CHAPTER 4

MODE ANALYSIS

4.1 INTRODUCTION

When a steel bar is hit by a hammer, a clear sound can be heard because the steel bar vibrates at its resonant frequency. If the bar is oscillated at this resonant frequency, it will be found that the vibration amplitude of the bar becomes very large. Therefore, when a machine is designed, it is important to know the resonant frequency of the machine. The analysis to obtain the resonant frequency and the vibration mode of an elastic body is called "mode analysis."

It is said that there are two methods for mode analysis. One is the theoretical analysis and the other is the finite-element method (FEM). Theoretical analysis is usually used for a simple shape of an elastic body, such as a flat plate and a straight bar, but theoretical analysis cannot give us the vibration mode for the complex shape of an elastic body. FEM analysis can obtain the vibration mode for it.

In this chapter, three examples are presented, which use beam elements, shell elements, and solid elements, respectively, as shown in Figure 4.1.
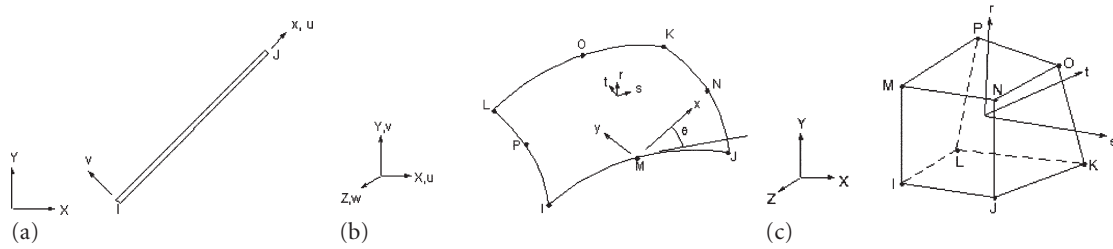
Figure 4.1 FEM element types: (a) beam element; (b) shell element; and (c) solid element.

4.2 MODE ANALYSIS OF A STRAIGHT BAR

4.2.1 PROBLEM DESCRIPTION

Obtain the lowest three vibration modes and resonant frequencies in the y direction of the straight steel bar shown in Figure 4.2.
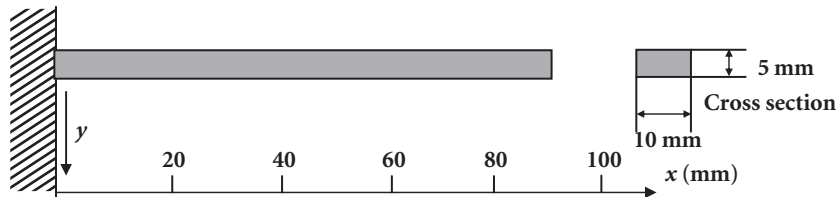

Figure 4.2 Cantilever beam for mode analysis.

Thickness of the bar is 0.005 m, width is 0.01 m, and the length is 0.09 m. Material of the bar is steel with Young's modulus, $E = 206$ GPa, and Poisson's ratio $\nu = 0.3$. Density $\rho = 7.8 \times 10^3$ kg/m^3.

Boundary condition: All freedoms are constrained at the left end.

4.2.2 ANALYTICAL SOLUTION

Before mode analysis is attempted using ANSYS program, an analytical solution for resonant frequencies will be obtained to confirm the validity of ANSYS solution. The analytical solution of resonant frequencies for a cantilever beam in y direction is given by:

$$f_i = \frac{\lambda_i^2}{2\pi L^2}\sqrt{\frac{EI}{M}} \quad (i = 1, 2, 3, \dots) \tag{4.1}$$

where length of the cantilever beam, $L = 0.09$ m, cross-section area of the cantilever beam, $A = 5 \times 10^{-5}$ m^2, and Young's modulus, $E = 206$ GPa.

The area moment of inertia of the cross-section of the beam is:

$$I = bt^3/12 = (0.01 \times 0.005^3)/12 = 1.042 \times 10^{-10} \, \text{m}^4$$

Mass per unit width $M = \rho AL/L = \rho A = 7.8 \times 10^3 \, \text{kg/m}^3 \times 5 \times 10^{-5} \, \text{m}^2 = 0.39 \, \text{kg/m}$

$$\lambda_1 = 1.875 \quad \lambda_2 = 4.694 \quad \lambda_3 = 7.855.$$

For that set of data the following solutions are obtained: $f_1 = 512.5$ Hz, $f_2 = 3212$ Hz, and $f_3 = 8994$ Hz.

Figure 4.3 shows the vibration modes and the positions of nodes obtained by Equation (4.1).
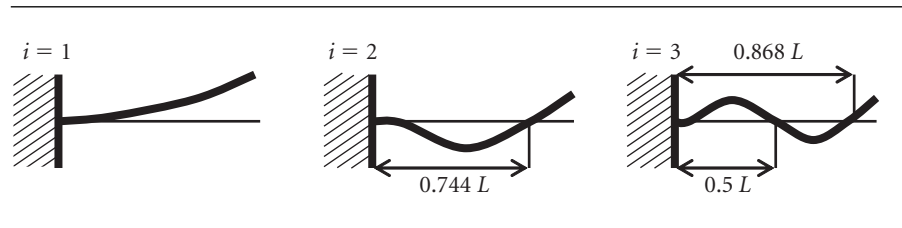

Figure 4.3   Analytical vibration mode and the node position.

4.2.3   Model for finite-element analysis

4.2.3.1   Element type selection

In FEM analysis, it is very important to select a proper element type which influences the accuracy of solution, working time for model construction, and CPU time. In this example, the two-dimensional elastic beam, as shown in Figure 4.4, is selected for the following reasons:
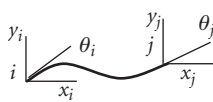
(a) Vibration mode is constrained in the two-dimensional plane.

(b) Number of elements can be reduced; the time for model construction and CPU time are both shortened.

Figure 4.4   Two-dimensional beam element.

Two-dimensional elastic beam has three degrees of freedom at each node (i, j), which are translatory deformations in the x and y directions and rotational deformation around the z-axis. This beam can be subjected to extension or compression bending due to its length and the magnitude of the area moment of inertia of its cross section.

C O M M A N D | **ANSYS Main Menu → Preprocessor → Element Type → Add/Edit/Delete**

Then the window **Element Types**, as shown in Figure 4.5, is opened.

(1) Click [A] **Add**. Then the window **Library of Element Types** as shown in Figure 4.6 opens.
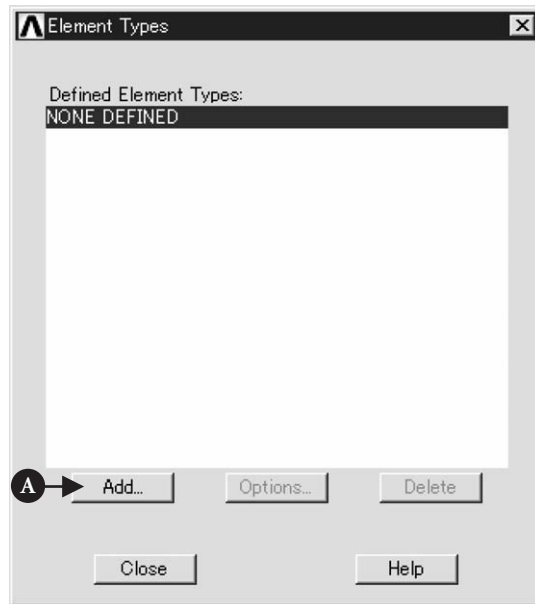
Figure 4.5   Window of **Element Types**.
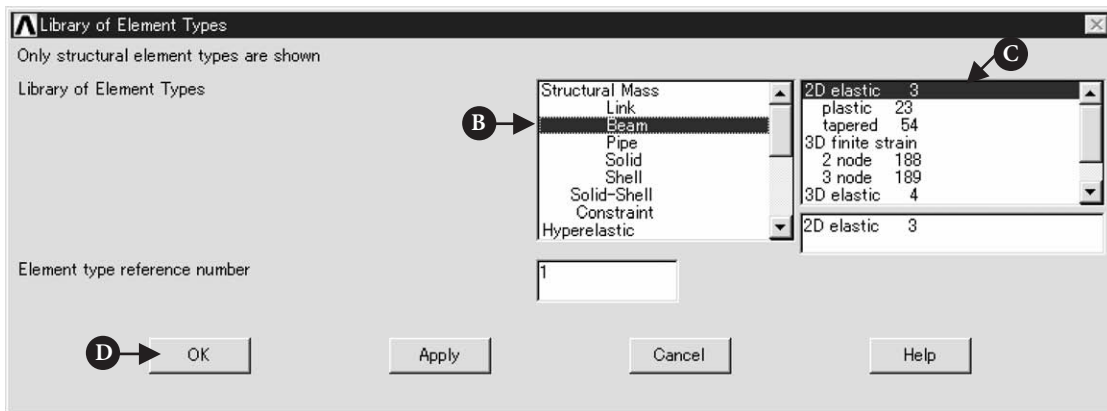
Figure 4.6   Window of **Library of Element Types**.

(2) Select [B] **Beam** in the table **Library of Element Types** and, then, select [C] **2D elastic 3**.

(3) **Element type reference number** is set to **1** and click D **OK** button. Then the window **Library of Element Types** is closed.

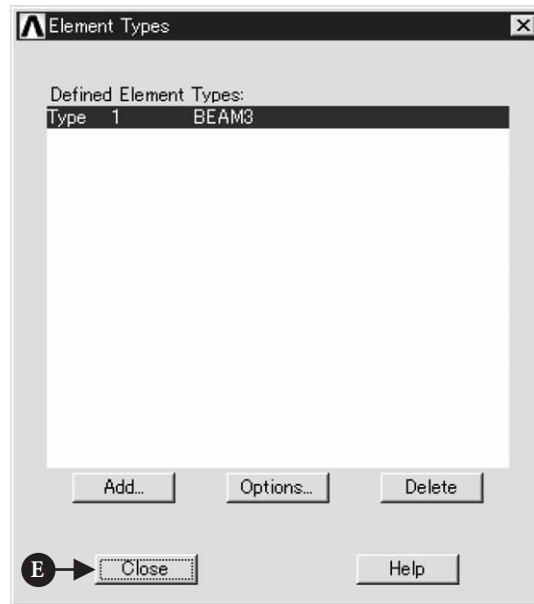(4) Click [E] **Close** button in the window of Figure 4.7.

Figure 4.7 Window of **Library of Element Types**.

4.2.3.2 REAL CONSTANTS FOR BEAM ELEMENT

C O M M A N D | **ANSYS Main Menu → Preprocessor → Real Constants → Add/Edit/Delete → Add**

(1) The window **Real Constants** opens. Click [A] **add** button, and the window **Element Type for Real Constants** appears in which the name of element type selected is listed as shown in Figure 4.8.

(2) Click [B] **OK** button to input the values of real constants and the window **Real Constant** for BEAM3 is opened (Figure 4.9).

(3) Input the following values in Figure 4.10. [C] Cross-sectional area = 5e−5; [D] Area moment of inertia = 1.042e−10; [E] Total beam height = 0.005. After inputting these values, click [F] **OK** button to close the window.

(4) Click [G] **Close** button in the window **Real Constants** (Figure 4.11).

4.2.3.3 MATERIAL PROPERTIES

This section describes the procedure of defining the material properties of the beam element.

C O M M A N D | **ANSYS Main Menu → Preprocessor → Material Props → Material Models**

(1) Click the above buttons in the specified order and the window **Define Material Model Behavior** opens (Figure 4.12).
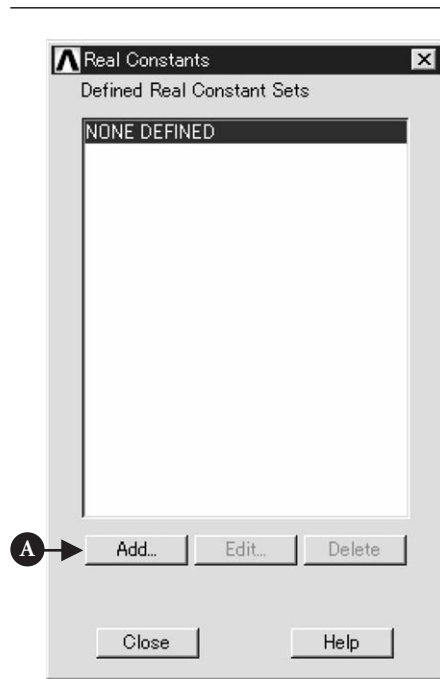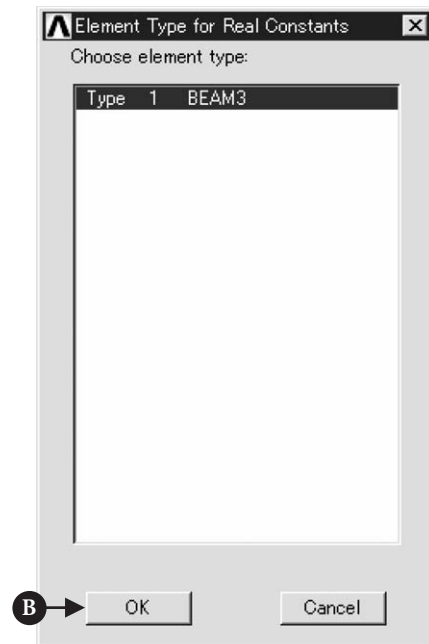
Figure 4.8 Window of **Real Constants**.

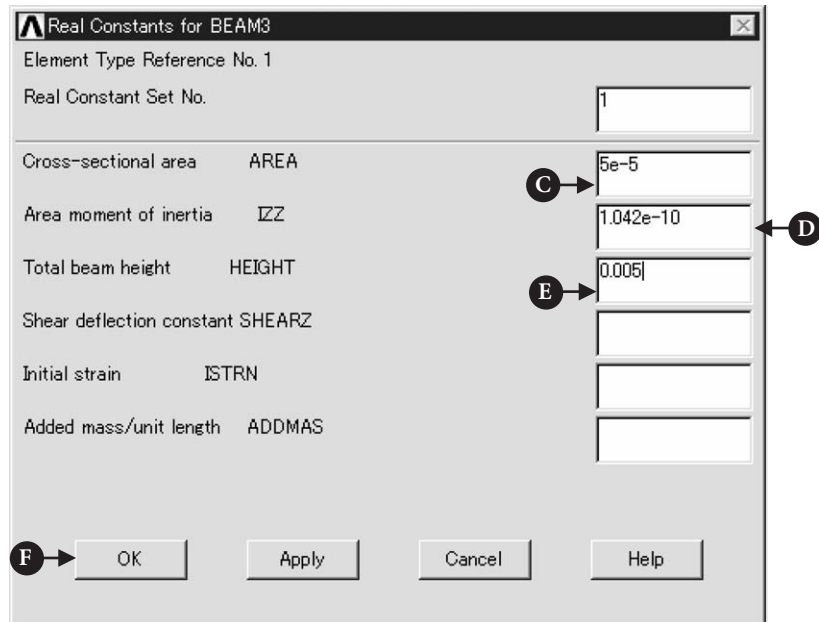**Figure 4.9** Window of **Element Type for Real Constants**.

Figure 4.10 Window of **Real Constants for BEAM3**.

Figure 4.11 Window of **Real Constants**.

(2) Double click the following terms in the window.

[A] **Structural → Linear → Elastic → Isotropic.**

As a result the window **Linear Isotropic Properties for Material Number 1** opens (Figure 4.13).

(3) Input Young's modulus of **206e9** to [B] **EX** box and Poisson ratio of **0.3** to [C] **PRXY** box. Then click [D] **OK** button.

Next, define the value of density of material.

(1) Double click the term of **Density** in Figure 4.12 and the window **Density for Material Number 1** opens (Figure 4.14).

(2) Input the value of density, **7800** to [F] **DENS** box and click [G] **OK** button. Finally, close the window **Define Material Model Behavior** by clicking [H] **X** mark at the upper right corner (Figures 4.12 and 4.14).

4.2.3.4 CREATE KEYPOINTS

To draw a cantilever beam for analysis, the method of using keypoints is described.

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

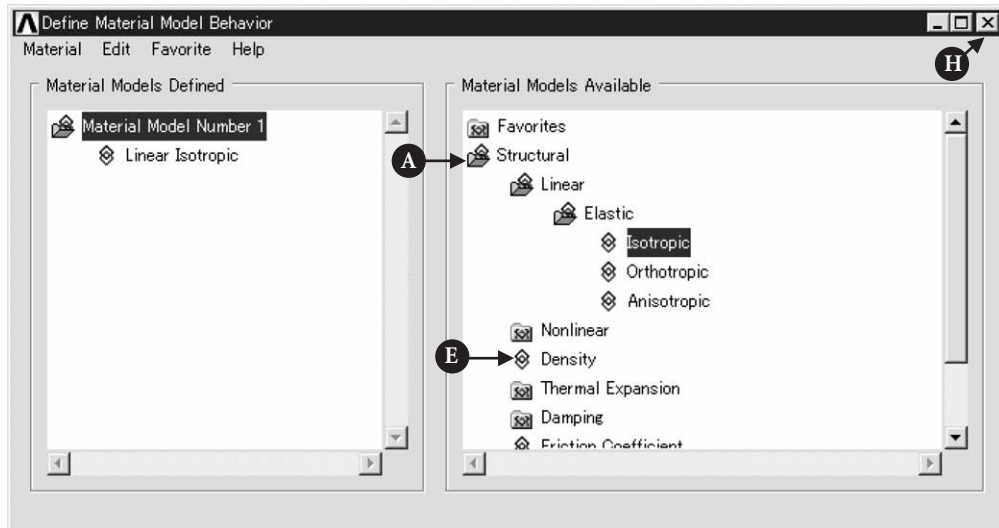The window **Create Keypoints in Active Coordinate System** opens.
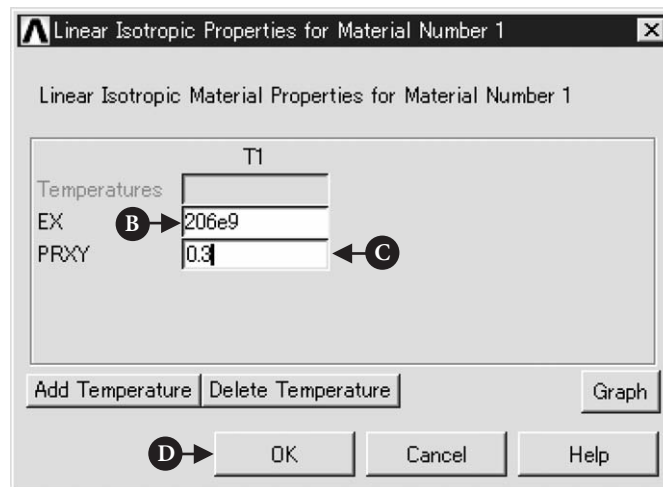
Figure 4.12 Window of **Define Material Model Behavior**.

Figure 4.13 Window of **Linear Isotropic Properties for Material Number 1**.

(1) Input **1** to [A] **NPT KeyPoint number** box, **0,0,0** to [B] **X, Y, Z Location in active CS** box, and then click [C] **Apply** button. Do not click **OK** button at this stage. If you click **OK** button, the window will be closed. In this case, open the window **Create Keypoints in Active Coordinate System** and then proceed to step 2 (Figure 4.15).
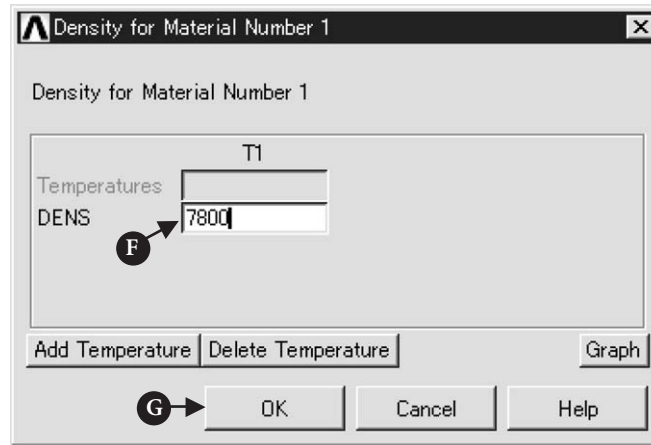
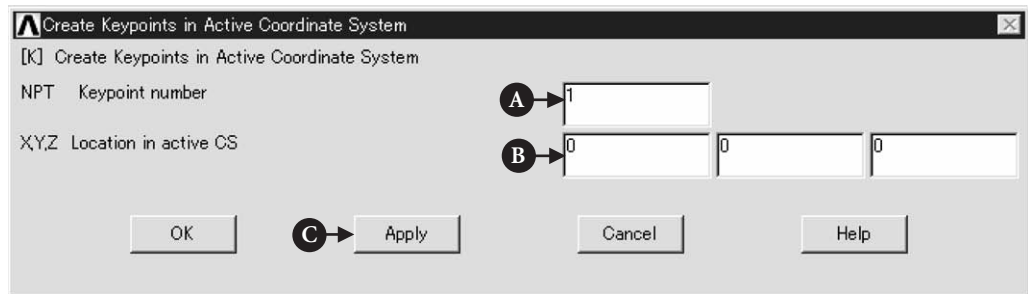**Figure 4.14**   Window of **Density for Material Number 1**.

Figure 4.15   Window of **Create Keypoint in Active Coordinate System**.

(2) In the same window, input **2** to [D] **NPT Keypoint number** box, **0.09, 0,0** to [E] **X, Y, Z Location in active CS** box, and then click [F] **OK** button (Figure 4.16).

(3) After finishing the above steps, two keypoints appear in the window (Figure 4.17).

4.2.3.5   CREATE A LINE FOR BEAM ELEMENT

By implementing the following steps, a line between two keypoints is created.

