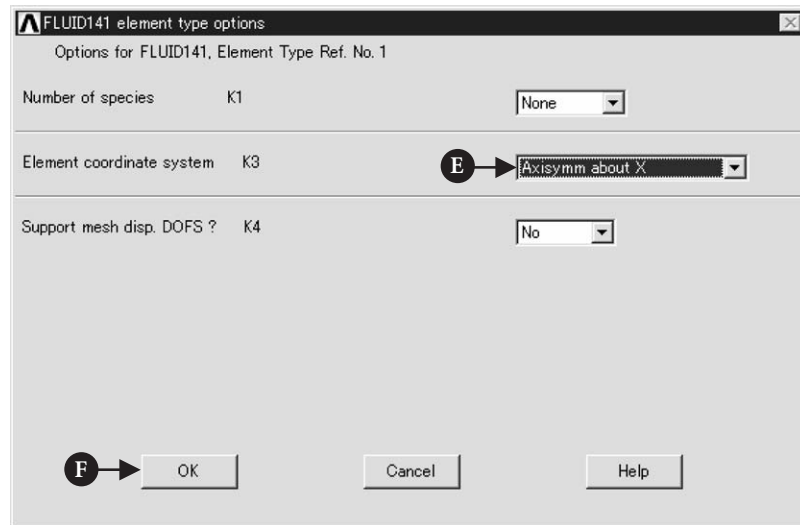


Figure 5.7 Window of **FLUID141 element type options**.

5.2.2.3 CREATE KEYPOINTS

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).

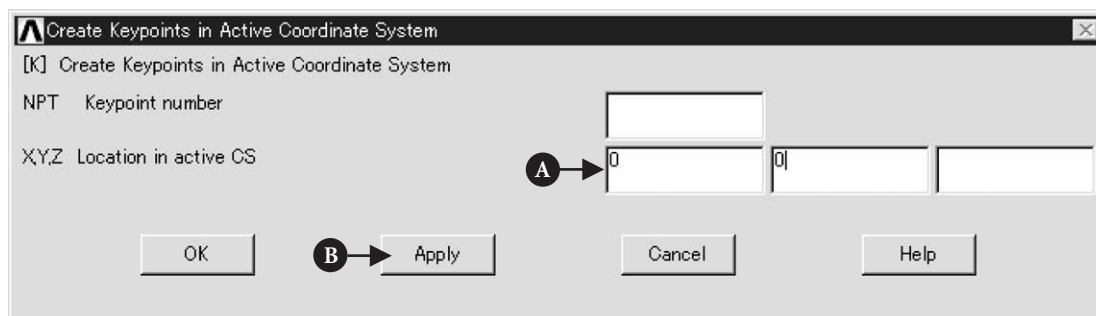


Figure 5.8 Window of **Create Keypoints in Active Coordinate System**.

- (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 5.1 Coordinates of keypoints

KP No.	X	Y
1	0	0
2	0.9	0
3	0.9	0.1
4	0	0.1
5	2.81	0
6	2.81	0.2
7	12.81	0
8	12.81	0.2

- (4) After finishing step (3), eight keypoints appear on the window as shown in Figure 5.9.

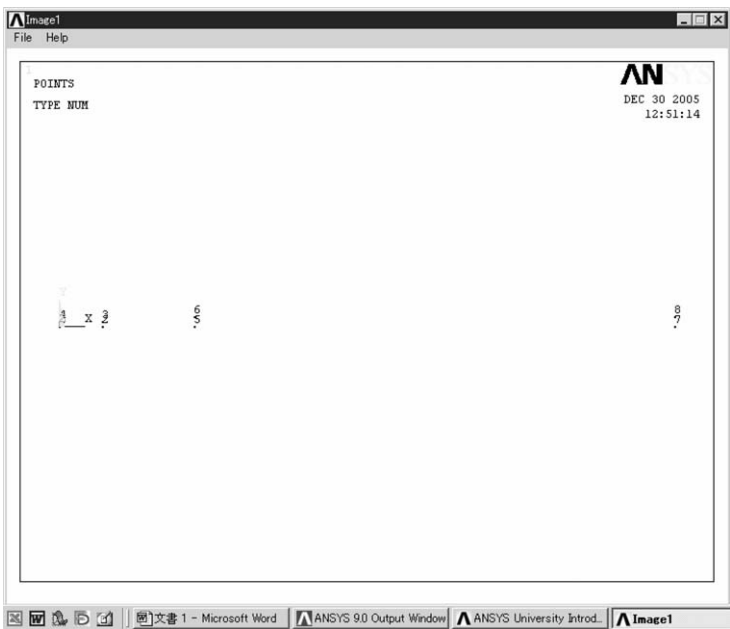


Figure 5.9 ANSYS Graphics window.

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.

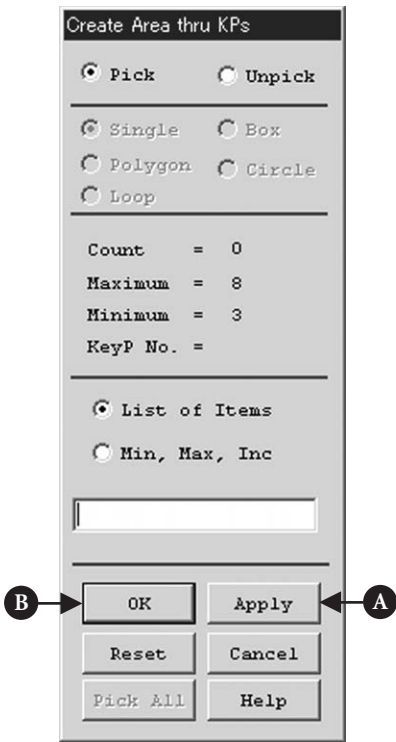


Figure 5.10 Window of Create Area thru KPs.

Table 5.2 Keypoint numbers for making areas

Area No.	KPs
1	1, 2, 3, 4
2	2, 5, 6, 3
3	5, 7, 8, 6

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.

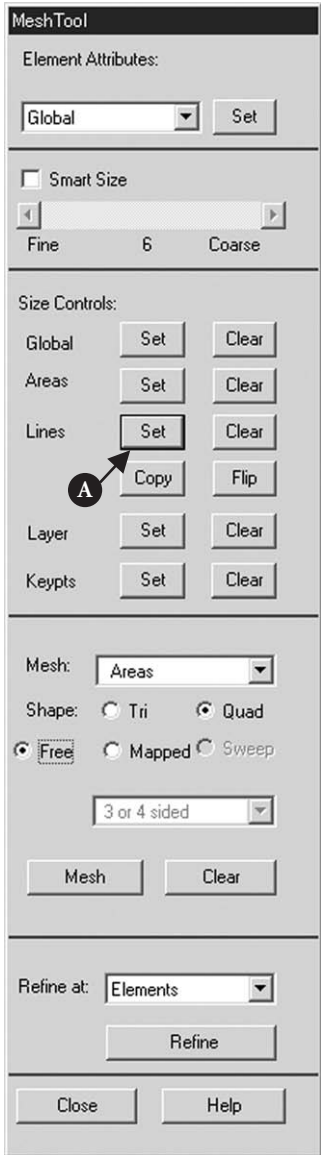


Figure 5.11 Window of **Mesh Tool**.

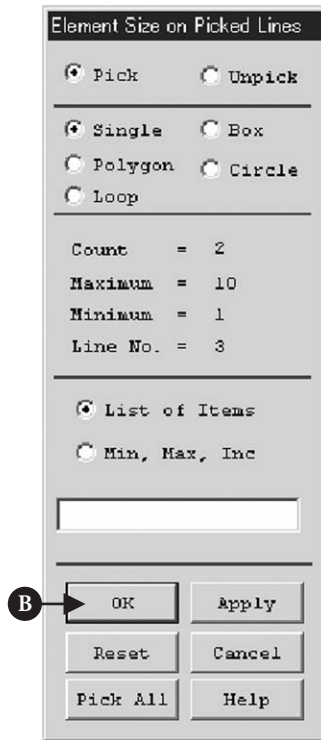


Figure 5.12 Window of Element Size on Picked Lines.

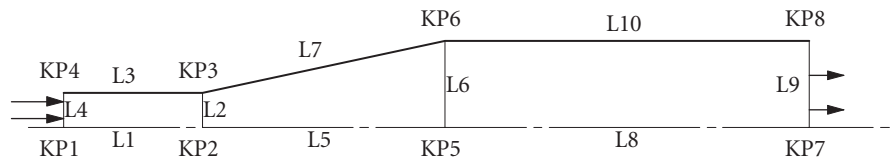


Figure 5.13 Keypoint and Line numbers for a diffuser.

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.

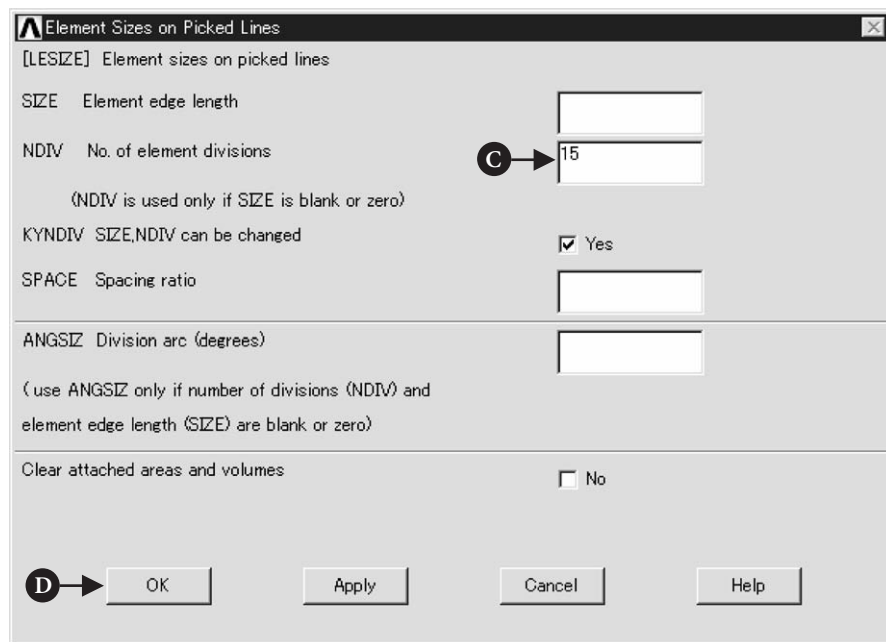


Figure 5.14 Window of **Element Sizes on Picked Lines**.

