Tutorial #2: Linear-Static Analysis.

In this tutorial you will analyze two simple structures: A beam and a flat plate. In your homework you will be asked to compare your FEA solutions to theoretical results.

BEAMS!

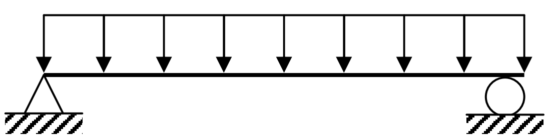
Part 1.Structural Analysis: Simple Geometry

I. Simply supported beam - Surface Load

A. Objective: This problem will familiarize you with the beam finite elements you are now learning in class. By comparing the ANSYS solution with simple beam theory, you will be able to understand the accuracy of your model.

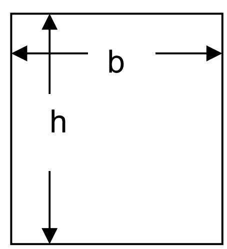
1. A schematic of a statically determinate beam with distributed load is shown below.

q = 10 kN/m



L= 1m

cross section:



b = 2 cm
h = 4 cm

E = 200 Gpa = 2 x 10^8 kPa

2. We will use the GUI to solve this problem. Follow along. The hollow boxes are provided for you to check off when you have completed the steps listed a), b), etc. in case you have to stop in the middle. Bulleted, italicized text contains details about the actions.

3. Preprocessing

- ☐ Choose the preprocessor from the ANSYS main menu
- ☐ Define the element type:

a) element type >> add/edit/delete >> add >>

- *The box on the left lists the general element categories, and the subheadings are lists of the element type. The category gives the element type a unique prefix.*

b) Choose Beam

- *A list of element types for beams should appear in the right box*

c) Choose 2D elastic 3 and click OK.

- *The three is the unique suffix for this element type. You have now specified that you will be using the ANSYS element Beam3, and it will be referred to as element type 1 in your model.*

d) Click Close

- ☐ To find out more about this element, consult the online manual.

a) In the ANSYS Input window type, "Help, Beam3"

- *This will usually be more than enough information. If not, you should consult the Theory Manual.*

- ☐ Define material properties:

a) material properties >> Material Models…

- *A two part box will open. The left part indicates you are defining Material Model 1. In the right part double click:*

b) Structural >> Linear Elastic >> Isotropic."

In the first box labeled EX input 2e11 and in the second box labeled PRXY input 0.35, click OK. Close the window.

- *This creates an isotropic material and assigns it the number 1.*

- *For this simple analysis, the only elastic constants that need to be defined are Young's modulus and Poisson's ratio. You're responsible for keeping track of units since ANSYS is "unitless" Young's modulus is in N/m^2 or Pa.*

- ☐ Create a set of real constants for the cross-section

  a) real constants >> add >>

  - *make sure the correct element type is shown*

  b) OK

  - *type the following in the correct boxes*

  c) area=8e-4, Izz=1.067e-7, height=.04 and click OK.

  d) Click Close

  - *Again, you're responsible for keeping track of units. These have been entered in meters.*

- ☐ Create nodes

  a) Modeling - create >> nodes >> in active cs>>

  - *and enter the following node numbers and coordinates*
  - *selecting apply will execute the command and query the user for more input*

  b) Node 1 is at 0,0

  c) Apply

  d) Node 2 is at .5,0

  e) Apply

  - *selecting OK will exit the node query and apply the current entry*

  f) Node 3 is at 1,0

- ☐ Create elements

  a) Create >> elements >> auto numbered >> thru nodes >>

- *You can either pick on the nodes or enter the node numbers in the ANSYS input box.*

b) Pick node 1 then node 2

c) Apply

d) Pick 2 then 3

e) OK, (you have created a 2-element model of the beam)

- You can plot the elements if they disappear.

  a) Utility Menu: plot >> elements

- Also under plot controls, you can opt to see the boundary conditions and loads that you are creating.

