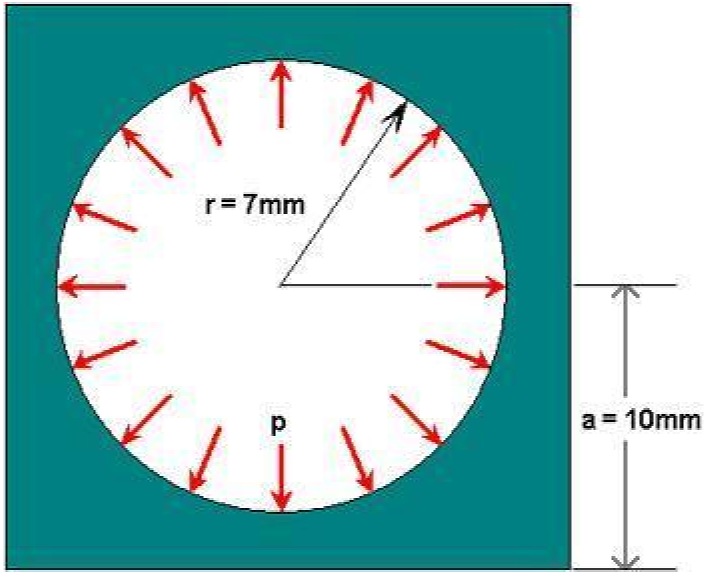
ANSYS - Plate with a hole (old version)

Author: Rajesh Bhaskaran, Cornell University

Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve
8. Postprocess the results
9. Validate the results
Problem Set 1

Problem Specification

Consider the square plate of uniform thickness with a circular hole with dimensions shown in the figure below. The thickness of the plate is 1 mm. The Young's modulus E =10^7 MPa and the Poisson ratio is 0.3. A uniform pressure p=1 MPa acts on the boundary of the hole. Assume that plane stress conditions prevail. Determine the displacement and stress fields using ANSYS. This problem is taken from section 6.14, p. 240-244 of Cook et al



Go to Step 1: Start-up and preliminary set-up

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Step 1: Start-up and preliminary set-up

Start ANSYS

Create a folder called *plate* at a convenient location. We'll use this folder to store files created during the ANSYS session.

Start > All Programs > ANSYS 12.0 > Mechanical APDL Product Launcher

Note that in version 11, it is at

Start > Programs > ANSYS 11.0 > ANSYS Product Launcher

In the window that comes up, enter the location of the folder you just created as your ***Working Directory*** by browsing to it. All files generated during the ANSYS run will be stored in this directory/folder.

Specify `plate` as your ***Job Name***. The job name is the prefix used for all files generated during the ANSYS session. For example, when you perform a save operation in ANSYS, it'll store your work in a file called *plate.db* in your working directory.

Click on ***Run***. This brings up the ANSYS interface. To make best use of screen real estate, move the windows around and resize them so that you approximate this screen arrangment. This way you can read instructions in the browser window and implement them in ANSYS.

You can resize the text in the browser window to your taste and comfort.

In Internet Explorer, use **Menubar > View > Text Size**, then choose the appropriate font size.

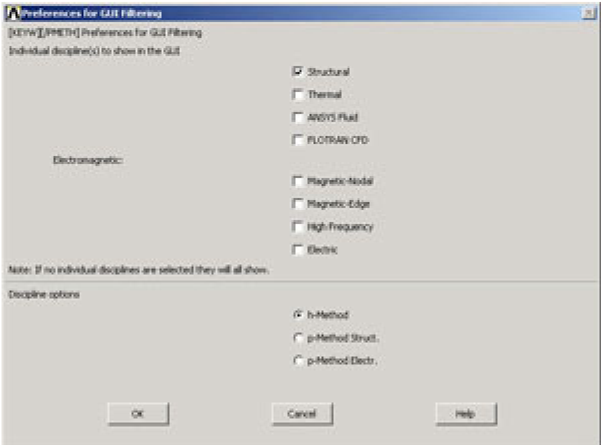
In Mozilla Firefox, use **Menubar > View > Zoom.**

Set Preferences

As before, we'll more or less work our way down the *Main Menu*.

Main Menu > Preferences

In the *Preferences for GUI Filtering* dialog box, click on the box next to ***Structural*** so that a tick mark appears in the box. Click ***OK***.



Recall that this is an optional step that customizes the graphical user interface so that only menu options valid for structural problems are made available during the ANSYS session.

Enter Parameters

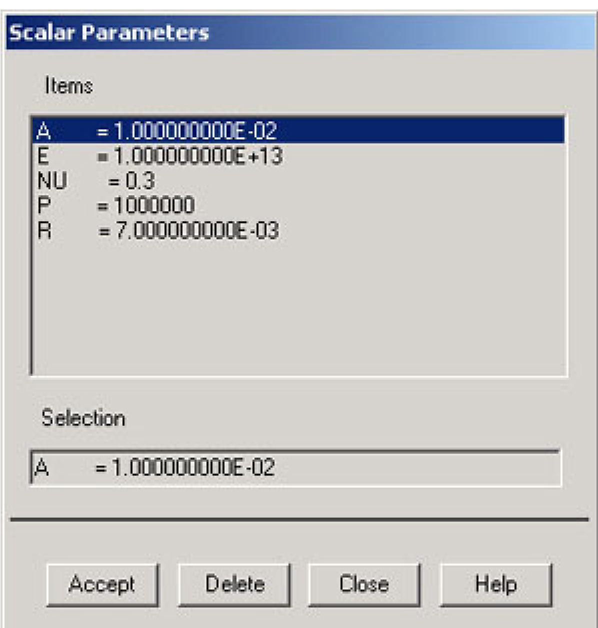
For convenience, we'll create scalar parameters corresponding to the plate half-width *a,* hole radius *r,* pressure *p,* and material properties *E* and *v.*

Utility Menu > Parameters > Scalar Parameters

Enter the parameter value for *a*:
`a=10e-3`
Click ***Accept***.

Similarly, enter the other parameter values and click ***Accept*** after each.

```
r=7e-3
p=1e6
E=1e13
nu=0.3
```



Close the *Scalar Parameters* window.

We can now enter these variable names instead of the corresponding values as we set up the problem in ANSYS. This is also helpful in carrying out parametric studies where one looks at the effect of changing a parameter.

