

[Tutorials](#) | [Chapter 2. Structural Tutorial](#) |

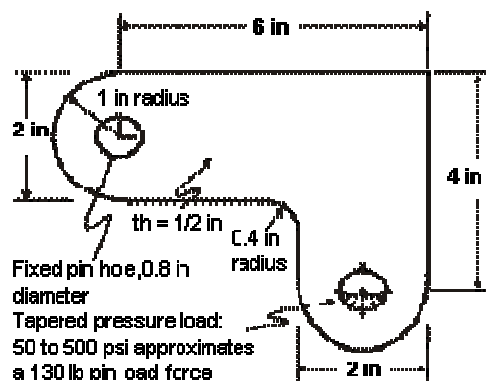
2.1. Static Analysis of a Corner Bracket

2.1.1. Problem Specification

Applicable ANSYS Products:	ANSYS Multiphysics, ANSYS Mechanical, ANSYS Structural, ANSYS ED
Level of Difficulty:	easy
Interactive Time Required:	60 to 90 minutes
Discipline:	structural
Analysis Type:	linear static
Element Types Used:	PLANE82
ANSYS Features Demonstrated:	solid modeling including primitives, Boolean operations, and fillets; tapered pressure load; deformed shape and stress displays; listing of reaction forces; examination of structural energy error
Applicable Help Available:	Structural Static Analysis in the ANSYS Structural Analysis Guide , PLANE82 in the ANSYS Elements Reference .

2.1.2. Problem Description

This is a simple, single load step, structural static analysis of the corner angle bracket shown below. 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.



2.1.2.1. Given

The dimensions of the corner bracket are shown in the accompanying figure. The bracket is made of A36 steel with a Young's modulus of 30E6 psi and Poisson's ratio of .27.

2.1.2.2. Approach and Assumptions

Assume [plane stress](#) for this analysis. Since the bracket is thin in the z direction (1/2 inch thickness) compared to its x and y dimensions, and since the pressure load acts only in the x - y plane, this is a valid assumption.

Your approach is to use solid modeling to generate the 2-D model and automatically mesh it with nodes and elements. (Another alternative in ANSYS is to create the nodes and elements directly.)

2.1.2.3. Summary of Steps

Use the information in the problem description and the steps below as a guideline in solving the problem on your own. Or, use the detailed interactive step-by-step solution by choosing the link for step 1.

Note

If your system includes a Flash player (from Macromedia, Inc.), you can view demonstration videos of each step by pointing your web browser to the following URL address:

http://www.ansys.com/techmedia/structural_tutorial_videos.html.

Build Geometry

1. [Define rectangles.](#)
2. [Change plot controls and replot.](#)
3. [Change working plane to polar and create first circle.](#)
4. [Move working plane and create second circle.](#)
5. [Add areas.](#)
6. [Create line fillet.](#)
7. [Create fillet area.](#)
8. [Add areas together.](#)
9. [Create first pin hole.](#)
10. [Move working plane and create second pin hole.](#)
11. [Subtract pin holes from bracket.](#)
12. [Save the database as model.db.](#)

[Back To Top](#)

Define Materials

13. [Set Preferences.](#)
14. [Define Material Properties.](#)
15. [Define element types and options.](#)
16. [Define real constants.](#)

[Back To Top](#)

Generate Mesh

17. [Mesh the area.](#)
18. [Save the database as mesh.db.](#)

[Back To Top](#)

Apply Loads

19. [Apply displacement constraints.](#)

20. [Apply pressure load.](#)

[Back To Top](#)

Obtain Solution

21. [Solve.](#)

[Back To Top](#)

Review Results

22. [Enter the general postprocessor and read in the results.](#)

23. [Plot the deformed shape.](#)

24. [Plot the von Mises equivalent stress.](#)

25. [List the reaction solution.](#)

26. [Exit the ANSYS program.](#)

2.1.3. Build Geometry

This is the beginning of [Preprocessing](#).

2.1.3.1. 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 rectangles 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**.

1. **Main Menu>Preprocessor>Modeling>Create>Areas>Rectangle>By Dimensions**

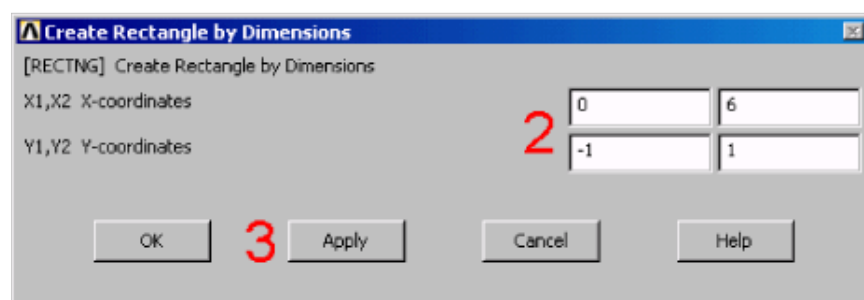
2. Enter the following:

X1 = 0

(Note:
Press
the Tab
key
between
entries)

X2 = 6

Y1 = -1



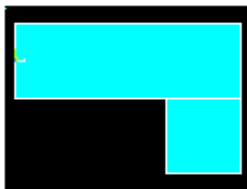
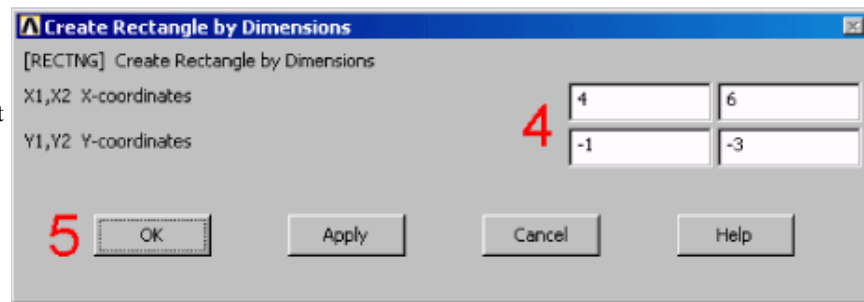
- Y2 = 1
→
3. Apply to create the first rectangle. →
 4. Enter the following:

X1 = 4

X2 = 6

Y1 = -1

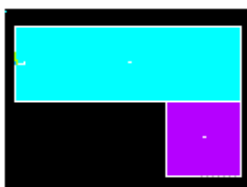
Y2 = -3
→
 5. OK to create the second rectangle and close the dialog box.
→



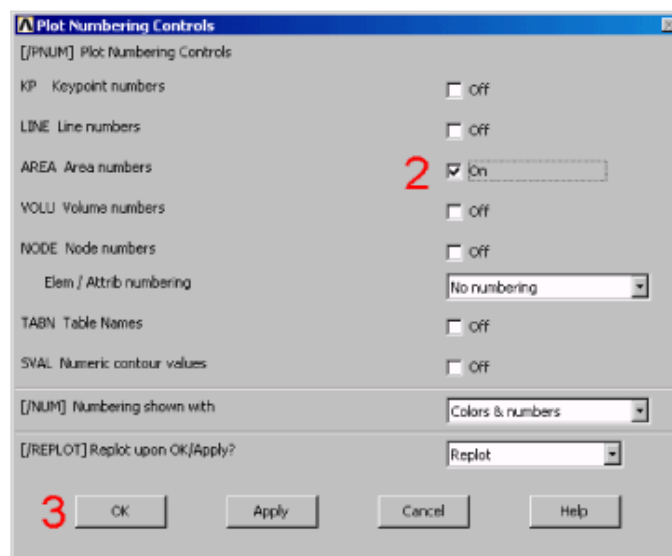
2.1.3.2. Step 2: Change plot controls and replot.

