Appendix C

Finite Element Analysis Using ANSYS

C.1 INTRODUCTION

ANSYS is the original (and commonly used) name for ANSYS Mechanical or ANSYS Multiphysics, general-purpose finite element analysis software. ANSYS, Inc actually develops a complete range of CAE products, but is perhaps best known for ANSYS Mechanical & ANSYS Multiphysics. The academic versions of these commercial products are referred to as ANSYS Academic Research, ANSYS Academic Teaching Advanced, Introductory etc. All of these products are general purpose finite element self contained analysis tools incorporating preprocessing (geometry creation, meshing), solver and post processing modules in a unified graphical user interface (GUI). For the remainder of this appendix, when we state ''ANSYS'', we mean the FEA capability of either the commercial or academic ANSYS Inc. products discussed above. Further we are limiting our discussion to the traditional (or ''classic'') user interface of ANSYS, not the ANSYS Workbench environment.

One of the advantages of ''ANSYS'' is the user-programmable capability. The ANSYS Command Language contains several thousand commands relating to creating geometry, mesh, boundary conditions, solver settings & many other features. A subset of these commands are termed ANSYS Parametric Design Language (APDL), and focus on managing parameters, macros, if-then-else branching, do-loops, and scalar, vector and matrix operations. For example, if multiple analyses are required with different values of parameters, do-loops can be used to change parameters, or the Optimization module (/OPT) can be used to automatically manage parameters & populate a response surface.

There are two different modes of performing finite element analysis in ANSYS: batch and interactive modes. The batch mode requires an input file of commands and executes it from the command line, while the interactive mode turns on the graphical user interface and shows the result of each user action (picked from a menu or typed as a command) in a graphics window. The batch mode is best suited for those experienced with the ANSYS Command Language. In this tutorial, only the interactive mode will be explained.

There are two different modes of inputting commands in ANSYS: graphic mode and text mode. In the graphic mode, the user can pick commands and type in required data values. Thus, there is no need to memorize all commands. This mode is beneficial in a sense that the user can see the result of each command in the graphic window. In the text mode, the user type in commands and data in the command area. In this case, the user should remember what command should be used to perform a particular action. However, the text mode has its own advantages. For example, during the design process, the user

may want to perform the analysis multiple times with different values of parameters. In such a case, the user can prepare a text file of all commands and then input the file as an input command file. In fact, all actions in the graphic mode are stored as commands in a text file, **jobname.log**. Thus, it would be a good practice to prepare the initial model using the graphic mode, and then, modify the input file for the further modification. In this chapter, we will present two tutorials using the graphic mode, and then will present several examples in Chapters 2–7 using the text mode.

C.2 GETTING START

ANSYS can be started in different ways. In this tutorial, we will start ANSYS in the traditional mode. To access ANSYS program on Windows systems, choose **Start**> **Programs**> **ANSYS 11.0**> **ANSYS**. This command will open ANSYS window as shown in Figure C.1. The ANSYS window is composed of different areas. The graphic area shows geometries and finite elements as well as results of simulation. The main menu contains primary ANSYS functions. The user can directly type in commands in the command area. The standard toolbox collects commands that are frequently used. The utility menu contains utility functions that are available throughout the ANSYS session.

In order to finish ANSYS, choose **File**> **Exit** command. Then, **Exit from ANSYS** window will show up as shown in the right-lower corner of Figure C.1. Choose a proper option and **OK** to finish the program.

ANSYS has its full documents in help menu. In the utility menu, choose **Help**> **Help Topics**. Then ANSYS 11.0 Documentation windows will show up. In this documentation, the user can find all manuals and tutorials available in ANSYS.
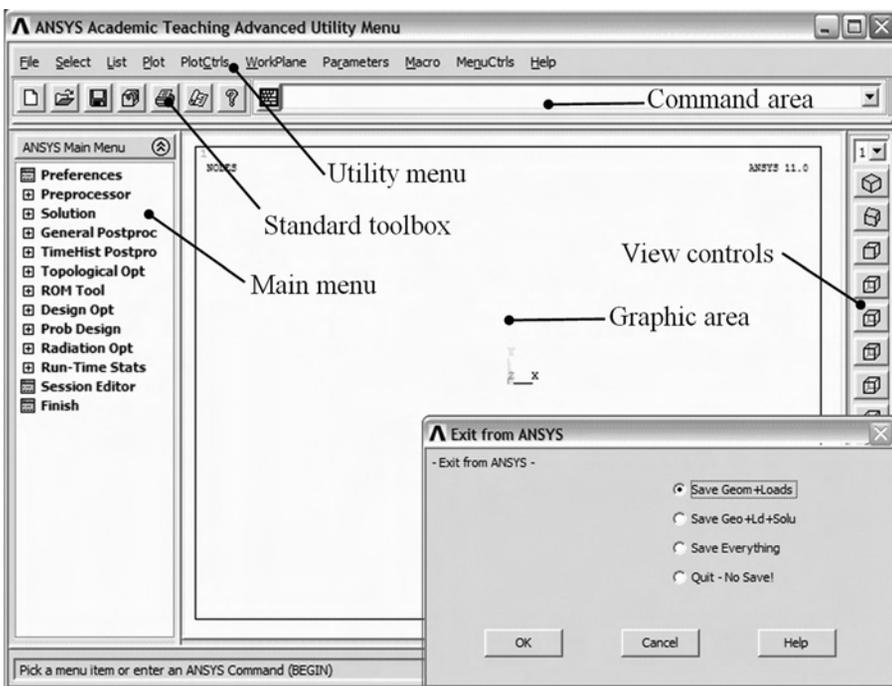

Figure C.1 Layout of ANSYS main window and exit window

C.3   STATIC ANALYSIS OF A CORNER BRACKET[2]

A.   Problem Description

This is a simple, single load step, structural static analysis of the corner angle bracket shown in Figure C.2. The upper left-hand pin hole is constrained (welded) around its entire circumference, and a tapered pressure load is applied to the bottom of the lower right-hand pin hole. The objective of the problem is to demonstrate the typical ANSYS analysis procedure. The US Customary system of units is used.

The dimensions of the corner bracket are shown in the figure. The bracket is made of A36 steel with a Young's modulus of 30E6 psi and Poisson's ratio of 0.27. Since the bracket is thin in the *z*-direction (1/2 inch thick) compared to its *x* and *y* dimensions, and since the pressure load acts only in the *x-y* plane, the problem can be assumed as plane stress. We will use solid modeling to generate the two-dimensional model and automatically mesh it with nodes and elements. (Another alternative in ANSYS is to create the nodes and elements directly.)

If your system includes a Flash player (Macromedia, Inc.), one can view demonstration videos of each step by pointing the web browser to the following URL address: http://www.ansys.com/techmedia/structural_tutorial_videos.html

B.   Building the Geometry

In order to perform the finite element analysis, we need to create the geometry, and elements, and to apply boundary conditions. This step is called pre-processing.

Step 1:   Define rectangles

There are several ways to create the model geometry within ANSYS, some more convenient than others. The first step is to recognize that you can construct the bracket easily with combinations of rectangle and circle primitives. Decide where the origin will be located and then define the rectangle and circle primitives relative to that origin. The location of the origin is arbitrary. Here, use the center of the upper left-hand hole. ANSYS does not need to know where the origin is. Simply begin by defining a rectangle relative to that location. In ANSYS, this origin is called the ***global origin***.
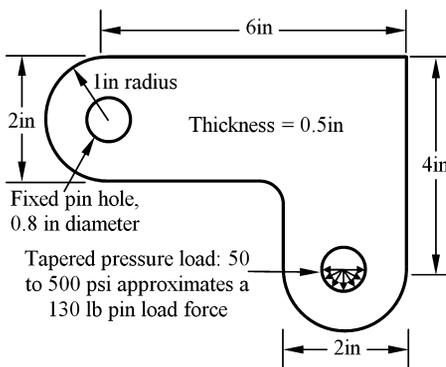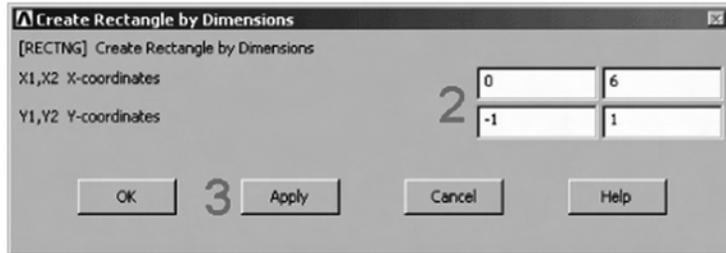

Figure C.2 Corner bracket stress analysis problem

---

[2] By the courtesy of ANSYS, Inc.