  a) Utility Menu: plot controls >> symbols >> All Applied BCs>>OK

- Now apply a set of boundary conditions

  The Loads menu is in both the Preprocessor menu and the Solutions menu. They are the same. From the Preprocessor menu, first you need to pick the analysis type:

  a) Loads >> new analysis >> analysis type >> static

  b) loads >> define loads >> apply >> structural >> displacement >> on nodes >>

  c) pick node 1

  d) apply

  - *A window will pop up asking you for the displacement*

  e) choose ux and uy, enter the value 0

  f) apply

  g) Pick node 3 this time

  h) apply

  - *If you click on ux, the highlighting should go away leaving the highlighting on uy.*

i) Select only uy and enter the value 0

j) OK

k) Boundary conditions “tie down” the structure at points. It is very important to tie the structure down, otherwise you’ll get zero stress everywhere, large deflections and an invalid solution in general. The only exception is in a modal analysis, where you can run the analysis “free-free”, that is, unconstrained.

- [ ] Apply the surface load

a) loads >> define loads >> apply >> Structural >> pressure >> on beams >> pick all

b) enter the value 10000 in the box for node I (ANSYS will automatically set the load for node J assuming a uniform pressure load)

- *The load is in N/m, thus we are using a fully consistent set of units – N, m, and Pa. ANSYS does not keep track of the units for you so beware!*

- *You should see the pressure loading graphically since you turned on the boundary conditions earlier.*

c) Click OK

The next command is very important to consider each time before you run an analysis. This is because during modeling you may have unselected certain sections of your model and if you don’t reselect them into the “active” set of entities the analysis will consider only part of your model. Do this before the SOLUTION phase each time you run an analysis!

- [ ] Select all of the nodes (go to the Utility Menu)

a) Select >> everything

b) OK

- *You just selected all of your nodes and put them into the “active set”, in other words, the set that all future commands will act on (until you select another set).*

- [ ] SAVE_DB (from the ANSYS Toolbar)

- Finish (ANSYS Main Menu, at the bottom)

  a) This takes you out of the preprocessor and cleans up the windows

4. Solution phase

- Select Solution from the ANSYS main menu.

  a) Solve ->> current LS

  b) OK

- Close the Information Window that says “solution is done”

5. Post-processing

- Select general post-processor from the ANSYS main menu.
- Read in the results

  a) Read results - first set

     - *This tells ANSYS which results set you want to work with. For this analysis you only have one set.*

- Plot the results

  a) Utility menu: Plot>>results>>deformed shape>>OK

     - *this plots the deformed shape.*
     - *Looking at the deformed shape always gives a first check to see if the loads, boundary conditions, etc. were applied correctly. If the shape does not make physical sense, examining this plot first will save time and paper.*
     - *You can obtain a hardcopy of the plot in many ways. The most popular way is to specify that you will be plotting the above results to a postscript file instead of the X11, and then print the file. This file can then be printed as follows:*

  b) Select PlotCtrls>>Hard copy>> To file and giving the *filename* then hit OK

     - *This basically dumps the graphics window image into a postscript file.*

c) You can view your postscript file before you print by using GhostView, type ghostview *filename* at a UNIX prompt or use the pull-down menu with your left mouse button. You may have to play with the color of the background to get good contrast on your printouts. For example try; PlotCtrls >> Style >> Background >> Textured Background… >> Plastic – ivory. Try other backgrounds.

d) Printing is accomplished by

Clicking Print All in ghostview and type in lpr -Pprintername (the printernames are sweet2 and sweet5) but the easiest is to launch mozilla firefox and email the file to yourself. NOTE: eps files can be inserted into a word document using insert>picture.

- ☐ Get a list of nodal displacements (General Postprocessor)

a) List results>>nodal solution>> Displacement vector sum>>OK

b) Print it out

c) Close

- ☐ Then obtain a list of element stresses

a) List results>>element solution>> Line Element Results >>Element Results>> OK

- *You can save these files to another location or print the file directly using the pull-down menus.*
- *What's EPELBYT? SBYB? Go to the online manual and look up BEAM3 again, it's all there.*
- *Close*

- ☐ List the reaction forces

a) List Results>>Reaction Solu…>>All items>>OK

- *Model Check: It is good to verify that your reaction loads at the end constraints equal your applied load.*

- ☐ Go to Part VIII to see what should be turned in with your homework.