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. Utility Menu> PlotCtrls> Numbering
2. Turn on area numbers. →
3. 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*. If you started ANSYS using the product launcher, you can specify a [jobname](#) at that point (the default jobname is *file*). 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.)

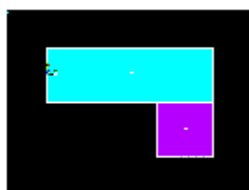
4. Toolbar: **SAVE_DB**.

2.1.3.3. 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 with a [Boolean](#) "add" operation (discussed in step 5.). 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. **Utility Menu> PlotCtrls> Pan, Zoom, Rotate**
2. Click on small dot once to zoom out. ➡
3. Close dialog box. ➡
4. **Utility Menu> WorkPlane> Display Working Plane** (toggle on)

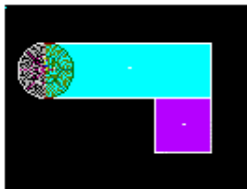


Notice the working plane origin is immediately plotted in the Graphics Window. It is indicated

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. **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. **Main Menu> Preprocessor> Modeling> Create> Areas> Circle> Solid Circle**

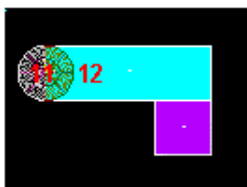
Be sure to read prompt before picking.

11. Pick center point at:

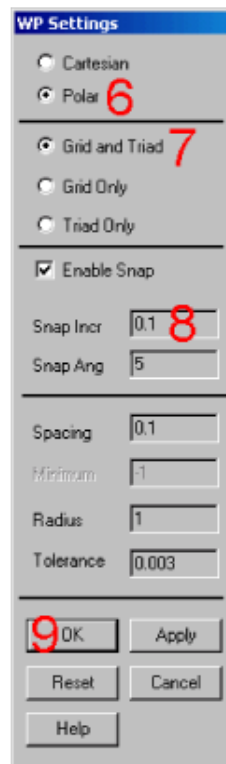
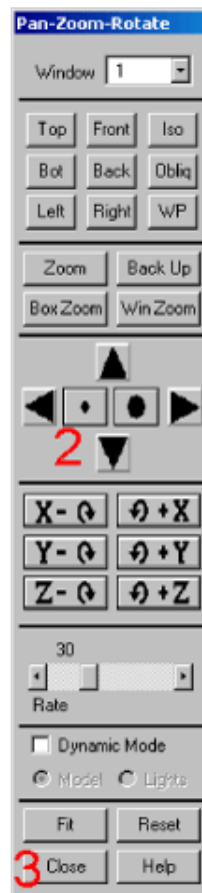
WP X = 0 (in Graphics Window shown below)

WP Y = 0

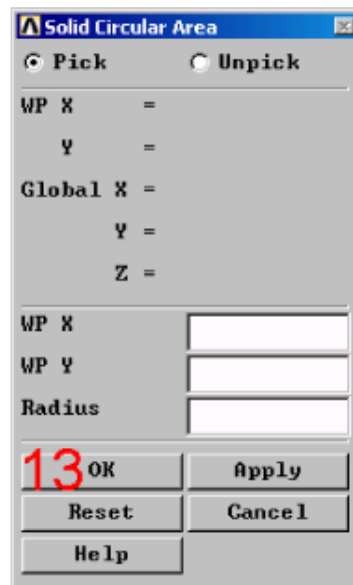
12. Move mouse to radius of 1 and click left button to create circle.



13. OK to close picking menu. ➔
14. Toolbar: **SAVE_DB**.



[C:\4] Pick 2 WP locations - center and radius

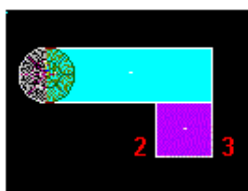
**Note**

While you are positioning the cursor for picking, the "dynamic" WP X and Y values are displayed in the Solid Circular Area dialog box. Also, as an alternative to picking, you can type these values along with the radius into the dialog box.

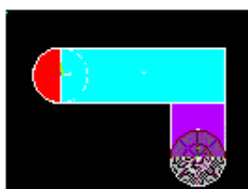
2.1.3.4. Step 4: Move working plane and create second circle.

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 offsets is to move the WP to an average keypoint location by picking the keypoints at the bottom corners of the lower, right rectangle.

1. **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.

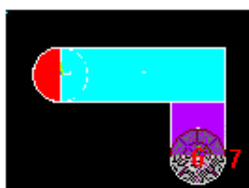


5. **Main Menu> Preprocessor> Modeling> Create> Areas> Circle> Solid Circle**
6. Pick center point at:

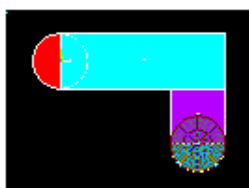
WP X = 0

WP Y = 0

7. Move mouse to radius of 1 and click left button to create circle.



8. OK to close picking menu.



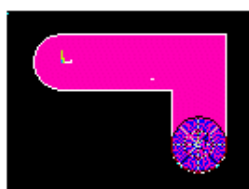
9. Toolbar: SAVE_DB.

2.1.3.5. Step 5: Add areas.

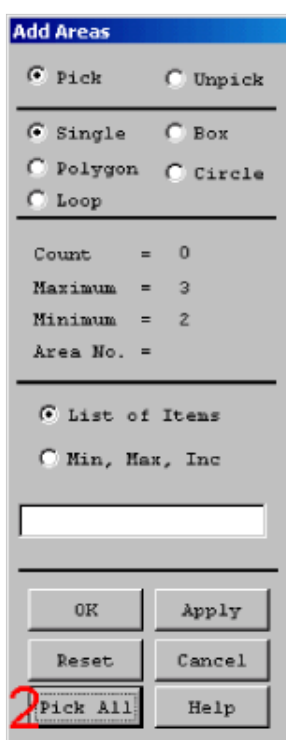
Now that the appropriate pieces of the model are defined (rectangles and circles), you need to add them together so the model becomes one continuous piece. You do this with the [Boolean](#) add operation for areas.

1. **Main Menu> Preprocessor> Modeling> Operate> Booleans> Add> Areas**

2. Pick All for all areas to be added. →



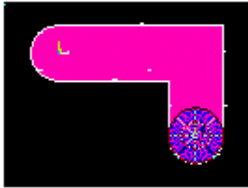
3. Toolbar: SAVE_DB.



