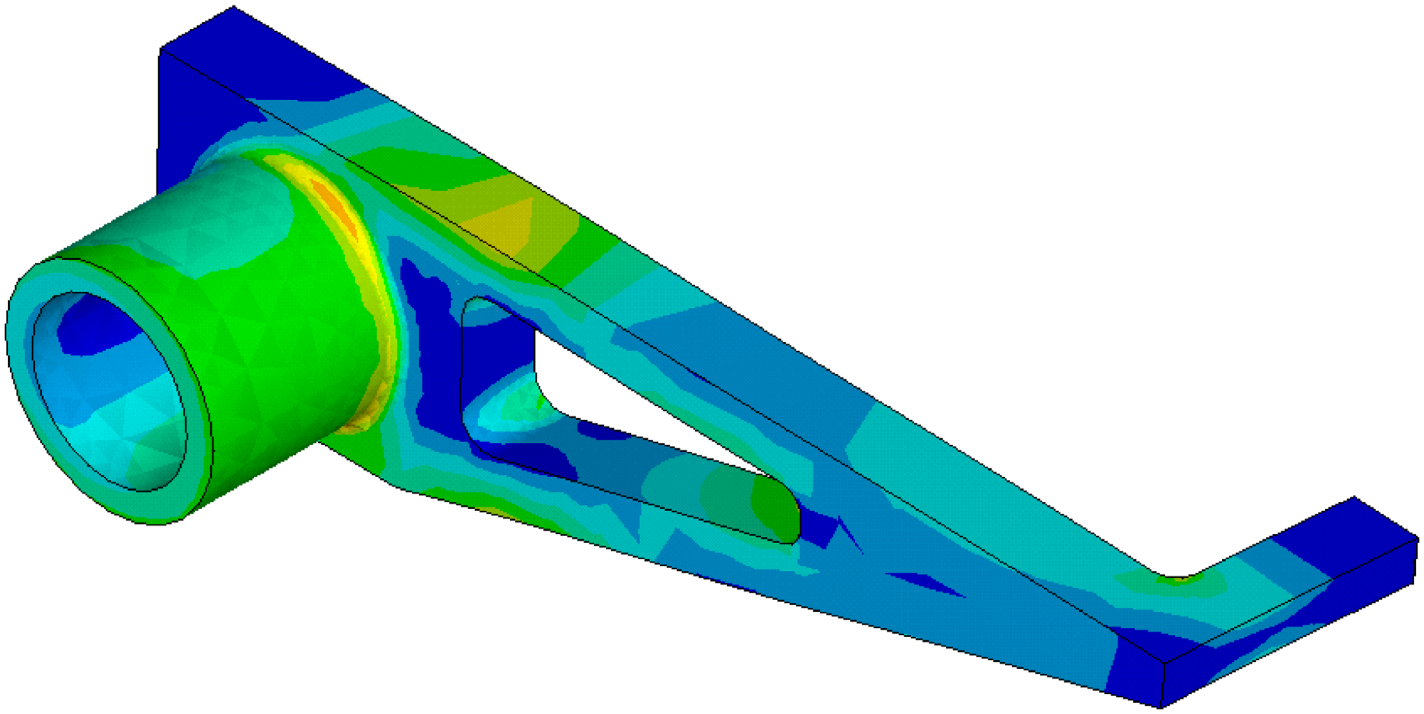
ANSYS® Tutorial Release 2020

Structural & Thermal Analysis Using the ANSYS® Mechanical APDL Release 2020 Environment



Kent L. Lawrence

SDC PUBLICATIONS
Better Textbooks. Lower Prices.
www.SDCpublications.com

Visit the following websites to learn more about this book:

SDC Publications

amazon.com

Google books

BARNES&NOBLE

Lesson 2

Plane Stress
Plane Strain

2-1 OVERVIEW

Plane stress and plane strain problems are an important subclass of general three-dimensional stress and strain problems. The tutorials in this lesson demonstrate:

- ♦ Solving planar stress concentration problems.
- ♦ Evaluating potential inaccuracies in the solutions.
- ♦ Using the various ANSYS 2D element formulations.

2-2 INTRODUCTION

It is possible for an object such as the one on the cover of this book to have six components of stress when subjected to arbitrary three-dimensional loadings. When referenced to a Cartesian x, y, z coordinate system these components of stress are:

Normal Stresses $\sigma_x,\ \sigma_y,\ \sigma_z$

Shear Stresses $\tau_{xy},\ \tau_{yz},\ \tau_{zx}$

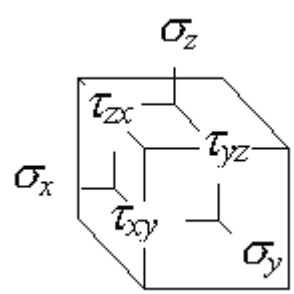


Figure 2-1 Stresses in 3 dimensions.

In general, the analysis of such objects requires three-dimensional modeling as discussed in Lesson 4. However, two-dimensional models are often easier to develop, easier to solve and can be employed in many situations **if** they can accurately represent the behavior of the object under loading.

A state of **Plane Stress** exists in a thin object loaded in the plane of its largest dimensions. Let the *X-Y* plane be the plane of analysis. The non-zero stresses σ_x, σ_y, and τ_{xy} lie in the *X* - *Y* plane and do not vary in the *Z* direction. Further, the other stresses (σ_z, τ_{yz} , and τ_{zx}) are all zero for this kind of geometry and loading. A thin beam loaded in its plane and a spur gear tooth are good examples of plane stress problems.

ANSYS provides a 6-node planar triangular element along with 4-node and 8-node quadrilateral elements for use in the development of plane stress models. We will use both triangles and quads in solution of the example problems that follow.

2-3 PLATE WITH CENTRAL HOLE

To start off, let's solve a problem with a known solution so that we can check our computed results as well as our understanding of the FEM process. The problem is that of a tensile-loaded thin plate with a central hole as shown in Figure 2-2.

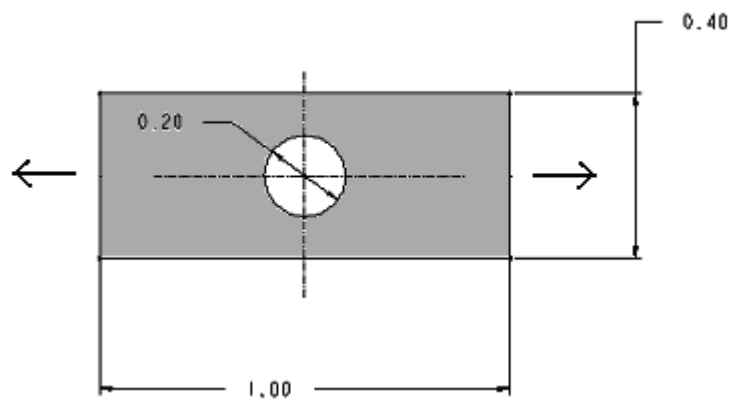


Figure 2-2 Plate of 0.01 meter thickness with a central hole.

The **1.0 m x 0.4 m** plate has a **thickness** of **0.01 m**, and a central hole **0.2 m** in **diameter**. It is made of steel with material properties; **elastic modulus**, $\boldsymbol{E}$ **= 2.07 x 10^{11} N/m^2** and **Poisson's ratio**, $\boldsymbol{\nu}$ **= 0.29**. We apply a **horizontal tensile loading** in the form of a **pressure** $\boldsymbol{p}$ **= -1.0 N/m^2** along the vertical edges of the plate.

Because holes are necessary for fasteners such as bolts, rivets, etc, the need to know stresses and deformations near them occurs very often and has received a great deal of study. The results of these studies are widely published, and we can look up the stress concentration factor for the case shown above. Before the advent of suitable computation methods, the effect of most complex stress concentration geometries had to be evaluated experimentally, and many available charts were developed from experimental results.

The uniform, homogeneous plate above is symmetric about horizontal axes in both geometry and loading. This means that the state of stress and deformation below a

horizontal centerline is a mirror image of that above the centerline, and likewise for a vertical centerline. We can take advantage of the symmetry and, by applying the correct boundary conditions, use only a quarter of the plate for the finite element model. For small problems using symmetry may not be too important; for large problems it can save modeling and solution efforts by eliminating one-half or a quarter or more of the work.

Place the origin of *X-Y* coordinates at the center of the hole. If we pull on both ends of the plate, points on the centerlines will move along the centerlines but not perpendicular to them. This indicates the appropriate displacement conditions to use as shown below.

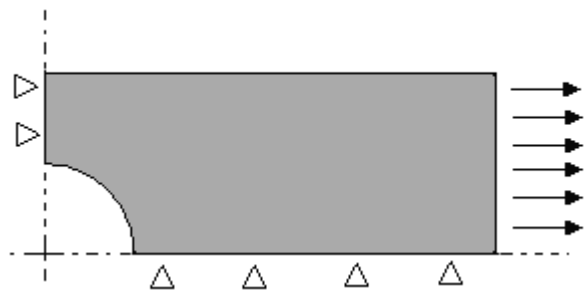


Figure 2-3 Quadrant used for analysis.

In Tutorial 2A we will use ANSYS to determine the maximum horizontal stress in the plate and compare the computed results with the maximum value that can be calculated using tabulated values for stress concentration factors. Interactive commands will be used to formulate and solve the problem.

2-4 TUTORIAL 2A - PLATE

Objective: Find the **maximum axial stress** in the plate with a central hole and compare your result with a computation using published stress concentration factor data.

PREPROCESSING

1. Start ANSYS, select the **Working Directory** where you will store the files associated with this problem. Also set the **Jobname** to **Tutorial2A** or something memorable and provide a **Title**.

(If you want to make changes in the Jobname, working Directory, or Title after you've started ANSYS, use **File > Change Jobname** or **Directory** or **Title**.)

Select the **six node triangular element** to use for the solution of this problem.

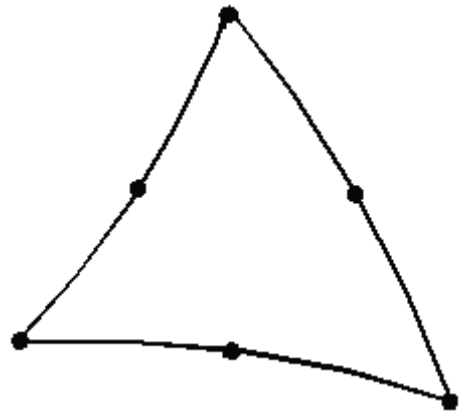


Figure 2-4 Six-node triangle.