II. Simply-supported Beam – Surface Load (Again!)

A. Model the beam with only *one* element and repeat the above analysis.

B. You can solve this problem by editing the model generated previously.

1. To do this, remember to start at the Preprocessor.

- ☐ To remove elements use

a) Modeling >> delete >> elements>> pick all

b) Modeling >> delete >> nodes >> pick all

- *You just removed the nodes, elements, and the surface load. You could have kept nodes 1 and 3.*

c) Now create only nodes 1 and 3 and put one element between them. Your element type and material are still active, however you must re-apply the constraints and load.

2. Review the results: Is this analysis useful? Is it correct? It is just as correct as the first analysis, but you only have nodes at the end of the beam, so you lose the information throughout the beam. You can get ANSYS to output results throughout the beam even if there are no nodes – look at the beam3 description in the help file to do it.

III. Simply-supported Beam – Surface Load (Again!!)

A. Do the same analysis as above, but use 4 nodes and three elements.

- *Create nodes at the following locations: (0,0), (.35,0), (.65,0), (1,0).*

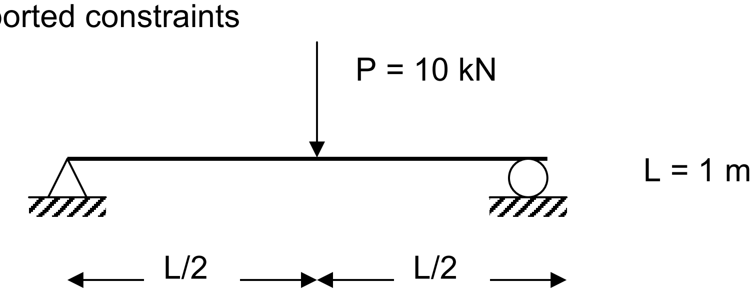
IV. Simply-supported Beam - Point Load

A. Consider the beam shown below with a point load applied at the mid-span:

B. You can solve this problem by editing the model generated previously.

1. To do this, remember to start at the Preprocessor.

- ☐ Re-model the beam with 2 elements and the same simply supported constraints



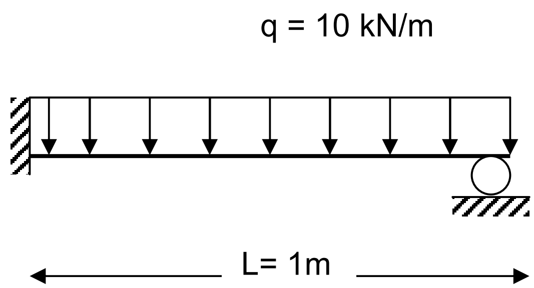
- Then to apply the nodal load use

  a) >> loads>> define loads >> apply >> structural >> force/moment >> on nodes

     - *Pick the center node. (Fy = -10,000) Then do the same analysis steps as before.*

V. Indeterminate (cantilever) Beam - Surface Load

A. Consider the beam shown below:



B. Do the same two element analysis.

C. Notice that you should change the boundary conditions at the left end to account for the fact that this node cannot rotate or translate, i.e. it’s fixed.

VI. Analytical Solutions

A. To find analytical solutions for these models, refer to your undergraduate structural mechanics book or to <u>Mechanics of Materials</u> by Gere & Timoshenko on reserve in the engineering library.

VII. Edit the log files

A. Open the ANSYS .log file and notice your commands. You can edit out errors and save this file for future reference to read parts of it into ANSYS.