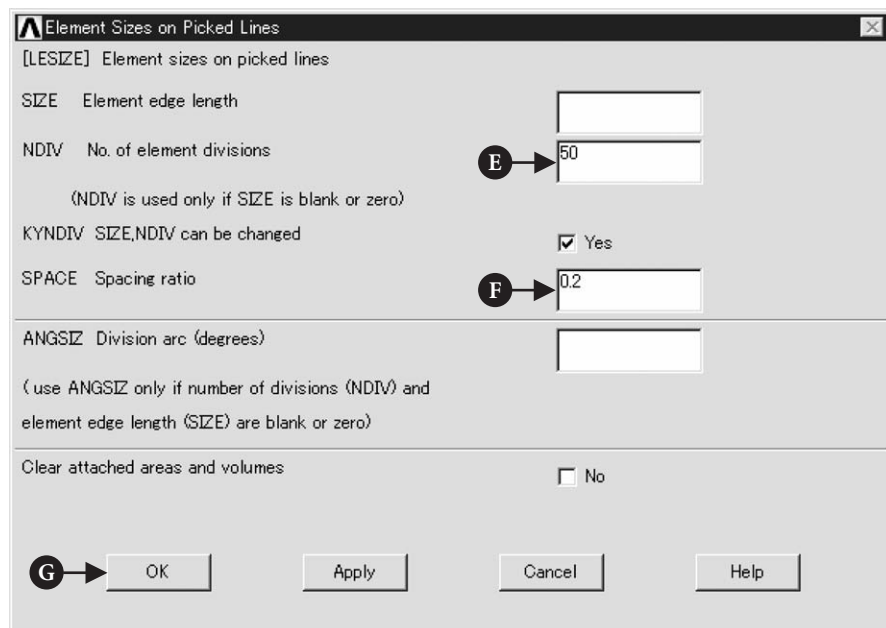
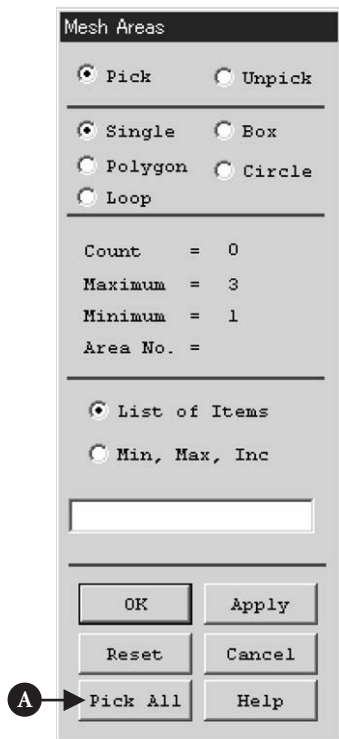
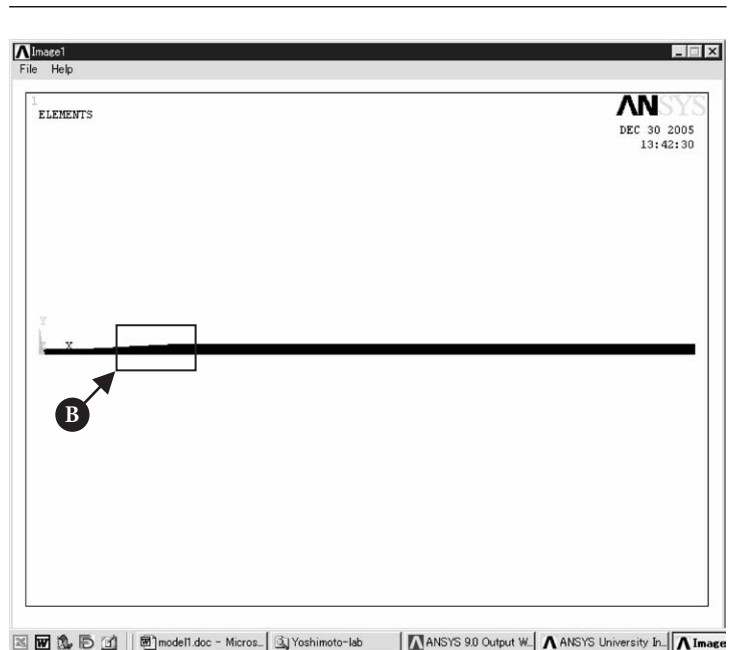


Figure 5.15 Window of **Element Sizes on Picked Lines**.

Table 5.3 Element sizes of picked lines

Line No.	NDIV	SPACE
1	15	
2	50	0.2
3	15	—
4	50	5
5	30	—
6	50	0.2
7	30	—
8	40	5
9	50	0.2
10	40	0.2

**Figure 5.16** Window of Mesh Areas.**Figure 5.17** ANSYS Graphics window.

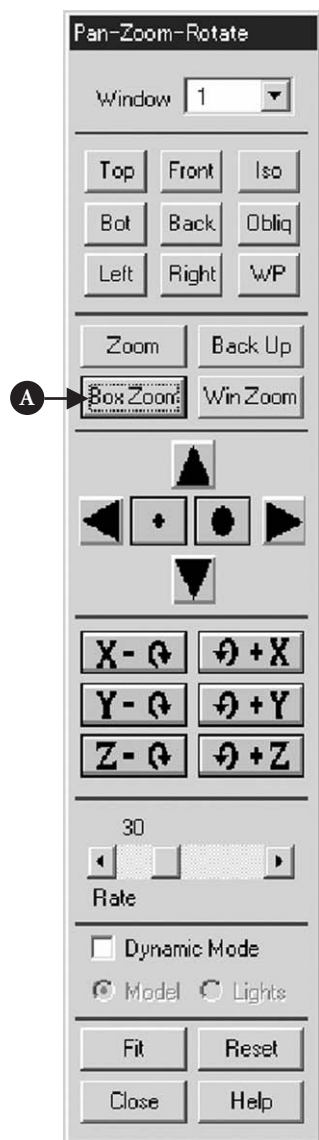


Figure 5.18 Window of Pan-Zoom-Rotate.

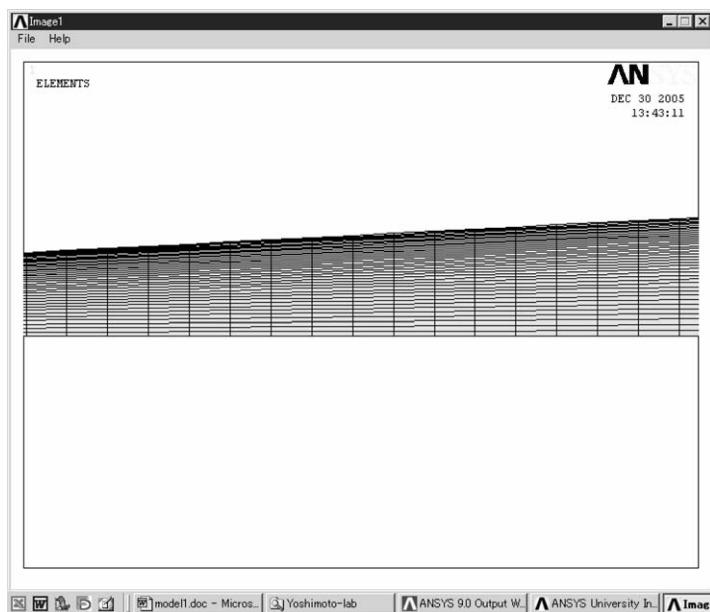


Figure 5.19 ANSYS Graphics window.

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.

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).

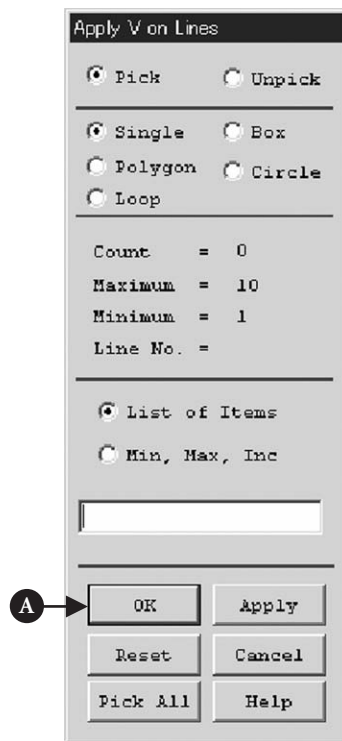


Figure 5.20 Window of **Apply V on Lines**.

- (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.

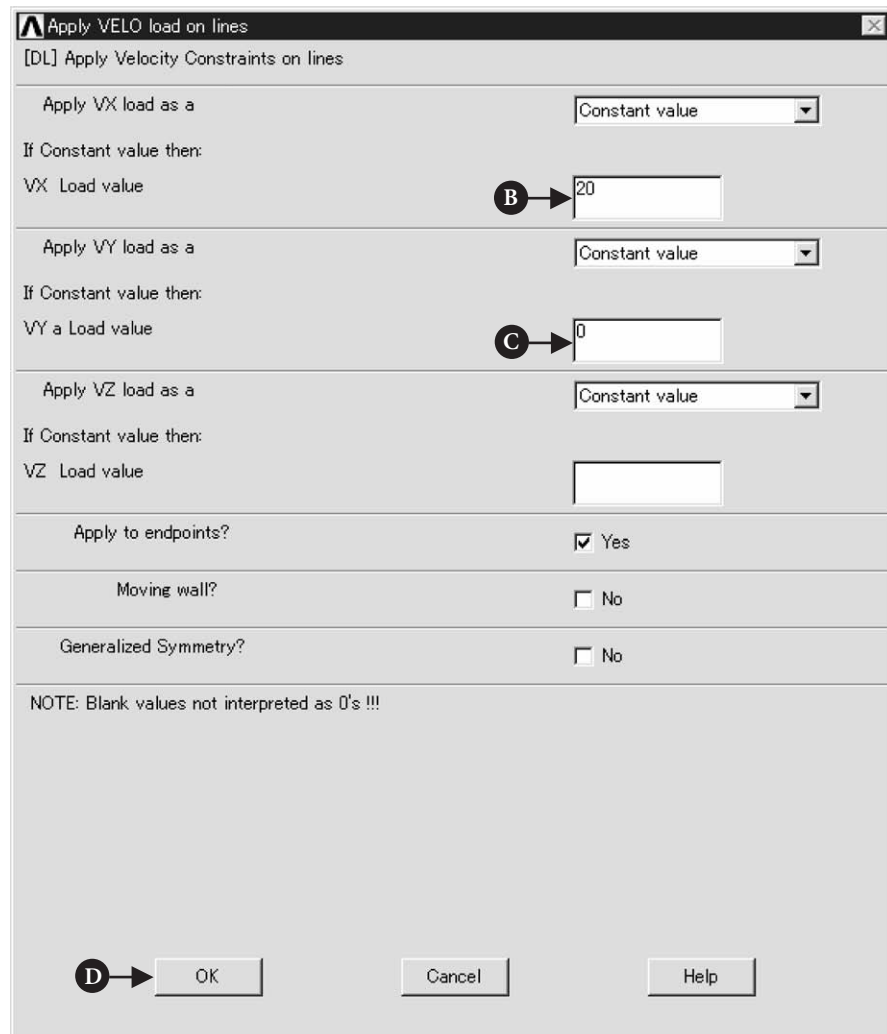


Figure 5.21 Window of **Apply VELO load on lines**.

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).

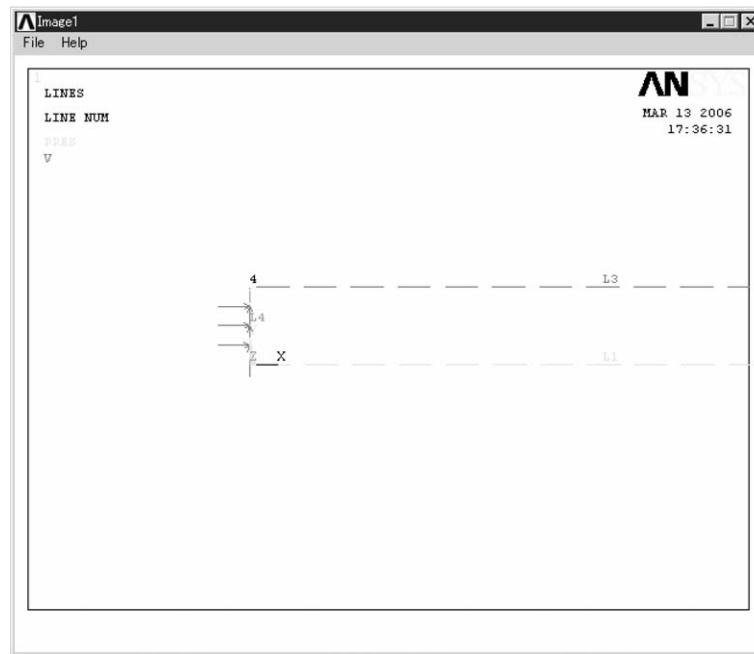


Figure 5.22 ANSYS Graphics window.