1. Choose **Main Menu> Preprocessor> Modeling> Create> Areas> Rectangle> By Dimensions**
2. Enter the following data to create the first rectangle: $X1 = 0$, $X2 = 6$, $Y1 = -1$, $Y2 = 1$ (Note: Press the Tab key between entries)
3. **Apply** to create the first rectangle.

4. Enter the following data to create the second rectangle: $X1 = 4$, $X2 = 6$, $Y1 = -1$, $Y2 = -3$
5. **OK** to create the second rectangle and close the dialog box.

Now the geometry should look like the figure below.

Step 2:   Change plot controls and replot

The area plot shows both rectangles, which are *areas*, in the same color. To more clearly distinguish between areas, turn on area numbers and colors. The ''Plot Numbering Controls'' dialog box on the Utility Menu controls how items are displayed in the Graphics Window. By default, a ''replot'' is automatically performed upon execution of the dialog box. The replot operation will repeat the last plotting operation that occurred (in this case, an area plot).
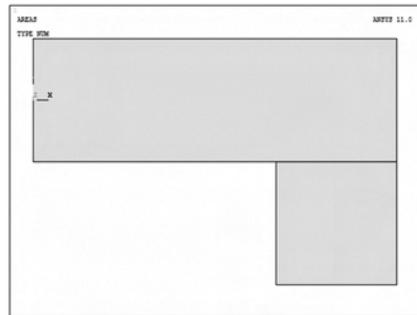
1. Choose **Utility Menu> Plot Ctrls> Numbering**
2. Turn on area numbers.
3. Choose **OK** to change controls, close the dialog box, and replot.
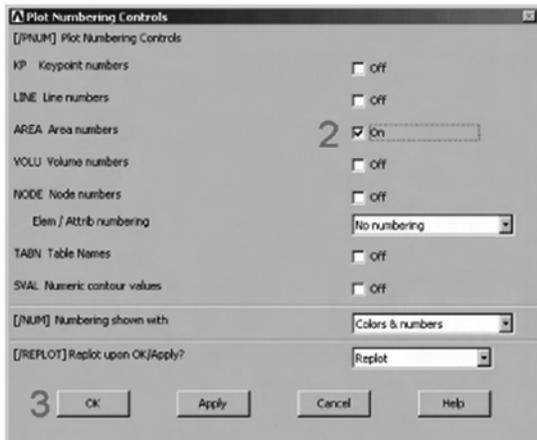
Before going to the next step, save the work you have done so far. ANSYS stores any input data in memory to the *ANSYS database*. To save that database to a file, use the SAVE operation, available as a tool on the Toolbar. ANSYS names the database file using the format *jobname.db*. You can check the current jobname at any time by choosing **Utility Menu> List> Status> Global Status**. You can also save the database at specific milestone points in the analysis (such as after the model is complete, or after the model is meshed) by choosing **Utility Menu> File> Save As** and specifying different jobnames (*model.db*, or *mesh.db*, etc.).

It is important to do an occasional save so that if you make a mistake, you can restore the model from the last saved state. You restore the model using the **RESUME** operation, also available on the Toolbar. (You can also find SAVE and RESUME on the Utility Menu, under File.)

**Step 3: Change working plane to polar and create first circle**

The next step in the model construction is to create the half circle at each end of the bracket. You will actually create a full circle on each end and then combine the circles and rectangles. To create the circles, you will use and display the working plane. You could have shown the working plane as you created the rectangles but it was not necessary.

Before you begin however, first ''zoom out'' within the Graphics Window so you can see more of the circles as you create them. You do this using the ''Pan-Zoom-Rotate'' dialog box, a convenient graphics control box you'll use often in any ANSYS session.

1. Choose **Utility Menu> PlotCtrls> Pan, Zoom, Rotate**
2. Click on small dot once to zoom out.
3. Close dialog box.
4. Choose **Utility Menu> WorkPlane> Display Working Plane** (toggle on)

Notice the working plane origin is immediately plotted in the Graphics Window. It is indicated by the WX and WY symbols; right now coincident with the global origin X and Y symbols. Next you will change the WP type to polar, change the snap increment, and display the grid.
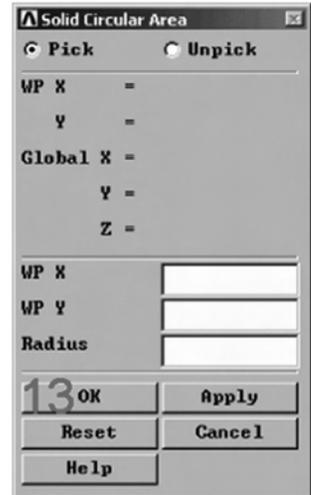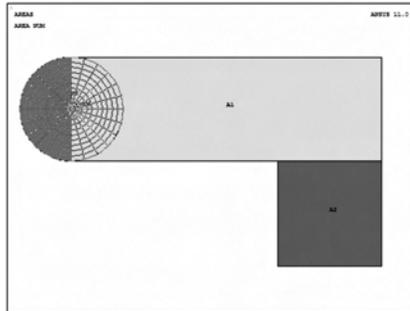
5. Choose **Utility Menu> WorkPlane> WP Settings**
6. Click on **Polar**.
7. Click on **Grid and Triad**.
8. Enter **.1** for snap increment.
9. **OK** to define settings and close the dialog box.
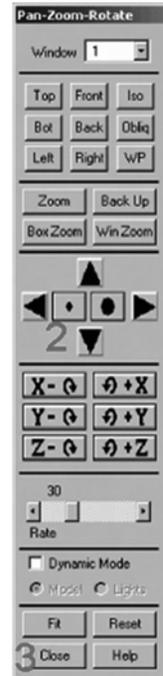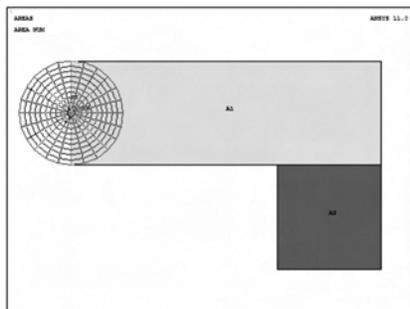
10. Choose **Main Menu**> **Preprocessor**> **Modeling**> **Create**> **Areas**> **Circle**> **Solid Circle**

   Be sure to read prompt before picking.

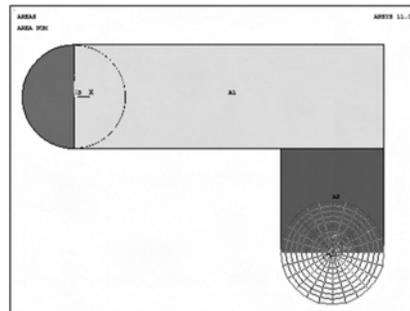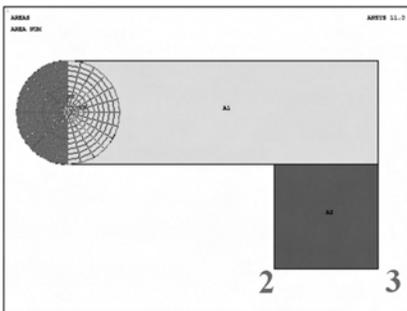11. Pick center point at: WP X = 0, WP Y = 0, Radius = 1 (in Graphics Window shown below)

12. Choose **OK** to close picking menu.
13. Toolbar: **SAVE_DB**.

Step 4:   Move working plane and create second circle

   To create the circle at the other end of the bracket in the same manner, you need to first move the working plane to the origin of the circle. The simplest way to do this without entering number off-sets is to move the WP to an average keypoint location by picking the keypoints at the bottom corners of the lower, right rectangle.

