CHALLENGE PROBLEM 2

Bar Under Gravity Force in Ansys

MAE 3150   Spring 2026

Due: Friday, 2/20/26 at 9 pm via Gradescope

Learning Goals

After completing this challenge problem, you should be able to:

- Solve simple boundary value problems (BVP) for static structural analysis using Ansys.
- Identify whether any option you pick in Ansys affects the mathematical model, numerical solution or post-processing.
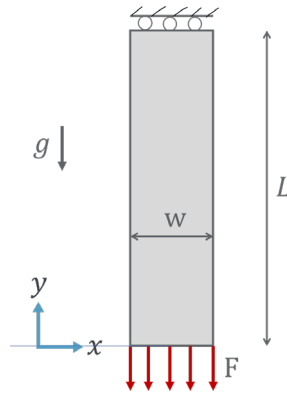- Explain at a conceptual level what's happening inside the blackbox as you set up and solve the problem in Ansys.

Instructions

Create a separate document with your responses to the following items for the "Bar under gravity force" challenge problem. Make sure to number each response to correspond to the question you are answering. Snapshots requested below can be captured using the Snipping tool in Windows or another app or through image export options within Ansys. After you have completed your responses to all the items, you can submit your document via Gradescope.

Problem Specification

Consider the bar shown in the figure below that is subject to gravity as well as the load F at the lower end. You previously derived the BVP for this problem and then carried out the finite-element (FE) solution for the BVP by hand. Here you'll carry out the FE solution for this BVP using Ansys and compare your Ansys FE results with the hand FE results.

The dimensions of the bar are:

- Length, $L = 2\,m$
- Width, $w = 0.5\,m$
- Thickness in the direction perpendicular to the screen, $t = 0.1\,m$

Note the co-ordinate system with $y = 0$ and $y = L$ corresponding to the lower and upper ends, respectively.

The material properties are:

- Young's modulus, $E = 1 \times 10^8\,Pa$
- Poisson's ratio, $v = 0.3$
- Density, $\rho = 10,000\,kg/m^3$

The force acting on the lower end is given as $F = 400\,N$.

Gravity is acting in the negative y-direction with acceleration due to gravity, $g = 9.8066\,\frac{m}{s^2}$.

In Ansys, you will need to solve this as a 2D plane stress problem. The primary unknowns in the 2D case are the displacement components u_x and u_y. Since we expect that u_x will be much smaller than u_y, we'll focus on the latter while evaluating our Ansys results. We'll guide you through the Ansys solution procedure which is similar to the preceding bar in extension example. Please read the following carefully and respond to the prompts.

Part One

Pre-analysis

Since we expect that $u_x \ll u_y$, the 2D plane stress BVP that Ansys solves reduces to the 1D BVP we saw in the earlier challenge problem. Recall that when using the 1D assumption for this problem, the only non-zero displacement and stress components are $u_y(y)$ and $\sigma_{yy}(y)$, respectively. The 1D governing equation in terms of the stress component is given by:

$$\frac{d\sigma_{yy}}{dx} + f_y = 0$$

Using the 1D constitutive model and strain-displacement relation, the above 1D governing equation can be written in terms of the displacement component u_y:

$$E\frac{d^2 u_y}{dy^2} + f_y = 0$$

The boundary conditions are:

$\frac{du_y}{dy} = \frac{F}{AE}$ at $y = 0$

$u_y = 0$ at $y = L$

1. List three assumptions used in deriving the above BVP for uy, apart from the 1D assumption.
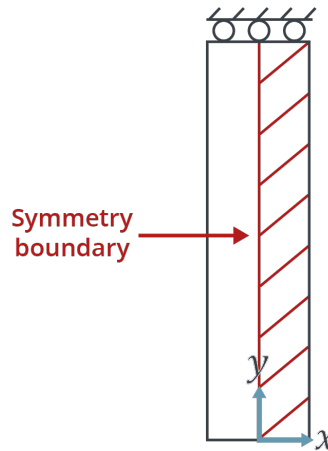
Part Two

Geometry

In the Geometry step, we specify the domain over which we need to solve the governing equations. The 1D domain is a vertical line from $y = 0$ to $y = L$. Since we are solving this as a 2D plane stress problem in Ansys, we'll have to use a 2D domain. By taking advantage of symmetry, we can use the **right half of the bar** for the domain as shown in the accompanying figure. Note the coordinate system.

Create the 2D domain in SpaceClaim using a procedure similar to the bar in extension example. As you work in Ansys, remember to save your project periodically from the Workbench window. If you are using Ansys Online or Apps on Demand, don't forget to copy the wbpz to a permanent location.

2. Turn in a snapshot of your rectangular domain in SpaceClaim. You can take a snapshot using the Snipping tool in Windows.

Part Three

Mesh

In the Mesh step, we divide the domain into elements. Create a mesh in Ansys using the following tips:

- Use 3 equal divisions in the vertical direction (i.e. the y-direction) and 1 division in the horizontal direction (i.e. the x-direction). Set Behavior to Hard in Sizing to enforce these.
- To ensure that you get a regular mesh, apply Face Meshing to the entire geometry.
- Use linear element type so that you don't have mid-side nodes.

After generating the mesh, check that you have 3 elements and 8 nodes.

3. Submit a snapshot of your mesh. Highlight the locations at which Ansys will later calculate the primary unknowns u_x and u_y by solving a set of algebraic equations. You can highlight the locations using the node selection option.