Part 2. Structural Analysis: Using 1/4 Symmetry

I. Hole in a square plate

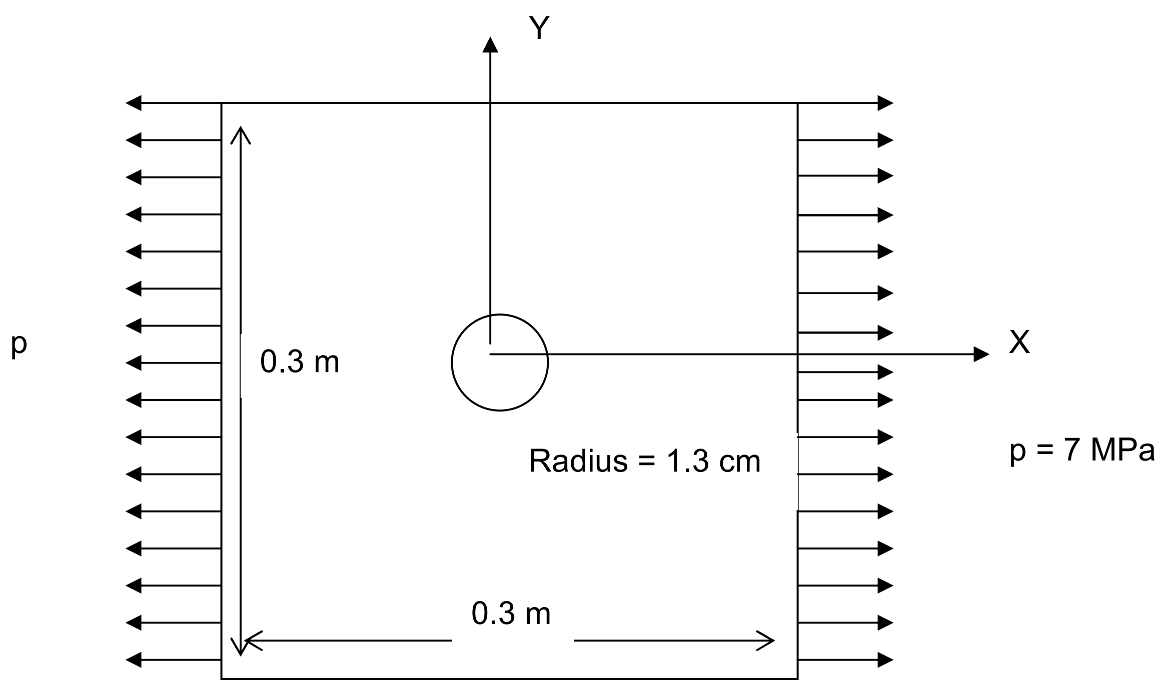
A. Objective: With this problem you will be introduced to the use of symmetry and to the strengths and weaknesses of the automesher, while performing a stress analysis on a common engineering model.

B. The following GUI guided section will develop the simple model by direct generation.

C. Commands that have been explained in Part 1 (i.e. defining element type placing nodes, etc.), are not given the same amount of detail as before. Instead, more emphasis is placed on new commands.

1. Consider a 0.3 m square steel plate of thickness 2.5 cm with a centrally located circular hole of 1.3 cm radius as shown below. It is subjected to a pressure on either side of 7 MPa as shown by p on the figure.

Young's Modulus $E = 2.1 \times 10^{11}$ N/m^2
Poisson's ratio $\nu = 0.27$
Plate thickness $t = 2.5$ cm



- ☐ Change the name of the analysis either by restarting ANSYS or simply go to the utility menu:

  File >> change jobname

- Declare element type 1 to be Solid>>Quad 4 node 42, but before you click OK, click on the Help button in that window. Double click on the "Pictorial Summary" and then find the Plane42 element. Click on it to find out more about the plane42 element. Or type "Help, Plane42" in the ANSYS Input command line.

- Set the isotropic Young's modulus of material type 1 to be 2.1e11 and set the Poisson's ratio (NUXY) of material type 1 to be 0.27.

    - *Watch out for units again. Modulus of elasticity here is considered to be units of N/m^2*

- Specify a state of plane stress

    a) Element type >> Add/edit/delete >> Options >> K3 >> Plane Stress with thickness

    - *This allows you to put a 2-D pressure load rather than a line load on the end of the plate. ANSYS calls these element options "keyopts", which stands for key options.*

- Define the real constant, thickness to be 0.025

    - *Watch out for units. Thickness here is in meters.*

- Place node 1 at .013,0

- Place node 13 at 0, .013

- Activate cylindrical coordinate system to create some nodes on an arc between nodes 1 and 13. Go to the Utility Menu at the top:

    a) WorkPlane>>Change active CS to >> Global cylindrical

- Now fill in nodes between nodes 1 and 13

    a) Modeling >> Create >> Nodes >> Fill between nds +>> pick 1 and 13

    b) OK

    c) Then choose the starting node number to be 2 and the number of nodes to fill to be 11 and the increment to be 1.