1. Choose **Utility Menu**> **WorkPlane**> **Offset WP to**> **Keypoints**
2. Pick keypoint at lower left corner of rectangle.
3. Pick keypoint at lower right of rectangle.


4. OK to close picking menu.
5. Choose **Main Menu**> **Preprocessor**> **Modeling**> **Create**> **Areas**> **Circle**> **Solid Circle**
6. Pick center point at: WP X = 0, WP Y = 0, Radius = 1
7. OK to close picking menu.
8. Toolbar: SAVE_DB.
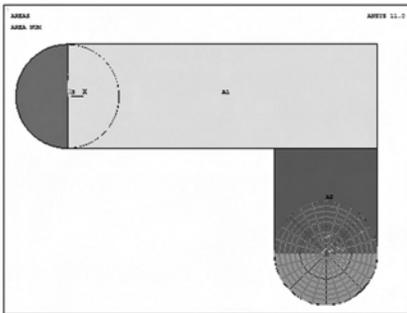
Step 5: Add areas.

Now that the appropriate pieces of the model are defined (rectangles and circles), you need to add them together so the model becomes one continuous piece. You do this with the Boolean add operation for areas.

1. Choose **Main Menu> Preprocessor> Modeling> Operate> Booleans> Add> Areas**
2. Choose **Pick All** for all areas to be added.
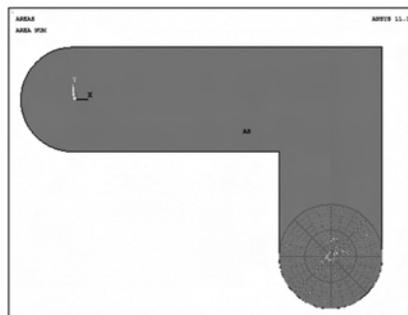
3. Toolbar: SAVE_DB.

Step 6:   Create line fillet.

1. Choose **Utility Menu> PlotCtrls> Numbering**
2. Turn on line numbering.
3. OK to change controls, close the dialog box, and automatically replot.
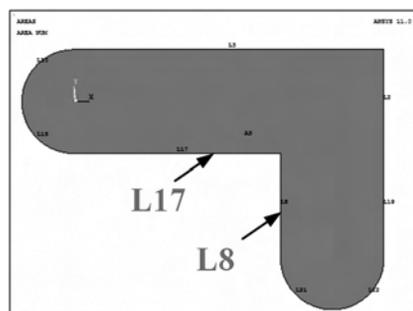
4. Choose **Utility Menu> WorkPlane> Display Working Plane** (toggle off)
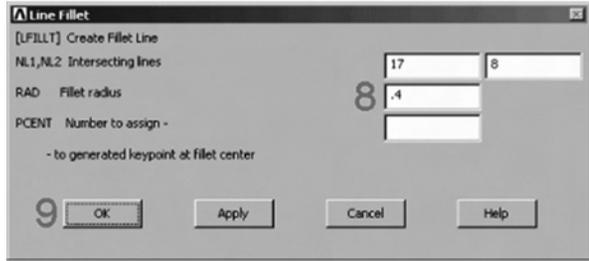
5. Choose **Main Menu> Preprocessor> Modeling> Create> Lines> Line Fillet**

6. Pick lines 17 and 8.
7. OK to finish picking lines (in picking menu).
8. Enter 0.4 as the radius.
9. OK to create line fillet and close the dialog box.
10. Choose **Utility Menu> Plot> Lines**

Step 7: Create fillet area.

1. Choose **Utility Menu> PlotCtrls> Pan, Zoom, Rotate**
2. Click on Zoom button.
3. Move mouse to fillet region, click left button, move mouse out and click again.
4. Choose **Main Menu> Preprocessor> Modeling> Create> Areas> Arbitrary> By Lines**
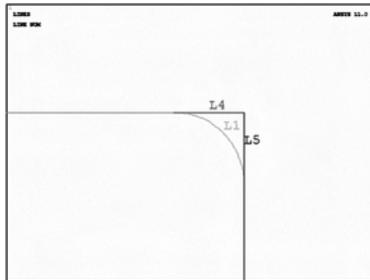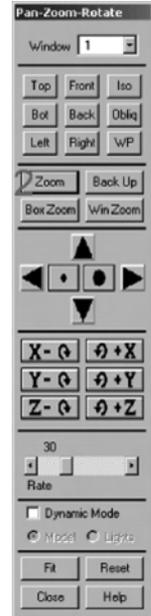
5. Pick lines 4, 5, and 1.
6. OK to create area and close the picking menu.
7. Click on Fit button in Pan, Zoom, Rotate dialog box.
8. Close the Pan, Zoom, Rotate dialog box.
9. Choose **Utility Menu> Plot> Areas**
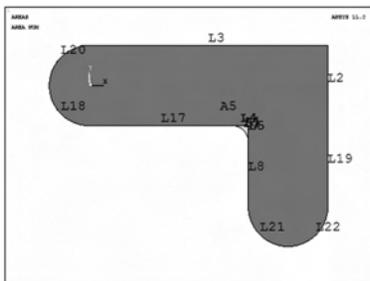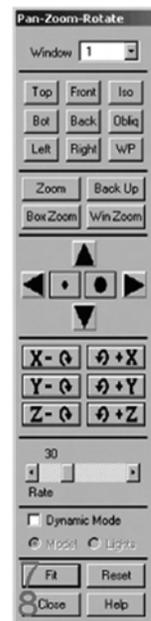
10. Toolbar: SAVE_DB.

Step 8:   Add areas together.

1. Choose **Main Menu> Preprocessor> Modeling> Operate> Booleans> Add> Areas**
2. Pick All for all areas to be added.
3. Toolbar: SAVE_DB.

Step 9:   Create first pin hole.

1. Choose **Utility Menu> WorkPlane> Display Working Plane** (toggle on)
2. Choose **Main Menu> Preprocessor> Modeling> Create> Areas> Circle> Solid Circle**
3. Pick center point at: WP X = 0, WP Y = 0, Radius = 0.4
4. OK to close picking menu.
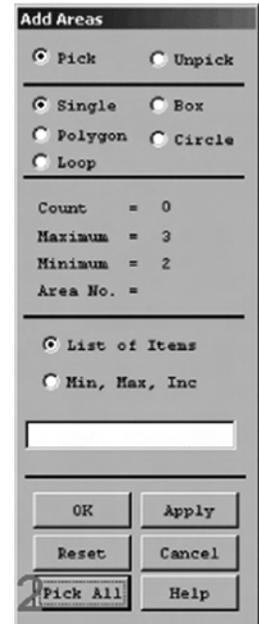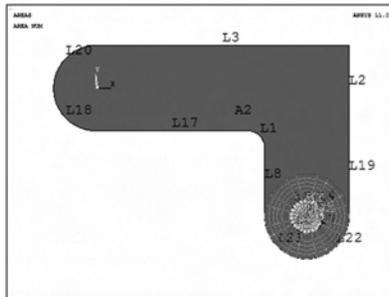
Step 10:   Move working plane and create second pin hole.

1. Choose **Utility Menu> WorkPlane> Offset WP to> Global Origin**
2. Choose **Main Menu> Preprocessor> Modeling> Create> Areas> Circle> Solid Circle**
3. Pick center point at: WP X = 0, WP Y = 0, Radius = 0.4 (in Graphics Window)
4. OK to close picking menu.
5. **Utility Menu> WorkPlane> Display Working Plane** (toggle off)
6. **Utility Menu> Plot> Lines**

If you plot areas, it appears that one of the pin hole areas is not there. However, it is there (as indicated by the presence of its lines), you just can't see it in the final display of the screen. That is because the bracket area is drawn on top of it. An easy way to see all areas is to plot the lines instead.