The six-node triangle is a sub-element of the eight-node quadrilateral.

2. Main Menu > Preprocessor > Element Type > Add/Edit/Delete > Add > Structural Solid > Quad 8node 183 > OK

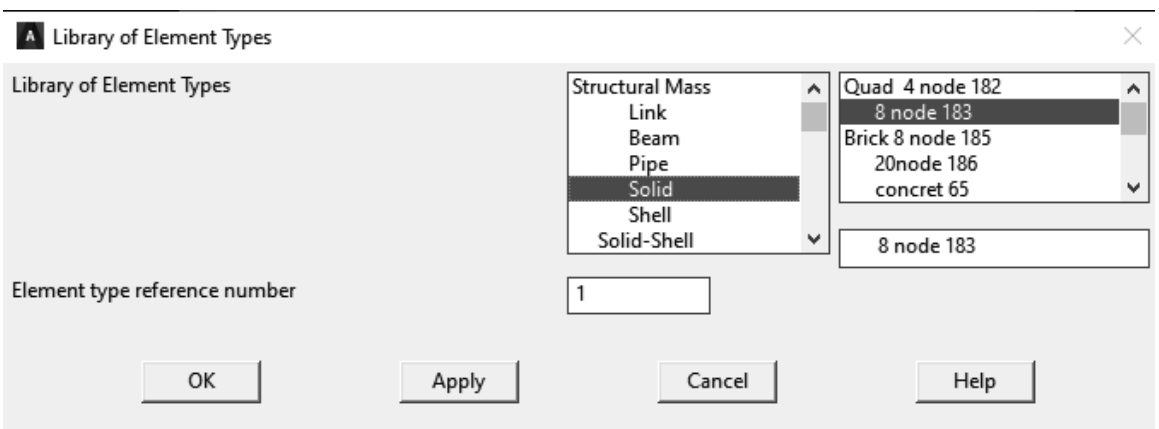


Figure 2-5 Element selection.

Select the triangle **option** and the option to define the plate thickness, otherwise a unit thickness is used.

3. Options (Element shape K1) **> Triangle,**

Options (Element behavior K3) **> Plane strs w/thk > OK > Close**

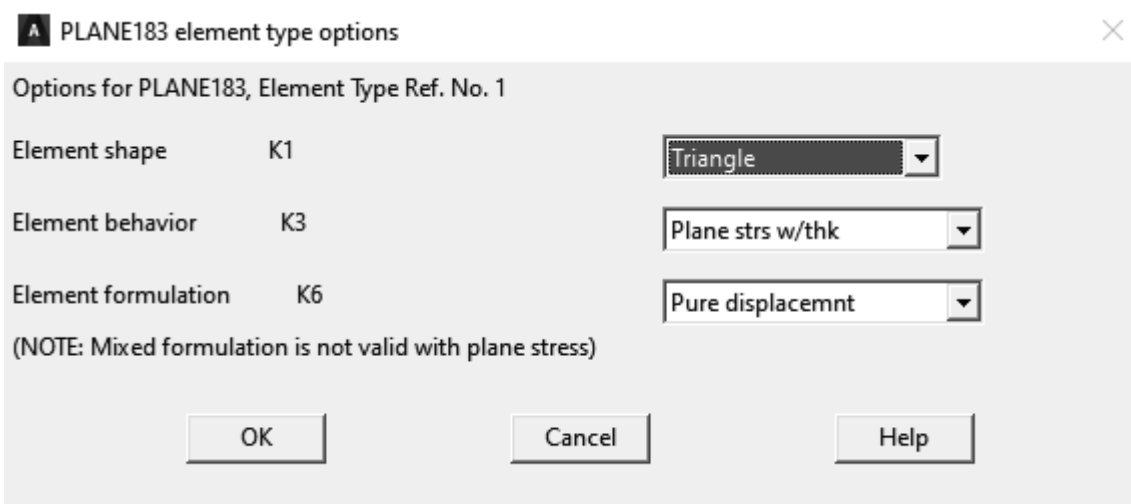


Figure 2-6 Element options.

4. Main Menu > Preprocessor > Real Constants > Add/Edit/Delete > Add > OK

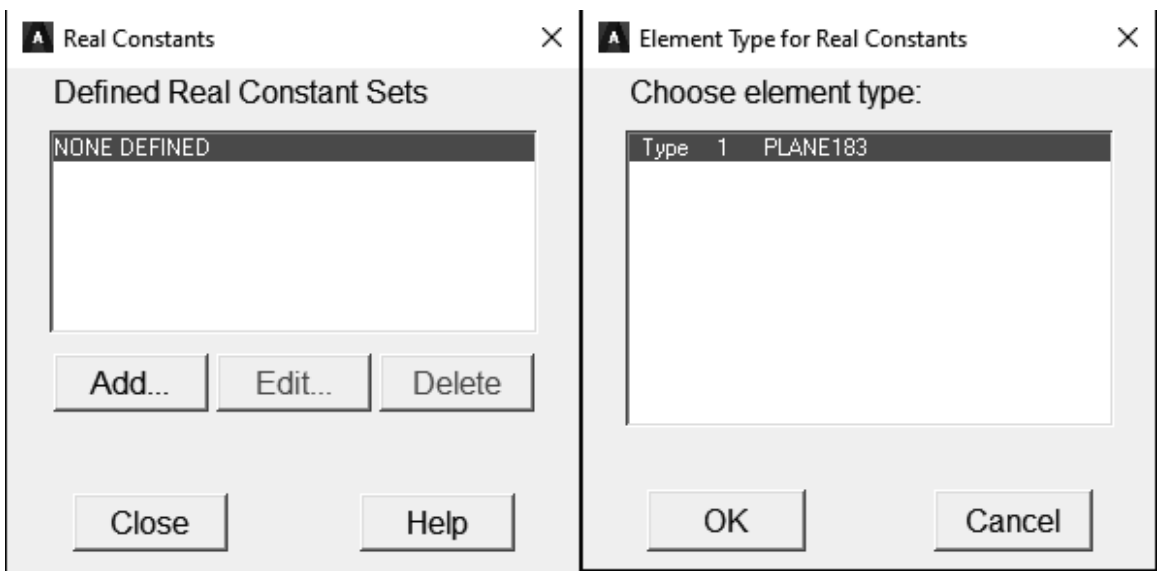


Figure 2-7 Real constants.

(Enter the plate thickness of 0.01 m.) >Enter **0.01 > OK > Close**

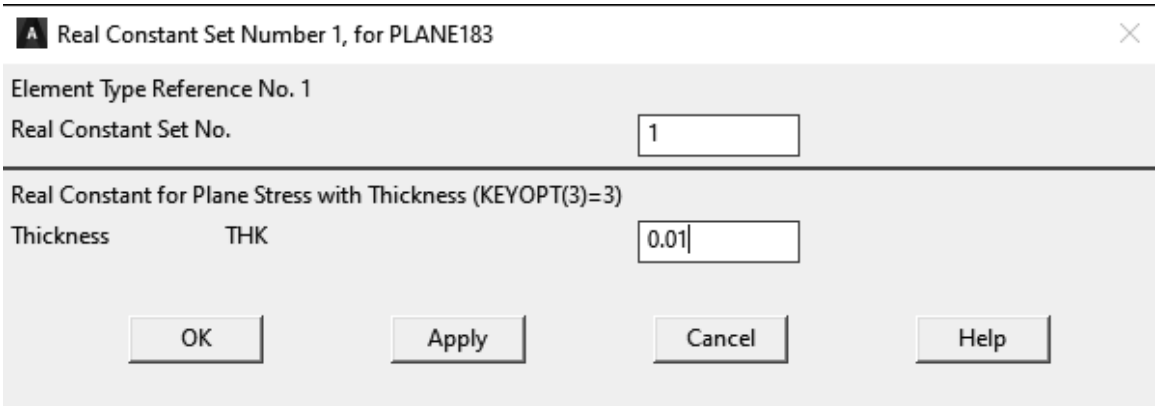


Figure 2-8 Enter the plate thickness.

Enter the material properties.

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

Material Model Number 1, click **Structural > Linear > Elastic > Isotropic**

Enter **EX = 2.07E11** and **PRXY = 0.29 > OK** (**Close** the Define Material Model Behavior window.)

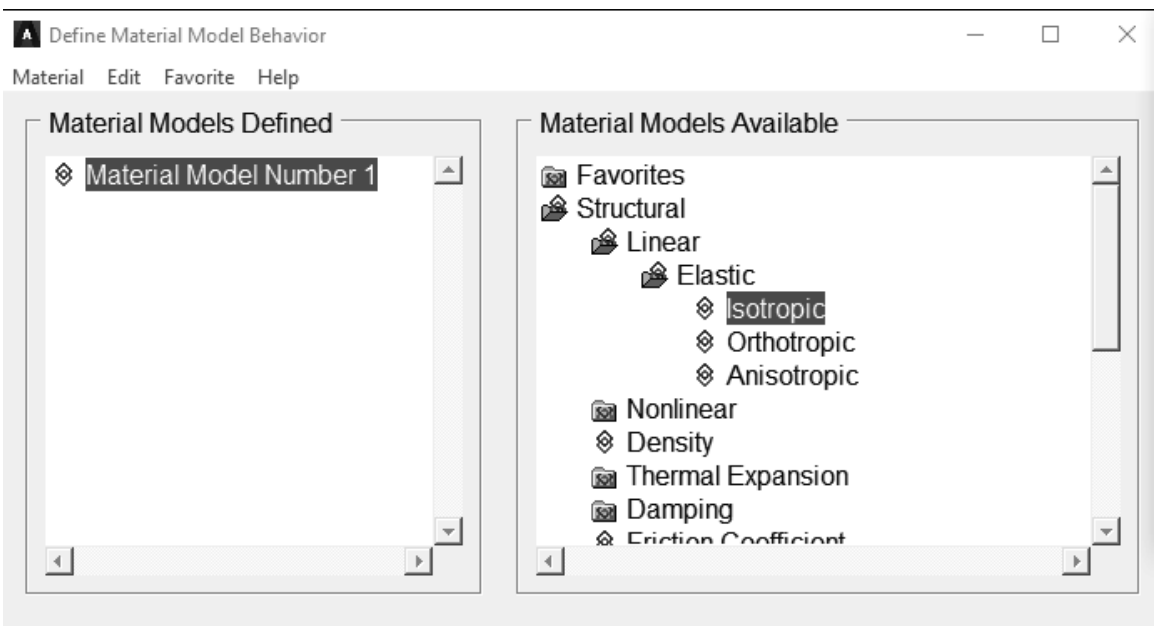


Figure 2-9 Material Models.