    d) OK

    - *This tells ANSYS to fill in the space between points 1 and 13 with 11 nodes starting with node number 2.*

- *Cylindrical coordinates can be very useful in model creation.*

- [ ] Don't forget to re-activate the Cartesian coordinate system

  a) WorkPlane>>Change active CS>> Global Cartesian

- [ ] You might want to occasionally see what coord. sys. is active at any given time – go to the Utility Menu

  a) List >> Status >> Global Status

  - *This shows you all of your current modeling settings. Scroll down until you see "Active coordinate system".*

- [ ] Now place node 157 at .15,0
- [ ] Node 163 at .15,.15
- [ ] And node 169 at 0,.15
- [ ] Create nodes 158-162 between nodes 157 and 163

  a) Modeling >> Create >> Nodes >> Fill between nds >> pick 157 and 163

  b) OK

  c) Starting node number 158, number of nodes to fill 5

  d) OK

- [ ] Fill in nodes 164-168 between 163 and 169 as above
- [ ] List coordinates of nodes 163-169. Go to the Utility Menu:

  a) List >> nodes >> ok

  - *As mentioned earlier, this allows you to check the coordinates of the nodes and to make sure that all nodes have been generated.*

  - *When modeling node for node in this matter it is important to be sure that your nodes are arranged in the manner that you intended, especially since we are going to create many more nodes based on these.*

- [ ] Fill in between nodes 1 and 157, and copy this pattern to create many more nodes (First SAVE_DB, in case it doesn't work)

a) Modeling >> Create >> Nodes>>Fill between nds >>

- *Pick 1 and 157*

b) number of nodes to be filled in 11

c) starting node number 14

d) inc between filled nodes 13

e) spacing ratio 10

f) number of fill operations 13

g) node number increment 1

h) OK

- *If you're wondering what that FILL command is all about, click on Help from the Fill window.*

- [ ] Zoom into the area around the hole. Go to the Utility Menu:

a) PlotCtrls>>Pan/zoom/rotate>>Box zoom

b) drag a box around the area

- [ ] Turn node numbering on.

a) PlotCtrls >> Numbering >> Node (on)

- [ ] Create elements with the given element type, Plane42, and material properties previously defined (SAVE_DB first!).

a) Modeling >> Create >> Elements >> Auto Num Thru Nodes +

- *Pick nodes 1, 14, 15, 2 in that order and click ok.*
- *Notice that the query box allows you the option to unpick any node if you make a mistake. (right click)*
- *You can always check the element list file to see that the elements are connected through the right nodes.*

- [ ] Turn on element numbering

a) PlotCtrls>> Numbering>> Elem/Attribute Numbering >> Element Numbers

- *You may have to plot the elements to see the numbers.*

- [ ] Generate 11 new elements by using the egen command

a) in the ANSYS input window type egen, 12,1,1

- *This tells ANSYS to create 12 more elements using element 1 as the master set (you could have done this with the GUI menu system).*

- *For elements, this is the way to “fill”, i.e. copy a pattern.*

- [ ] Generate 12 more sets by using the egen command again but on the row of elements just created (using GUI clicks).