7. Toolbar: SAVE_DB.
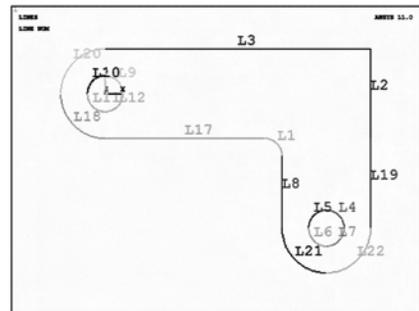
Step 11:   Subtract pin holes from bracket.

1. Choose **Main Menu> Preprocessor> Modeling> Operate> Booleans> Subtract> Areas**
2. Pick bracket as base area from which to subtract.

3. Apply (in picking menu).
4. Pick both pin holes as areas to be subtracted.
5. OK to subtract holes and close picking menu.
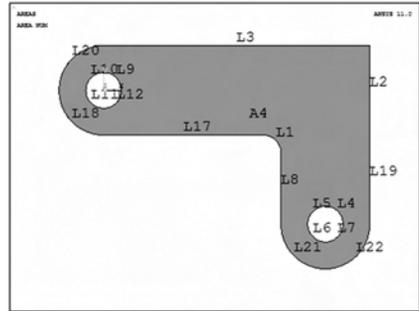
Step 12: Save the database as model.db.

At this point, you will save the database to a named file—a name that represents the model before meshing. If you decide to go back and remesh, you'll need to resume this database file. You will save it as model.db.

1. Choose **Utility Menu> File> Save As**
2. Enter model.db for the database file name.
3. OK to save and close dialog box.

C. Define Materials

Step 13: Set preferences.

In preparation for defining materials, you will set preferences so that only materials that pertain to a structural analysis are available for you to choose.

To set preferences:

1. Choose **Main Menu> Preferences**
2. Turn on structural filtering. The options may differ from what is shown here since they depend on the ANSYS product you are using.
3. OK to apply filtering and close the dialog box.
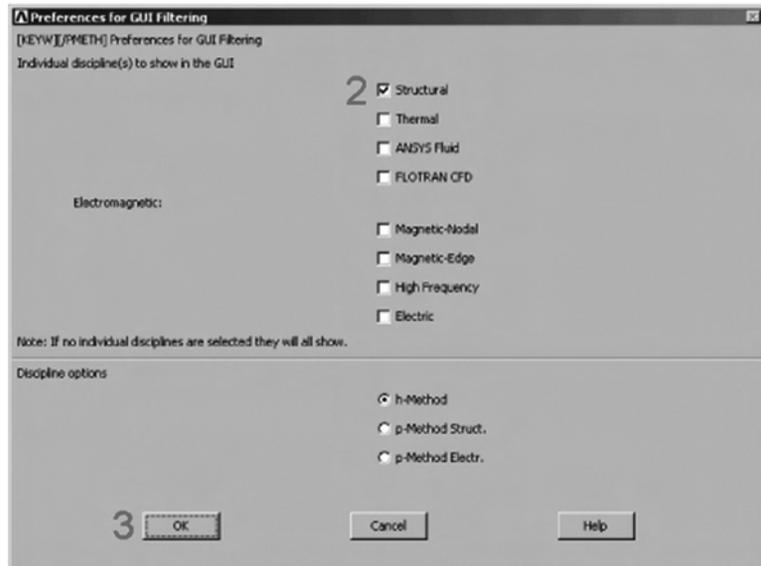
Step 14: Define material properties.

To define material properties for this analysis, there is only one material for the bracket, A36 Steel, with given values for Young's modulus of elasticity and Poisson's ratio.

1. Choose **Main Menu**> **Preprocessor**> **Material Props**> **Material Models**
2. Double-click on Structural, Linear, Elastic, Isotropic.
3. Enter 30e6 for EX and 0.27 for PRXY.
4. OK to define material property set and close the dialog box.
5. In the Define Material Model window, choose **Material**> **Exit**
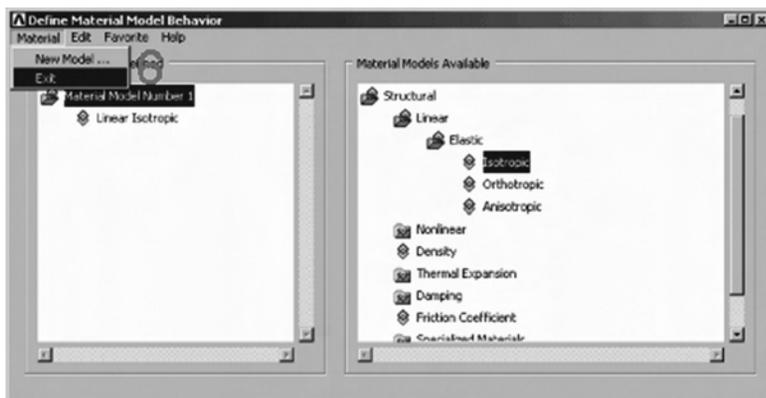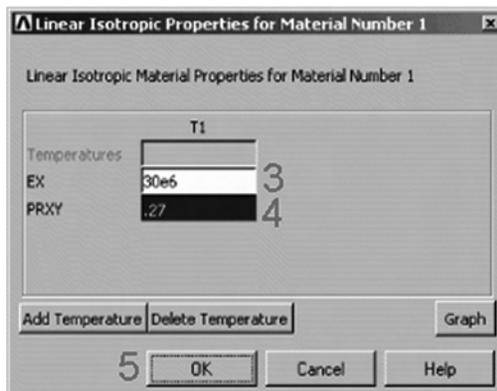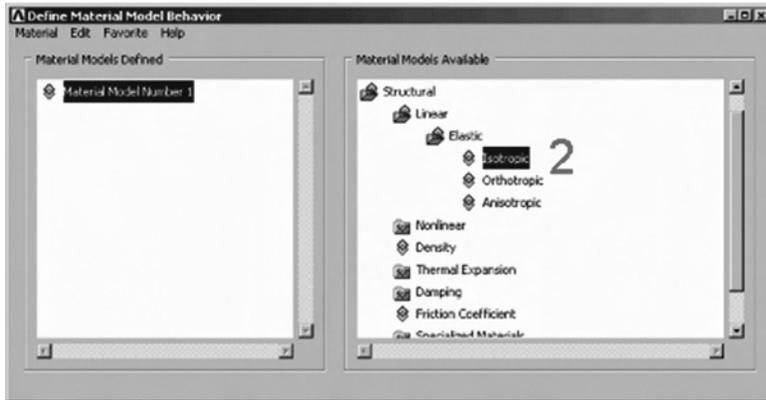
Step 15:   Define element types and options.

In any analysis, you need to select from a library of element types and define the appropriate ones for your analysis. For this analysis, you will use only one element type, **PLANE82**, which is a 2-D, quadratic, structural, higher-order element. The choice of a

higher-order element here allows you to have a coarser mesh than with lower-order elements while still maintaining solution accuracy. Also, ANSYS will generate some triangle shaped elements in the mesh that would otherwise be inaccurate if you used lower-order elements (**PLANE42**). You will need to specify plane stress with thickness as an option for **PLANE82**. (You will define the thickness as a real constant in the next step.)

1. Choose **Main Menu**> **Preprocessor**> **Element Type**> **Add/Edit/Delete**
2. Choose **Add**. . . button in Element Types window.
3. In Library of Element Types window, Choose **Solid**.
4. Choose **8node 82**, which is the 8-node quad element type (**PLANE82**).
5. OK to apply the element type and close the dialog box.
6. Choose **Options**. . . button in Element Types window.
7. Choose plane stress with thickness option for element behavior.
8. OK to specify options and close the options dialog box.
9. Close the element type dialog box.
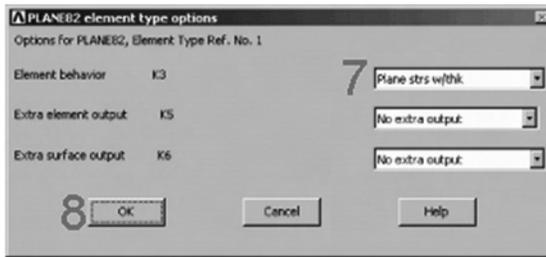
Step 16:   Define real constants.

For this analysis, since the assumption is plane stress with thickness, you will enter the thickness as a real constant for **PLANE82**. To find out more information about **PLANE82**, you will use the ANSYS Help System in this step by clicking on a Help button from within a dialog box.

