Exercise 4-1

Modeling a Simple 1D Cantilever Beam Using Beam Elements

Overview

In exercises 4-1, 4-2, and 4-3, you will perform a steady-state structural analysis of a cantilever beam using 1D, 2D, and 3D models meshed with beam, plane, and solid elements. The beam is made of aluminum with a Young's modulus of 73.1 GPa and a Poisson's ratio of 0.33. It is 1 m in length with a 10x10 cm cross section. It has a load of 5000 N applied to the unsupported end (Figure 4-1-1). The 1D model will be meshed with BEAM189 elements. The 2D model will be meshed with PLANE182 elements. And, the 3D model will be meshed with SOLID186 elements. The goal of each analysis is to determine the deflection at the end of the beam and the stresses throughout the beam. Once all three models have been solved, the differences and similarities among the results of the three models will be examined.

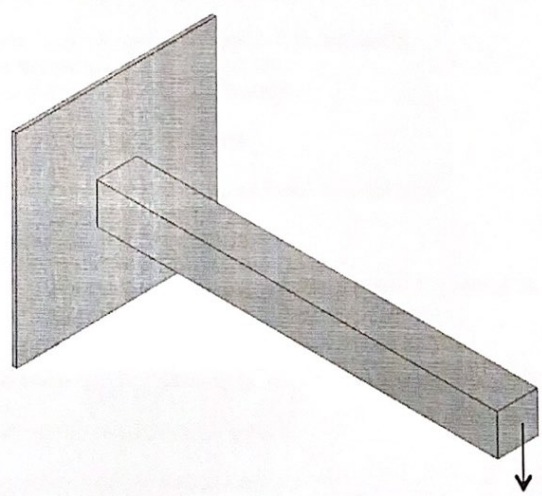

Figure 4-1-1 Schematic of Cantilever Beam with End Load.

ANSYS Mechanical APDL for Finite Element Analysis.
DOI: http://dx.doi.org/10.1016/B978-0-12-812981-4.00018-6

Model Attributes

Material Properties for 6061-T6 Aluminum

- Young's modulus—7.310e10 Pa
- Poisson's ratio—0.33

Loads

- 5000 N downward load applied to the center of the free end of the beam

Constraints

- The fixed end of the beam is fully constrained in *x*, *y*, and *z*

File Management

Create a new folder in your "Intro-to-ANSYS" folder named "Exercise4-1"

Open a new session of ANSYS using the Mechanical APDL Product Launcher

Change the Working Directory to the new "Exercise4-1" folder

Change the Jobname to "Exercise4-1"

Click Run to start ANSYS

Step 1: Define Geometry

1-1. Create keypoints to define the ends of the beam

- Preprocessor > Modeling > Create > Keypoints > In Active CS
- Supply (0,0,0) as the coordinates for the 1st KP
- Supply (1,0,0) as the coordinates for the 2nd KP

1-2. Create a line to connect the two keypoints

- Preprocessor > Modeling > Create > Lines > Lines > Straight Line

1-3. Save the model geometry

The solid model geometry specifies the location and length of the beam. The rest of the geometry is included via the beam's section properties including its moment of inertia.

Step 2: Define Element Types

2-1. Define the element type to use for this model

- Preprocessor > Element Type > Add/Edit/Delete
- Choose BEAM189 as the element type for this analysis

2-2. Define the section properties for your model

- Preprocessor > Sections > Beam > Common Sections
- Ensure that a rectangle is shown as the Sub-Type (Figure 4-1-2)
- Ensure that "Offset To" is set to "Centroid"
- Enter 0.1 for B (width)
- Enter 0.1 for H (height)
- Click "Preview" to view the defined cross section
- Click OK

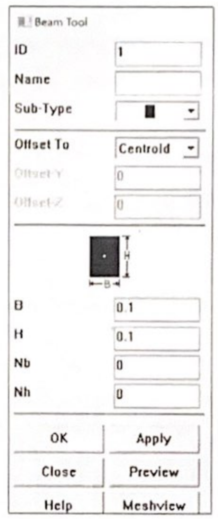

Figure 4-1-2 Defining Section Properties using the Beam Tool.

Step 3: Define Material Properties

3-1. Create a linear elastic material model for 6061-T6 aluminum

- Preprocessor > Material Props > Material Models
- Choose a structural, linear, elastic, isotropic material model
- Supply 7.310e10 as the value for Young's modulus (EX)
- Supply 0.33 as the value for Poisson's ratio (PRXY)

3-2. Save your progress

Step 4: Mesh

4-1. Create the mesh for the finite element model

- Preprocessor > Meshing > MeshTool
- Click the "Mesh" button
- Click the "Pick All" button in the Mesh Lines dialog box

4-2. Turn element numbering on

- Utility Menu > PlotCtrls > Numbering...
- Change "Elem/Attrib numbering" to "Element numbers"
- Click OK

4-3. Plot the finite element mesh

- Utility Menu > Plot > Elements

Note that this model only has two elements (element #1 and element #2).

4-4. Change to the isometric view

- Click the "Isometric View" button in the Pan Zoom Rotate menu

4-5. Turn element shape display on

- Utility Menu > PlotCtrls > Style > Size and Shape...
- Turn "[/ESHAPE] Display of element shapes based on real constant descriptions" on
- Click OK

*The **/ESHAPE** command displays elements and element results using shapes determined by the real constants or section properties that have been defined. This allows you to view the 3D representation of lower order models.*

4-6. Return to the front view

- Click the "Front View" button in the Pan Zoom Rotate menu

4-7. Turn element shape display off

- Utility Menu > PlotCtrls > Style > Size and Shape...
- Turn "[/ESHAPE] Display of element shapes based on real constant descriptions" off
- Click OK

4-8. Turn element numbering off

- Utility Menu > PlotCtrls > Numbering...
- Change "Elem/Attrib numbering" to "No numbering"
- Click OK

Step 5: Apply Constraint Boundary Conditions

5-1. Constrain the fixed end of the beam

- Solution > Define Loads > Apply > Structural > Displacement > On Keypoints
- Click on the keypoint at the origin or specify Keypoint 1 in the text box
- Click OK
- For "Lab2 DOFs to be constrained" choose "All DOF"
- For "VALUE Displacement value" enter 0
- Click OK

Two perpendicular little blue arrows should appear at the left end of the beam to indicate that the constraints have been successfully applied. In addition, two perpendicular orange double arrows should appear to indicate that rotations have also been constrained.

5-2. Save your constraints

Step 6: Apply Load Boundary Conditions

6-1. Apply a downward load to the free end of the beam

- Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints
- Click on the keypoint at the free (right) end of the beam or specify Keypoint 2 in the text box
- Click OK
- For "Lab Direction of force/mom" choose "FY"
- For "VALUE Force/moment value" enter −5000
- Click OK

A red arrow pointing downward should appear at the right end of the beam to indicate that the load has been applied successfully.

6-2. Save your loads

Step 7: Set the Solution Options

The default solution options can be used for this analysis.

Step 8: Solve

8-1. Select everything in your model

- Utility Menu > Select > Everything

8-2. Solve

- Solution > Solve > Current LS

8-3. Save your results

Step 9: Postprocess the Results

9-1. Change to the isometric view

- Click the "Isometric View" button in the Pan Zoom Rotate menu

9-2. Turn element shape display on

- Utility Menu > PlotCtrls > Style > Size and Shape. . .
- Turn "[/ESHAPE] Display of element shapes based on real constant descriptions" on
- Click OK

9-3. Plot the vertical deformation of the beam

- General Postproc > Plot Results > Contour Plot > Nodal Solu
- Choose "DOF Solution"
- Choose "Y-Component of displacement"
- Change "Scale Factor" to "True Scale"
- Click OK

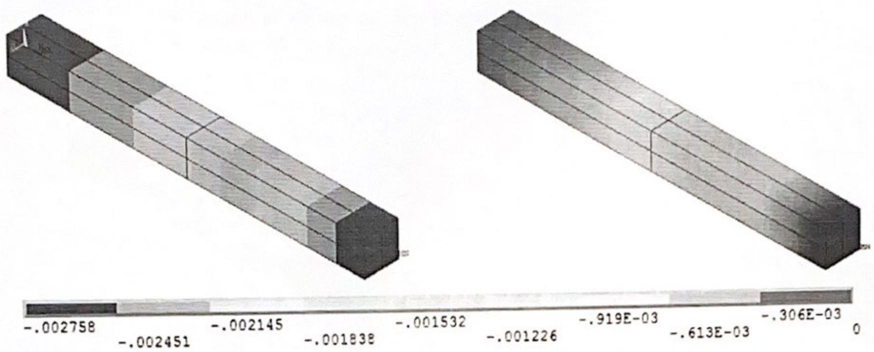

Figure 4-1-3 Plot of Displacement in Y (Isometric View, Element Shapes On): Win32 Graphics (left) and 3D Graphics (right).

Figure 4-1-3 shows that the deformations in the model are uniform through the cross section of the beam.

9-4. Change to the front view

- Click the "Front View" button in the Pan Zoom Rotate menu

9-5. Plot the x component of stress in the beam

- General Postproc > Plot Results > Contour Plot > Nodal Solu
- Choose "Stress"
- Scroll down and choose "X-Component of stress"
- Click OK

Figure 4-1-4 shows the longitudinal stress in the beam. You can see that the maximum stress occurs at the fixed end of the beam and has a value of 30 MPa. This is the same value as predicted by beam bending theory. The maximum stress on the top of the beam is equal and opposite in magnitude to the stress on the bottom of the beam, and there is no stress along the neutral axis or at the end of the beam where the load was applied. This is also consistent with our expectations.