Table 5.4 Boundary conditions for all lines

Line No.	VX	VY
1	—	0
2	—	—
3	0	0
4	20	0
5	—	0
6	—	—
7	0	0
8	—	0
9	—	—
10	0	0

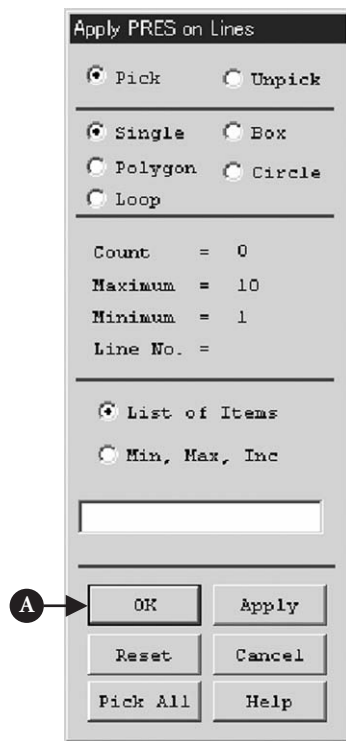


Figure 5.23 Window of Apply PRES on lines.

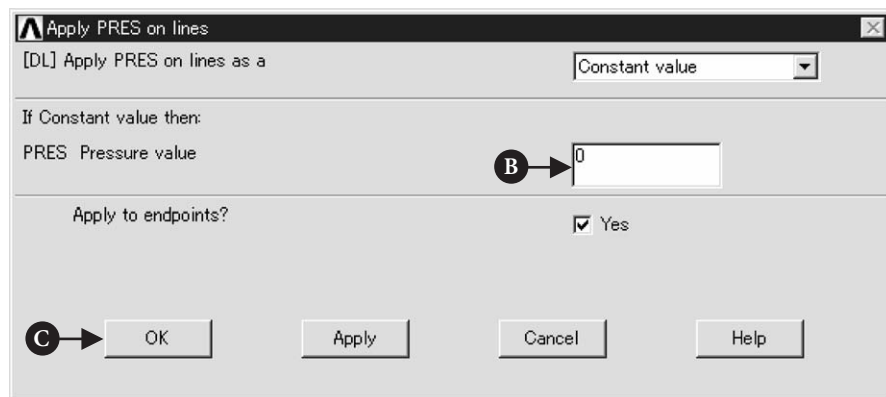


Figure 5.24 Window of Apply PRES on lines.

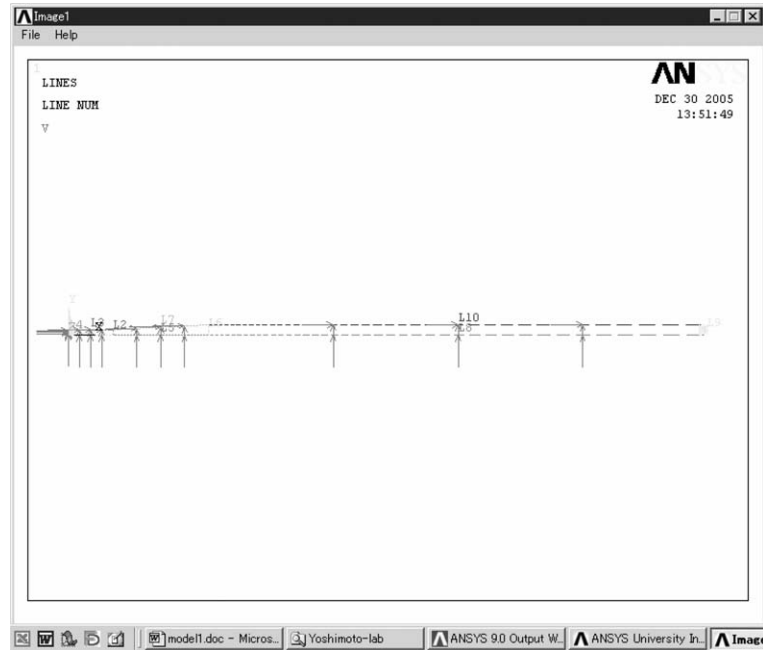


Figure 5.25 ANSYS Graphics window.

- (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.

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 [A] **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**

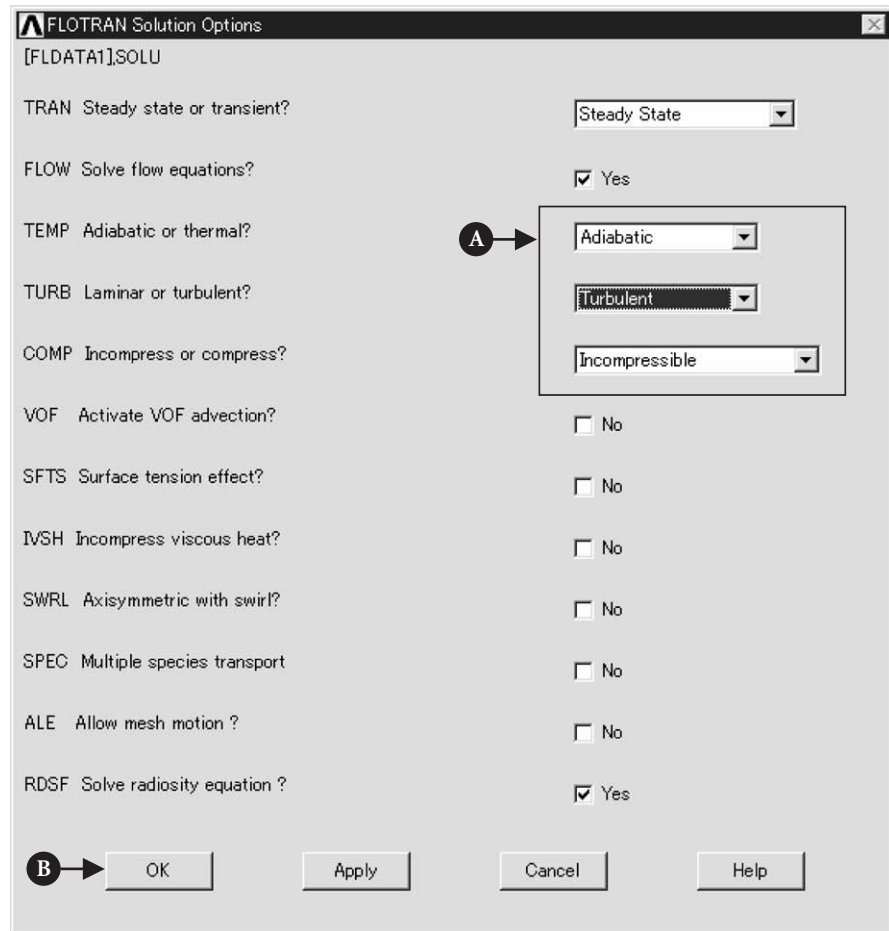


Figure 5.26 Window of FLOTRAN Solution Options.

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.

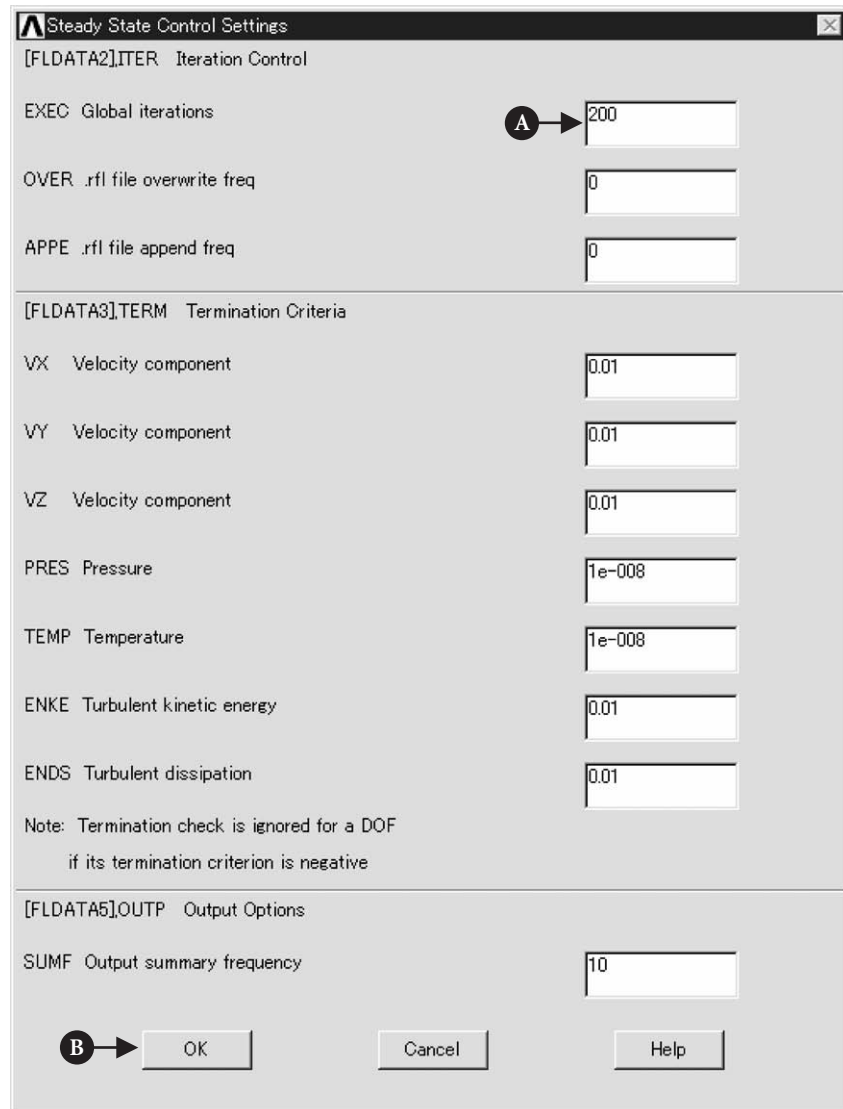


Figure 5.27 Window of **Steady State Control Settings**.

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.

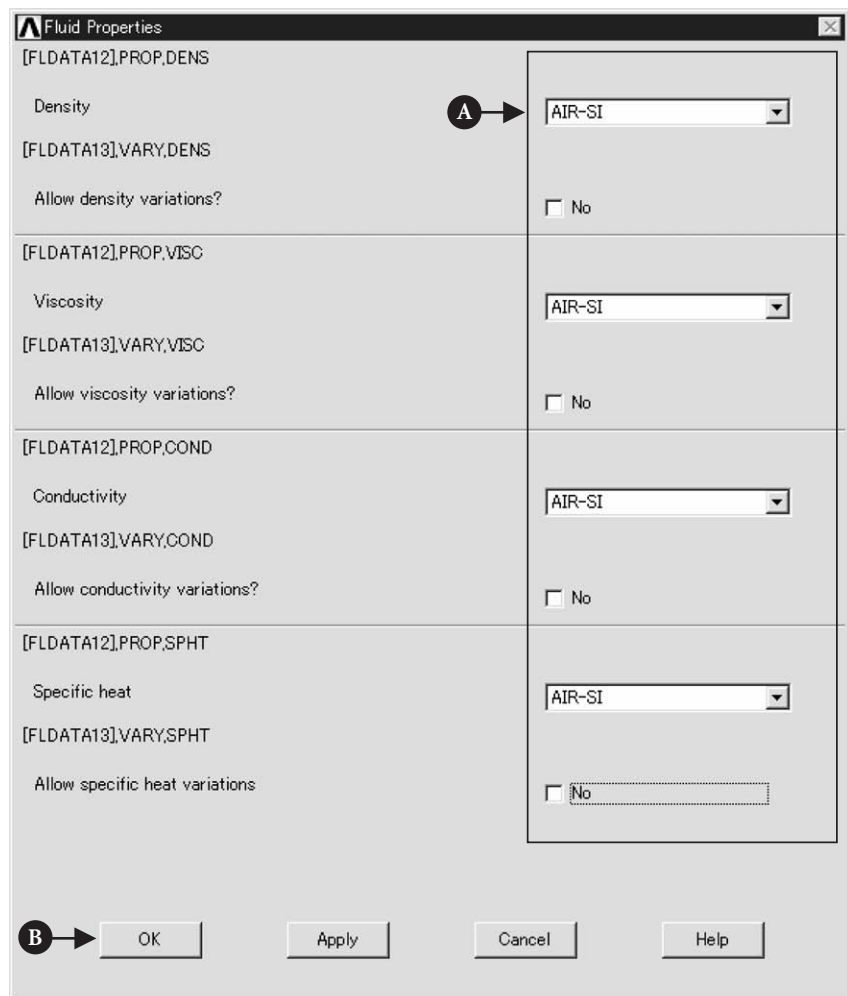


Figure 5.28 Window of **Fluid Properties**.