1. Choose **Main Menu> Preprocessor> Real Constants> Add/Edit/Delete**
2. Choose **Add**. . . button for a real constant set.
3. OK for **PLANE82**.
4. Help to get help on **PLANE82**.
5. Hold left mouse button down to scroll through element description.
6. Close the Help window
7. Enter .5 for THK.
8. OK to define the real constant and close the dialog box.
9. Close the real constant dialog box.

D.   Generate Mesh

Step 17:   Mesh the area.

One nice feature of the ANSYS program is that you can automatically mesh the model without specifying any mesh size controls. This is using what is called a **default mesh**. If you're not sure how to determine the mesh density, let ANSYS try it first! Meshing this model with a default mesh, however, generates more elements than are allowed in the

ANSYS ED program. Instead you will specify a global element size to control overall mesh density.

1. Choose **Main Menu> Preprocessor> Meshing> Mesh Tool**
2. Set Global Size control.
3. Type in 0.5.
4. OK.
5. Choose Area Meshing.
6. Click on Mesh.
7. Pick All for the area to be meshed (in picking menu). Close any warning messages that appear.
8. Close the Mesh Tool.
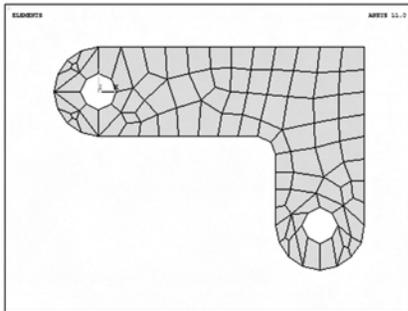
The mesh you see on your screen may vary slightly from the mesh shown here. As a result of this, you may see slightly different results during postprocessing.

Step 18:   Save the database as mesh.db.

Here again, you will save the database to a named file, this time mesh.db.

1. Choose **Utility Menu> File> Save as**
2. Enter mesh.db for database file name.
3. OK to save file and close dialog box.

E.   Apply Loads

Now, we finish preprocessing phase and move on to the solution phase. A new, static analysis is the default, so you will not need to specify analysis type for this problem. Also, there are no analysis options for this problem.

Step 19:   Apply displacement constraints.

You can apply displacement constraints directly to lines.

1. Choose **Main Menu> Solution> Define Loads> Apply> Structural> Displacement> On Lines**

2. Pick the four lines around left-hand hole (Line numbers 10, 9, 11, 12).
3. OK (in picking menu).
4. Click on All DOF.
5. Enter 0 for zero displacement.
6. OK to apply constraints and close dialog box.
7. Choose **Utility Menu> Plot> Lines**

Choose 4 lines
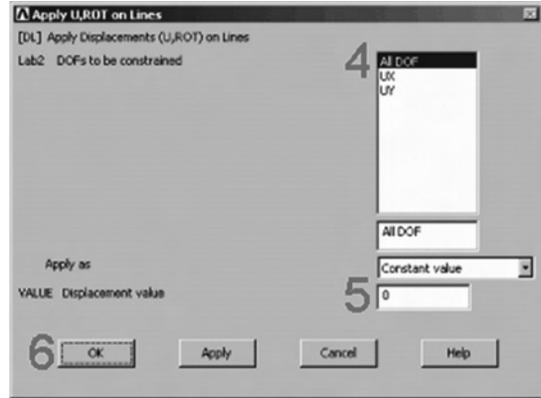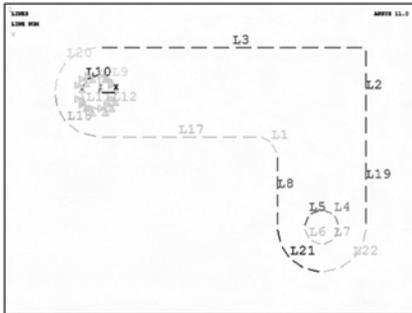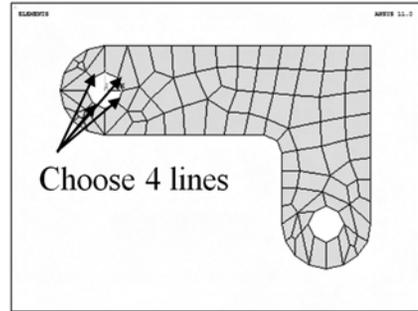
8. Toolbar: SAVE_DB.

Step 20:   Apply pressure load.

Now apply the tapered pressure load to the bottom, right-hand pin hole. (''Tapered'' here means varying linearly.) Note that when a circle is created in ANSYS, four lines define the perimeter. Therefore, apply the pressure to two lines making up the lower half of the circle. Since the pressure tapers from a maximum value (500 psi) at the bottom of the circle to a minimum value (50 psi) at the sides, apply pressure in two separate steps, with reverse tapering values for each line. The ANSYS convention for pressure loading is that a positive load value represents pressure into the surface (compressive).

1. Choose **Main Menu> Solution> Define Loads> Apply> Structural> Pressure> On Lines**
2. Pick line defining bottom left part of the circle (line 6).
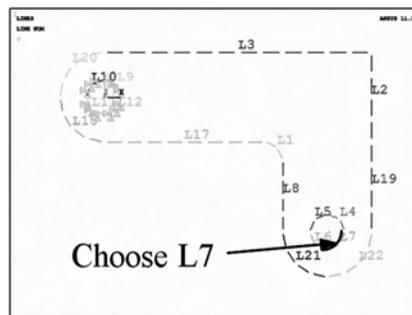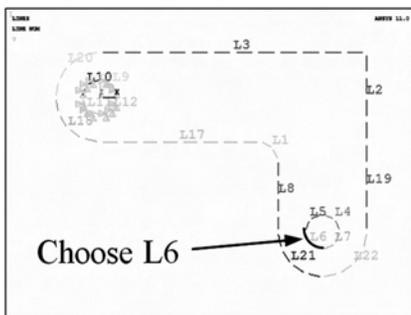
Choose L6

Choose L7

3. Apply.
4. Enter 50 for VALUE.

5. Enter 500 for optional value.
6. Apply.
7. Pick line defining bottom right part of circle (line 7).
8. Apply.
9. Enter 500 for VALUE.
10. Enter 50 for optional value.
11. OK.
12. Toolbar: SAVE_DB.
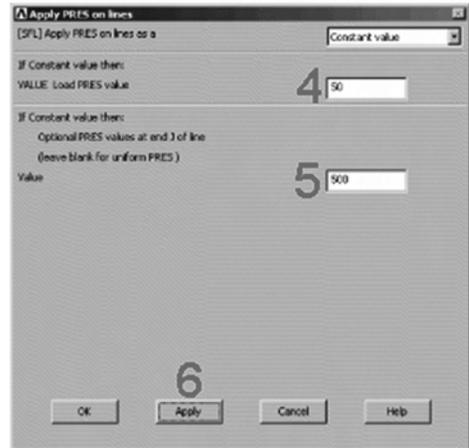
F. Obtain Solution

Step 21: Solve.

1. Choose **Main Menu> Solution> Solve> Current LS**
2. Review the information in the status window, then choose **File> Close** (Windows), or Close (X11/Motif), to close the window.
3. OK to begin the solution. Choose Yes to any Verify messages that appear.
4. Close the information window when solution is done.
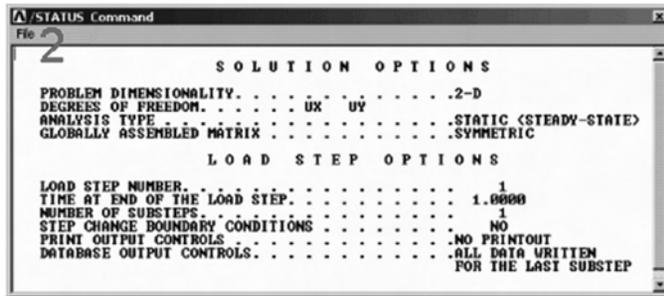
ANSYS stores the results of this one load step problem in the database and in the results file, Jobname.RST (or Jobname.RTH for thermal, Jobname.RMG for magnetic, and Jobname.RFL for fluid analyses). The database can actually contain only one set of results at any given time, so in a multiple load step or multiple substep analysis, ANSYS stores only the final solution in the database. ANSYS stores all solutions in the results file.