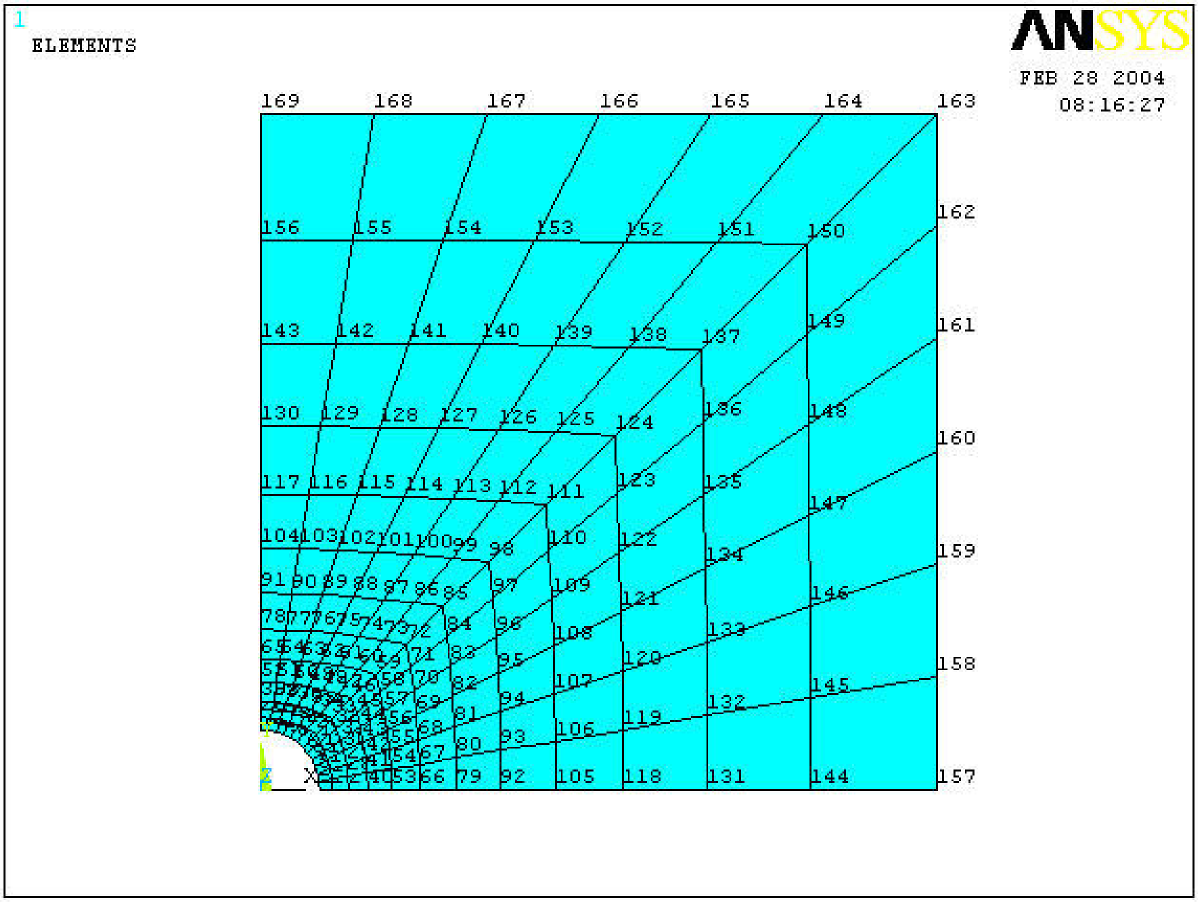
a) Copy >> Elements: auto-numbered

b) Select those 12 elements in order, click ok

c) ITIME = 12, NINC = 13

- *This tells ANSYS to generate 12 more sets of elements, incrementing the number by 13, using elements 1-12 as the master set. Select “Fit” on the Pan-zoom-rotate window to see the whole model.*

d) Your model should look something like this:



- Show all boundary conditions on subsequent plots
- Apply a pressure boundary condition of 7 MPa over a set of nodes

  a) Modeling >> Loads>>Apply>>Pressure>>On nodes>>

  b) Pick "Box", and select the nodes on the right edge of your model by dragging a box around them.

     - *You could also create an "active set" to select from, but it's not worth it for such a small procedure.*

     - *If you haven't noticed already, ANSYS likes to know information **before** the operation. Recall that the "active" set of real constants, "active" material type and the "active" element type, tell ANSYS what properties to apply to the **elements** that you are creating. You can see what is currently active by looking at the very bottom of the ANSYS window.*

