

APPLICATION OF ANSYS TO STRESS ANALYSIS

CHAPTER OUTLINE

3.1	CANTILEVER BEAM	51
3.2	THE PRINCIPLE OF ST. VENANT	84
3.3	STRESS CONCENTRATION DUE TO ELLIPTIC HOLES	93
3.4	STRESS SINGULARITY PROBLEM	106
3.5	TWO-DIMENSIONAL CONTACT STRESS	120
	REFERENCES	141

3.1 CANTILEVER BEAM

Beams are important fundamental structural and/or machine elements; they are found in buildings and in bridges. Beams are also used as shafts in cars and trains, as wings in aircrafts and bookshelves in bookstores. Arms and femurs of human beings and branches of trees are good examples of portions of living creatures which support their bodies. Beams play important roles not only in inorganic but also in organic structures.

Mechanics of beams is one of the most important subjects in engineering.

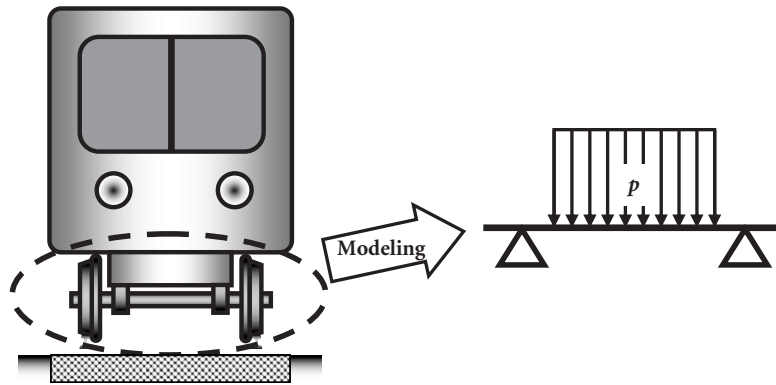


Figure 3.1 Modeling of an axle shaft by a simply supported beam.

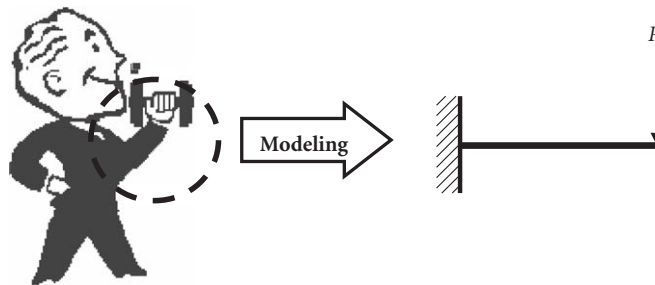


Figure 3.2 Modeling of an arm of a human being by a cantilever beam.

3.1.1 EXAMPLE PROBLEM: A CANTILEVER BEAM

Perform an finite-element method (FEM) analysis of a 2-D cantilever beam shown in Figure 3.3 below and calculate the deflection of the beam at the loading point and the longitudinal stress distribution in the beam.

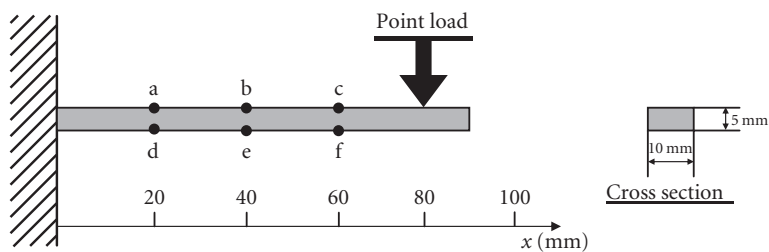


Figure 3.3 Bending of a cantilever beam to solve.

3.1.2 PROBLEM DESCRIPTION

Geometry: length $l = 90$ mm, height $h = 5$ mm, thickness $b = 10$ mm

Material: mild steel having Young's modulus $E = 210$ GPa and Poisson's ratio $\nu = 0.3$.

Boundary conditions: The beam is clamped to a rigid wall at the left end and loaded at $x = 80$ mm by a point load of $P = 100$ N.

3.1.2.1 REVIEW OF THE SOLUTIONS OBTAINED BY THE ELEMENTARY BEAM THEORY

Before proceeding to the FEM analysis of the beam, let us review the solutions to the example problem obtained by the elementary beam theory. The maximum deflection of the beam δ_{\max} can be calculated by the following equation:

$$\delta_{\max} = \frac{Pl_1^3}{3EI} \left(1 + \frac{3l_1}{l_2} \right) \quad (3.1)$$

where l_1 ($=80$ mm) is the distance of the application point of the load from the rigid wall and $l_2 = l - l_1$.

The maximum tensile stress $\sigma_{\max}(x)$ at x in the longitudinal direction appears at the upper surface of the beam in a cross section at x from the wall;

$$\sigma_{\max}(x) = \begin{cases} \frac{P(l_1-x)}{I} \frac{h}{2} & (0 \leq x \leq l_1) \\ 0 & (0 \leq x) \end{cases} \quad (3.2)$$

where l ($=90$ mm) is the length, h ($=5$ mm) the height, b ($=10$ mm) the thickness, E ($=210$ GPa) Young's modulus and I the area moment of inertia of the cross section of the beam. For a beam having a rectangular cross section of a height h by a thickness b , the value of I can be calculated by the following equation:

$$I = \frac{bh^3}{12} \quad (3.3)$$

3.1.3 ANALYTICAL PROCEDURES

Figure 3.4 shows how to make structural analyses by using FEM. In this chapter, the analytical procedures will be explained following the flowchart illustrated in Figure 3.4.

3.1.3.1 CREATION OF AN ANALYTICAL MODEL

[1] *Creation of a beam shape to analyze*

Here we will analyze a rectangular slender beam of 5 mm (0.005 m) in height by 90 mm (0.09 m) in length by 10 mm (0.01 m) in width as illustrated in Figure 3.3.

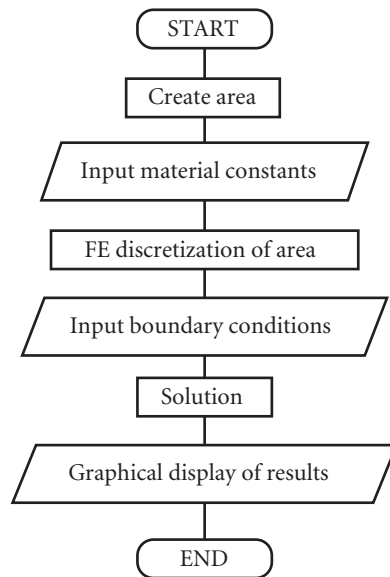


Figure 3.4 Flowchart of the structural analyses by ANSYS.

Figure 3.5 shows the “ANSYS Main Menu” window where we can find layered command options imitating folders and files in the Microsoft® Explorer folder window.

In order to prepare for creating the beam, the following operations should be made:

- (1) Click [A] **Preprocessor** to open its sub-menus in **ANSYS Main Menu** window.
- (2) Click [B] **Modeling** to open its sub-menus and select **Create** menu.
- (3) Click [C] **Areas** to open its sub-menus and select **Rectangle** menu.
- (4) Click to select [D] **By 2 corners** menu.

After carrying out the operations above, a window called **Rectangle by 2 corners** as shown in Figure 3.6 appears for the input of the geometry of a 2-D rectangular beam.

[2] Input of the beam geometry to analyze

The **Rectangle by 2 corners** window has four boxes for inputting the coordinates of the lower left corner point of the rectangular beam and the width and height of the beam to create. The following operations complete the creation procedure of the beam:

- (1) Input two 0's into [A] **WP X** and [B] **WP Y** to determine the lower left corner point of the beam on the Cartesian coordinates of the working plane.
- (2) Input **0.09** and **0.005 (m)** into [C] **Width** and [D] **Height**, respectively to determine the shape of the beam model.
- (3) Click [E] **OK** button to create the rectangular area, or beam on the **ANSYS Graphics** window as shown in Figure 3.7.

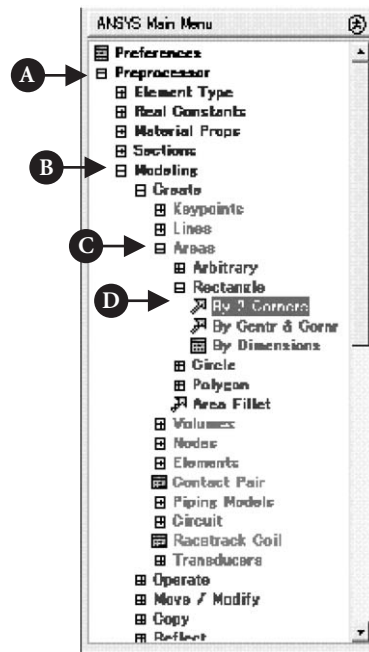


Figure 3.5 “ANSYS Main Menu” window.

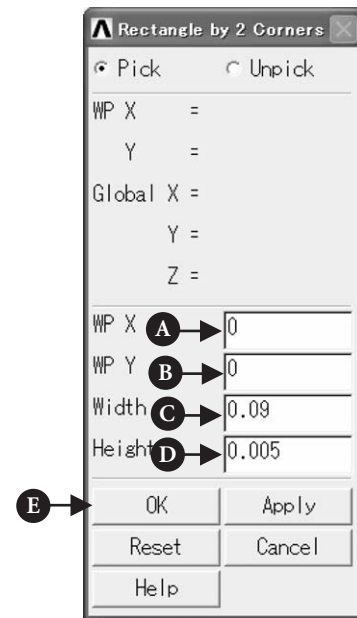


Figure 3.6 “Rectangle By 2 Corners” window.

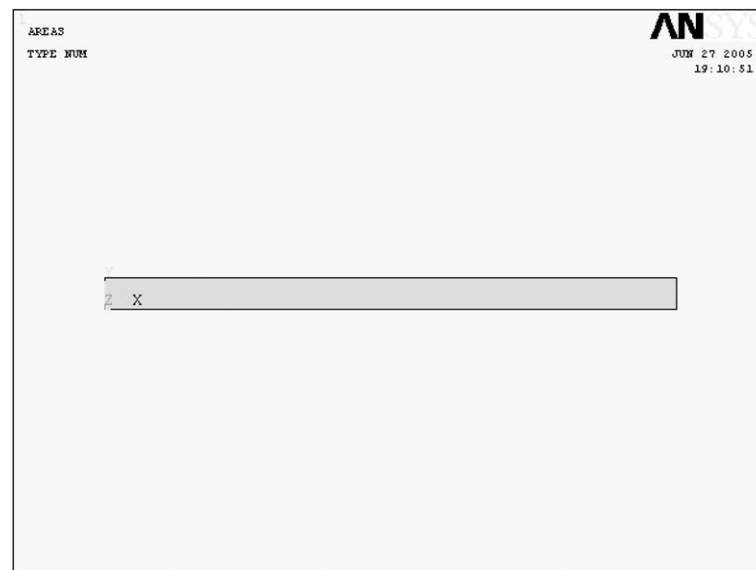


Figure 3.7 2-D beam created and displayed on the “ANSYS Graphics” window.

How to correct the shape of the model

In case of correcting the model, delete the area first, and repeat the procedures [1] and [2] above. In order to delete the area, execute the following commands:

COMMAND | ANSYS Main Menu → Preprocessor → Modeling → Delete → Area and Below

Then, the **Delete Area and Below** window opens and an upward arrow (↑) appears on the ANSYS Graphics window:

- (1) Move the arrow to the area to delete and click the left button of the mouse.
- (2) The color of the area turns from light blue into pink.
- (3) Click **OK** button and the area is to be deleted.

3.1.3.2 INPUT OF THE ELASTIC PROPERTIES OF THE BEAM MATERIAL

Next, we specify elastic constants of the beam. In the case of isotropic material, the elastic constants are Young's modulus and Poisson's ratio. This procedure can be performed any time before the solution procedure, for instance, after setting boundary conditions. If this procedure is missed, we cannot perform the solution procedure.

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

Then the **Define Material Model Behavior** window opens as shown in Figure 3.8:

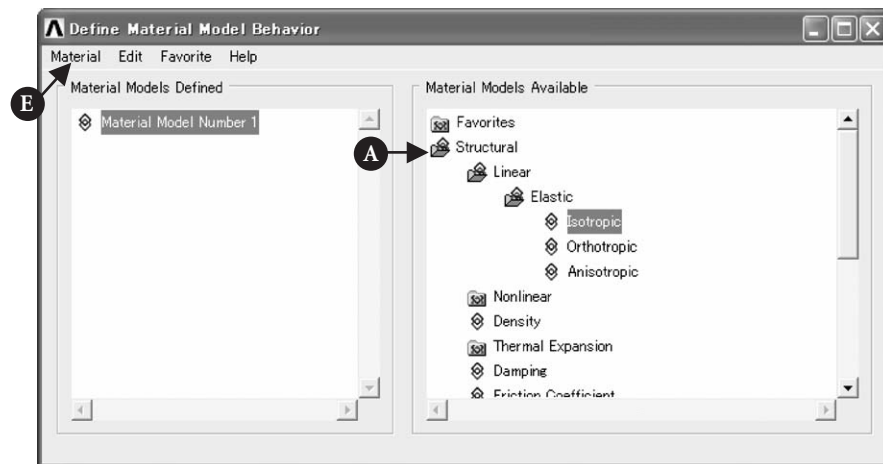


Figure 3.8 “Define Material Model Behavior” window.

- (1) Double-click [A] **Structural**, **Linear**, **Elastic**, and **Isotropic** buttons one after another.

- (2) Input the value of Young's modulus, **2.1e11** (Pa), and that of Poisson's ratio, **0.3**, into [B] **EX** and [C] **PRXY** boxes, and click [D] **OK** button of the **Linear Isotropic Properties for Material Number 1** as shown in Figure 3.9.

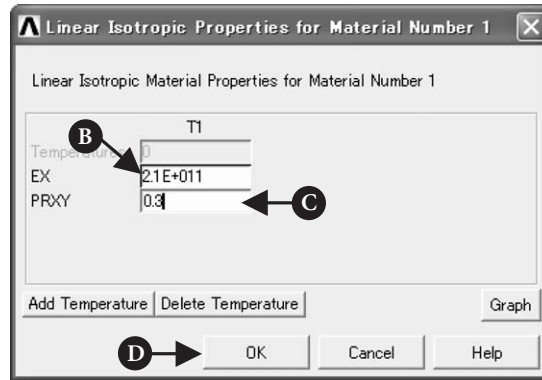


Figure 3.9 Input of elastic constants through the “Linear Isotropic Properties for Material Number 1” window.

- (3) Exit from the **Define Material Model Behavior** window by selecting **Exit** in [E] **Material** menu of the window (see Figure 3.8).

3.1.3.3 FINITE-ELEMENT DISCRETIZATION OF THE BEAM AREA

Here we will divide the beam area into finite elements. The procedures for finite-element discretization are firstly to select the element type, secondly to input the element thickness and finally to divide the beam area into elements.

[1] Selection of the element type

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

Then the **Element Types** window opens:

- (1) Click [A] **Add . . .** button to open the **Library of Element Types** window as shown in Figure 3.11 and select the element type to use.
- (2) To select the 8-node isoparametric element, select [B] **Structural Mass – Solid**.
- (3) Select [C] **Quad 8node 82** and click [D] **OK** button to choose the 8-node isoparametric element.
- (4) Click [E] **Options . . .** button in the **Element Types** window as shown in Figure 3.10 to open the **PLANE82 element type options** window as depicted in Figure 3.12. Select [F] **Plane strs w/thk** item in the **Element behavior** box and click [G] **OK** button to return to the **Element Types** window. Click [H] **Close** button to close the window.

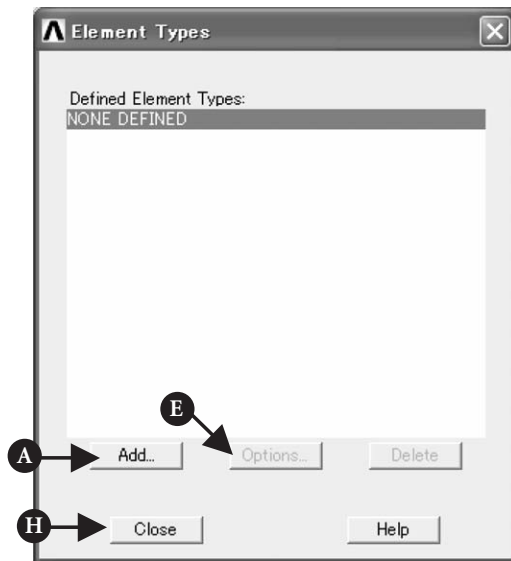


Figure 3.10 “Element Types” window.

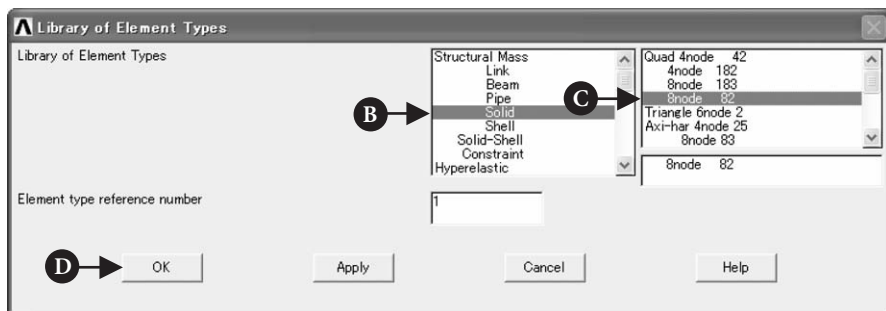


Figure 3.11 “Library of Element Types” window.

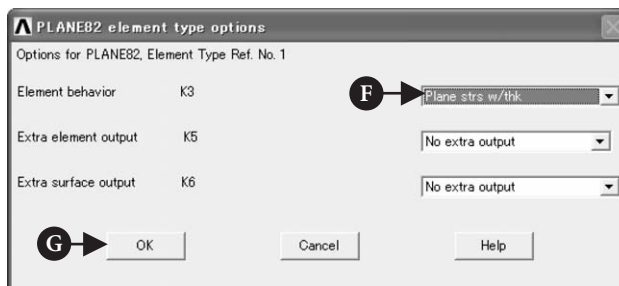


Figure 3.12 “PLANE82 element type options” window.

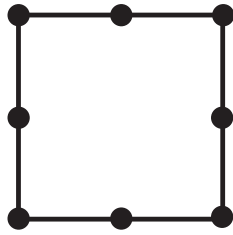


Figure 3.13 8-node isoparametric rectangular element.

The 8-node isoparametric element is a rectangular element which has four corner nodal points and four middle points as shown in Figure 3.13 and can realize the finite-element analysis with higher accuracy than the 4-node linear rectangular element. The beam area is divided into these 8-node rectangular #82 finite elements.

[2] Input of the element thickness

COMMAND

ANSYS Main Menu → Preprocessor → Real Constants

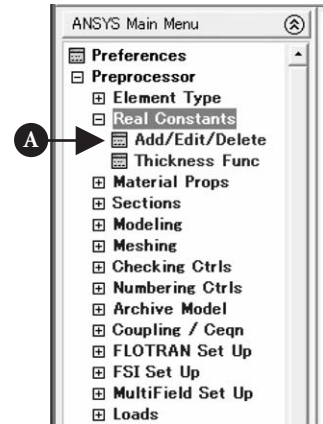


Figure 3.14 Setting of the element thickness from the real constant command.

Select [A] **Real Constants** in the **ANSYS Main Menu** as shown in Figure 3.14.

- (1) Click [A] **Add/Edit/Delete** button to open the **Real Constants** window as shown in Figure 3.15 and click [B] **Add ...** button.
- (2) Then the **Element Type for Real Constants** window opens (see Figure 3.16). Click [C] **OK** button.
- (3) The **Element Type for Real Constants** window vanishes and the **Real Constants Set Number 1. for PLANE82** window appears instead as shown in Figure 3.17.

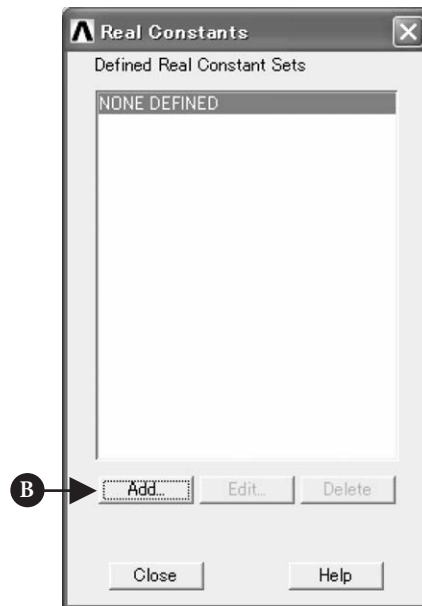


Figure 3.15 “Real Constants” window before setting the element thickness.

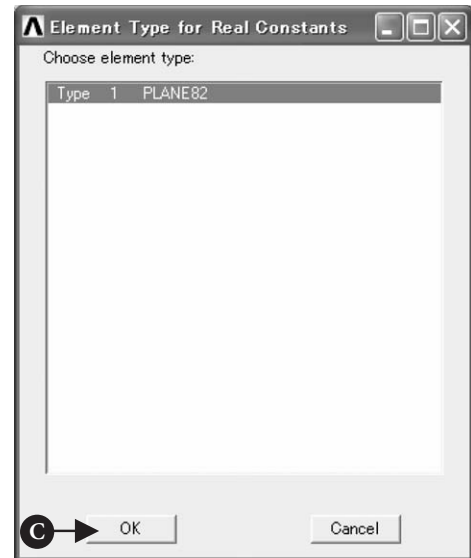


Figure 3.16 “Element Type for Real Constants” window.

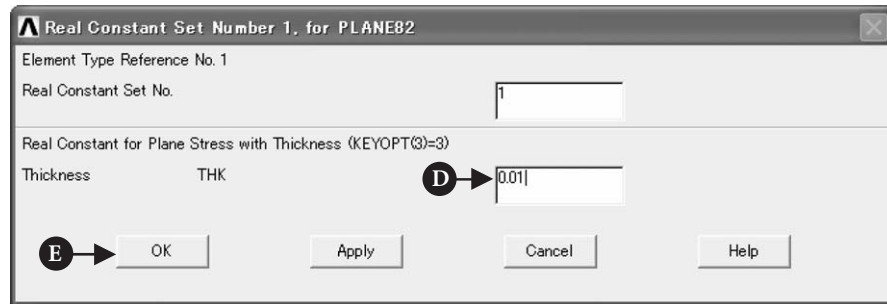


Figure 3.17 “Real Constants Set Number 1, for PLANE82” window.

Input a plate thickness of “0.01” (m) in [D] **Thickness** box and click [E] **OK** button.

- (4) The **Real Constants** window returns with the display of the **Defined Real Constants Sets** box changed to **Set 1** as shown in Figure 3.18. Click [F] **Close** button, which makes the operation of setting the plate thickness completed.



Figure 3.18 “Real Constants” window after setting the element thickness.

[3] Sizing of the elements

COMMAND | ANSYS Main Menu → Preprocessor → Meshing → Size Cntrls → Manual Size → Global → Size

The **Global Element Sizes** window opens as shown in Figure 3.19:

- (1) Input **0.002** (m) in [A] **SIZE** box and click [B] **OK** button.

By the operations above the element size of 0.002, i.e., 0.002 m, or 2 mm is specified and the beam of 5 mm by 90 mm is divided into rectangular finite-elements with one side 2 mm and the other side 3 mm in length.

[4] Meshing

COMMAND | ANSYS Main Menu → Preprocessor → Meshing → Mesh → Areas → Free

The **Mesh Areas** window opens as shown in Figure 3.20:

- (1) An upward arrow (↑) appears in the **ANSYS Graphics** window. Move this arrow to the beam area and click this area to mesh.
- (2) The color of the area turns from light blue into pink. Click [A] **OK** button to see the area meshed by 8-node rectangular isoparametric finite elements as shown in Figure 3.21.

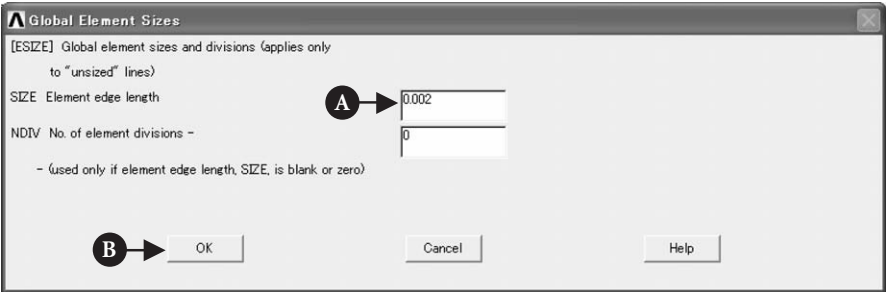


Figure 3.19 “Global Element Sizes” window.

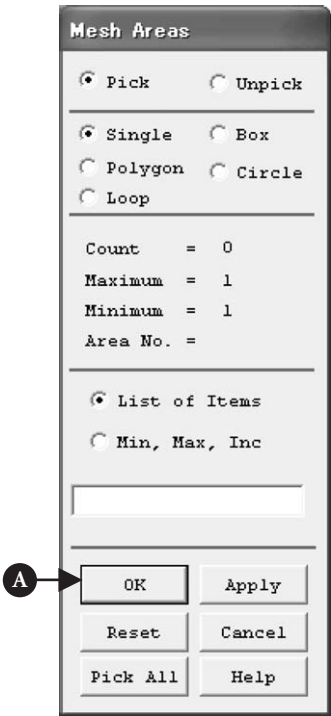


Figure 3.20 “Mesh Areas” window.

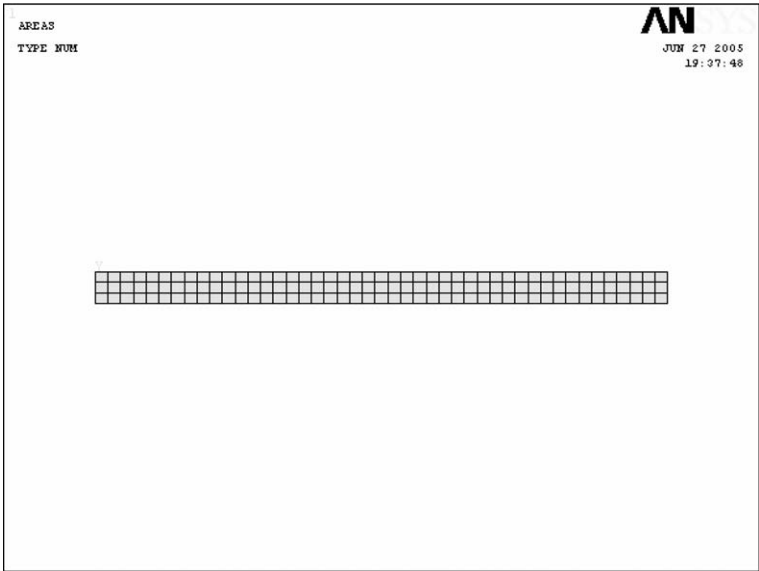


Figure 3.21 Beam area subdivided into 8-node isoparametric rectangular elements.

How to modify meshing

In case of modifying meshing, delete the elements, and repeat the procedures [1] through [4] above. Repeat from the procedures [1], from [2] or from [3] for modifying the element type, for changing the plate thickness without changing the element type, or for changing the element size only.

In order to delete the elements, execute the following commands:

COMMAND | **ANSYS Main Menu → Preprocessor → Meshing → Clear → Areas**

The **Clear Areas** window opens:

- (1) An upward arrow (↑) appears in the **ANSYS Graphics** window. Move this arrow to the beam area and click this area.
- (2) The color of the area turns from light blue into pink. Click **OK** button to delete the elements from the beam area. After this operation, the area disappears from the display. Execute the following commands to replot the area.

COMMAND | **ANSYS Utility Menu → Plot → Areas**

3.1.3.4 INPUT OF BOUNDARY CONDITIONS

Here we will impose constraint and loading conditions on nodes of the beam model. Display the nodes first to define the constraint and loading conditions.

[1] Nodes display

COMMAND | **ANSYS Utility Menu → Plot → Nodes**

The nodes are plotted in the **ANSYS Graphics** window opens as shown in Figure 3.22.



Figure 3.22 Plots of nodes.

[2] Zoom in the nodes display

Enlarged view of the models is often convenient when imposing constraint and loading conditions on the nodes. In order to zoom in the node display, execute the following commands:

COMMAND | ANSYS Utility Menu → PlotCtrls → Pan Zoom Rotate . . .

The **Pan-Zoom-Rotate** window opens as shown in Figure 3.23:

- (1) Click [A] **Box Zoom** button.
- (2) The shape of the mouse cursor turns into a magnifying glass in the **ANSYS Graphics** window. Click the upper left point and then the lower right point which enclose a portion of the beam area to enlarge as shown in Figure 3.24. Zoom in the left end of the beam.
- (3) In order to display the whole view of the beam, click [B] **Fit** button in the “**Pan-Zoom-Rotate**” window.

[3] Definition of constraint conditions**Selection of nodes**

COMMAND | ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Displacement → On Nodes

The **Apply U. ROT on Nodes** window opens as shown in Figure 3.25.

- (1) Select [A] **Box** button and drag the mouse in the **ANSYS Graphics** window so as to enclose the nodes on the left edge of the beam area with the yellow rectangular frame as shown in Figure 3.26. The **Box** button is selected to pick multiple nodes at once, whereas [B] the “**Single**” button is chosen to pick a single node.
- (2) After confirming that only the nodes to impose constraints on are selected, i.e., the nodes on the left edge of the beam area, click [C] **OK** button.