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



Figure 5.29 Window of CFD Fluid Properties.

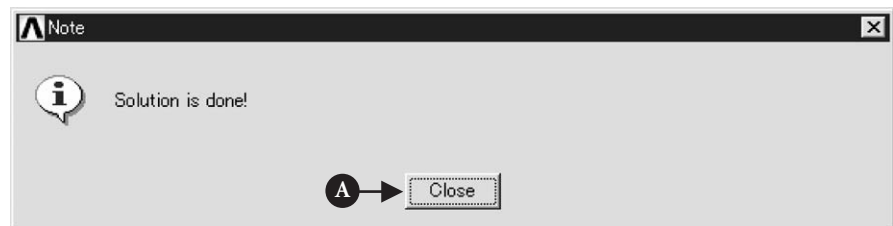


Figure 5.30 Window of Note.

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

- (1) Select [A] **DOF solution – Velocity V** and click [B] **OK**.
- (2) The calculated result for velocity vectors appears on **ANSYS Graphics** window as shown in Figure 5.32.

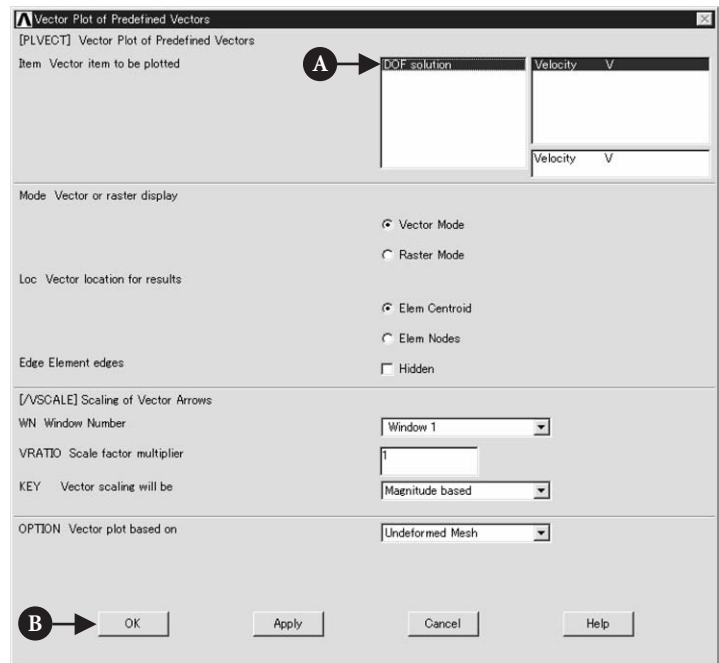


Figure 5.31 Window of **Vector Plot of Predefined Vectors**.

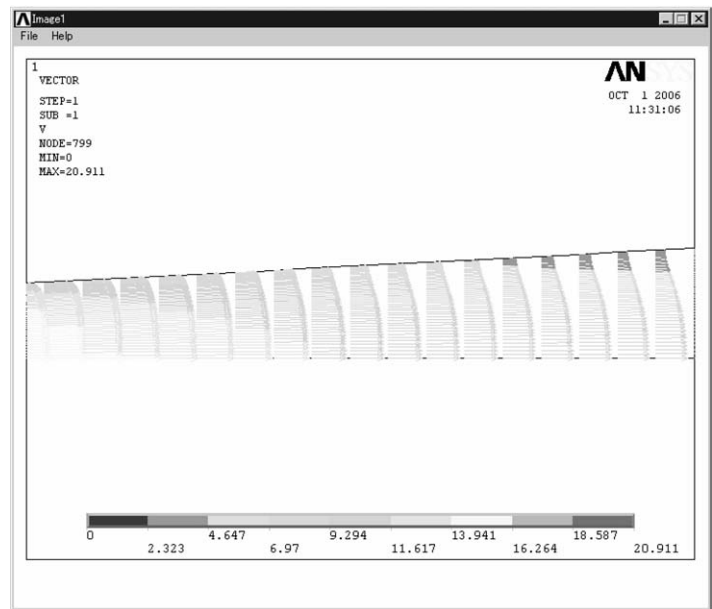


Figure 5.32 ANSYS Graphics window.

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.

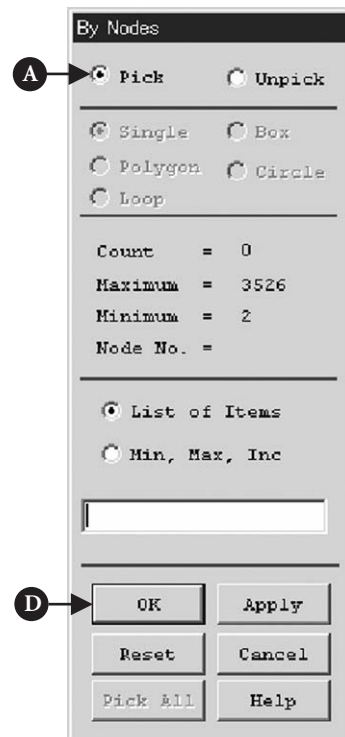


Figure 5.33 Window of **By Nodes**.

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

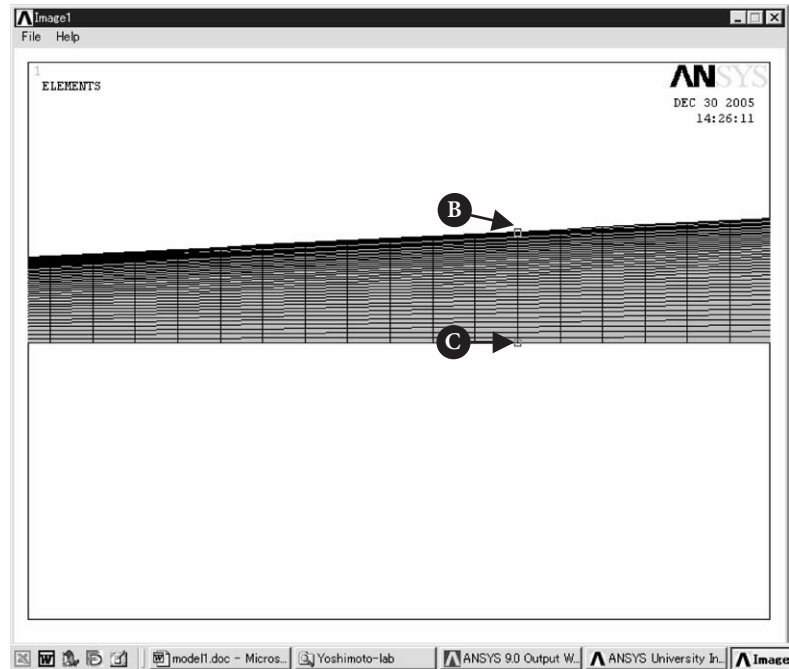


Figure 5.34 ANSYS Graphics window.

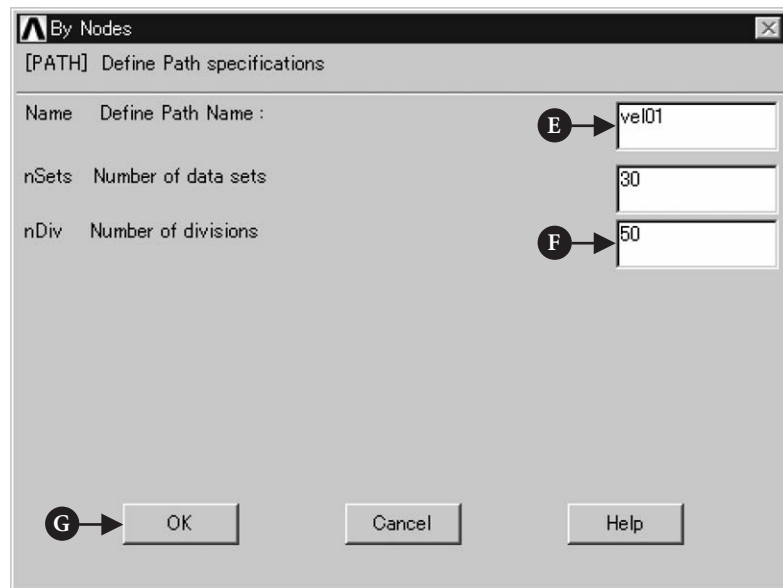


Figure 5.35 Window of By Nodes.

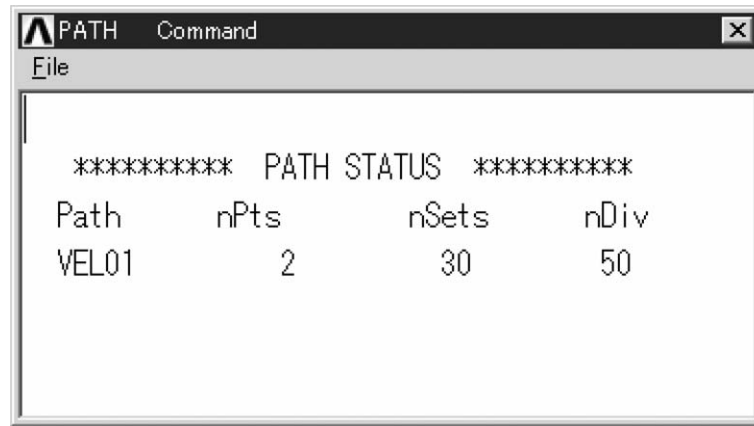


Figure 5.36 Window of PATH Command.

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.

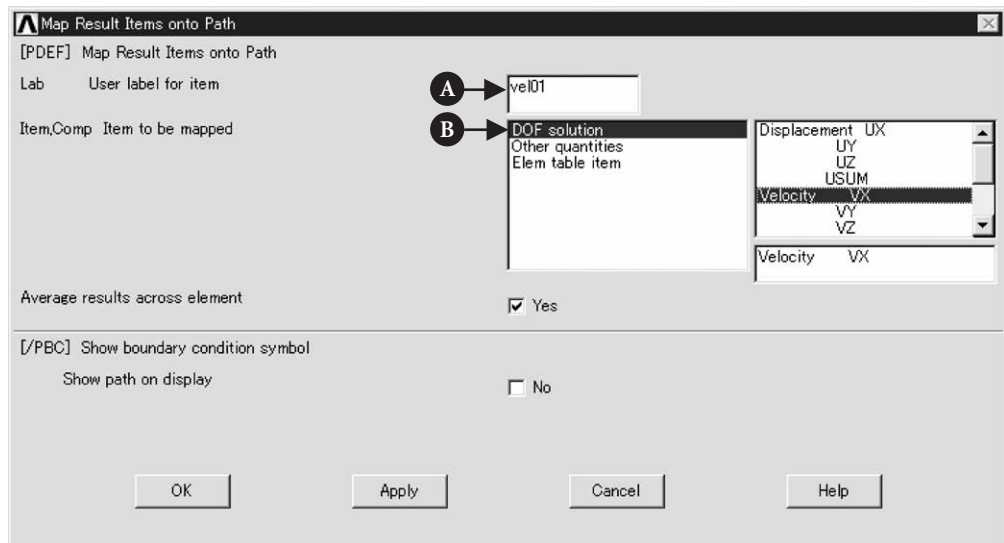


Figure 5.37 Window of Map Result Items onto Path.

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.

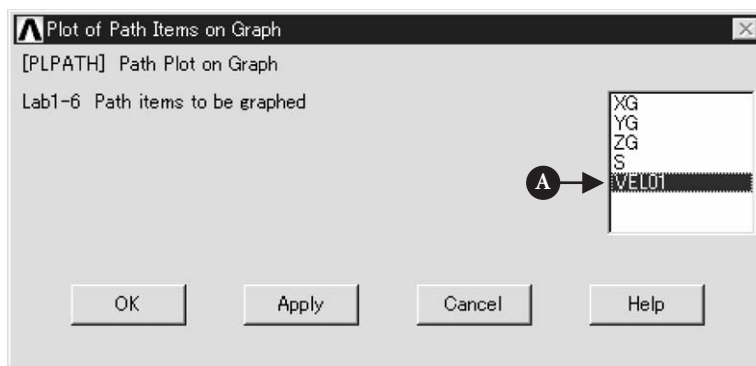


Figure 5.38 Window of **Plot of Path Items on Graph**.