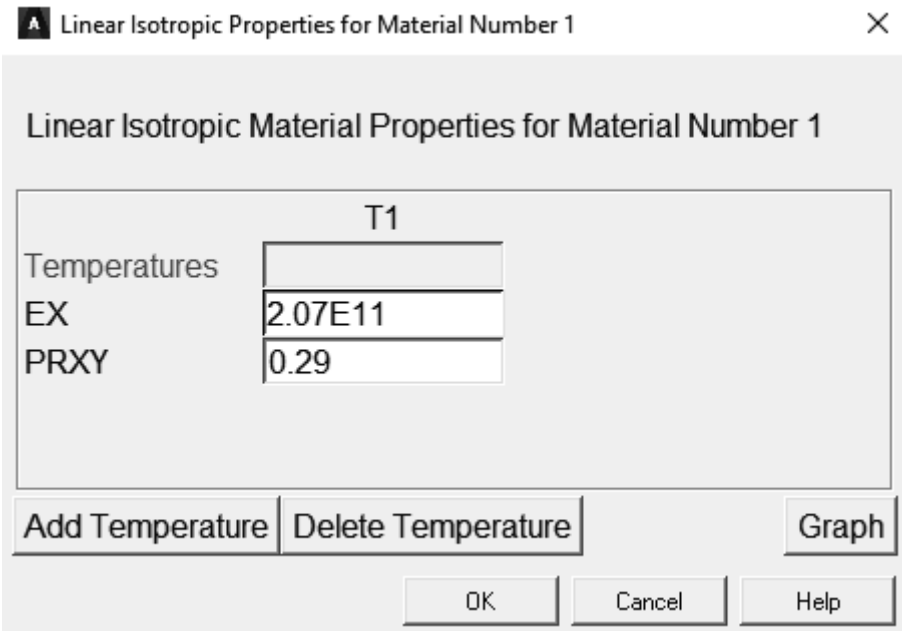


Figure 2-10 Material Properties.

Create the geometry for the upper right quadrant of the plate by **subtracting** a 0.2 m diameter circle from a 0.5 x 0.2 m rectangle. Generate the rectangle first.

6. Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By 2 Corners

Enter (lower left corner) **WP X = 0.0**, **WP Y = 0.0** and **Width = 0.5, Height = 0.2 > OK**

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

Enter **WP X = 0.0**, **WP Y = 0.0** and **Radius = 0.1 > OK**

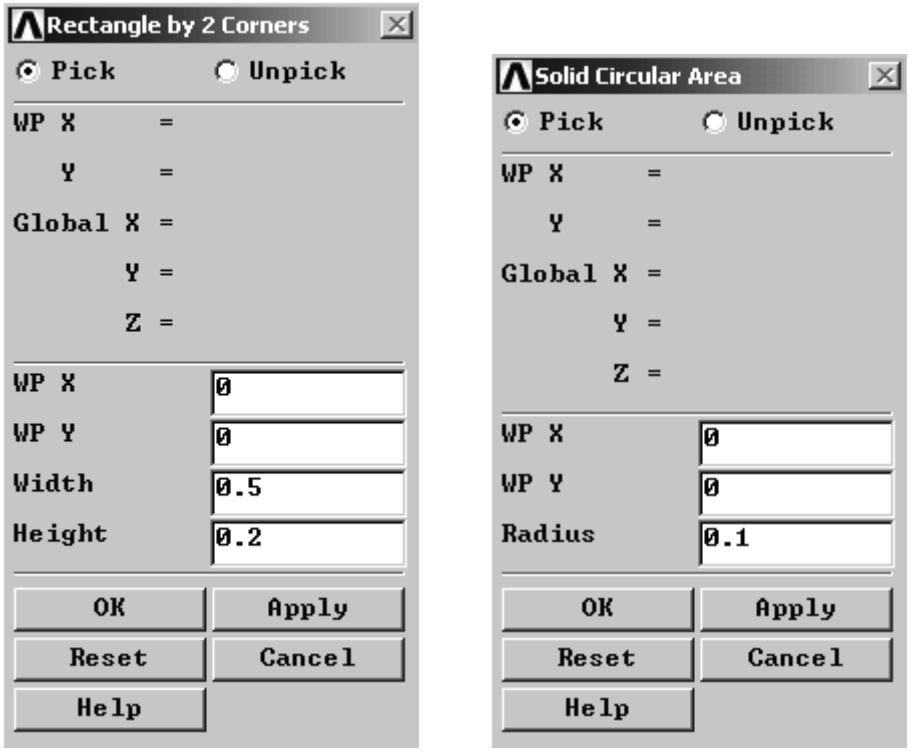


Figure 2-11 Create areas.

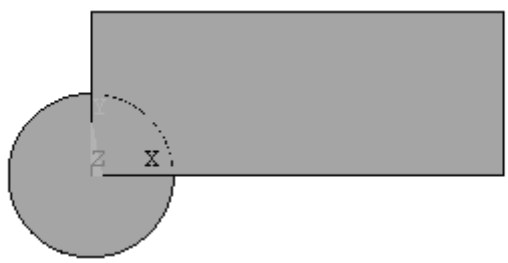


Figure 2-12 Rectangle and circle.

Now **subtract** the circle from the rectangle. (Read the messages in the window at the bottom of the screen as necessary.)

8. Main Menu > Preprocessor > Modeling > Operate > Booleans > Subtract > Areas > Pick the **rectangle > OK,** then pick the circle **> OK** (Use **Raise Hidden** and **Reset Picking** as necessary.)

Figure 2-13 Geometry for quadrant of plate.

Create a mesh of triangular elements over the quadrant area.

9. Main Menu > Preprocessor > Meshing > Mesh > Areas > Free Pick the quadrant **> OK** (Note that your mesh may differ from that shown here.)

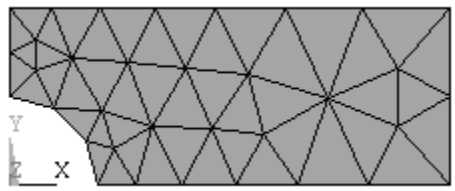


Figure 2-14 Triangular element mesh.

Apply the displacement boundary conditions and loads to the **geometry (lines)** instead of the nodes as we did in the previous lesson. These conditions will be applied to the FEM model when the solution is performed.

10. Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Lines Pick the left edge of the quadrant **> OK > UX = 0. > OK**
11. Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Lines Pick the bottom edge of the quadrant **> OK > UY = 0. > OK**

Apply the loading.

12. Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Pressure > On Lines. Pick the right edge of the quadrant **> OK > Pressure = -1.0 > OK** (A positive pressure would be a compressive load, so we use a *negative* pressure. The pressure is shown by the two arrows.)

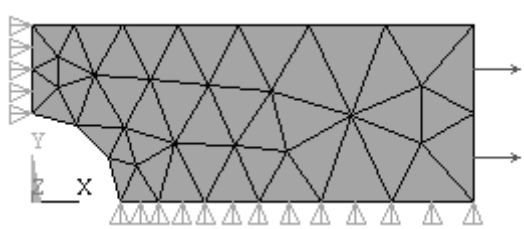


Figure 2-15 Model with loading and displacement boundary conditions.

The model-building step is now complete, and we can proceed to the solution. First, to be safe, save the model.

13. Utility Menu > File > Save as Jobname.db (Or **Save as ….** ; use a new name)

SOLUTION

The interactive solution proceeds as illustrated in the tutorials of Lesson 1.

14. Main Menu > Solution > Solve > Current LS > OK

The **/STATUS Command** window displays the problem parameters and the **Solve Current Load Step** window is shown. Check the solution options in the /STATUS window and if all is OK, select **File > Close**

In the **Solve Current Load Step** window, select **OK**, and when the solution is complete, **Close** the **'Solution is Done!'** window.

POSTPROCESSING

We can now **plot the results** of this analysis and also **list the computed values**. First examine the **deformed shape**. **PlotCtrls > Symbols > All Applied BCs**

15. Main Menu > General Postproc > Plot Results > Deformed Shape > Def. + Undef. > OK

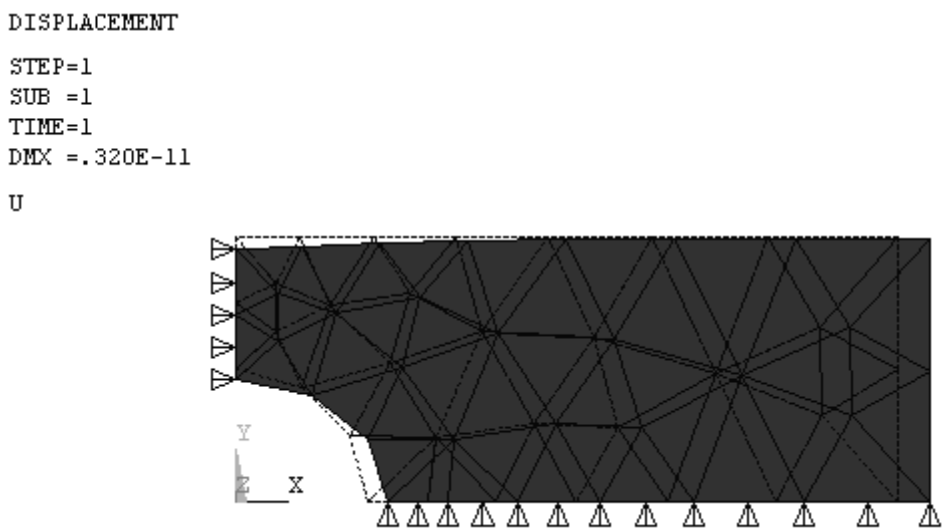


Figure 2-16 Plot of Deformed shape.

The deformed shape looks correct. (The undeformed shape is indicated by the dashed lines.) The right end moves to the right in response to the tensile load in the *X* direction, the circular hole ovals out, and the top moves down because of Poisson's effect. Note that the element edges on the circular arc are represented by straight lines. This is an artifact of the plotting routine not the analysis. The six-node triangle has curved sides, and if you pick on a mid-side of one these elements, you will see that a node is placed on the curved edge.

The maximum displacement is shown on the graph legend as 0.32e-11 which seems reasonable. The **units** of displacement are **meters** because we employed meters and N/m^2 in the problem formulation. Now plot the stress in the *X* direction.