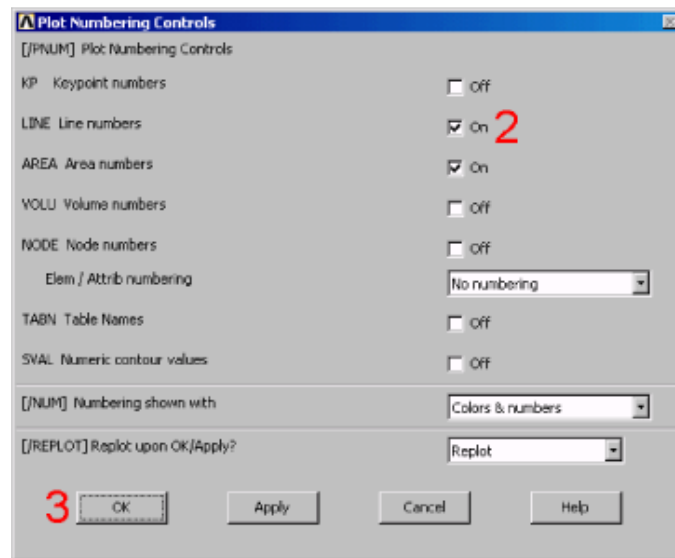
2.1.3.6. Step 6: Create line fillet.

1. **Utility Menu> PlotCtrls> Numbering**
2. Turn on line numbering. →

3. OK to change controls, close the dialog box, and automatically replot. →

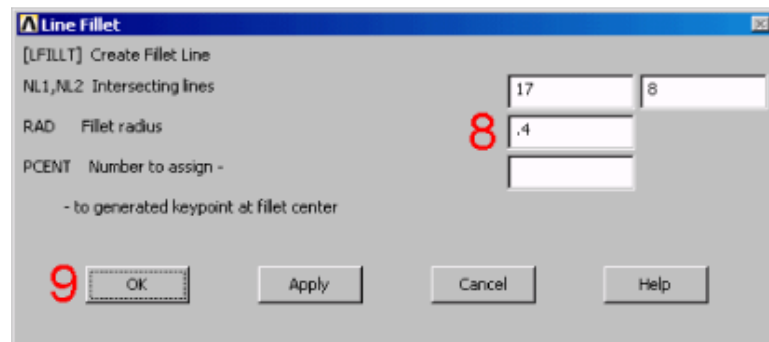
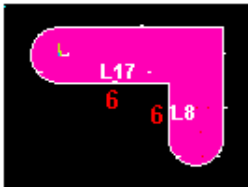


4. **Utility Menu> WorkPlane> Display Working Plane** (toggle off)



5. **Main Menu> Preprocessor> Modeling> Create> Lines> Line Fillet**

6. Pick lines 17 and 8.

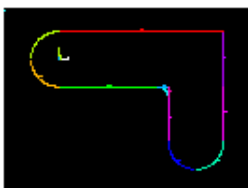


7. OK to finish picking lines (in picking menu).

8. Enter .4 as the radius. →

9. OK to create line fillet and close the dialog box. →

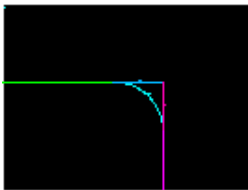
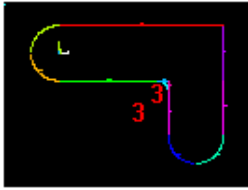
10. **Utility Menu> Plot> Lines**



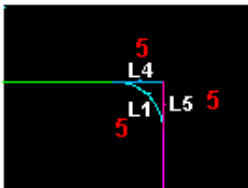
2.1.3.7. Step 7: Create fillet area.

1. **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.



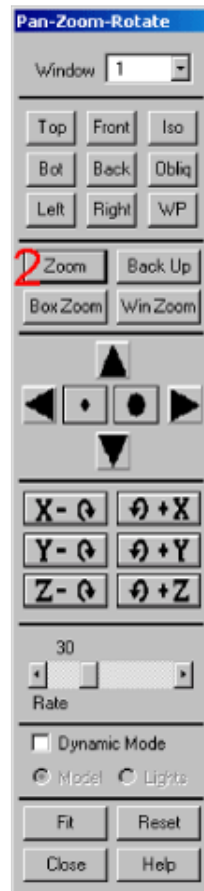
4. **Main Menu> Preprocessor> Modeling> Create> Areas> Arbitrary> By Lines**
5. Pick lines 4, 5, and 1.

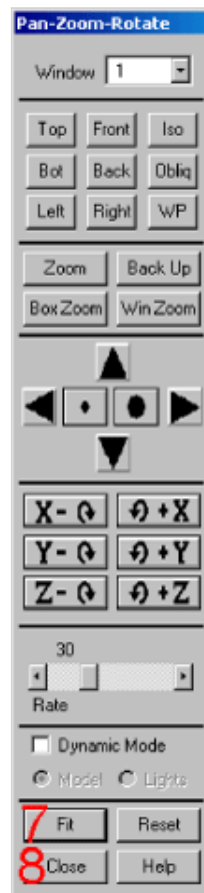


6. OK to create area and close the picking menu.
7. Click on Fit button. ➡
8. Close the Pan, Zoom, Rotate dialog box. ➡
9. **Utility Menu> Plot> Areas**



10. Toolbar: **SAVE_DB**.



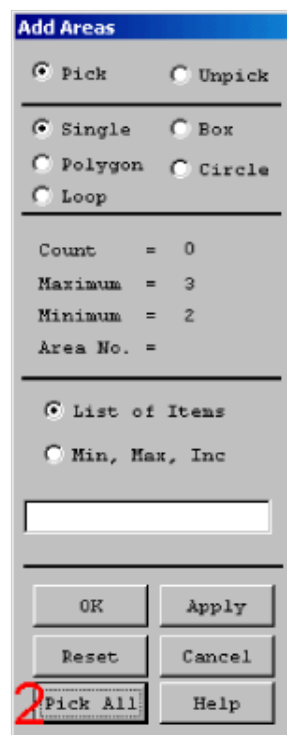


2.1.3.8. Step 8: Add areas together.

1. Main Menu> Preprocessor> Modeling> Operate> Booleans> Add> Areas
2. Pick All for all areas to be added. →

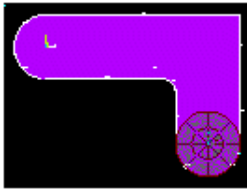


3. Toolbar: SAVE_DB.



2.1.3.9. Step 9: Create first pin hole.

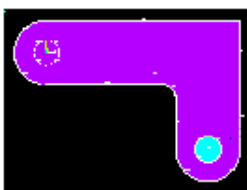
1. Utility Menu> WorkPlane> Display Working Plane (toggle on)



2. **Main Menu> Preprocessor> Modeling> Create> Areas> Circle> Solid Circle**
3. Pick center point at:
 - WP X = 0 (in Graphics Window)
 - WP Y = 0
4. Move mouse to radius of .4 (shown in the picking menu) and click left button to create circle.
5. OK to close picking menu.

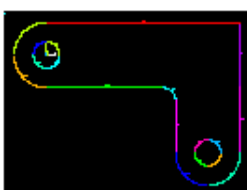
2.1.3.10. Step 10: Move working plane and create second pin hole.

1. **Utility Menu> WorkPlane> Offset WP to> Global Origin**
2. **Main Menu> Preprocessor> Modeling> Create> Areas> Circle> Solid Circle**
3. Pick center point at:
 - WP X = 0 (in Graphics Window)
 - WP Y = 0
4. Move mouse to radius of .4 (shown in the picking menu) and click left mouse button to create circle.
5. OK to close picking menu.
6. **Utility Menu> WorkPlane> Display Working Plane (toggle off)**
7. **Utility Menu> Plot> Replot**



From this area plot, 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.

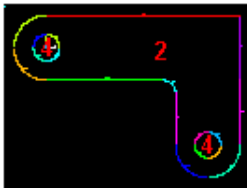
8. **Utility Menu> Plot> Lines**



9. Toolbar: **SAVE_DB**.

2.1.3.11. Step 11: Subtract pin holes from bracket.

1. **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.



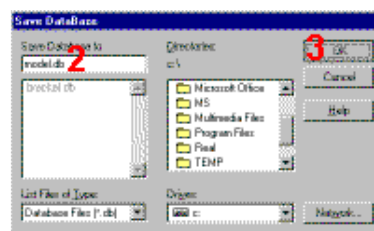
5. OK to subtract holes and close picking menu.



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



2.1.4. Define Materials

2.1.4.1. Step 13: Set preferences.

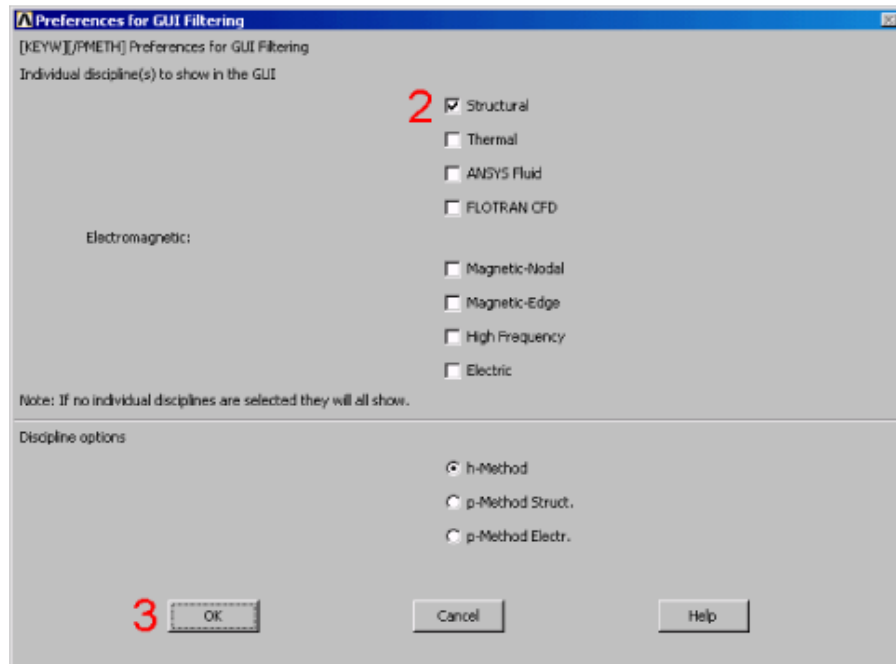
In preparation for defining materials, you will set [preferences](#) so that only materials that pertain to a structural analysis are available for you to choose.

To set preferences:

1. **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. →

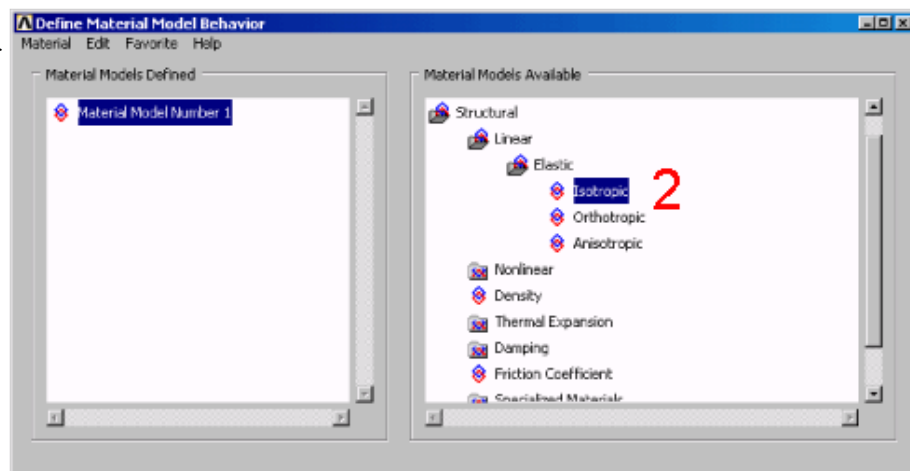
3. OK to apply filtering and close the dialog box. →

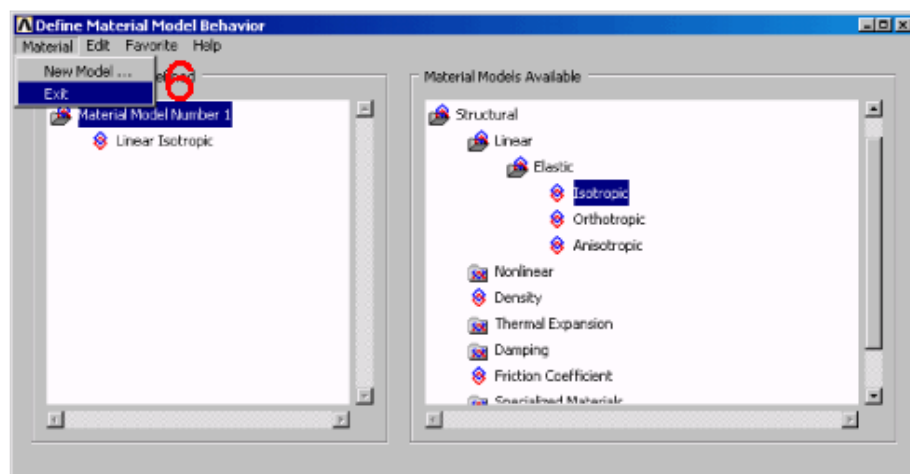
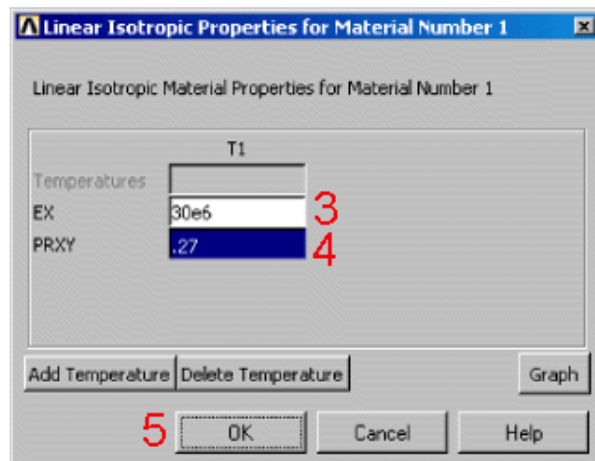