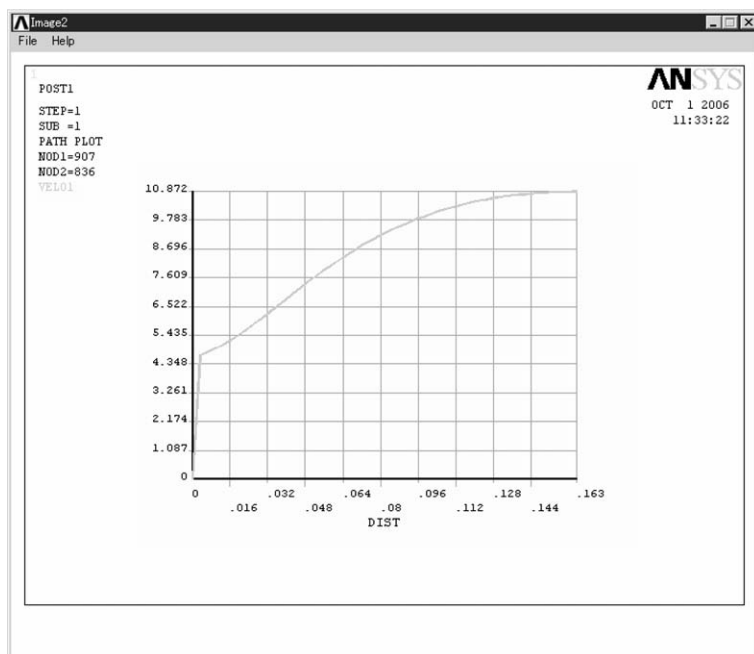


Figure 5.39 **ANSYS Graphics** window.

In addition, when the values of the defined path are needed, the following steps can be used.

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.

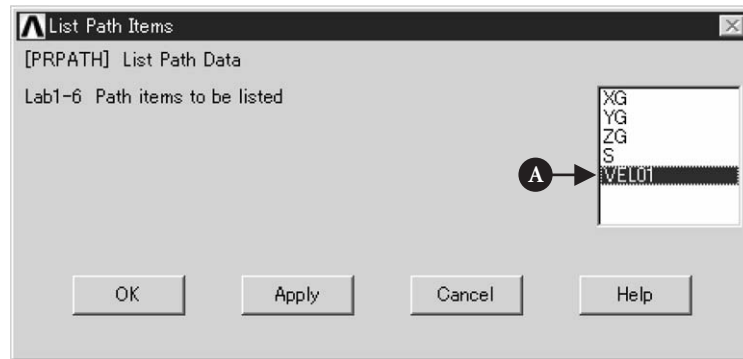


Figure 5.40 Window of **List Path Items**.

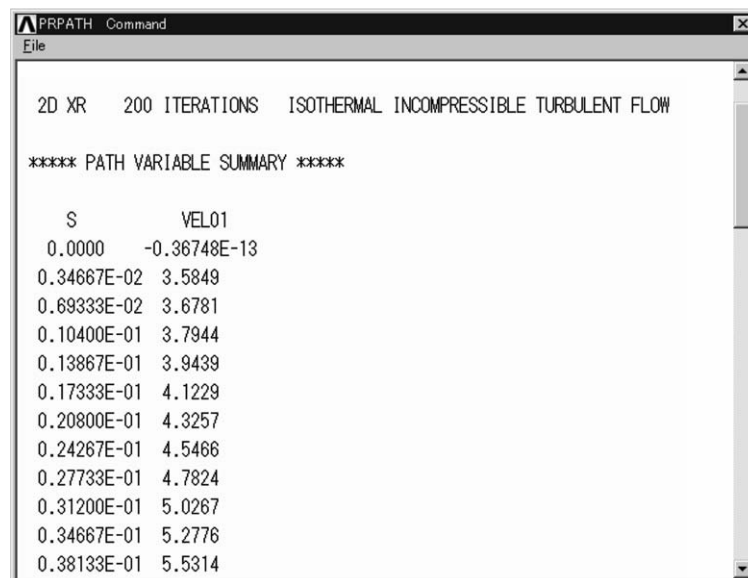


Figure 5.41 Window of **PRPATH Command**.

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.

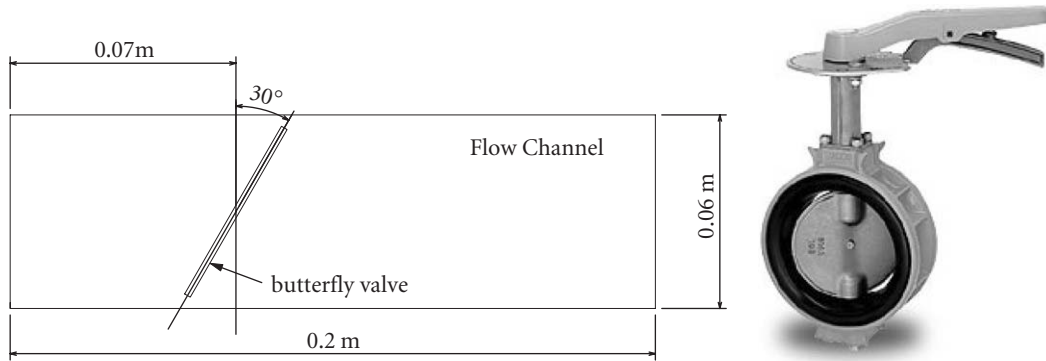


Figure 5.42 Flow channel with a butterfly valve (photo is quoted from <http://www.kitz.co.jp/product/bidg-jutaku/kyutouyou/index.html>).

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 **FLOTRAN CFD-2D FLOTRAN 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**.

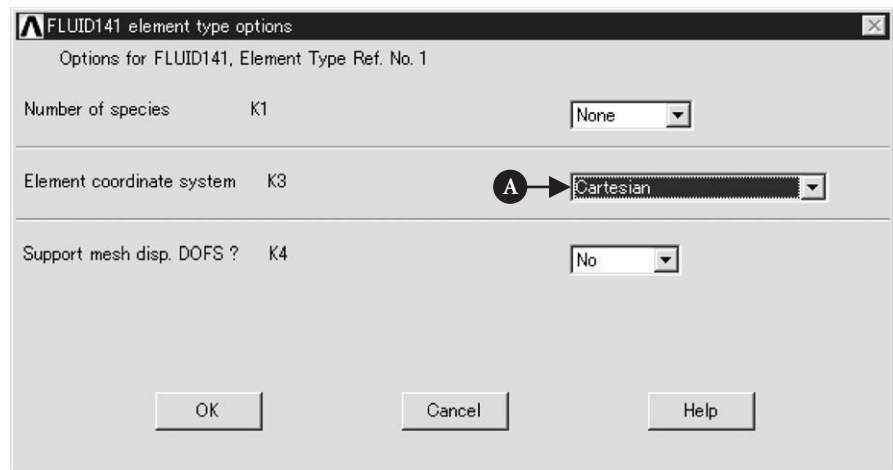


Figure 5.43 Window of FLUID141 element type options.

5.3.2.3 CREATE KEYPOINTS

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

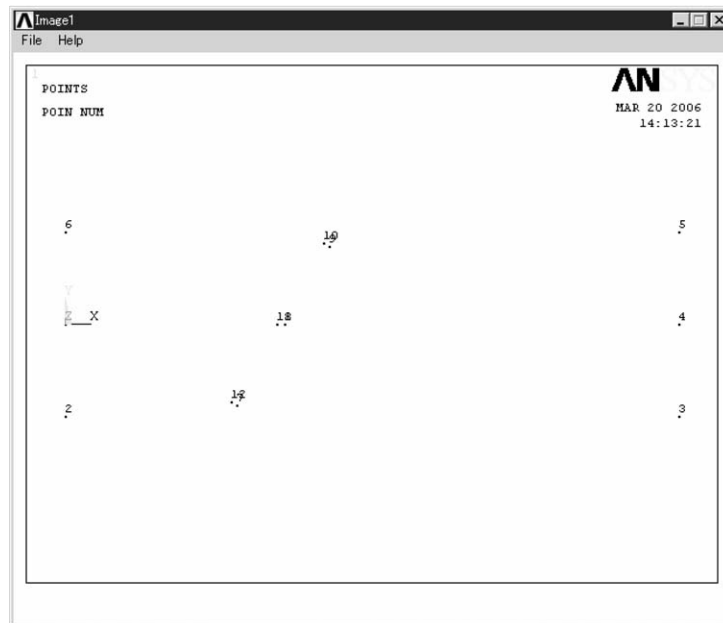
COMMAND | ANSYS Main Menu → Preprocessor → Modeling → Create → Keypoints → In Active CS

- (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

KP No.	X	Y
1	0	0
2	0	-0.03
3	0.2	-0.03
4	0.2	0
5	0.2	0.03
6	0	0.03
7	0.05587	-0.02648
8	0.0716	0
9	0.08587	0.02548
10	0.08413	0.02648
11	0.06885	0
12	0.05413	-0.02548

- (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.

**Figure 5.44** ANSYS Graphics window.

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**.

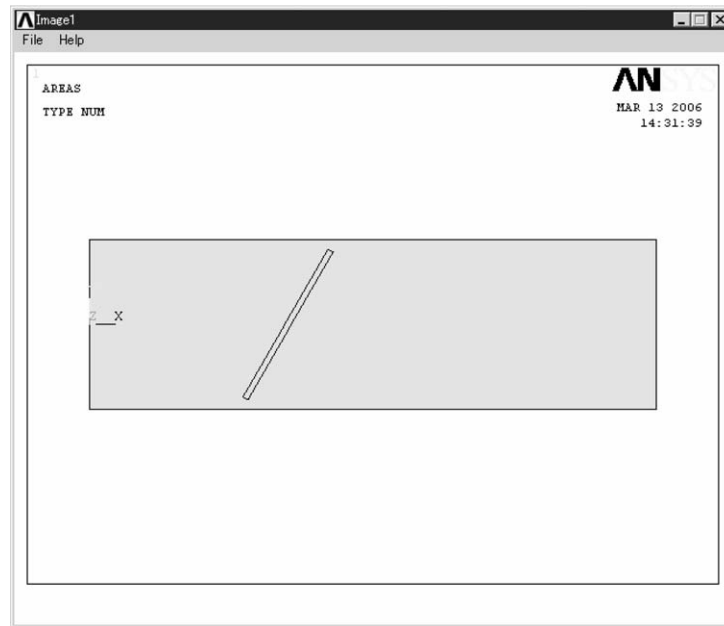


Figure 5.45 ANSYS Graphics window.

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.

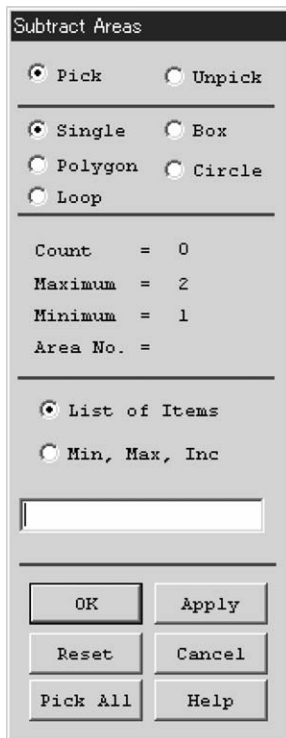


Figure 5.46 Window of Subtract Areas.