G.   Review Results

Now, we finish the solution phase and move to the postprocessing phase. The results you see may vary slightly from what is shown here due to variations in the mesh.

Step 22:   Enter the general postprocessor and read in the results.

  1. Choose **Main Menu> General Postproc> Read Results> First Set**

Step 23:   Plot the deformed shape.

  1. Choose **Main Menu> General Post-proc> Plot Results> Deformed Shape**
  2. Choose Def + undeformed.
  3. OK. You can also produce an animated version of the deformed shape:
  4. Choose **Utility Menu> Plot Ctrls> Animate> Deformed Shape**
  5. Choose Def + undeformed.
  6. OK to start animation.
  7. Make choices in the Animation Controller, if necessary, then choose Close.
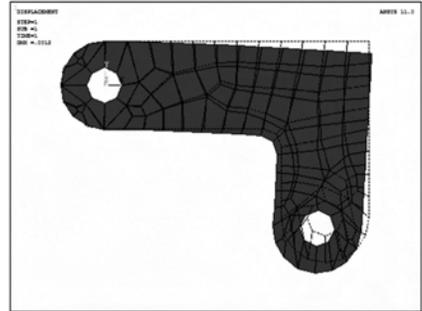
Step 24:   Plot the von Mises equivalent stress.

  1. Choose **Main Menu> General Postproc> Plot Results> Contour Plot> Nodal Solu**
  2. Choose **Stress** item to be contoured.
  3. Scroll down and choose **von Mises stress**.
  4. OK.
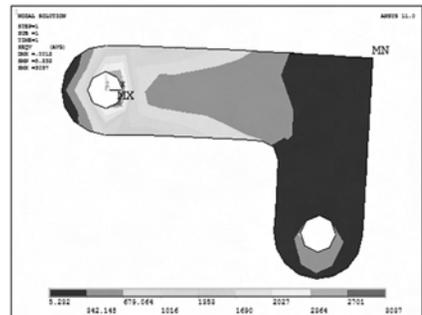
The colors in the contour plot stand for the level of von Mises stress. The maximum stress occurs at the fixed pin hole. The value of stress can be read from the color legend in the bottom of the graphic screen. Note that the color legend can be shown on the right side of graphic screen. You can also produce an animated version of these results:

  5. Choose **Utility Menu> Plot Ctrls> Animate> Deformed Results**

6. Choose **Stress** item to be contoured.
7. Scroll down and choose **von Mises SEQV**.
8. OK.
9. Make choices in the Animation Controller, if necessary, then choose Close.

Step 25: List reaction solution.

1. Choose **Main Menu> General Postproc> List Results> Reaction Solu**
2. OK to list all items and close the dialog box.
3. Scroll down and find the total vertical force, FY.
4. Choose **File> Close** (Windows), or Close (X11/Motif), to close the window.

The value of 134.61 is comparable to the total pin load force. Note that the values shown are representative and may vary from the values you obtain. There are many other options available for reviewing results in the general postprocessor. You have finished the analysis. Exit the program in the next step.

Step 26: Exit the ANSYS program.

When exiting the ANSYS program, you can save the geometry and loads portions of the database (default), save geometry, loads, and solution data (one set of results only), save geometry, loads, solution data, and postprocessing data (i.e., save everything), or save nothing. You can save nothing here, but you should be sure to use one of the other save options if you want to keep the ANSYS data files.

1. Toolbar: Quit.
2. Choose Quit - No Save!
3. OK.

C.4 EXAMPLES IN THE TEXT

The tutorial in the previous section was performed using graphical user interface (GUI) in which the user chooses commands using the menu. Then the GUI interprets these inputs into commands and sends them to ANSYS program. However, it is possible to type in ANSYS commands directly in the Command area. You can prepare commands in text editor and copy and paste them to the Command area. Or, you can save commands as a file and read them using **Utility Menu> File> Read Input From**... In this section, we present various example problems in the text using the command mode.

Example 2.4 – Two–bar Truss Structure

The two–bar truss shown in Figure C.3 has circular cross-sections with diameter of 0.25cm and Young's modulus $E = 30 \times 10^6 \, \text{N/cm}^2$. An external force $F = 50 \, \text{N}$ is applied in the horizontal direction at Node 2. Calculate the displacement of each node and stress in each element. The commands that can perform the finite element analysis is listed in Table C.1.

The detailed explanation of each command can be found by typing `HELP, COMMAND` in the command area. In ANSYS command lines, any text after ''!'' is considered as a comment; i.e., ignored by the program. The above commands are composed of three phases: preprocessing (`/PREP7`), solution (`/SOLUTION`), and postprocessing (`/POST1`). Each phase is ended using `FINISH` command. In this simple example, we generate nodes

Figure C.3 Two–bar truss

Table C.1 ANSYS Commands for Two–bar Truss Analysis

```
/PREP7                        ! Start preprocessing
ET,1,LINK1                    ! Element type 1 = 2D Truss element
MP,EX,1,3E7                   ! Elastic modulus of material set 1
R,1,3.1416*0.125**2          ! Cross-sectional area of
!                                real constant set 1
!
N,1,0,0                       ! Define Node 1 (0, 0)
N,2,12,8                      ! Define Node 2 (12, 8)
N,3,12,0                      ! Define Node 3 (12, 0)
!
TYPE,1                        ! Use Element type 1
MAT,1                         ! Use material set 1
REAL,1                        ! Use real constant set 1
E,1,2                         ! Create Element 1 (N1 - N2)
E,2,3                         ! Create Element 2 (N2 - N3)
!
D,1,ALL,0                     ! Fix all DOFs of Node 1
D,3,ALL,0                     ! Fix all DOFs of Node 3
F,2,FX,50                     ! Apply Fx = 50N at Node 2
FINISH                        ! Finish preprocessing
!
!
/SOLUTION                     ! Start simulation
OUTRES,ALL
OUTPR,ALL
SOLVE                         ! Solve the matrix equation
FINISH                        ! Finish simulation
!
/POST1                        ! Start postprocessing
SET,FIRST                     ! Choose the first result set
PLDISP,1                      ! Plot the deformed geometry
PRDISP                        ! Print out the nodal displacement
PRESOL,ELEM                   ! Print out all element results
FINISH                        ! Finish postprocessing
```

and elements directly (N and E commands). Displacement boundary conditions are imposed using D command, and the nodal force is applied using F command. The output from the PRDISP command is shown below:

```
  NODE           UX                    UY
     1          0.0000                0.0000
     2       0.82803E-03          -0.18108E-03
     3          0.0000                0.000
```

Note that the nodal displacements are the same with Example 2.4 in Chapter 2. The output from PRESOL command is shown below:

```
   EL= 1      NODES= 1    2              MAT= 1  LINK1
   MFORX=     60.093
   SAXL=      1224.2       EPELAXL= 0.000041


   EL= 2      NODES= 2    3              MAT= 1  LINK1
   MFORX=     -33.333
   SAXL=      -679.06      EPELAXL=-0.000023
```

MFORX, SAXL, and EPELAXL are, respectively, element force, axial stress, and axial strain. Note that Element 1 is in tension, while Element 2 is in compression. The output from PLDISP command is shown below:
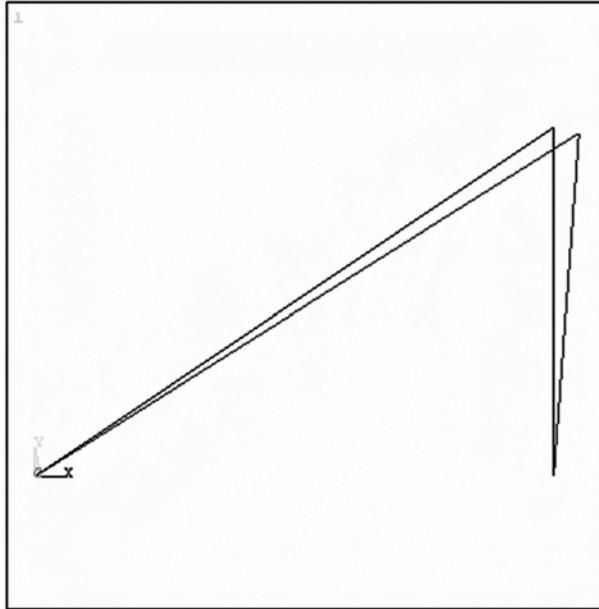
In general, the displacements are relatively small. Thus, most graphical postprocessors amplify the displacement so that the deformed shape of the truss can be shown in the figure.