Main Menu > General Postproc > Read Results > First Set

16. **Main Menu > General Postproc > Plot Results > Contour Plot > Element Solu > Stress > X-Component of stress > OK**

Use **PlotCtrls > Symbols [/PSF] Surface Load Symbols** (set to **Pressures**) and **Show pre and convect as** (set to **Arrows**) to display the pressure loads.

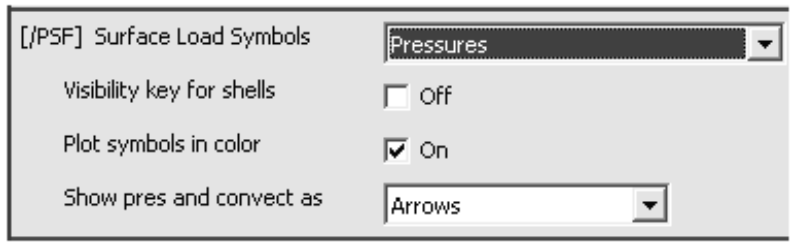


Figure 2-17 Surface load symbols.

Also select **Display All Applied BCs**

NOTE: We are plotting the **ELEMENT SOLUTION**

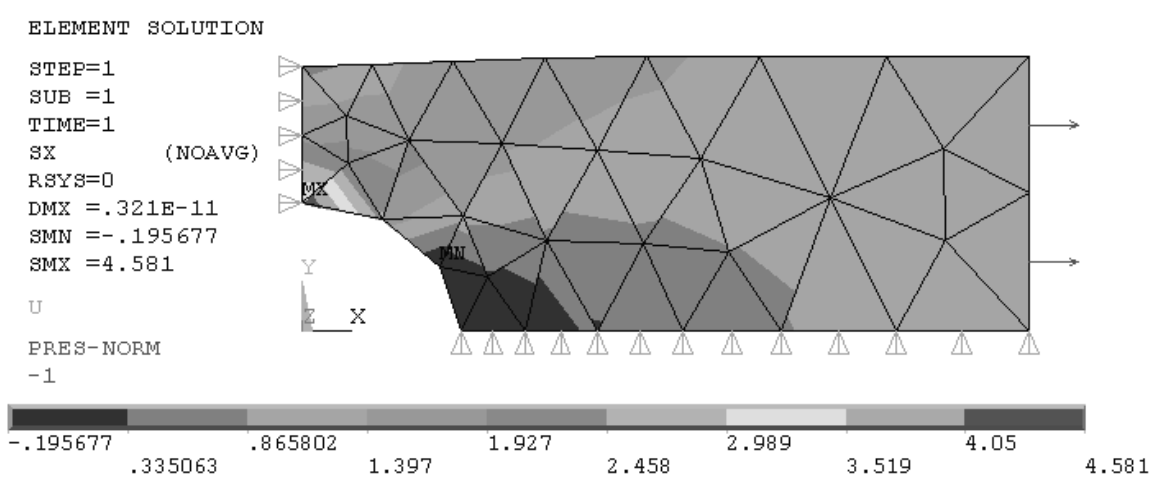


Figure 2-18 Element SX stresses.

The minimum, **SMN**, and maximum, **SMX,** stresses as well as the color bar legend give an overall evaluation of the σ_x (SX) stress state. We are interested in the maximum stress at the hole. Use the **Zoom** to focus on the area with highest stress. (Your meshes and results may differ a bit from those shown here.)

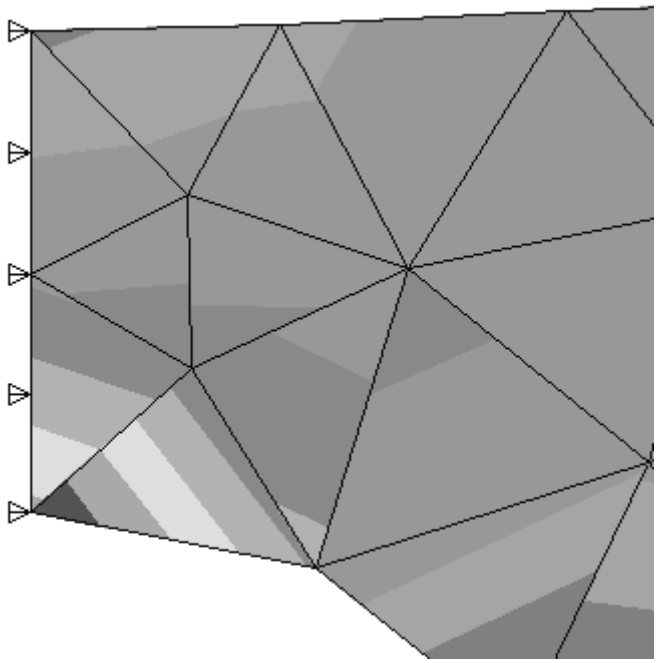


Figure 2-19 SX stress detail.

Stress variations in the actual isotropic, homogeneous plate should be smooth and continuous across elements. The **discontinuities** in the SX stress contours above indicate that the number of elements used in this model is too few to calculate with complete accuracy the stress values near the hole because of the stress gradients there. We **will not accept** this stress solution. More six-node elements are needed in the region near the hole to find accurate values of the stress. On the other hand, in the right half of the model, away from the stress riser, the calculated stress contours are smooth, and SX would seem to be accurately determined there.

It is **important** to note that in the plotting we selected **Element Solu** (Element Solution) in order to look for stress contour discontinuities. If you pick **Nodal Solu** to plot instead, for problems like the one in this tutorial, the stress values will be **averaged** before plotting, and any contour discontinuities (and thus **errors**) will be hidden. If you plot nodal solution stresses you will always see smooth contours.

A word about **element accuracy:** The FEM implementation of the **truss** element is taken directly from solid mechanics studies, and there is no approximation in the solutions for node-loaded truss structures formulated and solved in the ways discussed in Lesson 1.

The **continuum elements** such as the ones for plane stress and plane strain, on the other hand, are normally developed using displacement functions of a polynomial type to represent the displacements within the element. The higher the polynomial, the greater the accuracy. The ANSYS six-node triangle uses a quadratic polynomial and is capable of representing **linear stress and strain variations** within an element. Note the SX plot.

Near stress concentrations the stress gradients vary quite sharply. To capture this variation, the number of elements near the stress concentrations must be increased proportionately.

To obtain more elements in the model, return to the Preprocessor and refine the mesh, first remove the pressure. All elements are subdivided and the mesh below is created.

17. Main Menu > Preprocessor > Loads > Define Loads > Delete > Structural > Pressure > On Lines. Pick the right edge of the quadrant. **Main Menu > Preprocessor > Meshing > Modify Mesh > Refine At > All** (Select **Level of refinement 1**.)

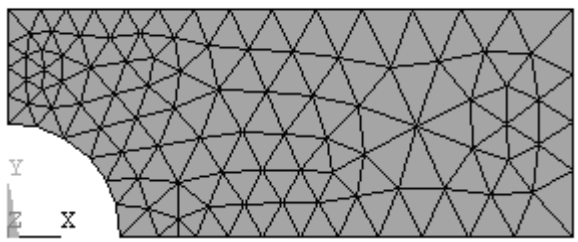


Figure 2-20 Global mesh refinement.

We will also refine the mesh selectively near the hole.

18. Main Menu > Preprocessor > Meshing > Modify Mesh > Refine At > Nodes. (Select the three nodes shown.) **> OK** (Select the **Level of refinement = 1**) **> OK**

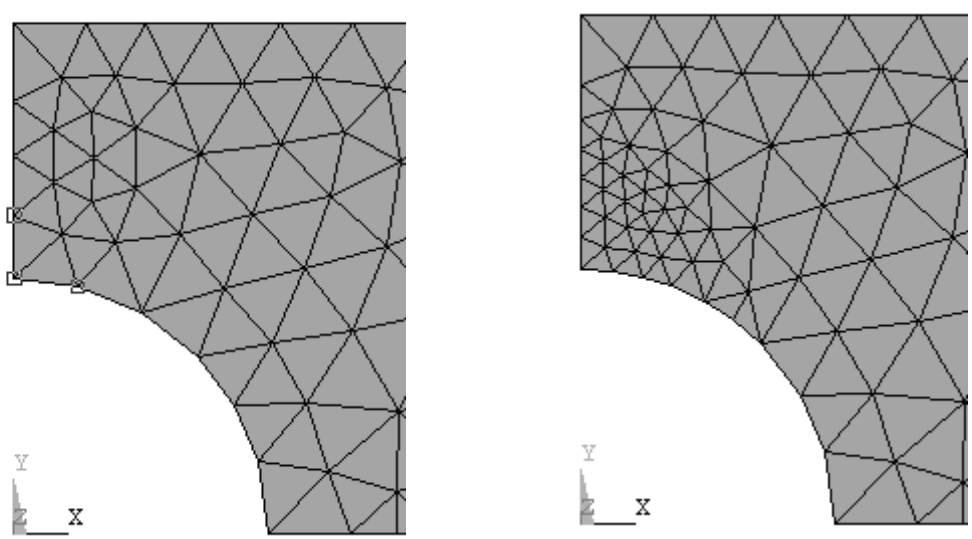


Figure 2-21 Selective refinement at nodes.

(Note: Alternatively you can use **Preprocessor > Meshing > Clear > Areas** to remove all elements and build a completely new mesh. **Plot > Areas** afterwards to view the area again. Note also that too much local refinement can create a mesh with too rapid a transition between fine and coarse mesh regions.)

Reapply the pressure loading, repeat the solution, and replot the stress SX.

19. Main Menu > Solution > Solve > Current LS > OK

Save your work.

20. File > Save as Jobname.db (Or **Save as ….** ; use a new name)

Plot the stresses in the *X* direction.