How to reselect nodes Click [D] **Reset** button to clear the selection of the nodes before clicking [C] **OK** button in the procedure (2) above, and repeat the procedures (1) and (2) above. The selection of

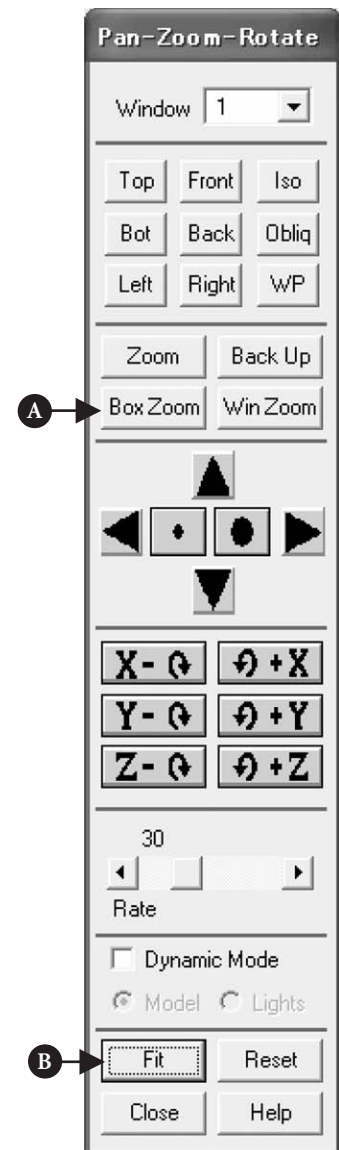


Figure 3.23 “Pan-Zoom-Rotate” window.

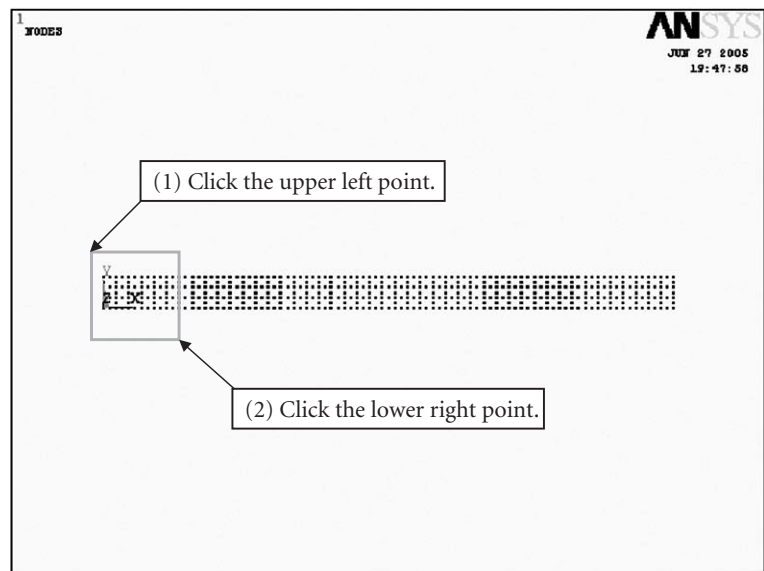


Figure 3.24 Magnification of an observation area.

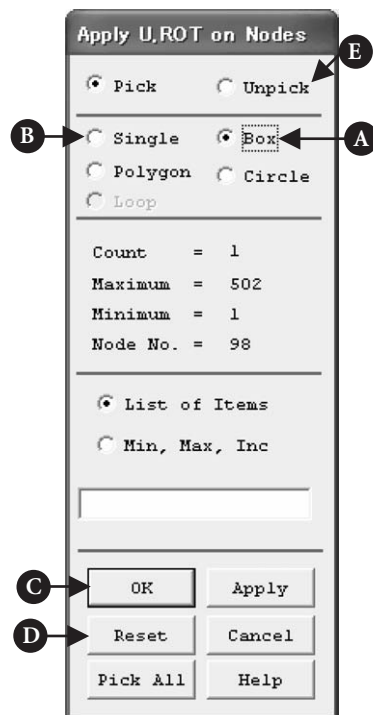


Figure 3.25 "Apply U. ROT on Nodes" window.

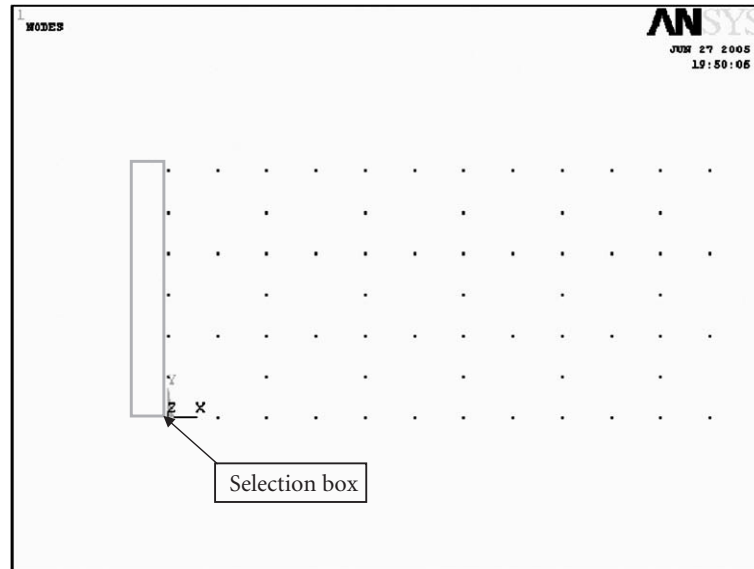


Figure 3.26 Picking multiple nodes by box.

nodes can be reset either by picking selected nodes after choosing [E] **Unpick** button or clicking the right button of the mouse to turn the upward arrow upside down.

Imposing constraint conditions on nodes

The **Apply U, ROT on Nodes** window (see Figure 3.27) opens after clicking [C] **OK** button in the procedure (2) in the subsection “Selection of nodes” above.

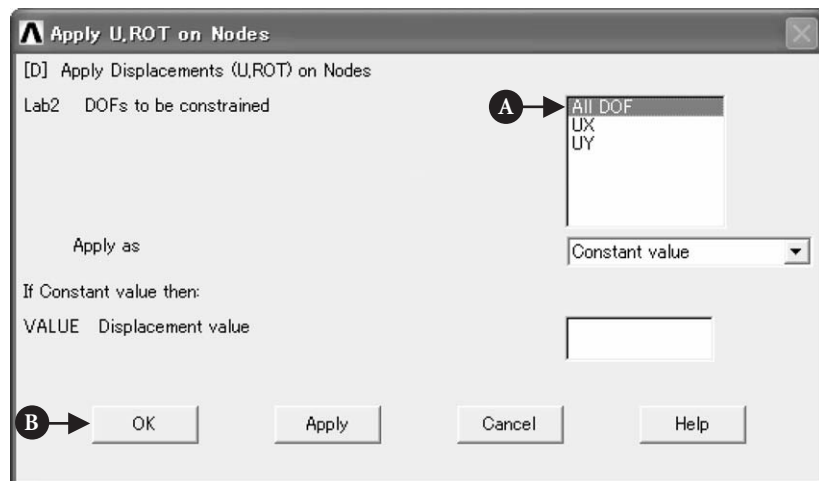


Figure 3.27 “Apply U, ROT on Nodes” window.

- (1) In case of selecting [A] **ALL DOF**, the nodes are to be clamped, i.e., the displacements are set to zero in the directions of the x - and y -axes. Similarly, the selection of **UX** makes the displacement in the x -direction equal to zero and the selection of **UY** makes the displacement in the y -direction equal to zero.
- (2) Click [B] **OK** button and blue triangular symbols, which denote the clamping conditions, appear in the **ANSYS Graphics** window as shown in Figure 3.28. The upright triangles indicate that each node to which the triangular symbol is attached is fixed in the y -direction, whereas the tilted triangles indicate the fixed condition in the x -direction.

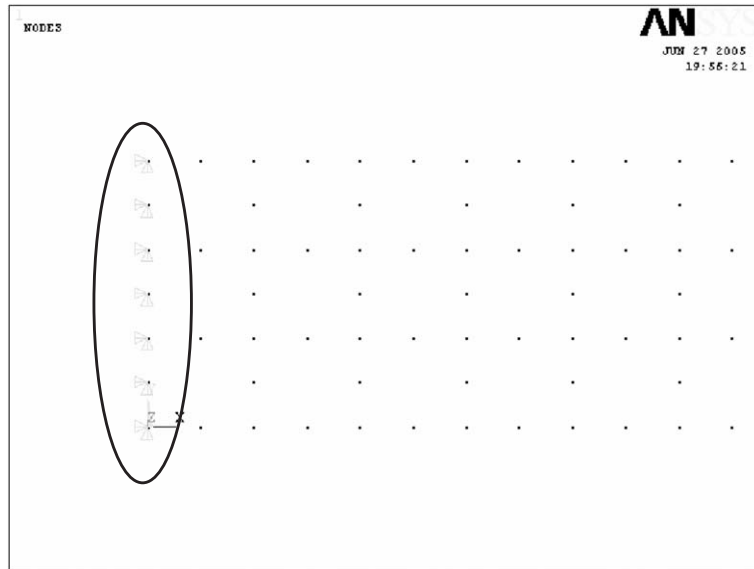


Figure 3.28 Imposing the clamping conditions on nodes.

How to clear constraint conditions

COMMAND | **ANSYS Main Menu → Solution → Define Loads → Delete → Structural → Displacement → On Nodes**

The **Delete Node Constrai...** window opens.

- (1) Click **Pick All** button to delete the constraint conditions of all the nodes that the constraint conditions are imposed. Select **Single** button and pick a particular node by the upward arrow in the **ANSYS Graphics** window and click **OK** button.
- (2) The **Delete Node constraints** window appears. Select **ALL DOF** and click **OK** button to delete the constraint conditions both in the x - and y -directions. Select **UX** and **UY** to delete the constraints in the x - and the y -directions, respectively.

[4] Imposing boundary conditions on nodes

Before imposing load conditions, click **Fit** button in the **Pan-Zoom-Rotate** window (see Figure 3.23) to get the whole view of the area and then zoom in the right end of the beam area for ease of the following operations.

Selection of the nodes

(1) Pick the node at a point where $x = 0.08$ m and $y = 0.005$ m for this purpose, click

COMMAND | ANSYS Utility Menu → PlotCtrls → Numbering . . .

consecutively to open the **Plot Numbering Controls** window as shown in Figure 3.29.

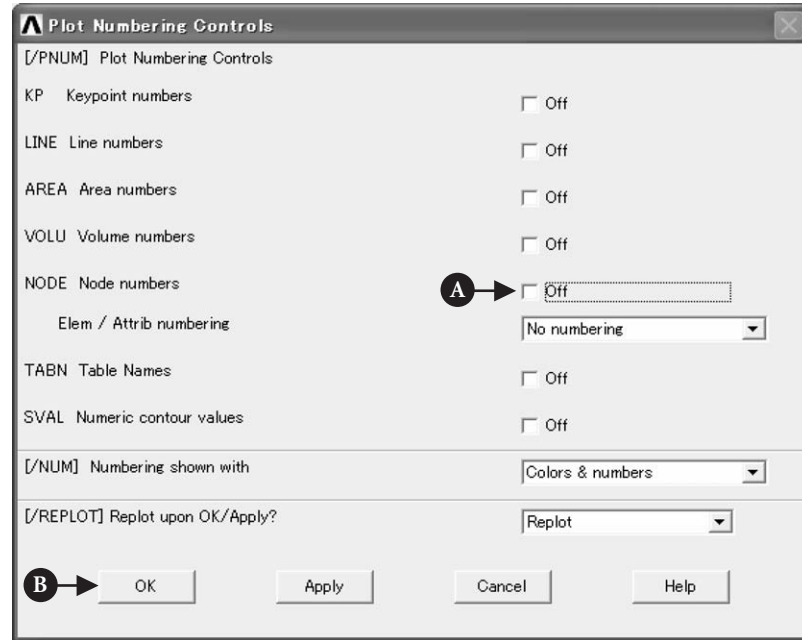


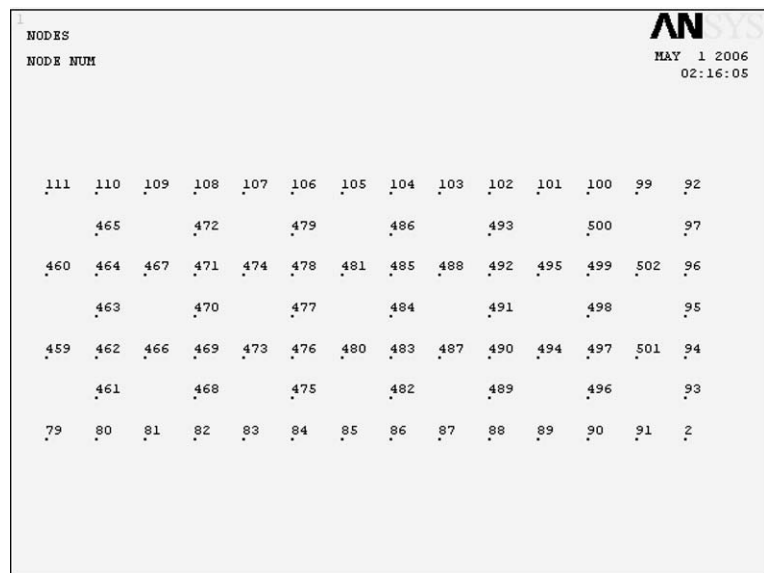
Figure 3.29 “Plot Numbering Controls” window.

- (2) Click [A] **NODE** ☐ **Off** box to change it to ☒ **On** box.
- (3) Click [B] **OK** button to display node numbers adjacent to corresponding nodes in the **ANSYS Graphics** window as shown in Figure 3.30.
- (4) To delete the node numbers, click [A] **NODE** ☒ **On** box again to change it to ☐ **Off** box.
- (5) Execute the following commands:

COMMAND | ANSYS Utility Menu → List → Nodes . . .

and the **Sort NODE Listing** window opens (see Figure 3.31). Select [A] **Coordinates only** button and then click [B] **OK** button.

- (6) The “**NLIST Command**” window opens as shown in Figure 3.32. Find the number of the node having the coordinates $x = 0.08$ m and $y = 0.005$ m, namely node #108 in Figure 3.32.



The image shows the ANSYS Graphics window with a list of nodes and nodal numbers. The window title is "AN". The date and time are "MAY 1 2006 02:16:05". The text "NODES" and "NODE NUM" is displayed at the top left. The list of nodes and nodal numbers is as follows:

NODES	
NODE NUM	
111	110
109	108
107	106
105	104
103	102
101	100
99	92
465	472
479	486
493	500
97	
460	464
467	471
474	478
481	485
488	492
495	499
502	96
463	470
477	484
491	498
95	
459	462
466	469
473	476
480	483
487	490
494	497
501	94
461	468
475	482
489	496
93	
79	80
81	82
83	84
85	86
87	88
89	90
91	2

Figure 3.30 Nodes and nodal numbers displayed on the ANSYS Graphics window.

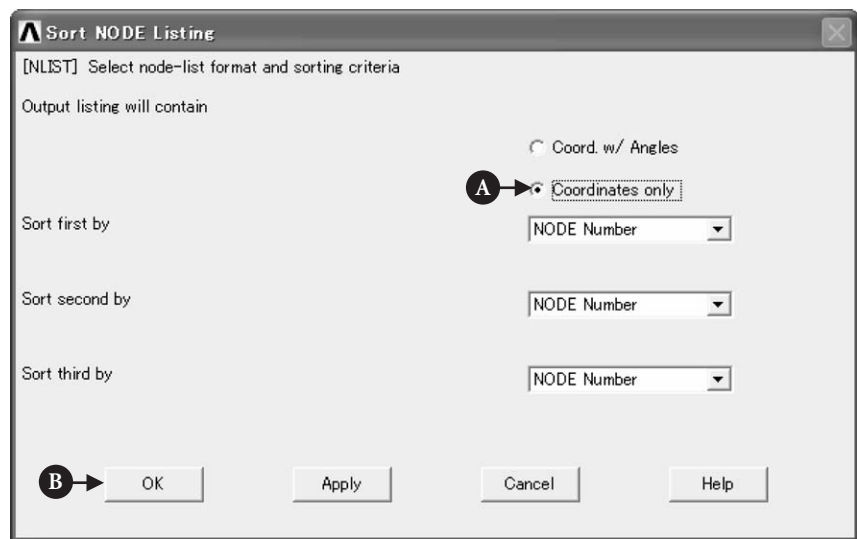
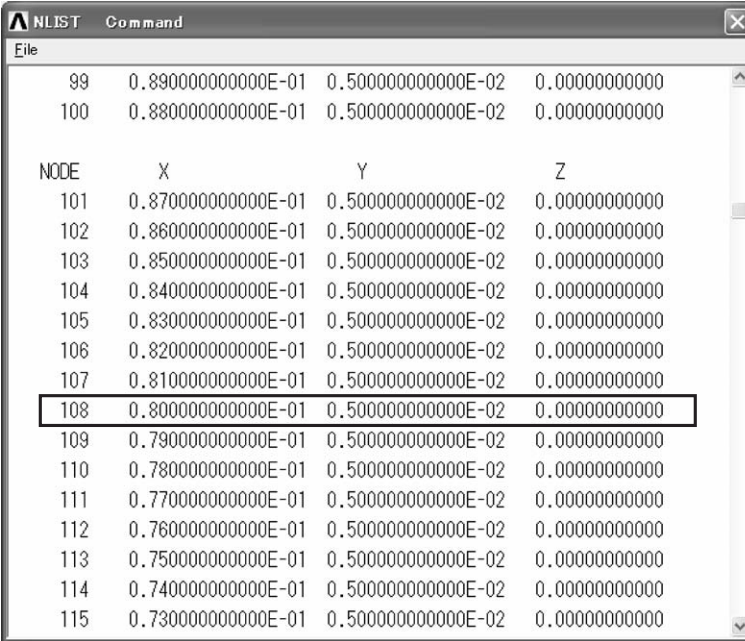


Figure 3.31 "Sort NODE Listing" window.

(7) Execute the following commands:

COMMAND | ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Force/
Moment → On Nodes



File				
99	0.890000000000E-01	0.500000000000E-02	0.000000000000	
100	0.880000000000E-01	0.500000000000E-02	0.000000000000	
NODE	X	Y	Z	
101	0.870000000000E-01	0.500000000000E-02	0.000000000000	
102	0.860000000000E-01	0.500000000000E-02	0.000000000000	
103	0.850000000000E-01	0.500000000000E-02	0.000000000000	
104	0.840000000000E-01	0.500000000000E-02	0.000000000000	
105	0.830000000000E-01	0.500000000000E-02	0.000000000000	
106	0.820000000000E-01	0.500000000000E-02	0.000000000000	
107	0.810000000000E-01	0.500000000000E-02	0.000000000000	
108	0.800000000000E-01	0.500000000000E-02	0.000000000000	
109	0.790000000000E-01	0.500000000000E-02	0.000000000000	
110	0.780000000000E-01	0.500000000000E-02	0.000000000000	
111	0.770000000000E-01	0.500000000000E-02	0.000000000000	
112	0.760000000000E-01	0.500000000000E-02	0.000000000000	
113	0.750000000000E-01	0.500000000000E-02	0.000000000000	
114	0.740000000000E-01	0.500000000000E-02	0.000000000000	
115	0.730000000000E-01	0.500000000000E-02	0.000000000000	

Figure 3.32 “NLIST Command” window showing the coordinates of the nodes; the framed portion indicates the coordinates of the node for load application.

to open the **Apply F/M on Nodes** window (see Figure 3.33)

- (8) Pick only the #108 node having the coordinates $x = 0.08$ m and $y = 0.005$ m with the upward arrow as shown in Figure 3.34.
- (9) After confirming that only the #108 node is enclosed with the yellow frame, click [A] **OK** button in the **Apply F/M on Nodes** window.

How to cancel the selection of the nodes of load application Click **Reset** button before clicking **OK** button or click the right button of the mouse to change the upward arrow to the downward arrow and click the yellow frame. The yellow frame disappears and the selection of the node(s) of load application is canceled.

Imposing load conditions on nodes

Click [A] **OK** in the **Apply F/M on Nodes** window to open another **Apply F/M on Nodes** window as shown in Figure 3.35.

- (1) Choose [A] **FY** in the **Lab Direction of force/mom** box and input [B] **–100** (N) in the **VALUE** box. A positive value for load indicates load in the upward or rightward direction, whereas a negative value load in the downward or leftward direction.
- (2) Click [C] **OK** button to display the red downward arrow attached to the #108 node indicating the downward load applied to that point as shown in Figure 3.36.

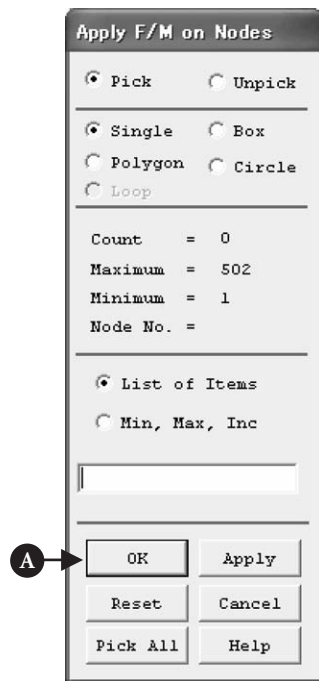


Figure 3.33 “Apply F/M on Nodes” window.

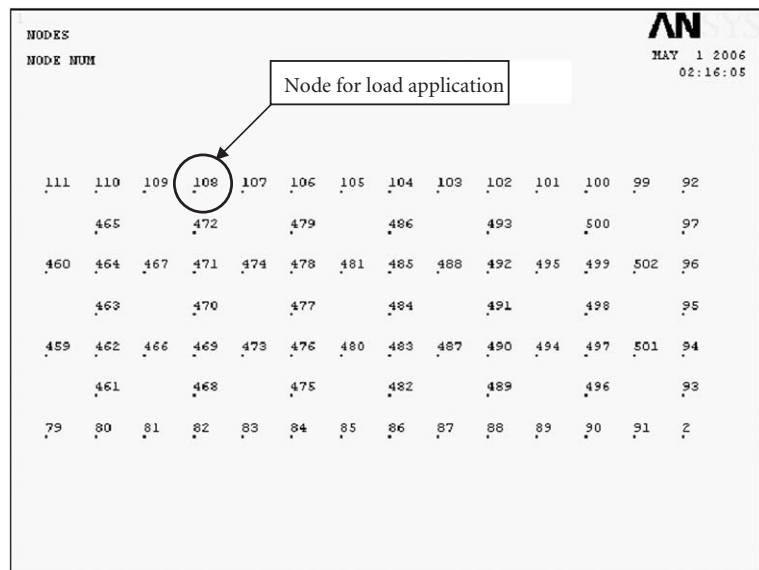


Figure 3.34 Selection of a node for load application.

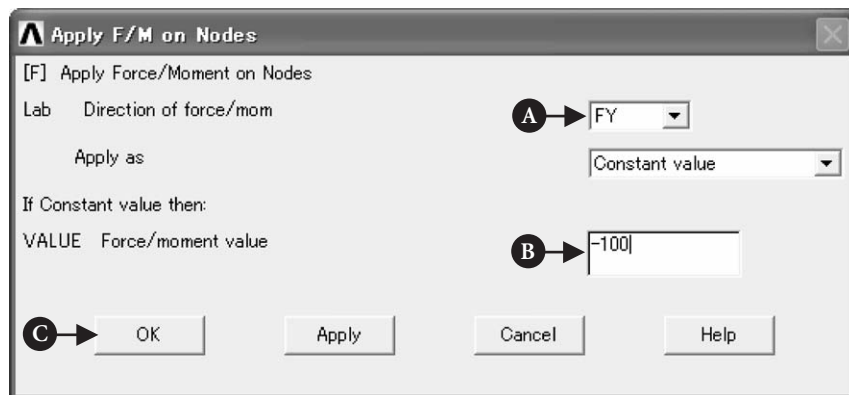


Figure 3.35 “Apply F/M on Nodes” window.

How to delete load conditions Execute the following commands:

COMMAND | ANSYS Main Menu → Solution → Define Loads → Delete → Structural → Force/
Moment → On Nodes

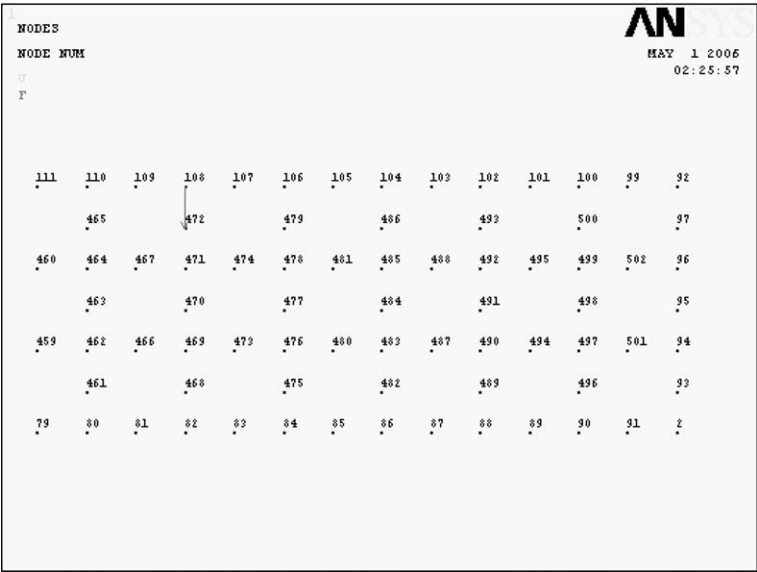


Figure 3.36 Display of the load application on a node by arrow symbol.

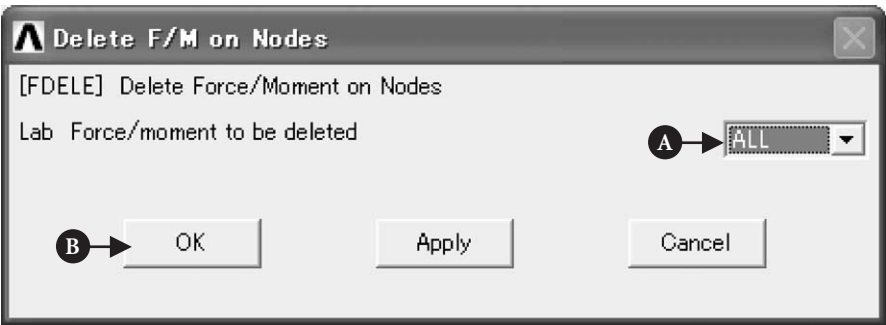


Figure 3.37 “Delete F/M on Nodes” window.

to open the **Delete F/M on Nodes** window (see Figure 3.37). Choose [A] **FY** or **ALL** in the **Lab Force/moment to be deleted** and click **OK** button to delete the downward load applied to the #108 node.

3.1.3.5 SOLUTION PROCEDURES

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

The **Solve Current Load Step** and **/STATUS Command** windows appear as shown in Figures 3.38 and 3.39, respectively.



Figure 3.38 “Solve Current Load Step” window.

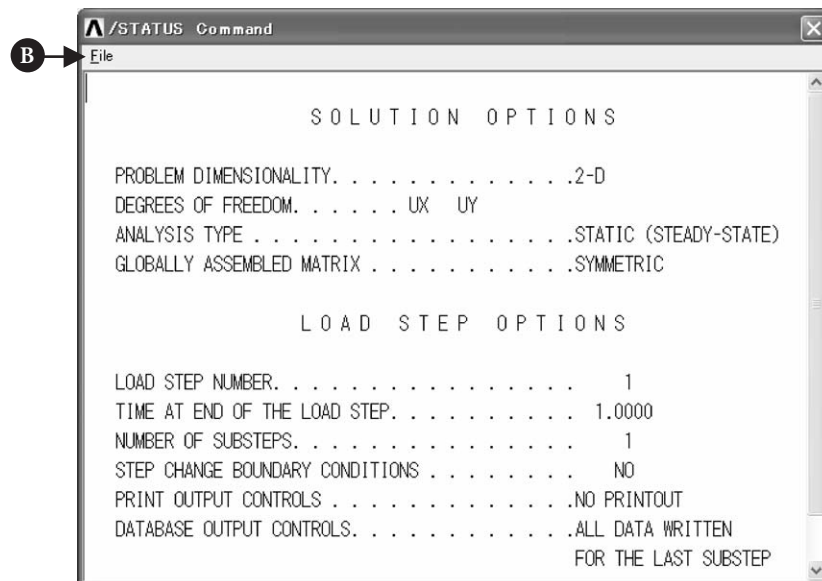


Figure 3.39 “/STATUS Command” window.

- (1) Click [A] **OK** button in the **Solve Current Load Step** window as shown in Figure 3.38 to begin the solution of the current load step.
- (2) The **/STATUS Command** window displays information on solution and load step options. Select [B] **File** button to open the submenu and select **Close** button to close the **/STATUS Command** window.
- (3) When solution is completed, the **Note** window (see Figure 3.40) appears. Click [C] **Close** button to close the window.

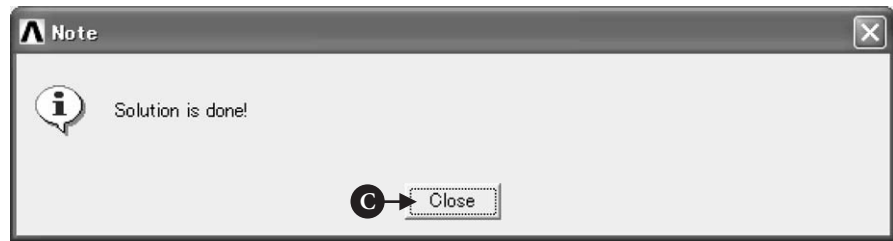


Figure 3.40 “Note” window.

3.1.3.6 GRAPHICAL REPRESENTATION OF THE RESULTS

[1] Contour plot of displacements

COMMAND | ANSYS Main Menu → General Postproc → Plot Results → Contour Plot → Nodal Solution

The **Contour Nodal Solution Data** window opens as shown in Figure 3.41.