2.1.4.2. Step 14: Define material properties.

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. **Main Menu> Preprocessor> Material Props> Material Models**
2. Double-click on Structural, Linear, Elastic, Isotropic. →
3. Enter 30e6 for EX. →
4. Enter .27 for PRXY. →
5. OK to define material property set and close the dialog box. →
6. **Material> Exit** →



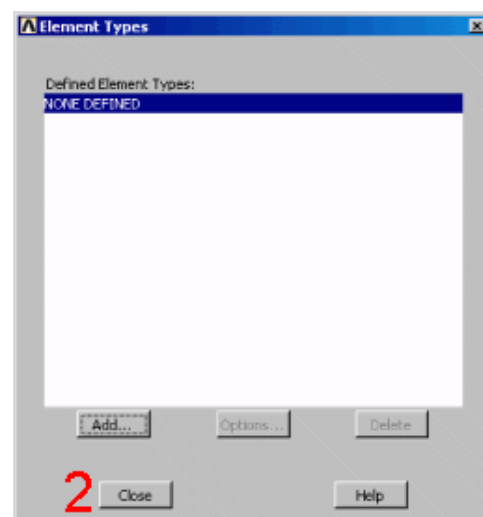


2.1.4.3. Step 15: Define element types and options.

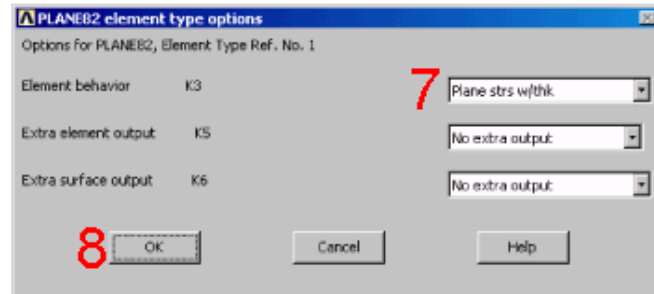
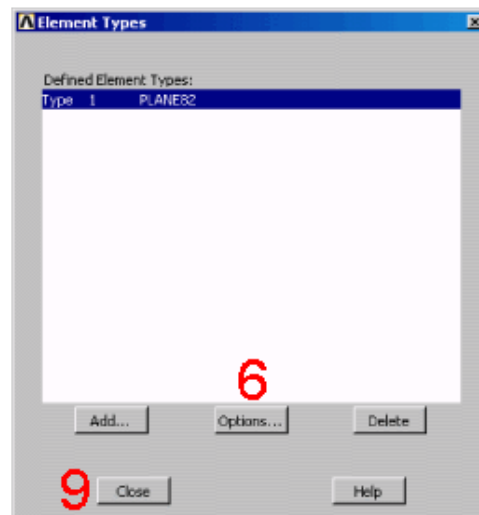
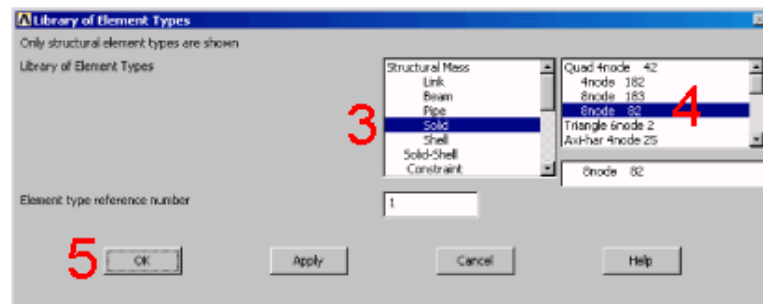
In any analysis, you need to select from a library of [element types](#) and define the appropriate ones for your analysis.

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. **Main Menu> Preprocessor> Element Type> Add/Edit/Delete**
2. Add an element type. →
3. Structural solid family of elements. →
4. Choose the 8-node quad ([PLANE82](#)). →
5. OK to apply the element type and close the dialog box. →



6. Options for PLANE82 are to be defined. →
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. →



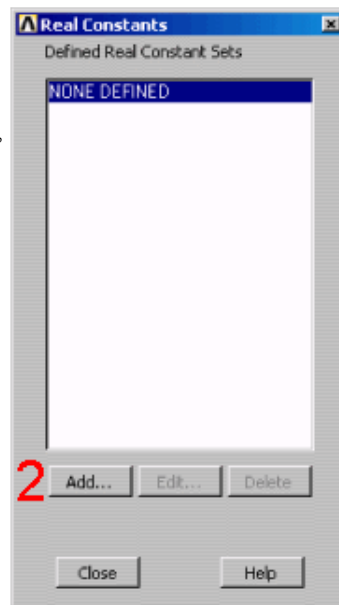
2.1.4.4. Step 16: Define real constants.

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.

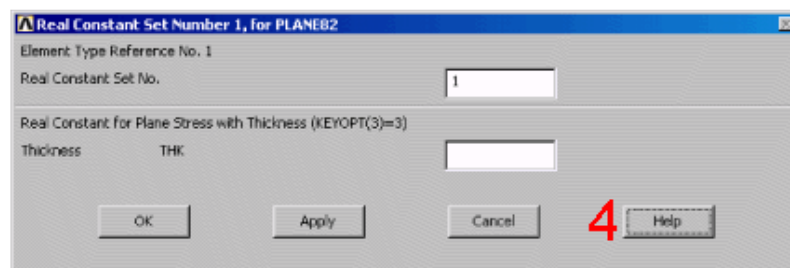
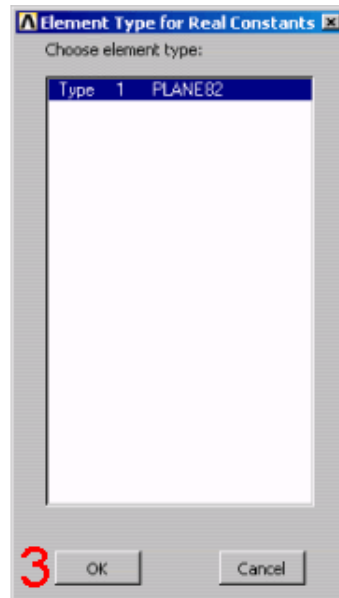
1. **Main Menu>
Preprocessor> Real
Constants>
Add/Edit/Delete**
2. Add a real constant set. →
3. OK for PLANE82. →

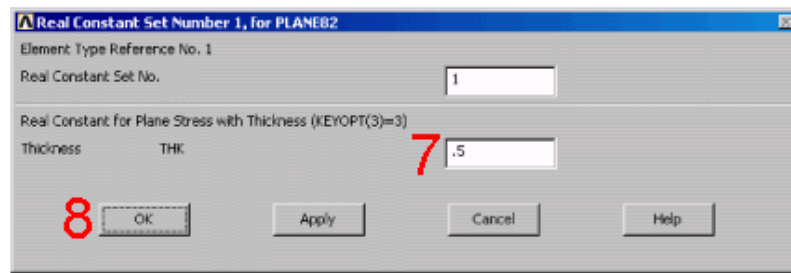
Before clicking on the Help button in the next

step, you should be aware that the help information may appear in the same window as this tutorial, *replacing* the contents of the tutorial. After reading the help information, click on the Back button to return to this tutorial. If the help information appears in a *separate* window from the tutorial, minimize or close the help window after you read the help information.