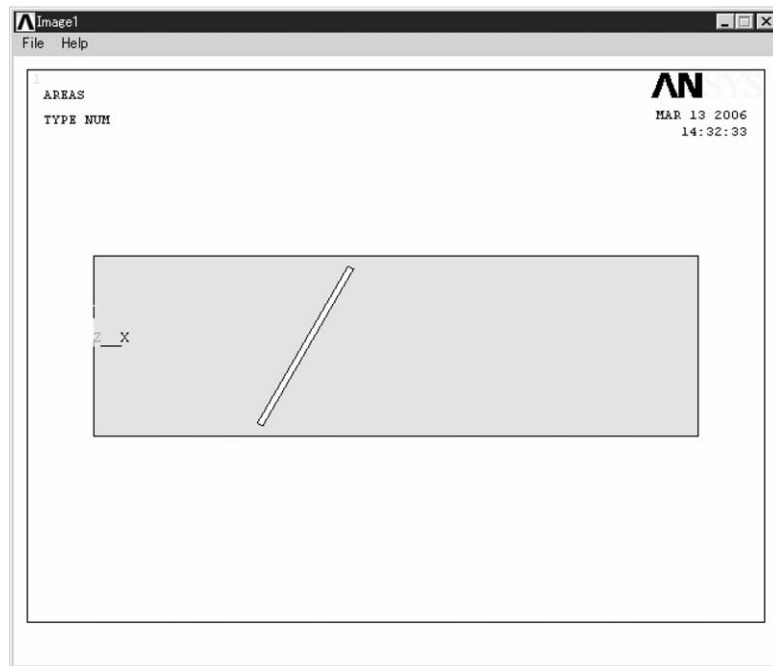


Figure 5.47 ANSYS Graphics window.

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

- (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**.

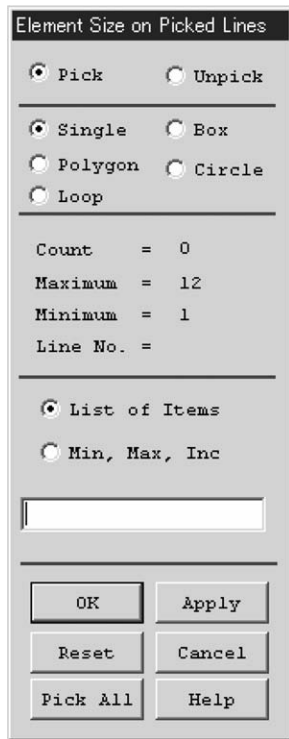


Figure 5.48 Window of Element Size on Picked Lines.

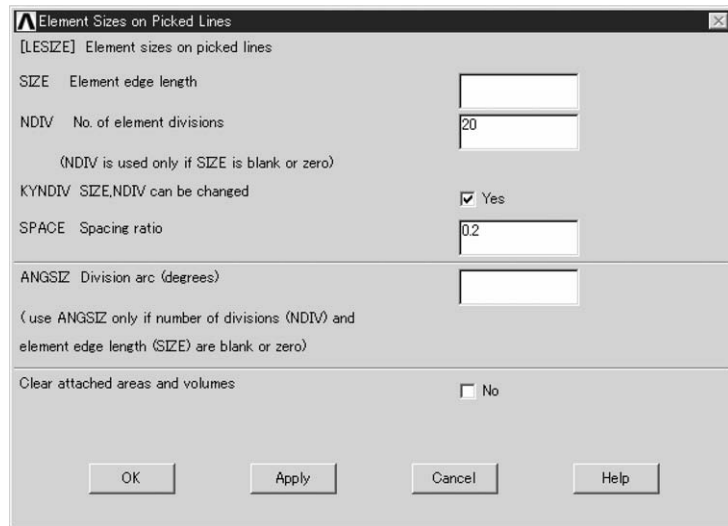


Figure 5.49 Window of Element Sizes on Picked Lines.

Table 5.6 Element sizes of picked lines

Line No.	NDIV	SPACE
1, 4, 7, 10	20	0.2
3, 6, 9 12	20	5
2, 5	100	—
8, 11	2	—

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.

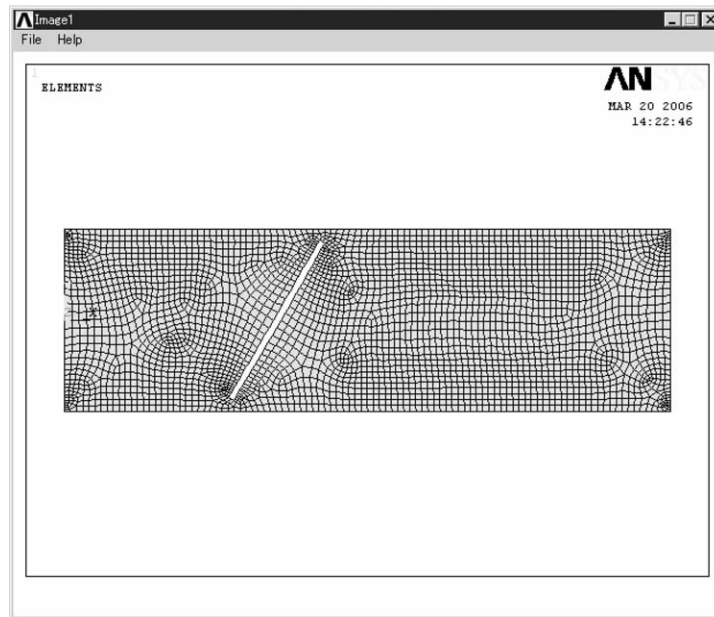


Figure 5.50 ANSYS Graphics window.

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.

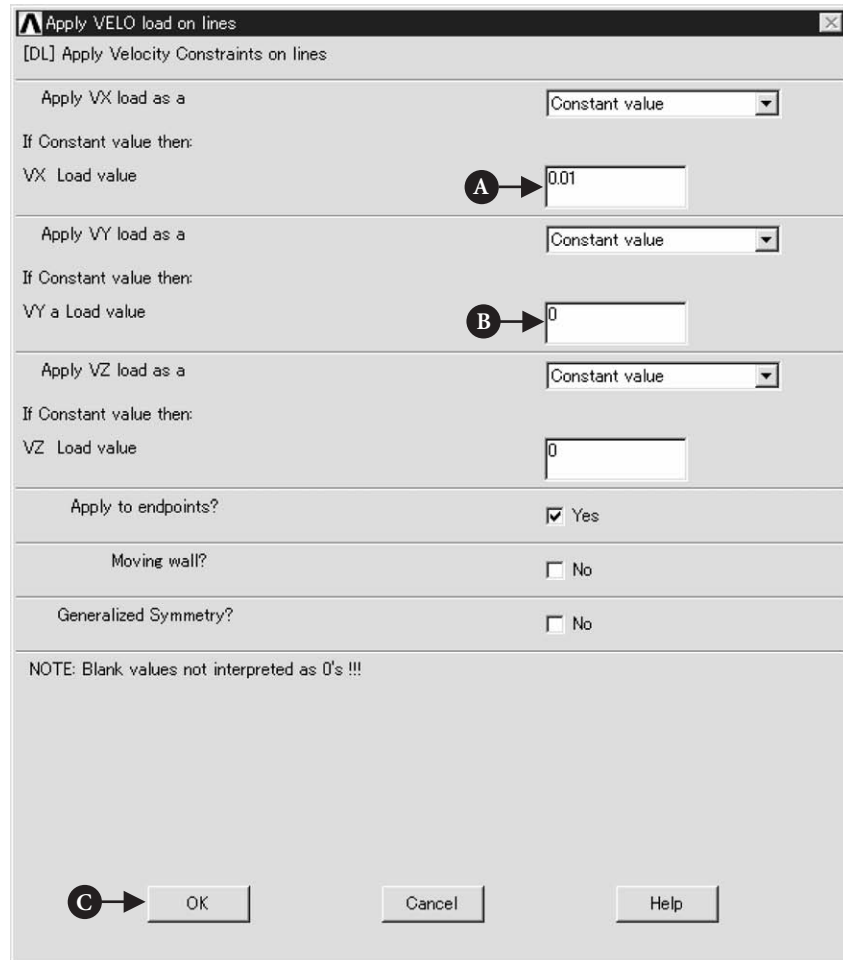


Figure 5.51 Window of Apply VELO load on lines.

- (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.

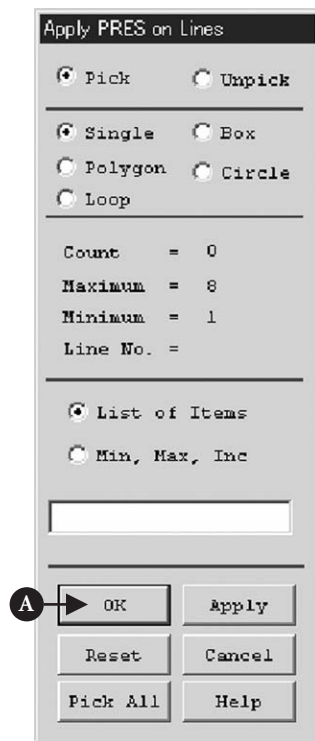


Figure 5.52 Window of Apply PRES on lines.

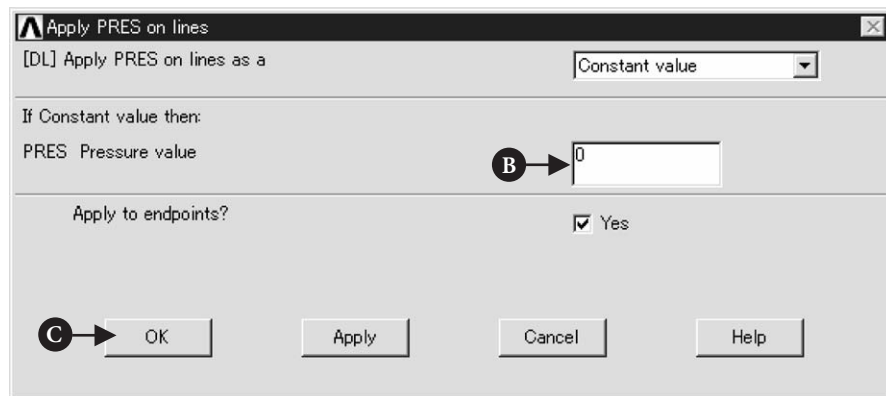


Figure 5.53 Window of Apply PRES on lines.

5.3.3 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.

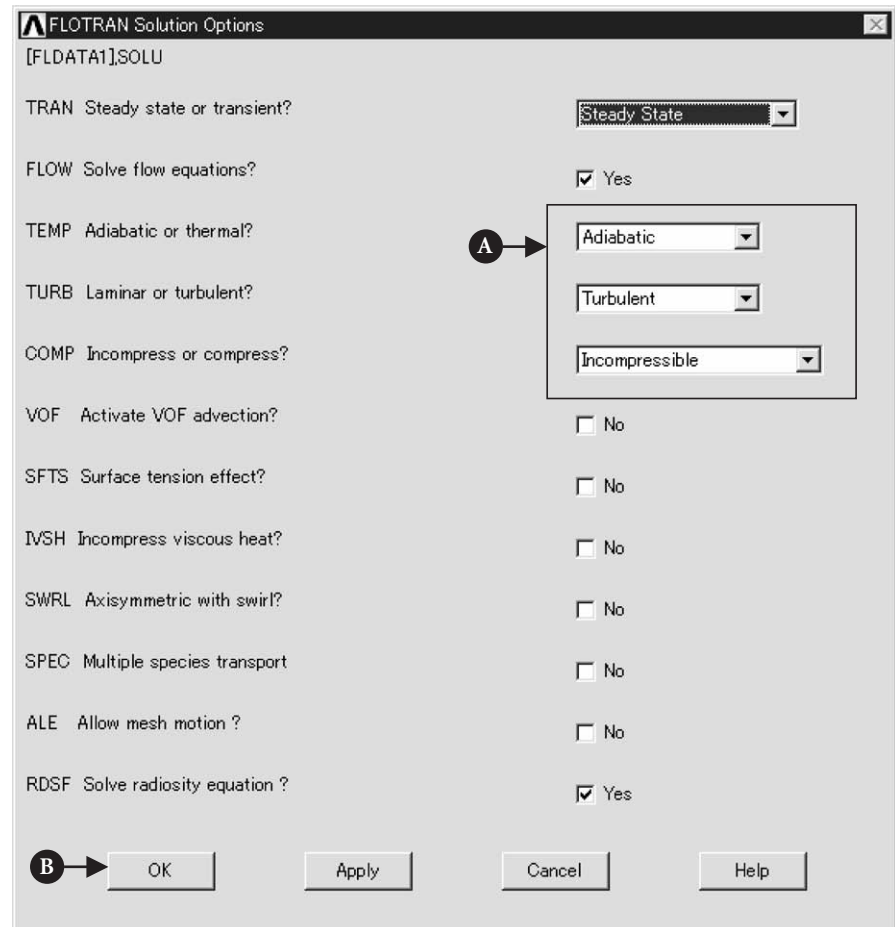


Figure 5.54 Window of FLOTRAN Solution Options.

