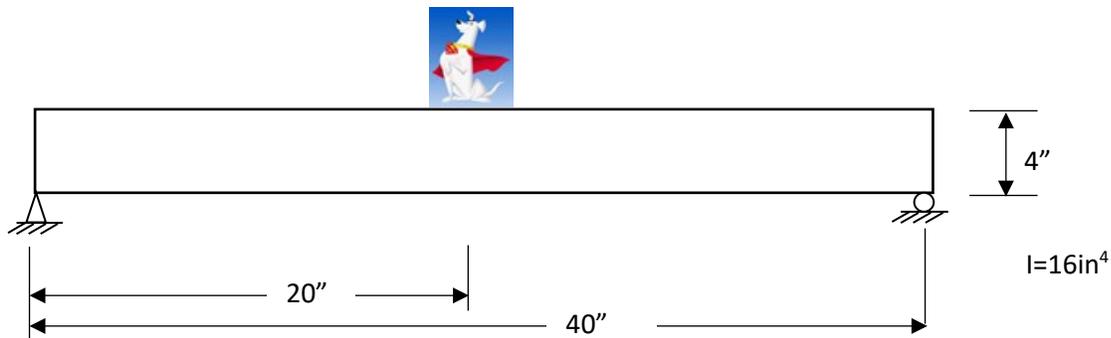


ME304 Finite Element Analysis, Fall 2019

HW 7: Line body, 2D and 3D Elements

Purpose: use ANSYS scripts provided to study the performance of line body elements (BEAM 188 which is a linear line body element and BEAM189 which is a quadratic line body element), a 2D element (PLANE182) and a 3D element (SOLID185). By filling in the tables below, you will perform a small convergence study (show how the results vary based on the number of elements) for each of the different elements.

Model the following beam. Note that this is one-fourth the length of the beam in the workbook, but Super-FEA-Dog weighs 4000 pounds compared to the 1000 pound aardvark in the workbook. The beam's width (z-axis direction) is 3 inches (therefore, $I = bh^3/12 = (3\text{in})(4\text{in})^3/12 = 16\text{in}^4$ – just like the workbook).



Assignment summary: this assignment document is several pages long. I strongly recommend you read through the entire assignment before beginning. In summary, here is what is expected:

- Calculate stresses and displacements by hand (problem 1 and 9).
- Calculate the numbers of nodes, elements, and stiffness matrix sizes for each of the 18 models
- Calculate stresses and displacements using ANSYS (4 different elements, 3 different sized elements for each – 12 total ANSYS models). Calculate for each of the 12 models:
 - maximum σ_x
 - maximum compressive σ_x
 - maximum compressive σ_y
 - maximum magnitude displacement

Problem 1: before beginning any FEA work, you should develop a sense for reasonable answers. So, do that: Using equation from beam displacement charts, determine the maximum displacement for this beam. Using bending stress equation (My/I), determine the bending stress in this beam (the bending stress caused by the bending moment, M . It is the normal stress in the axial direction; same direction as the neutral axis. Be sure to follow the standard problem format – given, find, assumptions, FREE BODY DIAGRAMS, etc...(do not forget to include a clear and complete FBD!). Put your answers here and attach your separate work.

$$\delta_{\max} = \underline{\hspace{2cm}} \quad \sigma_{\max \text{ bending}} = \underline{\hspace{2cm}} \quad \sigma_{\min \text{ bending}} = \underline{\hspace{2cm}} \quad \sigma_{y-\min} = \sigma_{y-\max} = \underline{\text{zero}}$$

FEA Problems:

Just like with the turbine project, we want to make sure element size does not affect our answer, so we need to do a convergence study. For each element, you will run 3 different models each with different sized elements as defined in the table below.

To change the size of the elements, look for the ESIZE command in the script. Change the value of ESIZE in the script to change the element size.

	Size of “big” elements	Size of “medium” elements	Size of “small” elements
Line body (BEAM188)	20 inch	5 inch	0.25 inch
Line body (BEAM189)	20 inch	5 inch	0.25 inch
2D Plate (PLANE182)	20 inch*	5 inch*	0.25 inch
3D Brick (SOLID185)	20 inch*	5 inch*	0.25 inch

(*October 31 note: if you have already completed the 2D and 3D work with 2” and 0.5” that is fine – use those results).