4. Help to get help on [PLANE82](#). →
5. Hold left mouse button down to scroll through element description.
6. If the help information replaced the tutorial, click on the Back button to return to the tutorial.
7. Enter .5 for THK. →
8. OK to define the real constant and close the dialog box. →
9. Close the real constant dialog box. →





2.1.5. Generate Mesh

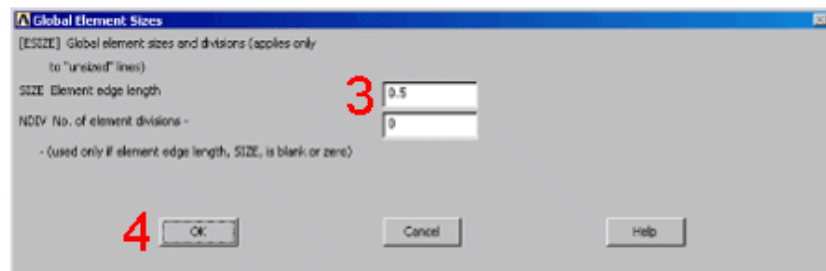
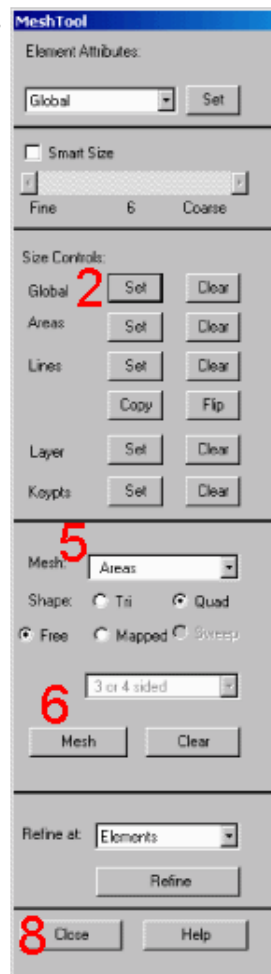
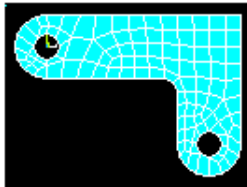
2.1.5.1. Step 17: Mesh the area.

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. **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.



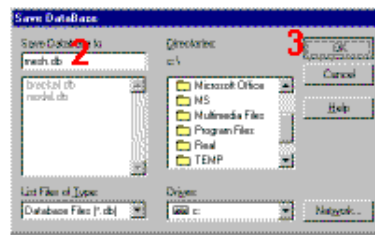
Note

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. For a discussion of results accuracy, see [Planning Your Approach](#) in the [ANSYS Modeling and Meshing Guide](#).

2.1.5.2. Step 18: Save the database as mesh.db.

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

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



2.1.6. Apply Loads

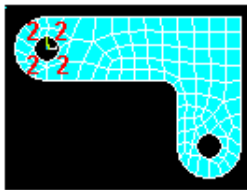
The beginning of the [solution](#) phase.

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.

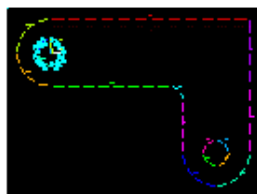
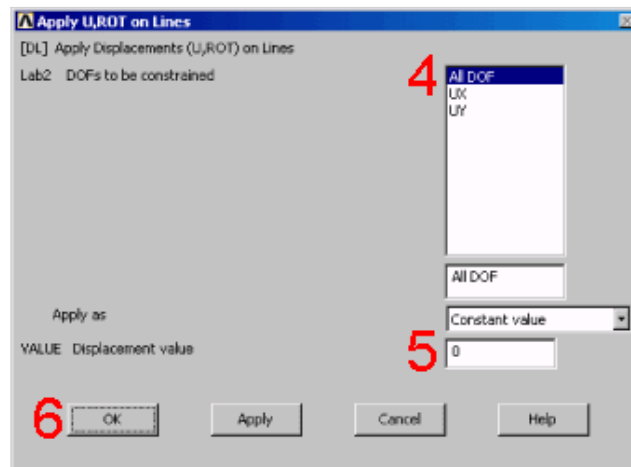
2.1.6.1. Step 19: Apply displacement constraints.

You can apply displacement constraints directly to lines.

1. **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. **Utility Menu> Plot Lines**



8. Toolbar: **SAVE_DB**.

2.1.6.2. Step 20: Apply pressure load.

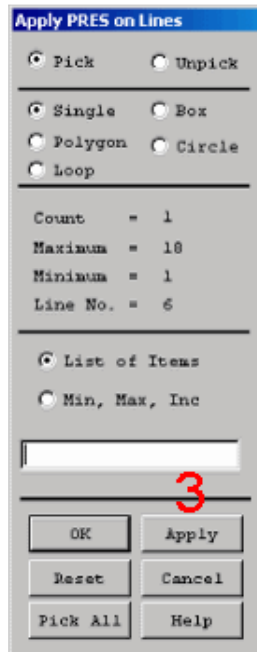
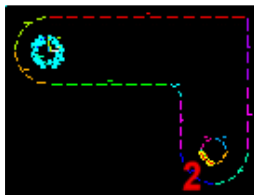
Now apply the tapered pressure load to the bottom, right-hand pin hole. ("Tapered" here means varying linearly.) Note that when a circle is created in ANSYS, four lines define the perimeter. Therefore, apply the pressure to *two* lines making up the lower half of the circle. Since the pressure tapers 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).

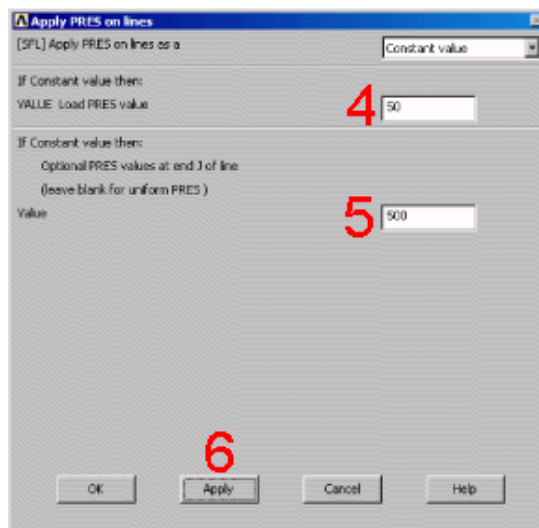
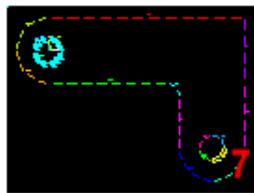
1. **Main Menu> Solution> Define Loads> Apply> Structural> Pressure> On Lines**

2. Pick line defining bottom *left* part of the circle (line 6).

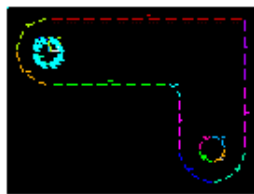


3. Apply. →
4. Enter 50 for VALUE. →
5. Enter 500 for optional value. →
6. Apply. →

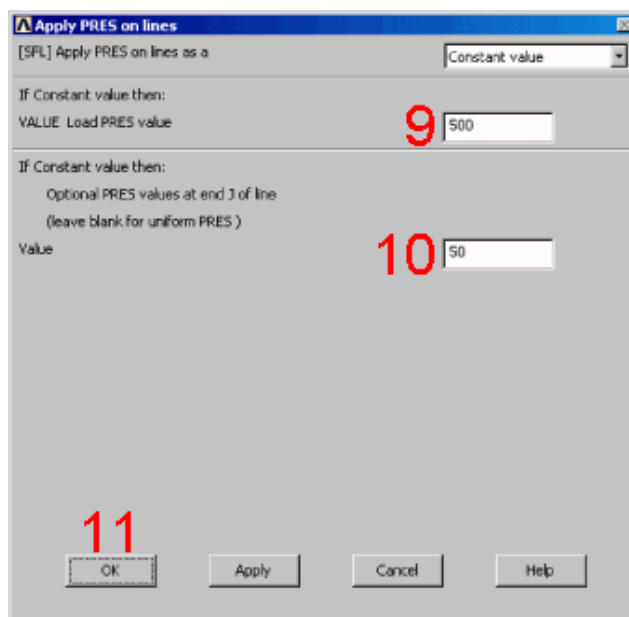
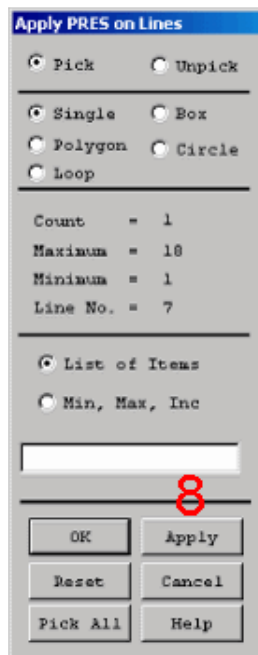
7. Pick line defining bottom *right* part of circle (line 7).



8. Apply. →
9. Enter 500 for VALUE. →
10. Enter 50 for optional value. →
11. OK. →



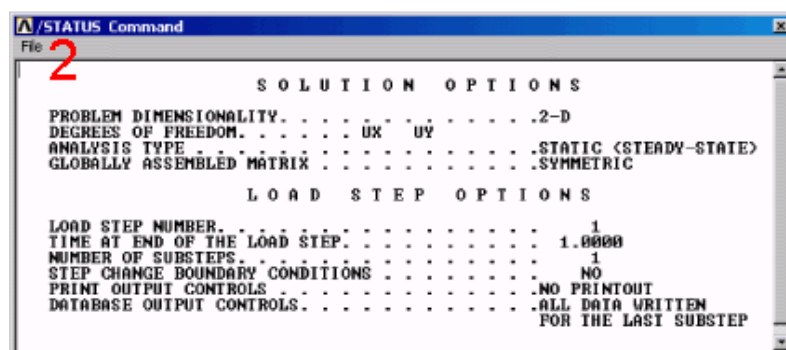
12. Toolbar: **SAVE_DB**.



2.1.7. Obtain Solution

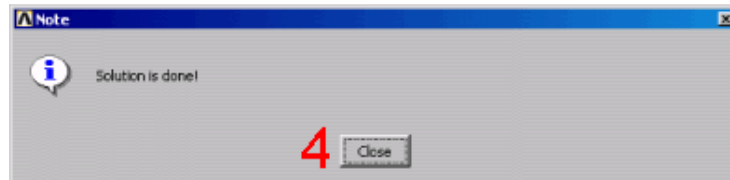
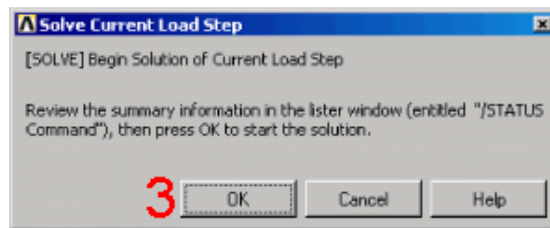
2.1.7.1. Step 21: Solve.

1. **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.

2.1.8. Review Results

The beginning of the [postprocessing](#) phase.

Note

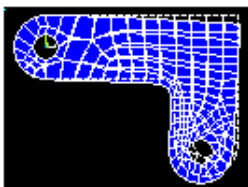
The results you see may vary slightly from what is shown here due to variations in the mesh.

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

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

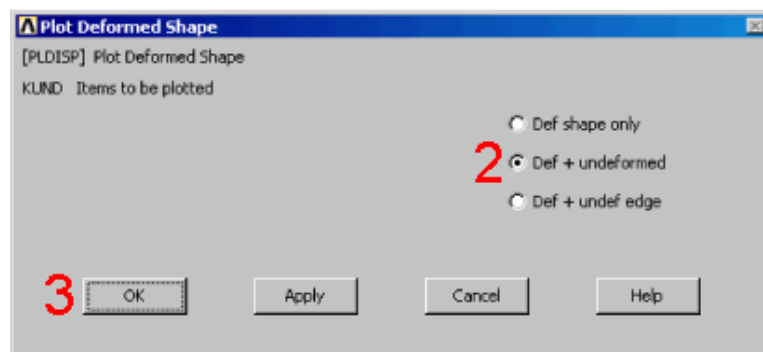
2.1.8.2. Step 23: Plot the deformed shape.