Example 2.5 – Plane Truss with Three Elements

The plane truss shown in Figure C.4 consists of three members connected to each other and to the walls by pin joints. The members make equal angles with each other and element 2 is vertical. The members are identical to each other with properties: Young's modulus $E = 206 \times 10^9$ Pa, cross-sectional area $A = 1 \times 10^{-4}$ m^2, and length $L = 1$ m. An inclined force $F = 20,000$ N is applied at node 1. Solve for the displacements at node 1 and stresses in the three elements.

One of powerful capabilities in ANSYS is parametric programming. The user can assign a value to a variable and use it as an argument in the following commands. For example, we define a variable val $= 1/\text{sqrt}(2)$, and then, use it in the definition of
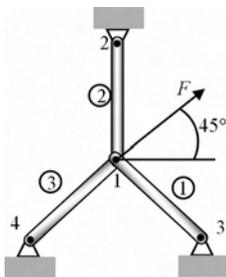
Figure C.4 Plane structure with three truss elements

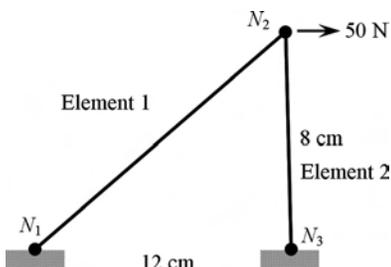**Table C.2**   ANSYS Commands for Three Truss Elements Analysis

```
/PREP7                          ! Start preprocessing
ET,1,LINK1                      ! Element type 1 = 2D Truss element
MP,EX,1,206E9                   ! Elastic modulus of material set 1
R,1,0.0001                      ! Cross-sectional area of
!                                  real constant set 1
!
ang=3.14159/6
N,1,0,0                         ! Define nodal coordinates
N,2,0,1
N,3,cos(ang),-sin(ang)
N,4,-cos(ang),-sin(ang)
!
TYPE,1                          ! Use Element type 1
MAT,1                           ! Use material set 1
REAL,1                          ! Use real constant set 1
E,1,3                           ! Create Element 1
E,1,2                           ! Create Element 2
E,1,4                           ! Create Element 3
!
D,2,ALL,0                       ! Fix all DOFs of Node 2
D,3,ALL,0                       ! Fix all DOFs of Node 3
D,4,ALL,0                       ! Fix all DOFs of Node 4
F,1,FX,20000/sqrt(2)            ! Apply force at Node 2
F,1,Fy,20000/sqrt(2)            ! Apply force at Node 2
FINISH                          ! Finish preprocessing
!
/SOLUTION                       ! Start simulation
OUTRES,ALL
OUTPR,ALL
SOLVE                           ! Solve the matrix equation
FINISH                          ! Finish simulation
!
/POST1                          ! Start postprocessing
SET,FIRST                       ! Choose the first result set
PLDISP,1                        ! Plot the deformed truss geometry
PRDISP                          ! Print out the nodal displacement
PRESOL,ELEM                     ! Print out all element results
FINISH                          ! Finish postprocessing
```

nodal coordinates, as `N,3,val,−val`. The variable can be an array or matrix. Table C.2 shows the list of ANSYS commands for plane truss. The output from the `PRDISP` command is shown below:

```
NODE              UX                    UY
  1           0.45767E-03           0.45767E-03
  2             0.0000                0.0000
  3             0.0000                0.0000
  4             0.0000                0.0000
```

Note that the nodal displacements are the same as in Example 2.5 in Chapter 2. The output from `PRESOL` command is shown below:

```
EL= 1 NODES=      1            3 MAT=        1
MFORX= -3450.9
SAXL=-0.34509E+08    EPELAXL=-0.000168

EL= 2 NODES=      1            2 MAT=        1
MFORX= -9428.1
SAXL=-0.94281E+08 EPELAXL=-0.000458

EL= 3 NODES=      1            4 MAT=        1
MFORX= 12879.
SAXL= 0.12879E+09 EPELAXL= 0.000625
```

The output from PLDISP command is shown in Figure C.5.

Example 2.6 – Space Truss Problem

Use the finite element method to determine the displacements and forces in the space truss, shown in Figure C.6. The coordinates of the nodes in meter units are given in the
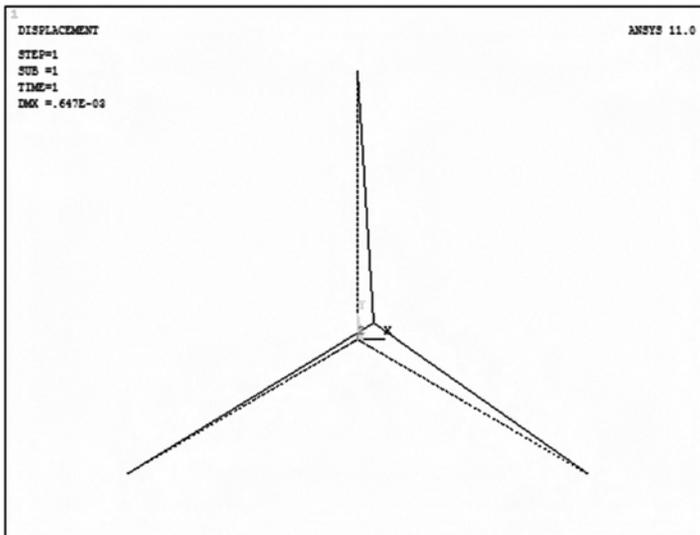
Figure C.5 Deformed shape of the plane structure with three truss elements
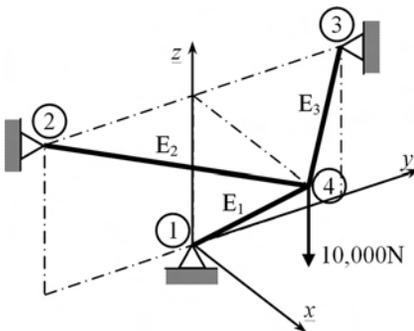
Figure C.6 Three–bar space truss structure

table. Assume Young's modulus $E = 70\,GPa$ and area of cross section $A = 1\,cm^2$. The magnitude of the downward force at Node 4 is equal to 10,000 N.

Node	x	y	z
1	0	0	0
2	0	−1	1
3	0	1	1
4	1	0	1

Table C.3   ANSYS Commands for Space Truss Problem

```
/PREP7                    ! Start preprocessing
ET,1,LINK8                ! Element type 1 = 3D Truss element
MP,EX,1,70E9              ! Elastic modulus of material set 1
R,1,0.0001               ! Cross-sectional area of
!                            real constant set 1
!
Val=1/sqrt(2)
N,1,0,0,0                ! Define Node 1
N,2,0,-1,1               ! Define Node 2
N,3,0,1,1                ! Define Node 3
N,4,1,0,1                ! Define Node 4
!
TYPE,1                    ! Use Element type 1
MAT,1                     ! Use material set 1
REAL,1                    ! Use real constant set 1
E,1,4                     ! Create Element 1
E,2,4                     ! Create Element 2
E,3,4                     ! Create Element 3
!
D,1,ALL,0                 ! Fix all DOFs of Node 1
D,2,ALL,0                 ! Fix all DOFs of Node 2
D,3,ALL,0                 ! Fix all DOFs of Node 3
F,4,Fz,-10000             ! Apply Fz = -10000N at Node 4
FINISH                    ! Finish preprocessing
!
/SOLUTION                 ! Start simulation
OUTRES,ALL
OUTPR,ALL
SOLVE                     ! Solve the matrix equation
FINISH                    ! Finish simulation
!
/POST1                    ! Start postprocessing
SET,FIRST                 ! Choose the first result set
/VIEW,1,1,2,3             ! Set isometric view
PLDISP,1                  ! Plot the deformed truss geometry
PRDISP                    ! Print out the nodal displacement
PRESOL,ELEM               ! Print out all element results
FINISH                    ! Finish postprocessing
```

Table C.3 shows the list of ANSYS Commands for space truss. Note that different element type, LINK8, is used for three-dimensional truss. The list of available elements can be found in ANSYS Help. /VIEW command controls the viewpoint direction. ANSYS provides various view controls including dynamic view. The output from the PRDISP command is shown below:

NODE	UX	UY	UZ
1	0.0000	0.0000	0.0000
2	0.0000	0.0000	0.0000
3	0.0000	0.0000	0.0000
4	0.20203E-02	0.0000	-0.60609E-02

Note that the nodal displacements are the same as in Example 2.6 in Chapter 2. The output from PRESOL command is shown below:

```
EL=      1 NODES=   1     4 MAT= 1    LINK8
MFORX= -14142.
SAXL=-0.14142E+09 EPELAXL=-0.002020


EL=      2 NODES=   2     4 MAT= 1    LINK8
MFORX=  7071.1
SAXL= 0.70711E+08 EPELAXL= 0.001010


EL=      3 NODES=   3     4 MAT= 1    LINK8
MFORX=  7071.1
SAXL= 0.70711E+08 EPELAXL= 0.001010
```

The figure below shows the deformed shape of the space truss.
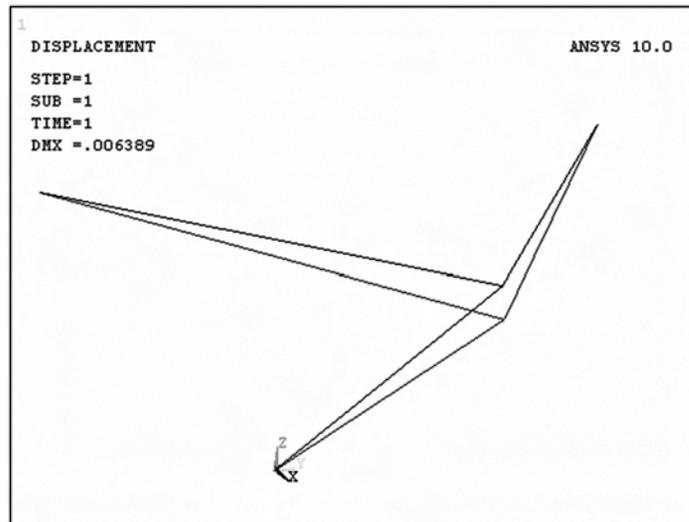
Simply Supported Beam Problem

Consider a simply supported beam, as shown in Figure C.7, loaded by a single load $F = 10$ kN, applied at a point 3.0 m from the left support. Plot the deflection curve, shear

Figure C.7 Simply supported beam

Table C.4   ANSYS Commands for Space Truss Problem

```
/PREP7                       ! Start preprocessing
ET,1,BEAM3                   ! Element type = 2D Beam element
MP,EX,1,210E9                ! Elastic modulus
R,1,4.53E-3,2.51E-5,0.1      ! Area and Izz
!
N,1,0,0                      ! Define Node 1
N,2,3,0                      ! Define Node 2
N,3,6,0                      ! Define Node 3
N,4,9,0                      ! Define Node 4
!
TYPE,1                       ! Use Element type 1
MAT,1                        ! Use material set 1
REAL,1                       ! Use real constant set 1
E,1,2                        ! Create Element 1
E,2,3                        ! Create Element 2
E,3,4                        ! Create Element 3
!
D,1,UX,0,,,,Uy               ! Fix x- & y-displacement of Node 2
D,4,Uy,0                     ! Fix y-displacement of Node 4
F,2,Fy,-10000                ! Apply Fy = -10000N at Node 2
FINISH                       ! Finish preprocessing
!
/SOLUTION                    ! Start simulation
OUTRES,ALL
OUTPR,ALL
SOLVE                        ! Solve the matrix equation
FINISH                       ! Finish simulation
!
/POST1                       ! Start postprocessing
SET,FIRST                    ! Choose the first result set
PLDISP,1                     ! Plot the deformed geometry
PRDISP                       ! Print out the nodal displacement
!
ETABLE,MzzI,SMISC,6          ! Bending moment at Node I
ETABLE,MzzJ,SMISC,12         ! Bending moment at Node J
ETABLE,VyI,SMISC,2           ! Shear force at Node I
ETABLE,VyJ,SMISC,8           ! Shear force at Node J
PRETAB,MzzI,MzzJ,VyI,VyJ     ! Print results
PLLS,MzzI,MzzJ               ! Plot bending moment diagram
PLLS,VyI,VyJ                 ! Plot shear force diagram
FINISH                       ! Finish postprocessing
```

force and bending moment diagrams using three equal–length beam elements. The following data apply to the beam: Young's modulus $E = 210\,\text{GPa}$, cross-sectional area $A = 4.53 \times 10^{-3}\,\text{m}^2$, and moment of inertia $I = 2.51 \times 10^{-5}\,\text{m}^4$.
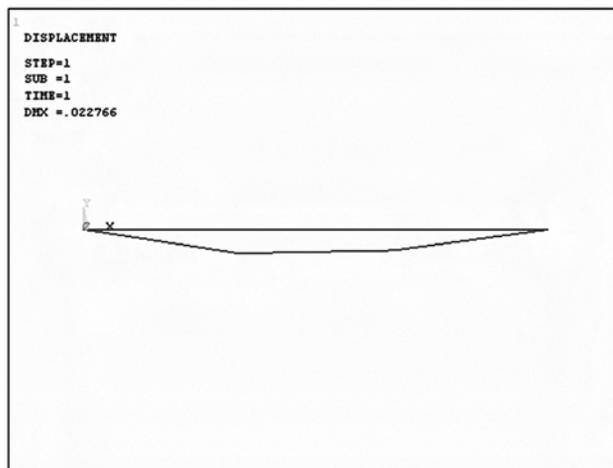
Table C.4 shows the list of ANSYS commands for simply supported beam. For beam analysis, element type BEAM3 is used, which is 2-D beam element. For the beam analysis, both cross-sectional area and the moment of inertia are required as real constants. In order to plot the shear force and bending moment diagram, ETABLE command is used, which defines a table of values per element (the element table) for use in further processing. The element table is organized as a ''worksheet,'' with the rows representing all selected elements, and the columns consisting of result items which have been moved into the table with ETABLE. Each column of data is identified by a user-defined label for listings and displays. PRETAB command prints the element table, and PLLS plots the data in the element table. The output from the PRDISP command is shown below:

NODE	UX	UY	ROTZ
1	0.0000	0.0000	−0.94859E−02
2	0.0000	−0.22766E−01	−0.37943E−02
3	0.0000	−0.19920E−01	0.47429E−02
4	0.0000	0.0000	0.75887E−02

The output from PRETAB command is shown below:

ELEM	MZZI	MZZJ	VYI	VYJ
1	−0.54570E−11	20000.	−6666.7	−6666.7
2	20000.	10000.	3333.3	3333.3
3	10000.	−0.14552E−10	3333.3	3333.3

The above table shows bending moments at the first node (MZZI) and at the second node (MZZJ), as well as the shear force at the first node (VYI) and at the second node (VYJ). The figure below shows the deformed shape of the beam.

DISPLACEMENT
STEP=1
SUB =1
TIME=1
DMX =.022766

The bending moment and shear force diagrams are shown below:

LINE STRESS

STEP=1
SUB =1
TIME=1
SMIS6   SMIS12
MIN =-.146E-10
ELEM=3
MAX =20000
ELEM=1

-.146E-10   4444   8889   13333   17778
        2222   6667   11111   15556   20000

LINE STRESS

STEP=1
SUB =1
TIME=1
SMIS2   SMIS8
MIN =-6667
ELEM=1
MAX =3333
ELEM=3

-6667   -4444   -2222   -.909E-12   2222
    -5556   -3333   -1111   1111   3333