- *Conversely, an active **entity** set (i.e., **node**, **element**, etc.) tells ANSYS where to apply the next operations (i.e. forces, pressures, etc.). By box-selecting entities, you are in effect creating a "temporary" active set, which is usually the safest way to do things.*

c) VALUE = 7e6

- *Watch units. Pressure is in Pa (N/m^2)*

- *The pressure will probably be displayed as a line on the edge of the mesh. This is useless. Turn on arrows:*

d) PlotCtrls >> Symbols

Surface Load Symbols: pressure

Show pres and convect as: arrows

- *Now you can see the direction of the pressure, and Oops!, it's in the wrong direction. Delete those pressures and create a new pressure load with a magnitude of –7e6. We want a tensile pressure.*

☐ Create an active set of nodes for the next operation by typing into the ANSYS input window

a) nsel, s, node,,1,157,13

- *This selects the nodes along the horizontal symmetry plane. Again, you could have done this by dragging a box.*

- *Go to the utility menu and plot nodes to see the set that is active.*

☐ Constrain this plane of symmetry in the y-direction

a) Loads>>DefineLoads>>Apply>>Structural>>Displacement>>On Nodes>>pick all >> uy>> enter the value 0

- *Notice that these nodes are considered to be constrained from moving only in the y direction because there is a plane of symmetry there. This is because both the structure AND the loading is symmetric about this line.*

- *Remember that "pick all" refers only to the "active" set of nodes.*

- *Now you must select >> everything from the Utility Menu so that all nodes are active again.*

☐ Using yet another method of selecting, create a new active set consisting of the vertical symmetry line (Go to the Utility Menu).

a) Select >> Entities: Nodes >> By Location >> X coordinates

- *Type 0 for "Min,Max". This picks all the nodes at x = 0. If you want to review the nodes you just picked, go to the Utility Menu and List >> Nodes.*

b) Fix the x-displacements for these nodes, as before.

c) Loads >> Apply >> displacement >> on nodes >> pick all >> ux >> 0

- *Similarly, these nodes are constrained in the x-direction because of the symmetry condition.*

☐ Select all nodes for the solution, another way. Type into the command line:

a) allsel

- *Make sure to do this. Otherwise, ANSYS will only solve for the active set of nodes.*

☐ Save_db

☐ Finish

☐ Solution

a) Solve-Current LS >> OK

☐ Load desired results:

a) General Postproc>> Read Results >> First Set

☐ Plot the deformed shape

a) Plot Results >> Deformed Shape

☐ Animate the displacement results

a) PlotCtrls >> Animate >> Deformed Shape

- *Animation is a great way to gain insight into the response of your model: Note how the symmetry boundary conditions are working properly i.e. Poisson "breathing" has not been suppressed by the vertical B.C. , and the horizontal BC is free to translate in the x-direction.*

- *Keep in mind that this is a static analysis, not dynamic.*

☐ Plot the element stresses

a) Plot Results>> Contour Plot >> Element Solu…>>Stress >>X-Component of stress

- *Record the max stress and location for your homework.*

- *You can also plot the nodal stresses for nicer contours that are averaged at the nodes.*

b) Plot Results>> Contour Plot >> nodal Solu…>>Stress >> X-Component of stress

☐ List the reaction forces (Utility Menu).

a) List Results >> Reaction Sol…>> All items >> OK

- *Model Check: It is good to verify that your reaction loads equal your applied loads.*

> Whew! That was an example of **direct** model generation: You had to control nearly every node, but it created a decent model. While there were some good lessons in there that you will need, THERE MUST BE A BETTER WAY!

II. Create a structured model in ANSYS from scratch, by first creating an ANSYS "CAD" model, then using automatic mesh generation to create an FEA model from the CAD data.

A. Simple CAD modeling in ANSYS.

1. Delete your past model by using "clear and start new", or whatever. Change job names too.

2. Create the underlying CAD geometry for your model.

- ☐ From the Preprocessor, create a rectangle

  a) Modeling >> Create >> Areas >> Rectangle >> By 2 Corners

  b) WP X = 0, WP Y = 0, Width = .15, Height = .15

- ☐ Create a circle

  a) Modeling >> Create >> Areas >> Circle >> Solid Circle

  - *Center at 0,0 and radius of .013.*

- ☐ Using Boolean operations to subtract the circle from the rectangle.

  a) Modeling >> Operate >> Booleans >> Subtract >> Areas

  - *Pick the rectangle and click ok.*
  - *Pick the circle and click ok.*
  - *The order you pick them is critical. Otherwise you'll end up subtracting the square from the circle.*

3. Now you're ready to start meshing the area, but first you need to define the active element type, real constants, and material property, as before. Go do it!