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.

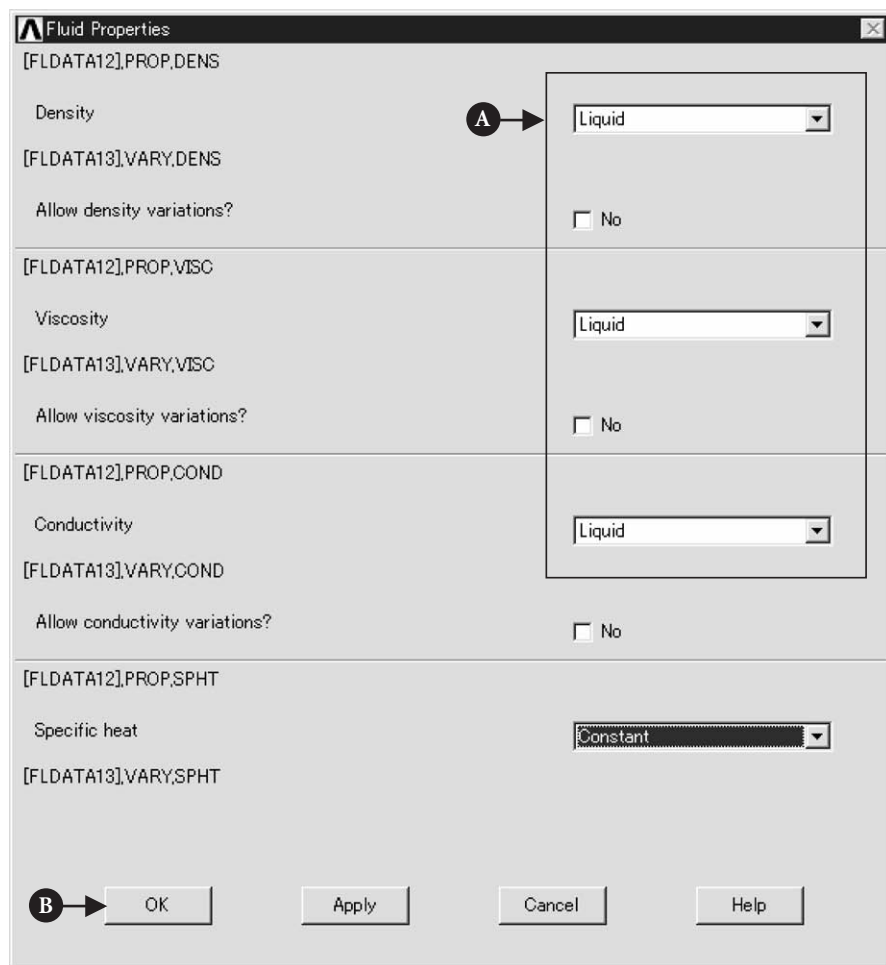


Figure 5.55 Window of **Fluid Properties**.

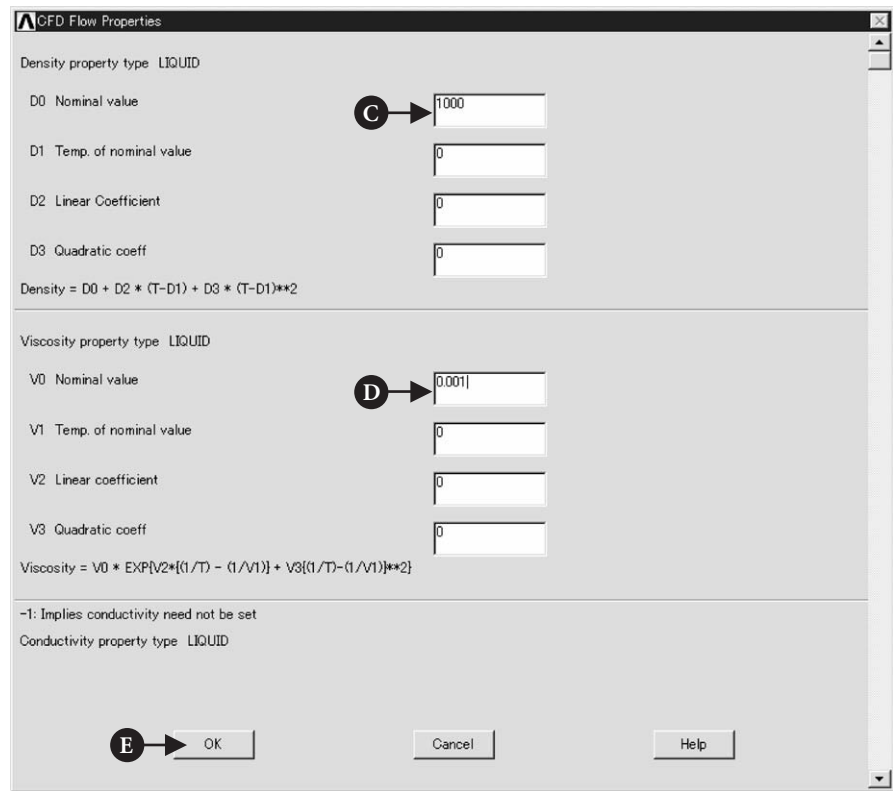


Figure 5.56 Window of CFD Flow Properties.

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.

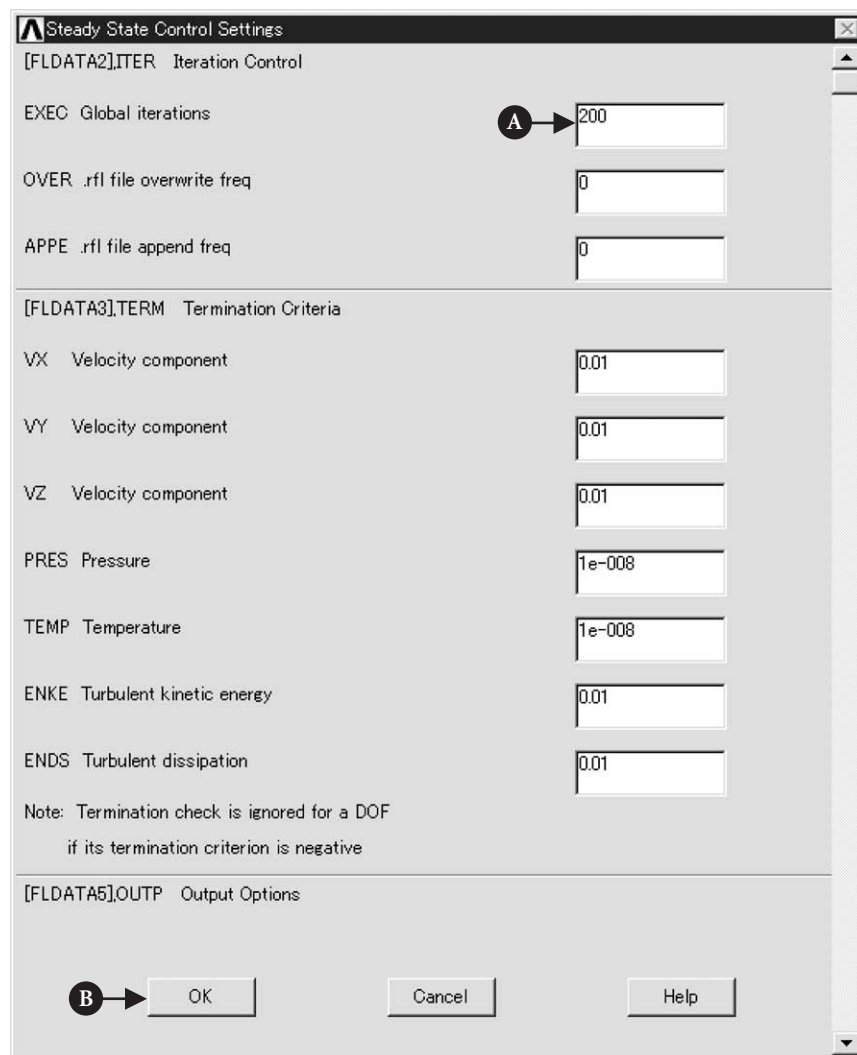


Figure 5.57 Window of **Steady State Control Settings**.

5.3.5 POSTPROCESSING

5.3.5.1 READ THE CALCULATED RESULTS

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

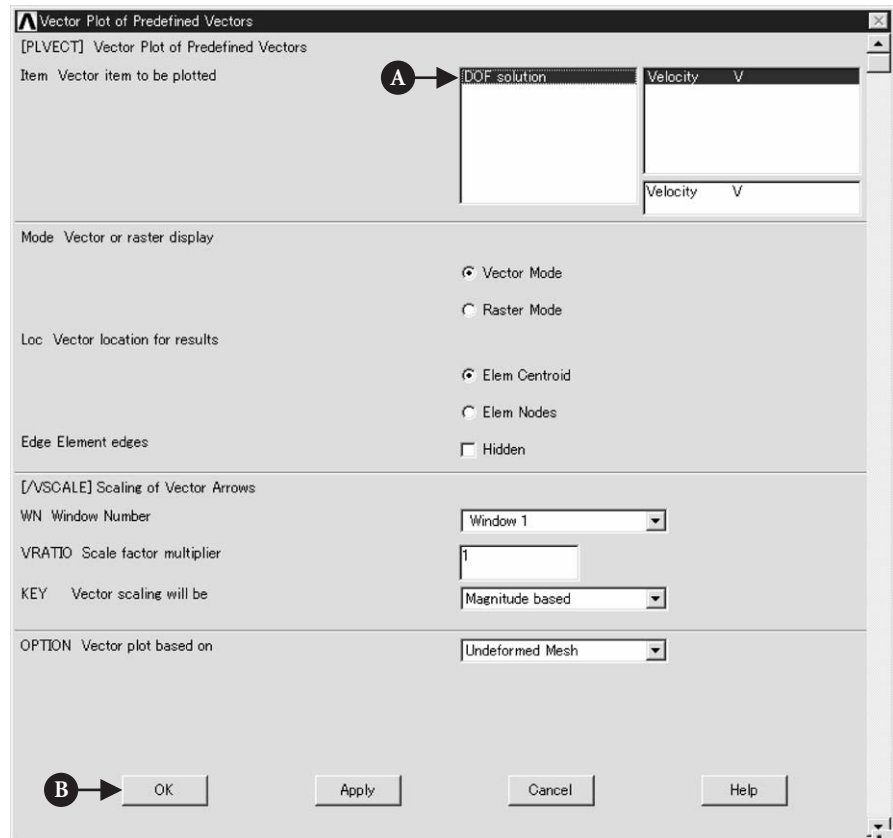


Figure 5.58 Window of **Vector Plot of Predefined Vectors**.

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).

- (1) Select [A] **DOF solution** and **Velocity V** and click [B] **OK**.
- (2) The calculated result for velocity vectors appears on **ANSYS Graphics** window as shown in Figure 5.59.

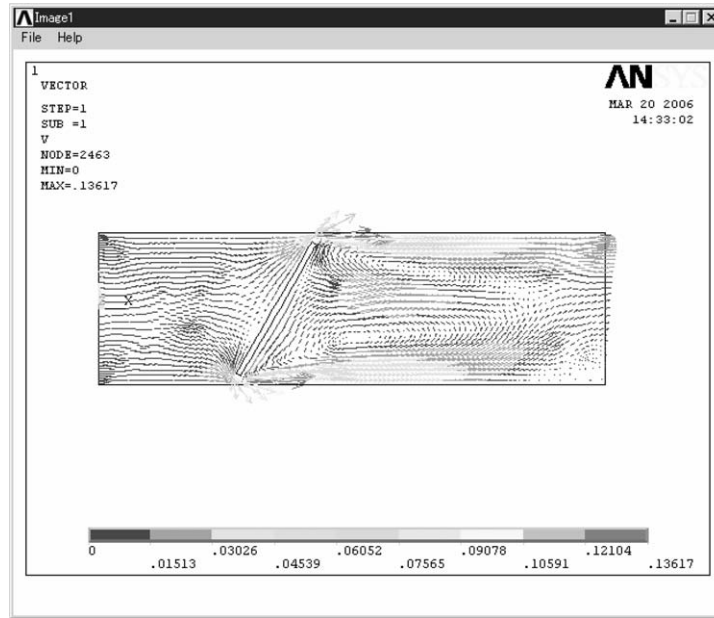


Figure 5.59 ANSYS Graphics window.

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

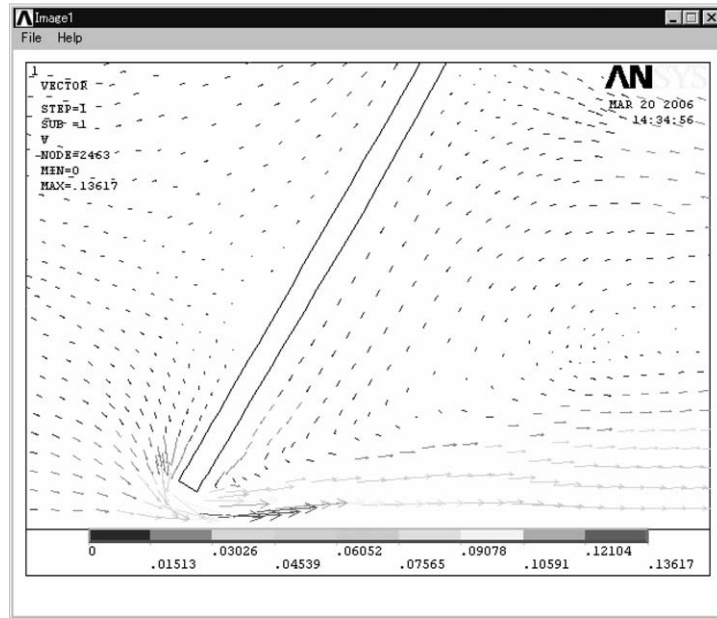


Figure 5.60 Enlarged view near the valve in ANSYS Graphics window.

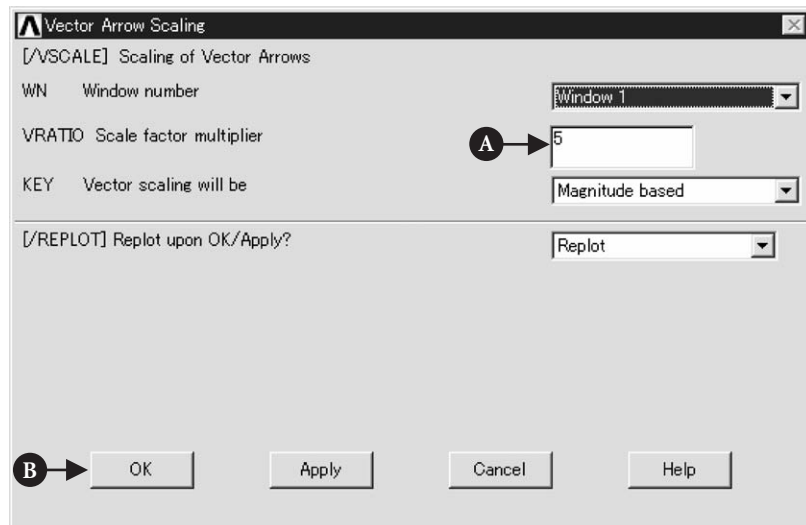


Figure 5.61 Window of Vector Arrow Scaling.

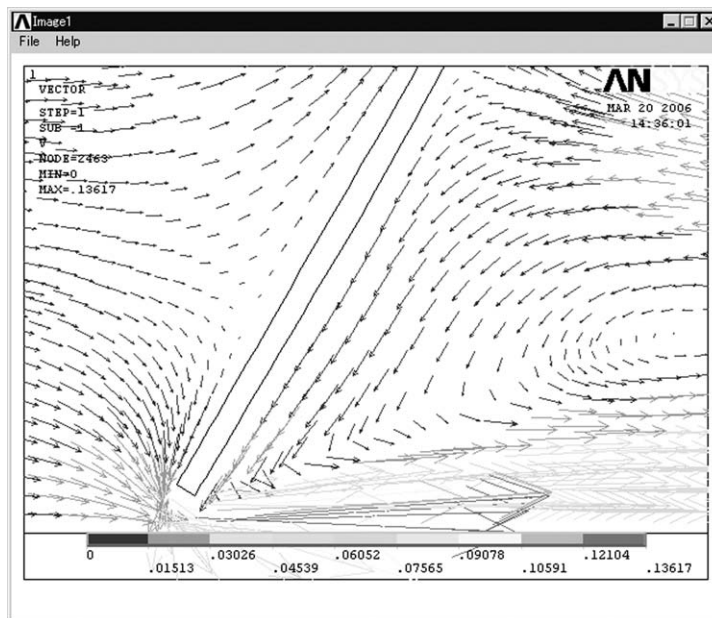


Figure 5.62 ANSYS Graphics window.

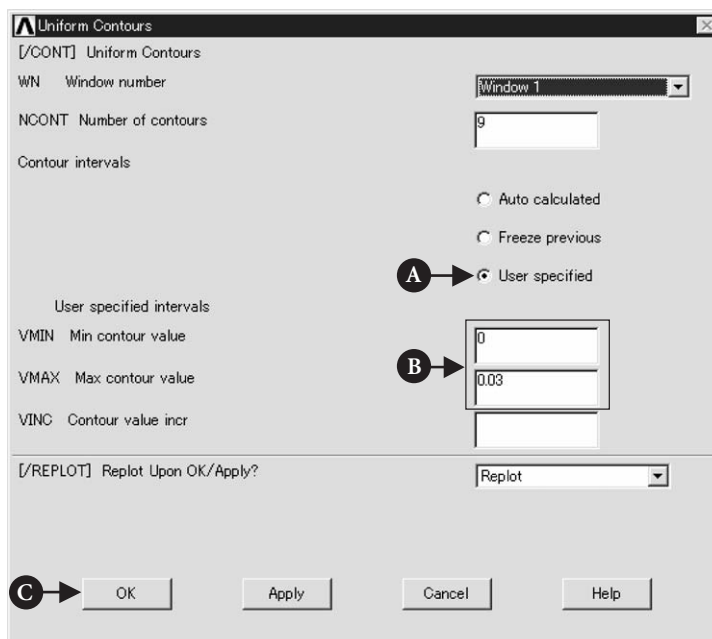


Figure 5.63 Window of Uniform Contours.

displayed on **ANSYS Graphics** window are colored only from **0** to **0.03** as shown in Figure 5.64.

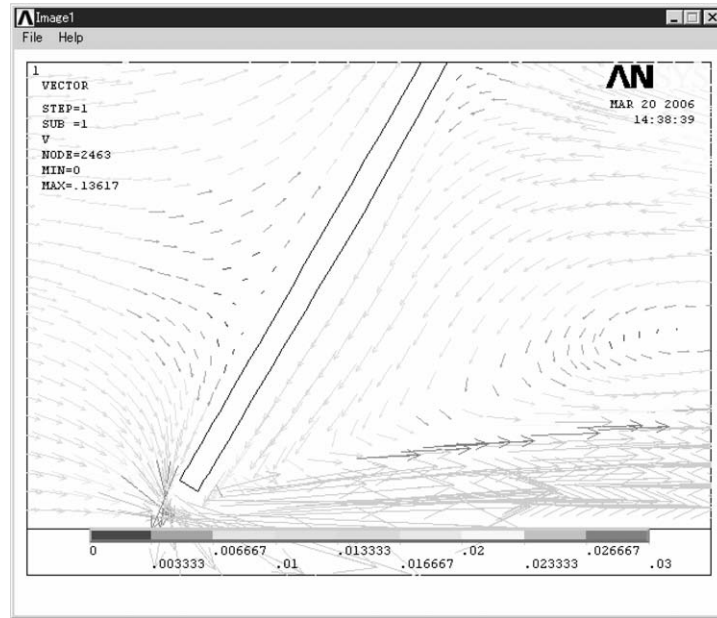


Figure 5.64 ANSYS Graphics window.

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.

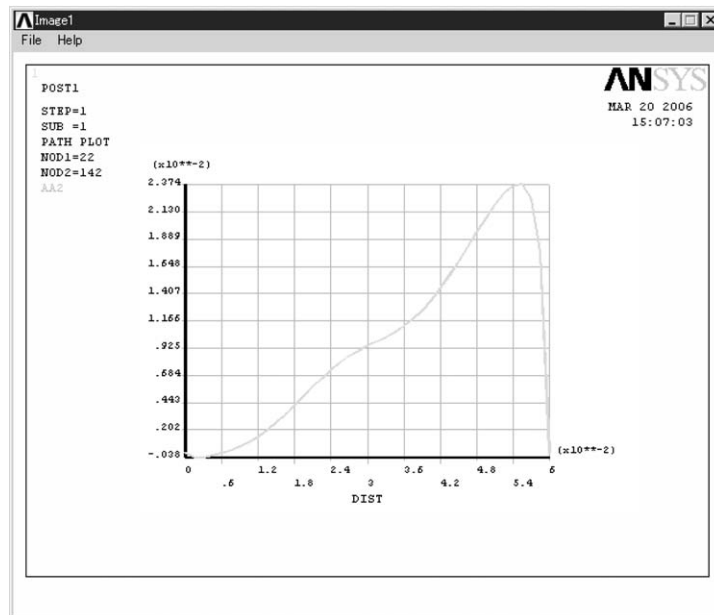


Figure 5.65 ANSYS Graphics window.

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.

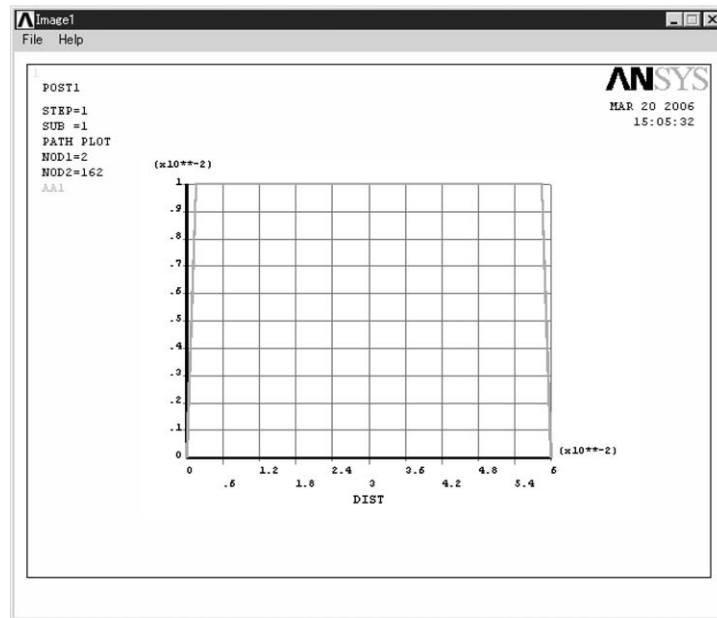


Figure 5.66 ANSYS Graphics window.

The values of flow velocities of the defined paths can be listed by the following commands.

COMMAND | ANSYS Main Menu → General Postproc → Path Operations → Plot Path Item → List Path Items

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.