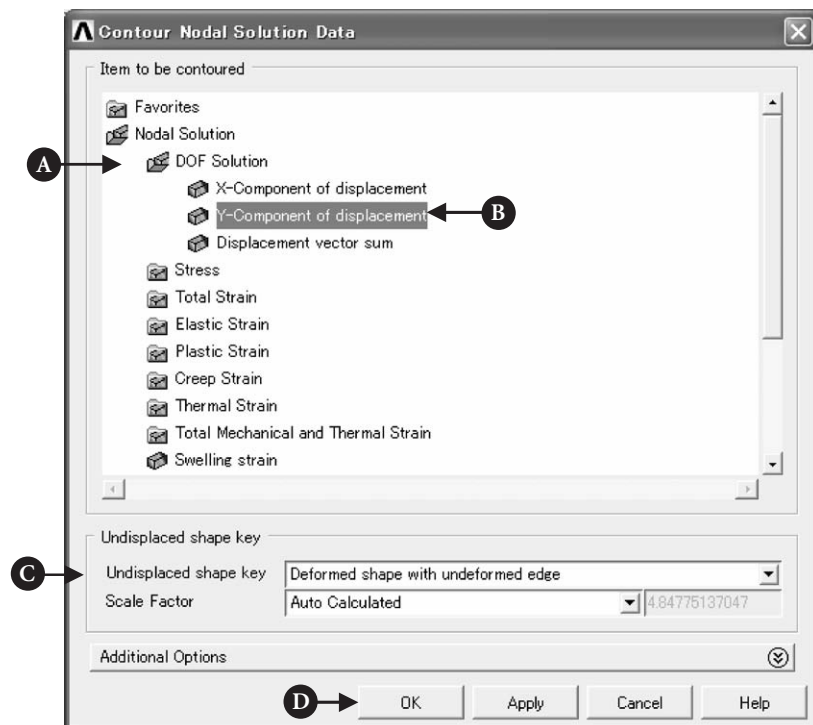


Figure 3.41 “Contour Nodal Solution Data” window.

- (1) Select [A] **DOF Solution** and [B] **Y-Component of displacement**.
- (2) Select [C] **Deformed shape with undeformed edge** in the **Undisplaced shape** key box to compare the shapes of the beam before and after deformation.

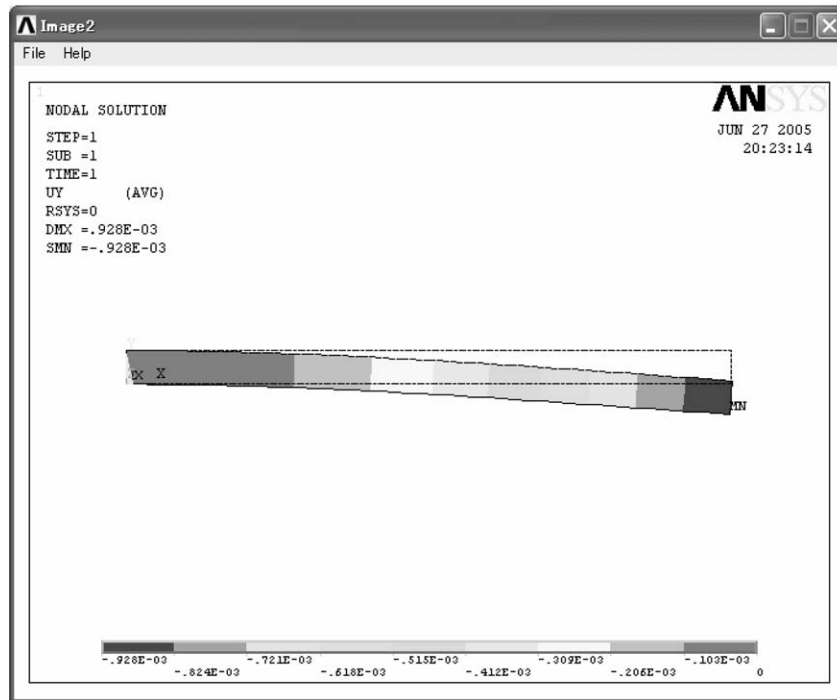


Figure 3.42 Contour map representation of the distribution of displacement in the y - or vertical direction.

- (3) Click [D] **OK** button to display the contour of the y -component of displacement, or the deflection of the beam in the **ANSYS Graphics** window (see Figure 3.42). The **DMX** value shown in the Graphics window indicates the maximum deflection of the beam.

[2] Contour plot of stresses

- (1) Select [A] **Stress** and [B] **X-Component of stress** as shown in Figure 3.43.
- (2) Click [C] **OK** button to display the contour of the x -component of stress, or the bending stress in the beam in the **ANSYS Graphics** window (see Figure 3.44). The **SMX** and **SMN** values shown in the Graphics window indicate the maximum and the minimum stresses in the beam, respectively.
- (3) Click [D] **Additional Options** bar to open additional option items to choose. Select [E] **All applicable** in the **Number of facets per element edge** box to calculate stresses and strains at middle points of the elements.

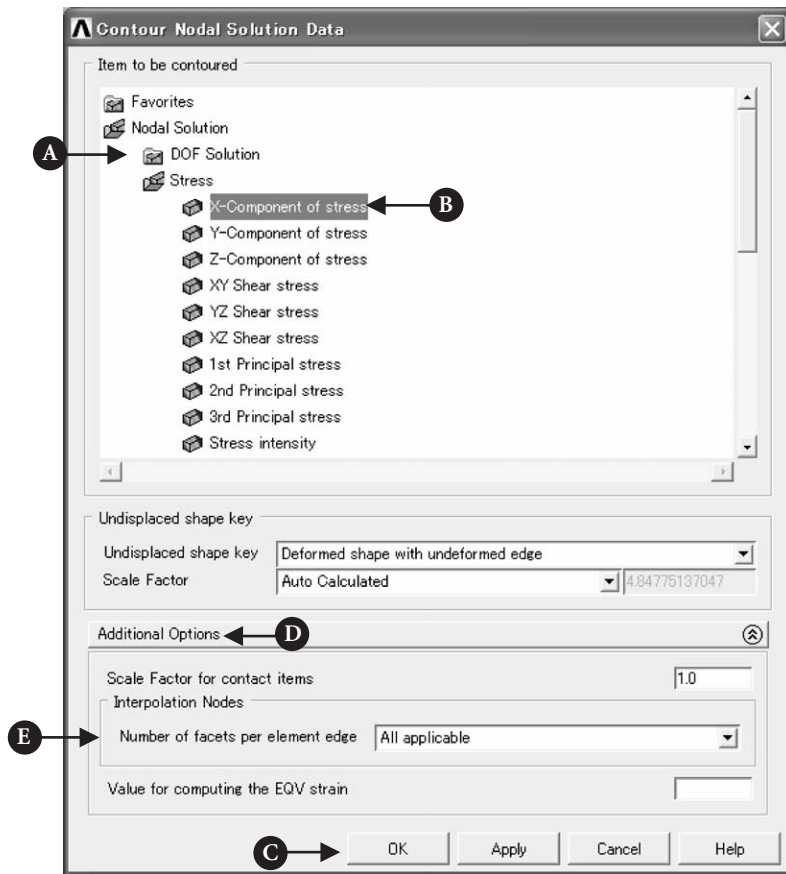


Figure 3.43 “Contour Nodal Solution Data” window.

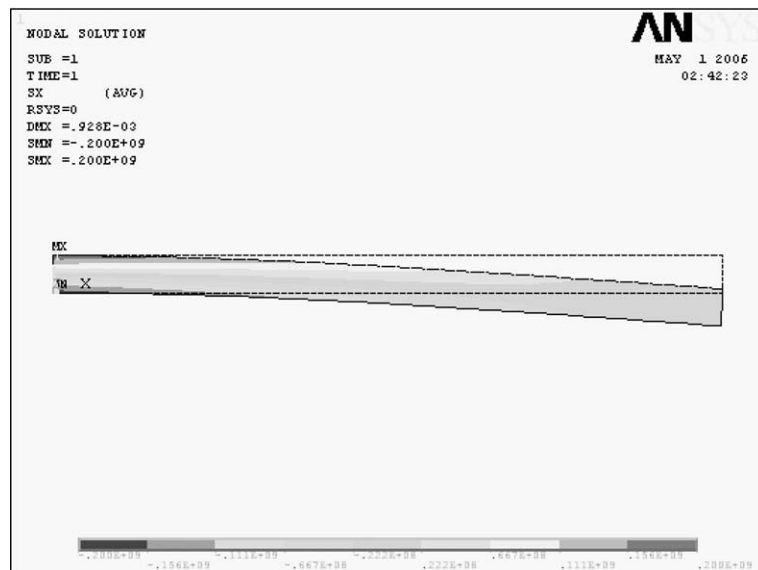


Figure 3.44 Contour map representation of the distribution of normal stress in the x- or horizontal direction.

3.1.4 COMPARISON OF FEM RESULTS WITH EXPERIMENTAL ONES

Figure 3.45 compares longitudinal stress distributions obtained by ANSYS with those by experiments and by the elementary beam theory. The results obtained by three different methods agree well with one another. As the applied load increases, however, errors among the three groups of the results become larger, especially at the clamped end. This tendency arises from the fact that the clamped condition can be hardly realized in the strict sense.

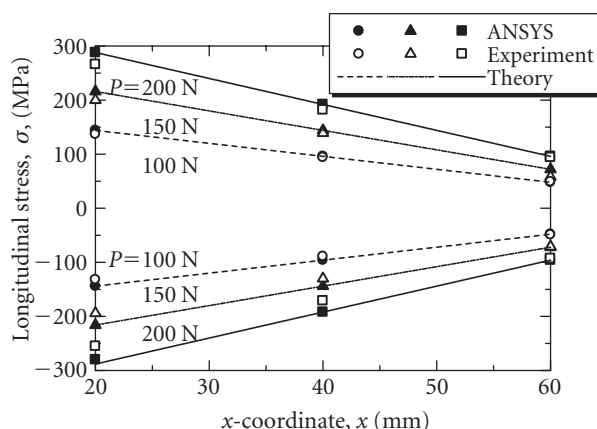


Figure 3.45 Comparison of longitudinal stress distributions obtained by ANSYS with those by experiments and by the elementary beam theory.

3.1.5 PROBLEMS TO SOLVE

PROBLEM 3.1

Change the point of load application and the intensity of the applied load in the cantilever beam model shown in Figure 3.3 and calculate the maximum deflection.

PROBLEM 3.2

Calculate the maximum deflection in a beam clamped at the both ends as shown in Figure P3.2 where the thickness of the beam in the direction perpendicular to the page surface is 10 mm.

(Answer: 0.00337 mm)

PROBLEM 3.3

Calculate the maximum deflection in a beam simply supported at the both ends as shown in Figure P3.3 where the thickness of the beam in the direction perpendicular to the page surface is 10 mm.

(Answer: 0.00645 mm)

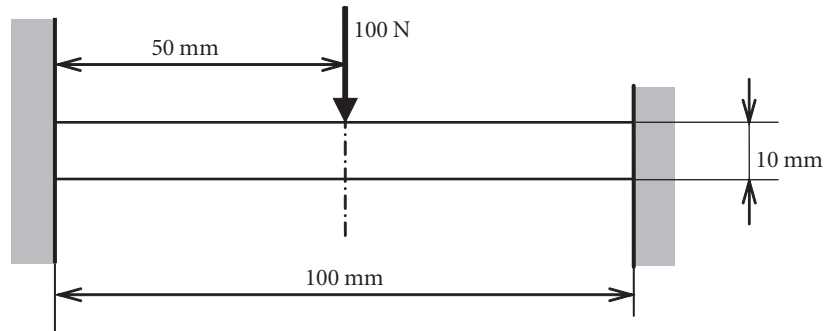


Figure P3.2 A beam clamped at the both ends and subjected to a concentrated force of 100 N at the center of the span.

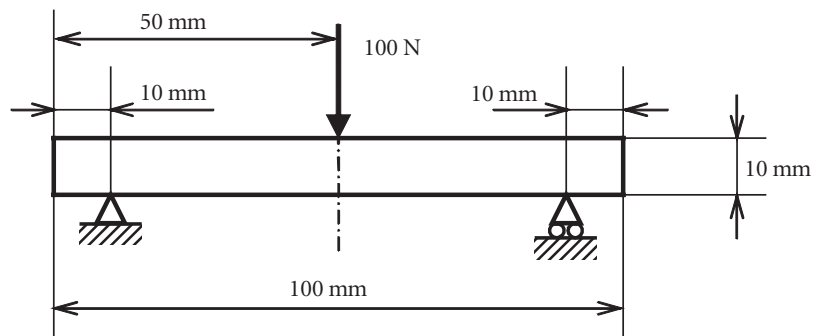


Figure P3.3 A beam simply supported at the both ends and subjected to a concentrated force of 100 N at the center of the span.

PROBLEM 3.4

Calculate the maximum deflection in a beam shown in Figure P3.4 where the thickness of the beam in the direction perpendicular to the page surface is 10 mm. Choose an element size of 1 mm.

(Answer: 0.00337 mm)

Note that the beam shown in Figure P3.2 is bilaterally symmetric so that the x -component of the displacement (DOF X) is zero at the center of the beam span. If the beam shown in Figure P3.2 is cut at the center of the span and the finite-element calculation is made for only the left half of the beam by applying a half load of 50 N to its right end which is fixed in the x -direction but is deformed freely in the y -direction as shown in Figure P3.4, the solution obtained is the same as that for the left half of the beam in Problem 3.2. Problem 3.2 can be solved by its half model as shown in Figure P3.4. A half model can achieve the efficiency of finite-element calculations.

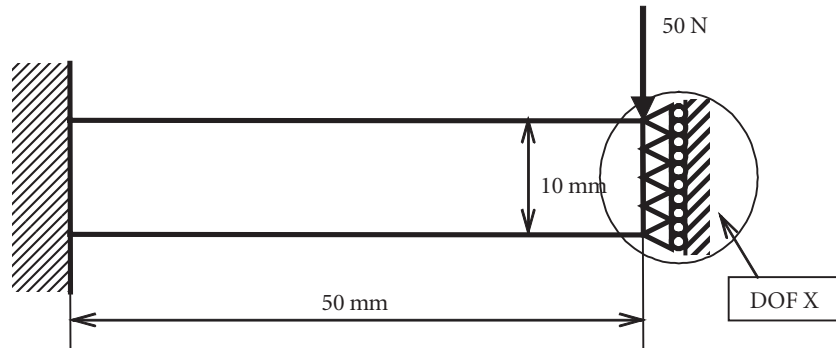


Figure P3.4 A half model of the beam in Problem 3.2.

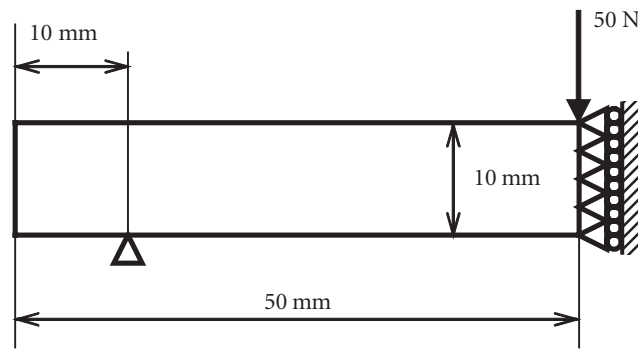


Figure P3.5 A half model of the beam in Problem 3.3.

PROBLEM 3.5

Calculate the maximum deflection in a beam shown in Figure P3.5 where the thickness of the beam in the direction perpendicular to the page surface is 10 mm. This beam is the half model of the beam of Problem 3.3.

(Answer: 0.00645 mm)

PROBLEM 3.6

Calculate the maximum value of the von Mises stress in the stepped beam as shown in Figure P3.6 where Young's modulus $E = 210$ GPa, Poisson's ratio $\nu = 0.3$, the element size is 2 mm and the beam thickness is 10 mm. Refer to the Appendix to create the stepped beam. The von Mises stress σ_{eq} is sometimes called the equivalent stress or

the effective stress and is expressed by the following formula:

$$\sigma_{eq} = \frac{1}{\sqrt{2}} \sqrt{(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)} \quad (\text{P3.6})$$

in three-dimensional elasticity problems. It is often considered that a material yields when the value of the von Mises stress reaches the yield stress of the material σ_Y which is determined by the uniaxial tensile tests of the material.

(Answer: 40.8 MPa)

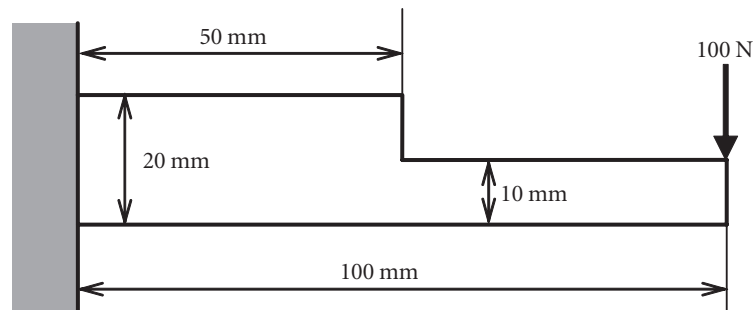


Figure P3.6 A stepped cantilever beam subjected to a concentrated force of 100 N at the free end.

PROBLEM 3.7

Calculate the maximum value of the von Mises stress in the stepped beam with a rounded fillet as shown in Figure P3.7 where Young's modulus $E = 210$ GPa, Poisson's ratio $\nu = 0.3$, the element size is 2 mm and the beam thickness is 10 mm. Refer to the Appendix to create the stepped beam with a rounded fillet.

(Answer: 30.2 MPa)

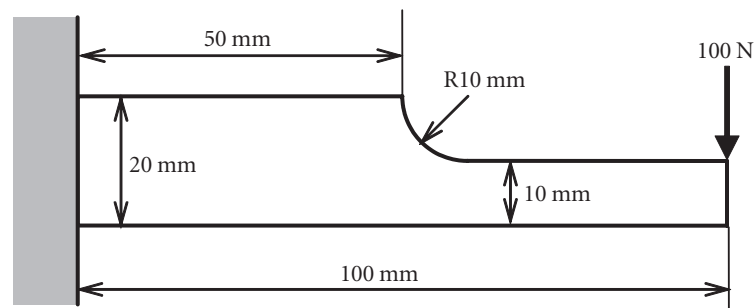


Figure P3.7 A stepped cantilever beam with a rounded fillet subjected to a concentrated force of 100 N at the free end.

APPENDIX: PROCEDURES FOR CREATING STEPPED BEAMS

A3.1 CREATION OF A STEPPED BEAM

A stepped beam as shown in Figure P3.6 can be created by adding two rectangular areas of different sizes:

- (1) Create two rectangular areas of different sizes, say 50 mm by 20 mm with WP X = -50 mm and WP Y = -10 mm, and 60 mm by 10 mm with WP X = -10 mm and WP Y = -10 mm, following operations described in 3.1.3.1.
- (2) Select the Boolean operation of adding areas as follows to open the **Add Areas** window (see Figure 3.46)

COMMAND | ANSYS Main Menu → Preprocessor →
Modeling → Operate → Booleans →
Add → Areas

- (3) Pick all the areas to add by the upward arrow.
- (4) The color of the areas picked turns from light blue into pink (see Figure 3.47). Click [A] **OK** button to add the two rectangular areas to create a stepped beam area as shown in Figure 3.48.

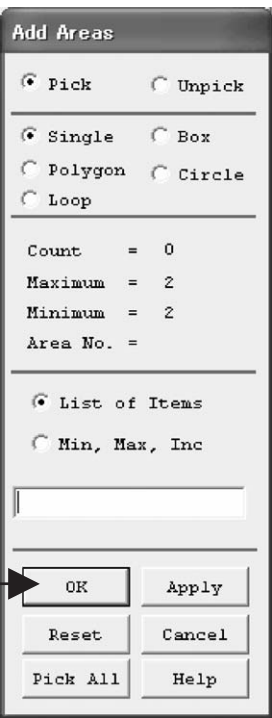


Figure 3.46 “Add Areas” window.

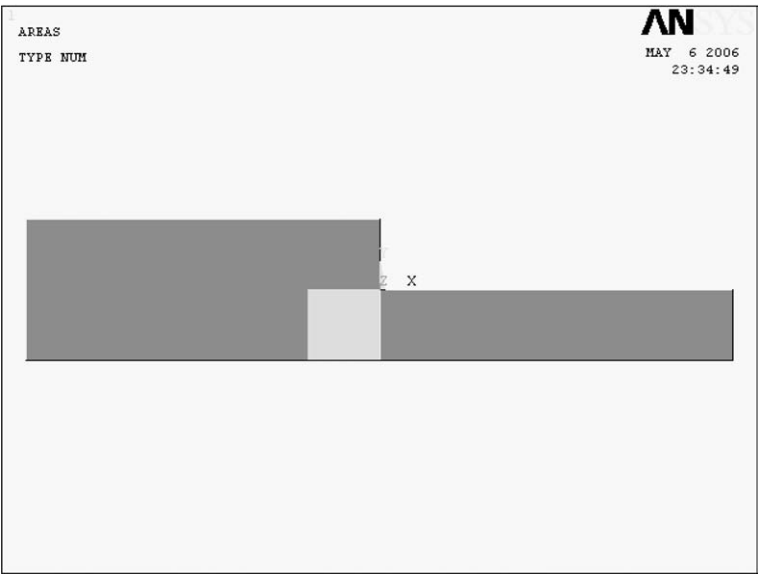


Figure 3.47 Two rectangular areas of different sizes to be added to create a stepped beam area.

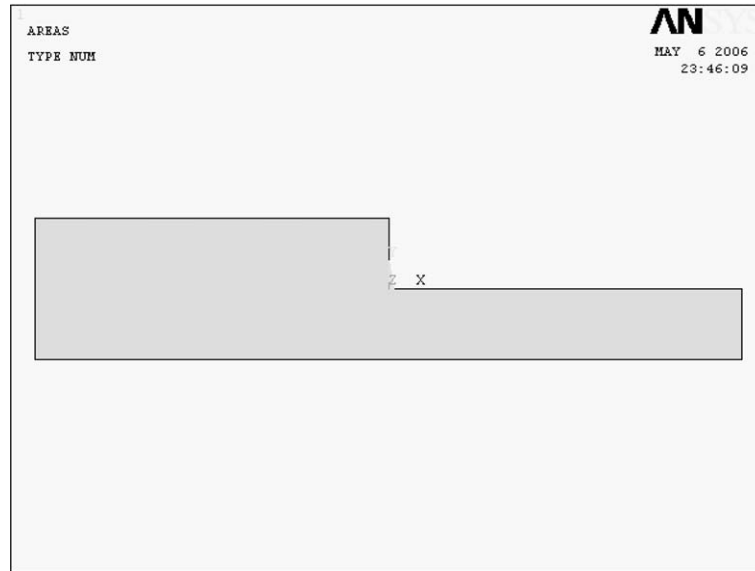


Figure 3.48 A stepped beam area created by adding two rectangle areas.

A3.1.1 HOW TO CANCEL THE SELECTION OF AREAS

Click **Reset** button or click the right button of the mouse to change the upward arrow to the downward arrow and click the area(s) to pick. The color of the unpicked area(s) turns pink into light blue and the selection of the area(s) is canceled.

A3.2 CREATION OF A STEPPED BEAM WITH A ROUNDED FILLET

A stepped beam with a rounded fillet as shown in Figure P3.7 can be created by subtracting a smaller rectangular area and a solid circle from a larger rectangular area:

- (1) Create a larger rectangular area of 100 mm by 20 mm with WP X = -100 mm and WP Y = -10 mm, a smaller rectangular area of, say 50 mm by 15 mm with WP X = 10 mm and WP Y = 0 mm, and a solid circular area having a diameter of 10 mm with WP X = 10 mm and WP Y = 10 mm as shown in Figure 3.49. The solid circular area can be created by executing the following operation:

COMMAND | ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Circle → Solid Circle

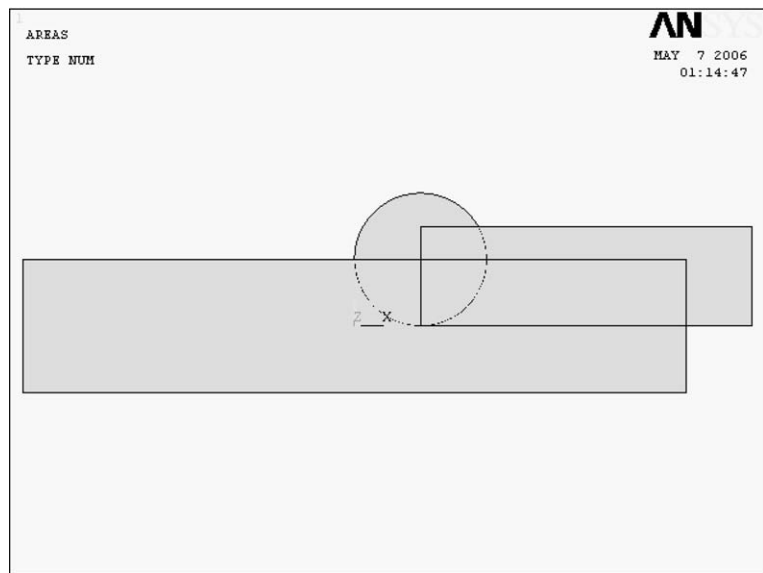


Figure 3.49 A larger rectangular area, a smaller rectangular area and a solid circular area to create a stepped beam area with a rounded fillet.

to open the **Solid Circular Area** window. Input the coordinate of the center ([A] **WP X**, [B] **WP Y**) and [C] **Radius** of the solid circle and click [D] **OK** button as shown in Figure 3.50.

- (2) Select the Boolean operation of subtracting areas as follows to open the **Subtract Areas** window (see Figure 3.51):

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

- (3) Pick the larger rectangular area by the upward arrow as shown in Figure 3.52 and the color of the area picked turns from light blue into pink. Click [A] **OK** button.
- (4) Pick the smaller rectangular area and the solid circular area by the upward arrow as shown in Figure 3.53 and the color of the two areas picked turns from light blue into pink. Click [A] **OK** button to subtract the smaller rectangular and solid

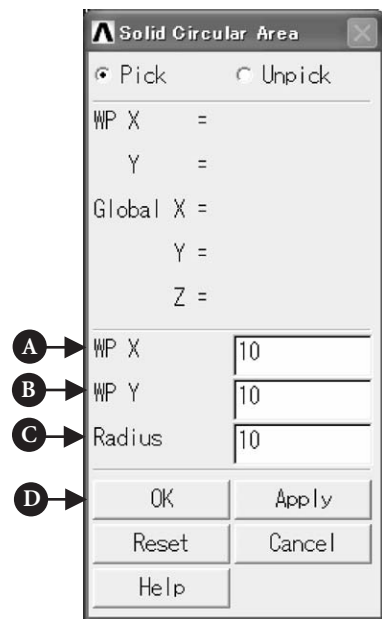


Figure 3.50 “Solid Circular Area” window.

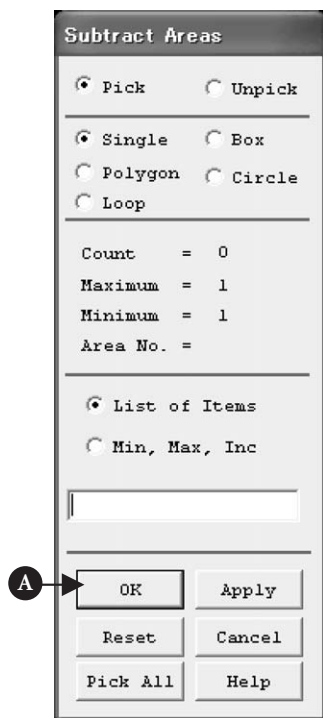


Figure 3.51 “Subtract Areas” window.

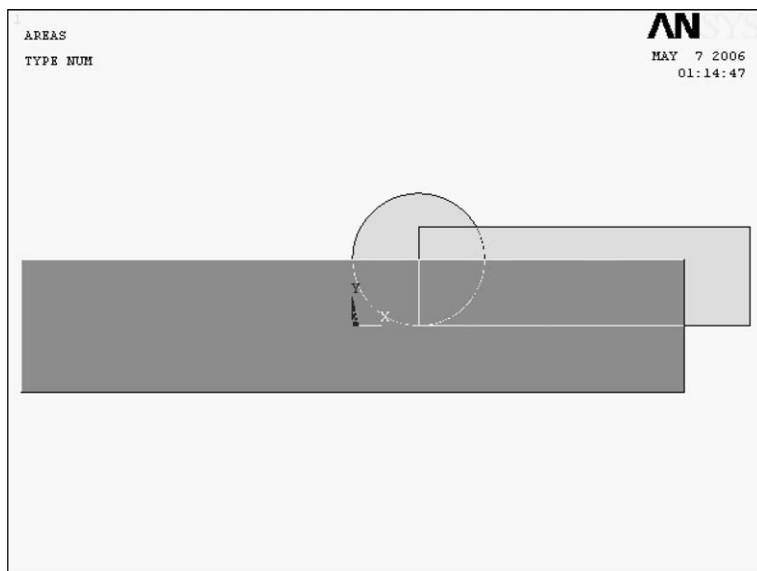


Figure 3.52 The larger rectangular area picked.

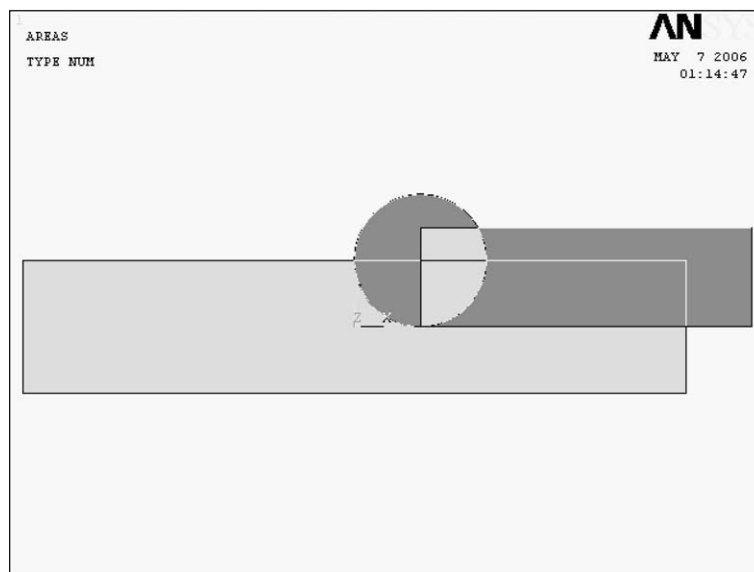


Figure 3.53 The smaller rectangular area and the solid circular area picked to be subtracted from the larger rectangular area to create a stepped beam area with a rounded fillet.

circular areas from the larger rectangular area to create a stepped beam area with a rounded fillet as shown in Figure 3.54.

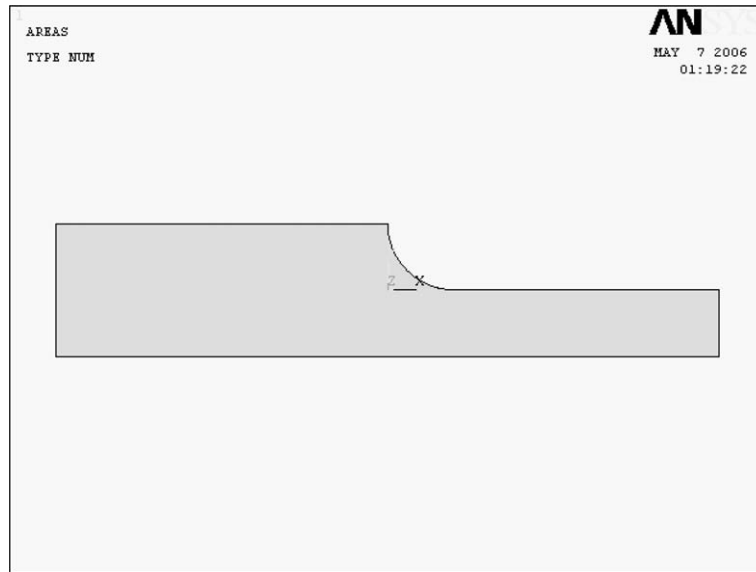


Figure 3.54 A stepped cantilever beam area with a rounded fillet.

A3.2.1 HOW TO DISPLAY AREA NUMBERS

Area numbers can be displayed in the “ANSYS Graphics” window by the following procedure.

COMMAND | ANSYS Utility Menu → PlotCtrls → Numbering . . .

- (1) The **Plot Numbering Controls** window opens as shown in Figure 3.29.
- (2) Click **AREA** ☐ **Off** box to change it to ☒ **On** box.
- (3) Click **OK** button to display area numbers in the corresponding areas in the **ANSYS Graphics** window.
- (4) To delete the area numbers, click **AREA** ☒ **On** box again to change it to ☐ **Off** box.

3.2 THE PRINCIPLE OF ST. VENANT

3.2.1 EXAMPLE PROBLEM

An elastic strip subjected to distributed uniaxial tensile stress or negative pressure at one end and clamped at the other end.

Perform an FEM analysis of a 2-D elastic strip subjected to a distributed stress in the longitudinal direction at one end and clamped at the other end (shown in Figure 3.55 below) and calculate the stress distributions along the cross sections at different distances from the loaded end in the strip.

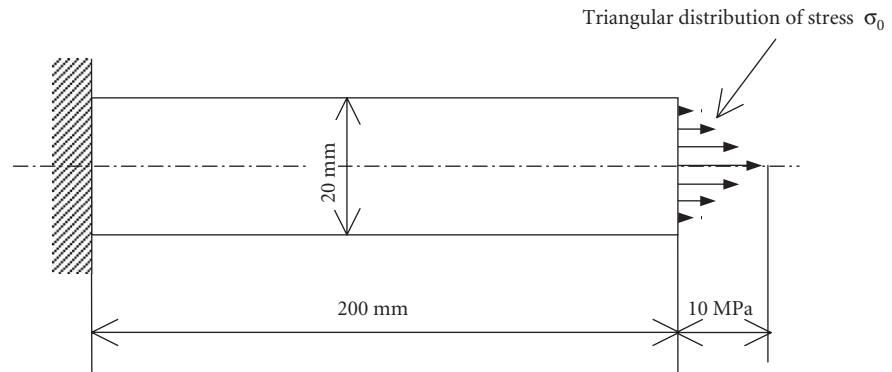


Figure 3.55 A 2-D elastic strip subjected to a distributed force in the longitudinal direction at one end and clamped at the other end.

3.2.2 PROBLEM DESCRIPTION

Geometry: length $l = 200$ mm, height $h = 20$ mm, thickness $b = 10$ mm.

Material: mild steel having Young's modulus $E = 210$ GPa and Poisson's ratio $\nu = 0.3$.

Boundary conditions: The elastic strip is subjected to a triangular distribution of stress in the longitudinal direction at the right end and clamped to a rigid wall at the left end.

3.2.3 ANALYTICAL PROCEDURES

3.2.3.1 CREATION OF AN ANALYTICAL MODEL

COMMAND | ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Rectangle → By 2 Corners

- (1) Input two 0's into the "WP X" and "WP Y" boxes in the "Rectangle by 2 Corners" window to determine the lower left corner point of the elastic strip on the Cartesian coordinates of the working plane.
- (2) Input 200 and 20 (mm) into the Width and Height boxes, respectively, to determine the shape of the elastic strip model.
- (3) Click the OK button to create the rectangular area, or beam on the ANSYS Graphics window.

In the procedures above, the geometry of the strip is input in millimeters. You must decide what kind of units to use in finite-element analyses. When you input the geometry of a model to analyze in millimeters, for example, you must input applied loads in N (Newton) and Young's modulus in MPa, since 1 MPa is equivalent to 1 N/mm^2 . When you use meters and N as the units of length and load, respectively, you must input Young's modulus in Pa, since 1 Pa is equivalent to 1 N/m^2 . You can choose any system of unit you would like to, but your unit system must be consistent throughout the analyses.

3.2.3.2 INPUT OF THE ELASTIC PROPERTIES OF THE STRIP MATERIAL

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

- (1) The **Define Material Model Behavior** window opens.
- (2) Double-click **Structural**, **Linear**, **Elastic**, and **Isotropic** buttons one after another.
- (3) Input the value of Young's modulus, **2.1e5** (MPa), and that of Poisson's ratio, **0.3**, into **EX** and **PRXY** boxes, and click the **OK** button of the **Linear Isotropic Properties for Materials Number 1** window.
- (4) Exit from the **Define Material Model Behavior** window by selecting **Exit** in the **Material** menu of the window.

3.2.3.3 FINITE-ELEMENT DISCRETIZATION OF THE STRIP AREA

[1] Selection of the element type

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

- (1) The **Element Types** window opens.
- (2) Click the **Add ...** button in the **Element Types** window to open the **Library of Element Types** window and select the element type to use.
- (3) Select **Structural Mass – Solid** and **Quad 8node 82**.
- (4) Click the **OK** button in the **Library of Element Types** window to use the 8-node isoparametric element.
- (5) Click the **Options ...** button in the **Element Types** window to open the **PLANE82 element type options** window. Select the **Plane strs w/thk** item in the **Element behavior** box and click the **OK** button to return to the **Element Types** window. Click the **Close** button in the **Element Types** window to close the window.

[2] Input of the element thickness

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

- (1) The **Real Constants** window opens.
- (2) Click [A] **Add/Edit/Delete** button to open the **Real Constants** window and click the **Add ...** button.

- (3) The **Element Type for Real Constants** window opens. Click the **OK** button.
- (4) The **Element Type for Real Constants** window vanishes and the **Real Constants Set Number 1. for PLANE82** window appears instead. Input a strip thickness of 10 (mm) in the **Thickness** box and click the **OK** button.
- (5) The **Real Constants** window returns with the display of the **Defined Real Constants Sets** box changed to **Set 1**. Click the **Close** button.

[3] Sizing of the elements

COMMAND | ANSYS Main Menu → Preprocessor → Meshing → Size Cntrls → Manual Size → Global → Size

- (1) The **Global Element Sizes** window opens.
- (2) Input 2 in the **SIZE** box and click the **OK** button.

[4] Dividing the right-end side of the strip area into two lines

Before proceeding to meshing, the right-end side of the strip area must be divided into two lines for imposing the triangular distribution of the applied stress or pressure by executing the following commands.

COMMAND | ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Divide → Lines w/Options

- (1) The **Divide Multiple Lines ...** window opens as shown in Figure 3.56.
- (2) When the mouse cursor is moved to the **ANSYS Graphics** window, an upward arrow (↑) appears.
- (3) Confirming that the **Pick** and **Single** buttons are selected, move the upward arrow onto the right-end side of the strip area and click the left button of the mouse.
- (4) Click the **OK** button in the **Divide Multiple Lines ...** window to display the **Divide Multiple Lines with Options** window as shown in Figure 3.57.
- (5) Input 2 in [A] **NDIV** box and 0.5 in [B] **RATIO** box, and select **Be modified** in [C] **KEEP** box.
- (6) Click [D] **OK** button.

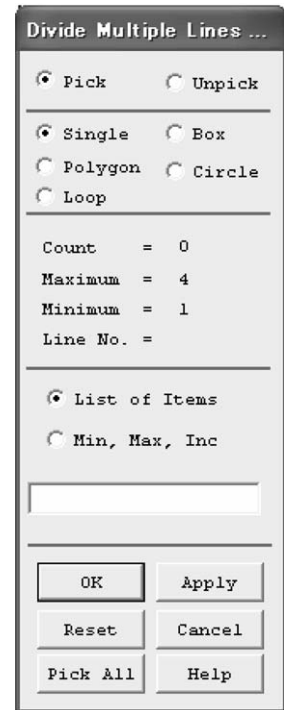


Figure 3.56 “Divide Multiple Lines ...” window.

[5] Meshing

COMMAND | ANSYS Main Menu → Preprocessor → Meshing → Mesh → Areas → Free

- (1) The **Mesh Areas** window opens.

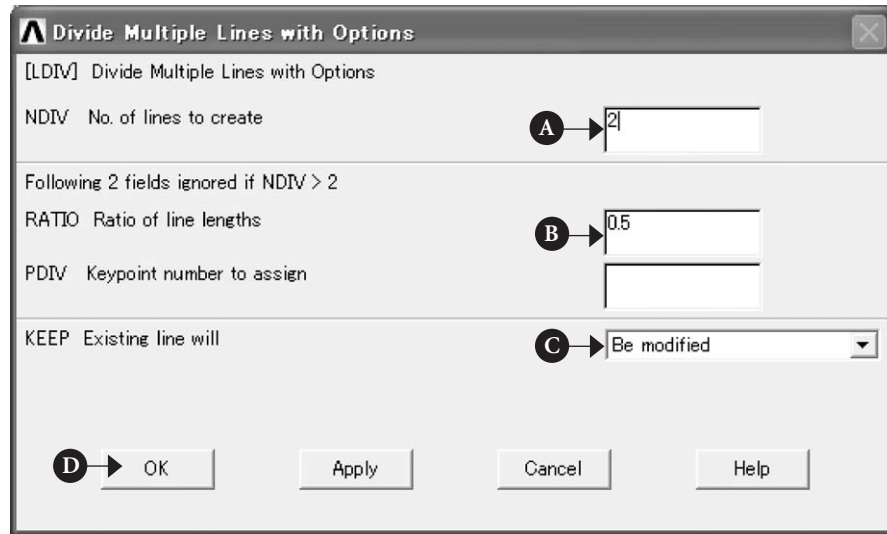


Figure 3.57 “Divide Multiple Lines with Options” window.

- (2) The upward arrow appears in the **ANSYS Graphics** window. Move this arrow to the elastic strip area and click this area.
- (3) The color of the area turns from light blue into pink. Click the **OK** button to see the area meshed by 8-node rectangular isoparametric finite elements.

3.2.3.4 INPUT OF BOUNDARY CONDITIONS

[1] *Imposing constraint conditions on the left end of the strip*

COMMAND | **ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Displacement → On Lines**

- (1) The **Apply U. ROT on Lines** window opens and the upward arrow appears when the mouse cursor is moved to the **ANSYS Graphics** window.
- (2) Confirming that the **Pick** and **Single** buttons are selected, move the upward arrow onto the left-end side of the strip area and click the left button of the mouse.
- (3) Click the **OK** button in the **Apply U. ROT on Lines** window to display another **Apply U. ROT on Lines** window.
- (4) Select **ALL DOF** in the **Lab2** box and click **OK** button in the **Apply U. ROT on Lines** window.

[2] *Imposing a triangular distribution of applied stress on the right end of the strip*

Distributed load or stress can be defined by pressure on lines and the triangular distribution of applied load can be defined as the composite of two linear distributions

of pressure which are symmetric to each other with respect to the center line of the strip area.

COMMAND | ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Pressure → On Lines

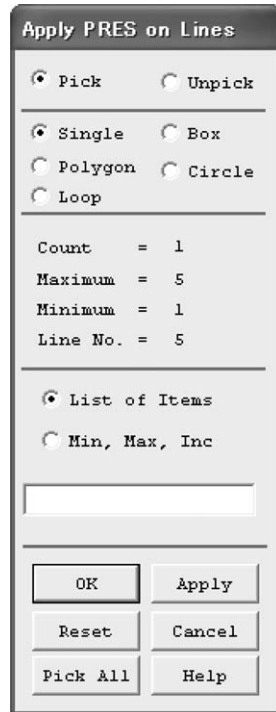


Figure 3.58 “Apply PRES on Lines” window for picking the lines to which pressure is applied.

- (1) The **Apply PRES on Lines** window opens (see Figure 3.58) and the upward arrow appears when the mouse cursor is moved to the **ANSYS Graphics** window.
- (2) Confirming that the **Pick** and **Single** buttons are selected, move the upward arrow onto the upper line of the right-end side of the strip area and click the left button of the mouse. Then, click the **OK** button. Remember that the right-end side of the strip area was divided into two lines in Procedure [4] in the preceding Section 3.2.3.3.
- (3) Another **Apply PRES on Lines** window opens (see Figure 3.59). Select **Constant value** in [A] [SFL] **Apply PRES on lines as a box** and input [B] **−10 (MPa)** in **VALUE Load PRES value** box and [C] **0 (MPa)** in **Value** box.
- (4) Click [D] **OK** button in the window to define a linear distribution of pressure on the upper line which is zero at the upper right corner and **−10 (MPa)** at the center of the right-end side of the strip area (see Figure 3.60).
- (5) For the lower line of the right-end side of the strip area, repeat the commands above and Procedures (2) through (4).
- (6) Select **Constant value** in [A] [SFL] **Apply PRES on lines as a box** and input [B] **0 (MPa)** in **VALUE Load PRES value** box and [C] **−10 (MPa)** in **Value** box as shown in Figure 3.61. Note that the values to input in the lower two boxes in the **Apply PRES on Lines** window is interchanged, since the distributed pressure on the lower line of the right-end side of the strip area is symmetric to that on the upper line with respect to the center line of the strip area.
- (7) Click [D] **OK** button in the window shown in Figure 3.60 to define a linear distribution of pressure on the lower line which is **−10 MPa** at the center and zero at the lower right corner of the right-end side of the strip area as shown in Figure 3.62.

3.2.3.5 SOLUTION PROCEDURES

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

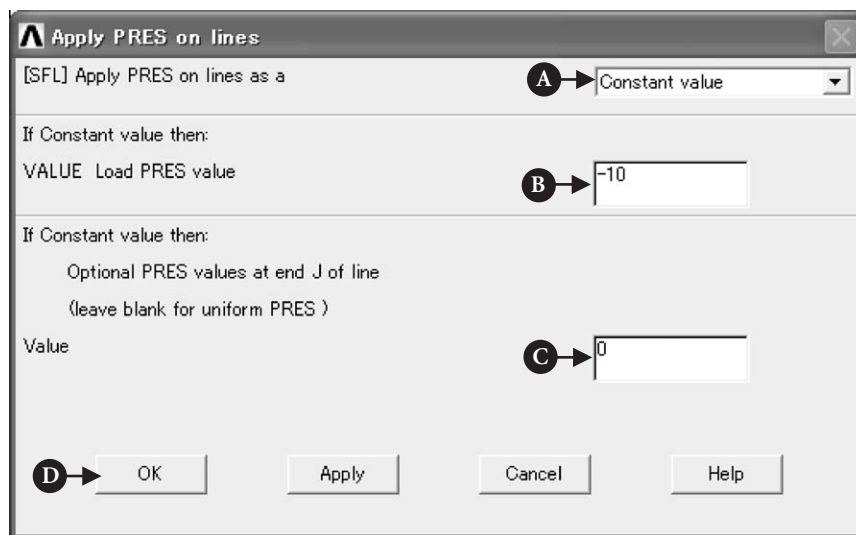


Figure 3.59 “Apply PRES on Lines” window for applying linearly distributed pressure to the upper half of the right end of the elastic strip.



Figure 3.60 Linearly distributed negative pressure applied to the upper half of the right-end side of the elastic strip.

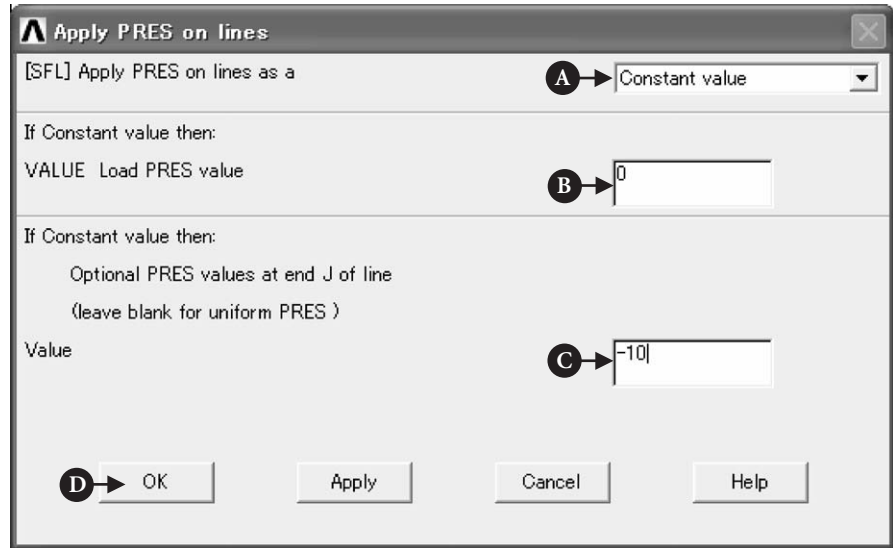


Figure 3.61 “Apply PRES on Lines” window for applying linearly distributed pressure to the lower half of the right end of the elastic strip.

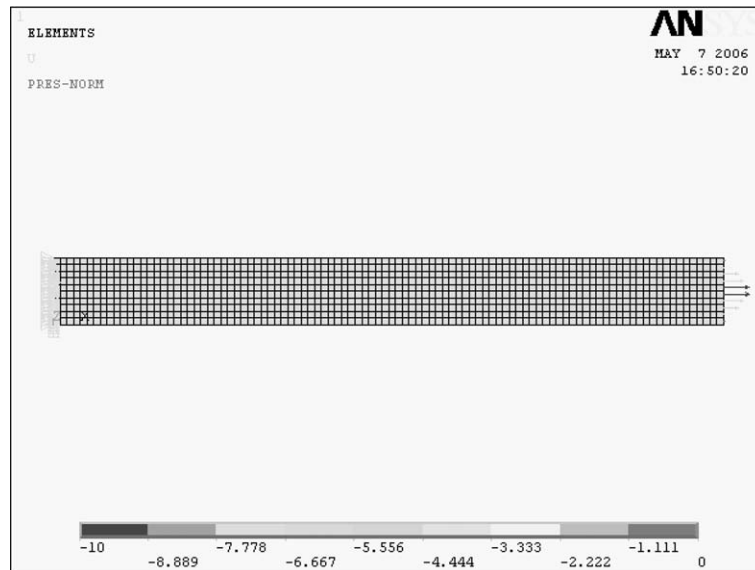


Figure 3.62 Triangular distribution of pressure applied to the right end of the elastic strip.

- (1) The **Solve Current Load Step** and **/STATUS Command** windows appear.
- (2) Click the **OK** button in the **Solve Current Load Step** window to begin the solution of the current load step.

- (3) Select the **File** button in **/STATUS Command** window to open the submenu and select the **Close** button to close the **/STATUS Command** window.
- (4) When solution is completed, the **Note** window appears. Click the **Close** button to close the “**Note**” window.

3.2.3.6 CONTOUR PLOT OF STRESS

COMMAND | ANSYS Main Menu → General Postproc → Plot Results → Contour Plot → Nodal Solution

- (1) The **Contour Nodal Solution Data** window opens.
- (2) Select **Stress** and **X-Component of stress**.
- (3) Click the **OK** button to display the contour of the x -component of stress in the elastic strip in the **ANSYS Graphics** window as shown in Figure 3.63.

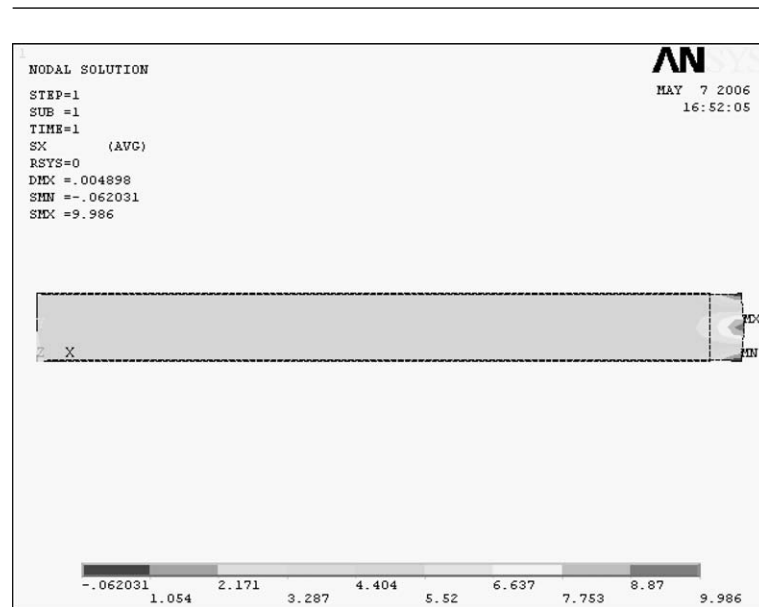


Figure 3.63 Contour of the x -component of stress in the elastic strip showing uniform stress distribution at one width or larger distance from the right end of the elastic strip to which triangular distribution of pressure is applied.

3.2.4 DISCUSSION

Figure 3.64 shows the variations of the longitudinal stress distribution in the cross section with the x -position of the elastic strip. At the right end of the strip, or at

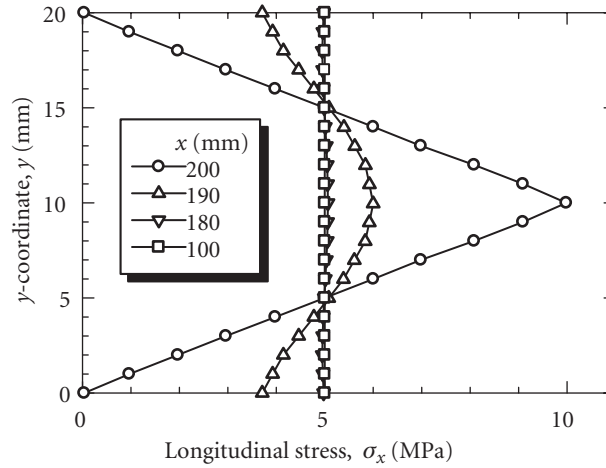


Figure 3.64 Variations of the longitudinal stress distribution in the cross section with the x -position of the elastic strip.

$x = 200$ mm, the distribution of the applied longitudinal stress takes the triangular shape which is zero at the upper and lower corners and 10 MPa at the center of the strip. The longitudinal stress distribution varies as the distance of the cross section from the right end of the strip increases, and the distribution becomes almost uniform at $x = 180$ mm, i.e., at one width distance from the end of the stress application. The total amount of stress in any cross section is the same, i.e., 1 kN in the strip and, stress is uniformly distributed and the magnitude of stress becomes 5 MPa at any cross section at one width or larger distance from the end of the stress application.

The above result is known as the principle of St. Venant and is very useful in practice, or in the design of structural components. Namely, even if the stress distribution is very complicated at the loading points due to the complicated shape of load transfer equipment, one can assume a uniform stress distribution in the main parts of structural components or machine elements at some distance from the load transfer equipment.

3.3 STRESS CONCENTRATION DUE TO ELLIPTIC HOLES

3.3.1 EXAMPLE PROBLEM

An elastic plate with an elliptic hole in its center is subjected to uniform longitudinal tensile stress σ_0 at one end and clamped at the other end in Figure 3.65. Perform the FEM stress analysis of the 2-D elastic plate and calculate the maximum longitudinal stress σ_{\max} in the plate to obtain the stress concentration factor $\alpha = \sigma_{\max}/\sigma_0$. Observe the variation of the longitudinal stress distribution in the ligament between the foot of the hole and the edge of the plate.

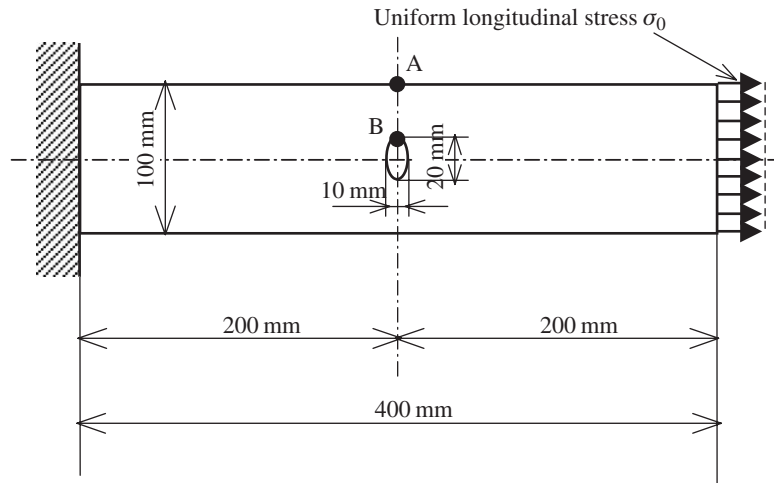


Figure 3.65 A 2-D elastic plate with an elliptic hole in its center subjected to a uniform longitudinal stress at one end and clamped at the other end.

3.3.2 PROBLEM DESCRIPTION

Plate geometry: $l = 400$ mm, height $h = 100$ mm, thickness $b = 10$ mm.

Material: mild steel having Young's modulus $E = 210$ GPa and Poisson's ratio $\nu = 0.3$.

Elliptic hole: An elliptic hole has a minor radius of 5 mm in the longitudinal direction and a major radius of 10 mm in the transversal direction.

Boundary conditions: The elastic plate is subjected to a uniform tensile stress of $\sigma_0 = 10$ Mpa in the longitudinal direction at the right end and clamped to a rigid wall at the left end.

3.3.3 ANALYTICAL PROCEDURES

3.3.3.1 CREATION OF AN ANALYTICAL MODEL

Let us use a quarter model of the elastic plate with an elliptic hole as illustrated later in Figure 3.70, since the plate is symmetric about the horizontal and vertical centerlines. The quarter model can be created by a slender rectangular area from which an elliptic area is subtracted by using the Boolean operation described in Section A3.1.

First, create the rectangular area by the following operation:

COMMAND | ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Rectangle → By 2 Corners

- (1) Input two 0's into the **WP X** and **WP Y** boxes in the **Rectangle by 2 Corners** window to determine the lower left corner point of the elastic plate on the Cartesian coordinates of the working plane.

- (2) Input **200** and **50** (mm) into the **Width** and **Height** boxes, respectively, to determine the shape of the quarter elastic plate model.
- (3) Click the **OK** button to create the quarter elastic plate on the **ANSYS Graphics** window.

Then, create a circular area having a diameter of 10 mm and then reduce its diameter in the longitudinal direction to a half of the original value to get the elliptic area. The following commands create a circular area by designating the coordinates (UX, UY) of the center and the radius of the circular area:

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Circle → Solid Circle**

- (1) The **Solid Circular Area** window opens as shown in Figure 3.66.
- (2) Input two **0**'s into [A] **WP X** and [B] **WP Y** boxes to determine the center position of the circular area.
- (3) Input [C] **10** (mm) in **Radius** box to determine the radius of the circular area.
- (4) Click [D] **OK** button to create the circular area superimposed on the rectangular area in the **ANSYS Graphics** window as shown in Figure 3.67.

In order to reduce the diameter of the circular area in the longitudinal direction to a half of the original value, use the following **Scale → Areas** operation:

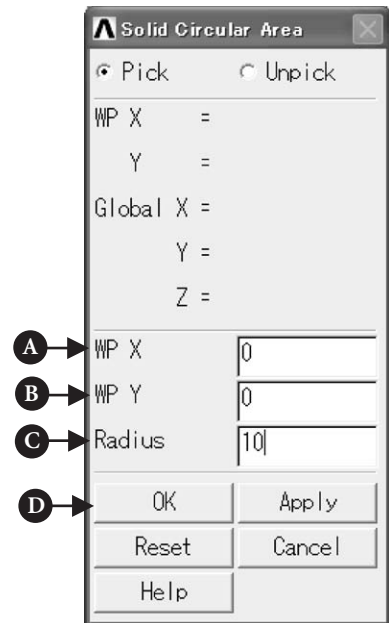


Figure 3.66 “Solid Circular Area” window.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Operate → Scale → Areas**

- (1) The **Scale Areas** window opens as shown in Figure 3.68.
- (2) The upward arrow appears in the **ANSYS Graphics** window. Move the arrow to the circular area and pick it by clicking the left button of the mouse. The color of the circular area turns from light blue into pink and click [A] **OK** button.
- (3) The color of the circular area turns into light blue and another **Scale Areas** window opens as shown in Figure 3.69.
- (4) Input [A] **0.5** in **RX** box, select [B] **Areas only** in **NOELEM** box and [C] **Moved** in **IMOVE** box.
- (5) Click [D] **OK** button. An elliptic area appears and the circular area still remains. The circular area is an afterimage and does not exist in reality. To erase this

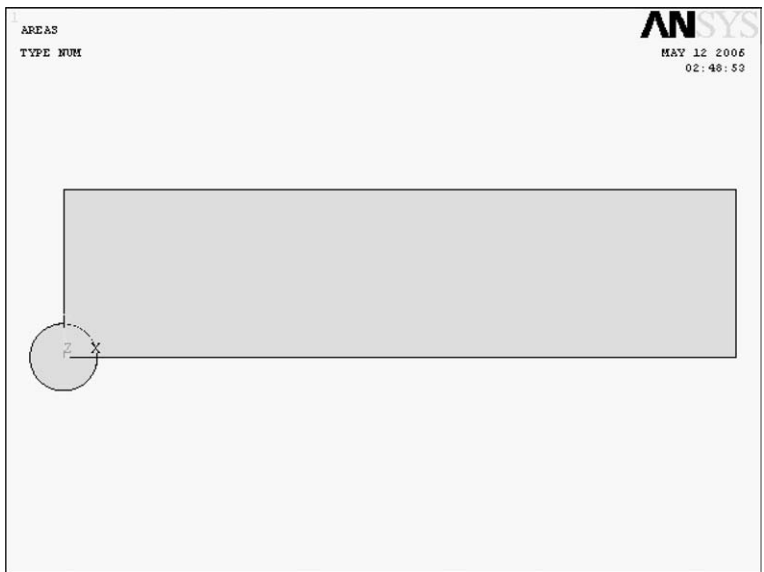


Figure 3.67 Circular area superimposed on the rectangular area.

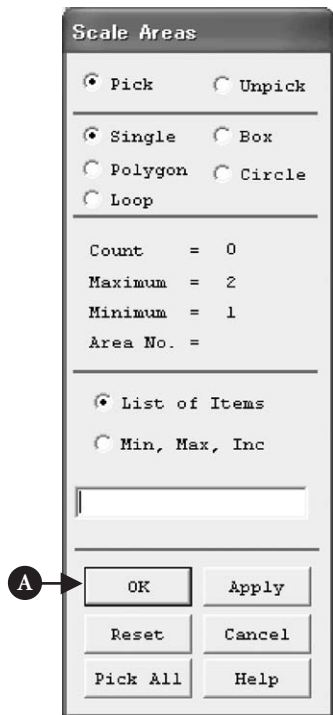


Figure 3.68 "Scale Areas" window.

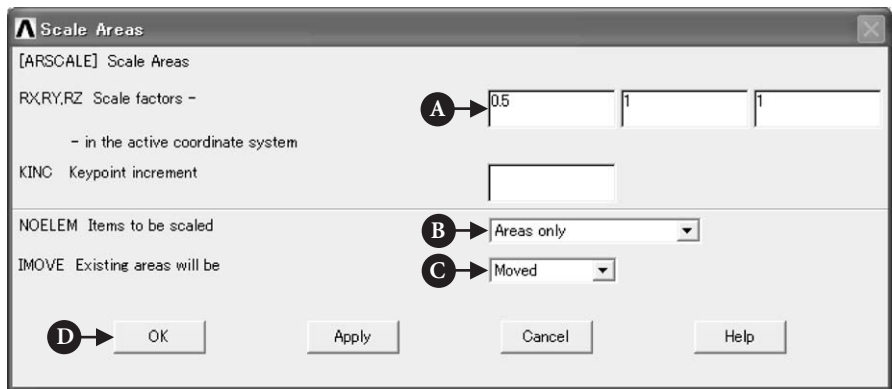


Figure 3.69 "Scale Areas".

after-image, perform the following commands:

COMMAND | **ANSYS Utility Menu → Plot → Replot**

The circular area vanishes. Subtract the elliptic area from the rectangular area in a similar manner as described in Section A3.2, i.e.,

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

- (1) Pick the rectangular area by the upward arrow and confirm that the color of the area picked turns from light blue into pink. Click the **OK** button.
- (2) Pick the elliptic area by the upward arrow and confirm that the color of the elliptic area picked turns from light blue into pink. Click the **OK** button to subtract the elliptic area from the rectangular area to get a quarter model of a plate with an elliptic hole in its center as shown in Figure 3.70.

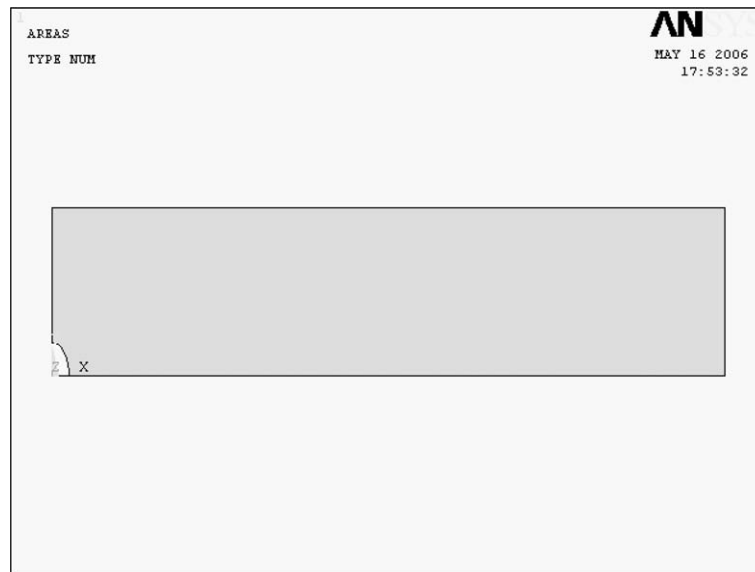


Figure 3.70 Quarter model of a plate with an elliptic hole in its center created by subtracting an elliptic area from a rectangular area.

3.3.3.2 INPUT OF THE ELASTIC PROPERTIES OF THE PLATE MATERIAL

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

- (1) The **Define Material Model Behavior** window opens.
- (2) Double-click **Structural**, **Linear**, **Elastic**, and **Isotropic** buttons one after another.

- (3) Input the value of Young's modulus, **2.1e5** (MPa), and that of Poisson's ratio, **0.3**, into **EX** and **PRXY** boxes, and click the **OK** button of the **Linear Isotropic Properties for Materials Number 1** window.
- (4) Exit from the **Define Material Model Behavior** window by selecting **Exit** in the **Material** menu of the window.

3.3.3.3 FINITE-ELEMENT DISCRETIZATION OF THE QUARTER PLATE AREA

[1] Selection of the element type

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

- (1) The **Element Types** window opens.
- (2) Click the **Add . . .** button in the **Element Types** window to open the **Library of Element Types** window and select the element type to use.
- (3) Select **Structural Mass – Solid** and **Quad 8node 82**.
- (4) Click the **OK** button in the **Library of Element Types** window to use the 8-node isoparametric element.
- (5) Click the **Options . . .** button in the **Element Types** window to open the **PLANE82 element type options** window. Select the **Plane strs w/thk** item in the **Element behavior** box and click the **OK** button to return to the **Element Types** window. Click the **Close** button in the **Element Types** window to close the window.

[2] Input of the element thickness

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

- (1) The **Real Constants** window opens.
- (2) Click **[A] Add/Edit/Delete** button to open the **Real Constants** window and click the **Add . . .** button.
- (3) The **Element Type for Real Constants** window opens. Click the **OK** button.
- (4) The **Element Type for Real Constants** window vanishes and the **Real Constants Set Number 1. for PLANE82** window appears instead. Input a strip thickness of **10** (mm) in the **Thickness** box and click the **OK** button.
- (5) The **Real Constants** window returns with the display of the **Defined Real Constants Sets** box changed to **Set 1**. Click the **Close** button.

[3] Sizing of the elements

COMMAND | ANSYS Main Menu → Preprocessor → Meshing → Size Cntrls → Manual Size → Global → Size

- (1) The **Global Element Sizes** window opens.
- (2) Input **1.5** in the **SIZE** box and click the **OK** button.

[4] Meshing

COMMAND | ANSYS Main Menu → Preprocessor → Meshing → Mesh → Areas → Free

- (1) The **Mesh Areas** window opens.
- (2) The upward arrow appears in the **ANSYS Graphics** window. Move this arrow to the quarter plate area and click this area.
- (3) The color of the area turns from light blue into pink. Click the **OK** button to see the area meshed by 8-node isoparametric finite elements as shown in Figure 3.71.

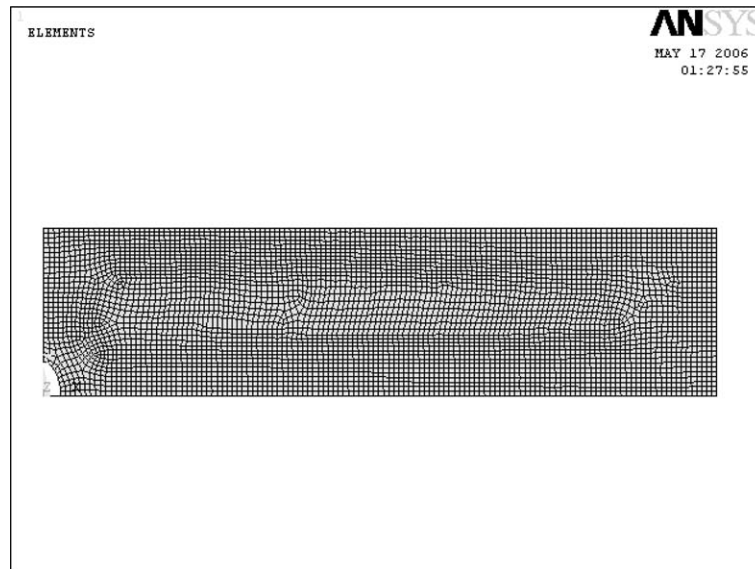


Figure 3.71 Quarter model of a plate with an elliptic hole meshed by 8-node isoparametric finite elements.

3.3.3.4 INPUT OF BOUNDARY CONDITIONS

[1] Imposing constraint conditions on the left end and the bottom side of the quarter plate model

Due to the symmetry, the constraint conditions of the quarter plate model are **UX**-fixed condition on the left end and **UY**-fixed condition on the bottom side of the quarter plate model. Apply the constraint conditions onto the corresponding lines by the following commands:

COMMAND | ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Displacement → On Lines

- (1) The **Apply U, ROT on Lines** window opens and the upward arrow appears when the mouse cursor is moved to the **ANSYS Graphics** window.

- (2) Confirming that the **Pick** and **Single** buttons are selected, move the upward arrow onto the left-end side of the quarter plate area and click the left button of the mouse.
- (3) Click the **OK** button in the **Apply U. ROT on Lines** window to display another **Apply U. ROT on Lines** window.
- (4) Select **UX** in the **Lab2** box and click the **OK** button in the **Apply U. ROT on Lines** window.

Repeat the commands and operations (1) through (3) above for the bottom side of the model. Then, select **UY** in the **Lab2** box and click the **OK** button in the **Apply U. ROT on Lines** window.

[2] Imposing a uniform longitudinal stress on the right end of the quarter plate model

A uniform longitudinal stress can be defined by pressure on the right-end side of the plate model as described below:

COMMAND | **ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Pressure → On Lines**

- (1) The **Apply PRES on Lines** window opens and the upward arrow appears when the mouse cursor is moved to the **ANSYS Graphics** window.
- (2) Confirming that the **Pick** and **Single** buttons are selected, move the upward arrow onto the right-end side of the quarter plate area and click the left button of the mouse.
- (3) Another **Apply PRES on Lines** window opens. Select **Constant value** in the **[SFL] Apply PRES on lines as a** box and input **−10** (MPa) in the **VALUE Load PRES value** box and leave a blank in the **Value** box.
- (4) Click the **OK** button in the window to define a uniform tensile stress of 10 MPa applied to the right end of the quarter plate model.

Figure 3.72 illustrates the boundary conditions applied to the center notched plate model by the above operations.

3.3.3.5 SOLUTION PROCEDURES

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

- (1) The **Solve Current Load Step** and **/STATUS Command** windows appear.
- (2) Click the **OK** button in the **Solve Current Load Step** window to begin the solution of the current load step.
- (3) Select the **File** button in **/STATUS Command** window to open the submenu and select the **Close** button to close the **/STATUS Command** window.
- (4) When the solution is completed, the **Note** window appears. Click the **Close** button to close the **Note** window.

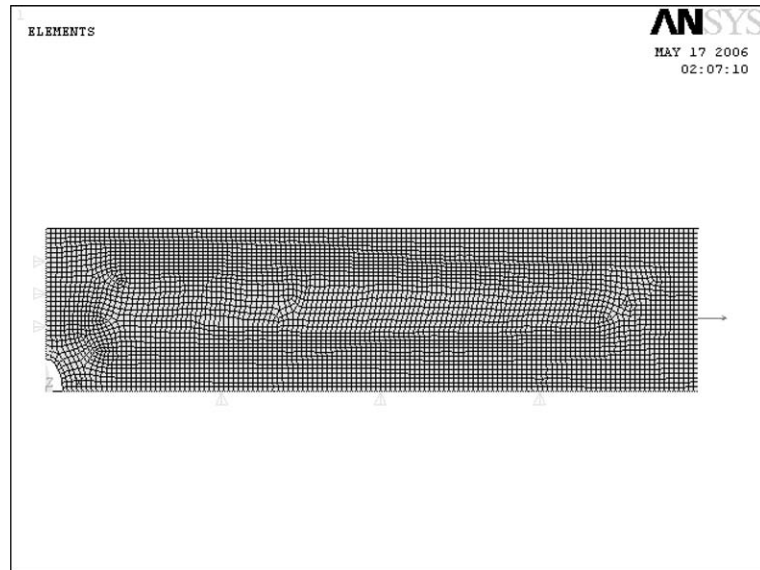


Figure 3.72 Boundary conditions applied to the quarter model of the center notched plate.

3.3.3.6 CONTOUR PLOT OF STRESS

COMMAND | ANSYS Main Menu → General Postproc → Plot Results → Contour Plot → Nodal Solution

- (1) The **Contour Nodal Solution Data** window opens.
- (2) Select **Stress** and **X-Component of stress**.
- (3) Select **Deformed shape with undeformed edge** in the **Undisplaced shape key** box to compare the shapes of the notched plate before and after deformation.
- (4) Click the **OK** button to display the contour of the x -component of stress in the quarter model of the center notched plate in the **ANSYS Graphics** window as shown in Figure 3.73.

3.3.3.7 OBSERVATION OF THE VARIATION OF THE LONGITUDINAL STRESS DISTRIBUTION IN THE LIGAMENT REGION

In order to observe the variation of the longitudinal stress distribution in the ligament between the foot of the hole and the edge of the plate, carry out the following commands:

COMMAND | ANSYS Utility Menu → PlotCtrls → Symbols ...

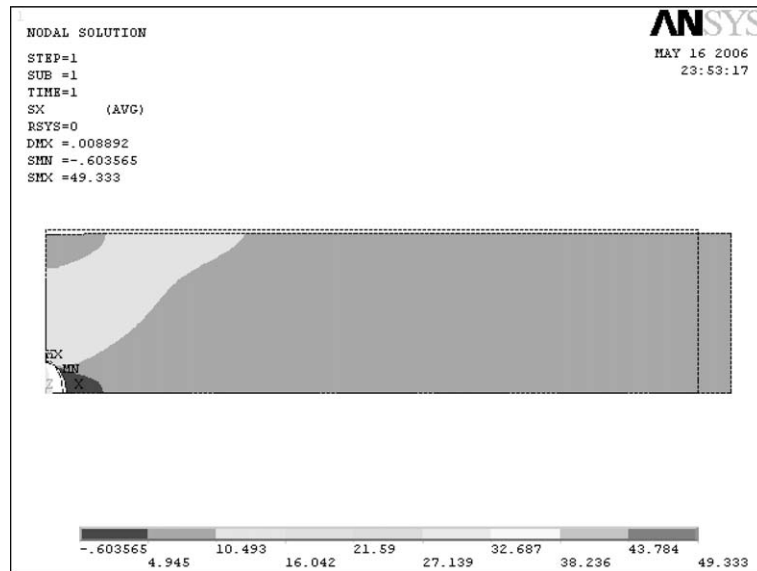


Figure 3.73 Contour of the x -component of stress in the quarter model of the center notched plate.

The **Symbols** window opens as shown in Figure 3.74:

- (1) Select [A] **All Reactions** in the [/PBC] **Boundary condition symbol** buttons, [B] **Pressures** in the [/PSF] **Surface Load Symbols** box and [C] **Arrows** in the [/PSF] **Show pres and convect as** box.
- (2) The distributions of the reaction force and the longitudinal stress in the ligament region (see [A] in Figure 3.75) as well as that of the applied stress on the right end of the plate area are superimposed on the contour x -component of stress in the plate in the **ANSYS Graphics** window. The reaction force is indicated by the leftward red arrows, whereas the longitudinal stress by the rightward red arrows in the ligament region.

The longitudinal stress reaches its maximum value at the foot of the hole and is decreased approaching to a constant value almost equal to the applied stress σ_0 at some distance, say, about one major diameter distance, from the foot of the hole. This tendency can be explained by the principle of St. Venant as discussed in the previous section.

3.3.4 DISCUSSION

If an elliptical hole is placed in an infinite body and subjected to a remote uniform stress of σ_0 , the maximum stress σ_{\max} occurs at the foot of the hole, i.e., Point B in

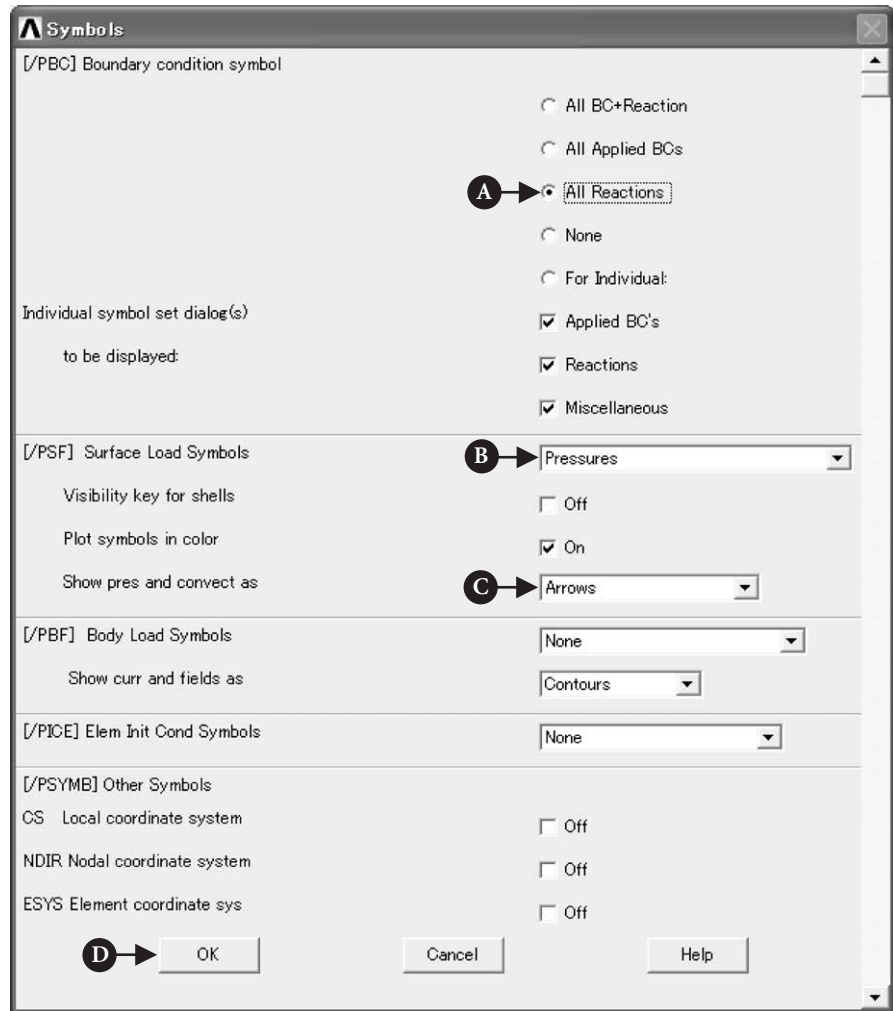


Figure 3.74 “Symbols” window.

Figure 3.65, and is expressed by the following formula:

$$\sigma_{\max} = \left(1 + 2\frac{a}{b}\right) \sigma_0 = \left(1 + \sqrt{\frac{a}{\rho}}\right) \sigma_0 = \alpha_0 \sigma_0 \quad (3.4)$$

where a is the major radius, b the minor radius, ρ the local radius of curvature at the foot of the elliptic hole and α_0 the stress concentration factor which is

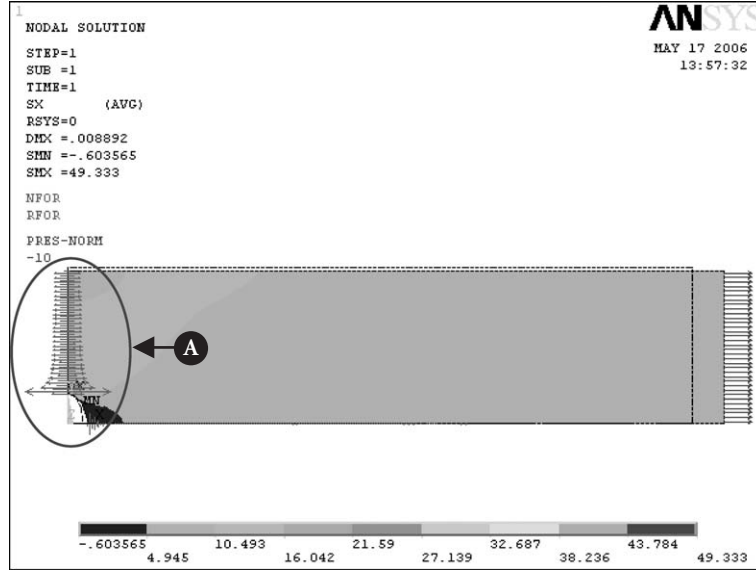


Figure 3.75 Applied stress on the right end side of the plate and resultant reaction force and longitudinal stress in its ligament region.

defined as,

$$\alpha_0 \equiv \frac{\sigma_{\max}}{\sigma_0} = \left(1 + 2\frac{a}{b}\right) = \left(1 + \sqrt{\frac{a}{\rho}}\right) \quad (3.5)$$

The stress concentration factor α_0 varies inversely proportional to the aspect ratio of elliptic hole b/a , namely the smaller the value of the aspect ratio b/a or the radius of curvature ρ becomes, the larger the value of the stress concentration factor α_0 becomes.

In a finite plate, the maximum stress at the foot of the hole is increased due to the finite boundary of the plate. Figure 3.76 shows the variation of the stress concentration factor α for elliptic holes having different aspect ratios with normalized major radius $2a/h$, indicating that the stress concentration factor in a plate with finite width h is increased dramatically as the ligament between the foot of the hole and the plate edge becomes smaller.

From Figure 3.76, the value of the stress intensity factor for the present elliptic hole is approximately 5.16, whereas Figure 3.75 shows that the maximum value of the longitudinal stress obtained by the present FEM calculation is approximately 49.3 MPa, i.e., the value of the stress concentration factor is approximately $49.3/10 = 4.93$. Hence, the relative error of the present calculation is approximately, $(4.93 - 5.16)/5.16 \approx -0.0446 = 4.46\%$ which may be reasonably small and so be acceptable.

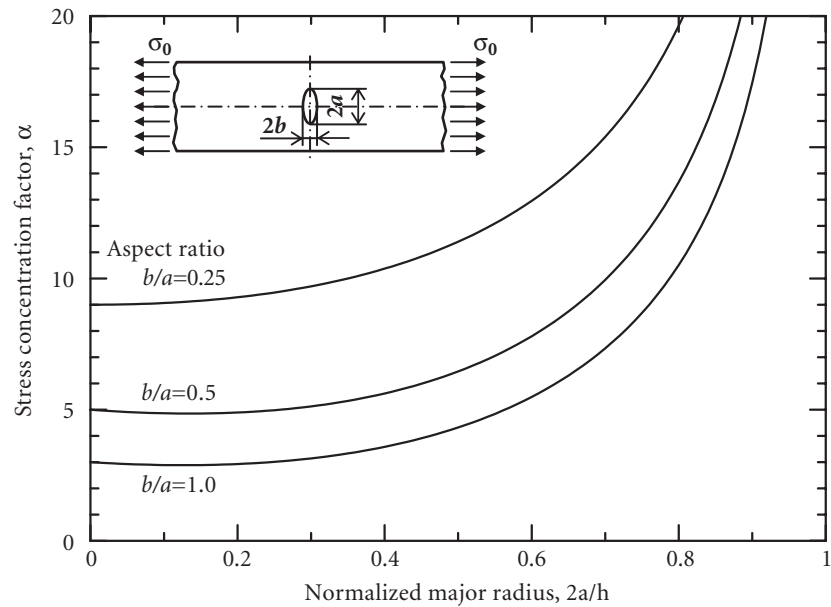


Figure 3.76 Variation of the stress concentration factor α for elliptic holes having different aspect ratios with normalized major radius $2a/h$.

3.3.5 PROBLEMS TO SOLVE

PROBLEM 3.8

Calculate the value of stress concentration factor for the elliptic hole shown in Figure 3.65 by using the whole model of the plate and compare the result with that obtained and discussed in 3.3.4.

PROBLEM 3.9

Calculate the values of stress concentration factor α for circular holes for different values of the normalized major radius $2a/h$ and plot the results as the α versus $2a/h$ diagram as shown in Figure 3.76.

PROBLEM 3.10

Calculate the values of stress concentration factor α for elliptic holes having different aspect ratios b/a for different values of the normalized major radius $2a/h$ and plot the results as the α versus $2a/h$ diagram.

PROBLEM 3.11

For smaller values of $2a/h$, the disturbance of stress in the ligament between the foot of a hole and the plate edge due to the existence of the hole is decreased and stress in the ligament approaches to a constant value equal to the remote stress σ_0 at some distance from the foot of the hole (remember the principle of St. Venant in the previous section). Find how much distance from the foot of the hole the stress in the ligament region can be considered to be almost equal to the value of the remote stress σ_0 .

3.4 STRESS SINGULARITY PROBLEM

3.4.1 EXAMPLE PROBLEM

An elastic plate with a crack of length $2a$ in its center subjected to uniform longitudinal tensile stress σ_0 at one end and clamped at the other end as illustrated in Figure 3.77. Perform an FEM analysis of the 2-D elastic center-cracked tension plate illustrated in Figure 3.77 and calculate the value of the mode I (crack-opening mode) stress intensity factor for the center-cracked plate.

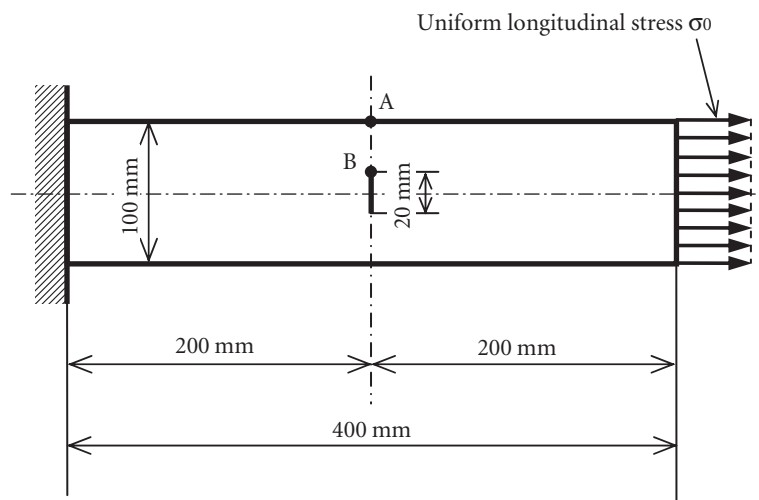


Figure 3.77 Two dimensional elastic plate with a crack of length $2a$ in its center subjected to a uniform tensile stress σ_0 in the longitudinal direction at one end and clamped at the other end.

3.4.2 PROBLEM DESCRIPTION

Specimen geometry: length $l = 400$ mm, height $h = 100$ mm.

Material: mild steel having Young's modulus $E = 210$ GPa and Poisson's ratio $\nu = 0.3$.

Crack: A crack is placed perpendicular to the loading direction in the center of the plate and has a length of 20 mm. The center-cracked tension plate is assumed to be in the plane strain condition in the present analysis.

Boundary conditions: The elastic plate is subjected to a uniform tensile stress in the longitudinal direction at the right end and clamped to a rigid wall at the left end.

3.4.3 ANALYTICAL PROCEDURES

3.4.3.1 CREATION OF AN ANALYTICAL MODEL

Let us use a quarter model of the center-cracked tension plate as illustrated in Figure 3.77, since the plate is symmetric about the horizontal and vertical center lines.

Here we use the singular element or the quarter point element which can interpolate the stress distribution in the vicinity of the crack tip at which stress has the $1/\sqrt{r}$ singularity where r is the distance from the crack tip ($r/a \ll 1$). An ordinary isoparametric element which is familiar to you as “**Quad 8node 82**” has nodes at the corners and also at the midpoint on each side of the element. A singular element, however, has the midpoint moved one-quarter side distance from the original midpoint position to the node which is placed at the crack tip position. This is the reason why the singular element is often called the quarter point element instead. ANSYS software is equipped only with a 2-D triangular singular element, but neither with 2-D rectangular nor with 3-D singular elements. Around the node at the crack tip, a circular area is created and is divided into a designated number of triangular singular elements. Each triangular singular element has its vertex placed at the crack tip position and has the quarter points on the two sides joining the vertex and the other two nodes.

In order to create the singular elements, the plate area must be created via keypoints set at the four corner points and at the crack tip position on the left-end side of the quarter plate area.

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

- (1) The **Create Keypoints in Active Coordinate System** window opens as shown in Figure 3.78.
- (2) Input [A] each **keypoint number** in the **NPT** box and [B] **x**-, **y**-, and **z**-**coordinates** of each keypoint in the three **X**, **Y**, **Z** boxes, respectively. Figure 3.78 shows the case of Key point #5, which is placed at the crack tip having the coordinates (0, 10, 0). In the present model, let us create Key points #1 to #5 at the coordinates (0, 0, 0), (200, 0, 0), (200, 50, 0), (0, 50, 0), and (0, 10, 0), respectively. Note that the z-coordinate is always 0 in 2-D elasticity problems.
- (3) Click [C] **Apply** button four times and create Key points #1 to #4 without exiting from the window and finally click [D] **OK** button to create key point #5 at the crack tip position and exit from the window (see Figure 3.79).

Then create the plate area via the five key points created in the procedures above by the following commands:

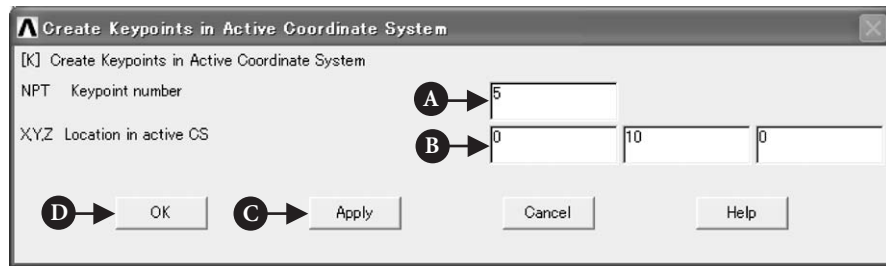


Figure 3.78 “Create Keypoints in Active Coordinate System” window.

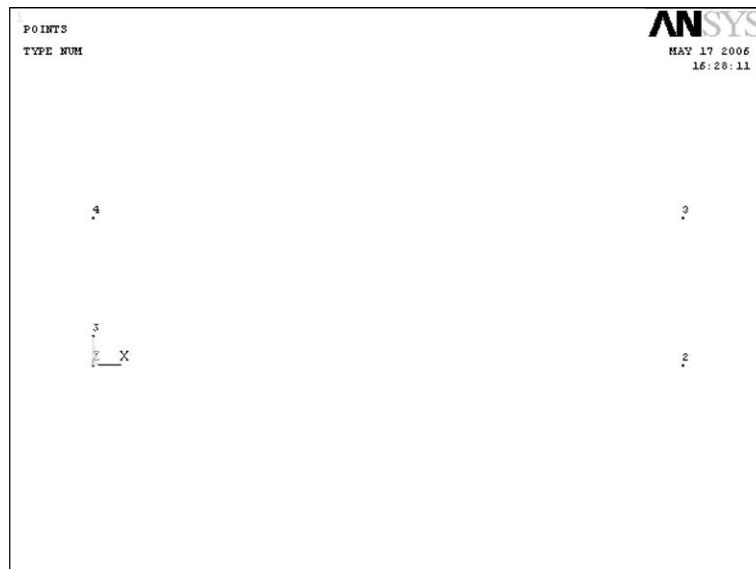


Figure 3.79 Five key points created in the “ANSYS Graphics” window.

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

- (1) The **Create Area thru KPs** window opens.
- (2) The upward arrow appears in the **ANSYS Graphics** window. Move this arrow to Key point #1 and click this point. Click Key points #1 through #5 one by one counterclockwise (see Figure 3.80).
- (3) Click the **OK** button to create the plate area as shown in Figure 3.81.

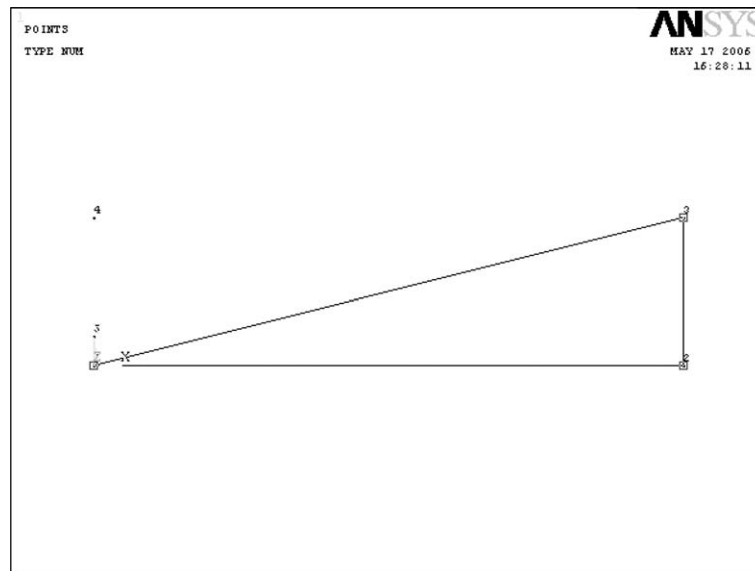


Figure 3.80 Clicking Key points #1 through #5 one by one counterclockwise to create the plate area.

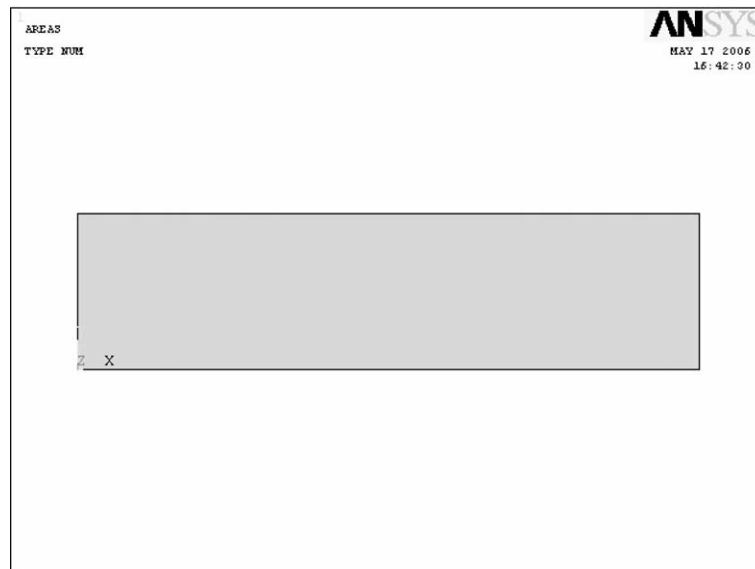


Figure 3.81 Quarter model of the center cracked tension plate.

3.4.3.2 INPUT OF THE ELASTIC PROPERTIES OF THE PLATE MATERIAL

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

- (1) The **Define Material Model Behavior** window opens.
- (2) Double-click **Structural**, **Linear**, **Elastic**, and **Isotropic** buttons one after another.
- (3) Input the value of Young's modulus, **2.1e5** (MPa), and that of Poisson's ratio, **0.3**, into **EX** and **PRXY** boxes, and click the **OK** button of the **Linear Isotropic Properties for Materials Number 1** window.
- (4) Exit from the **Define Material Model Behavior** window by selecting **Exit** in the **Material** menu of the window.

3.4.3.3 FINITE-ELEMENT DISCRETIZATION OF THE CENTER-CRACKED TENSION PLATE AREA

[1] Selection of the element type

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

- (1) The **Element Types** window opens.
- (2) Click the **Add ...** button in the **Element Types** window to open the **Library of Element Types** window and select the element type to use.
- (3) Select **Structural Mass – Solid** and **Quad 8 node 82**.
- (4) Click the **OK** button in the **Library of Element Types** window to use the 8-node isoparametric element.
- (5) Click the **Options ...** button in the **Element Types** window to open the **PLANE82 element type options** window. Select the **Plane strain** item in the **Element behavior** box and click the **OK** button to return to the **Element Types** window. Click the **Close** button in the **Element Types** window to close the window.

[2] Sizing of the elements

Before meshing, the crack tip point around which the triangular singular elements will be created must be specified by the following commands:

COMMAND | ANSYS Main Menu → Preprocessor → Meshing → Size Cntrls → Concentrate KPs → Create

- (1) The **Concentration Keypoint** window opens.
- (2) Display the key points in the **ANSYS Graphics** window by

COMMAND | ANSYS Utility Menu → Plot → Keypoints → Keypoints.

- (3) Pick Key point #5 by placing the upward arrow onto Key point #5 and by clicking the left button of the mouse. Then click the **OK** button in the **Concentration Keypoint** window.
- (4) Another **Concentration Keypoint** window opens as shown in Figure 3.82.

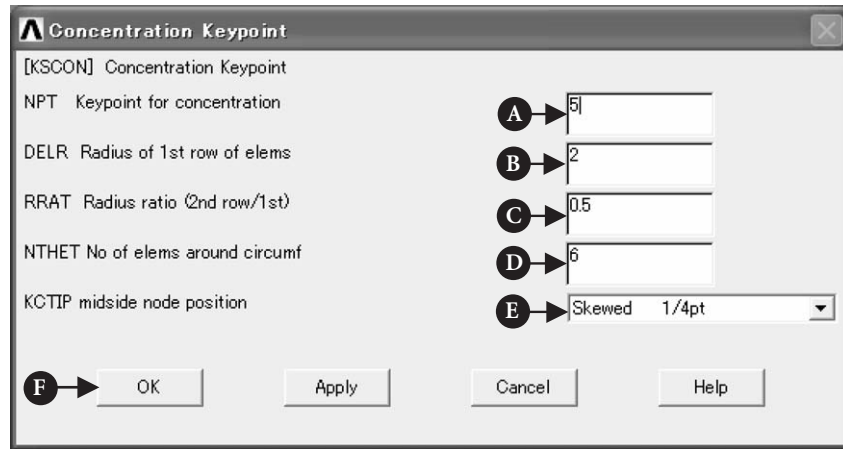


Figure 3.82 “Concentration Keypoint” window.

- (5) Confirming that [A] **5**, i.e., the key point number of the crack tip position is input in the **NPT** box, input [B] **2** in the **DELR** box, [C] **0.5** in the **RRAT** box and [D] **6** in the **NTHET** box and select [E] **Skewed 1/4pt** in the **KCTIP** box in the window. Refer to the explanations of the numerical data described after the names of the respective boxes on the window. **Skewed 1/4pt** in the last box means that the mid nodes of the sides of the elements which contain Key point #5 are the quarter points of the elements.
- (6) Click [F] **OK** button in the **Concentration Keypoint** window.

The size of the meshes other than the singular elements and the elements adjacent to them can be controlled by the same procedures as have been described in the previous sections of the present Chapter 3.

COMMAND | ANSYS Main Menu → Preprocessor → Meshing → Size Cntrls → Manual Size → Global → Size

- (1) The **Global Element Sizes** window opens.
- (2) Input **1.5** in the **SIZE** box and click the **OK** button.

[3] Meshing

The meshing procedures are also the same as before.

COMMAND | ANSYS Main Menu → Preprocessor → Meshing → Mesh → Areas → Free

- (1) The **Mesh Areas** window opens.
- (2) The upward arrow appears in the **ANSYS Graphics** window. Move this arrow to the quarter plate area and click this area.
- (3) The color of the area turns from light blue into pink. Click the **OK** button.
- (4) The **Warning** window appears as shown in Figure 3.83 due to the existence of six singular elements. Click [A] **Close** button and proceed to the next operation below.

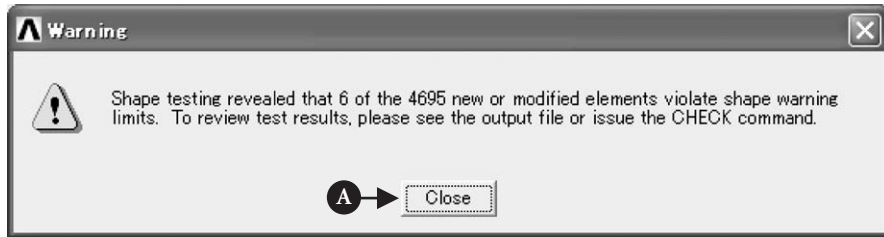


Figure 3.83 “Warning” window.

- (5) Figure 3.84 shows the plate area meshed by ordinary 8-node isoparametric finite elements except for the vicinity of the crack tip where we have six singular elements.

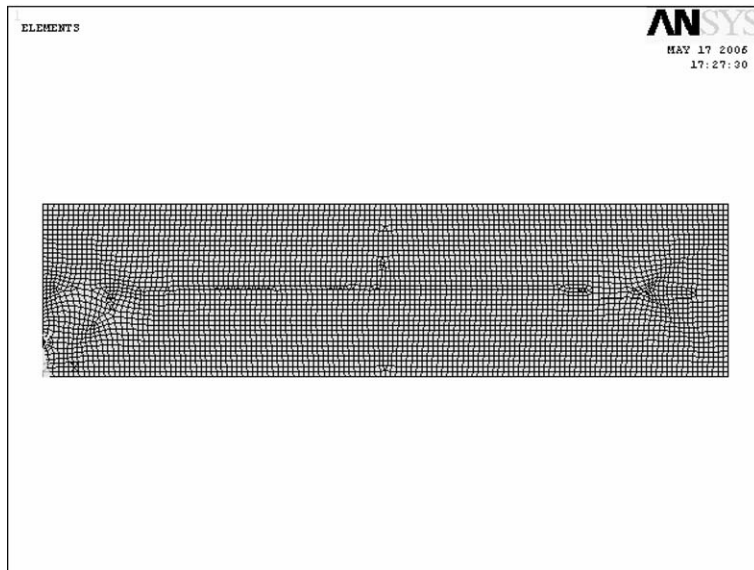


Figure 3.84 Plate area meshed by ordinary 8-node isoparametric finite elements and by singular elements.

Figure 3.85 is an enlarged view of the singular elements around [A] the crack tip showing that six triangular elements are placed in a radial manner and that the size of the second row of elements is half the radius of the first row of elements, i.e., triangular singular elements.

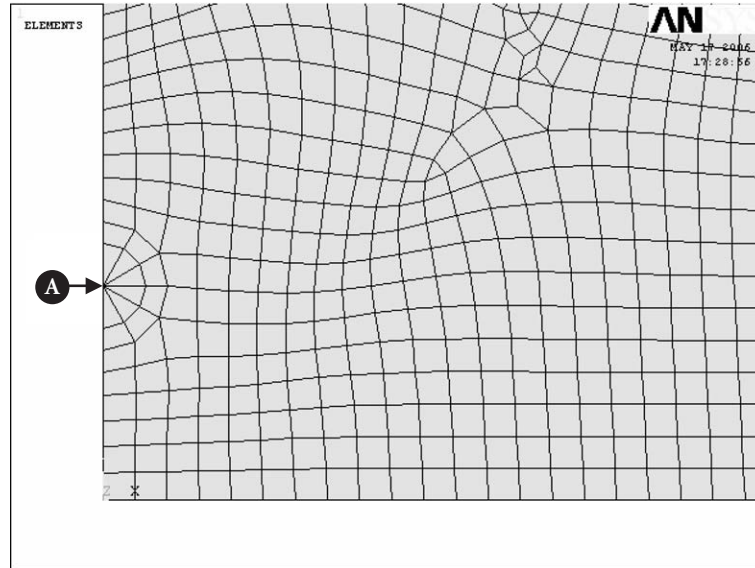


Figure 3.85 Enlarged view of the singular elements around the crack tip.

3.4.3.4 INPUT OF BOUNDARY CONDITIONS

[1] Imposing constraint conditions on the ligament region of the left end and the bottom side of the quarter plate model

Due to the symmetry, the constraint conditions of the quarter plate model are **UX**-fixed condition on the ligament region of the left end, that is, the line between Key points #4 and #5, and **UY**-fixed condition on the bottom side of the quarter plate model. Apply these constraint conditions onto the corresponding lines by the following commands:

COMMAND | ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Displacement → On Lines

- (1) The **Apply U. ROT on Lines** window opens and the upward arrow appears when the mouse cursor is moved to the **ANSYS Graphics** window.
- (2) Confirming that the **Pick** and **Single** buttons are selected, move the upward arrow onto the line between Key points #4 and #5 and click the left button of the mouse.
- (3) Click the **OK** button in the **Apply U. ROT on Lines** window to display another **Apply U. ROT on Lines** window.

- (4) Select **UX** in the **Lab2** box and click the **OK** button in the **Apply U. ROT on Lines** window.

Repeat the commands and operations (1) through (3) above for the bottom side of the model. Then, select **UY** in the **Lab2** box and click the **OK** button in the **Apply U. ROT on Lines** window.

[2] Imposing a uniform longitudinal stress on the right end of the quarter plate model

A uniform longitudinal stress can be defined by pressure on the right-end side of the plate model as described below:

COMMAND | **ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Pressure → On Lines**

- (1) The **Apply PRES on Lines** window opens and the upward arrow appears when the mouse cursor is moved to the **ANSYS Graphics** window.
- (2) Confirming that the **Pick** and **Single** buttons are selected, move the upward arrow onto the right-end side of the quarter plate area and click the left button of the mouse.
- (3) Another **Apply PRES on Lines** window opens. Select **Constant value** in the [SFL] **Apply PRES on lines as a** box and input **−10** (MPa) in the **VALUE Load PRES value** box and leave a blank in the **Value** box.
- (4) Click the **OK** button in the window to define a uniform tensile stress of 10 MPa applied to the right end of the quarter plate model.

Figure 3.86 illustrates the boundary conditions applied to the center-cracked tension plate model by the above operations.

3.4.3.5 SOLUTION PROCEDURES

The solution procedures are also the same as usual.

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

- (1) The **Solve Current Load Step** and **/STATUS Command** windows appear.
- (2) Click the **OK** button in the **Solve Current Load Step** window to begin the solution of the current load step.
- (3) The **Verify** window opens as shown in Figure 3.87. Proceed to the next operation below by clicking [A] **Yes** button in the window.
- (4) Select the **File** button in **/STATUS Command** window to open the submenu and select the **Close** button to close the **/STATUS Command** window.
- (5) When the solution is completed, the **Note** window appears. Click the **Close** button to close the **Note** window.

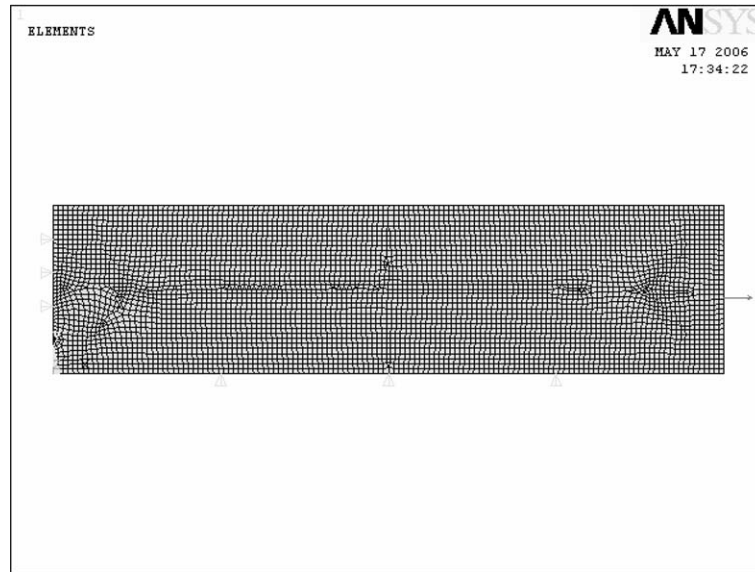


Figure 3.86 Boundary conditions applied to the center-cracked tension plate model.

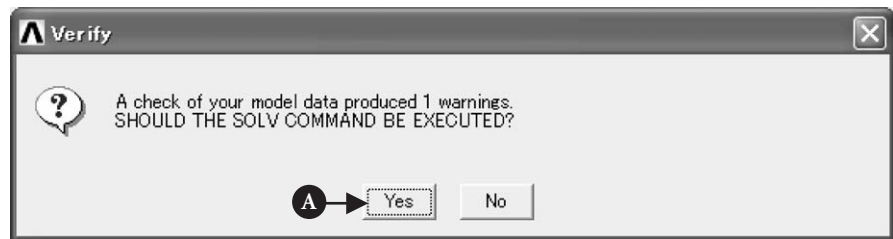


Figure 3.87 “Verify” window.

3.4.3.6 CONTOUR PLOT OF STRESS

COMMAND | ANSYS Main Menu → General Postproc → Plot Results → Contour Plot → Nodal Solution

- (1) The **Contour Nodal Solution Data** window opens.
- (2) Select **Stress** and **X-Component of stress**.
- (3) Select **Deformed shape with deformed edge** in the **Undisplaced shape key** box to compare the shapes of the cracked plate before and after deformation.
- (4) Click the **OK** button to display the contour of the x-component of stress, or longitudinal stress in the center-cracked tension plate in the **ANSYS Graphics** window as shown in Figure 3.88.

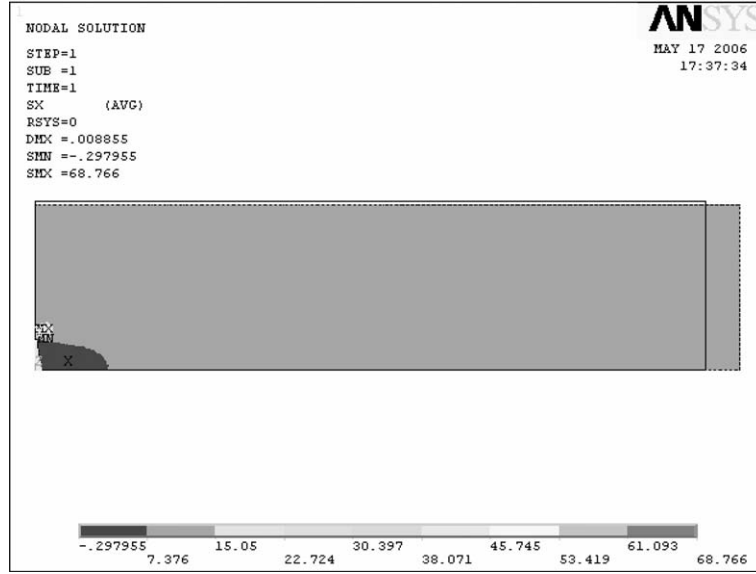


Figure 3.88 Contour of the x-component of stress in the center-cracked tension plate.

Figure 3.89 is an enlarged view of the longitudinal stress distribution around the crack tip showing that very high tensile stress is induced at the crack tip whereas almost zero stress around the crack surface and that the crack shape is parabolic.

3.4.4 DISCUSSION

Figure 3.90 shows extrapolation of the values of the correction factor, or the non-dimensional mode I (the crack-opening mode) stress intensity factor $F_I = K_I/(\sigma\sqrt{\pi a})$ to the point where $r = 0$, that is, the crack tip position by the hybrid extrapolation method [1]. The plots in the right region of the figure are obtained by the formula:

$$K_I = \lim_{r \rightarrow 0} \sqrt{2\pi r} \sigma_x(\theta = 0) \quad (3.6)$$

or

$$F_I(\lambda) = \frac{K_I}{\sigma\sqrt{\pi a}} = \lim_{r \rightarrow 0} \frac{\sqrt{\pi r} \sigma_x(\theta = 0)}{\sqrt{h} \sigma \sqrt{\pi \lambda}} \quad \text{where } \lambda = 2a/h \quad (3.6')$$

whereas those in the left region

$$K_I = \lim_{r \rightarrow 0} \sqrt{\frac{2\pi}{r}} \frac{E}{(1 + \nu)(\kappa + 1)} u_x(\theta = \pi) \quad (3.7)$$

or

$$F_I(\lambda) = \frac{K_I}{\sigma\sqrt{\pi a}} = \lim_{r \rightarrow 0} \sqrt{\frac{\pi}{hr}} \frac{E}{(1 + \nu)(\kappa + 1)\sigma\sqrt{\pi \lambda}} u_x(\theta = \pi) \quad (3.7')$$

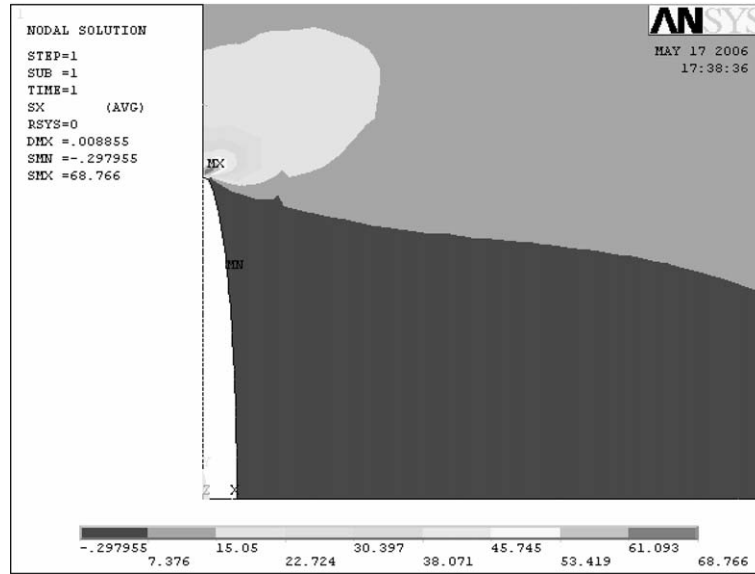


Figure 3.89 Enlarged view of the longitudinal stress distribution around the crack tip.

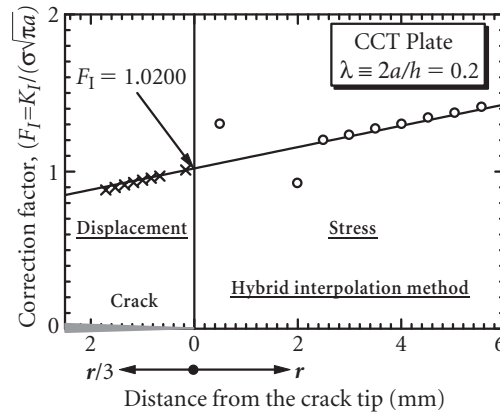


Figure 3.90 Extrapolation of the values of the correction factor $F_I = K_I / (\sigma \sqrt{\pi a})$ to the crack tip point where $r = 0$.

where r is the distance from the crack tip, σ_x ($\theta = 0$) the x -component of stress at $\theta = 0$, i.e., along the ligament, σ the applied uniform stress in the direction of the x -axis, a the half crack length, E Young's modulus, $u_x(\theta = \pi)$ the x -component of strain at $\theta = \pi$, i.e., along the crack surface, $\kappa = 3 - 4\nu$ for plane strain condition and $\kappa = (3 - \nu)/(1 + \nu)$ for plane stress condition and θ the angle around the crack tip. The correction factor F_I accounts for the effect of the finite boundary, or the edge effect of the plate. The value of F_I can be evaluated either by the stress extrapolation method

(the right region) or by the displacement extrapolation method (the left region). The hybrid extrapolation method [1] is a blend of the two types of the extrapolation methods above and can bring about better solutions. As illustrated in Figure 3.90, the extrapolation lines obtained by the two methods agree well with each other when the gradient of the extrapolation line for the displacement method is rearranged by putting $r' = r/3$. The value of F_I obtained for the present CCT plate by the hybrid method is 1.0200.

Table 3.1 Correction factor $F_I(\lambda)$ as a function of non-dimensional crack length $\lambda = 2a/h$

$\lambda = 2a/h$	Isida Equation (3.8)	Feddersen Equation (3.9)	Koiter Equation (3.10)	Tada Equation (3.11)
0.1	1.0060	1.0062	1.0048	1.0060
0.2	1.0246	1.0254	1.0208	1.0245
0.3	1.0577	1.0594	1.0510	1.0574
0.4	1.1094	1.1118	1.1000	1.1090
0.5	1.1867	1.1892	1.1757	1.1862
0.6	1.3033	1.3043	1.2921	1.3027
0.7	1.4882	1.4841	1.4779	1.4873
0.8	1.8160	1.7989	1.8075	1.8143
0.9	2.5776	—	2.5730	2.5767

Table 3.1 lists the values of F_I calculated by the following four equations [2–6] for various values of non-dimensional crack length $\lambda = 2a/h$:

$$F_I(\lambda) = 1 + \sum_{k=1}^{35} A_k \lambda^{2k} \quad (3.8)$$

$$F_I(\lambda) = \sqrt{\sec(\pi\lambda/2)} \quad (\lambda \leq 0.8) \quad (3.9)$$

$$F_I(\lambda) = \frac{1 - 0.5\lambda + 0.326\lambda^2}{\sqrt{1 - \lambda}} \quad (3.10)$$

$$F_I(\lambda) = (1 - 0.025\lambda^2 + 0.06\lambda^4)\sqrt{\sec(\pi\lambda/2)} \quad (3.11)$$

Among these, Isida's solution is considered to be the best. The value of $F_I(0.2)$ by Isida is 1.0246. The relative error of the present result is $(1.0200 - 1.0246)/1.0246 = -0.45\%$, which is a reasonably good result.

3.4.5 PROBLEMS TO SOLVE

PROBLEM 3.12

Calculate the values of the correction factor for the mode I stress intensity factor for the center-cracked tension plate having normalized crack lengths of $\lambda = 2a/h = 0.1$

to 0.9. Compare the results with the values of the correction factor $F(\lambda)$ listed in Table 3.1

PROBLEM 3.13

Calculate the values of the correction factor for the mode I stress intensity factor for a double edge cracked tension plate having normalized crack lengths of $\lambda = 2a/h = 0.1$ to 0.9 (see Figure P3.13). Compare the results with the values of the correction factor $F(\lambda)$ calculated by Equation (P3.13) [7, 8]. Note that the quarter model described in the present section can be used for the FEM calculations of this problem simply by changing the boundary conditions.

$$F_I(\lambda) = \frac{1.122 - 0.561\lambda - 0.205\lambda^2 + 0.471\lambda^3 - 0.190\lambda^4}{\sqrt{1 - \lambda}} \quad \text{where } \lambda = 2a/h \quad (\text{P3.13})$$

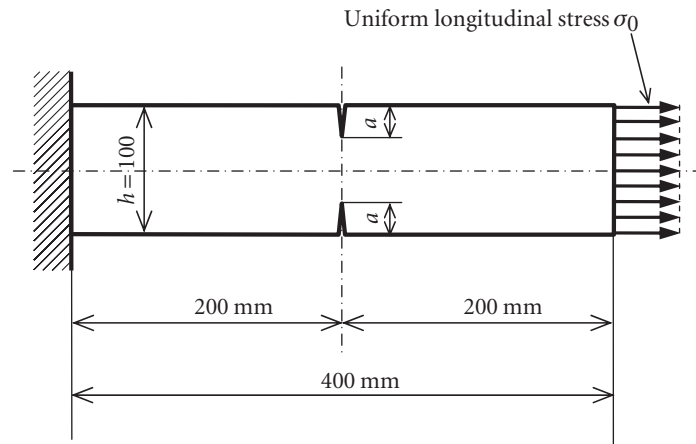


Figure P3.13 Double edge cracked tension plate subjected to a uniform longitudinal stress at one end and clamped at the other end.

PROBLEM 3.14

Calculate the values of the correction factor for the mode I stress intensity factor for a single edge cracked tension plate having normalized crack lengths of $\lambda = 2a/h = 0.1$ to 0.9 (see Figure P3.14). Compare the results with the values of the correction factor $F(\lambda)$ calculated by Equation (P3.14) [7, 9]. In this case, a half plate model which is clamped along the ligament and is loaded at the right end of the plate. Note that the whole plate model with an edge crack needs a somewhat complicated procedure. An

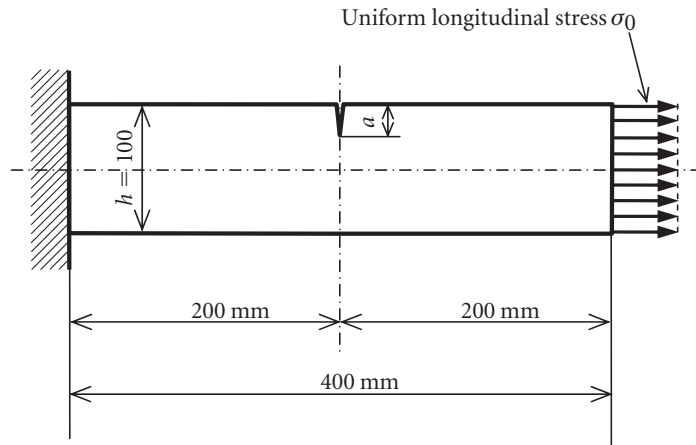


Figure P3.14 Single edge cracked tension plate subjected to a uniform longitudinal stress at one end and clamped at the other end.

approximate model as described above gives good solutions with an error of a few percent or less and is sometimes effective in practice.

$$F_I(\lambda) = \frac{0.752 + 2.02\lambda + 0.37[1 - \sin(\pi\lambda/2)]^3}{\cos(\pi\lambda/2)} \sqrt{\frac{2}{\pi\lambda} \tan\left(\frac{\pi\lambda}{2}\right)} \quad \text{where } \lambda = a/h \quad (\text{P3.14})$$

3.5 TWO-DIMENSIONAL CONTACT STRESS

3.5.1 EXAMPLE PROBLEM

An elastic cylinder of mild steel with a radius of R pressed against a flat surface of a linearly elastic medium of the same material by a force P' . Perform an FEM analysis of the elastic cylinder pressed against the elastic flat plate illustrated in Figure 3.91 and calculate the resulting contact stress in the vicinity of the contact point in the flat plate.

3.5.2 PROBLEM DESCRIPTION

Geometry of the cylinder: radius $R = 1000$ mm.

Geometry of the flat plate: width $W = 1000$ mm, height $h = 500$ mm.

Material: mild steel having Young's modulus $E = 210$ GPa and Poisson's ratio $\nu = 0.3$.
Boundary conditions: The elastic cylinder is pressed against the elastic flat plate by a force of $P' = 1$ kN/mm and the plate is clamped at the bottom. Note that the cylinder and the plate is considered to have a unit thickness in the case of plane stress condition.

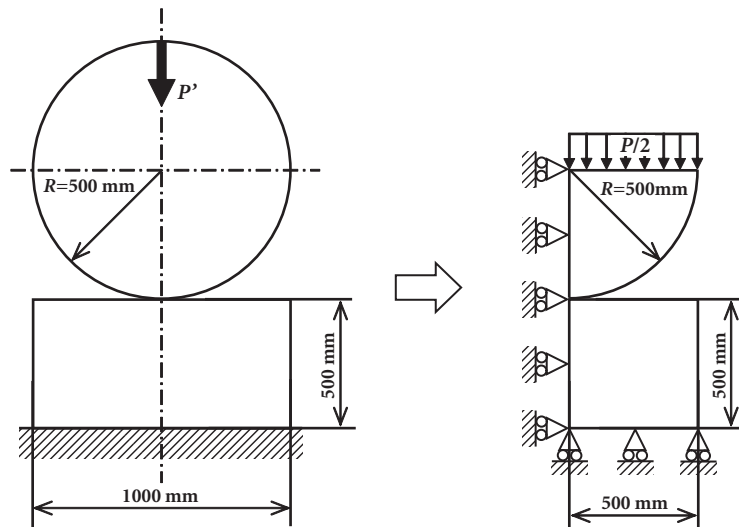


Figure 3.91 Elastic cylinder of mild steel with a radius of R pressed against an elastic flat plate of the same steel by a force P' and the FEM model of the cylinder-plate system.

3.5.3 ANALYTICAL PROCEDURES

3.5.3.1 CREATION OF AN ANALYTICAL MODEL

Let us use a half model of the indentation problem illustrated in Figure 3.91, since the problem is symmetric about the vertical center line.

[1] *Creation of an elastic flat plate*

COMMAND | ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Rectangle → By 2 Corners

- (1) Input two **0**'s into the **WP X** and **WP Y** boxes in the **Rectangle by 2 Corners** window to determine the upper left corner point of the elastic flat plate on the Cartesian coordinates of the working plane.
- (2) Input **500** and **-500** (mm) into the **Width** and **Height** boxes, respectively, to determine the shape of the elastic flat plate model.
- (3) Click the **OK** button to create the plate on the **ANSYS Graphics** window.

[2] *Creation of an elastic cylinder*

COMMAND | ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Circle → Partial Annulus

- (1) The **Part Annular Circ Area** window opens as shown in Figure 3.92.
- (2) Input **0** and **500** into [A] **WP X** and [B] **WP Y** boxes, respectively, in the **Part Annular Circ Area** window to determine the center of the elastic cylinder on the Cartesian coordinates of the working plane.
- (3) Input two **500**'s (degree) into [C] **Rad-1** and [E] **Rad-2** boxes to designate the radius of the elastic cylinder model.
- (4) Input **-90** and **0** (mm) into [D] **Theta-1** and [F] **Theta-2** boxes, respectively, to designate the starting and ending angles of the quarter elastic cylinder model.
- (5) Click [G] **OK** button to create the quarter cylinder model on the **ANSYS Graphics** window as shown in Figure 3.93.

[3] Combining the elastic cylinder and the flat plate models

The elastic cylinder and the flat plate models must be combined into one model so as not to allow the rigid-body motions of the two bodies.

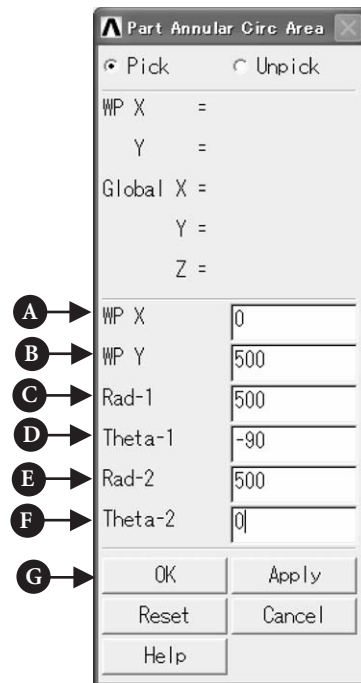


Figure 3.92 “Part Annular Circ Area” window.

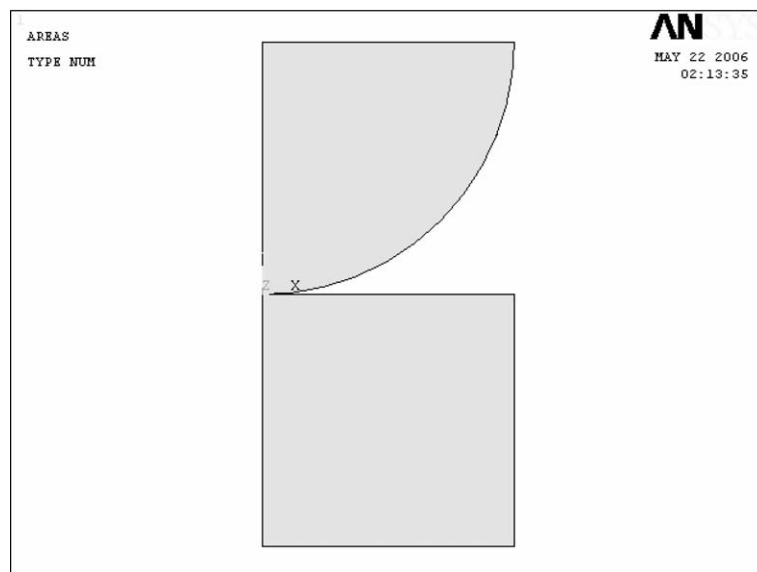


Figure 3.93 Areas of the FEM model for an elastic cylinder pressed against an elastic flat plate.

COMMAND

ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Glue → Areas

- (1) The **Glue Areas** window opens as shown in Figure 3.94.
- (2) The upward arrow appears in the **ANSYS Graphics** window. Move this arrow to the quarter cylinder area and click this area, and then move the arrow on to the half plate area and click this area to combine these two areas.
- (3) Click [A] **OK** button to create the contact model on the **ANSYS Graphics** window as shown in Figure 3.93.

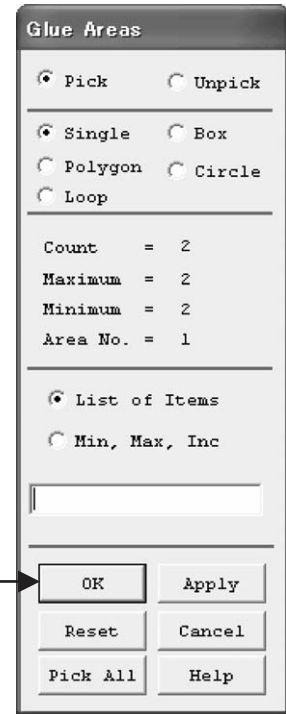


Figure 3.94 “Glue Areas” window.

3.5.3.2 INPUT OF THE ELASTIC PROPERTIES OF THE MATERIAL FOR THE CYLINDER AND THE FLAT PLATE

COMMAND

ANSYS Main Menu → Preprocessor → Material Props → Material Models

- (1) The **Define Material Model Behavior** window opens.
- (2) Double-click **Structural**, **Linear**, **Elastic**, and **Isotropic** buttons one after another.
- (3) Input the value of Young’s modulus, **2.1e5** (MPa), and that of Poisson’s ratio, **0.3**, into **EX** and **PRXY** boxes, and click the **OK** button of the **Linear Isotropic Properties for Materials Number 1** window.
- (4) Exit from the **Define Material Model Behavior** window by selecting **Exit** in the **Material** menu of the window.

3.5.3.3 FINITE-ELEMENT DISCRETIZATION OF THE CYLINDER AND THE FLAT PLATE AREAS

[1] Selection of the element types

Select the 8-node isoparametric elements as usual.

COMMAND

ANSYS Main Menu → Preprocessor → Element Type → Add/Edit/Delete

- (1) The **Element Types** window opens.
- (2) Click the **Add ...** button in the **Element Types** window to open the **Library of Element Types** window and select the element type to use.

- (3) Select **Structural Mass – Solid** and **Quad 8 node 82**.
- (4) Click the **OK** button in the **Library of Element Types** window to use the 8-node isoparametric element. Note that this element is defined as Type 1 element as indicated in the **Element type reference number** box.
- (5) Click the **Options ...** button in the **Element Types** window to open the **PLANE82 element type options** window. Select the **Plane strain** item in the **Element behavior** box and click the **OK** button to return to the **Element Types** window. Click the **Close** button in the **Element Types** window to close the window.

In contact problems, two mating surfaces come into contact with each other exerting great force on each other. Contact elements must be used in contact problems for preventing penetration of one object into the other:

- (6) Repeat clicking the **Add ...** button in the **Element Types** window to open the **Library of Element Types** window and select **Contact** and **2D target 169**.
- (7) Click the **OK** button in the **Library of Element Types** window to select the target element. Note that this target element is defined as Type 2 element as indicated in the **Element type reference number** box.
- (8) Clicking again the **Add ...** button in the **Element Types** window to open the **Library of Element Types** window and select **Contact** and **3 nd surf 172**.
- (9) Click the **OK** button in the **Library of Element Types** window to select the surface effect element. Note that this target element is defined as Type 3 element as indicated in the **Element type reference number** box.

[2] Sizing of the elements

Here let us designate the element size by specifying number of divisions of lines picked. For this purpose, carry out the following commands:

COMMAND | ANSYS Main Menu → Preprocessor →
Meshing → Size Cntrls → Manual Size →
Lines → Picked Lines

- (1) The **Element Size on Pick ...** window opens as shown in Figure 3.95.
- (2) Move the upward arrow to the upper and the right sides of the flat plate area and click [A] **OK** button and the **Element Sizes on Picked Lines** window opens as shown in Figure 3.96.
- (3) Input [A] **60** in **NDIV** box and [B] **1/10** in **SPACE** box to divide the upper and right side of the plate area into 60 subdivisions. The size of the subdivisions decreases in geometric progression as approaching to the left side of the

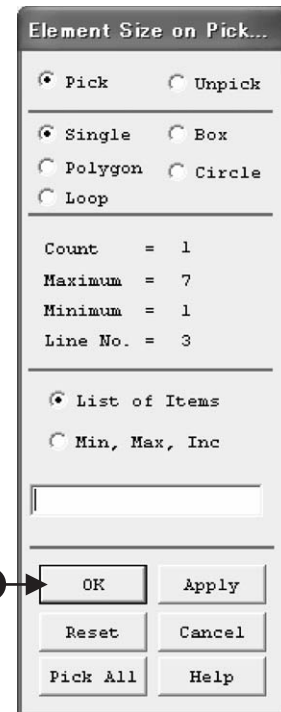


Figure 3.95 “Element Size on Pick ...” window.

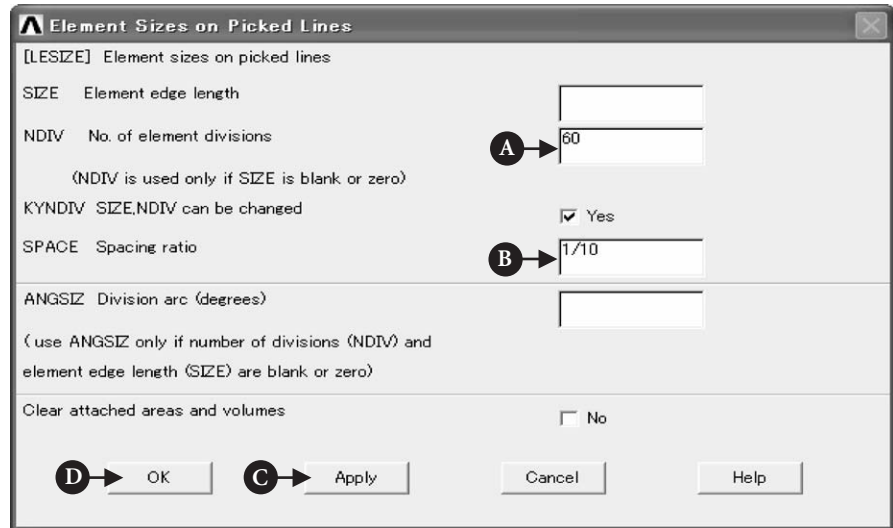


Figure 3.96 “Element Sizes on Picked Lines” window.

plate area. It would be preferable that we may have smaller elements around the contact point where high stress concentration occurs. Click [C] **Apply** button to continue to input more **NDIV** and **SPACE** data. Input **60** in the **NDIV** box and **10** in the **SPACE** box for the left and the bottom sides of the plate, whereas **40** and **10** for the left side and the circumference of the quarter circular area and **40** and **1/10** for the upper side of the quarter circular area in the respective boxes. Finally, click [D] **OK** button to close the window.

[3] Meshing

The meshing procedures are the same as usual:

COMMAND | **ANSYS Main Menu → Preprocessor → Meshing → Mesh → Areas → Free**

- (1) The **Mesh Areas** window opens.
- (2) The upward arrow appears in the **ANSYS Graphics** window. Move this arrow to the quarter cylinder and the half plate areas and click these areas.
- (3) The color of the areas turns from light blue into pink. Click the **OK** button to see the areas meshed by 8-node isoparametric finite elements as shown in Figure 3.97.
- (4) Figure 3.98 is an enlarged view of the finer meshes around the contact point.

[4] Creation of target and contact elements

First select the lower surface of the quarter cylinder to which the contact elements are attached:

COMMAND | **ANSYS Utility Menu → Select → Entities ...**

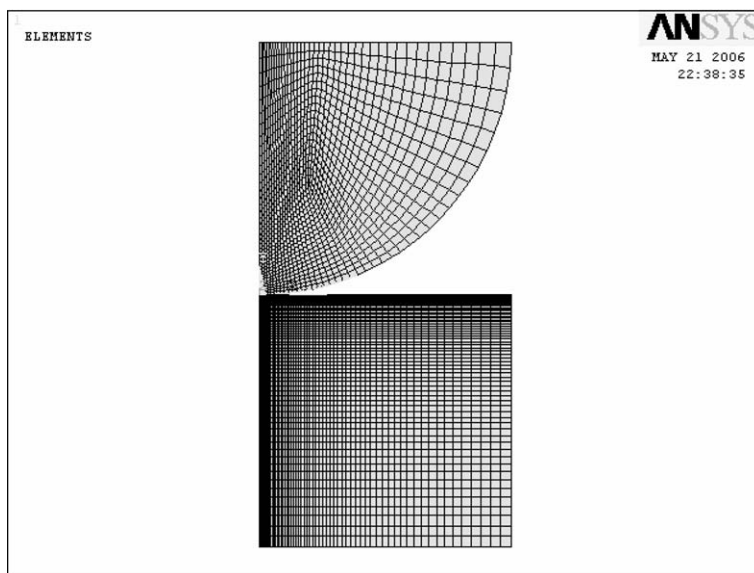


Figure 3.97 Cylinder and flat plate areas meshed by 8-node isoparametric finite elements.

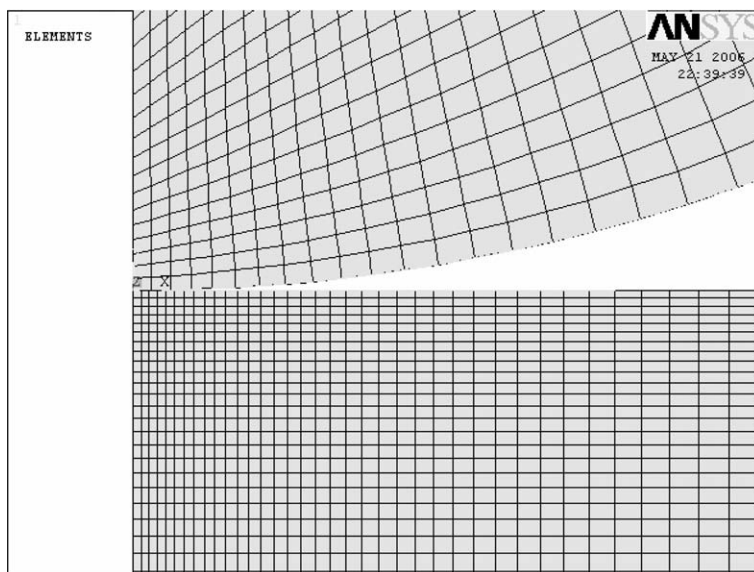


Figure 3.98 Enlarged view of the finer meshes around the contact point.

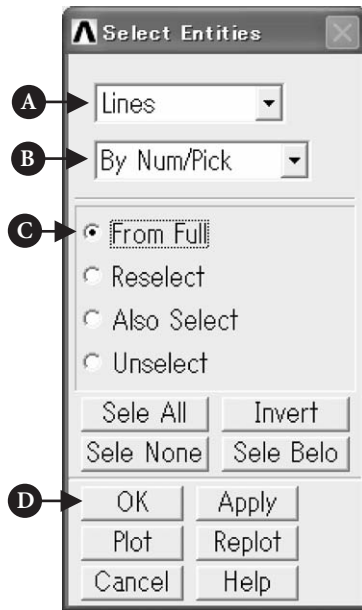


Figure 3.99 “Select Entities” window.

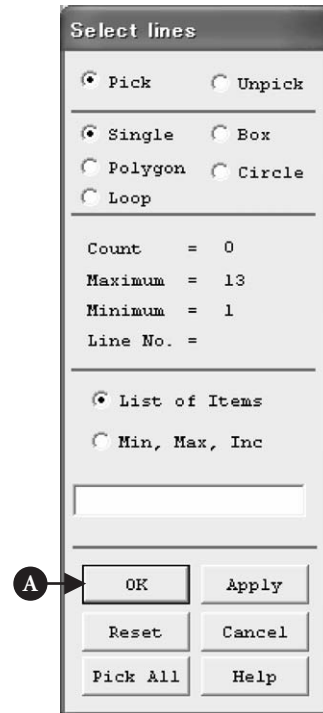


Figure 3.100 “Select lines” window.

- (1) The **Select Entities** window opens as shown in Figure 3.99.
- (2) Select [A] **Lines**, [B] **By Num/Pick** and [C] **From Full** in the **Select Entities** window.
- (3) Click [D] **OK** button in the **Select Entities** window and the **Select lines** window opens as shown in Figure 3.100.
- (4) The upward arrow appears in the **ANSYS Graphics** window. Move this arrow to the quarter circumference of the cylinder and click this line. The lower surface of the cylinder is selected as shown in Figure 3.101.

Repeat the **Select** → **Entities ...** commands to select nodes on the lower surface line of the cylinder:

COMMAND | **ANSYS Utility Menu** → **Select** → **Entities ...**

- (1) The **Select Entities** window opens as shown in Figure 3.102.
- (2) Select [A] **Nodes**, [B] **Attached to**, [C] **Lines, all** and [D] **Reselect** in the **Select Entities** window.

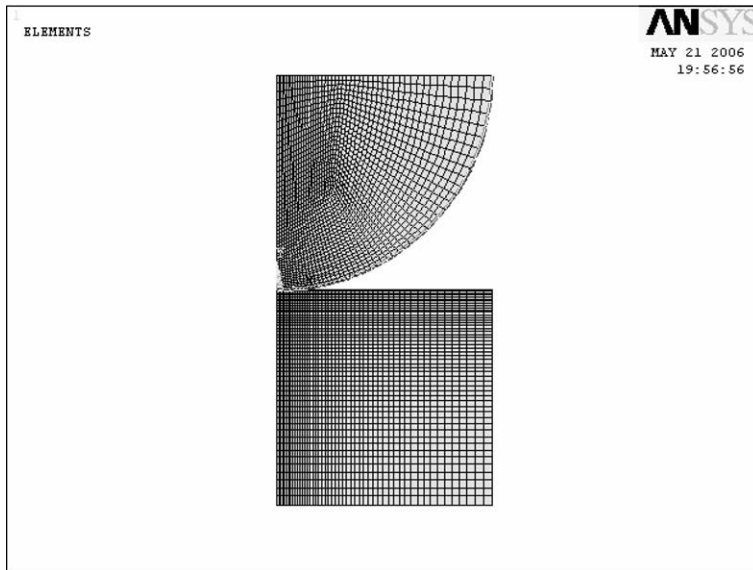


Figure 3.101 Lower surface of the cylinder selected.

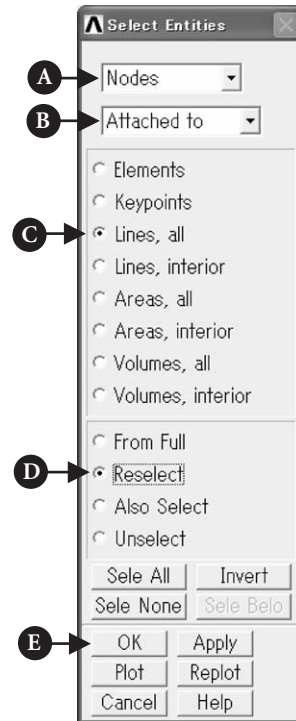


Figure 3.102 “Select Entities” window.

- (3) Click [E] **OK** button and perform the following commands:

COMMAND | **ANSYS Utility Menu → Plot → Nodes**

Only nodes on the lower surface of the cylinder are plotted in the **ANSYS Graphics** window as shown in Figure 3.103.

Repeat the **Select → Entities ...** commands again to select a fewer number of the nodes on the portion of the lower surface of the cylinder in the vicinity of the contact point:

COMMAND | **ANSYS Utility Menu → Select → Entities ...**

- (1) The **Select Entities** window opens as shown in Figure 3.104.
- (2) Select [A] **Nodes**, [B] **By Num/Pick** and [C] **Reselect** in the **Select Entities** window.
- (3) Click [D] **OK** button and the **Select nodes** window opens as shown in Figure 3.105.
- (4) The upward arrow appears in the **ANSYS Graphics** window. Select [A] **Box** instead of **Single**, then move the upward arrow to the leftmost end of the array of nodes and select nodes by enclosing them with a rectangle formed by dragging

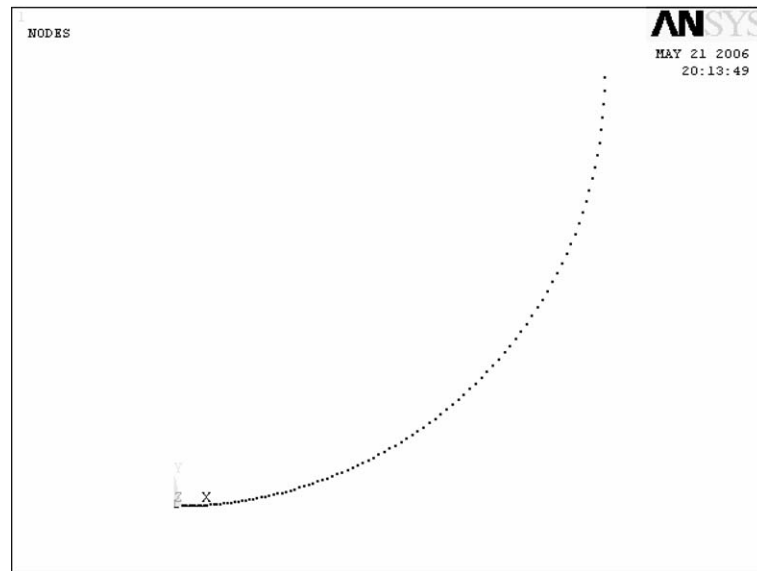


Figure 3.103 Nodes on the lower surface of the cylinder plotted in the “ANSYS Graphics” window.

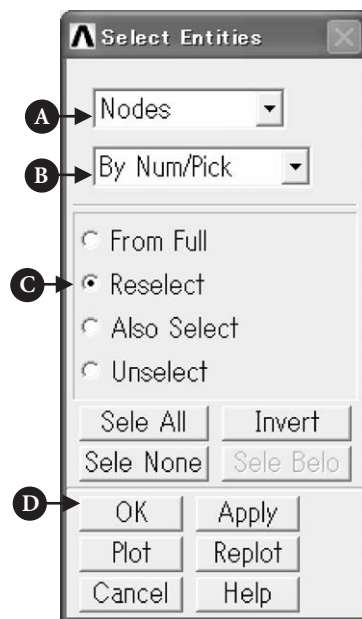


Figure 3.104 “Select Entities” window.

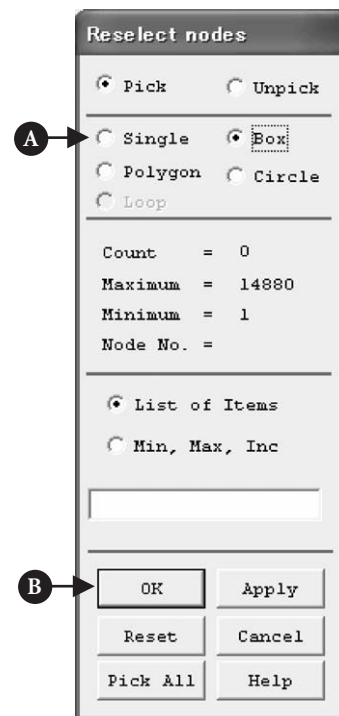


Figure 3.105 “Reselect nodes” window.

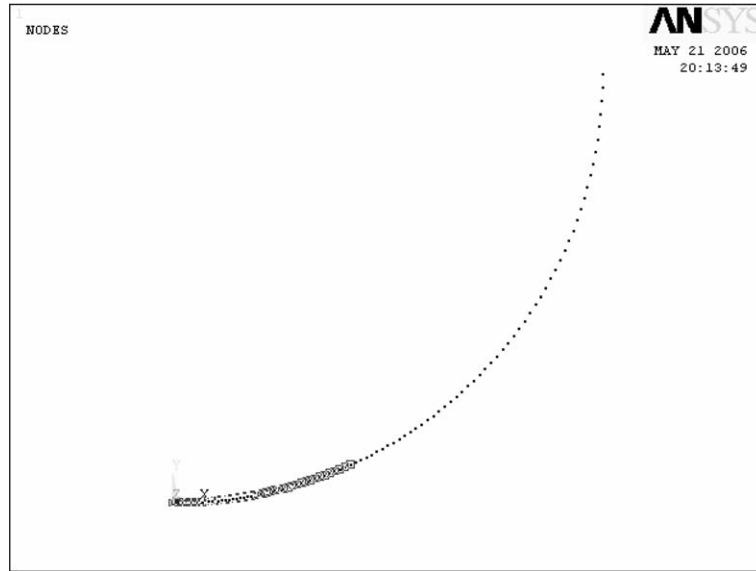


Figure 3.106 Selection of nodes on the portion of the bottom surface of the cylinder.

the mouse as shown in Figure 3.106. Click [B] **OK** button to select the nodes on the bottom surface of the cylinder.