4. Mesh the area using default settings first.

  a) Meshing >> Mesh >> Areas >> Free

  - *Select the area by picking it, click ok.*
  - *The resulting mesh is quite poor. Clear the mesh and do it again while controlling the mesh a little more.*

- *Delete the elements so that you can re-mesh it.*

b) Meshing >> Clear >> Areas (Pick the area)

c) Plot >> Areas (to show the area again)

5. Try the Meshtool. It allows you to mesh and clear rapidly so you can hone in on the best mesh.

a) Meshing >> Meshtool

b) Lines >> Set

- *Pick both lines of symmetry: NDIV = 15, SPACE = 1/10*
- *Pick the circular-arc line, NDIV = 10, and SPACE = 1.*
- *Pick the top line and the right most line, make NDIV = 5*
- *Go to Help to find out what SIZE, NDIV, and SPACE do.*
- *Do a plot >> lines to see how the perimeter of the geometry is divided into segments representing the element sizes along those edges. Notice how you scaled the two line divisions so that the mesh will be finer around the hole.*
- *Now click mesh on the MeshTool, select the area, and click ok.*
- *That produces a decent mesh, but you can still do better, in this case, if you do what ANSYS calls a mapped mesh: Basically the ANSYS pre-processor is real good at mapping a rectangular mesh onto things that have 4 sides. So you need to make your plate into 2 quadrilateral sections. First, clear the old mesh.*

c) Mesh Tool >> Clear >> Pick the area and click ok.

6. Create a mapped mesh.

☐ Divide the areas with a line, so that they each have 4 sides.

a) Modeling >> Create >> Keypoints >> In Active CS

- *Create the Keypoint at 0,0*

b) Modeling >> Create >> Lines >> Lines >> Straight Line

- *Make the line through that keypoint and the upper right vertex of the plate.*

c) Modeling >> Operate >> Booleans >> Divide >> Area by Line

- *Pick the area and click ok, then pick the line and click ok, and you will have two areas.*

☐ Now go back to the mesh tool and set up the lines such that:

- *The two symmetry lines and the diagonal have: NDIV = 13, and SPACE = 1/10.*

- *The rest of the lines have NDIV = 5, SPACE = 1.*

- *Select Mesher: toggle Mapped, and mesh each area.*

- *This should produce a good mesh resembling the one that was generated by hand earlier. Note how interactive and fast it is to use the MeshTool, and how quickly you can generate and iterate to a nice mesh (in this case).*

III. Reflecting on the modeling process:

A. On Symmetry:

1. Symmetry is a very powerful way to reduce your model size. It has some advantages and disadvantages:

a) Advantages

- *Less disk space is required to store your model.*

- *The model requires less memory during run-time, and it runs faster.*

- *It's usually easy to reflect the model to make a full model if required.*

b) Disadvantages

- *The symmetry boundary conditions (loads) can be complicated. i.e. you have to make sure that your load* *condition is symmetric as well as your constraints before you attempt to use symmetry.*

- *The model is not easy to integrate into other models, however you can reflect it easily and create a full model. You can't run your model with a more general asymmetric load at a later time.*

- *The results can be confusing when you present them because people expect to see the full structure.*

2. In general, if you don't have model size or speed limitations it doesn't make sense to use a sub-model.

B. On Meshing

1. From this tutorial you can see that there are many ways to model the same thing in FEA. This is especially true for mesh generation. Creating a good model in a reasonable amount of time will involve taking advantage of the techniques and tricks that ANSYS (or any pre-processor) has to offer. Thus, you need to add to your bag of tricks whenever possible. The best, and perhaps only, way to do this is by playing around with ANSYS – do this throughout the quarter - don't do the tutorials word for word – try other things – surf the help menus a bit, etc. It's much less fun to "play" with ANSYS at 3:00 AM the night before your project is due!