21. Main Menu > General Postproc > Plot Results > Contour Plot > Element Solu > Stress > X-Component of stress > OK

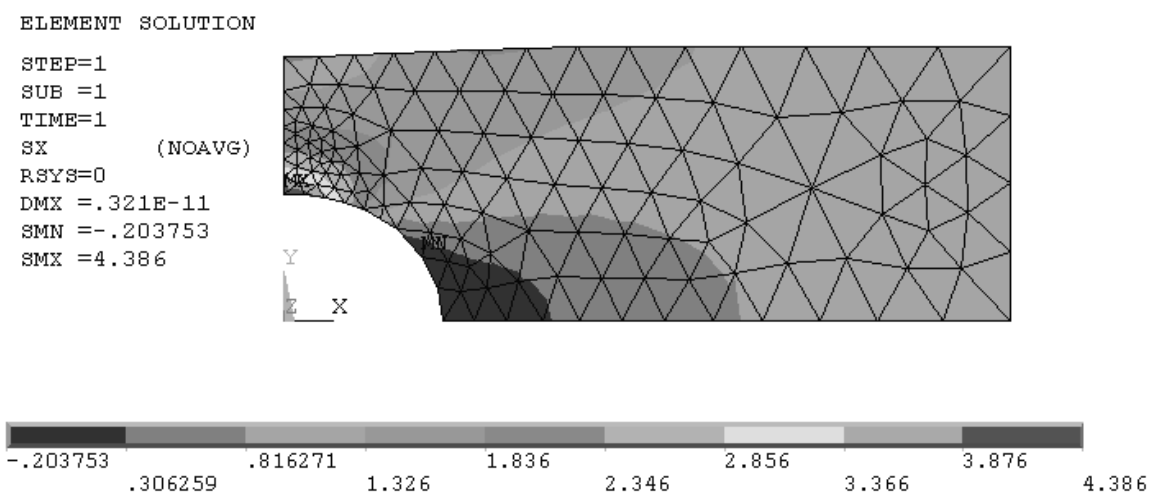


Figure 2-22 SX stress contour after mesh refinement.

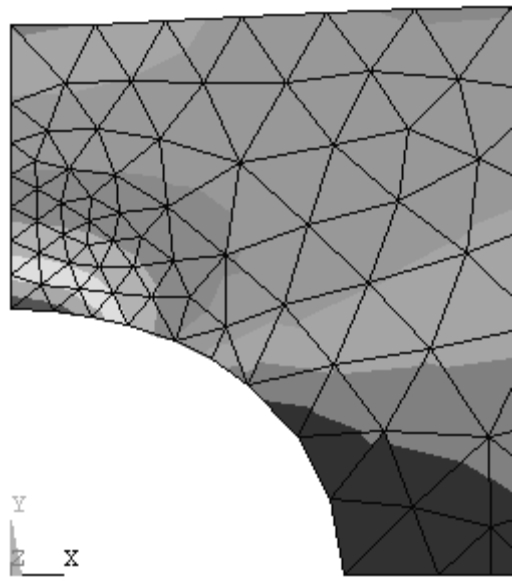


Figure 2-23 SX stress detail contour after mesh refinement.

The element solution stress contours are now smooth across element boundaries, and the stress legend shows a maximum value of 4.386 *Pa*, a 4.3 percent change in the SX stress computed using the previous mesh.

To check this result, find the stress concentration factor for this problem in a text or reference book or from a suitable web site. For the geometry of this example we find K_t = 2.17. We can compute the maximum stress using (K_t)(load)/(net cross sectional area). Using the pressure p = 1.0 *Pa* we obtain. (Load = pressure x area.)

$$\sigma_{x\,MAX} = 2.17 * p * (0.4)(0.01)/[(0.4-0.2)*0.01] = 4.34 Pa$$

The computed maximum value is **4.39 *Pa*** which is around **one percent in error**, *assuming* that the value of K_t is exact.

2-5 THE APPROXIMATE NATURE OF FEM

As mentioned above, the stiffness matrix for the truss elements of Lesson 1 can be developed directly and simply from elementary solid mechanics principles. For continuum problems in two and three-dimensional stress, this is generally no longer possible, and the element stiffness matrices are usually developed by assuming something specific about the characteristics of the displacements that can occur within an element.

Ordinarily this is done by specifying the highest degree of the polynomial that governs the displacement distribution within an element. For **h-method** elements, the polynomial degree depends upon the **number of nodes** used to describe the element, and the interpolation functions that relate displacements within the element to the displacements at the nodes are called **shape functions**. In ANSYS, 2-dimensional problems can be

modeled with **six-node triangles**, **four-node quadrilaterals** or **eight-node quadrilaterals**.

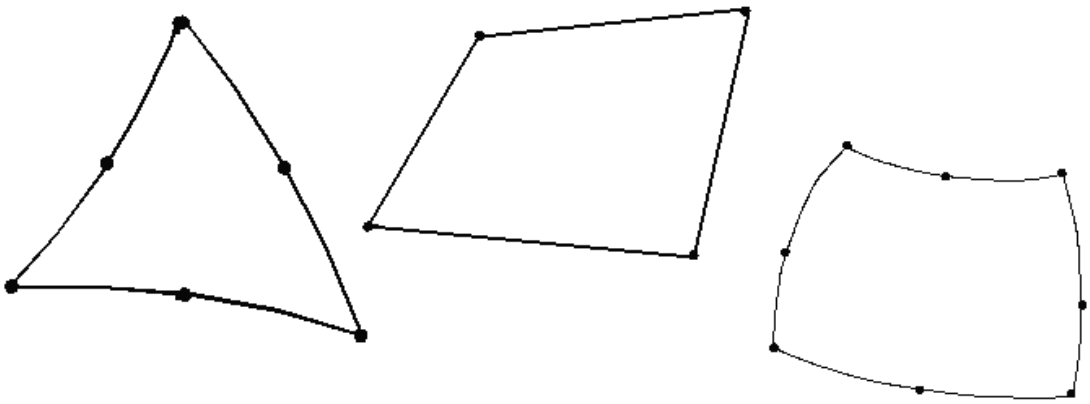


Figure 2-24 Triangular and quadrilateral elements.

The greater the number of nodes, the higher the order of the polynomial and the greater the accuracy in describing displacements, stresses and strains within the element. If the stress is constant throughout a region, a very simple model is sufficient to describe the stress state, perhaps only one or two elements. If there are gradients in the stress distributions within a region, high-degree displacement polynomials and/or many elements are required to accurately analyze the situation.

These comments explain the variation in the accuracy of the results as different numbers of elements were used to solve the problem in the previous tutorial and why the engineer must carefully prepare a model, start with small models, grow the models as understanding of the problem develops and carefully interpret the calculated results. The ease with which models can be prepared and solved sometimes leads to careless evaluation of the computed results.

2-6 ANSYS FILES

The files created during the solution were saved in step 20 of Tutorial 2A. Look in the working directory and you see Tutorial2A files with extensions **BCS**, **db**, **dbb**, **esav**, **full**, **mntr**, **rst**, and **stat**. However, the Tutorial 2A problem can be reloaded using only **Tutorial2A.db**, so if you want to save disk space, you can delete the others.

2-7 ANSYS GEOMETRY

The finite element model consists of elements and nodes and is separate from the geometry on which it may be based. It is possible to build the finite element model without consideration of any underlying geometry as was done in the truss examples of Lesson 1, but in many cases, development of the geometry is the first task.

Two-dimensional geometry in ANSYS is built from **keypoints**, **lines** (straight, arcs, splines), and **areas**. These geometric items are assigned numbers and can be listed, numbered, manipulated, and plotted. The keypoints (2,3,4,5,6), lines (2,3,5,9,10), and area (3) for Tutorial 2A are shown below. (Your numbering may differ.)

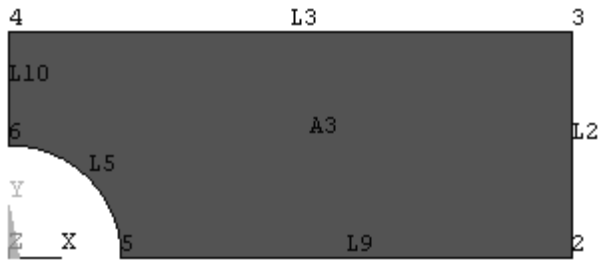


Figure 2-25 Keypoints, lines and areas.

The finite element model developed previously for this part used the area A3 for development of the node/element FEM mesh. The loads, displacement boundary conditions and pressures were applied to the geometry lines. When the solution step was executed, the loads were transferred from the lines to the FEM model nodes. Applying boundary conditions and loads to the geometry facilitates remeshing the problem. The geometry does not change, only the number and location of nodes and elements, and at solution time, the loads are transferred to the new mesh.

Geometry can be created in ANSYS interactively (as was done in the previous tutorial) or it can be created by reading a text file. For example, the geometry of Tutorial 2A can be generated with the following text file using the **File > Read Input from** command sequence. (The keypoint, line, etc. numbers will be different from those shown above.)

```
/FILNAM,Geom
/title, Stress Concentration Geometry
! Example of creating geometry using keypoints, lines, arcs
/prep7
! Create geometry
k, 1,  0.0,  0.0     ! Keypoint 1 is at 0.0, 0.0
k, 2,  0.1,  0.0
k, 3,  0.5,  0.0
k, 4,  0.5,  0.2
k, 5,  0.0,  0.2
k, 6,  0.0,  0.1

L, 2, 3             ! Line from keypoints 2 to 3
L, 3, 4
L, 4, 5
L, 5, 6

! arc from keypoint 2 to 6, center kp 1, radius 0.1
LARC, 2, 6, 1, 0.1

AL, 1, 2, 3, 4, 5   ! Area defined by lines 1,2,3,4,5
```

Geometry for FEM analysis also can be created with solid modeling **CAD** or other software and imported into ANSYS. The **IGES** (Initial Graphics Exchange Specification) neutral file is a common format used to exchange geometry between computer programs. Tutorial 2B demonstrates this option for ANSYS geometry development.

2-8 TUTORIAL 2B – SEATBELT COMPONENT

Objective: Determine the stresses and deformation of the prototype seatbelt component shown in the figure below if it is subjected to tensile load of **1000 lbf**.

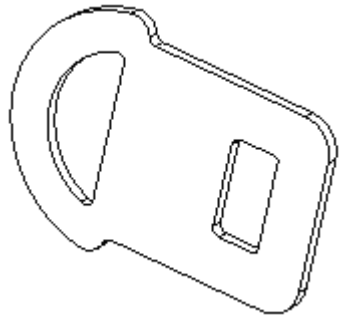


Figure 2-26 Seatbelt component.

The seatbelt component is made of steel, has an overall length of about 2.5 inches and is 3/32 = 0.09375 inches thick. A solid model of the part was developed in a CAD system and exported as an IGES file. The file is imported into ANSYS for analysis. For simplicity we will analyze only the right, or 'tongue' portion of the part in this tutorial.