1. Main Menu> General Postproc> Plot Results> Deformed Shape
2. Choose Def + undeformed. →
3. OK. →



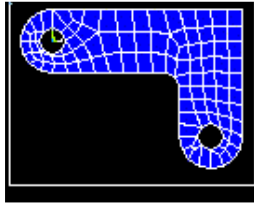
You can also produce an animated version of the deformed shape:

4. Utility Menu> Plot
Ctrl+P> Animate>
Deformed Shape
5. Choose Def + undeformed.

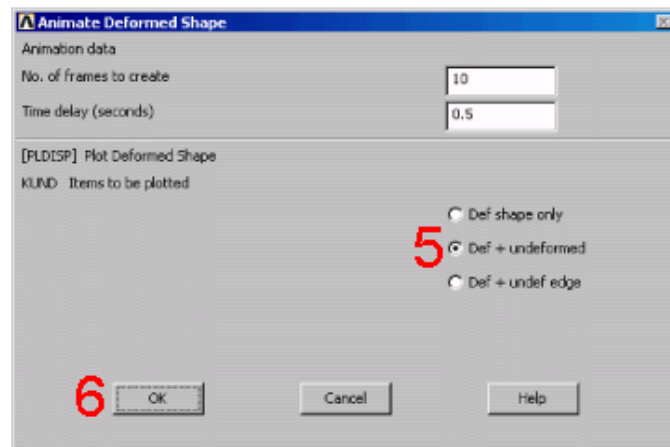




6. OK. →

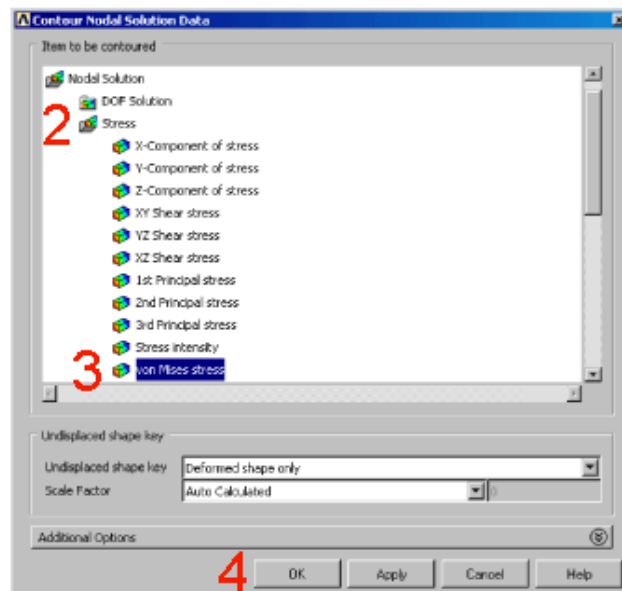
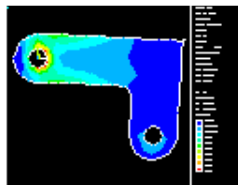


7. Make choices in the Animation Controller (not shown), if necessary, then choose Close.



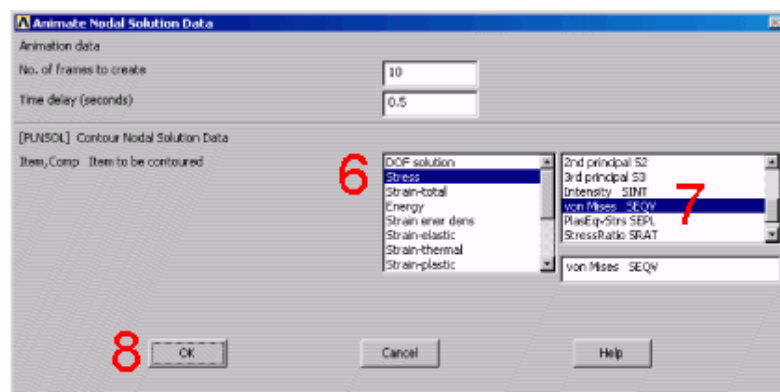
2.1.8.3. Step 24: Plot the von Mises equivalent stress.

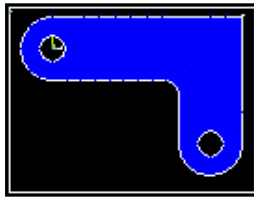
1. **Main Menu> General Postproc> Plot Results> Contour Plot> Nodal Solu**
2. Choose Stress item to be contoured. →
3. Scroll down and choose von Mises (SEQV). →
4. OK. →



You can also produce an animated version of these results:

5. **Utility Menu> PlotCtrls> Animate> Deformed Results**
6. Choose Stress item to be contoured. →
7. Scroll down and choose von Mises (SEQV). →
8. OK. →

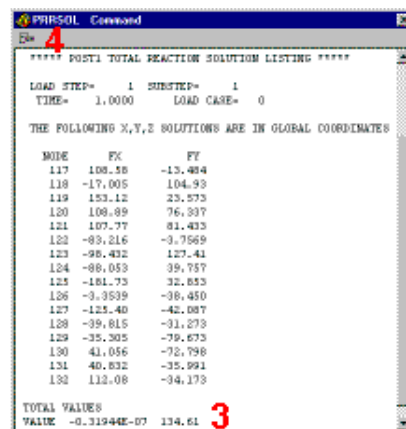
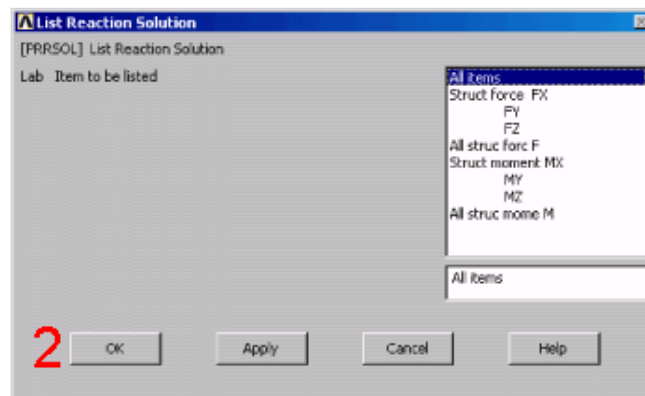




9. Make choices in the Animation Controller (not shown), if necessary, then choose Close.

2.1.8.4. Step 25: List reaction solution.

1. **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. **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

The values shown are representative and may vary from the values you obtain.

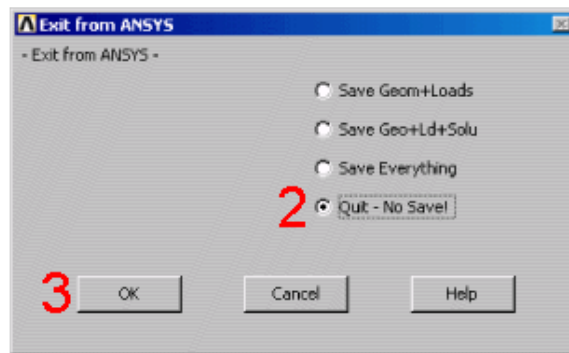
There are many other options available for reviewing results in the general postprocessor. You'll see some of these demonstrated in other tutorials. You have finished the analysis. Exit the program in the next step.

2.1.8.5. 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. →



Congratulations! You have completed this tutorial.

Even though you have exited the ANSYS program, you can still view animations [using the ANSYS ANIMATE program](#). The ANIMATE program runs only on the PC and is extremely useful for:

- Viewing ANSYS animations on a PC regardless of whether the files were created on a PC (AVI files) or on a UNIX workstation (ANIM files).
- Converting ANIM files to AVI files.
- Sending animations over the web.