Next, let us define contact elements to be attached to the lower surface of the cylinder by the commands as follows:

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Create → Elements → Elem Attributes**

- (1) The **Element Attributes** window opens as shown in Figure 3.107.
- (2) Select [A] **3 CONTA172** in the **TYPE** box and click [B] **OK** button in the **Element Attributes** window.

Then, perform the following commands to attach the contact elements to the lower surface of the cylinder.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Create → Elements → Surf/Contact → Surf to Surf**

- (1) The **Mesh Free Surfaces** window opens as shown in Figure 3.108.
- (2) Select [A] **Bottom surface** in the **Tlab** box and click [B] **OK** button in the window.
- (3) Another **Mesh Free Surfaces** like the **Reselect notes** windows as shown in Figure 3.105 window opens. The upward arrow appears in the **ANSYS Graphics** window. Select the **Box** radio button and move the upward arrow to the left end of the array of nodes. Enclose all the nodes as shown in Figure 3.16 with a rectangle formed

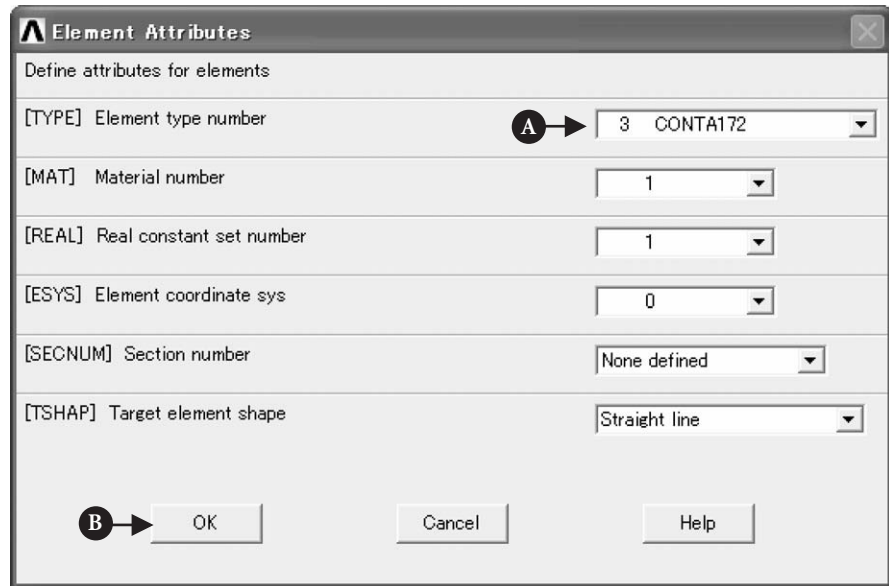


Figure 3.107 “Element Attributes” window.

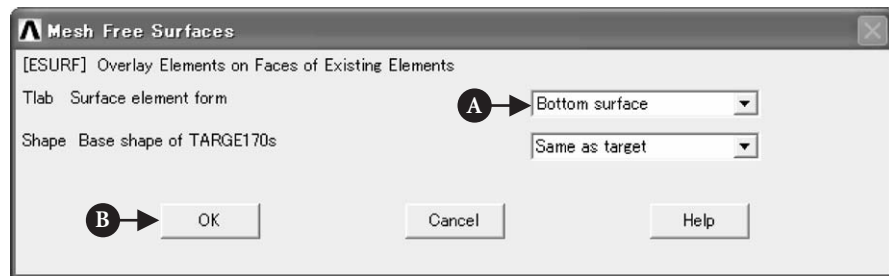


Figure 3.108 “Mesh Free Surfaces” window.

by dragging mouse to the right end of the node array. Click the **OK** button to select the nodes to which **CONTA172** element are to be attached.

- (4) **CONTA172** elements are created on the lower surface of the cylinder as shown in Figure 3.109.

Before proceeding to the next step, **Select → Everything** command in the **ANSYS Utility Menu** must be carried out in order to select nodes other than those selected by the previous procedures.

Repeat operations similar to the above in order to create **TARGET169** elements on the top surface of the flat plate: Namely, perform the **Select → Entities ...** commands to select only nodes on the portion of the top surface of the flat plate in the vicinity of

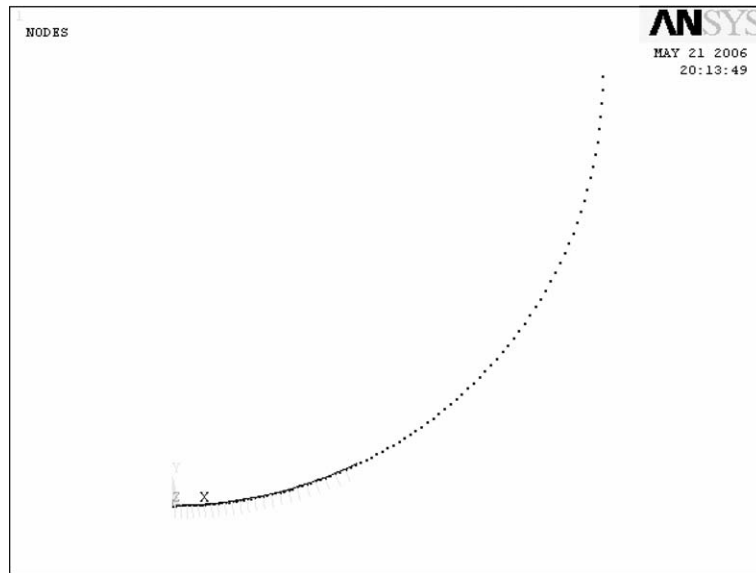


Figure 3.109 “CONTA172” elements created on the lower surface of the cylinder.

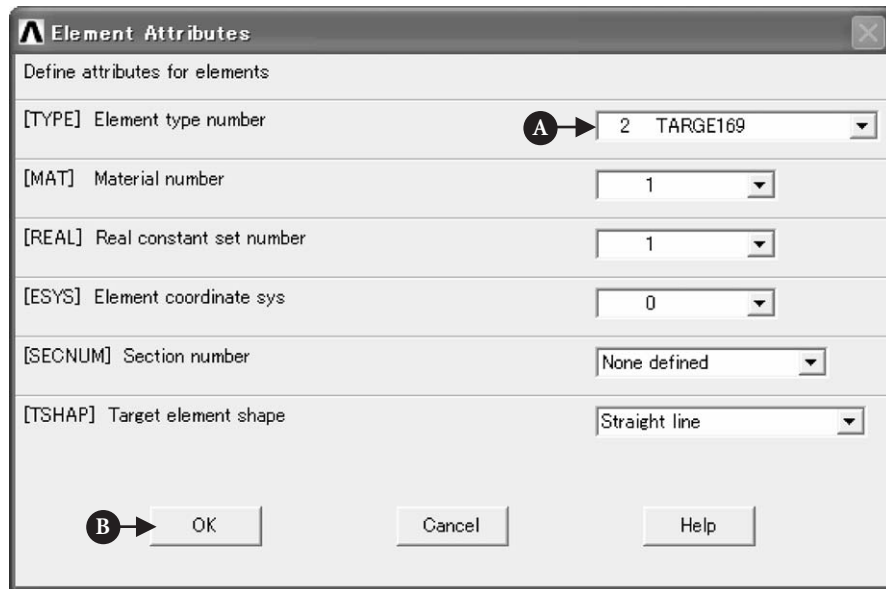


Figure 3.110 “Element Attributes” window.

the contact point. Then, define the target elements to be attached to the top surface of the flat plate by the commands as follows:

COMMAND | ANSYS Main Menu → Preprocessor → Modeling → Create → Elements → Elem Attributes

- (1) The **Element Attributes** window opens as shown in Figure 3.110.
- (2) Select [A] **2 TARGET169** in the **TYPE** box and click [B] **OK** button in the **Element Attributes** window.

Then, perform the following commands to attach the target elements to the top surface of the flat plate:

COMMAND | ANSYS Main Menu → Preprocessor → Modeling → Create → Elements → Surf/Contact → Surf to Surf

- (3) The **Mesh Free Surfaces** window opens as shown in Figure 3.111.

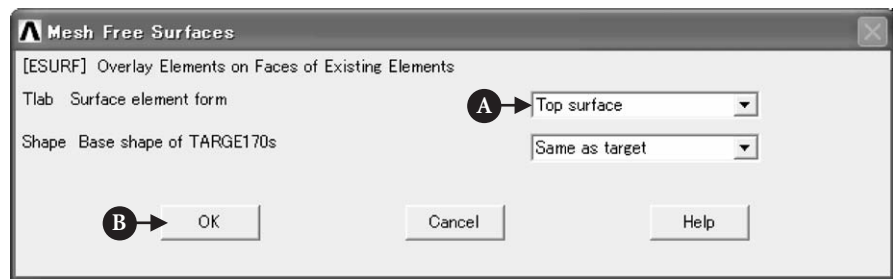


Figure 3.111 “Mesh Free Surfaces” window.

- (4) Select [A] **Top surface** in the **Tlabb** box and click [B] **OK** button in the window.
- (5) **TARGET169** elements are created on the top surface of the flat plate as shown in Figure 3.112.
- (6) Perform the **Select → Everything** command in the **ANSYS Utility Menu** in order to impose boundary conditions on nodes other than those selected by the previous procedures.

3.5.3.4 INPUT OF BOUNDARY CONDITIONS

[1] Imposing constraint conditions on the left sides of the quarter cylinder and the half flat plate models

Due to the symmetry, the constraint conditions of the present models are **UX**-fixed condition on the left sides of the quarter cylinder and the half flat plate models, and **UY**-fixed condition on the bottom side of the half flat plate model. Apply these constraint conditions onto the corresponding lines by the following commands:

COMMAND | ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Displacement → On Lines

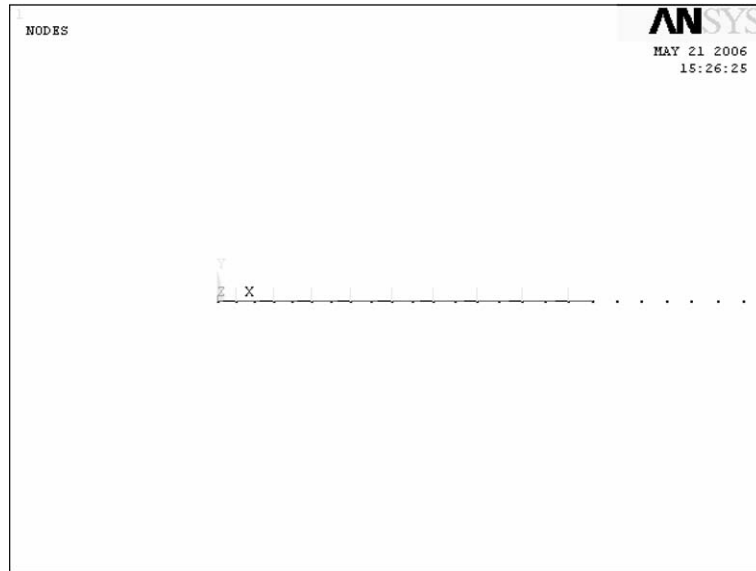


Figure 3.112 “TARGET169” elements created on the top surface of the flat plate.

- (1) The **Apply U. ROT on Lines** window opens and the upward arrow appears when the mouse cursor is moved to the **ANSYS Graphics** window.
- (2) Confirming that the **Pick** and **Single** buttons are selected, move the upward arrow onto the left sides of the quarter cylinder and the half flat plate models and click the left button of the mouse on each model.
- (3) Click the **OK** button in the **Apply U. ROT on Lines** window to display another **Apply U. ROT on Lines** window.
- (4) Select **UX** in the **Lab2** box and click the **OK** button in the **Apply U. ROT on Lines** window.

Repeat the commands and operations (1) through (3) above for the bottom side of the flat plate model. Then, select **UY** in the **Lab2** box and click the **OK** button in the **Apply U. ROT on Lines** window.

[2] Imposing a uniform pressure on the upper side of the quarter cylinder model

COMMAND | ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Pressure → On Lines

- (1) The **Apply PRES on Lines** window opens and the upward arrow appears when the mouse cursor is moved to the **ANSYS Graphics** window.

- (2) Confirming that the **Pick** and **Single** buttons are selected, move the upward arrow onto the upper side of the quarter cylinder model and click the left button of the mouse.
- (3) Another **Apply PRES on Lines** window opens. Select **Constant value** in the [SFL] **Apply PRES on lines as a** box and input **1** (MPa) in the **VALUE Load PRES value** box and leave a blank in the **Value** box.
- (4) Click the **OK** button in the window to define a downward force of $P' = 1 \text{ kN/mm}$ applied to the elastic cylinder model.

Figure 3.113 illustrates the boundary conditions applied to the finite-element model of the present contact problem by the above operations.

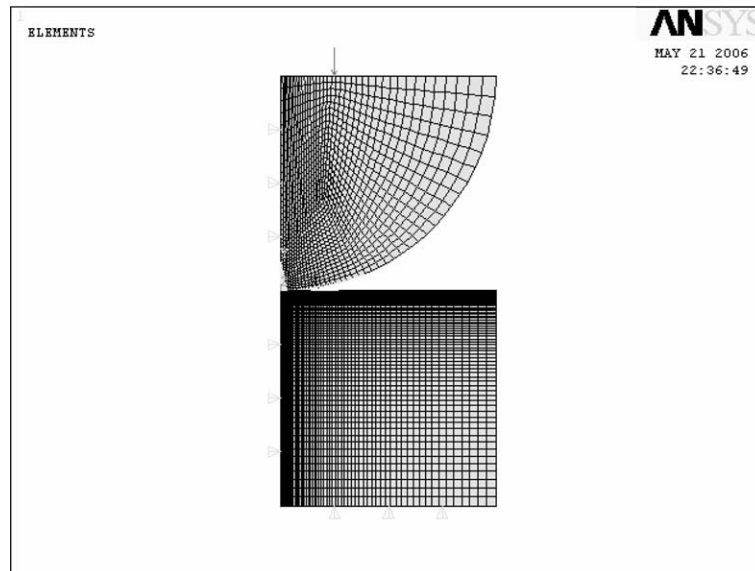


Figure 3.113 Boundary conditions applied to the finite-element model of the present contact problem.

3.5.3.5 SOLUTION PROCEDURES

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

- (1) The **Solve Current Load Step** and **/STATUS Commands** windows appear.
- (2) Click the **OK** button in the **Solve Current Load Step** window to begin the solution of the current load step.
- (3) Select the **File** button in **/STATUS Commands** window to open the submenu and select the **Close** button to close the **/STATUS Commands** window.
- (4) When the solution is completed, the **Note** window appears. Click the **Close** button to close the **Note** window.

3.5.3.6 CONTOUR PLOT OF STRESS

COMMAND | ANSYS Main Menu → General Postproc → Plot Results → Contour Plot → Nodal Solution

- (1) The **Contour Nodal Solution Data** window opens.
- (2) Select **Stress** and **Y-Component of stress**.
- (3) Click the **OK** button to display the contour of the y -component of stress in the present model of the contact problem in the **ANSYS Graphics** window as shown in Figure 3.114.

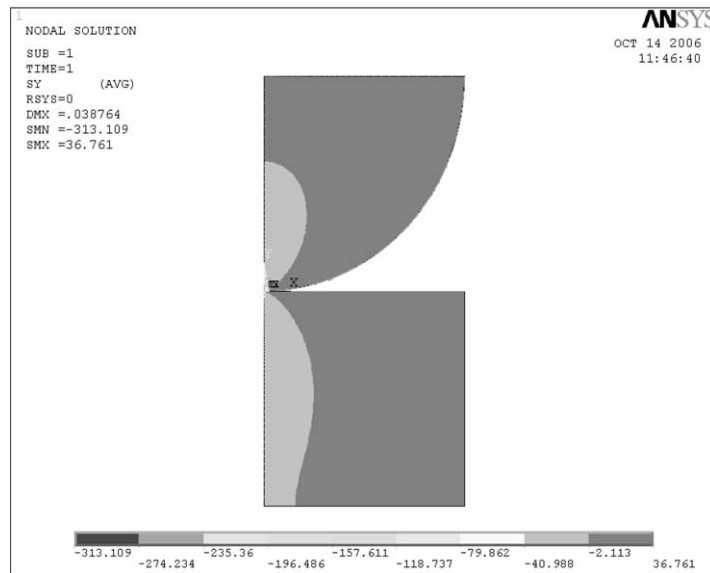


Figure 3.114 Contour of the y -component of stress in the present model of the contact problem.

Figure 3.115 is an enlarged contour of the y -component of stress around the contact point showing that very high compressive stress is induced in the vicinity of the contact point known as “Hertz contact stress.”

3.5.4 DISCUSSION

Open circular symbols in Figure 3.116 show the plots of the y -component stress, or vertical stress along the upper surface of the flat plate obtained by the present finite-element calculation. The solid line in the figure indicates the theoretical curve $p(x)$ expressed by a parabola [10, 11], i.e.,

$$p(x) = k\sqrt{b^2 - x^2} = 115.4\sqrt{2.349^2 - x^2} \quad (3.12)$$

where x is the distance from the center of contact, $p(x)$ the contact stress at a point x , k the coefficient given by Equation (3.13) below, and b half the width of contact

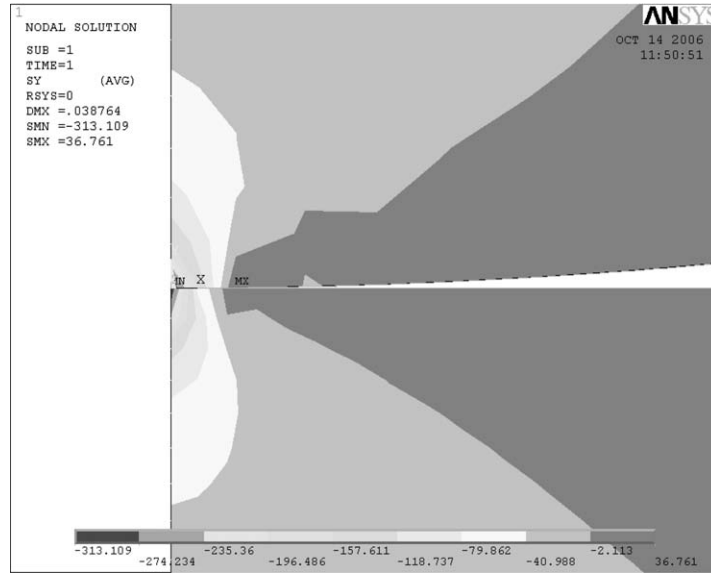


Figure 3.115 Enlarged contour of the y-component of stress around the contact point.

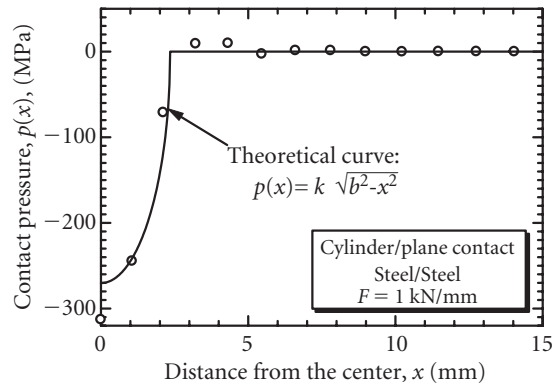


Figure 3.116 Plots of the y-component stress along the upper surface of the flat plate obtained by the present finite-element calculation.

given by Equation (3.14) below.

$$k = \frac{2P'}{\pi b^2} = 115.4 \text{ N/mm}^2 \quad (3.13)$$

$$b = 1.522 \sqrt{\frac{P'R}{E}} = 2.348 \text{ mm} \quad (3.14)$$

The maximum stress p_0 is obtained at the center of contact and expressed as,

$$p_0 = 0.418 \sqrt{\frac{P'E}{R}} = 270.9 \text{ MPa} \quad (3.15)$$

The length of contact is as small as 4.7 mm, so that the maximum value of the contact stress is as high as 270.9 MPa. The number of nodes which fall within the contact region is only 3, although the total number of nodes is approximately 15,000 and meshes are finer in the vicinity of the contact point. The three plots of the present results agree reasonably well with the theoretical curve as shown in Figure 3.116.

3.5.5 PROBLEMS TO SOLVE

PROBLEM 3.15

Calculate the contact stress distribution between a cylinder having a radius of curvature $R_1 = 500$ mm and another cylinder having $R_2 = 1000$ mm against which the smaller cylinder is pressed by a force $P' = 1$ kN/mm as shown in Figure P3.15. The two cylinders are made of the same steel having Young's modulus = 210 GPa and Poisson's ratio $\nu = 0.3$. The theoretical expression for the contact stress distribution $p(x)$ is given by Equation (P3.15a) with the parameters given by Equations (P3.15b)–(P3.15d).

$$p(x) = k\sqrt{b^2 - x^2} \quad (P3.15a)$$

$$k = \frac{2P'}{\pi b^2} \quad (P3.15b)$$

$$b = 1.522 \sqrt{\frac{P'}{E} \cdot \frac{R_1 R_2}{R_1 + R_2}} \quad (P3.15c)$$

$$p_0 = 0.418 \sqrt{\frac{P'E(R_1 + R_2)}{R_1 R_2}} \quad (P3.15d)$$

PROBLEM 3.16

Calculate the contact stress distribution between a cylinder having a radius of curvature $R = 500$ mm and a flat plate of 1000 mm width by 500 mm height against which the cylinder is pressed by a force of $P' = 1$ kN/mm as shown in Figure P3.16. The cylinder is made of fine ceramics having Young's modulus $E_1 = 320$ GPa and Poisson's ratio $\nu_1 = 0.3$, and the flat plate stainless steel having $E_2 = 192$ GPa and $\nu_2 = 0.3$. The theoretical curve of the contact stress $p(x)$ is given by Equation (P3.16a) with the parameters expressed by Equations (P3.16b) to (P3.16d) on page 140. Compare the result obtained by the FEM calculation with the theoretical distribution given by Equations (P3.16a)–(P3.16d).

$$p(x) = k\sqrt{b^2 - x^2} \quad (P3.16a)$$

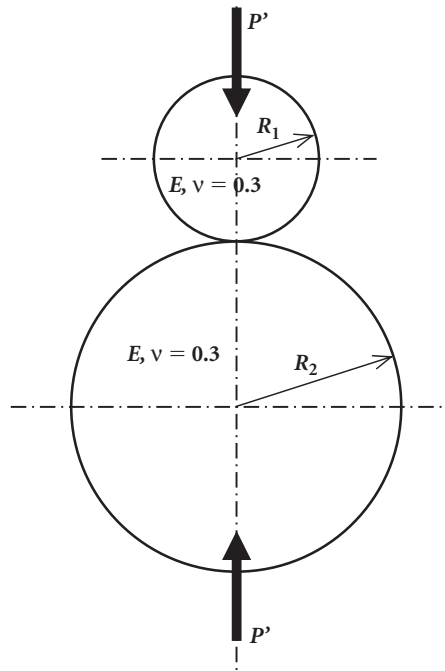


Figure P3.15 A cylinder having a radius of curvature $R_1 = 500$ mm pressed against another cylinder having $R_2 = 1000$ mm by a force $P' = 1$ kN/mm.

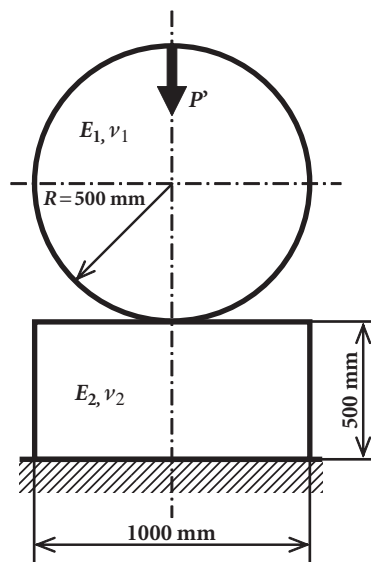


Figure P3.16 A cylinder having a radius of curvature $R = 500$ mm pressed against a flat plate.

$$k = \frac{2P'}{\pi b^2} \quad (\text{P3.16b})$$

$$b = \frac{2}{\sqrt{\pi}} \sqrt{P'R_1 \left(\frac{1-\nu_1^2}{E_1} + \frac{1-\nu_2^2}{E_2} \right)} \quad (\text{P3.16c})$$

$$p_0 = \frac{1}{\sqrt{\pi}} \sqrt{\frac{P'}{R_1} \cdot \frac{1}{\left(\frac{1-\nu_1^2}{E_1} + \frac{1-\nu_2^2}{E_2} \right)}} \quad (\text{P3.16d})$$

PROBLEM 3.17

Calculate the contact stress distribution between a cylinder having a radius of curvature $R_1 = 500$ mm and another cylinder having $R_2 = 1000$ mm against which the smaller cylinder is pressed by a force $P' = 1$ kN/mm as shown in Figure P3.15. The upper cylinder is made of fine ceramics having Young's modulus $E_1 = 320$ GPa and Poisson's ratio $\nu_1 = 0.3$, and the lower cylinder stainless steel having $E_2 = 192$ GPa and $\nu_2 = 0.3$. The theoretical curve of the contact stress $p(x)$ is given by Equation (P3.17a) with the parameters expressed by Equations (P3.17b) to (P3.17d) below. Compare the result obtained by the FEM calculation with the theoretical distribution given by Equations (P3.17a)–(P3.17d).

$$p(x) = k\sqrt{b^2 - x^2} \quad (\text{P3.17a})$$

$$k = \frac{2P'}{\pi b^2} \quad (\text{P3.17b})$$

$$b = \frac{2}{\sqrt{\pi}} \sqrt{\frac{P'R_1R_2}{R_1 + R_2} \left(\frac{1-\nu_1^2}{E_1} + \frac{1-\nu_2^2}{E_2} \right)} \quad (\text{P3.17c})$$

$$p_0 = \frac{1}{\sqrt{\pi}} \sqrt{\frac{P'(R_1 + R_2)}{R_1R_2} \cdot \frac{1}{\left(\frac{1-\nu_1^2}{E_1} + \frac{1-\nu_2^2}{E_2} \right)}} \quad (\text{P3.17d})$$

PROBLEM 3.18

Calculate the contact stress distribution between a cylinder of fine ceramics having a radius of curvature $R_1 = 500$ mm and a cylindrical seat of stainless steel having $R_2 = 1000$ mm against which the cylinder is pressed by a force $P' = 1$ kN/mm as shown in Figure P3.18. Young's module of fine ceramics and stainless steel are $E_1 = 320$ GPa and $E_2 = 192$ GPa, respectively and Poisson's ratio of each material is the same, i.e., $\nu_1 = \nu_2 = 0.3$. The theoretical curve of the contact stress $p(x)$ is given by Equation (P3.18a) with the parameters expressed by Equations (P3.18b) to (P3.18d) below.

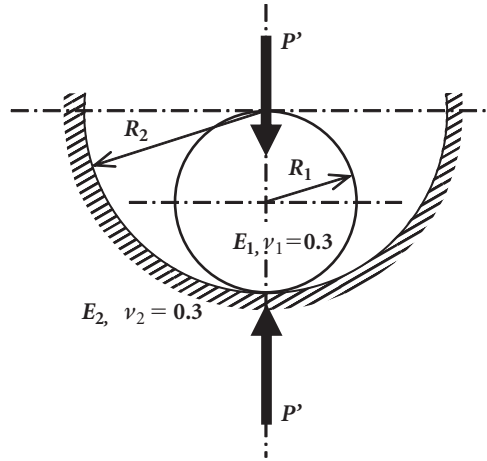


Figure P3.18 A cylinder having a radius of curvature $R_1 = 500$ mm pressed against a cylindrical seat having $R_2 = 1000$ mm by a force $P' = 1$ kN/mm.

Compare the result obtained by the FEM calculation with the theoretical distribution given by Equations (P3.18a)–(P3.18d).

$$p(x) = k\sqrt{b^2 - x^2} \quad (\text{P3.18a})$$

$$k = \frac{2P'}{\pi b^2} \quad (\text{P3.18b})$$

$$b = \frac{2}{\sqrt{\pi}} \sqrt{\frac{P'R_1R_2}{R_2 - R_1} \left(\frac{1 - \nu_1^2}{E_1} + \frac{1 - \nu_2^2}{E_2} \right)} \quad (\text{P3.18c})$$

$$p_0 = \frac{1}{\sqrt{\pi}} \sqrt{\frac{P'(R_2 - R_1)}{R_1R_2} \cdot \frac{1}{\left(\frac{1 - \nu_1^2}{E_1} + \frac{1 - \nu_2^2}{E_2} \right)}} \quad (\text{P3.18d})$$

Note that Equations (P3.18a) to (P3.18d) cover Equations (P3.15a)–(P3.15d), (P3.16a)–(P3.16d) and (P3.17a)–(P3.17d).

REFERENCES

1. R. Yuuki and H. Kisu, *Elastic Analysis with the Boundary Element Method*, Baifukan Co., Ltd., Tokyo (1987) (in Japanese).
2. M. Isida, *Elastic Analysis of Cracks and their Stress Intensity Factors*, Baifukan Co., Ltd., Tokyo (1976) (in Japanese).

3. M. Isida, *Engineering Fracture Mechanics*, Vol. 5, No. 3, pp. 647–665 (1973).
4. R. E. Feddersen, Discussion, *ASTM STP 410*, pp. 77–79 (1967).
5. W. T. Koiter, *Delft Technological University, Department of Mechanical Engineering*, Report No. 314 (1965).
6. H. Tada, *Engineering Fracture Mechanics*, Vol. 3, No. 2, pp. 345–347 (1971).
7. H. Okamura, *Introduction to the Linear Elastic Fracture Mechanics*, Baifukan Co., Ltd., Tokyo (1976) (in Japanese).
8. H. L. Ewalds and R. J. H. Wanhill, *Fracture Mechanics*, Edward Arnold, Ltd., London (1984).
9. A. Saxena, *Nonlinear Fracture Mechanics for Engineers*, CRC Press, Florida (1997).
10. I. Nakahara, *Strength of Materials*, Vol. 2, Yokendo Co., Ltd., Tokyo (1966) (in Japanese).
11. S. P. Timoshenko and J. N. Goodier, *Theory of Elasticity* (3rd edn), McGraw-Hill Kogakusha, Ltd., Tokyo (1971).