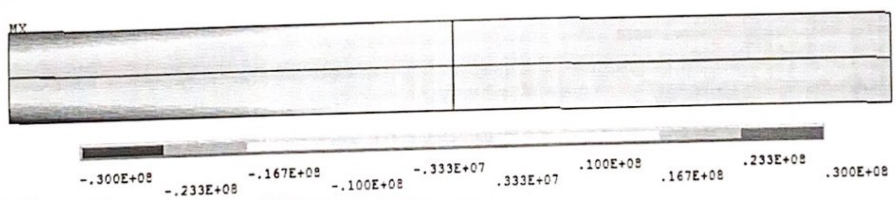

Figure 4-1-4 Plot of X-Component of Stress (Front View, Element Shapes On, 3D Graphics).

9-6. Change to the Isometric view

- Click the "Isometric View" button in the Pan Zoom Rotate menu

9-7. Plot the equivalent stress in the beam

- General Postproc > Plot Results > Contour Plot > Nodal Solu
- Choose "Stress"
- Scroll down and choose "von Mises stress"
- Click OK

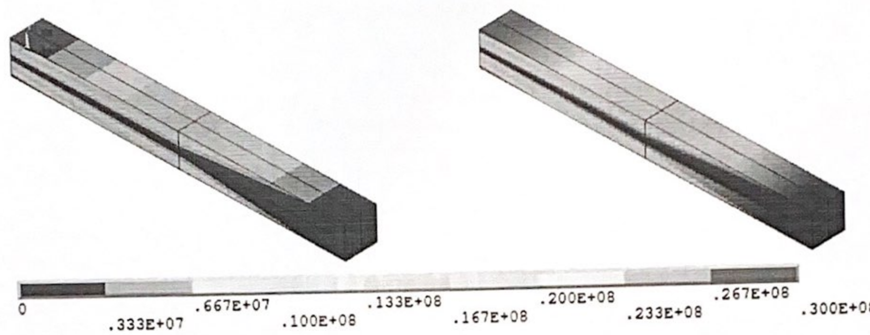

Figure 4-1-5 Plot of von Mises Stress (Isometric View, Element Shapes On): Win32 Graphics (left) and 3D Graphics (right).

Figure 4-1-5 shows that the stress at the free end of the beam is uniformly zero.

Step 10: Compare and Verify the Results

Figure 4-1-3 shows the vertical displacement in the beam. You can see that there is zero displacement at the fixed end of the beam and a maximum displacement of 2.758E-3 m at the free end of the beam. Beam bending theory predicts a maximum displacement of 2.736E-3 m. Thus, the results of this model are within 1% of the theory.

Figure 4-1-4 shows the longitudinal stress in the beam. You can see that the maximum stress occurs at the fixed end of the beam and has a value of 30 MPa. This is the same value as predicted by beam bending theory. The maximum stress on the top of the beam is equal and opposite in magnitude to the stress on the bottom of the beam, and there is no stress along the neutral axis or at the end of the beam where the load was applied. This is also consistent with our expectations.

Figure 4-1-5 shows the equivalent stress (von Mises stress) in the beam. The calculation of equivalent stress involves a square root so equivalent stress is always positive. In this plot, you can see that the equivalent stress is symmetric about the centerline of the beam. Equivalent stress is used in the Theory of Plasticity to determine if the material has yielded. This plot shows that the maximum stress is much lower than the yield stress of 6061-T6 aluminum (275 MPa). This validates our use of a linear elastic material model.

Close the Program

Utility Menu > File > Exit...

Sample Input File

```
/PREP7                    ! Enter the Preprocessor
K,,0,0,0                  ! Create keypoint at (0,0,0)
K,,1,0,0                  ! Create keypoint at (1,0,0)
L,1,2                     ! Create a line between Keypoints 1 and 2
ET,1,BEAM189              ! Use BEAM189 elements
SECTYPE,1,BEAM,RECT       ! Use a rectangular cross section for beam
SECOFFSET,CENT            ! Offset beam node to the centroid
SECDATA,0.1,0.1,          ! Use 0.1 x 0.1 cross section
MP,EX,1,7.31e10           ! Define Young's modulus for material #1
MP,PRXY,1,0.33            ! Define Poisson's ratio for material #1
LMESH,ALL                 ! Mesh the line
FINISH                    ! Finish and Exit Preprocessor

/SOLU                     ! Enter the Solution Processor
DK,1,ALL,0                ! Constrain KP 1 in all DOFs
FK,2,FY,-5000             ! Apply -5000 N in y direction to KP 2
ALLSEL                    ! Select everything
SOLVE                     ! Solve the model
FINISH                    ! Finish and Exit Solution

/POST1                    ! Enter the General Postprocessor
/ESHAPE,1                 ! Display element shapes using section data
/DSCALE,ALL,1             ! Plot using true scale
/VIEW,1,1,1,1             ! Change to isometric view
PLNSOL,U,Y,0,1            ! Plot displacement in y
/VIEW,1,,,1               ! Change to front view
PLNSOL,S,X,0,1            ! Plot stress in x
/VIEW,1,1,1,1             ! Change to isometric view
PLNSOL,S,EQV,0,1          ! Plot the equivalent stress
FINISH                    ! Finish and Exit Postprocessor

SAVE                      ! Save the database
!/EXIT                    ! Exit ANSYS
```