COMMAND   **ANSYS Main Menu → Preprocessor → Modeling → Create → Lines → Lines → Straight Line**
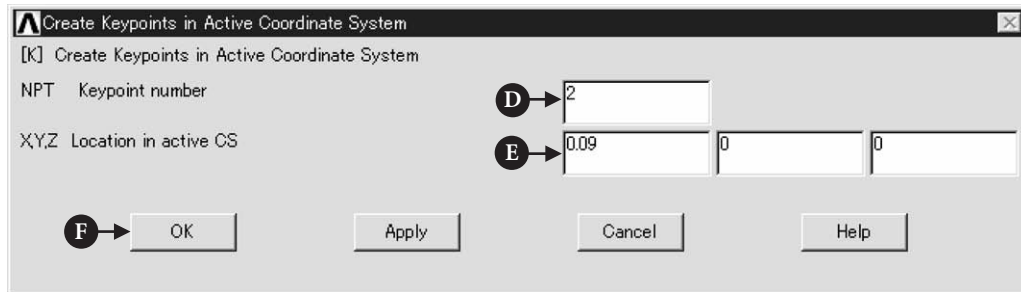
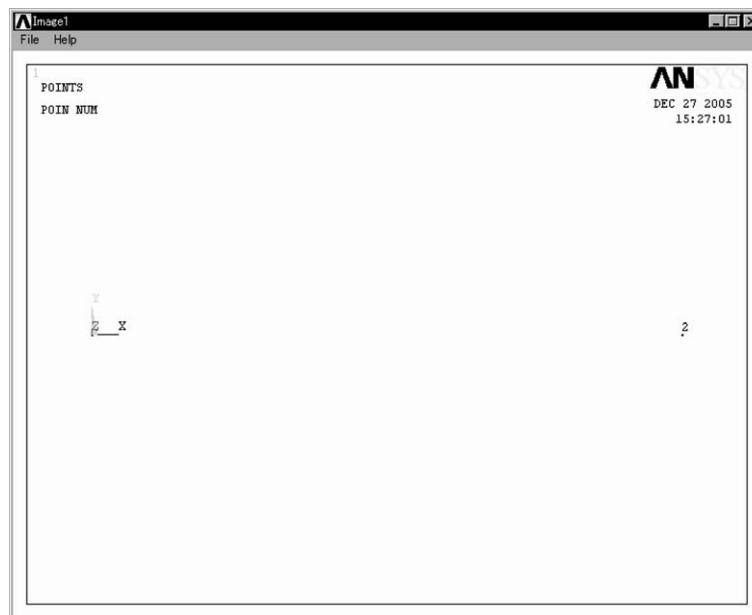**Figure 4.16**   Window of **Create Keypoint in Active Coordinate System**.

Figure 4.17   **ANSYS Graphics** window.

The window **Create Straight Line**, as shown in Figure 4.18, is opened.

(1) Pick the keypoints [A] **1** and [B] **2** (as shown in Figure 4.19) and click [C] **OK** button in the window **Create Straight Line** (as shown in Figure 4.18). A line is created.

4.2.3.6   CREATE MESH IN A LINE

COMMAND   **ANSYS Main Menu → Preprocessor → Meshing → Size Cntrls → Manual Size → Lines → All Lines**
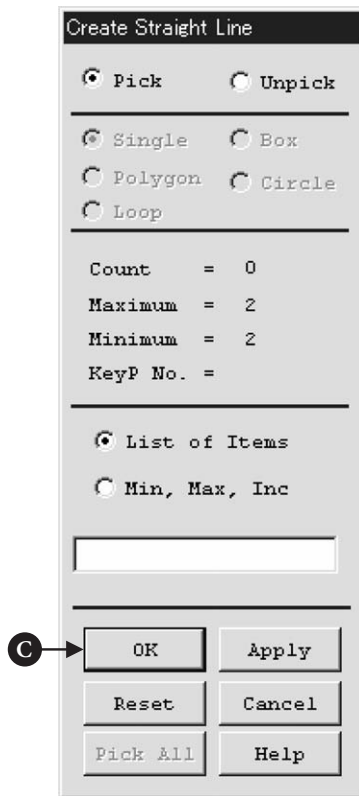
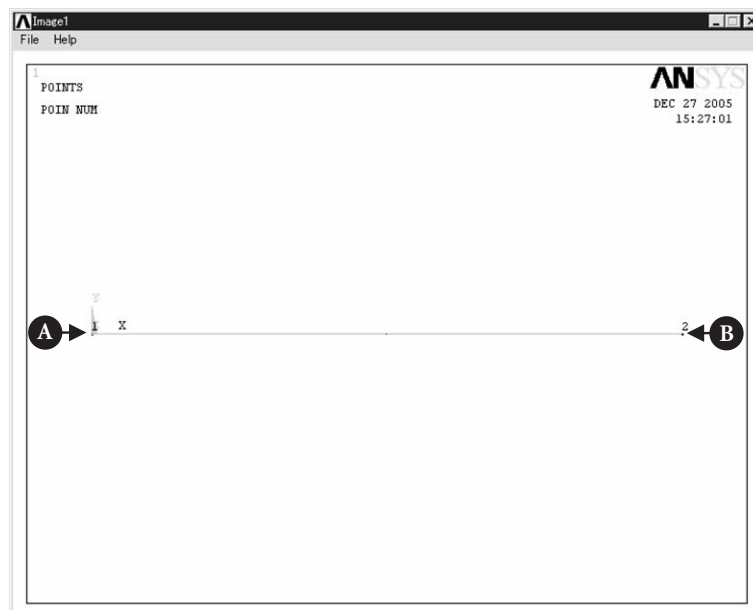**Figure 4.18** Window of **Create Straight Line**.

Figure 4.19 **ANSYS Graphics** window.

The window **Element Sizes on All Selected Lines**, as shown in Figure 4.20, is opened.

(1) Input [A] the number of **20** to **NDIV** box. This means that a line is divided into 20 elements.
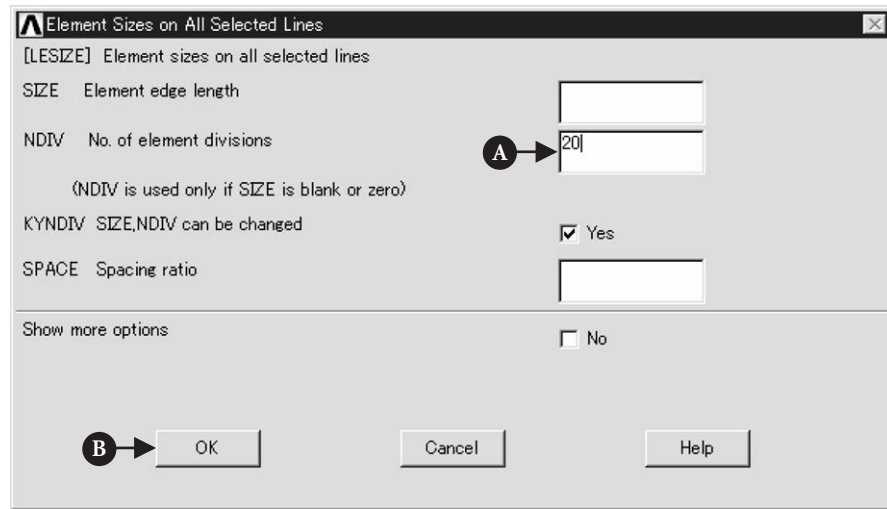
(2) Click [B] **OK** button and close the window.

Figure 4.20   Window of **Element Sizes on All Selected Lines**.

From **ANSYS Graphics** window, the preview of the divided line is available, as shown in Figure 4.21, but the line is not really divided at this stage.

COMMAND | **ANSYS Main Menu → Preprocessor → Meshing → Mesh → Lines**

The window **Mesh Lines**, as shown in Figure 4.22, opens.

(1) Click [C] the line shown in **ANSYS Graphics** window and, then, [D] **OK** button to finish dividing the line.

4.2.3.7   BOUNDARY CONDITIONS

The left end of nodes is fixed in order to constrain the left end of the cantilever beam.

COMMAND | **ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Displacement → On Nodes.**
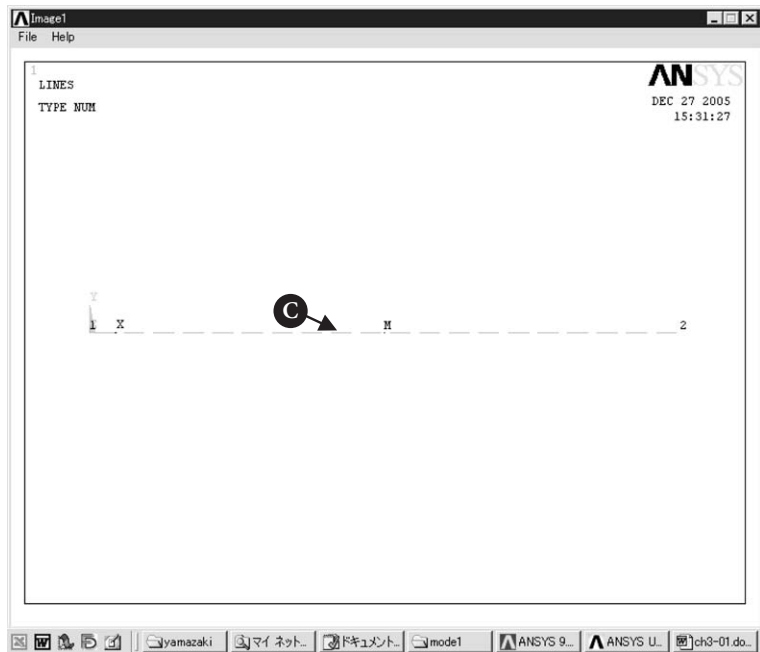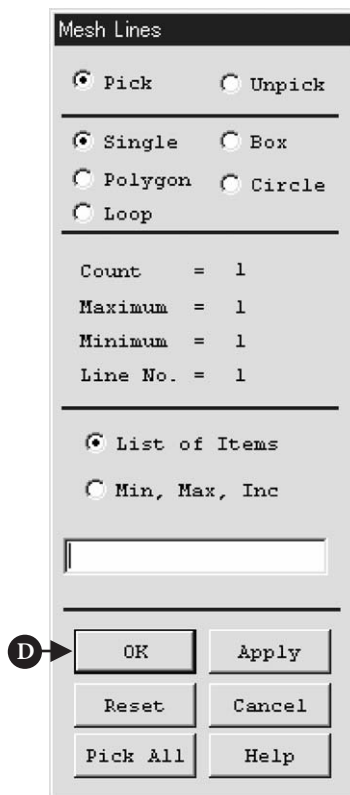
Figure 4.21 Preview of the divided line.

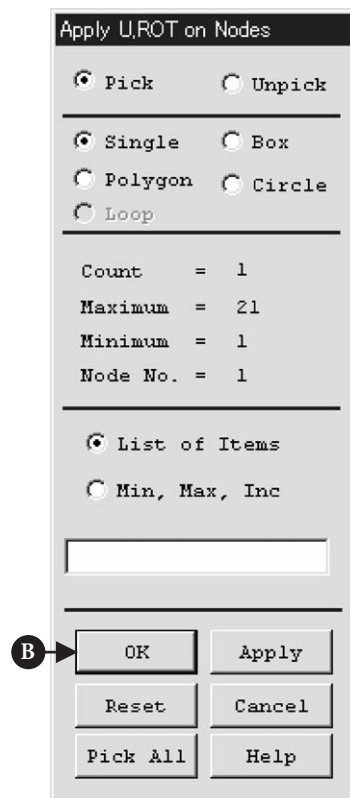**Figure 4.22** Window of **Mesh Lines**.

Figure 4.23 Window of **Apply U,ROT on Nodes**.

The window **Apply U,ROT on Nodes**, as shown in Figure 4.23, opens.

(1) Pick [A] the node at the left end in Figure 4.24 and click [B] **OK** button. Then the window **Apply U,ROT on Nodes** as shown in Figure 4.25 opens.
(2) In order to set the boundary condition, select [C] **All DOF** in the box **Lab2**. In the box **VALUE** [D] input **0** and, then, click [E] **OK** button. After these steps, **ANSYS Graphics** window is changed as shown in Figure 4.26.
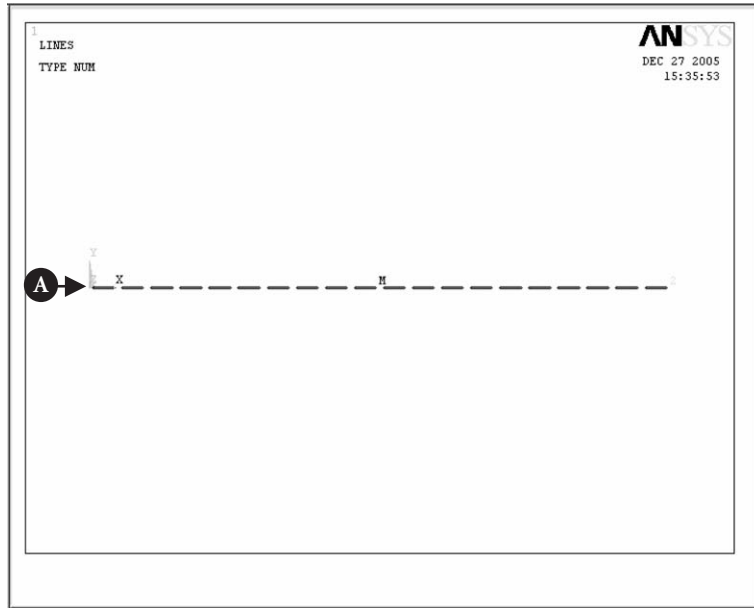

Figure 4.24   **ANSYS Graphics** window.
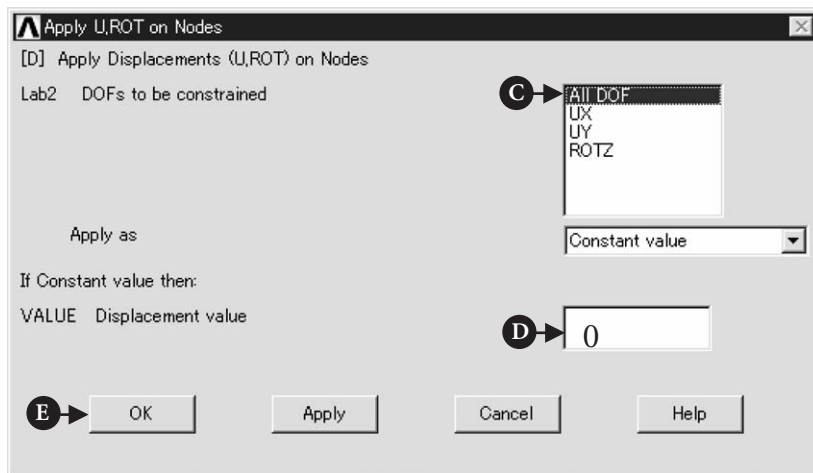

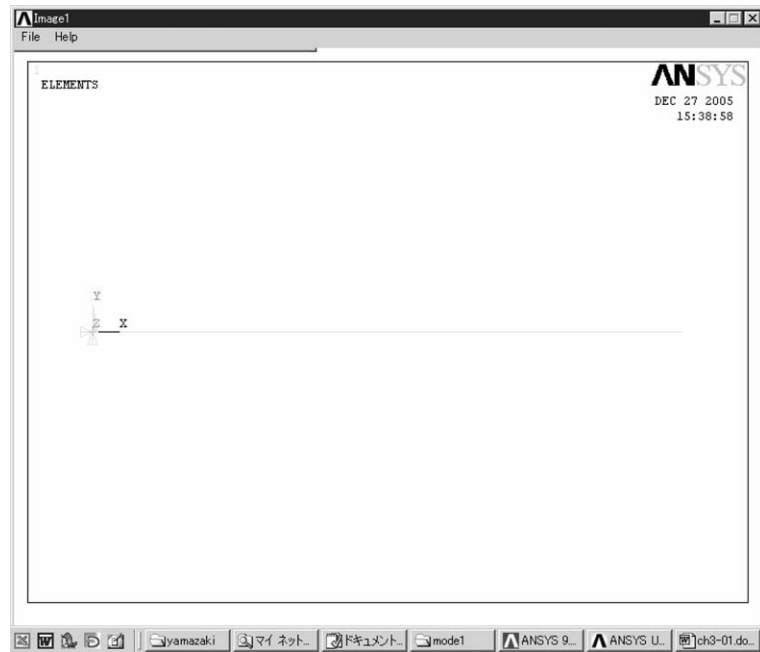**Figure 4.25**   Window of **Apply U,ROT on Nodes**.

Figure 4.26 Window after the boundary condition was set.

4.2.4 EXECUTION OF THE ANALYSIS

4.2.4.1 DEFINITION OF THE TYPE OF ANALYSIS

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

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

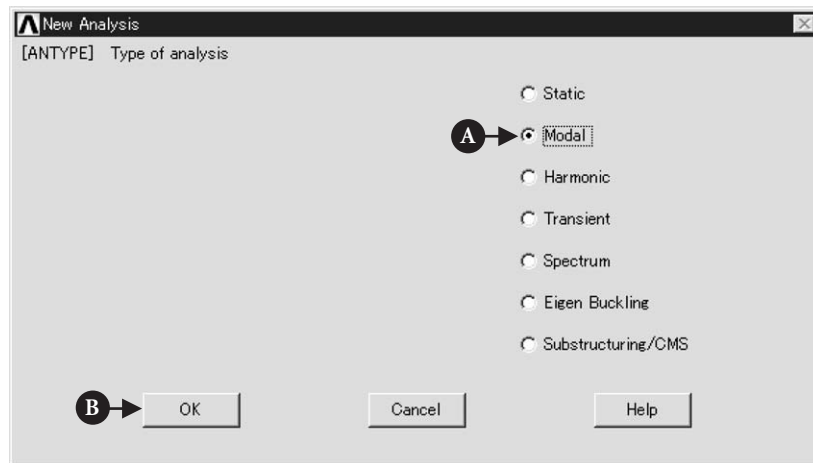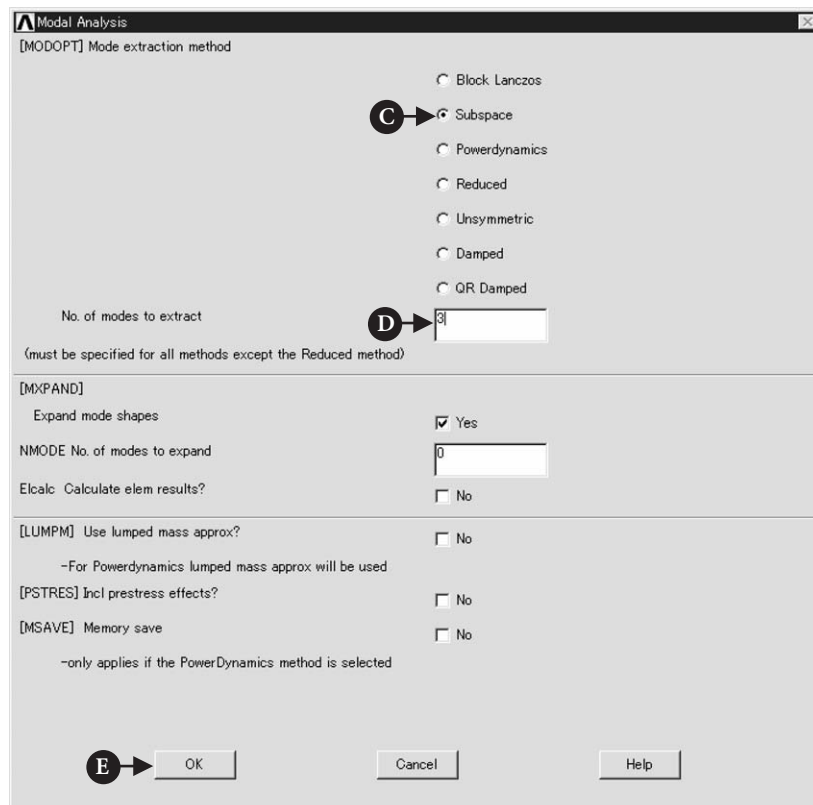The window **New Analysis**, as shown in Figure 4.27, opens.

(1) Check [A] **Modal** and, then, click [B] **OK** button.

In order to define the number of modes to extract, the following procedure is followed.

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

The window **Modal Analysis**, as shown in Figure 4.28, opens.

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

Figure 4.27 Window of **New Analysis**.

Figure 4.28 Window of **Modal Analysis**.

(2) Then, the window **Subspace Modal Analysis**, as shown in Figure 4.29, opens. Input [F] **10000** in the box of **FREQE** and click [G] **OK** button.
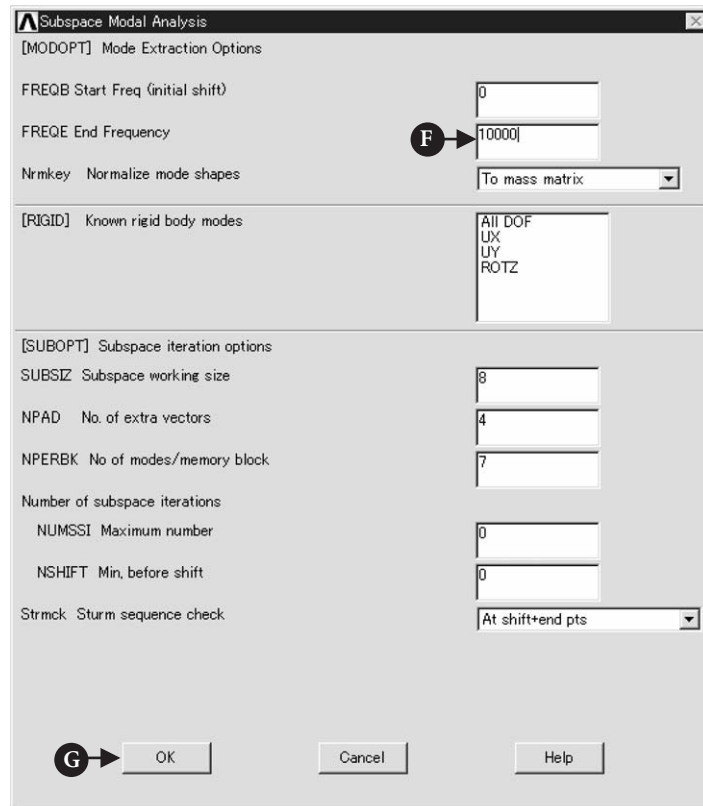

Figure 4.29   Window of **Subspace Modal Analysis**.

4.2.4.2 EXECUTE CALCULATION

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

The window **Solve Current Load Step**, as shown in Figure 4.30, opens.

(1) Click [A] **OK** button to initiate calculation. When the window **Note**, as shown in Figure 4.31 appears, the calculation is finished.

(2) Click [B] **Close** button and to close the window. The window **/STATUS Command**, as shown in Figure 4.32, also opens but this window can be closed by clicking [C] the mark **X** at the upper right-hand corner of the window.
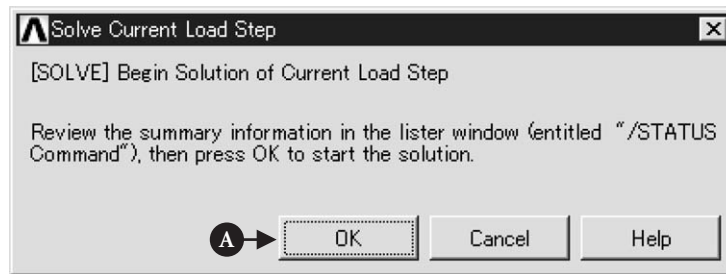
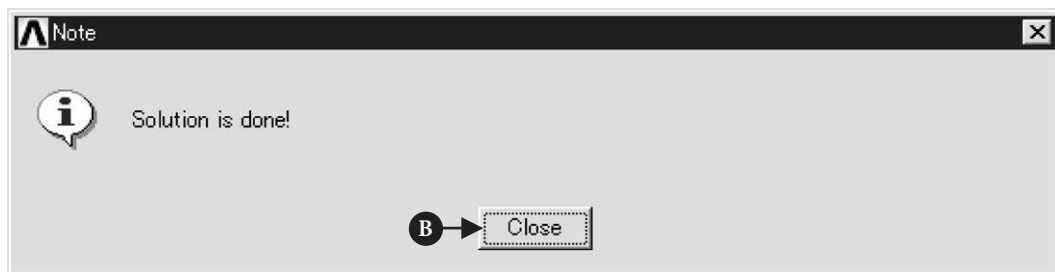**Figure 4.30** Window of **Solve Current Load Step**.