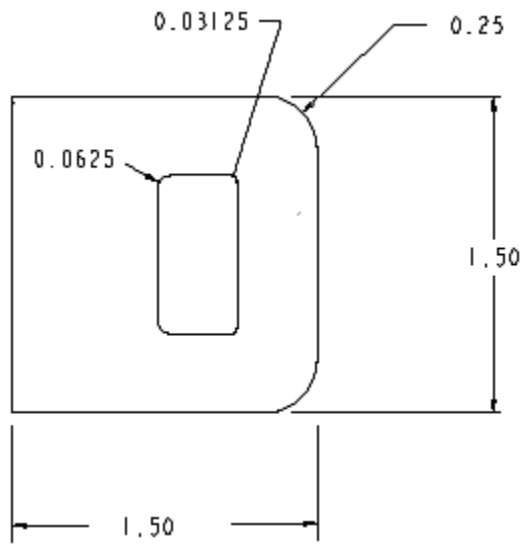


Figure 2-27 Seatbelt 'tongue'.

PREPROCESSING

1. Start ANSYS, Run Interactive, set jobname, and working directory.

Create the top half of the geometry above. The latch retention slot is 0.375 x 0.8125 inches and is located 0.375 inch from the right edge.

If you are not using an IGES file to define the geometry for this exercise, you can create the geometry directly in ANSYS with key points, lines, and arcs by selecting **File > Read Input from** to read in the text file given below and by **skipping the IGES import steps 2, 3, 4,** and **10 below**.

```
/FILNAM,Seatbelt
/title, Seatbelt Geometry
! Example of creating geometry using keypoints, lines, arcs
/prep7
! Create geometry
k, 1,  0.0,     0.0       ! Keypoint 1 is at 0.0, 0.0
k, 2,  0.75,    0.0
k, 3,  1.125,   0.0
k, 4,  1.5,     0.0
k, 5,  1.5,     0.5
k, 6,  1.25,    0.75
k, 7,  0.0,     0.75
k, 8,  1.125,   0.375
k, 9,  1.09375, 0.40625
k, 10, 0.8125,  0.40625
k, 11, 0.75,    0.34375
k, 12, 1.25,    0.5
k, 13, 1.09375, 0.375
k, 14, 0.8125,  0.34375
```

```
L, 1, 2                    ! Line from keypoints 1 to 2
L, 3, 4
L, 4, 5
L, 6, 7
L, 7, 1
L, 3, 8
L, 9, 10
L, 11, 2

! arc from keypoint 5 to 6, center kp 12, radius 0.25, etc.
LARC, 5,6, 12, 0.25
LARC, 8, 9, 13, 0.03125
LARC, 10, 11, 14, 0.0625

AL,all                     ! Use all lines to create the area.
```

2. Alternatively, use a solid modeler to create the top half of the component shown above in the X-Y plane and export an IGES file of the part.

To import the IGES file

3. Utility Menu > File > Import > IGES > OK

Select the IGES file you created earlier. Accept the ANSYS import default settings. Merging is used to remove inconsistencies that may exist in the IGES file, for example keypoints that, because of the modeling or the file translation process, do not quite join to digital precision accuracy.

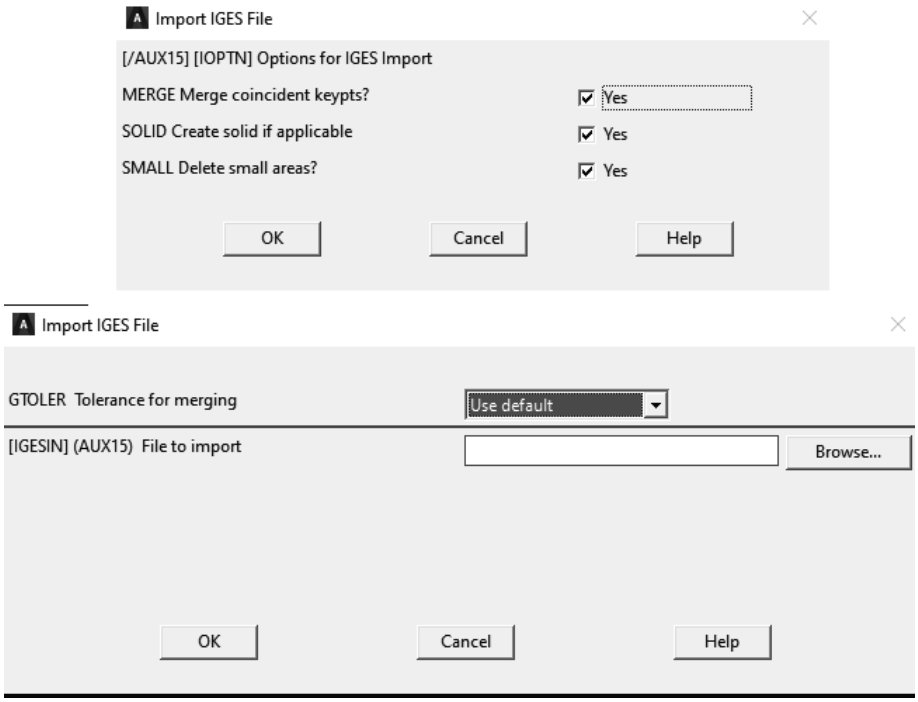


Figure 2-28 IGES import.

Turn the IGES solid model around if necessary so you can easily select the X-Y plane.
4. Utility Menu > PlotCtrls > Pan, Zoom, Rotate > Back, or use the side-bar icon.

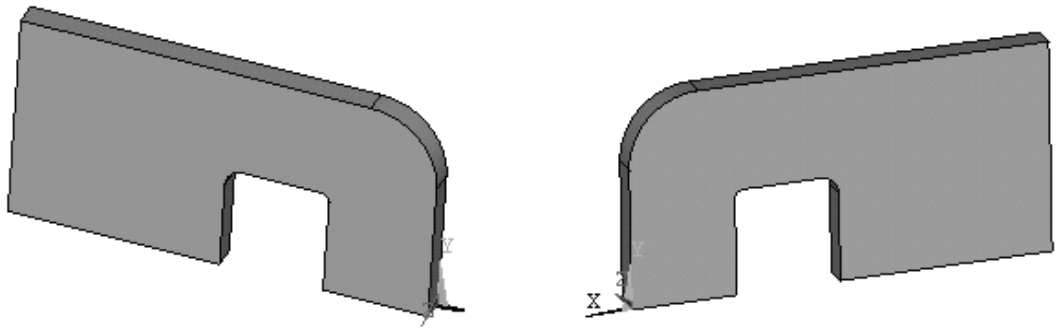


Figure 2-29 Seatbelt solid, front and back.

5. Main Menu > Preprocessor > Element Type > Add/Edit/Delete > Add > Solid > Quad 8node 183 > OK (Use the 8-node quadrilateral element for this problem.)

6. Options > Plane strs w/thk > OK > Close

Enter the thickness

7. Main Menu > Preprocessor > Real Constants > Add/Edit/Delete > Add > (Type 1 Plane 183) **> OK >** Enter **0.09375 > OK > Close**

Enter the material properties

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

Material Model Number 1, click **Structural > Linear > Elastic > Isotropic**

Enter **EX = 3.0E7** and **PRXY = 0.3 > OK** (Close Define Material Model Behavior window.)

Now mesh the X-Y plane area. (Turn on area numbers if it helps.)

9. Main Menu > Preprocessor > Meshing > Mesh > Areas > Free. Pick the X-Y planar area **> OK**

IMPORTANT NOTE: The mesh below was developed from an **IGES** geometry file. Using the text file geometry definition may produce **a much different mesh**. If so, use the **Modify Mesh** refinement tools to obtain a mesh density that produces results with accuracies comparable to those given below. Computed **stress values** can be surprisingly **sensitive** to mesh differences.

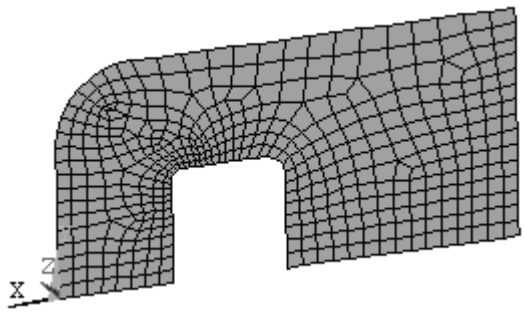


Figure 2-30 Quad 8 mesh.

The IGES **solid model** is no longer needed, and since its lines and areas may interfere with subsequent modeling operations, we can delete it from the session.

10. Main Menu > Preprocessor > Modeling > Delete > Volume and Below (Don't be surprised if everything disappears. Just **Plot > Elements** to see the mesh again.)

11. Utility Menu > PlotCtrls > Pan, Zoom, Rotate > Front (If necessary to see the front side of mesh.)

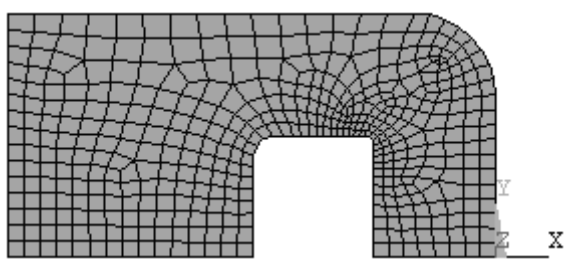


Figure 2-31 Mesh, front view.

Now apply displacement and pressure boundary conditions. Zero displacement UX along left edge and zero UY along bottom edge.

12. Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Lines Pick the left edge **> UX = 0. > OK**

13. Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Lines Pick the lower edge **> UY = 0. > OK**

The 1000 lbf load corresponds to a uniform pressure of about 14,000 psi along the ¾ inch vertical inside edge of the latch retention slot. [1000 lbf/(0.09375 in. x 0.75 in.)].

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

Select the inside line and set **pressure = 14000 > OK**

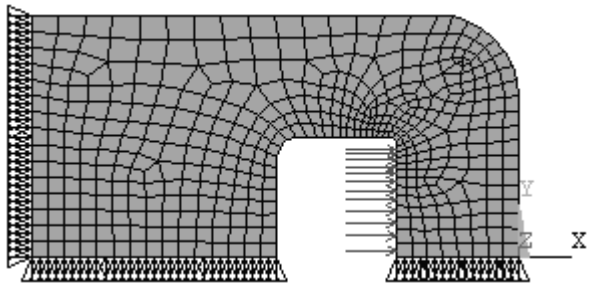


Figure 2-32 Applied displacement and pressure conditions.

Solve the equations.

SOLUTION

15. Main Menu > Solution > Solve > Current LS > OK

POSTPROCESSING

Comparing the **von Mises stress** with the **material yield stress** is an accepted way of evaluating static load yielding for **ductile metals** in a combined stress state, so we enter the postprocessor and plot the **element solution** of von Mises stress, **SEQV**.