Part Four

Mathematical Model Setup

In the Mathematical Model Setup step, we specify the governing equations and boundary conditions. Complete the mathematical model setup in Ansys Mechanical by following a similar procedure to the bar in extension example. Here are some tips:

- Apply half of the load F at the bottom boundary since we are using symmetry.
- To specify gravity as the body force, select the option Inertial > Standard Earth Gravity. Specify gravity as acting in the negative y-direction. This will set $f_y = -\rho g$.
- At the symmetry boundary (i.e. the left edge), set $u_x = 0$ and $\tau_{xy} = 0$. Impose this using the Displacement rather than Symmetry boundary condition in Ansys Mechanical.

4. Explain briefly how you specify the value of the constant E, the Young's modulus, in the governing equation in Ansys. Hint: This involves two separate steps, one in Engineering Data and the other in the main Mechanical interface.

5. Explain how you imposed the above symmetry boundary conditions at the left edge in Ansys using the Displacement boundary condition. Did you have to do anything to set $\tau_{xy} = 0$? Explain why or why not?

Part Five

Numerical Solution

In the Numerical Solution step, we get the Ansys FE solver to generate algebraic equations corresponding to our mathematical model and solve the algebraic equations to determine the degrees of freedom. Obtain a numerical solution of the mathematical model in Ansys Mechanical by hitting the Solve button.

6. What is the total number of degrees of freedom in your Ansys model? Of these, how many degrees of freedom can be determined from the essential boundary conditions? List these degrees of freedom and their corresponding values. Number the nodes on the left edge starting from the bottom. Then continue the node numbering on the right edge, again starting from the bottom.

7. How many algebraic equations will Ansys have to solve simultaneously to determine the remaining degrees of freedom?

Part Six

Post-Processing

At this point, Ansys should have determined all the degrees of freedom. You can move on to the Post-Processing step. Obtain the results outlined below and answer the related questions. If a question asks you to plot a result, make sure to include the plot as a part of your answer.

8. Plot contours of the displacement magnitude (i.e. "Total Deformation"). Use Auto Scale and turn on the undeformed wireframe. Does your solution satisfy the essential boundary conditions? Explain why or why not.

9. Plot the variation of u_y vs. y along the symmetry boundary. For this, you will need to create a "path" along that boundary. Set the number of sampling points to 2 to get a total of 4 points along the path. This should give you the values of u_y at the 4 nodes on this boundary. Use mm as the unit.

Take a snapshot of the table of u_y values along the path that Ansys displays and include that as part of your answer to this question. Comment on how well the 4 u_y values from Ansys compare with the corresponding values from the hand FE solution given in the following table. The hand FE solution is from the previous challenge.

y (m)	u_y (mm) Hand FE solution
0	-2.12
0.67	-1.85
1.33	-1.14
2	0

10. Calculate the reaction in the y direction at the top boundary in Ansys. Include a snapshot that shows this reaction value in the Ansys interface. Double this value to determine the reaction for the full bar. Compare this value to the reaction F_R at $y = L$ you calculated in Challenge 1 using the algebraic equation at node 4. Report reaction results to one decimal place. Do you get good agreement for the reaction at $y = L$ between the Ansys FE and your hand FE solution?

Explain briefly how Ansys calculates this reaction force. Use an explanation similar to the one in the Reaction Calculation video in the bar in extension example.

11. Plot σ_{yy} contours over the undeformed bar. Use the Unaveraged display option. Turn on the elements in the view.

Part (a): Is the stress discontinuous across element interfaces? Briefly explain why based on the interpolation for u_y.

Part (b): To obtain σ_{yy} at $y = 0$ in the Ansys solution, probe the unaveraged σ_{yy} value at the bottom left corner where the symmetry and bottom boundaries intersect. What is the value you get? Report your result in Pa to one decimal place. Note that you will have to change units to meters to get stress results in Pa.

Compare this value to the stress component σ_{yy} at $y = 0$ you got in Challenge 1, #13. Do you get reasonably good agreement for σ_{yy} at $y = 0$ between the Ansys FE and your hand FE solution?

Part (c): Comment on how well the Ansys value for σ_{yy} at $y = 0$ matches the applied traction in the y direction which is a boundary condition. What is the percent error in the Ansys value? Do you expect the Ansys value for σ_{yy} at $y = 0$ to match the boundary condition exactly? Why or why not? (Think about whether this is an essential or natural boundary condition.)

Part Seven

Verification and Validation

12. Refine the mesh by increasing the number of elements to 6 while keeping the linear interpolation. You just need to change the sizing for the vertical edges. Obtain the numerical solution on the refined mesh. Submit a snapshot of your refined mesh.

    What is the value of u_y at the bottom left corner when using the refined mesh? You can get this value from the table under the path plot for u_y that you created previously. (Report displacement results here and below in mm to an accuracy of two decimal places.) This value gives the overall extension of the bar in the y-direction. Compare this to the corresponding value obtained on the 3-element mesh. Also, compare the unaveraged σ_{yy} value at the lower left corner with the applied traction in the y direction. Is the agreement with the applied traction better on refining the mesh?

13. Revert to the original 3-element mesh but now use quadratic interpolation. What is the value of u_y at the lower left corner for this case? Compare this to the corresponding value obtained above on the 6-element mesh. Also, compare the unaveraged σ_{yy} value at the lower left corner with the applied traction. Is the agreement with the applied traction better than the 6-element mesh with linear interpolation?

To submit this assignment, please go to Gradescope.