Figure 4.31 Window of **Note**.


```
           S O L U T I O N   O P T I O N S

PROBLEM DIMENSIONALITY. . . . . . . . . . . . .2-D
DEGREES OF FREEDOM. . . . . .UX   UY   ROTZ
ANALYSIS TYPE . . . . . . . . . . . . . . . . .MODAL
    EXTRACTION METHOD. . . . . . . . . . . . .SUBSPACE
NUMBER OF MODES TO EXTRACT. . . . . . . . . . .    3
MODAL EXTRACTION RANGE. . . . . . . . . . . . . 0.0000    TO   10000.
SUBSPACE OPTIONS
    SUBSPACE WORKING SIZE. . . . . . . . . . .    8
    NUMBER OF EXTRA VECTORS. . . . . . . . . .    4
    NUMBER OF MODES PER MEMORY BLOCK . . . . .    7
    MAXIMUM NUMBER OF SUBSPACE ITERATIONS. . .   100
    NUMBER OF ITERATIONS PER SHIFT . . . . . .    5
GLOBALLY ASSEMBLED MATRIX . . . . . . . . . . .SYMMETRIC

           L O A D   S T E P   O P T I O N S
```
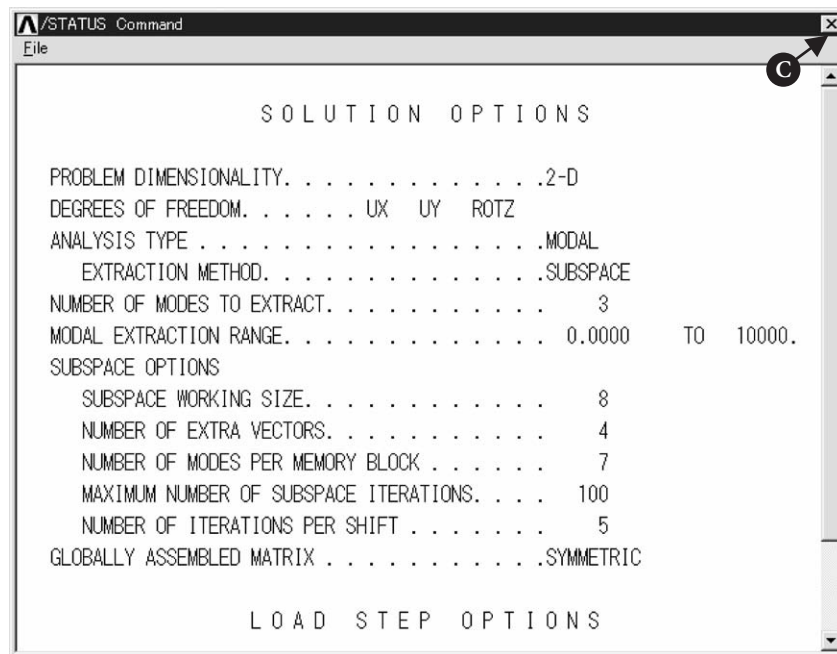
Figure 4.32 Window of **/STATUS Command**.

4.2.5 POSTPROCESSING

4.2.5.1 READ THE CALCULATED RESULTS OF THE FIRST MODE OF VIBRATION

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

4.2.5.2 PLOT THE CALCULATED RESULTS

COMMAND | **ANSYS Main Menu → General Postproc → Plot Results → Deformed Shape**

The window **Plot Deformed Shape**, as shown in Figure 4.33, opens.

(1) Select [A] **Def+Undeformed** and click [B] **OK**.

(2) Calculated result for the first mode of vibration is displayed in the **ANSYS Graphics** window as shown in Figure 4.34. The resonant frequency is shown as FRQE at the upper left-hand side on the window.
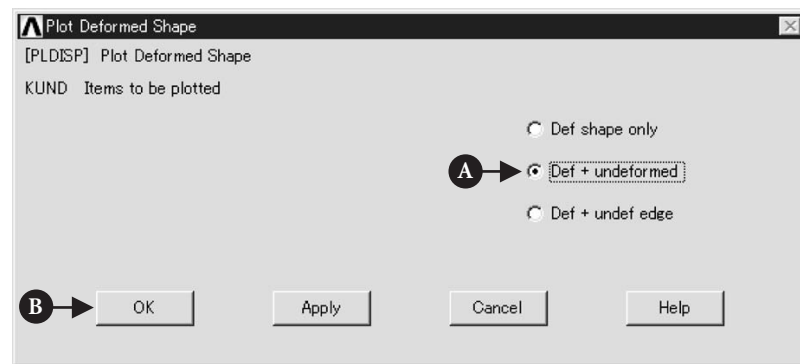
Figure 4.33 Window of **Plot Deformed Shape**.

4.2.5.3 READ THE CALCULATED RESULTS OF THE SECOND AND THIRD MODES OF VIBRATION

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

Follow the same steps outlined in Section 4.2.5.2 and calculated results for the second and third modes of vibration. Results are plotted in Figures 4.35 and 4.36. Resonant frequencies obtained by ANSYS show good agreement of those by analytical solution indicated in page 145 though they show slightly lower values.
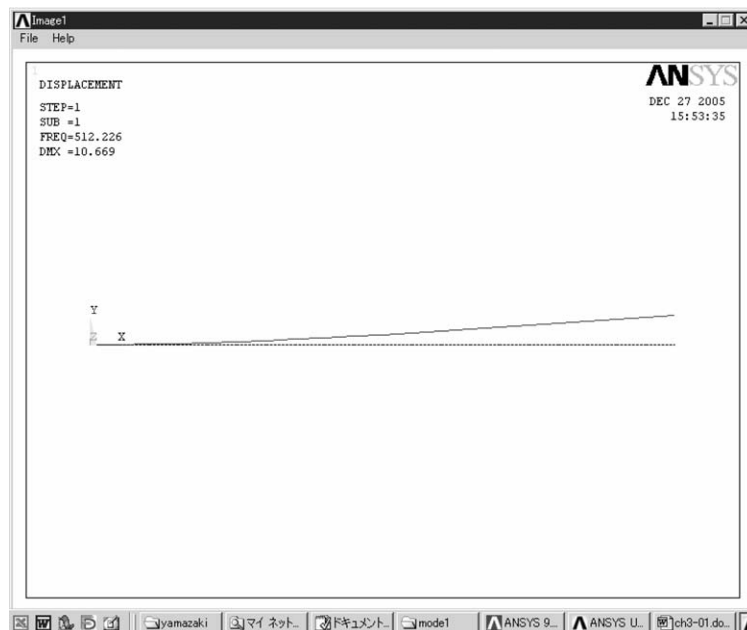
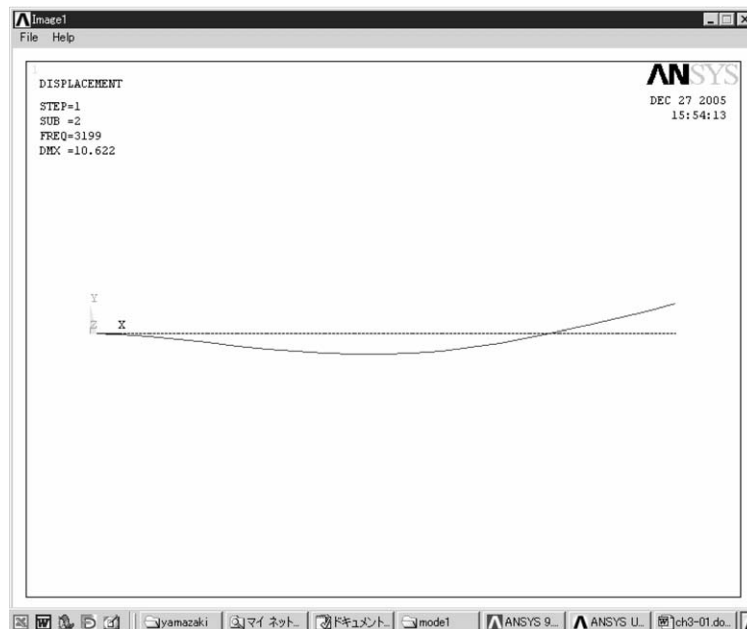**Figure 4.34** Window for the calculated result (the first mode of vibration).

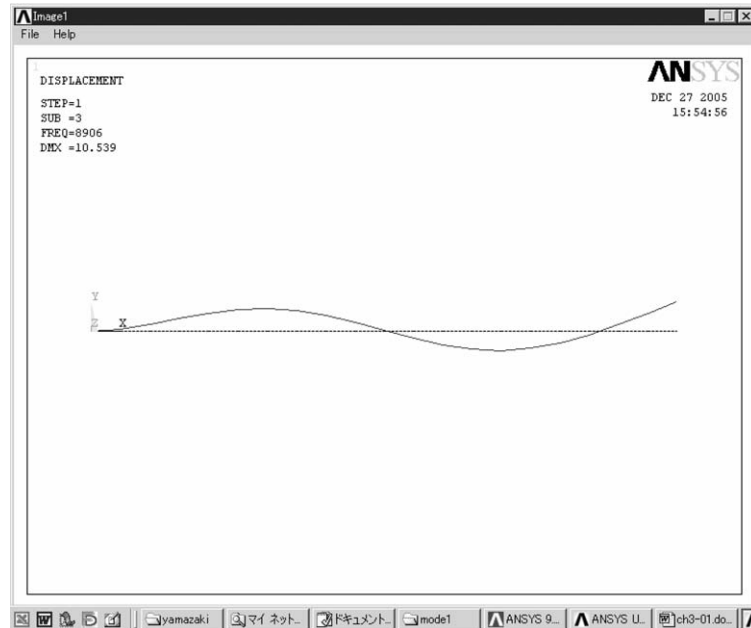**Figure 4.35** Window for the calculated result (the second mode of vibration).

Figure 4.36 Window for the calculated result (the third mode of vibration).

4.3 Mode analysis of a suspension for hard-disc drive

4.3.1 Problem description

A suspension of hard-disc drive (HDD) has many resonant frequencies with various vibration modes and it is said that the vibration mode with large radial displacement causes the tracking error. So the suspension has to be operated with frequencies of less than this resonant frequency.

Obtain the resonant frequencies and determine the vibration mode with large radial displacement of the HDD suspension as shown in Figure 4.37:

- Material: Steel, thickness of suspension: 0.05×10^{-3}(m)
- Young's modulus, $E = 206$ GPa, Poisson's ratio $\nu = 0.3$
- Density $\rho = 7.8 \times 10^3$ kg/m^3
- Boundary condition: All freedoms are constrained at the edge of a hole formed in the suspension.

4.3.2 Create a model for analysis

4.3.2.1 Element type selection

In this example, the two-dimensional elastic shell is selected for calculations as shown in Figure 4.37(c). Shell element is very suitable for analyzing the characteristics of thin material.
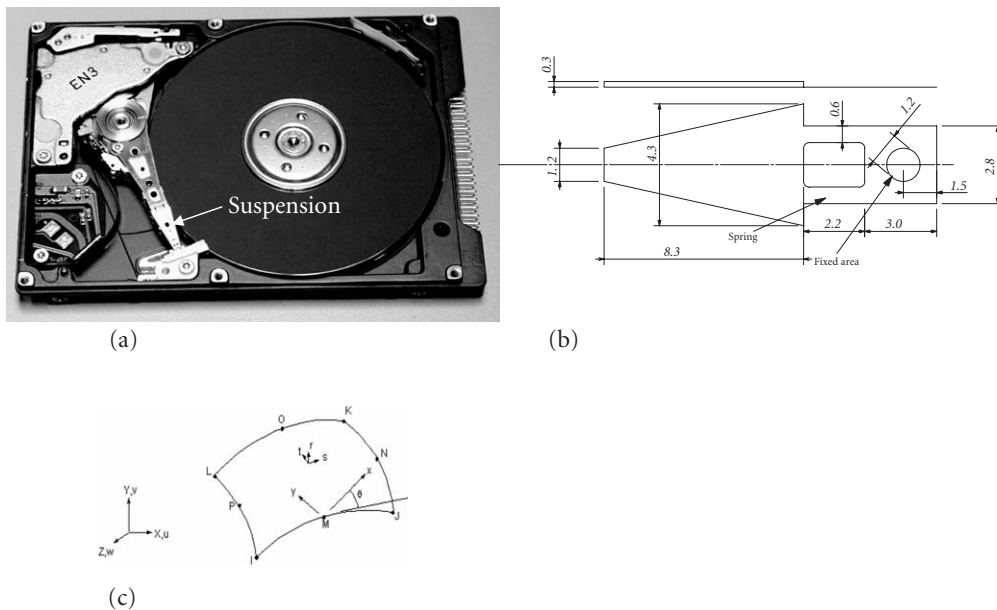
(a)   (b)

(c)

Figure 4.37   Mode analysis for an HDD suspension: (a) HDD (referenced from http://www-5.ibm.com/es/press/fotos/.informaticapersonal/i/hdd.jpg); (b) drawing of the analyzed suspension; and (c) shell element.

Figure 4.38   Window of **Element Types**.

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

Then the window **Element Types**, as shown in Figure 4.38, is opened.

(1) Click [A] **add**. Then the window **Library of Element Types**, as shown in Figure 4.39, opens.
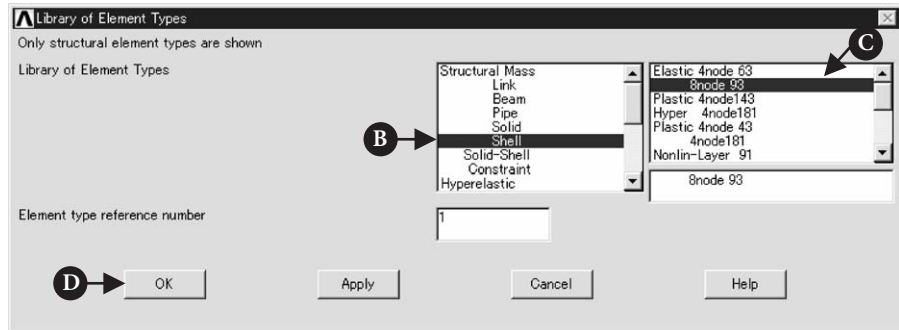
Figure 4.39   Window of **Library of Element Types**.

(2) Select [B] **Shell** in the table of **Library of Element Types** and, then, select [C] **Elastic 8node 93**.

(3) **Element type reference number** is set to **1**. Click [D]**OK** button to close the window **Library of Element Types**.

(4) Click [E] **Close** button in the window of Figure 4.40.

4.3.2.2   REAL CONSTANTS FOR BEAM ELEMENT

COMMAND | **ANSYS Main Menu → Preprocessor → Real Constants → Add/Edit/ Delete → Add**

(1) The window **Real Constants** opens (Figure 4.41). Click [A]**add** button, and the window **Element Type for Real Constants** appears in which the name of element type selected is listed as shown in Figure 4.42.

(2) Click [B] **OK** button to input the values of real constants and the window **Real Constant Set Number1, for Shell** opens (Figure 4.43).

(3) Input the following values in Figure 4.43: [C] shell thickness at nodes I, J, K, and L = 0.05e−3. After inputting these values, click [D] **OK** button to close the window.

(4) Click [E] **Close** button in the window of **Real Constants** (Figure 4.44).

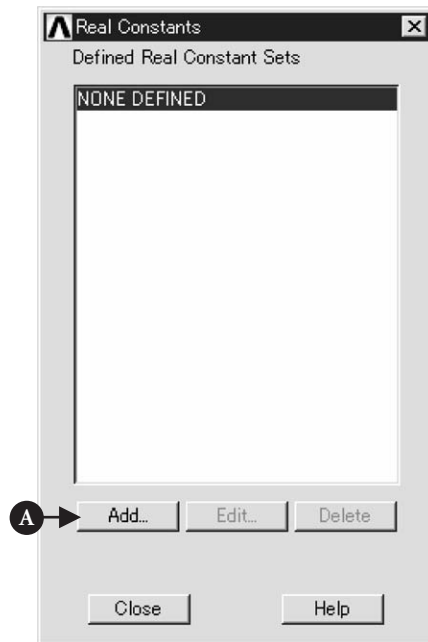**Figure 4.40** Window of **Library of Element Types**.
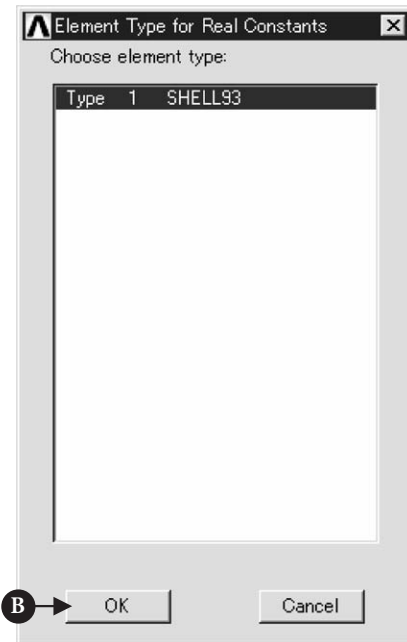

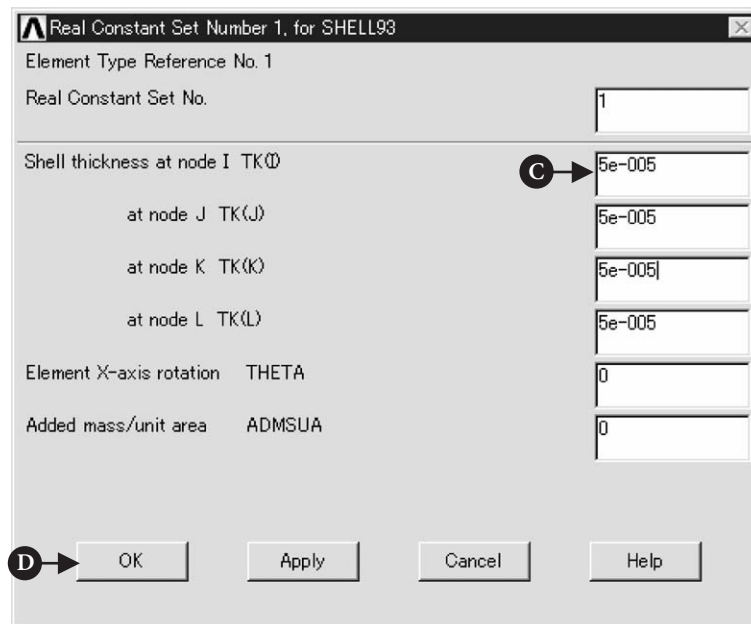**Figure 4.41** Window of **Real Constants**.

Figure 4.42 Window of **Real Constants**.

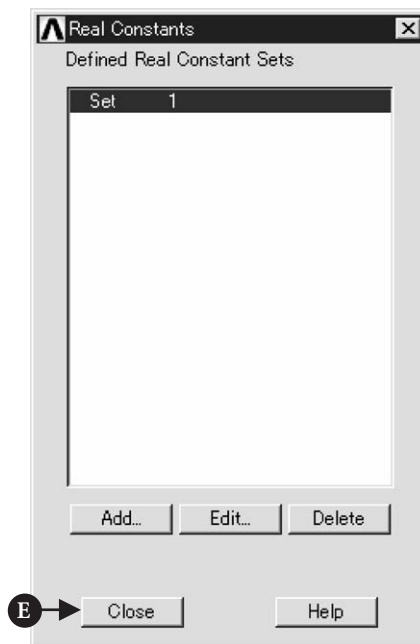**Figure 4.43**   Window of **Real Constants Set Number 1, for Shell**.

Figure 4.44   Window of **Real Constants**.

4.3.2.3   MATERIAL PROPERTIES

This section describes the procedure of defining the material properties of shell element.

COMMAND | ANSYS Main Menu → Preprocessor → Material Props → Material Models

(1) Click the above buttons in order and the window **Define Material Model Behavior**, as shown in Figure 4.12, opens .

(2) Double click the following terms in the window:

    Structural → Linear → Elastic → Isotropic

    Then the window of **Linear Isotropic Properties for Material Number 1** opens.

(3) Input Young's modulus of **206e9** to [A] **EX** box and Poisson ratio of **0.3** to [B] **PRXY** box. Then click [C] **OK** button (Figure 4.45).

    Next, define the value of density of material.

(1) Double click the term **Density** and the window **Density for Material Number 1** opens (Figure 4.46).

(2) Input the value of Density, **7800** to [D] **DENS** box and click [E] **OK** button (Figure 4.46). Finally close the window **Define Material Model Behavior** by clicking **X** mark at the upper right end.

4.3.2.4   CREATE KEYPOINTS

To draw a suspension for the analysis, the method of keypoints is described in this section.
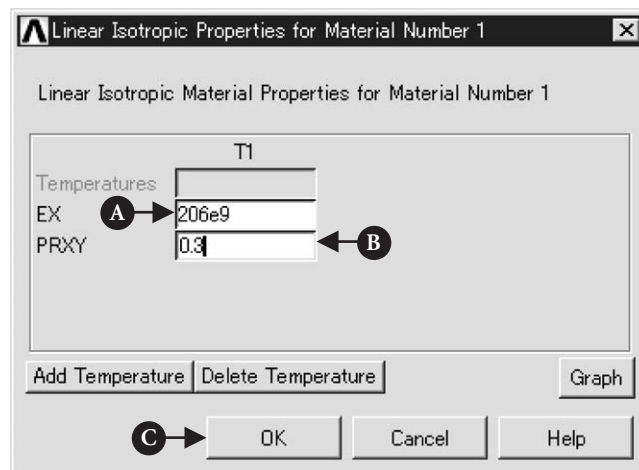

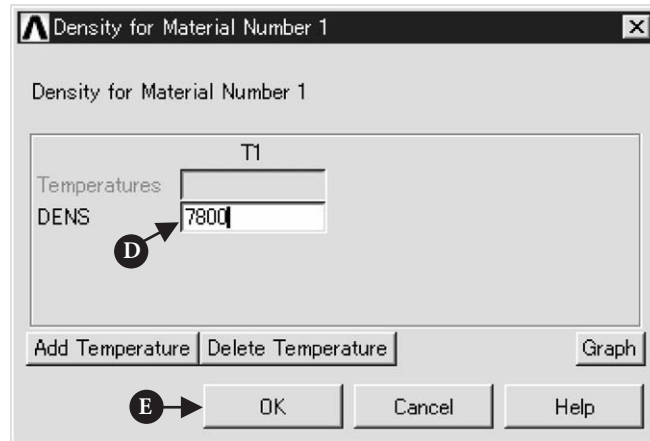**Figure 4.45**   Window of **Linear Isotropic Properties for Material Number 1**.
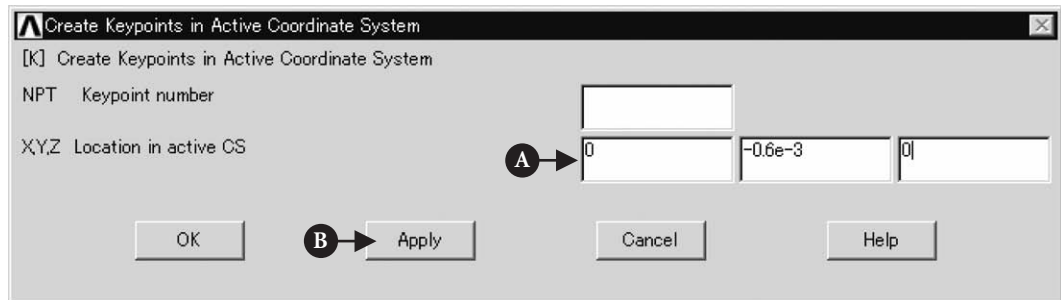
Figure 4.46   Window of **Density for Material Number 1**.


Figure 4.47   Window of **Create Keypoints in Active Coordinate System**.

C O M M A N D | **ANSYS Main Menu → Preprocessor → Modeling → Create → Keypoints → In Active CS**

(1) The window **Create Keypoints in Active Coordinate System** opens (Figure 4.47).

(2) Input [A] **0, −0.6e−3, 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 a figure is not inputted into [C] **NPT Key Point number** box, the number of keypoint is automatically assigned in order.

(3) In the same window, input the values as shown in Table 4.1 in order. When all values are inputted, click **OK** button.

(4) Then all inputted keypoints appear on **ANSYS Graphics** window as shown in Figure 4.48.

Table 4.1   X, Y, and Z coordinates of keypoints for suspension

KP No.	X	Y	Z
1	0	−0.6e−3	
2	8.3e−3	−2.15e−3	
3	8.3e−3	−1.4e−3	
4	13.5e−3	−1.4e−3	
5	13.5e−3	1.4e−3	
6	8.3e−3	1.4e−3	
7	8.3e−3	2.15e−3	
8	0	0.6e−3	
9	8.3e−3	−0.8e−3	
10	10.5e−3	−0.8e−3	
11	10.5e−3	0.8e−3	
12	8.3e−3	0.8e−3	
13	0	−0.6e−3	0.3e−3
14	8.3e−3	−2.15e−3	0.3e−3
15	8.3e−3	2.15e−3	0.3e−3
16	0	0.6e−3	0.3e−3
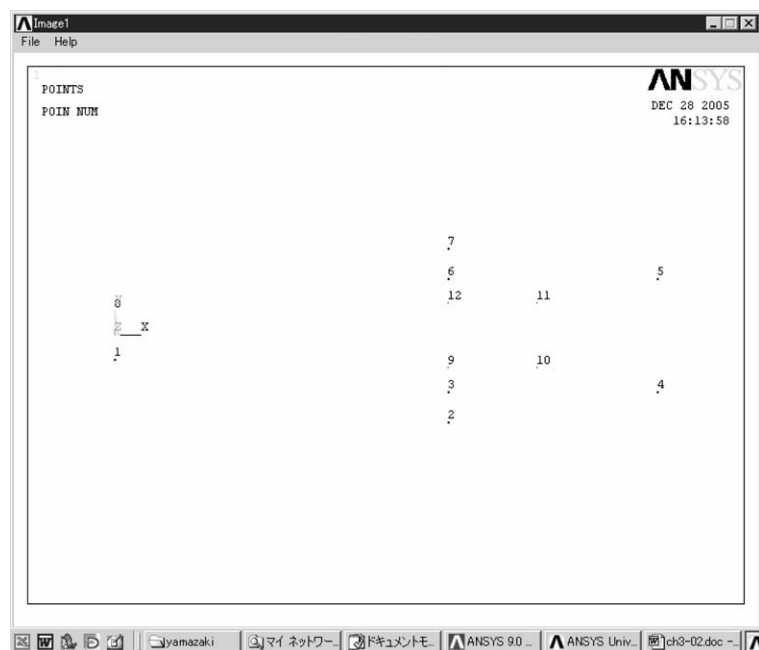
Figure 4.48   **ANSYS Graphics** window.

4.3.2.5   CREATE AREAS FOR SUSPENSION

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 (Figure 4.49).

(2) Pick the keypoints, [A] **9**, **10**, **11**, and **12** in Figure 4.50 in order and click [B] **Apply** button in Figure 4.49. An area is created on the window as shown in Figure 4.51.

(3) By performing the same steps, other areas are made on the window. Click keypoints listed in Table 4.2 and make other areas. When you make area No. 3 and 4, you have to rotate the drawing of suspension, using **PlotCtrls—Pan-Zoom-Rotate** in Utility Menu.
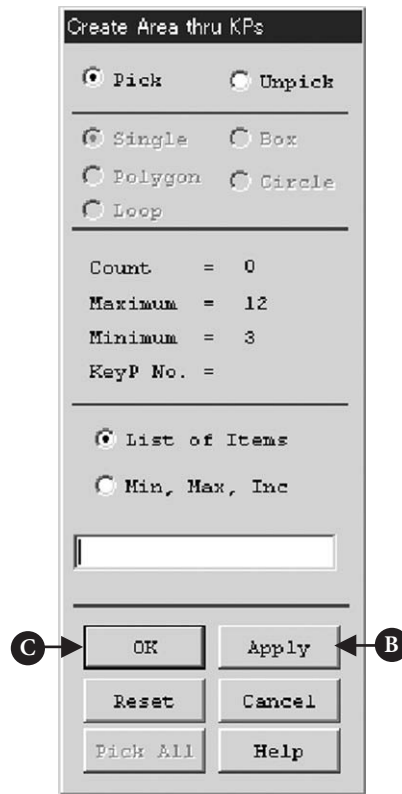
Figure 4.49   Window of **Create Area thru KPs**.
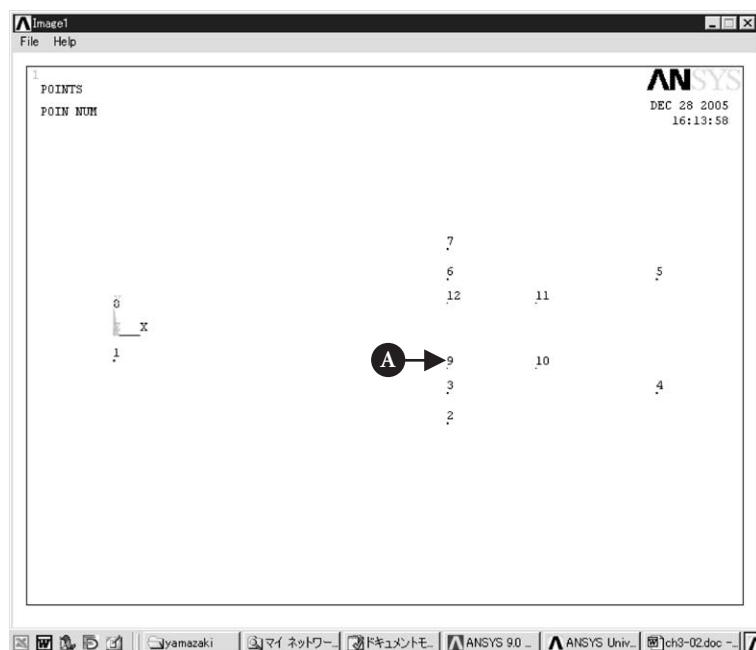
Figure 4.50 **ANSYS Graphics** window.
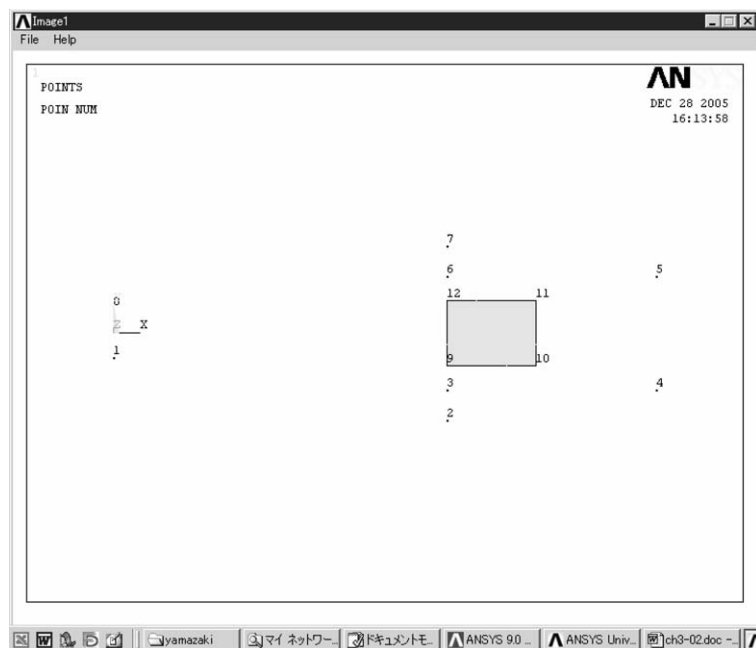
Figure 4.51 **ANSYS Graphics** window.

Table 4.2   Keypoint numbers for making areas of a suspension

Area No.	Keypoint number
1	9, 10, 11, 12
2	1, 2, 3, 4, 5, 6, 7, 8
3	1, 2, 14, 13
4	8, 7, 15, 16
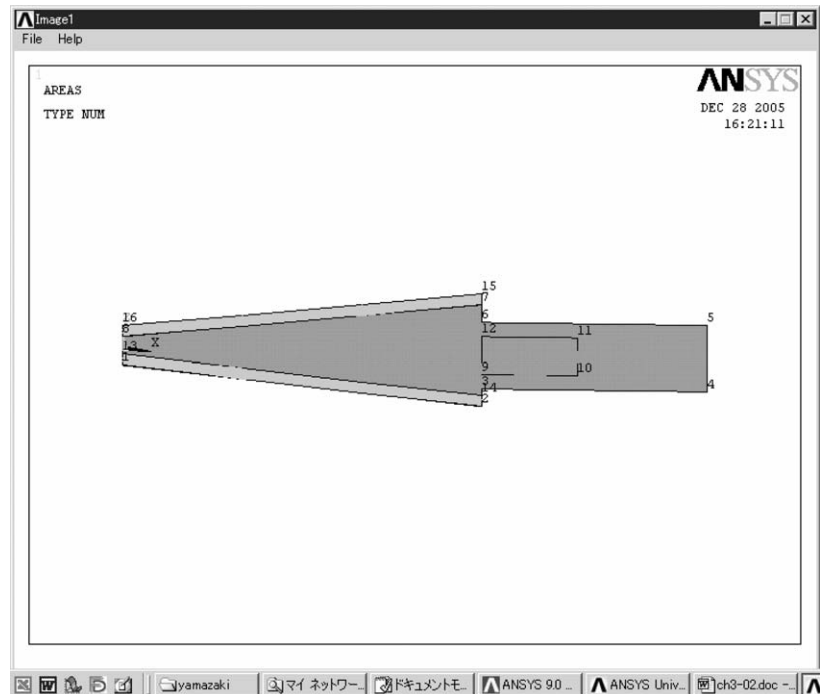
Figure 4.52   **ANSYS Graphics** window.

(4) When all areas are made on the window, click [C] **OK** button in Figure 4.49. Then the drawing of the suspension in Figure 4.52 appears.

C O M M A N D   **ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Circle → Solid Circle**

The window **Solid Circle Area** opens, Figure 4.53. Input [D]the values of **12.0e−3, 0**, **0.6e−3** to **X**, **Y** and **Radius** boxes as shown in Figure 4.53, respectively, and click [E] **OK** button. Then the solid circle is made in the drawing of suspension as shown in Figure 4.54.
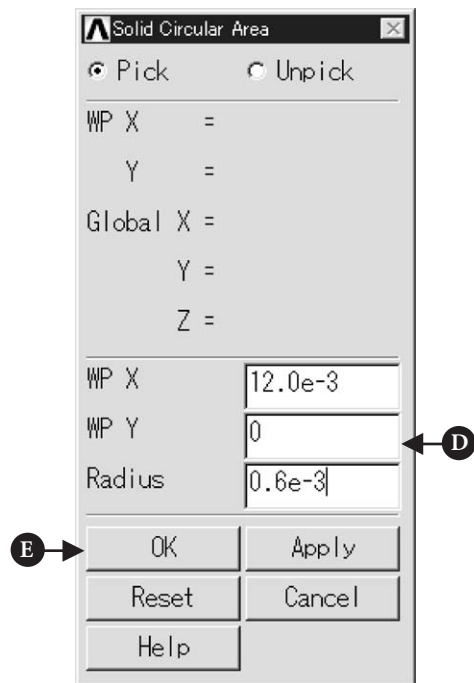
Figure 4.53   Window of **Solid Circular Area**.
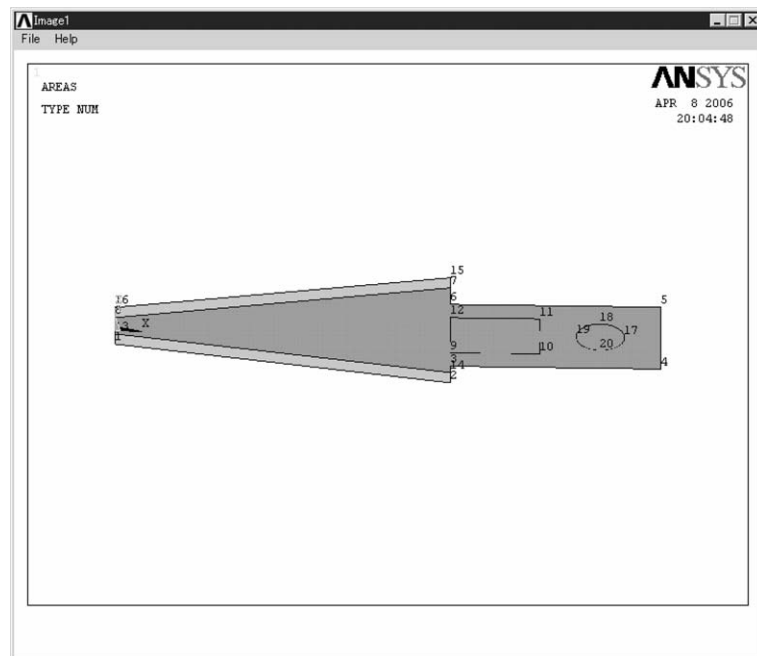
Figure 4.54   **ANSYS Graphics** window.
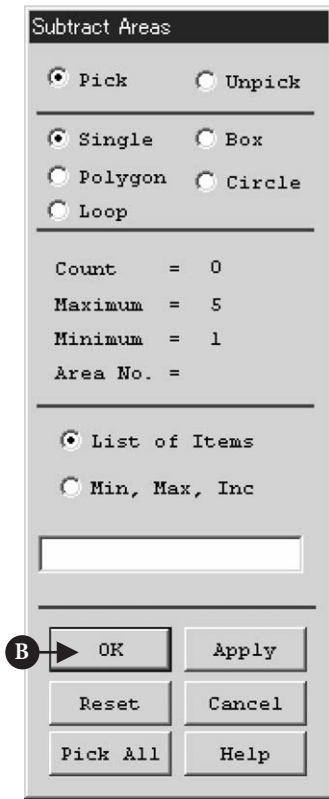
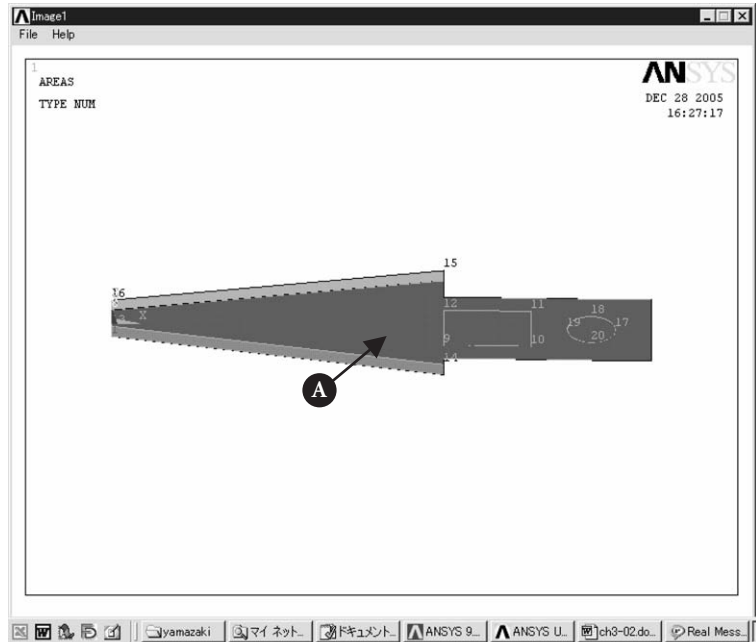**Figure 4.55**   Window of **Subtract Areas**.

Figure 4.56   **ANSYS Graphics** window.

4.3.2.6   BOOLEAN OPERATION

In order to make a spring region and the fixed region of the suspension, the rectangular and circular areas in the suspension are subtracted by Boolean operation.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Subtract → Areas**

(1) The window **Subtract Areas** opens (Figure 4.55).

(2) Click [A] the area of the suspension in **ANSYS Window** and [B] **OK** button in Figure 4.56. Then click [C] rectangular and [D] circular areas as shown in Figure 4.57 and [B] **OK** button in Figure 4.54. The drawing of the suspension appears as shown in Figure 4.58.

(3) In analysis, all areas have to be glued.
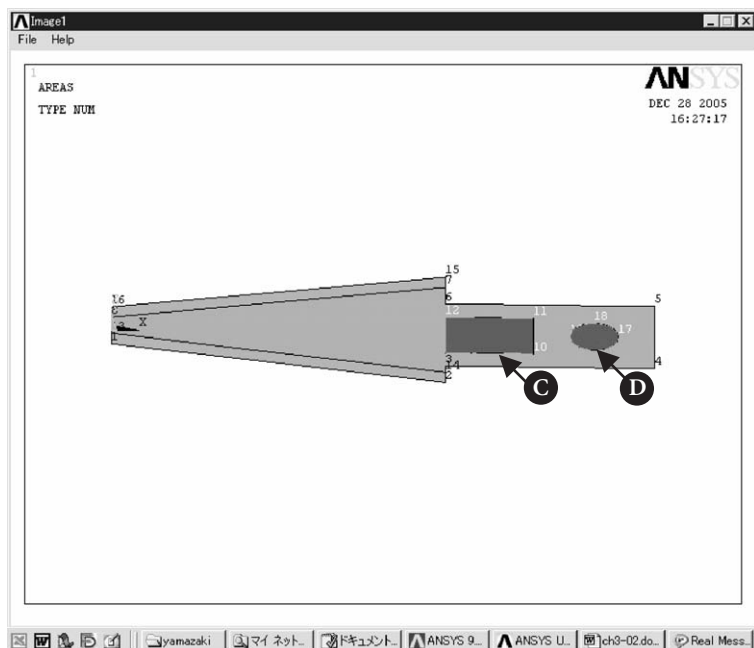
Figure 4.57 **ANSYS Graphics** window.
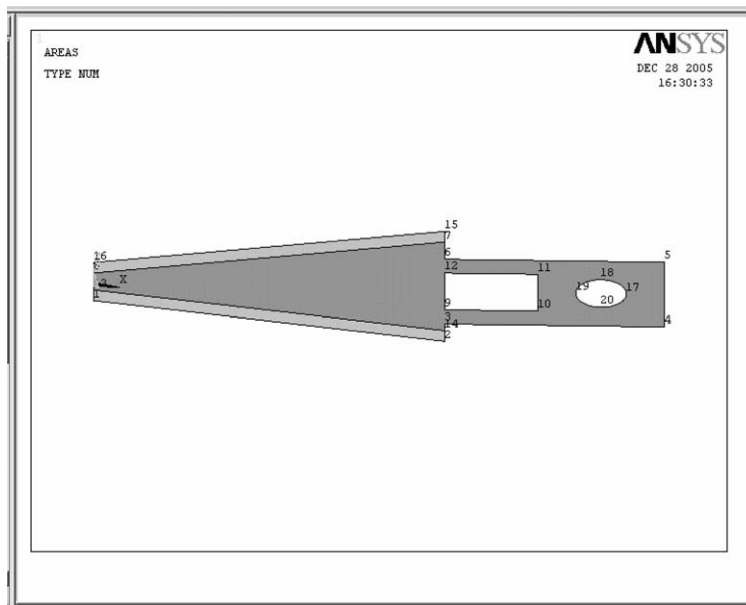
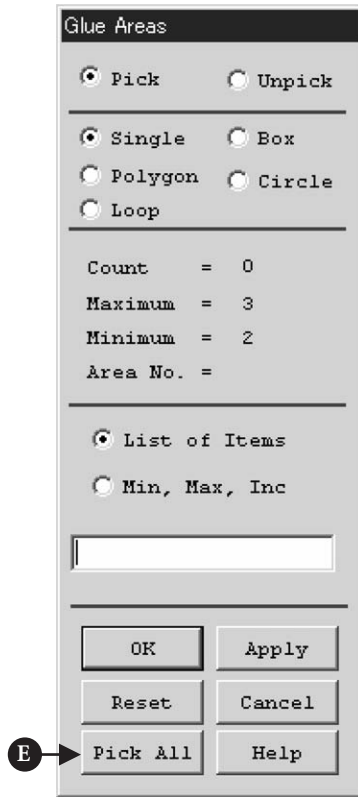**Figure 4.58** **ANSYS Graphics** window.

Figure 4.59   Window of **Glue Areas**.

C O M M A N D | **ANSYS Main Menu → Preprocessor → Modeling   → Operate → Booleans → Glue → Areas**

The window **Glue Areas** opens (Figure 4.59). Click [E] **Pick All** button.

4.3.2.7   CREATE MESH IN AREAS

C O M M A N D | **ANSYS Main Menu → Preprocessor → Meshing → Size Cntrls → Manual Size → Areas → All Areas**

The window **Element Sizes on All Selected Areas** opens (Figure 4.60).

(1) Input [A] **0.0002** to **SIZE** box. This means that areas are divided by meshes of edge length of 0.0002 m.

(2) Click [B] **OK** button and close the window.
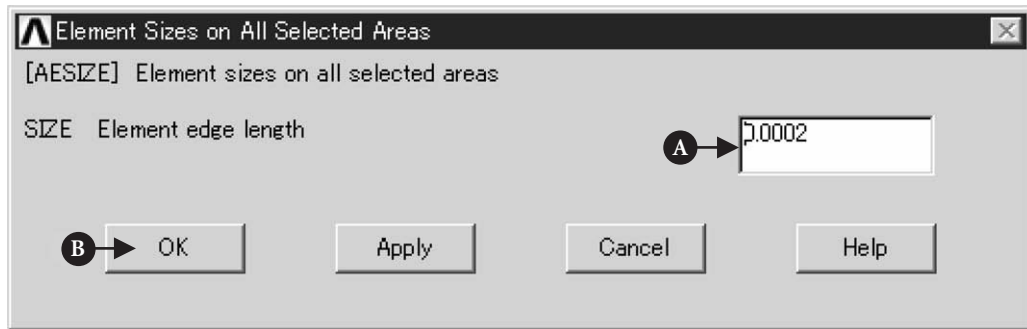
Figure 4.60 Window of **Element Sizes on All Selected Areas**.
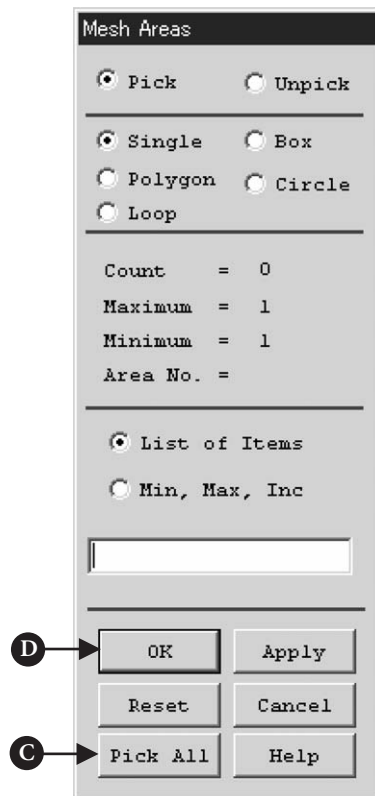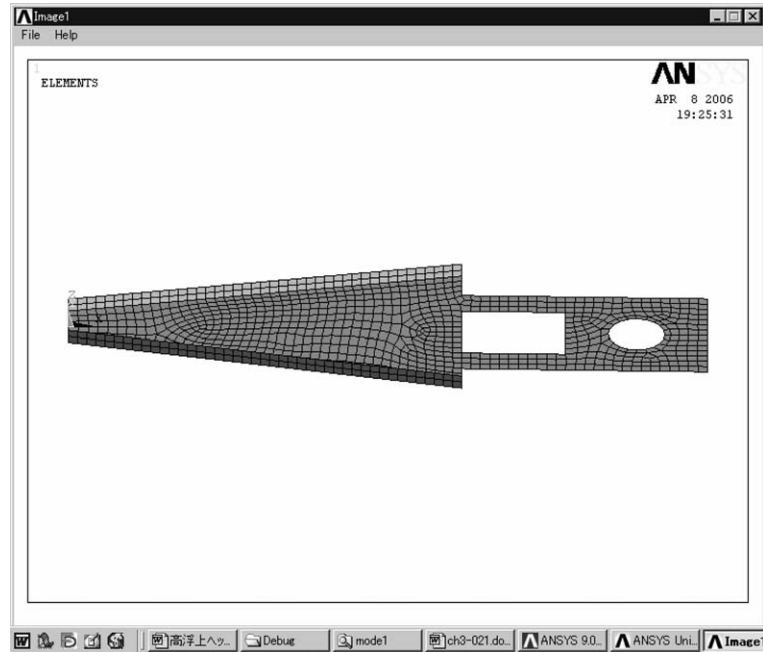

Figure 4.61 Window of **Mesh Areas**.

Figure 4.62   **ANSYS Graphics** window.

C O M M A N D | **ANSYS Main Menu → Preprocessor → Meshing → Mesh → Areas → Free**

The window **Mesh Areas** opens (Figure 4.61).

(1) Click [C] **Pick ALL** and, then, [D] **OK** button to finish dividing the areas as shown in Figure 4.62.

4.3.2.8   BOUNDARY CONDITIONS

The suspension is fixed at the edge of the circle.

C O M M A N D | **ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Displacement → On Lines**

The window **Apply U,ROT on Lines** opens (Figure 4.63).

(1) Pick [A] four lines included in the circle in Figure 4.64 and click [B] **OK** button in Figure 4.63. Then the window **Apply U,ROT on Lines** opens (Figure 4.65).

(2) In order to set the boundary condition, select [C] **All DOF** in the box of **Lab2**. Input [D] **0** to the box of **VALUE** and, then, click [E] **OK** button. After these steps, the **ANSYS Graphics** window is changed as shown in Figure 4.66.
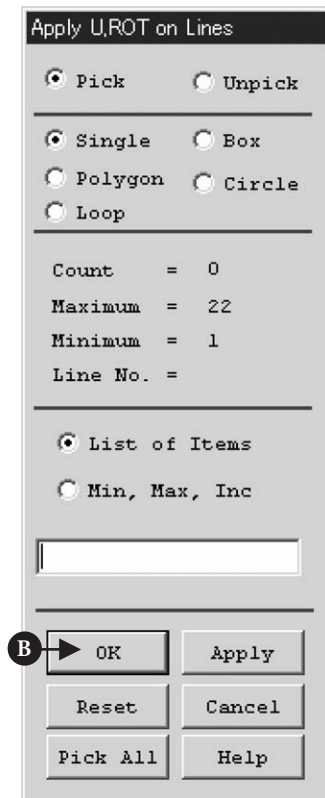
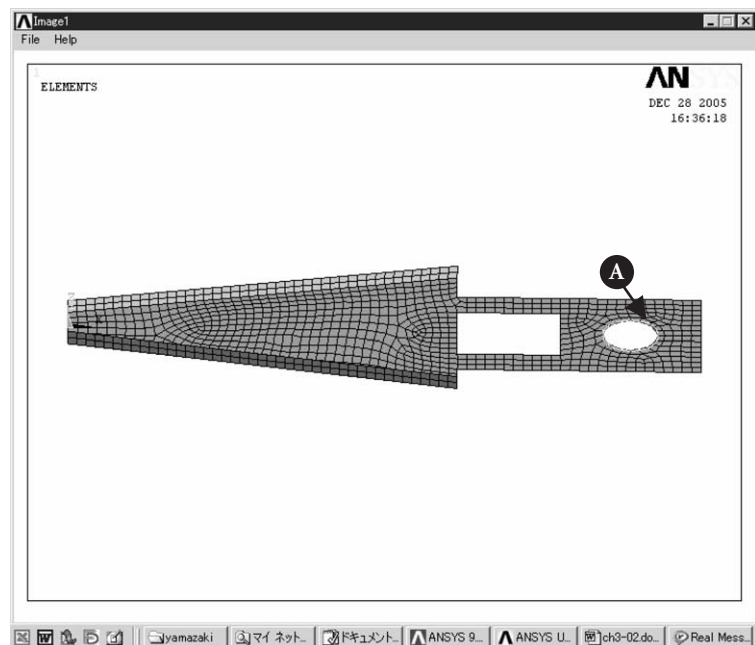**Figure 4.63**   Window of **Apply U,ROT on Lines**.
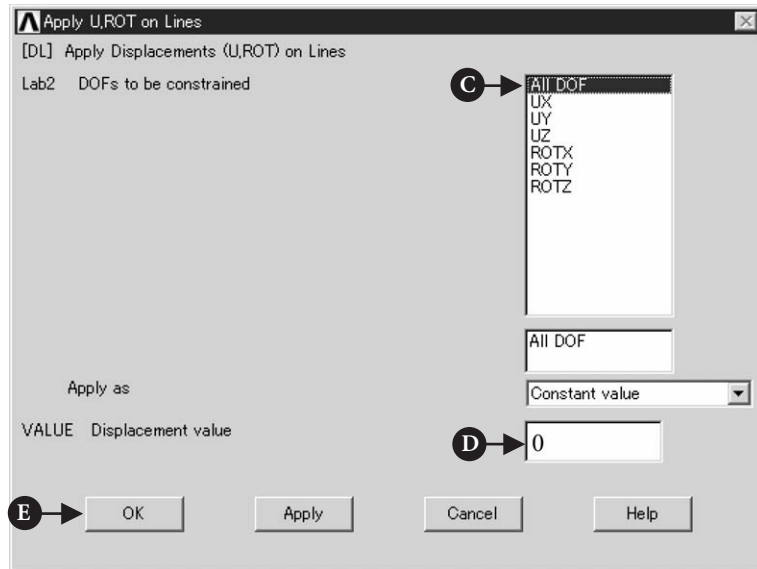

Figure 4.64   **ANSYS Graphics** window.

Figure 4.65   Window of **Apply U,ROT on Lines**.
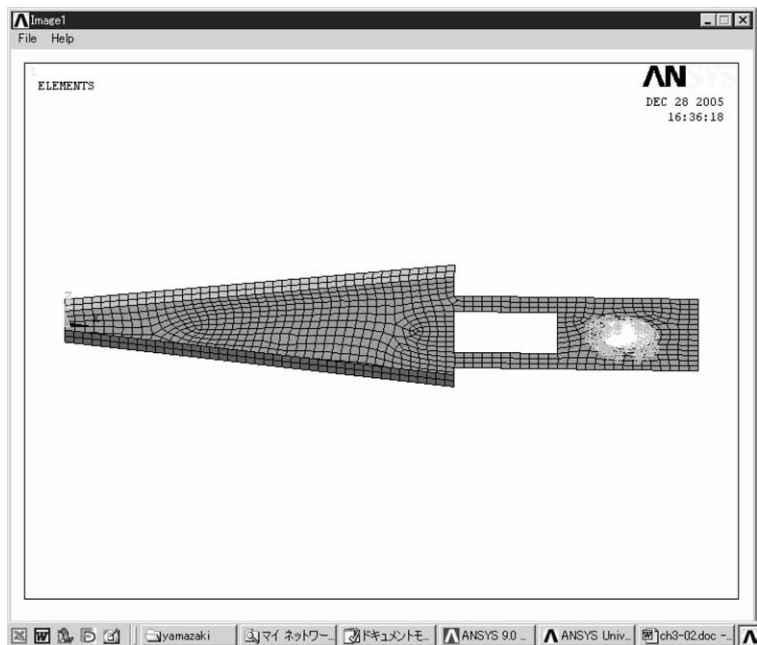

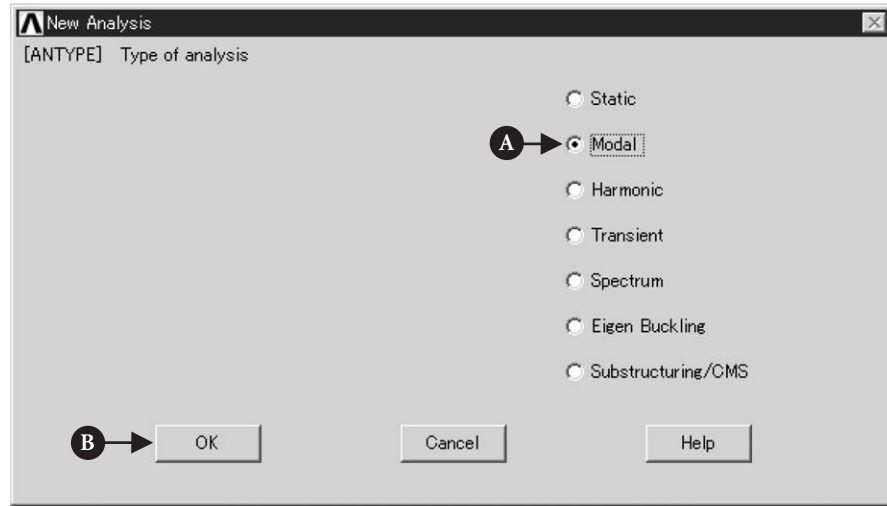**Figure 4.66**   **ANSYS Graphics** window.

Figure 4.67 Window of **New Analysis**.

4.3.3 ANALYSIS

4.3.3.1 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 4.67).

(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 4.68).

(1) Check [C]**Subspace** of **MODOPT** and input [D] **10** in the box of **No. of modes to extract** and click [E] **OK** button.

(2) Then, the window **Subspace Modal Analysis** as shown in Figure 4.69 opens. Input [F] **20000** in the box of **FREQE** and click [G] **OK** button.

4.3.3.2 EXECUTE CALCULATION

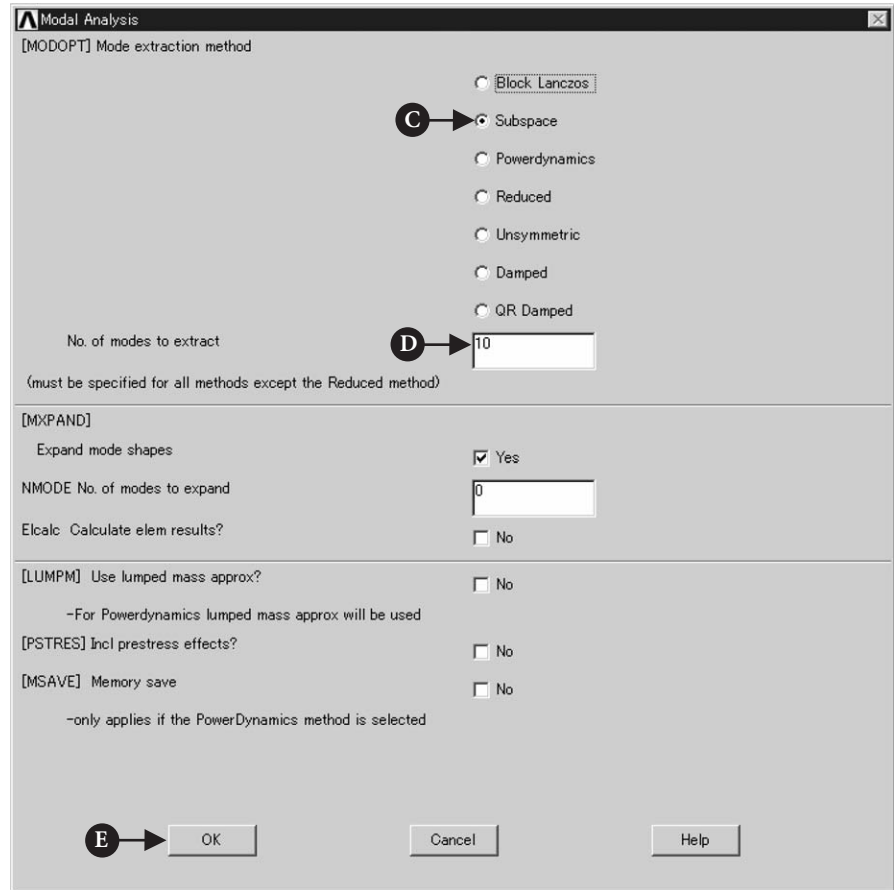COMMAND | **ANSYS Main Menu → Solution → Solve → Current LS**

Figure 4.68   Window of **Modal Analysis**.

4.3.4   POSTPROCESSING

4.3.4.1   READ THE CALCULATED RESULTS OF THE FIRST MODE OF VIBRATION

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

4.3.4.2   PLOT THE CALCULATED RESULTS

COMMAND | **ANSYS Main Menu → General Postproc → Plot Results → Deformed Shape**

The window **Plot Deformed Shape** opens (Figure 4.70).
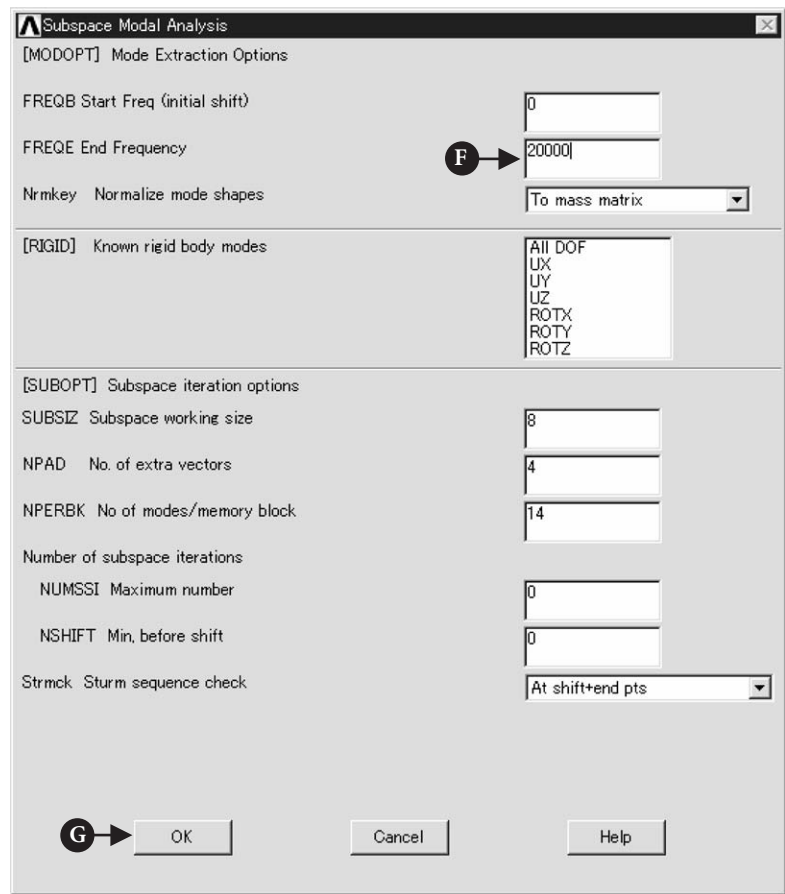
(1) Select [A] **Def+Undeformed** and click [B] **OK**.

Figure 4.69 Window of **Subspace Modal Analysis**.

(2) The calculated result for the first mode of vibration appears on **ANSYS Graphics** window as shown in Figure 4.71. The resonant frequency is shown as FRQE at the upper left side on the window.

4.3.4.3 READ THE CALCULATED RESULTS OF HIGHER MODES OF VIBRATION

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

Perform the same steps described in Section 4.2.4.2 and the calculated results from the second mode to the sixth mode of vibration are displayed on the windows as shown in Figures 4.72–4.76. From these vibration modes, it is found that a large radial displacement appears at the fifth mode of vibration.
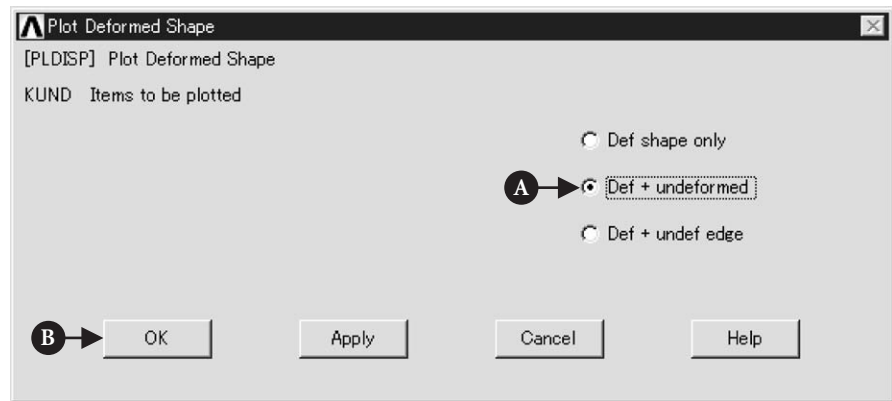
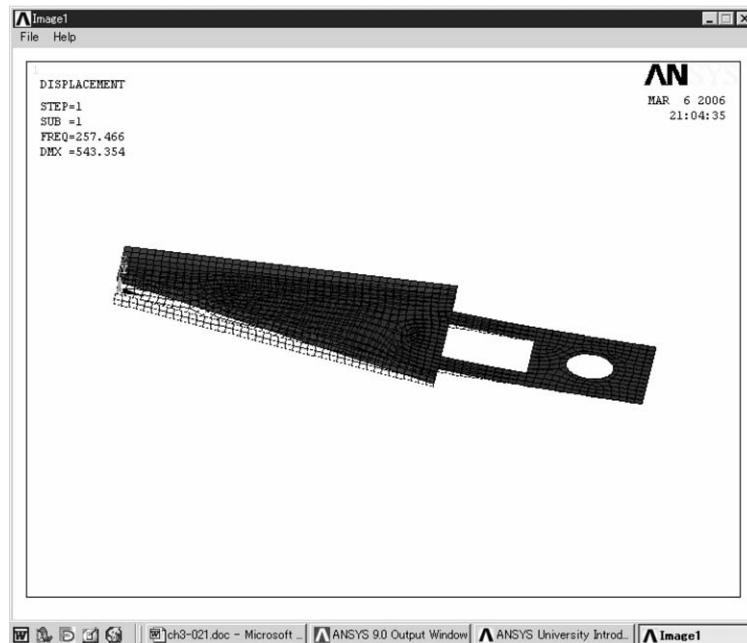**Figure 4.70**   Window of **Plot Deformed Shape**.

Figure 4.71   The first mode of vibration.
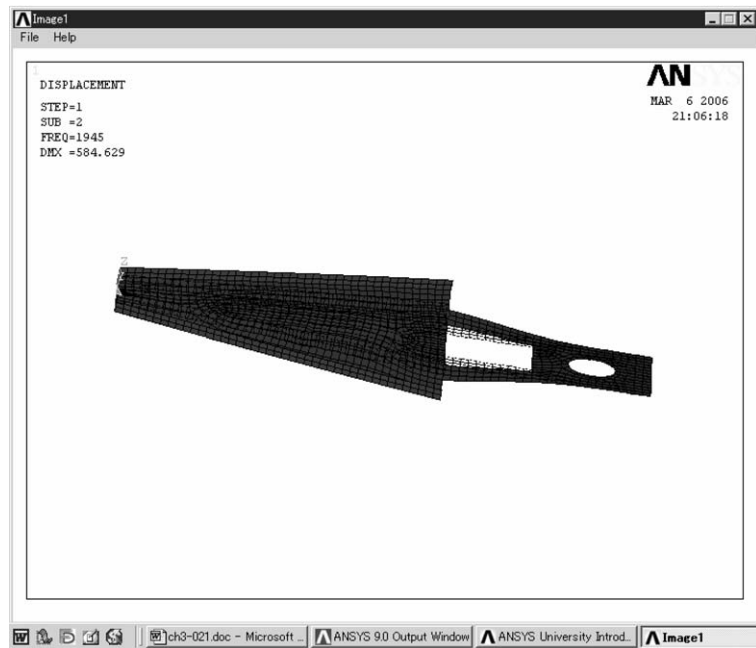
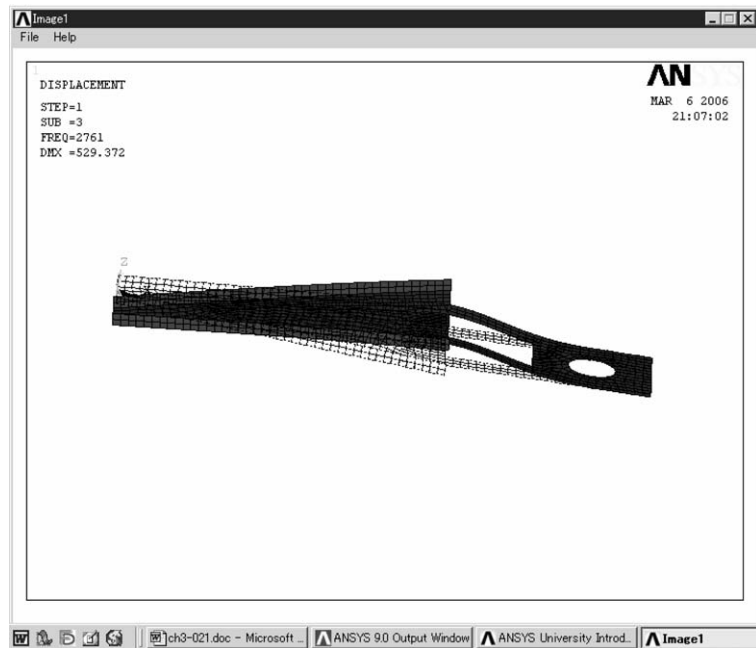**Figure 4.72**   The second mode of vibration.

Figure 4.73   The third mode of vibration.

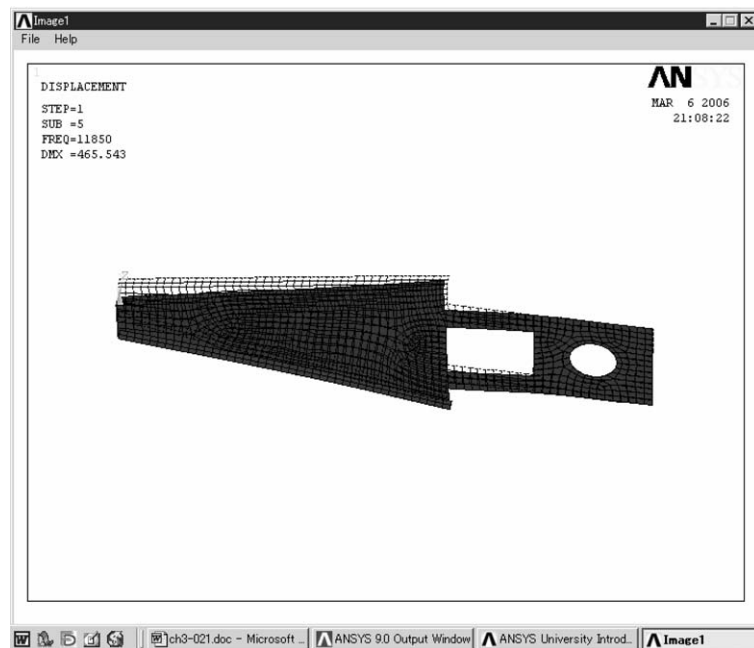**Figure 4.74**   The fourth mode of vibration.

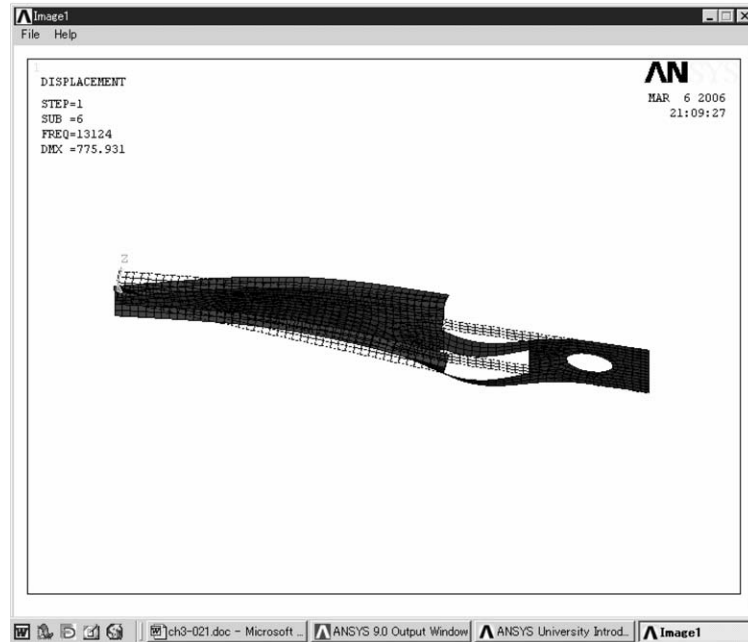**Figure 4.75**   The fifth mode of vibration.

Figure 4.76   The sixth mode of vibration.

4.4   MODE ANALYSIS OF A ONE-AXIS PRECISION MOVING TABLE USING ELASTIC HINGES

4.4.1   PROBLEM DESCRIPTION

A one-axis table using elastic hinges has been often used in various precision equipment, and the position of a table is usually controlled at nanometer-order accuracy using a piezoelectric actuator or a voice coil motor. Therefore, it is necessary to confirm the resonant frequency in order to determine the controllable frequency region.

Obtain the resonant frequency of a one-axis moving table using elastic hinges when the bottom of the table is fixed and a piezoelectric actuator is selected as an actuator:

- Material: Steel, thickness of the table: 5 mm
- Young's modulus, $E = 206$ GPa, Poisson's ratio $\nu = 0.3$
- Density $\rho = 7.8 \times 10^3$ kg/m^3
- Boundary condition: All freedoms are constrained at the bottom of the table and the region A indicated in Figure 4.77, where a piezoelectric actuator is glued.
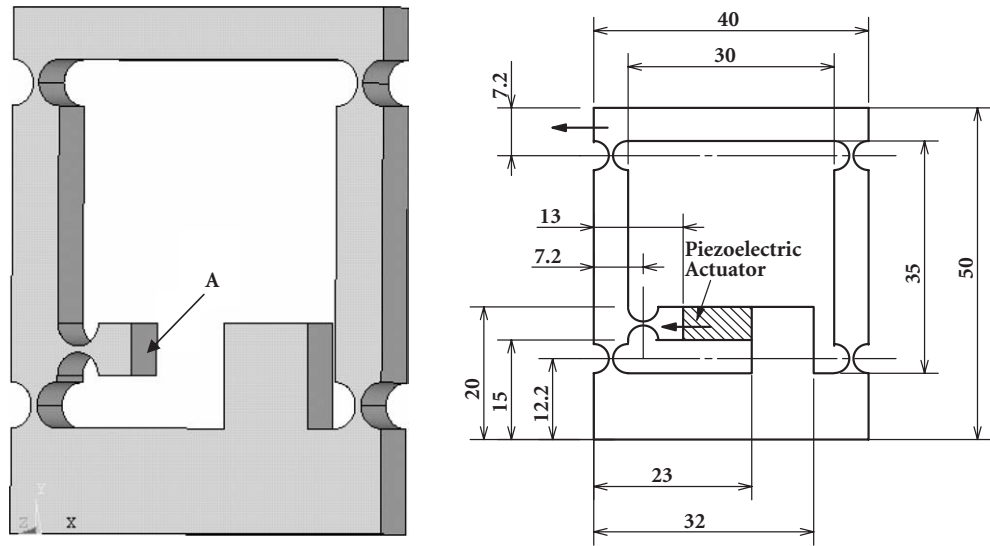
Figure 4.77   A one-axis moving table using elastic hinges.

4.4.2   CREATE A MODEL FOR ANALYSIS

4.4.2.1   SELECT ELEMENT TYPE

In this example, the solid element is selected to analyze the resonant frequency of the moving table.
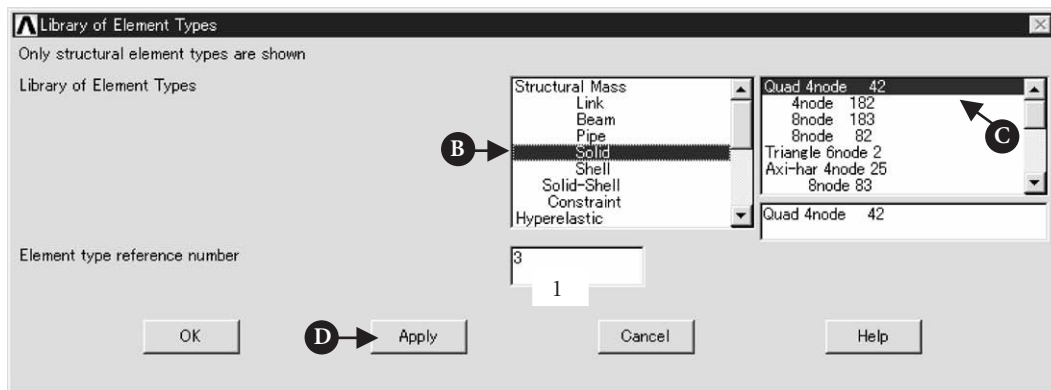
COMMAND | **ANSYS Main Menu → Preprocessor → Element Type → Add/Edit/Delete**

Then the window **Element Types** as shown in Figure 4.78 opens.

(1) Click [A] **add** button. Then the window **Library of Element Types** as shown in Figure 4.79 opens.
(2) Select [B] **Solid** in the table of **Library of Element Types** and, then, select [C] **Quad 4node 42** and **Element type reference number** is set to **1** and click [D] **Apply**. Select [E] **Brick 8node 45**, as shown in Figure 4.80.
(3) **Element type reference number** is set to **2** and click [F] **OK** button. Then the window **Library of Element Types** is closed.
(4) Click [G] **Close** button in the window of Figure 4.81 where the names of selected elements are indicated.

4.4.2.2   MATERIAL PROPERTIES

This section describes the procedure of defining the material properties of solid element.

Figure 4.78 Window of **Element Types**.

Figure 4.79 Window of **Library of Element Types**.

C O M M A N D | **ANSYS Main Menu → Preprocessor → Material Props → Material Models**

(1) Click the above buttons in order and the window **Define Material Model Behavior** opens as shown in Figure 4.82.

(2) Double click the following terms in the window.

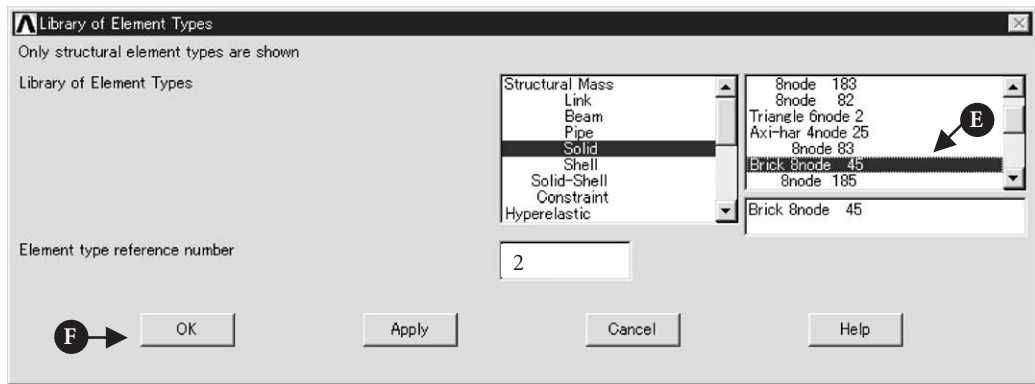**Structural → Linear → Elastic → Isotropic**
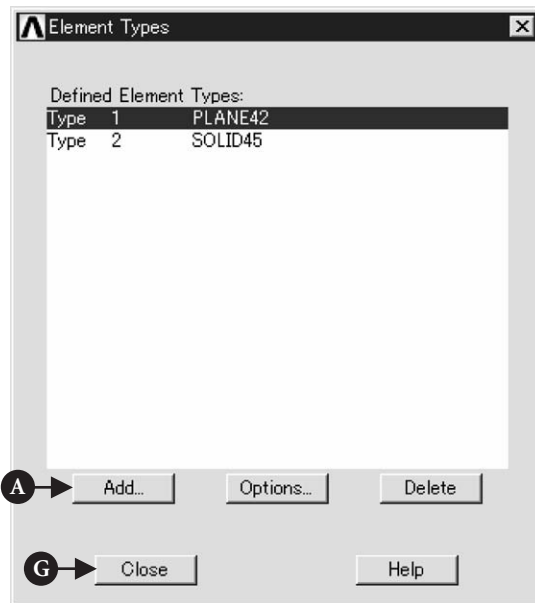
Figure 4.80 Window of **Library of Element Types**.

Figure 4.81 Window of **Element Types**.

Then the window **Linear Isotropic Properties for Material Number 1** opens.

(3) Input Young's modulus of **206e9** to **EX** box and Poisson's ratio of **0.3** to **PRXY** box.

Next, define the value of density of material.

(1) Double click the term of **Density**, and the window **Density for Material Number 1** opens.

(2) Input the value of Density, **7800** to **DENS** box and click **OK** button. Finally, close the window **Define Material Model Behavior** by clicking **X** mark at the upper right end.
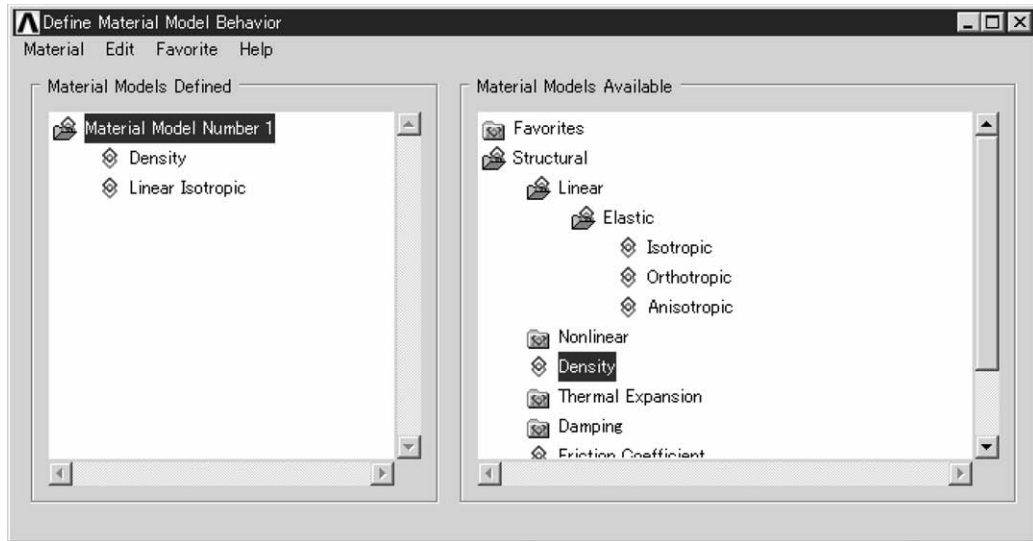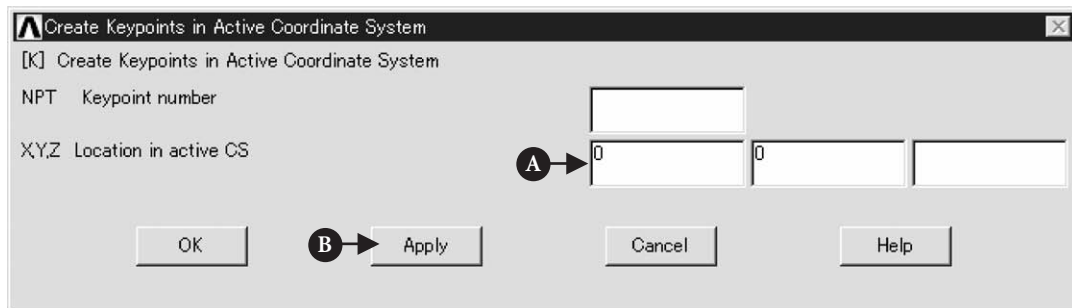
Figure 4.82 Window of **Define Material Model Behavior**.

Figure 4.83 Window of **Create Keypoints in Active Coordinate System**.

4.4.2.3 CREATE KEYPOINTS

To draw the moving table for analysis, the method using keypoints is described in this section.

C O M M A N D | **ANSYS Main Menu → Preprocessor → Modeling → Create → Keypoints → In Active CS**

The window **Create Keypoints in Active Coordinate System** opens (Figure 4.83).

(1) 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.

Table 4.3   Coordinates of KPs

KP No.	X	Y
1	0	0
2	0.04	0
3	0.04	0.05
4	0	0.05
5	0.005	0.01
6	0.035	0.01
7	0.035	0.045
8	0.005	0.045
9	0.005	0.015
10	0.013	0.015
11	0.013	0.02
12	0.005	0.02
13	0.023	0.01
14	0.032	0.01
15	0.032	0.02
16	0.023	0.02

(2) In the same window, input the values as shown in Table 4.3 in order. When all values are inputted, click **OK** button.

(3) Then all input keypoints appear in **ANSYS Graphics** window as shown in Figure 4.84.

4.4.2.4   CREATE AREAS FOR THE TABLE

Areas are created from keypoints by proceeding the following steps.

COMMAND   **ANSYS Main Menu → Preprocessor → Modeling → Create → Arbitrary → Through KPs**

The window **Create Area thru KPs** opens (Figure 4.85).

(1) Pick the keypoints, **1**, **2**, **3**, and **4** in order and click [A] **Apply** button in Figure 4.85. Then pick keypoints **5**, **6**, **7**, and **8** and click [B] **OK** button. Figure 4.86 appears.

COMMAND   **ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Subtract → Areas**

(2) Click the area of KP No. **1–4** and **OK** button. Then click the area of KP No. **5–8** and **OK** button. The drawing of the table appears as shown in Figure 4.87.
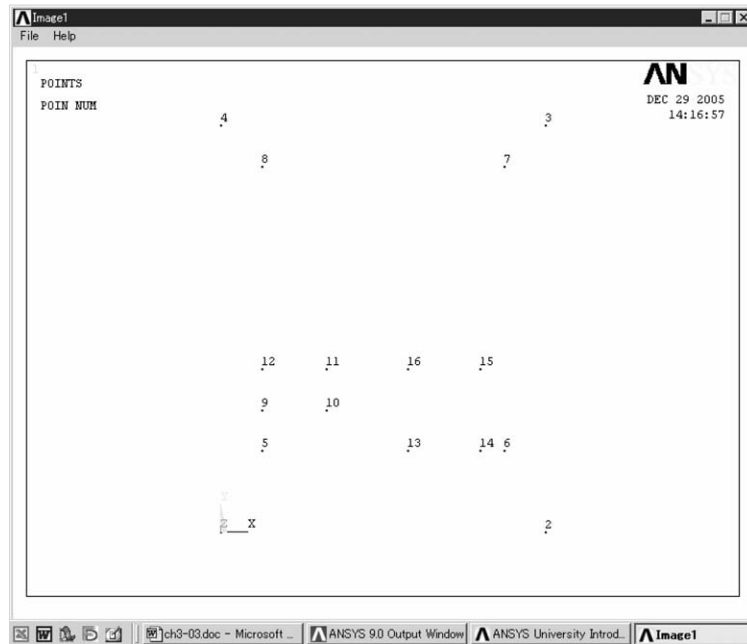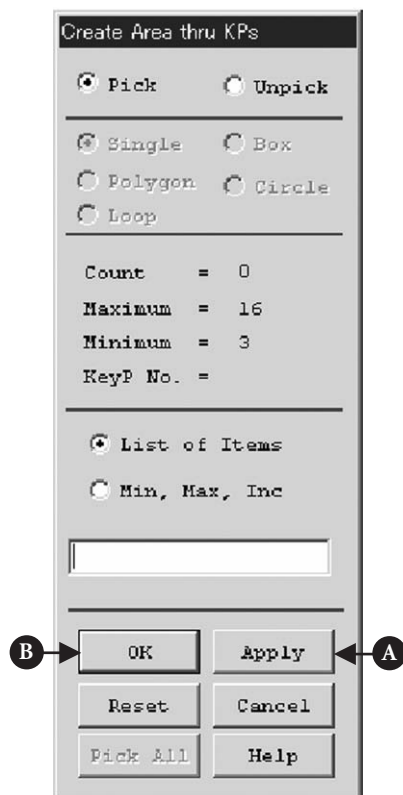
Figure 4.84 **ANSYS Graphics** window.
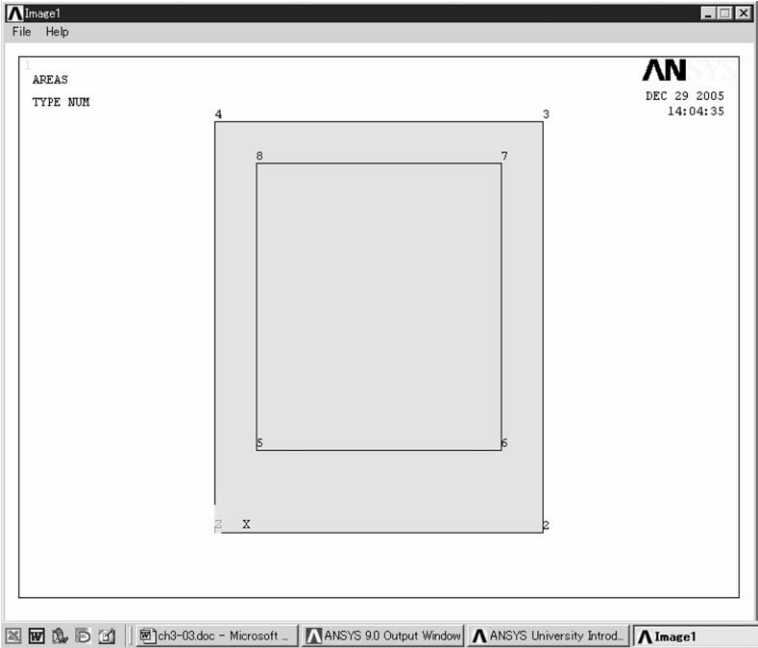

Figure 4.85 Window of **Create Area thru KPs**.

Figure 4.86 **ANSYS Graphics** window.
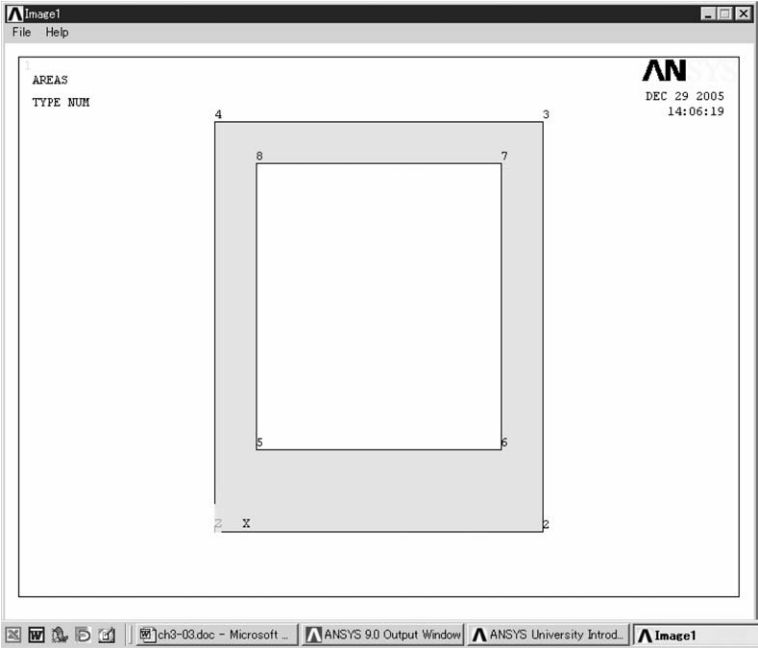

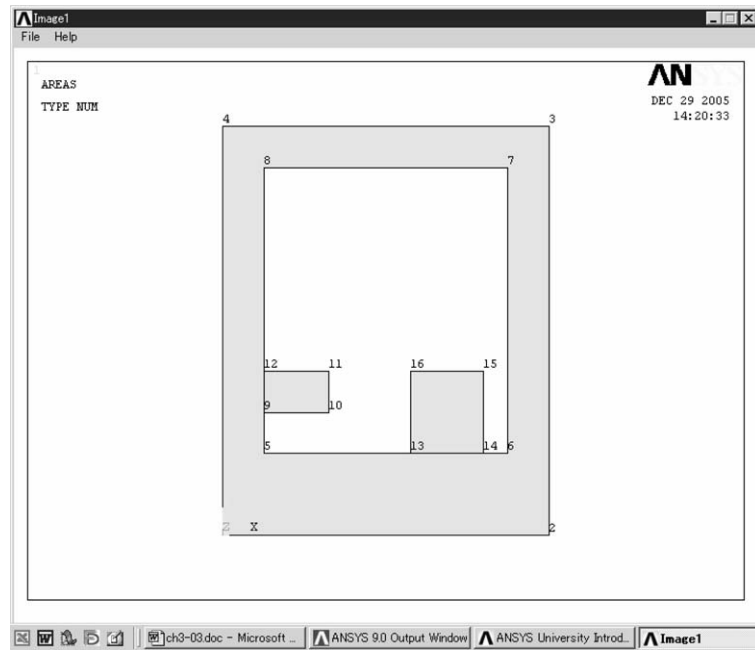**Figure 4.87** **ANSYS Graphics** window.

Figure 4.88 **ANSYS Graphics** window.

C O M M A N D | **ANSYS Main Menu → Preprocessor → Modeling → Create → Arbitrary → Through KPs**

(3) Pick keypoints **9, 10, 11,** and **12** in order and click [A] **Apply** button in Figure 4.85. Then pick keypoints **13, 14, 15,** and **16** and click [B] **OK** button. Figure 4.88 appears.

C O M M A N D | **ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Add → Areas**

The window **Add Areas** opens (Figure 4.89).

(4) Pick three areas on **ANSYS Graphics** window and click [A] **OK** button. Then three areas are added as shown in Figure 4.90.

C O M M A N D | **ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Circle → Solid Circle**

The window **Solid Circle Area** opens (Figure 4.91).

(5) Input the values of **0, 12.2e−3, 2.2e−3** to [X], [Y] and **Radius** boxes as shown in Figure 4.91 and click [A] **Apply** button. Then continue to input the coordinates
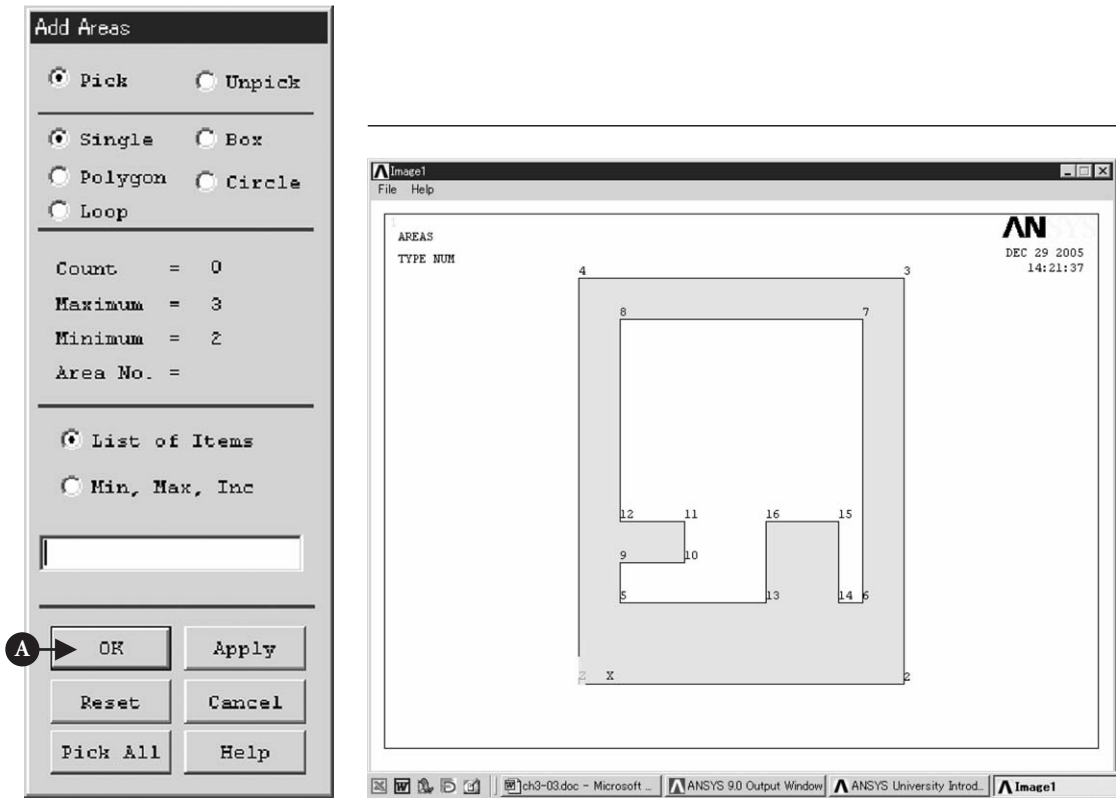
Figure 4.89 Window of **Add Areas**. **Figure 4.90** **ANSYS Graphics** window.

of the solid circle as shown in Table 4.4. The radius of all solid circles is 0.0022. When all values are inputted, the drawing of the table appears as shown in Figure 4.92. Click [B] **OK** button.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Subtract → Areas**

(1) Subtract all circular areas from the rectangular area by executing the above steps. Then Figure 4.93 is displayed.

4.4.2.5 CREATE MESH IN AREAS

COMMAND | **ANSYS Main Menu → Preprocessor → Meshing → Mesh Tool**

The window **Mesh Tool** opens (Figure 4.94).

(1) Click [A] **Lines Set** and the window **Element Size on Picked Lines** opens (Figure 4.95).
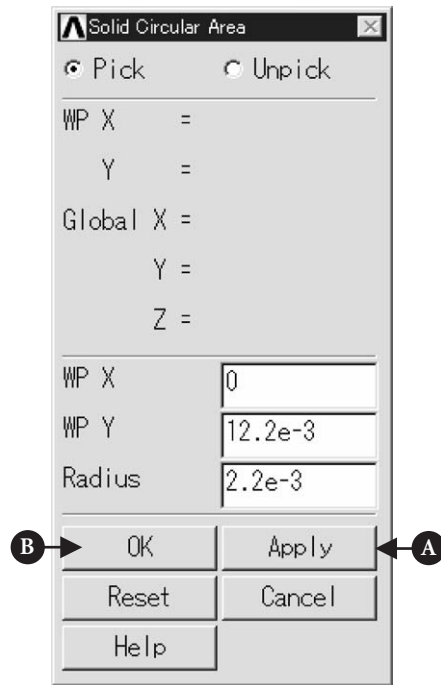
Figure 4.91 Window of **Solid Circular Area**.

Table 4.4 Coordinates of solid circles

No.	X	Y	Radius
1	0	0.0122	
2	0.005	0.0122	
3	0.035	0.0122	
4	0.04	0.0122	
5	0	0.0428	0.0022
6	0.005	0.0428	
7	0.035	0.0428	
8	0.04	0.0428	
9	0.0072	0.015	
10	0.0072	0.02	

(2) Click [B] **Pick All** button and the window **Element Sizes on Picked Lines** opens (Figure 4.96).

(3) Input [C] **0.001** to **SIZE** box and click [D] **OK** button.

(4) Click [E] **Mesh** of the window **Mesh Tool** (Figure 4.97) and, then, the window **Mesh Areas** opens (Figure 4.98).
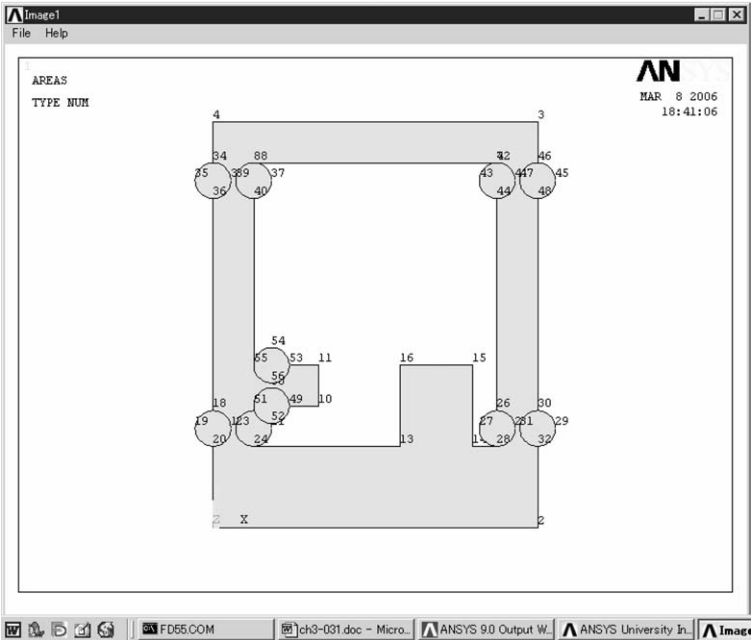
Figure 4.92   **ANSYS Graphics** window.
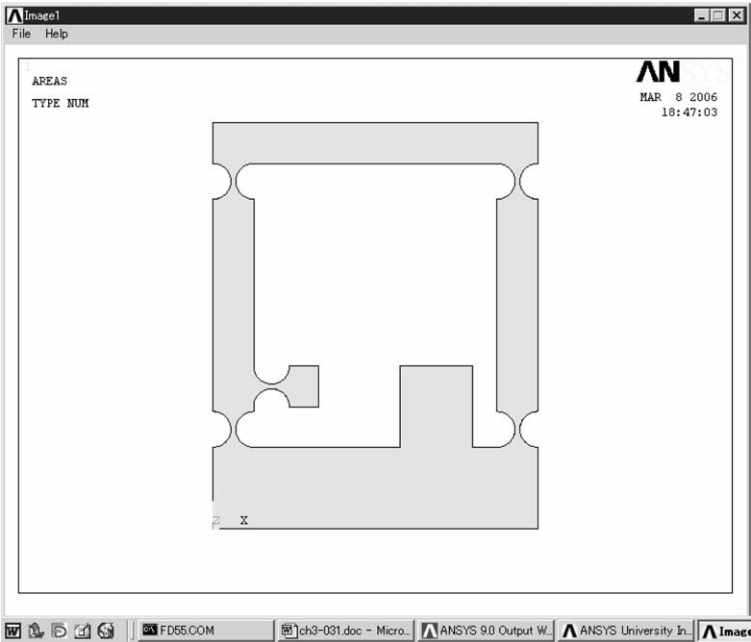
Figure 4.93   **ANSYS Graphics** window.
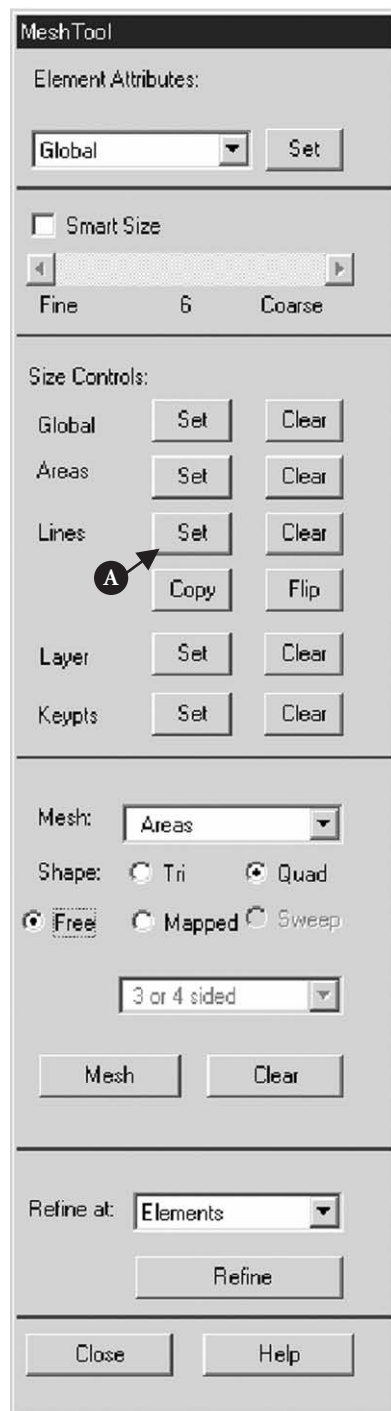
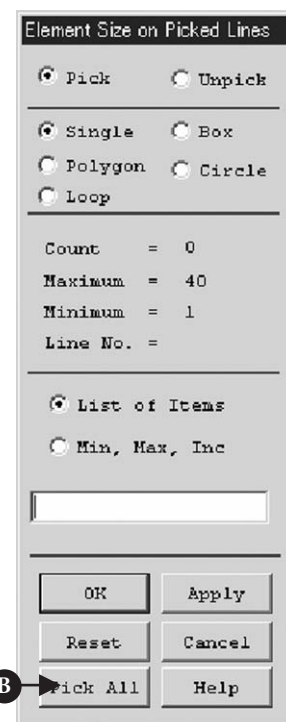**Figure 4.94**　Window of **Mesh Tool**.

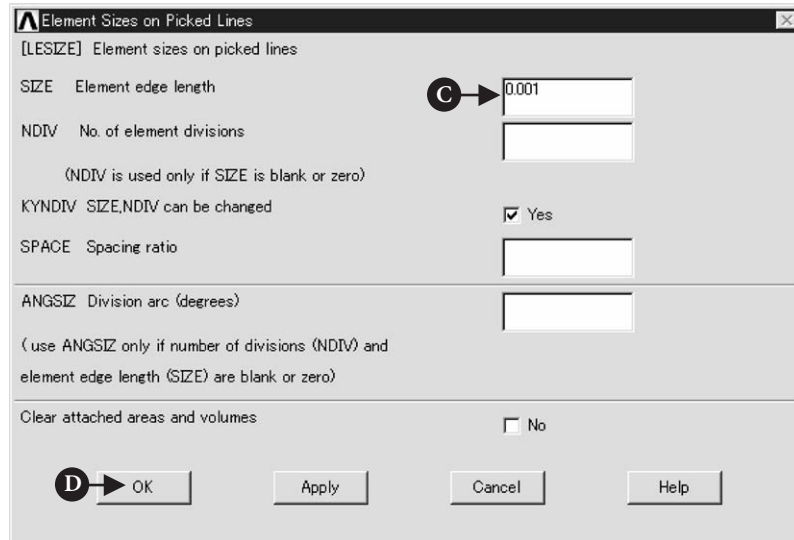**Figure 4.95**　Window of **Element Size on Picked Lines**.

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.

C O M M A N D   **ANSYS Main Menu → Preprocessor → Modeling → Operate → Extrude → Elem Ext Opts**

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.

C O M M A N D   **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

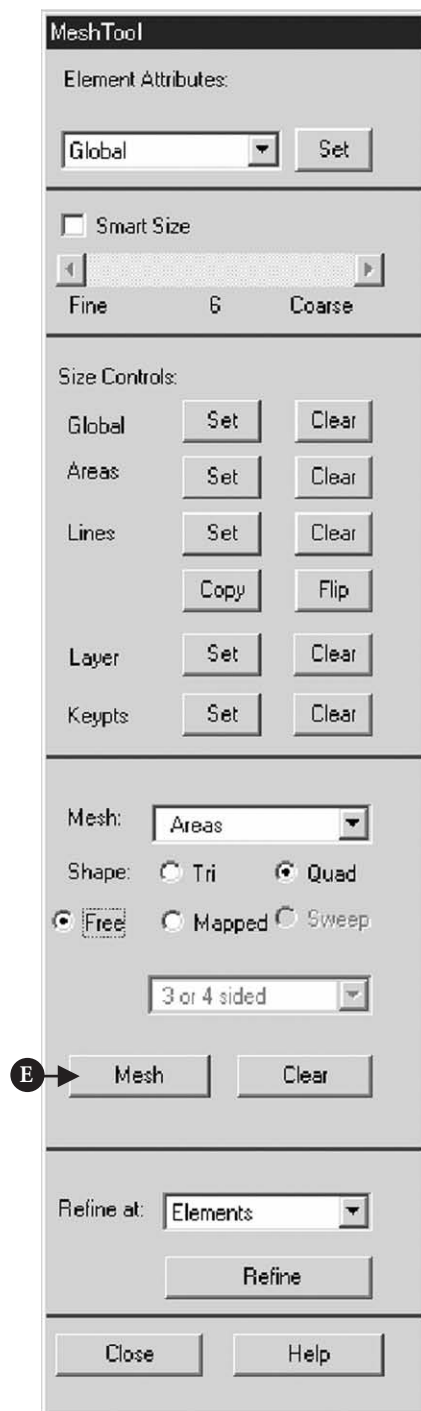The table is fixed at both the bottom and the region A of the table.
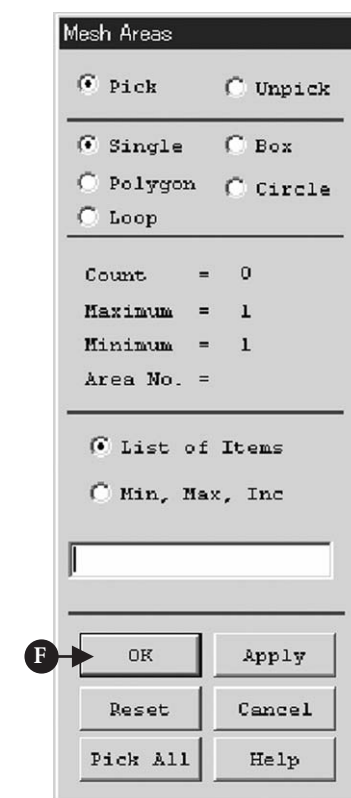
Figure 4.97 Window of **Mesh Tool**.
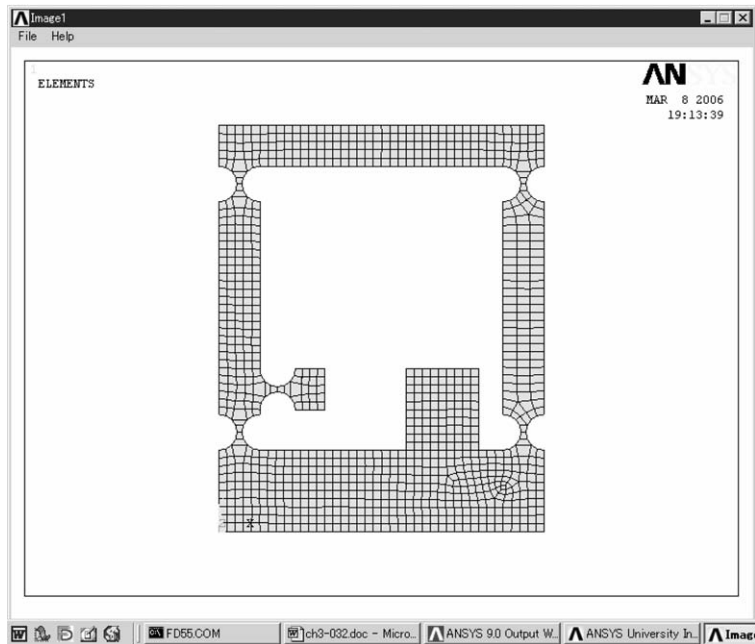

Figure 4.98 Window of **Mesh Areas**.

Figure 4.99   **ANSYS Graphics** window.
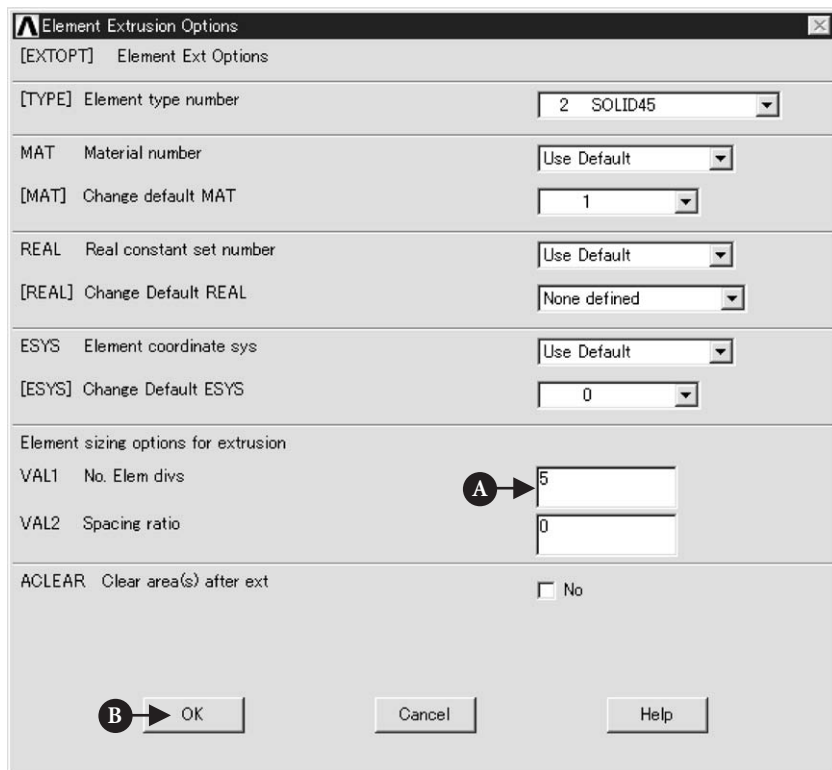

Figure 4.100   Window of **Element Extrusion Options**.
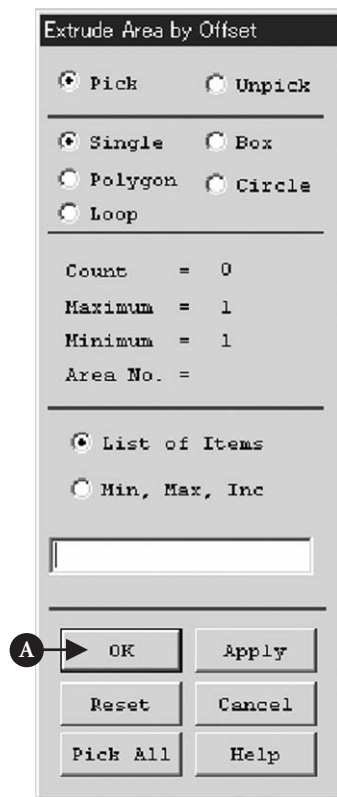
Figure 4.101   Window of **Extrude Area by Offset**.
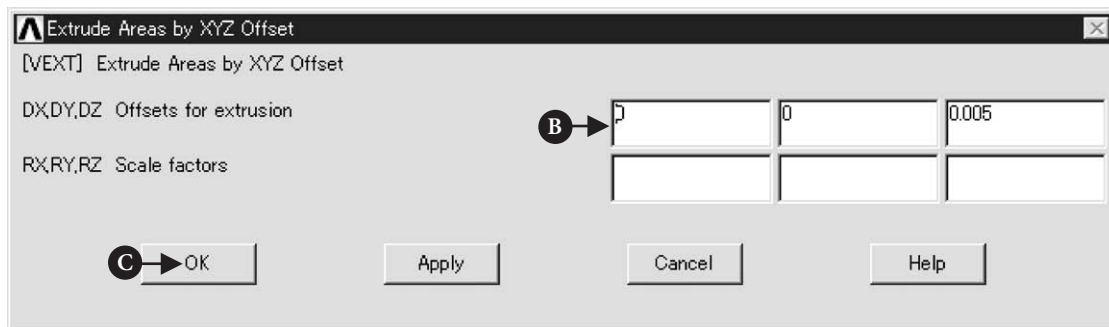

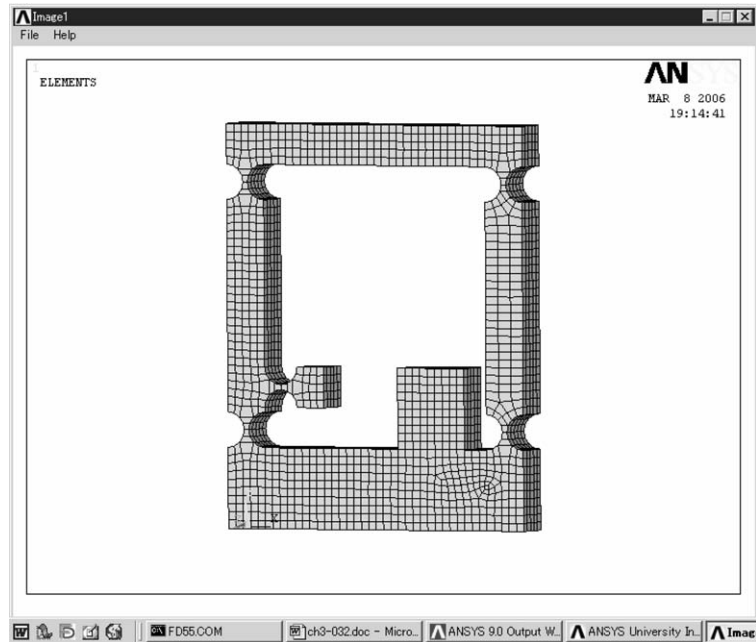**Figure 4.102**   Window of **Extrude Areas by XYZ Offset**.

Figure 4.103   **ANSYS Graphics** window.

ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Displacement → On Areas

The window **Apply U,ROT on Areas** opens (Figure 4.104).

(1) Pick [A] the side wall for a piezoelectric actuator in Figure 4.105 and [B] the bottom of the table. Then click [C] **OK** button and the window **Apply U,ROT on Areas** opens (Figure 4.106).

(2) Select [D] **All DOF** in the box of **Lab2** and, then, click [E] **OK** button. After these steps **ANSYS Graphics** window is changed as shown in Figure 4.107.

4.4.3   ANALYSIS

4.4.3.1   DEFINE THE TYPE OF ANALYSIS

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

ANSYS Main Menu → Solution → Analysis Type → New Analysis
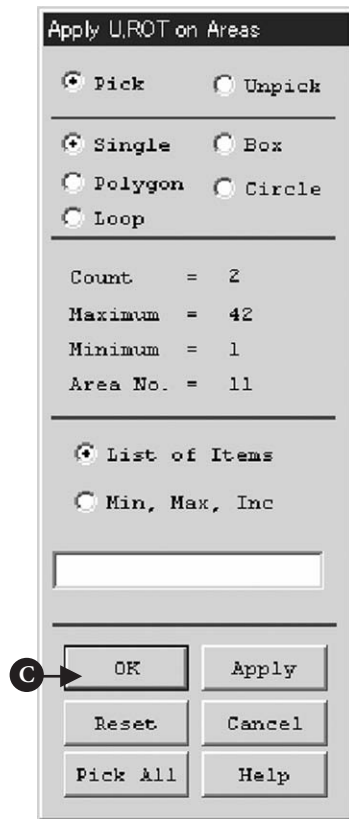
The window **New Analysis** opens (Figure 4.108).
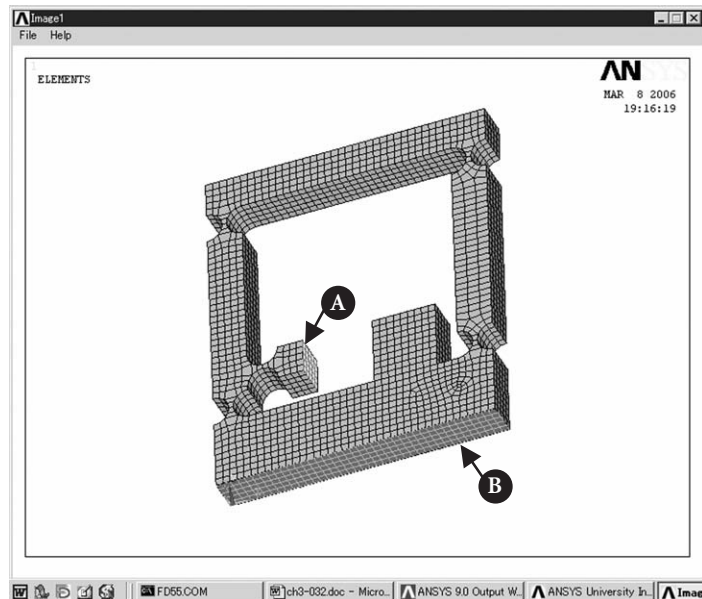
Figure 4.104 Window of **Apply U,ROT on Areas**.

Figure 4.105 **ANSYS Graphics** window.
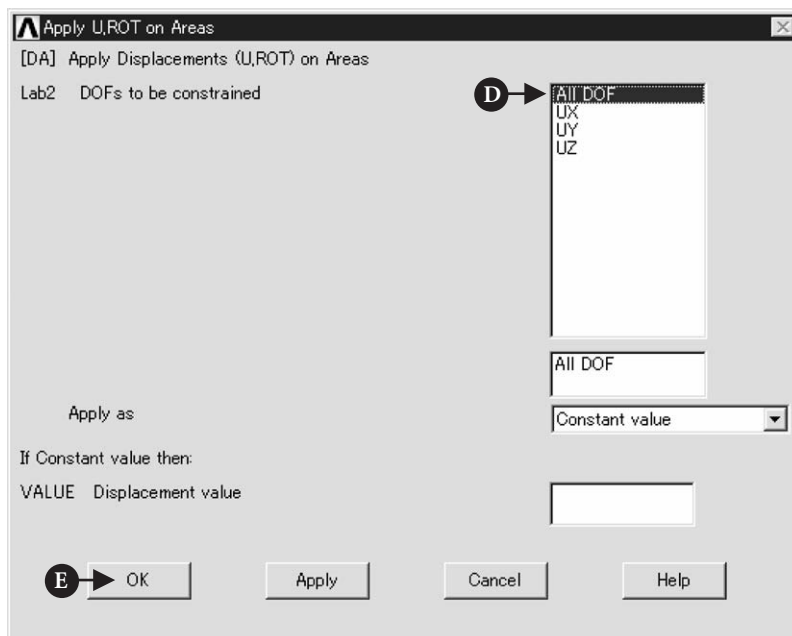
Figure 4.106   Window of **Apply U,ROT on Areas**.
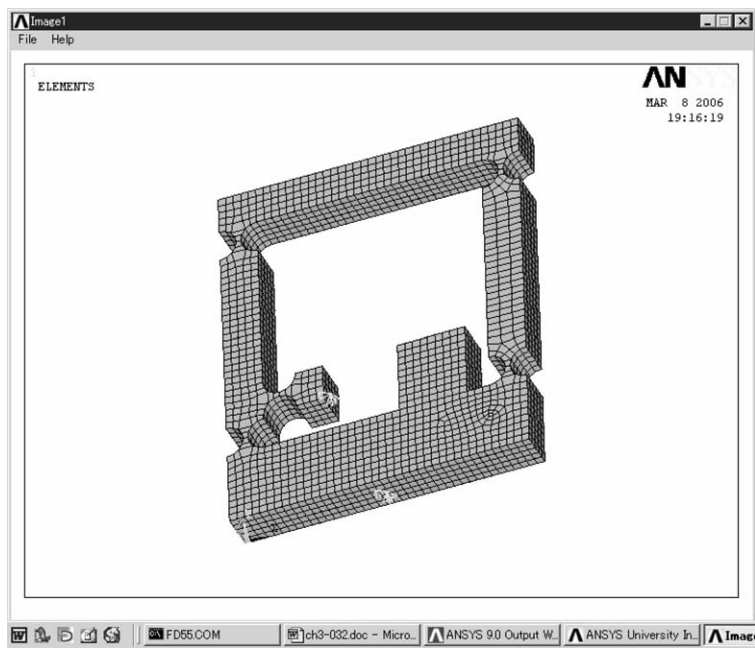

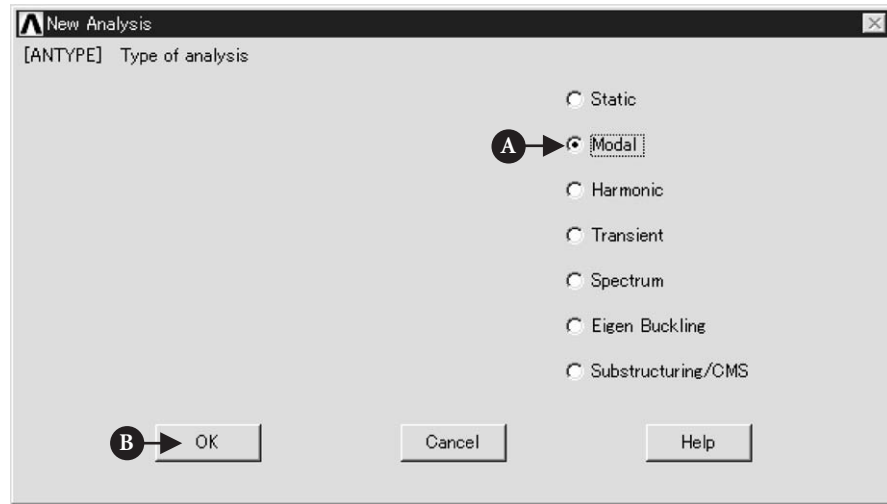**Figure 4.107**   **ANSYS Graphics** window.

Figure 4.108 Window of **New Analysis**.

(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.

C O M M A N D | **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

C O M M A N D | **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.
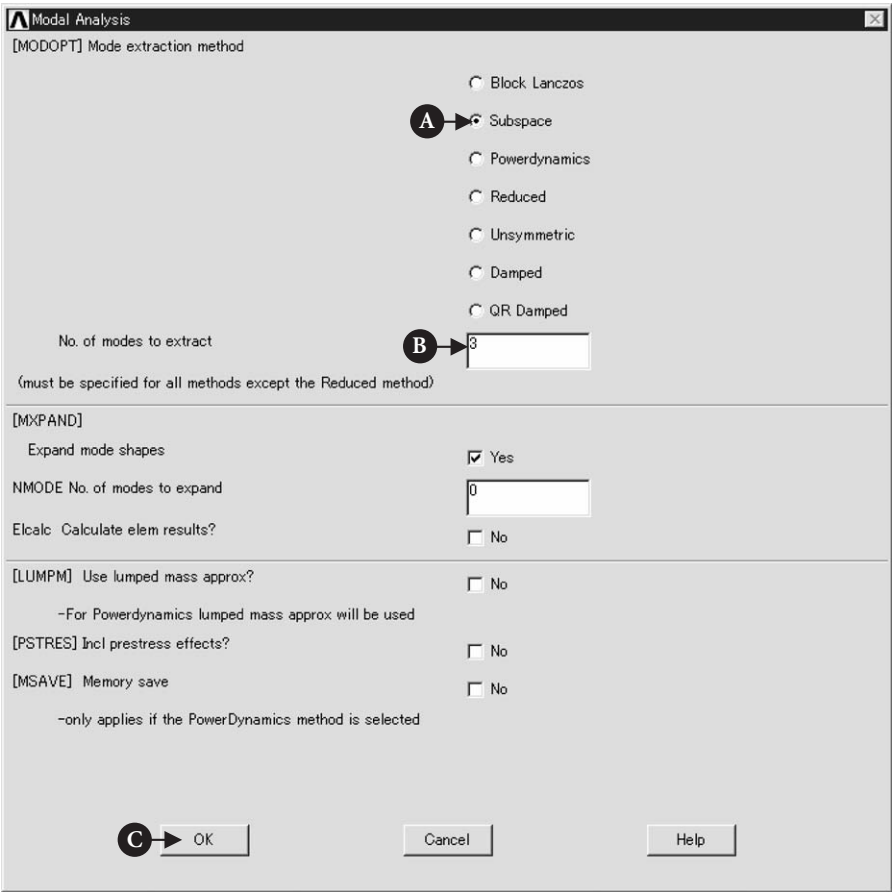
Figure 4.109 Window of **Modal Analysis**.

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**
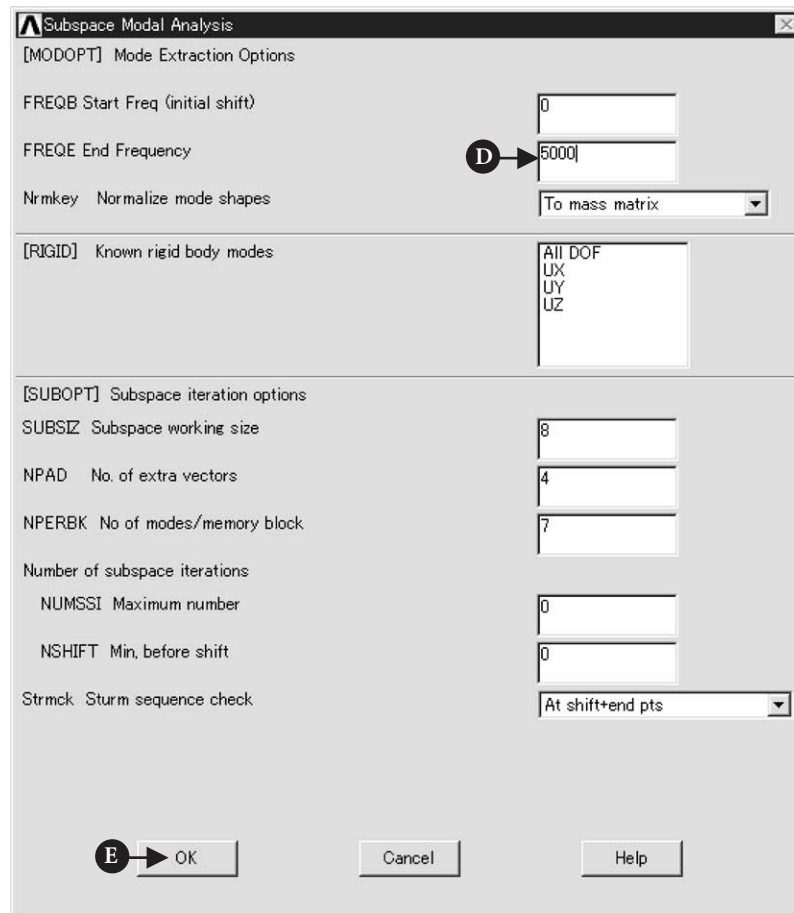
Figure 4.110   Window of **Subspace Modal Analysis**.

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

C O M M A N D | **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.
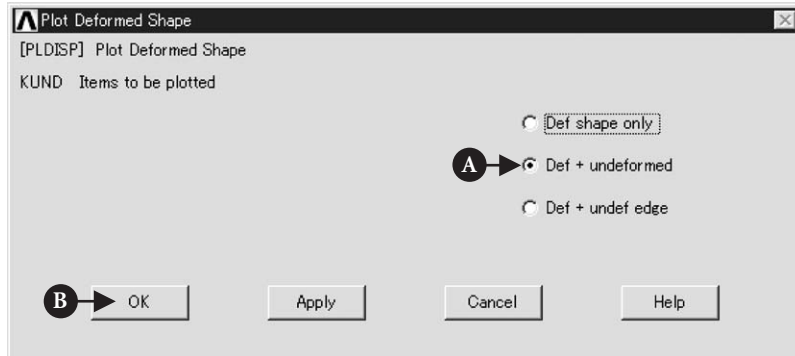
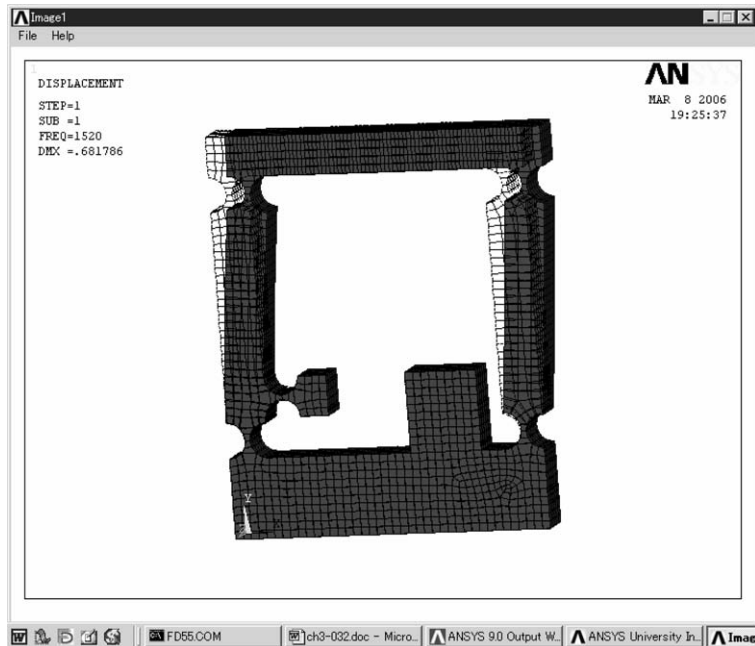**Figure 4.111**   Window of **Plot Deformed Shape**.

Figure 4.112   **ANSYS Graphics** window for the first mode.

4.4.4.4   ANIMATE THE VIBRATION MODE SHAPE

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

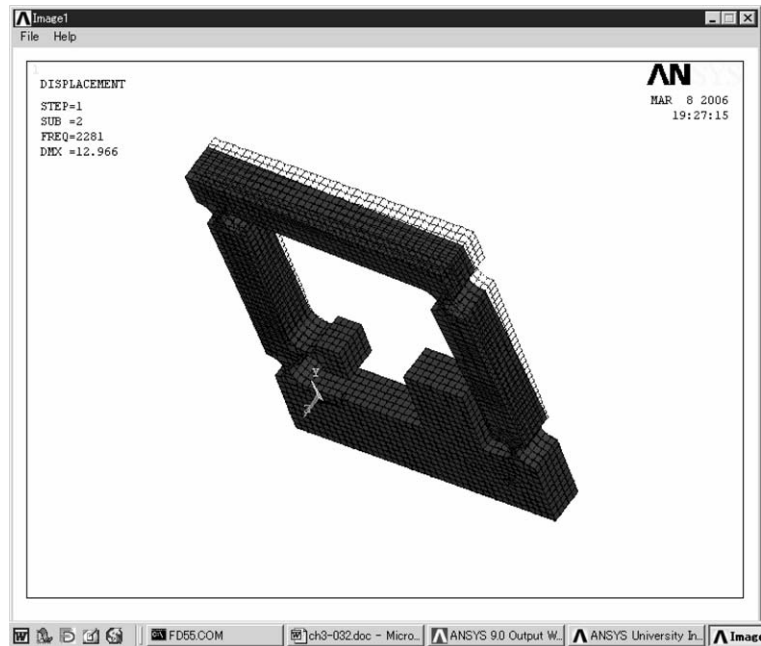COMMAND | **Utility Menu → PlotCtrls → Animate → Mode Shape**

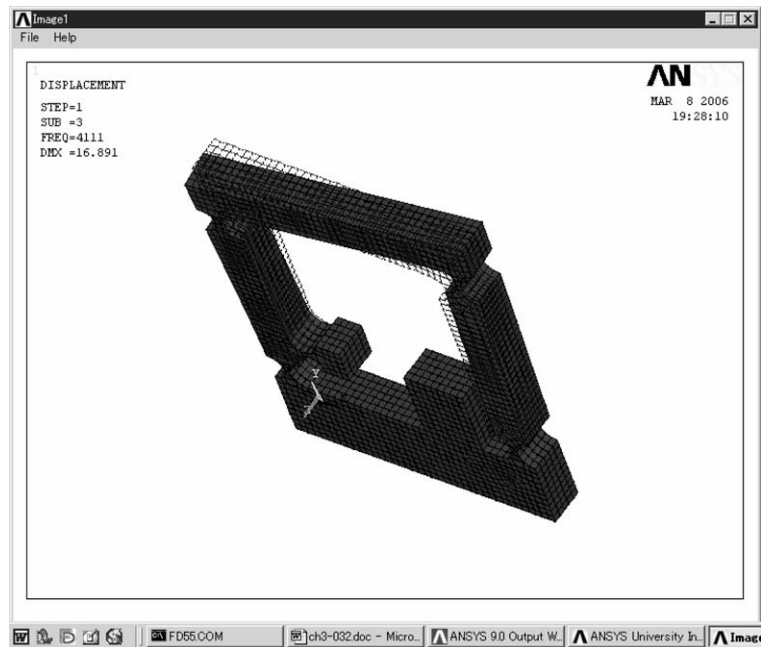**Figure 4.113** **ANSYS Graphics** window for the second mode.

Figure 4.114 **ANSYS Graphics** window for the third mode.

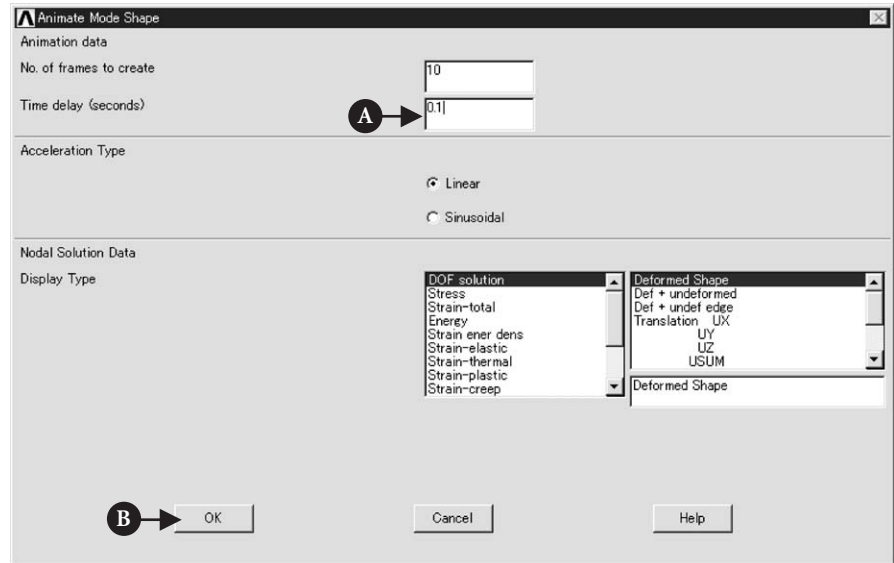The window **Animate Mode Shape** opens (Figure 4.115).

Figure 4.115   Window of **Animate Mode Shape**.

(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.