16. Main Menu > General Postproc > Plot Results > Contour Plot > Element Solu > Stress > (scroll down) **von Mises > OK**

Zoom in on the small fillet where the maximum stresses occur. The element solution stress contours are reasonably smooth, and the maximum von Mises stress is around **118,000 psi**. Further **mesh refinement** gives a stress value of approximately **140,000 psi**. The small fillet radius of this geometry illustrates the challenges that can arise in creating accurate solutions; however, you can easily come within a few percent of the most likely true result using the methods discussed thus far.

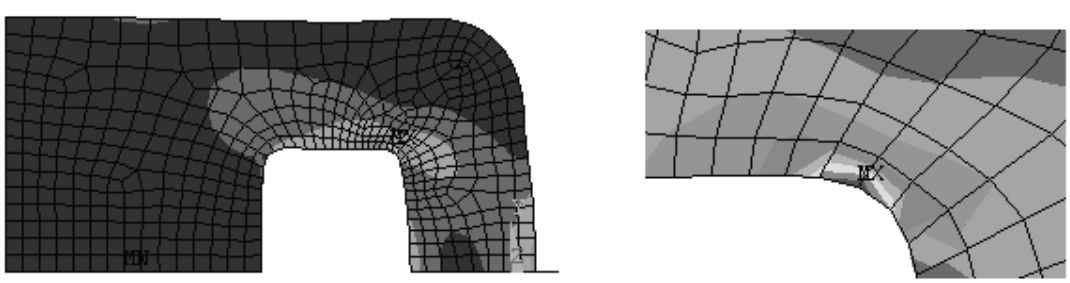


Figure 2-33 Von Mises stresses.

Redesign to reduce the maximum stress requires an increase in the thickness or fillet radius. Look at charts of stress concentration factors, and you notice that the **maximum stress increases** as the **radius of the stress raiser decreases**, approaching **infinite values at zero radii**.

If your model has a **zero radius notch**, your finite-size elements will show a very high stress but not infinite stress. If you refine the mesh, the stress will increase but not reach infinity. The finite element technique necessarily describes finite quantities and cannot directly treat an infinite stress at a **singular point**, so don't 'chase a singularity'. If you do not care what happens at the notch (static load, ductile material, etc.) do not worry about this location but examine the stresses and strains in other regions.

If you really are concerned about the maximum stress in a particular location (fatigue loads or brittle material), then use the actual part notch radius however small (1/32 for this tutorial); **do not use a zero radius**. Also examine the stress gradient in the vicinity of the notch to make sure the mesh is sufficiently refined near the notch. If a crack tip is the object of the analysis, you should look at **fracture mechanics** approaches to the problem. (See ANSYS help topics on fracture mechanics.)

The engineer's responsibility is not only to build useful models, but also to interpret the results of such models in intelligent and meaningful ways. This can often get overlooked in the rush to get answers.

Continue with the evaluation and check the strains and deflections for this model as well.

17. Main Menu > General Postproc > Plot Results > Contour Plot > Element Solu > Strain-total > 1st prin > OK

The maximum principal normal strain value is found to be approximately **0.004 in/in**.

18. Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu > DOF Solution > X-Component of displacement > OK

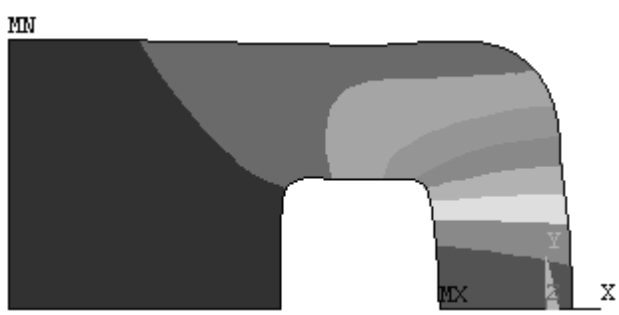


Figure 2-34 UX displacements.

The maximum deflection in the *X* direction is about **0.00145 inches** and occurs as expected at the center of the right-hand edge of the latch retention slot.

2-9 MAPPED MESHING

Quadrilateral meshes can also be created by mapping a square with a regular array of cells onto a general quadrilateral or triangular region. To illustrate this, **delete the last line,** `AL,all`, **from the text file** above so that the **area is not created** (just the lines) and read it into ANSYS. Use **PlotCtrls** to turn **Keypoint Numbering On**. Then use

1. Main Menu > Preprocessor > Modeling > Create > Lines > Lines > Straight Line. Successively pick pairs of keypoints until the four interior lines shown below are created.

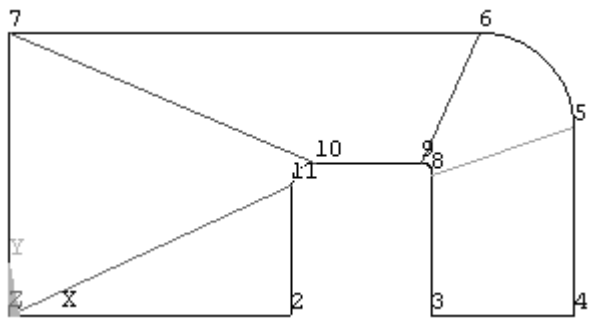


Figure 2-35 Lines added to geometry.

2. Main Menu > Preprocessor > Modeling > Create > Areas > Arbitrary > By Lines Pick the three lines defining the lower left triangular area. **> Apply >** Repeat for the quadrilateral areas. **> Apply > OK**

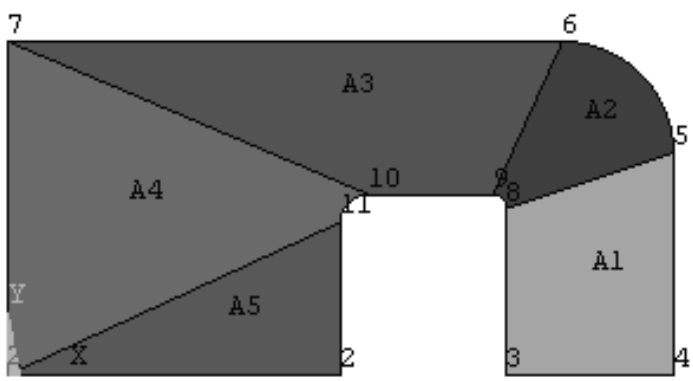


Figure 2-36 Quadrilateral/Triangular regions.

3. Main Menu > Preprocessor > Modeling > Operate > Booleans > Glue > Areas > Pick All

The **glue** operation preserves the boundaries between areas that we will need for mapped meshing.

4. Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > All Lines Enter **4 for NDIV, No. element divisions > OK**

All lines will be divided into four segments for mesh creation.

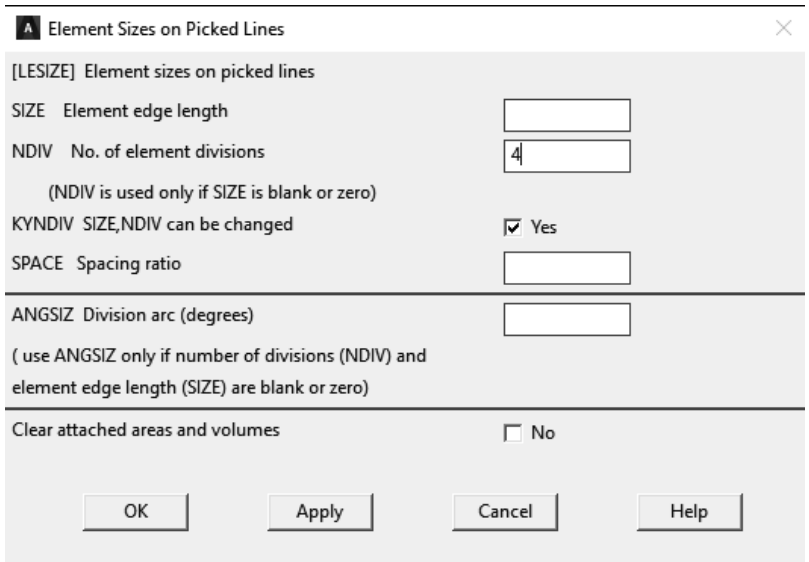


Figure 2-37 Element size on picked lines.

5. Main Menu > Preprocessor > Element Type > Add/Edit/Delete > Add > Solid > Quad 8node 183 > OK (Use the 8-node quadrilateral element for the mesh.)

6. Main Menu > Preprocessor > Meshing > Mesh > Areas > Mapped > 3 or 4 sided > Pick All

The mesh below is created. Applying boundary and load conditions and solving gives the von Mises stress distribution shown. The stress contours are discontinuous because of the poor mesh quality. Notice the long and narrow quads near the point of maximum stress. We need more elements and they need to be better shaped with smaller aspect ratios to obtain satisfactory results.

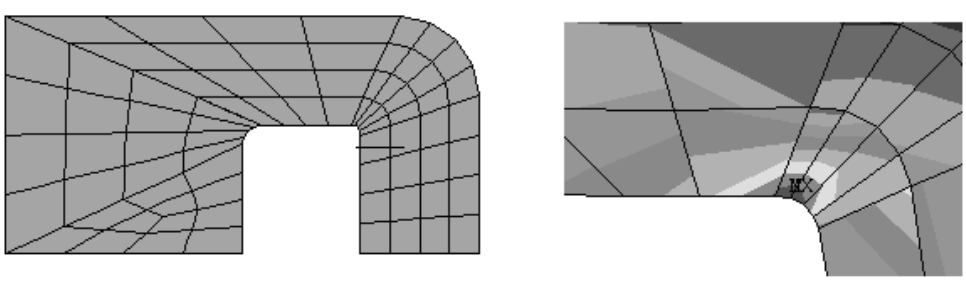


Figure 2-38 Mapped mesh and von Mises results.

One can tailor the mapped mesh by specifying how many elements are to be placed along which lines. This allows much better control over the quality of the mesh, and an example of using this approach is described in Lesson 4.

2-10 CONVERGENCE

The goal of finite element analysis as discussed in this lesson is to arrive at computed estimates of deflection, strain and stress that converge to definite values as the number of elements in the mesh increases, just as a convergent series arrives at a definite value once enough terms are summed.

For elements based on assumed displacement functions that produce continuum models, the computed displacements are smaller in theory than the true displacements because the assumed displacement functions place an artificial constraint on the deformations that can occur. These constraints are relaxed as the element polynomial is increased or as more elements are used. Thus your computed displacements usually converge smoothly from below to fixed values.