Go to Step 2: Specify element type and constants

See and rate the complete Learning Module

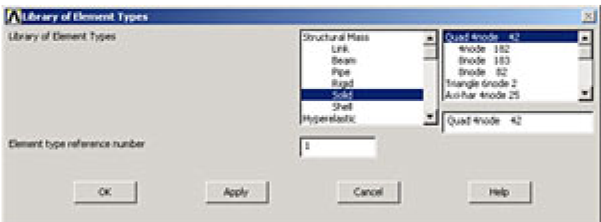
Go to all ANSYS Learning Modules

Step 2: Specify element type and constants

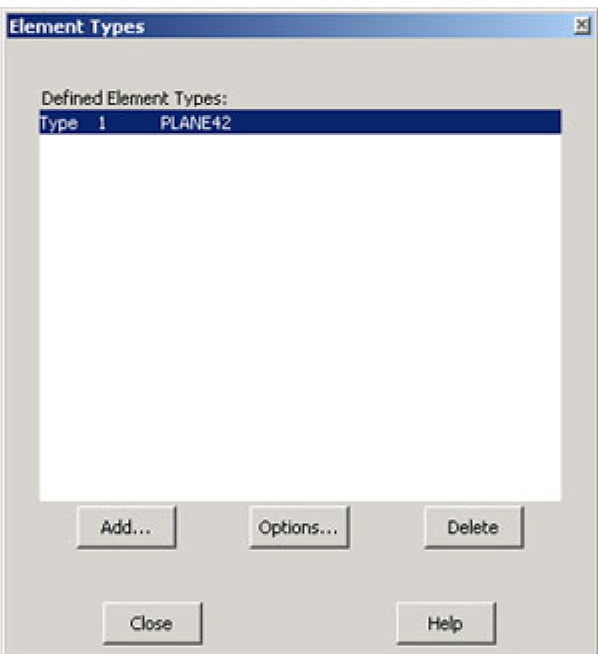
Specify Element Type

Main Menu > Preprocessor> Element Type > Add/Edit/Delete > Add...

Pick ***Structural Mass*** with subtype ***Solid*** in the left field. Pick ***Quad 4 node 42*** in the right field. Click ***OK*** to select this element.



You'll now see the ***Element Types*** menu with ***PLANE42*** as the only defined element type.



Let's take a look at the online help pages to learn about the properties of this element.

Utility Menu > Help > Help Topics

Select the ***Search*** tab, type in ***pictorial summary*** as the keyword and click ***List Topics***. You should see ***Pictorial Summary*** as one of the topics listed; double-click on this. This brings up the *Pictorial Summary of Element Types* help page. Scroll down to ***Plane42*** under *Structural 2-D Solid.* Note that the *PL ANE42* element is defined by four nodes with two degrees of freedom at each node: translations *UX* and *UY* in the (nodal) x and y-directions.

Click on the ***PLANE42*** box to bring up the help page for this element. Read the *Element Description* and take a look at the figure of the element. Think about why this element is appropriate for the problem at hand. Minimize the help window.

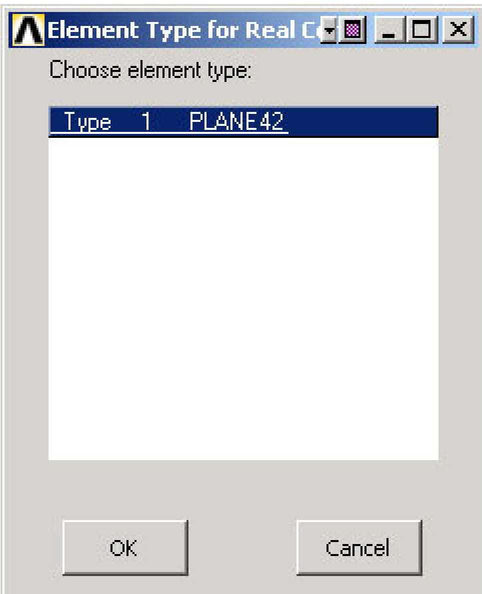
If you actually read the *Element Description* for PLANE42, you'd have noticed that this element can also be used for axisymmetric problems. In the axisymmetric case, you would choose ***Options***... for the element in the *Element Types* menu. Note that in the *PLANE42 element type options* menu that comes up, under ***Element behavior***, you have the option of ***Axisymmetric***. For the current problem, we'll of course use the default of ***Plane stress***. Click ***Cancel*** to exit the *PLANE42 element type options* menu retaining the defaults.

Close the *Element Types* menu.

Specify Element Constants

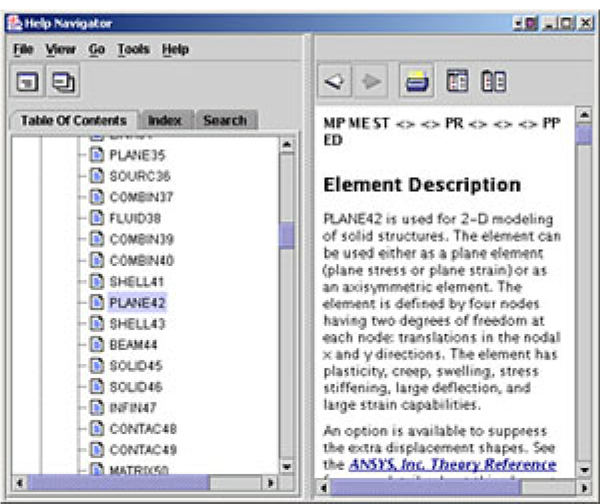
Main Menu > Preprocessor> Real Constants > Add/Edit/Delete > Add

This brings up the *Element Type for Real Constants* menu with a list of the element types defined in the previous step. We have only one element type and it is automatically selected.



Click ***OK***.

You should get a note saying "Please check and change keyopt setting for element PLANE42 before proceeding." Close the warning window and the *Real Constants* menu. To see what this message implies, let's again take a look at the help pages for PLANE42.



Under *PLANE42 Input Summary*, the documentation says that there are no real constants for this element when KEYOPT(3)=0, 1, 2.

To see what the value of KEYOPT(3) is, bring up the Element Type menu again:

Main Menu > Preprocessor> Element Type > Add/Edit/Delete > Options

K3 i.e. KEYOPT(3) is set to ***Plane stress***. In the help page, under *PLANE42 Input Summary*, you can check that plane stress corresponds to KEYOPT(3) =0. Thus, there are no real constants to be specified. That's why we got the "Please check and change keyopt settings..." warning message. Of course, the ANSYS warning could have been less cryptic but what fun would that be.

Cancel the *PLANE42 element type options* menu, ***Close*** the *Element Types* menu and close the *Element Type* sticky menu.

Save your work

Toolbar > SAVE_DB

Go to Step 3: Specify material properties

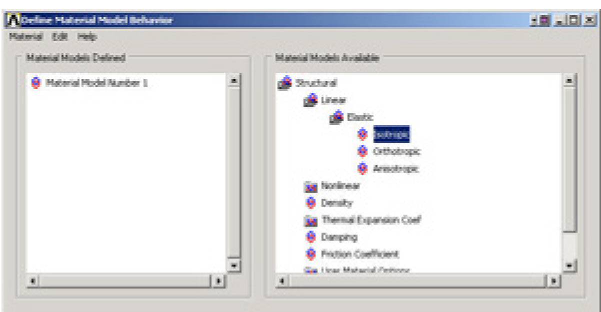
See and rate the complete Learning Module

Go to all ANSYS Learning Modules

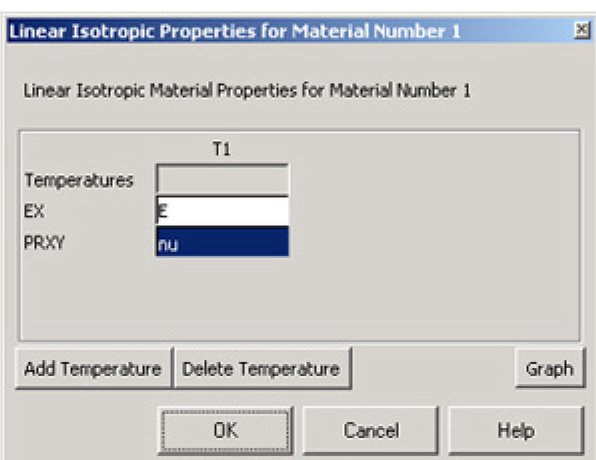
Step 3: Specify material properties

Main Menu > Preprocessor >Material Props > Material Models ....

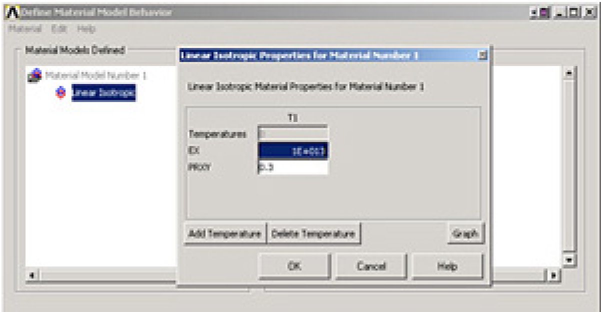
In the *Define Material Model Behavior* menu, double-click on ***Structural, Linear, Elastic,*** and ***Isotropic.***



We'll use the previously defined parameter names while specifying the material properties. Enter *E* for Young's modulus ***EX***, *nu* for Poisson's Ratio ***PRXY***. Click ***OK***



To double-check the material property values, double-click on ***Linear, Isotropic*** under ***Material Model Number 1*** in the *Define Material Model Behavior* menu. This will show you the current values for *EX* and *PRXY*. ***Cancel*** the *Linear Isotropic Properties* window.



When you enter parameter names, ANSYS substitutes the corresponding parameter values as soon as you click ***OK*** or ***Apply***.

This completes the specification of *Material Model Number 1*. When we mesh the geometry later on, we'll use the reference no. 1 to assign this material model. Close the *Define Material Model Behavior* menu.

Save your work

Toolbar > SAVE_DB

Go to Step 4: Specify geometry

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Step 4: Specify geometry

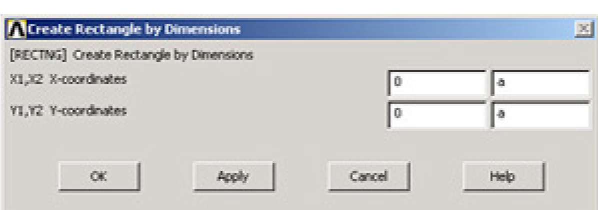
Since the geometry, material properties and loading are all symmetric with respect to the horizontal and vertical centerlines, we need to model only a quarter of the plate. We will take the origin of the coordinate system to be at the center of the hole and model only the top right quadrant. We'll create the geometry by creating a square area of side *a* and subtracting the circular sector of radius *r* from it.

Create the Square

Main Menu > Preprocessor > Modeling >Create > Areas >Rectangle > By Dimensions

X1 and ***X2*** are the x-coordinates of the left and right edges of the square, respectively. Enter `0` for ***X1***, a for ***X2***.

Y1 and ***Y2*** are the y-coordinates of the bottom and top edges of the square, respectively. Enter `0` for ***Y1***, a for ***Y2***.



Click ***OK***. You should see a square appear in the graphics window.

Create the Circular Sector

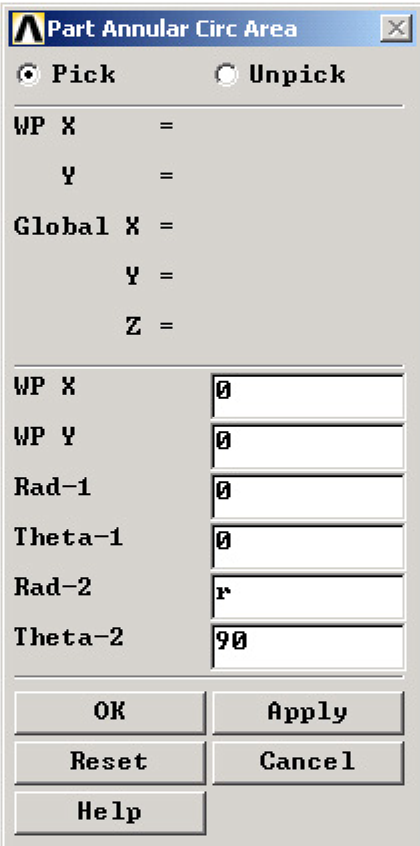
Main Menu > Preprocessor > Modeling > Create > Areas > Circle > Partial Annulus

WP X and ***WP Y*** are the x- and y-coordinates of the center of the circular arc. So enter 0 for both ***WP X*** and ***WP Y*** (*WP* refers to the *Working Plane* which by default coincides with the global Cartesian coordinate system. We won't have to worry about the working plane in this friendly example.)

Rad-1 is the radius of the inner circular arc. We want to create a solid rather than an annular arc. Enter `0` for ***Rad-1*** to create a solid arc.

Rad-2 is the (outer) radius of the arc. Since we had defined the hole radius as parameter *r* earlier, enter `r` for ***Rad-2***.

Theta-1 and ***Theta-2*** are the starting and ending angles of the arc, respectively. These angles need to be specified in degrees. Enter `0` for ***Theta-1*** and `90` for ***Theta-2***. Click ***OK***.

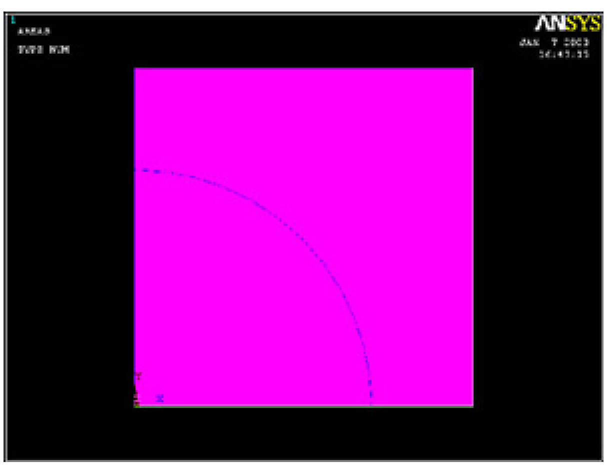


This will create and draw the circular sector. You'll see a white line denoting the circular sector.

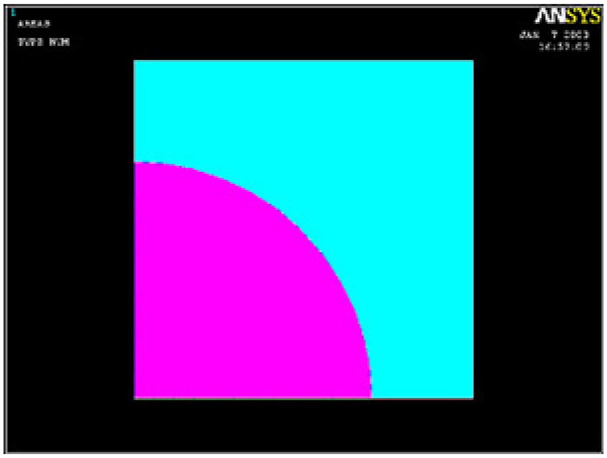
Subtract Circular Sector from Square

Main Menu > Preprocessor >Modeling > Operate > Booleans > Subtract > Areas

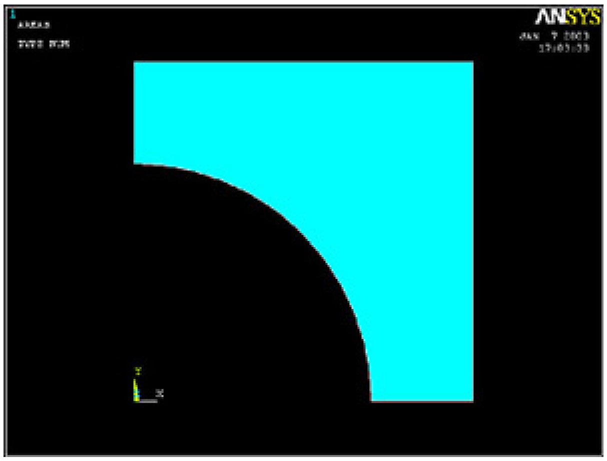
In the *Input* window, ANSYS tells you to "pick or enter base areas from which to subtract". So we pick the square area as follows: Hold down the left mouse button, move the cursor over the areas until the square is selected (it will change color) and release the left mouse button. Click ***OK***.



In the *Input* window, ANSYS now tells you to "pick or enter areas to be subtracted". So select the circular sector by holding down and releasing the left mouse button. Click ***OK***.



If you did this correctly, you will see that the circular sector has been subtracted out from the square area.



You can also select areas during the Boolean subtract operation by simply clicking on them but it becomes difficult to select areas (and other components) in this fashion in more complicated geometries. That's why I made you use the "holding-down-the-mouse-and-releasing" technique.

If you picked an area incorrectly, you can unpick it by clicking the right mouse button and selecting the area. The cursor changes to a downward arrow during an unpick operation. Right-click to return to pick mode.

Save Your Work

Toolbar > SAVE_DB

Go to Step 5: Mesh geometry

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Step 5: Mesh geometry

Bring up the *MeshTool*:

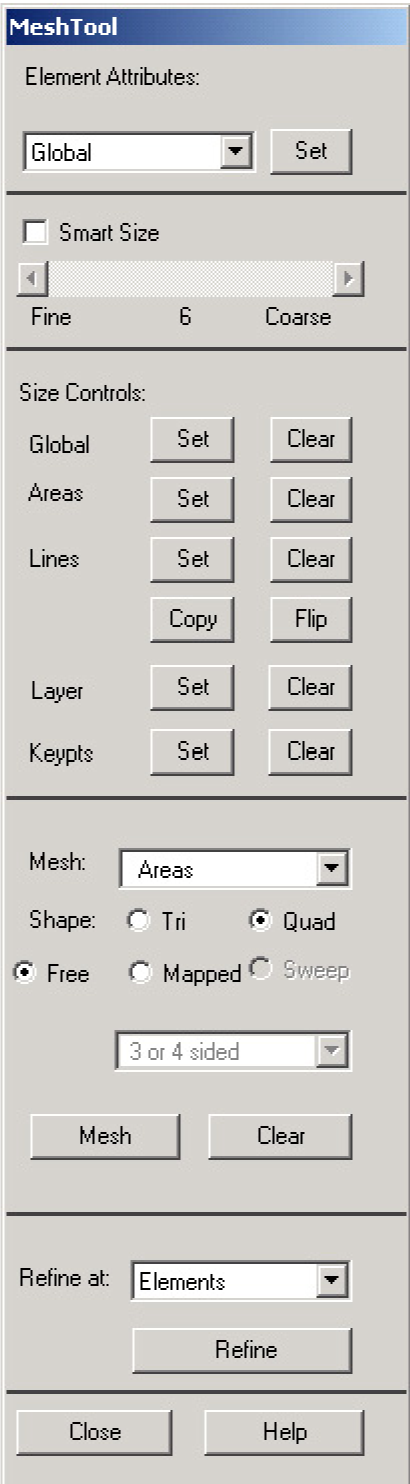
Main Menu > Preprocessor > Meshing > MeshTool

The *MeshTool* is used to control and generate the mesh.

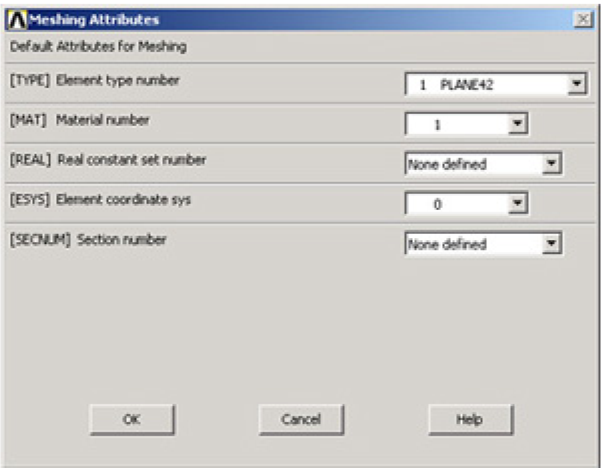
Set Meshing Parameters

We'll now specify the element type, real constant set and material property set to be used in the meshing. Since we have only one of each, we can assign them to the entire geometry using the ***Global*** option under ***Element Attributes***.

Make sure ***Global*** is selected under ***Element Attributes*** and click on Set.



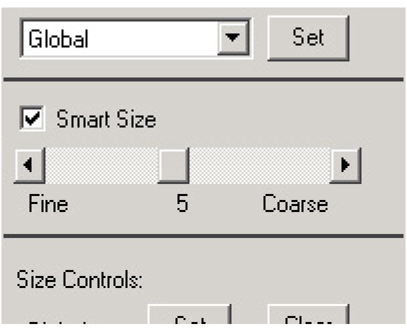
This brings up the *Meshing Attributes* menu. You will see that the correct element type and material number are already selected since we have only one of each. Recall that no real constants need to be defined for *PLANE42* element type with the plane stress keyoption.



Click ***OK***. ANSYS now knows what element type and material type to use for the mesh.

Set Mesh Size

Instead of setting the mesh size at each boundary, we'll use the *SmartSize* option which enables automatic element sizing. Click on the ***SmartSize*** checkbox so that a tickmark appears in it.

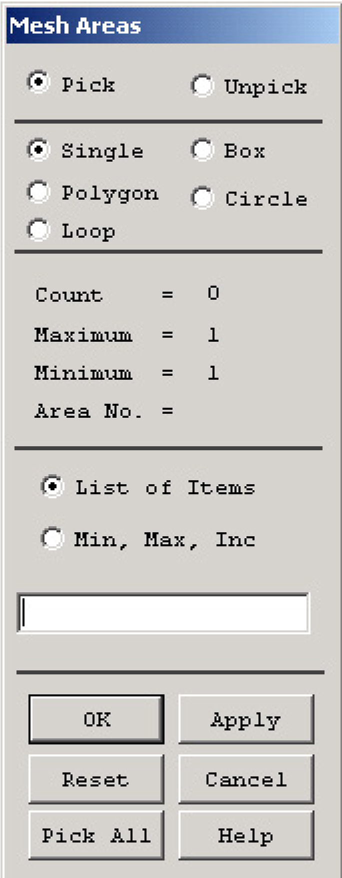


The only input necessary for the ***SmartSize*** option is the overall element size level for meshing. The element size level determines the fineness of the mesh. Its value is controlled by the slider shown in the above picture. Change the setting for the overall element size level to ***5*** by moving the slider under ***S martSize*** to the left.

Mesh Areas

In the *MeshTool*, make sure ***Areas*** is selected in the drop-down list next to ***Mesh***. This means the geometry components to be meshed are areas (as opposed to lines or volumes). We'll use quadrilateral elements. So make sure the default option of ***Quad*** is selected under ***Shape***. We'll also use the default of ***Free*** meshing.

Click on *the* ***Mesh*** button. This brings up the pick menu.



In the *Input* window, ANSYS tells you to "pick or enter areas to be meshed". Since we have only one area to be meshed, click on ***Pick All***. The geometry has been meshed and the elements are plotted in the *Graphics* window. ***Close*** the *MeshTool*.

The mesh statistics are reported in the *Output* window (usually hiding behind the *Graphics* window):

** AREA 3 MESHED WITH 105 QUADRILATERALS, 0 TRIANGLES **
** Meshing of area 3 completed ** 105 elements.

NUMBER OF AREAS MESHED = 1
MAXIMUM NODE NUMBER = 130
MAXIMUM ELEMENT NUMBER = 105

Save Your Work

Toolbar > SAVE_DB

Go to Step 6: Specify boundary conditions

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Step 6: Specify boundary conditions

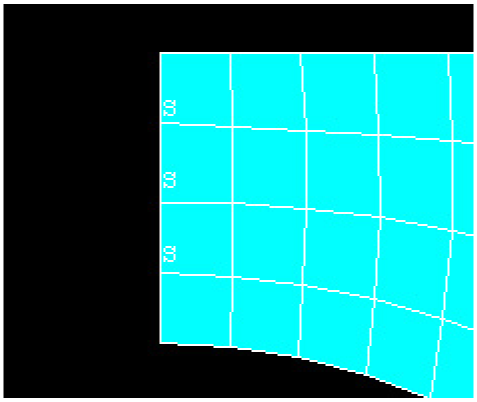
Next, we step up to the plate to define the displacement constraints and loads. Recall that in ANSYS terminology, the displacement constraints are also "loads". As in the truss tutorial, we'll apply the loads to the geometry rather than the mesh. That way we won't have to reapply the loads on changing the mesh.

Apply Symmetry Boundary Conditions

ANSYS provides the option of applying a "symmetry boundary condition" along lines of symmetry.

Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > Symmetry B.C. > On Lines

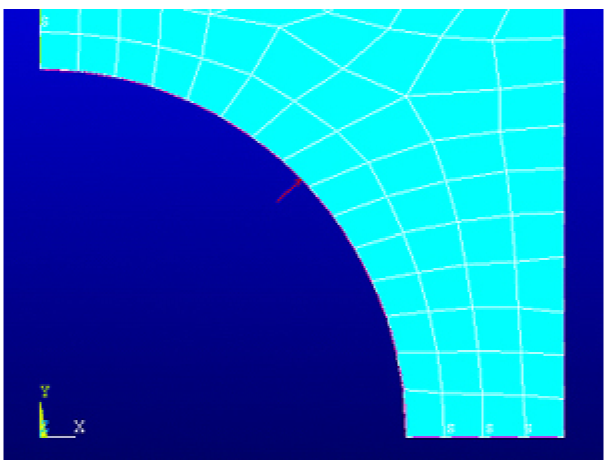
Select the straight lines corresponding to the left and bottom edges (which are the lines of symmetry for this problem) by clicking on them. Click ***OK*** in the pick menu. The symbol *s* appears along these lines indicating that the symmetry B.C. is applied along these lines.



Apply Pressure

Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Pressure > On Lines

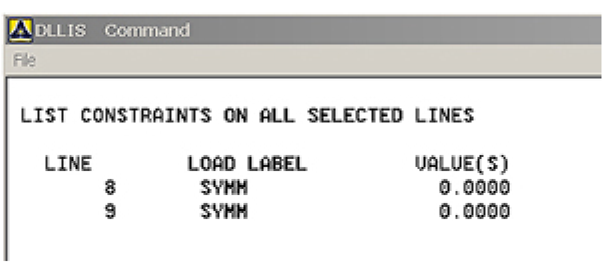
Select the circular arc and click ***OK***. This brings up the *Apply Pressure on Lines* menu. Enter *p* for ***Value*** and click ***OK***. A single red arrow denotes the pressure and the direction in which it is acting.



Check Loads

Let's check that the displacement constraints have been applied correctly.

Utility Menu > List > Loads > DOF constraints > On All Lines


DLLIS Command
File

LIST CONSTRAINTS ON ALL SELECTED LINES

LINE	LOAD LABEL	VALUE(S)
8	SYMM	0.0000
9	SYMM	0.0000

Symmetry BCs are applied on lines 8 and 9. Turn on line numbering:

Utility Menu > PlotCtrls > Numbering

Turn on ***Line numbers*** and click ***OK***. Are lines L8 and L9 the ones on which you want the symmetry BCs?

Similarly, check that the pressure is applied correctly using **Utility Menu > List > Loads > Surface Loads > On All Lines**. Note that ***VALI*** and ***VALJ*** would be different if the applied pressure were linearly varying along the line.

Turn off line numbering: **Utility Menu > PlotCtrls > Numbering**. Turn off Line numbers and click ***OK***.

Save Your Work

Toolbar > SAVE_DB

Go to Step 7: Solve

See and rate the complete Learning Module

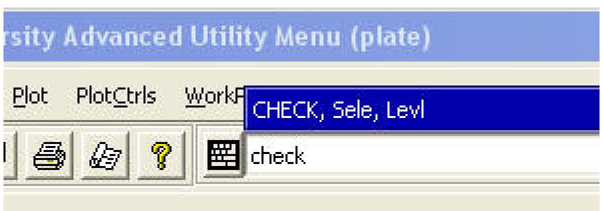
Go to all ANSYS Learning Modules

Step 7: Solve!

Enter solution module:

Main Menu > Solution

Enter `check` in the *Input* window and press Enter.



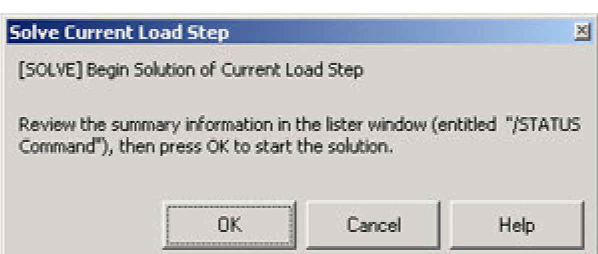
If the problem has been set up correctly, there will be no errors or warnings reported. If you look in the *Output* window, you should see the message: The analysis data was checked and no warnings or errors were found.

Main Menu > Solution > Solve > Current LS

Recall from the truss tutorial that this solves the current load step (LS) i. e. the current loading conditions. In this problem also, there is only one load step.

Review the information in the */STATUS Command* window. Close this window.

Click ***OK*** in *Solve Current Load Step* menu.



ANSYS performs the solution and a window should pop up saying "Solution is done!". Congratulations! Close the window.

Verify that ANSYS has created a file called *plate.rst* in your working directory. This file contains the results of the (previous) *solve*.

Go to Step 8: Postprocess the results

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Step 8: Postprocess the Results

Enter the postprocessing module to analyze the solution.

Main Menu > General Postproc

Plot Deformed Shape

Main Menu > General Postproc > Plot Results > Deformed Shape

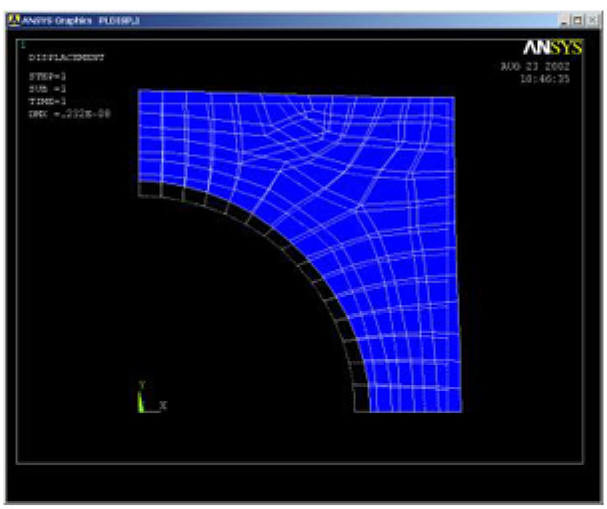
Select ***Def + undeformed*** and click ***OK***.

This plots the deformed and undeformed shapes in the *Graphics* window. The maximum deformation *DMX* is 0.232E-08m as reported in the *Graphics* window. Note that the deformation is magnified in the plot so as to be visible.

The deformation would be better visible if the foreground and background were not of the same color. Turn off the background:

Utility Menu > PlotCtrls > Style > Background > Display Picture Background

To get the background back, you just have to select this again.



Animate the deformation:

Utility Menu > PlotCtrls > Animate > Deformed Shape...

Select ***Def + undeformed*** and click ***OK***. Select ***Forward Only*** in the *Animation Controller*.

The left and bottom edges move parallel to themselves which means that the full deformed plate is also symmetric about these edges. This shows that the symmetry boundary condition at these edges is imposed correctly. The circular edge of the hole moves outward which is what one would expect from the outward pressure acting along it. Thus, the deformation of the structure agrees with the applied boundary conditions and matches with what one would expect from intuition.

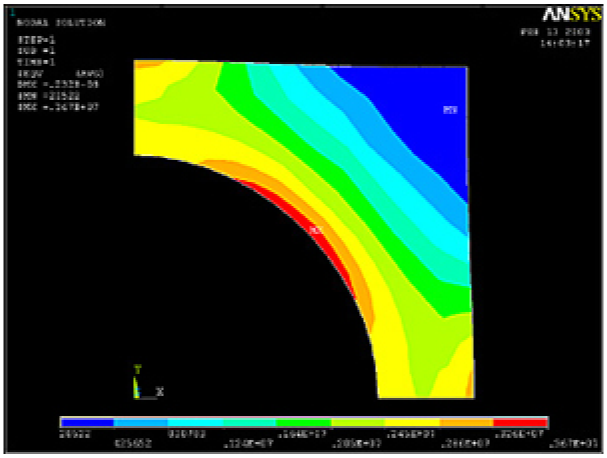
Close the *Animation Controller*.

Plot Nodal Solution of von Mises Stress

To display the von Mises stress distribution as *continuous* contours, select

Main Menu > General Postproc > Plot results > Contour Plot > Nodal Solu

Select ***Nodal Solution > Stress > von Mises stress*** and click ***OK***.



The contour plot will show you the locations of the maximum and minimum values with the labels *MX* and *MN*, respectively. Are these locations where you expect them? *SMX* and *SMN* values reported in the *Graphics* window are the corresponding maximum and minimum stress values.

The diagonal is an additional line of symmetry. How symmetric is your result about the diagonal?

Save this plot to a file:

Utility Menu > PlotCtrls > Hard Copy > To File

Select the file format you want and type in a filename of your choice under ***Save to:*** and click ***OK***. Check that the file has been created in your working directory.

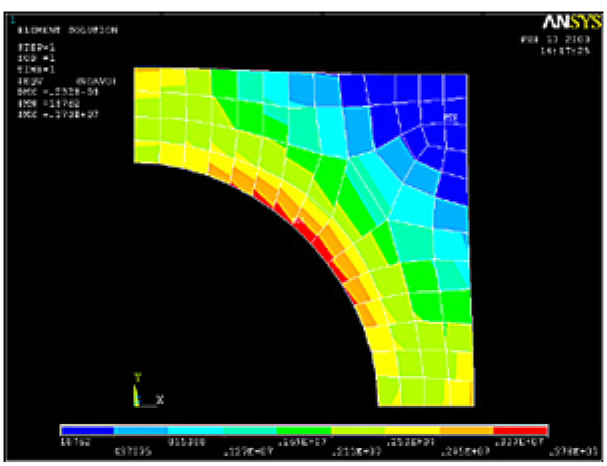
When you plot the "Nodal Solution", ANSYS obtains a continuous distribution as follows:
1. It determines the average at each node of the values of all elements connected to the node.
2. Within each element, it linearly interpolates the average nodal value obtained in the previous step.

Plot Element Solution of von Mises Stress

To obtain results without nodal averaging, select

Main Menu > General Postproc > Plot results > Contour Plot > Element Solu

Select ***Element Solution > Stress > von Mises stress*** and click ***OK***. This displays the von Mises stress results as discontinuous element contours.



Save this plot to a file: **Utility Menu > PlotCtrls > Hard Copy > To File**

Element solution contours are determined by linear interpolation within each element but no nodal averaging is performed. The discontinuity between contours of adjacent elements is an indication of the gradient across elements. The inter-element discontinuities in our solution are relatively small compared to the stress levels. This indicates that the mesh resolution is reasonably good.

Query Results

To determine the value of the first principal stress $_1$ at a selected location, select

Main Menu > General Postproc > Query Results > Subgrid Solu

This brings up the *Query Subgrid Solution Data* menu. Select ***Stress*** from the left list, ***1st principal S1*** from the right list and click ***OK***.

This brings up the *pick* menu. You can click on any location in the geometry and ANSYS will print the $_1$ value at that location. Try querying the values at a few locations. Note that the coordinates of the picked location and the corresponding solution value are reported in the *pick* menu.

Cancel the *pick* menu.

Go to Step 9: Validate the results

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Step 9: Validate the results

It is ***very important*** that you take the time to check the validity of your solution. This section leads you through some of the steps you can take to validate your solution.

Simple Checks

Does the deformed shape look reasonable and agree with the applied boundary conditions? We checked this in step 8.

Do the reactions at the supports balance the applied forces for static equilibrium? To check this, select

Main Menu > General Postproc > List Results > Reaction Solu

Select ***All struc forc F*** for ***Item to be listed*** and click ***OK***.

The total reaction force in the x-direction is -7000 N.

Applied force = (pressure) x (projected distance in x-direction of the line along which the constant pressure acts) = (p) (r) = 7000 N in positive x-direction.

So the reaction cancels out the applied force in the x-direction. Similarly, you can check that this is true in the y-direction also.

Refine Mesh

Let's repeat the calculations on a mesh with overall element size level under *SmartSize* set to 4 instead of 5 and compare the results on the two meshes. Delete the current mesh:

Main Menu > Preprocessor > Meshing > Mesh Tool

Select ***Clear*** under ***Mesh***: and ***Pick All*** in the *pick* menu. The mesh is deleted.

Set the overall element size level under *SmartSize* to 4 by dragging the slider to the left. Click on ***Mesh*** and ***Pick All***.

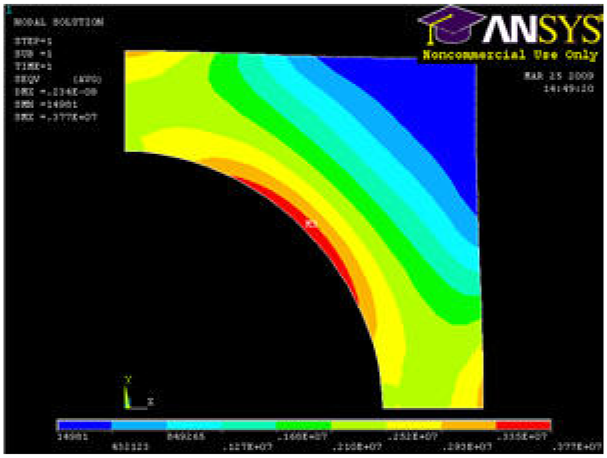
In the *Output* window, check how many elements are contained in this mesh? Your new mesh should have 320 quadrilateral elements.

Obtain a new solution: **Main Menu > Solution > Solve > Current LS**

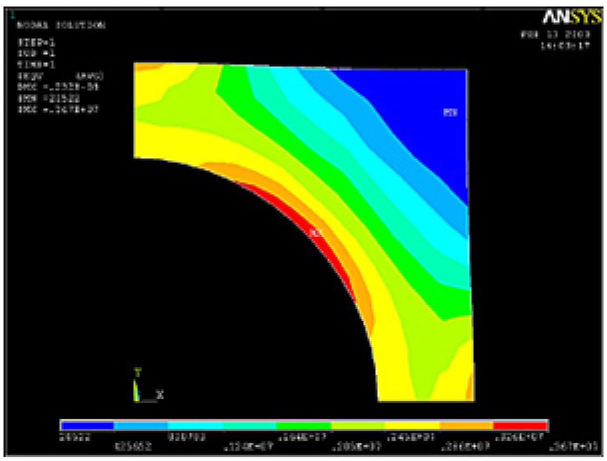
Plot nodal solution of the von Mises stress:

Main Menu > General Postproc > Plot results > Contour Plot > Nodal Solu

Select ***Nodal Solution > Stress > von Mises stress*** and click ***OK***



Compare this with the von Mises contours for the previous mesh:



The two results compare well with the finer mesh contours being smoother as expected. Compare the maximum stress and displacement values:

	Coarser Mesh	Finer Mesh
DMX	0.232e-8m	0.234e-8m
SMX	3.64MPa	3.77MPa

The maximum displacement value changes by less than 1% and the maximum von Mises stress value by less than 3%. This indicates that the meshes used provide adequate resolution.

Exit ANSYS

Utility Menu > File > Exit

Select ***Save Everything*** and click ***OK***.

Reference

Cook, R.D., Malkus, D.S., Plesha, M.E., and Witt, R.J., *Concepts and Applications of Finite Element Analysis*, Fourth Edition, John Wiley and Sons, Inc., 2002.

Go to Problem Set 1

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Problem Set 1

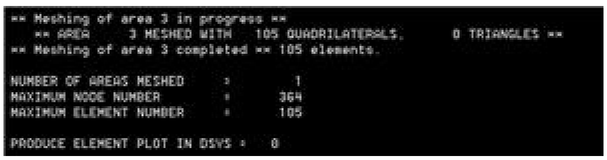
Problem Statement

We used a 4-node quad element (PLANE42) in the tutorial. ANSYS also offers a 8-node quad element (PLANE82). Re-solve the tutorial problem using the PLANE82 element. Compare plots of the nodal and element solution of the von Mises stress for the two cases. You may use either mesh for this problem (although the final results presented here are done using the coarser mesh).

Hints

Look at the steps and think about which ones you have to change.

When you remesh the object, notice the following changes:



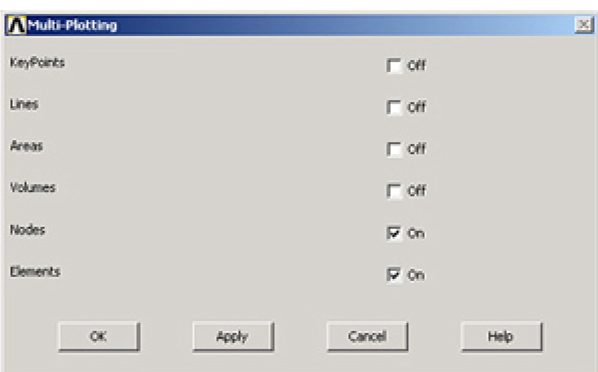
Higher Resolution Image

The number of nodes has increased!

To see why, do:

Utility Menu > PlotCtrls > Multi-plot Ctrls ...

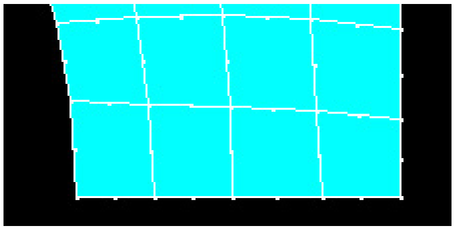
Click ***OK***. Then on the *Multi-Plotting Window* that comes up, deselect everything but ***Nodes*** and ***Elements***.



Click ***OK***.

Then go to ***Plot > Multi-Plots***

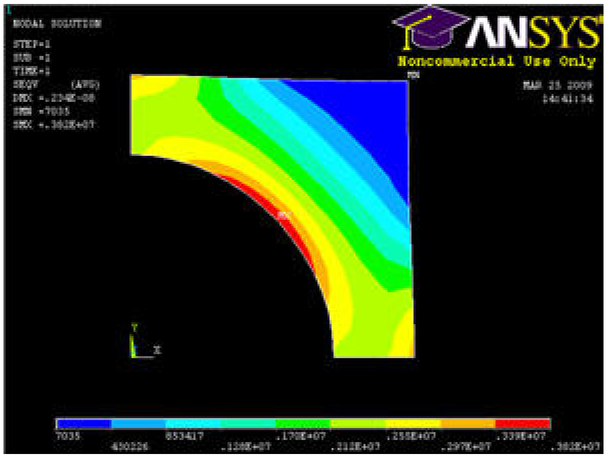
In the *Graphics Window*, you will now see the nodes in between the lines. There are 8 points for each quadrilateral area instead of the four we had before!



Final Result

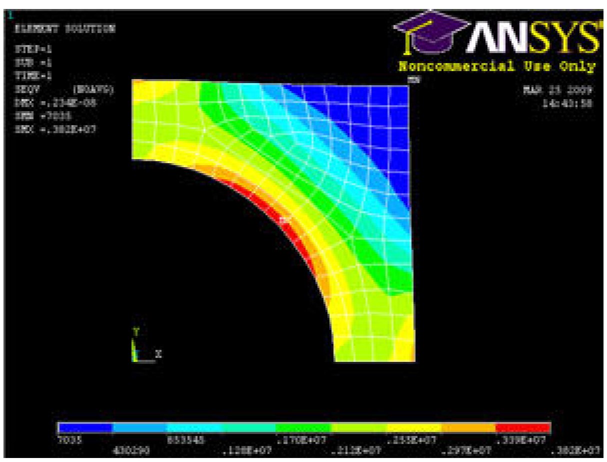
Here are the Nodal and Element Solutions you should have gotten:

Nodal Solution



Higher Resolution Image

Element Solution



Higher Resolution Image

Back to Problem Specification

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Go to all ANSYS Learning Modules