Problem 2: determine the **number of elements** for each element type and element “size”. For 2D and 3D elements, first calculate how many elements would lie across each edge, then multiply to get the number of elements on a surface, and for 3D multiply to get the number in the volume.

number of elements	“big” elements	“medium” elements	“small” elements
Line body (BEAM188)			
Line body (BEAM189)			
2D Plate (PLANE182)			
3D Brick (SOLID185)			

Problem 3: determine the **number of nodes** for each element type and element “size”. For BEAM188, there are 2 nodes per element and for BEAM189, there are 3 nodes per element – but remember that elements share nodes. For PLANE182, there are 4 nodes per elements... but elements share nodes. For SOLID185, there are 8 nodes per element... but elements share nodes. For 2D and 3D determine the number of nodes along each edge and multiply (similar to problem 2).

number of nodes	“big” elements	“medium” elements	“small” elements
Line body (BEAM188)			
Line body (BEAM189)			
2D Plate (PLANE182)			
3D Brick (SOLID185)			

Problem 4: estimate the **size of the stiffness matrix** for each model. Remember, the stiffness matrix is a square matrix with size {number of nodes * DOF per node}. Also remember how much more difficult it is to invert a 3X3 than a 2X2, and how fun a 4X4 must be...even computers have difficulty with large matrices. From ANSYS help, BEAM188 elements have 2 nodes and 6 DOF per node. BEAM189 elements

have 3 nodes and 6 DOF per node. PLANE182 elements have 4 nodes with 2 DOF per node (UX, UY). SOLID185 have 6 nodes per element and 3 DOF per node (UX, UY, UZ).

Matrix size	DOF/node	“big” elements	“medium” elements	“small” elements
Line body (BEAM188)	6			
Line body (BEAM189)	6			
2D Plate (PLANE182)	2			
3D Brick (SOLID185)	3			

For the following problems, run the provided script models (you will need to change ESIZE in each of the 4 provided scripts). Then plot requested information in the tables below.

Problem 5: enter the **maximum magnitude of displacement** in the table below. The “analytic answer” column is from the answer in Problem 1. Does it take a noticeable amount of time to run the “small element” 3D model?

max displacement	“big” elements	“medium” elements	“small” elements	Analytic answer
Line body (BEAM188)				
Line body (BEAM189)				
2D Plate (PLANE182)				
3D Brick (SOLID185)				

Problem 6: enter the **maximum stress in the x-direction** in the table below. The “analytic answer” is from the answer in Problem 1.

x-direction, max tension stress	“big” elements	“medium” elements	“small” elements	Analytic answer
Line body (BEAM188)				
Line body (BEAM189)				
2D Plate (PLANE182)				
3D Brick (SOLID185)				

Problem 7: enter the minimum stress (**maximum compressive stress**) in the **x-direction** in the table below. The “analytic answer” is from the answer in Problem 1.

x-direction, max compressive stress	“big” elements	“medium” elements	“small” elements	Analytic answer
Line body (BEAM188)				
Line body (BEAM189)				
2D Plate (PLANE182)				
3D Brick (SOLID185)				

Problem 8: enter the minimum stress (the **maximum compressive stress**) in the **y-direction** in the table below. What is the stress in the y-direction would you expect (enter that in the last column). Do you notice anything “unusual” or “unexpected” in the 2D or 3D models? Briefly discuss.

y-direction, max compressive stress	“big” elements	“medium” elements	“small” elements	“expected” answer
Line body (BEAM188)				
Line body (BEAM189)				
2D Plate (PLANE182)				
3D Brick (SOLID185)				

Problem 9: one of the primary objectives of this course is for you to develop the ability to have confidence in your engineering work. If you approach a problem using two different approaches, and you get the same results, that can give you a certain degree of confidence. Since the only difference between this beam and the aardvark beam in the workbook is this beam is $\frac{1}{4}$ the length of the workbook’s beam and the load is 4 times greater. Therefore, the bending stress in this beam should be (circle the best answer):

- $\frac{1}{4}$ the stresses in the workbook aardvark beam
- $\frac{1}{2}$ the stress in the workbook aardvark beam
- Equal to the stress in the workbook aardvark beam
- 4-times greater than the stress in the workbook aardvark beam
- None of the above, the relationships should be: (fill in the blank): _____

The maximum magnitude deflection in this beam should be (circle the best answer):

- $\frac{1}{4}$ the deflection in the workbook aardvark beam
- $\frac{1}{2}$ the deflection in the workbook aardvark beam
- Equal to the stress in the workbook aardvark beam
- 4-times greater than the deflection in the workbook aardvark beam
- None of the above, the relationships should be: (fill in the blank): _____

Compare the pretty pictures created in FEA with your pretty colored workbook results. Select the FEA model that most closely matches the analytic results. First, change the color bar in the FEA output to have the same colors as in the workbook:

- When the model is complete, from the top menu bar in ANSYS APDL, select PlotCtrls>Style>contours>uniform contours and then in the box NCONT enter 5 (which is the desired number of colors).
- Change from isometric view to front view (GUI on right side of screen)

Do the FEA and workbook results confirm each other?

Problem 10: why were the minimum stress in the x-direction (maximum compressive stress) for the 2D and 3D elements so different than the bending stress calculated by hand – and the difference got worse with smaller elements?

Problem 11: comment on which element type and element size worked best for determining bending stresses? Was the small 3D element model better in any way?