Strains are the x and/or y derivatives of the displacements and thus depend on the distribution of the displacements for any given mesh. The strains and stresses may change in an erratic way as the mesh is refined, first smaller than the final computed values, then larger, etc.

Not all elements are developed using the ideas discussed above, and some will give displacements that converge from above. (See Lesson 6.) In any case you should be alert to computed displacement and stress variations as you perform mesh refinement during the solution of a problem.

2-11 TWO-DIMENSIONAL ELEMENT OPTIONS

The analysis options for two-dimensional elements are **Plane Stress**, **Axisymmetric**, **Plane Strain**, **Plane Stress with Thickness** and **Generalized Plane Strain**. The two examples thus far in this lesson were of the third type, namely problems of plane stress in which we provided the thickness of the part.
The first analysis option, **Plane Stress**, is the ANSYS default and provides an analysis for a part with **unit thickness**. If you are working on a design problem in which the thickness is not yet known, you may wish to use this option and then select the thickness based upon the stress, strain, and deflection distributions found for a unit thickness.

The second option, **Axisymmetric** analysis, is covered in detail in Lesson 3.

Plane Strain occurs in a problem such as a cylindrical roller bearing caged against axial motion and uniformly loaded in a direction normal to the cylindrical surface. Because there is no axial motion, there is no axial strain. Each slice through the cylinder behaves like every other and the problem can be conveniently analyzed with a planar model.

Another plane strain example is that of a long retaining wall, restrained at each end and loaded uniformly by soil pressure on one or more faces.

Generalized Plane Strain allows a nonzero strain the Z direction as well as Z strain variations with X and Y.

2-12 SUMMARY

Problems of stress concentration in plates subject to in-plane loadings were used to illustrate ANSYS analysis of plane stress problems. Free **triangular** and **quadrilateral** element meshes were developed and analyzed. **Mapped meshing** with quads was also presented. Similar methods are used for solving problems involving plane strain; one only has to choose the appropriate option during element selection. The approach is also applicable to axisymmetric geometries as discussed in the next lesson.

2-13 PROBLEMS

In the problems below, use triangular and/or quadrilateral elements as desired. Triangles may produce more regular shaped element meshes with free meshing. The six-node triangles and eight-node quads can approximate curved surface geometries and, when stress gradients are present, give much better results than the four-node quad elements.

2-1 Find the maximum stress in the aluminum plate shown below. Convert the 12 kN concentrated force into an equivalent pressure applied to the edge. Use tabulated stress concentration factors to independently calculate the maximum stress. Compare the two results by determining the percent difference in the two answers.

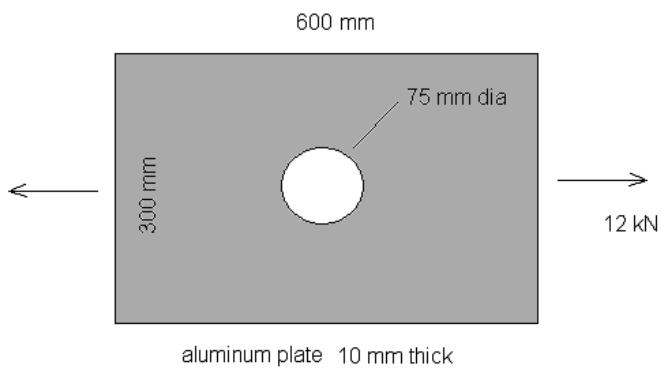


Figure P2-1

2-2 Find the maximum stress for the plate from 2-1 if the hole is located halfway between the centerline and top edge as shown. You will now need to model half of the plate instead of just one quarter and properly restrain vertical rigid body motion. One way to do this is to fix one keypoint along the centerline from UY displacement.

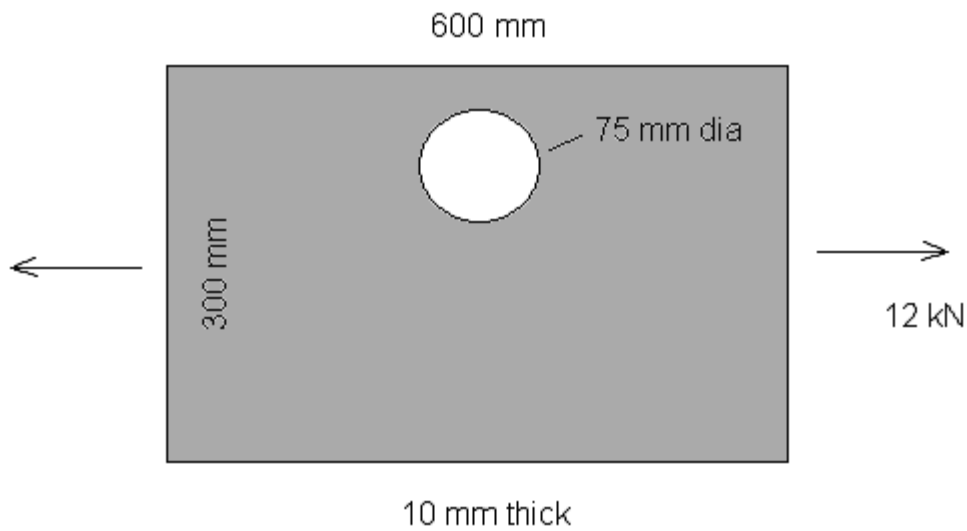


Figure P2-2

2-3 An aluminum square 10 inches on a side has a 5-inch diameter hole at the center. The object is in a state of **plane strain** with an internal pressure of 1500 psi. Determine the magnitude and location of the maximum principal stress, the maximum principal strain, and the maximum von Mises stress. Note that no thickness need be supplied for plane strain analysis.

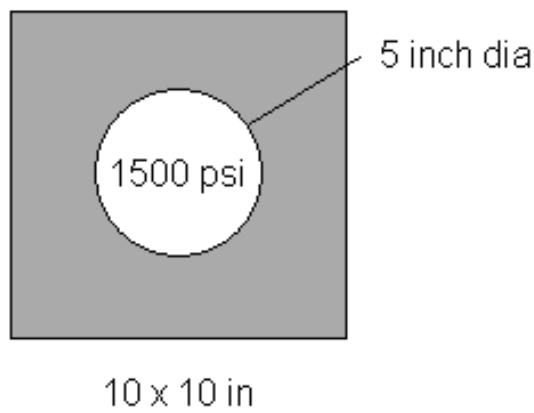


Figure P2-3

2-4 Repeat 2-3 for a steel plate one inch thick in a state of plane stress.

2-5 See if you can reduce the maximum stress for the plate of problem 2-1 by adding holes as shown below. Select a hole size and location that you think will smooth out the 'stress flow' caused by the load transmission through the plate.

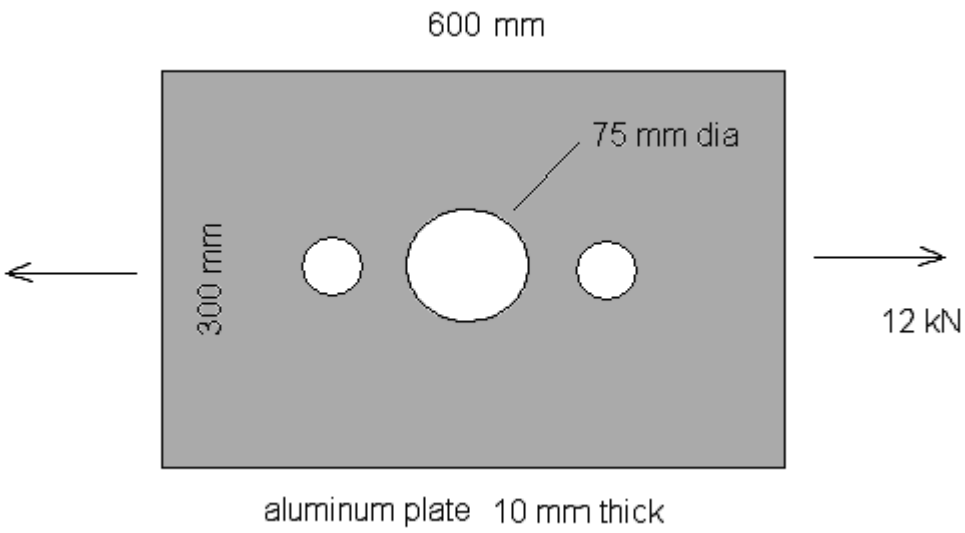


Figure P2-5

2-6 Repeat 2-1 but the object is now a plate with notches or with a step in the geometry. (See the next figure.) Select your own dimensions, materials, and loads. Use published stress concentration factor data to compare to your results. The published results are for plates that are relatively long so that there is a uniform state of axial stress at either end relatively far from notch or hole. Create your geometry accordingly.

Figure P2-6

2-7 Solve the seatbelt component problem of Tutorial 2B again using six node triangular elements instead of the quadrilaterals. Experiment with mesh refinement. Turn on **Smart Sizing** using size controls to examine the effect on the solution. See if you can compute a maximum von Mises stress of around 140 kpsi.

2-8 Determine the stresses and deflections in an object 'at hand' (such as a seatbelt tongue or retaining wall) whose geometry and loading make it suitable for plane stress or plane strain analysis. Do all the necessary modeling of geometry (use a CAD system if you wish), materials and loadings.

2-9 A cantilever beam with a unit width rectangular cross section is loaded with a uniform pressure along its upper surface. Model the beam as a problem in plane stress. Compute the end deflection and the maximum stress at the cantilever support. Compare your results to those you would find using elementary beam theory.

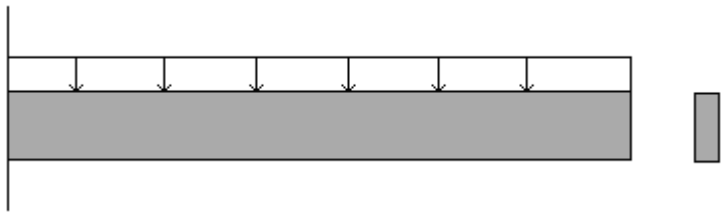


Figure P2-8

Restrain UX along the cantilever support line, but restrain UY at only one keypoint along this line. Otherwise, the strain in the Y direction due to the Poisson effect is prevented here, and the root stresses are different from elementary beam theory because of the singularity created. (Try fixing all node points in UX and UY and see what happens.)

Select your own dimensions, materials, and pressure. Try a beam that's long and slender and one that's short and thick. The effect of shear loading becomes more important in the deflection analysis as the slenderness decreases.

NOTES: