VIET NAM NATIONAL UNIVERSITY HO CHI MINH CITY

HCM UNIVERSITY OF TECHNOLOGY
Faculty of Transportation Engineering
Department of Automotive Engineering

PROJECT REPORT

BUCKLING BEAM

Instructor: PhD Trần Hữu Nhân

Students: Nguyễn Tuấn Kiệt - 2052143

Lâm Huỳnh Phát - 2053313

Course: Finite Element Analysis in Automotive Engineering

Course ID: TR3025 **Semester:** 232

TABLE OF CONTENT

I. Theoretical basis

1. Buckling beam

1.1. Beam

A beam is a structural element that primarily resists loads applied laterally across the beam's axis. Its mode of deflection is usually by bending, as loads produce reaction forces at the beam's support points and internal bending moments, shear, stresses, strains, and deflections.

In this project, we choose the BEAM 188, which is a 2-node beam (defined by nodes I and J in the global coordinate system. BEAM 188 is suitable for analyzing slender to moderately stubby/thick beam structures. It has six or seven degrees of freedom at each node. These include translations in the x, y, and z directions and rotations about the x, y, and z directions.

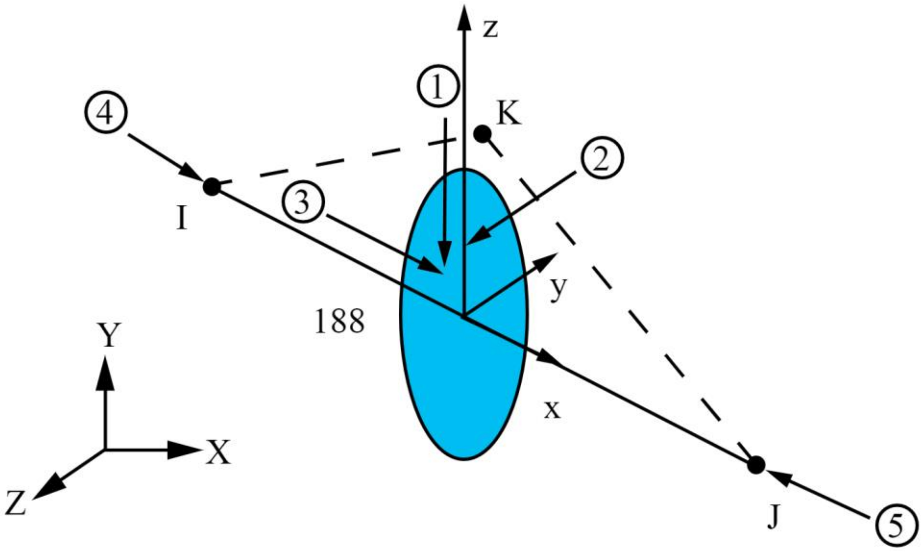

Fig. 1. BEAM188 Geometry.

The element includes stress stiffness terms, by default, in any analysis with large deflection. The provided stress-stiffness terms enable the elements to analyze flexural, lateral, and torsional stability problems.

Elasticity, plasticity, creep and other nonlinear material models are supported. A cross-section associated with this element type can be a built-up section referencing more than one material.

1.2. Buckling

Buckling loads are critical loads where certain types of structures become unstable. Each load has an associated buckled mode shape; this is the shape that the structure assumes in a buckled condition.

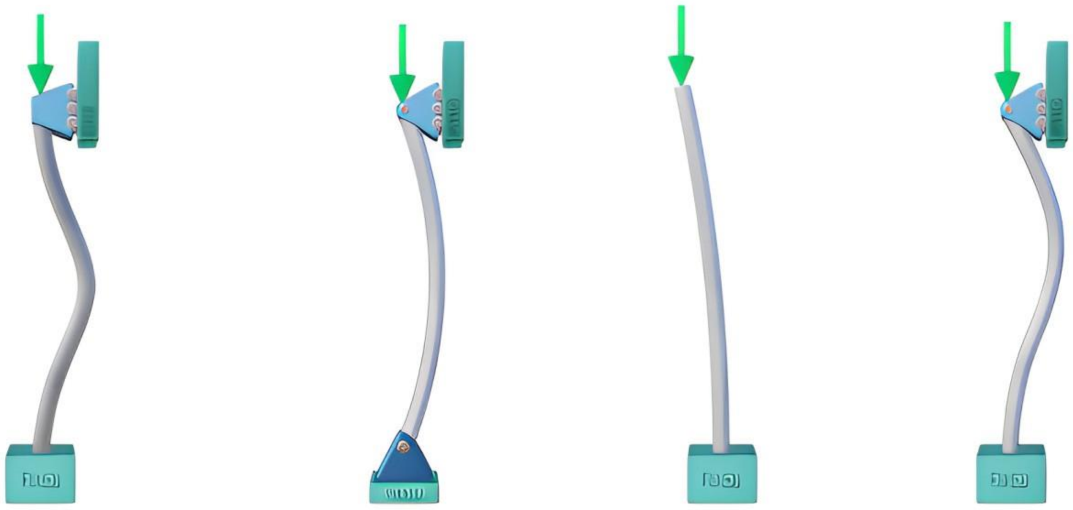

Fig. 2. A beam under buckling load.

If a structure is subjected to a gradually increasing load, when the load reaches a critical level, a member may suddenly change shape and the structure and component is said to have buckled.Euler's critical load and Johnson's parabolic formula are used to determine the buckling stress of a column.

2. Buckling analysis method

There are two primary means to perform a buckling analysis:

Eigenvalue buckling analysis: predicts the theoretical buckling strength of an ideal elastic structure. It computes the structural eigenvalues for the given system loading and constraints. This is known as classical Euler buckling analysis. Buckling loads for several configurations are readily available from tabulated solutions. However, in real life, structural imperfections and nonlinearities prevent most real-world structures from reaching their eigenvalue predicted buckling strength; ie. it over-predicts the expected buckling loads. This method is not recommended for accurate, real-world buckling prediction analysis.

Nonlinear buckling analysis: is more accurate than eigenvalue analysis because it employs non-linear, large-deflection, static analysis to predict buckling loads. Its mode of operation is very simple: it gradually increases the applied load until a load level is found whereby the structure becomes unstable (ie. suddenly a very small increase in the load will cause very large deflections). The true non-linear nature of this analysis thus permits the modeling of geometric imperfections, load perturbations, material nonlinearities, and gaps. For this type of analysis, note that small off-axis loads are necessary to initiate the desired buckling mode.

3. Calculation equations

Euler's buckling formula (1) is a simple equation that is used to calculate the axial load at which a column or beam will buckle. At the critical buckling load any small perturbation, whether it's a lateral force or a small imperfection in the column geometry, will cause the column to buckle.

$$P_{cr} = \frac{\pi^2 EI}{L_e^2} \tag{1}$$

The critical load depends on only three parameters, the Young's modulus E of the column material, the second moment I of area of the column cross-section, and the effective length L_e of the column.

Since the radius of gyration is defined as the square root of the ratio of the column's moment of inertia about an axis to its cross sectional area, the above Euler formula may be reformatted by substituting the radius of gyration Ar^2 for I:

$$\sigma = \frac{F}{A} = \frac{\pi^2 E}{\left(\frac{l}{r}\right)^2} \tag{2}$$

where σ is the stress that causes buckling.

In the Euler column formula, the quantity L/r is referred to as the slenderness ratio:

$$R_s = \frac{L}{r} \tag{3}$$

The slenderness ratio indicates the susceptibility of the column to buckling. Columns with a high slenderness ratio are more susceptible to buckling and are classified as "long" columns.

The transition slenderness ratio (3) can be calculated for a column which indicates the cut-off between long and intermediate columns. The transition slenderness ratio is the value of the slenderness ratio at which the Euler critical stress σ_{cr} is equal to half the material yield strength S_y:

$$R_{trans} = \frac{L_{trans}}{r} = \sqrt{\frac{2\pi^2 E}{K^2 S_y}} \tag{4}$$

Columns with a slenderness ratio less than the transition slenderness ratio are considered intermediate columns.

The maximum deflection in the column can be found by:

$$\delta_{max} = e\left[\sec\left(\frac{KL}{2r}\sqrt{\frac{P}{AE}}\right) - 1\right] \tag{5}$$

The maximum compressive stress in the column can be found by:

$$\sigma_{c,max} = \frac{P}{A}\left[1 + \frac{ec}{r^2}\sec\left(\frac{KL}{2r}\sqrt{\frac{P}{AE}}\right)\right] \tag{6}$$

Where

e - the eccentricity

sec =1/cos

K- effective length factor

L - the actual unsupported length of the column

A - the cross-sectional area

E - the elastic modulus of the material

P - the applied force

c - the centroidal distance

r - the radius of gyration

II. Buckling analysis

This project will use a steel beam with a 10 mm X 10 mm cross section, rigidly constrained at the bottom. The required load to cause buckling, applied at the top-center of the beam, will be calculated.

1. Eigenvalue Buckling Analysis

1.1. Preprocessing: Defining the Problem

<u>Open preprocessor menu:</u>

```
/PREP7
```

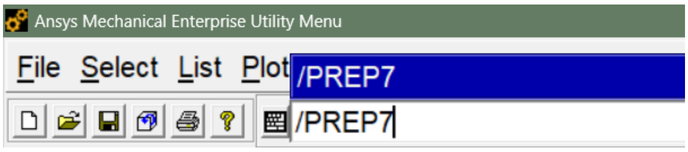

Fig. 3. Open preprocessor menu.

<u>Give example a Title:</u>

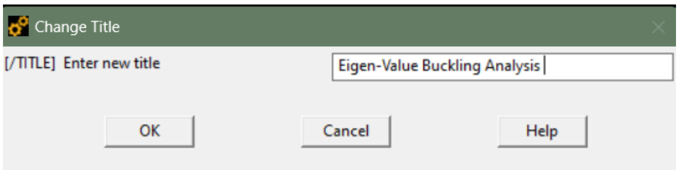

Fig. 4. Name the title.

<u>Define Keypoints:</u>

Preprocessor > Modeling > Create > Keypoints > In Active CS ...

We are going to define 2 Keypoints for this beam as given in the following table:

Keypoints	Coordinates (x,y)
1	(0,0)
2	(0,100)

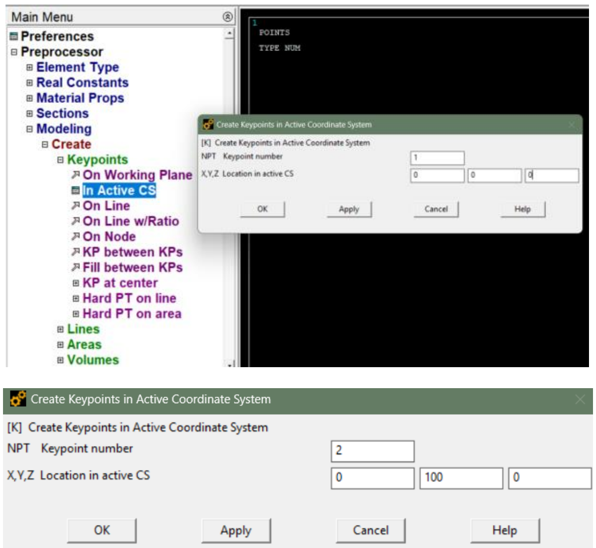

Fig. 5. Define keypoints.

Create Lines:

Preprocessor > Modeling > Create > Lines > Lines > In Active Coord

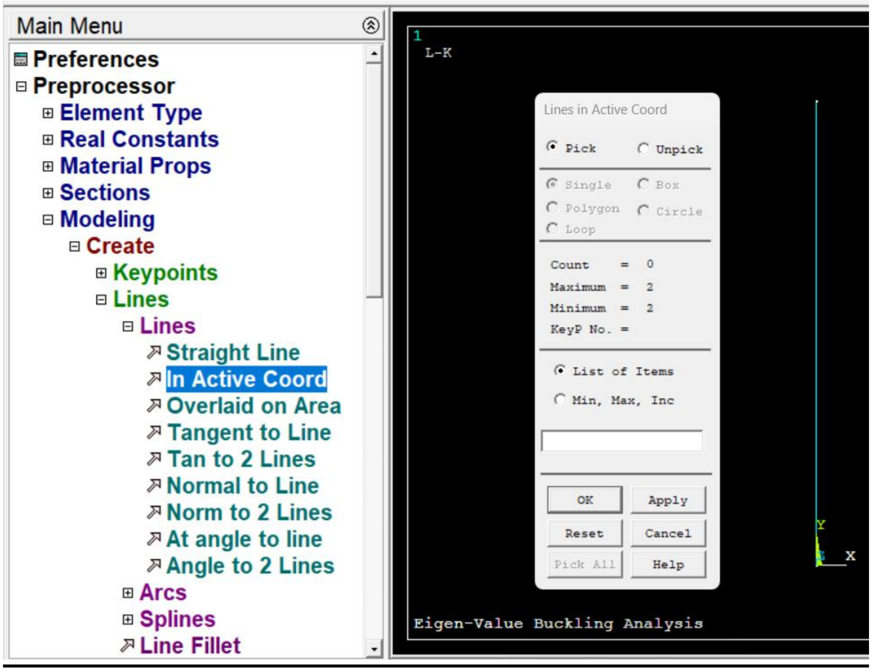

Fig. 6. Create lines.

Define the Type of Element:

Preprocessor > Element Type > Add/Edit/Delete

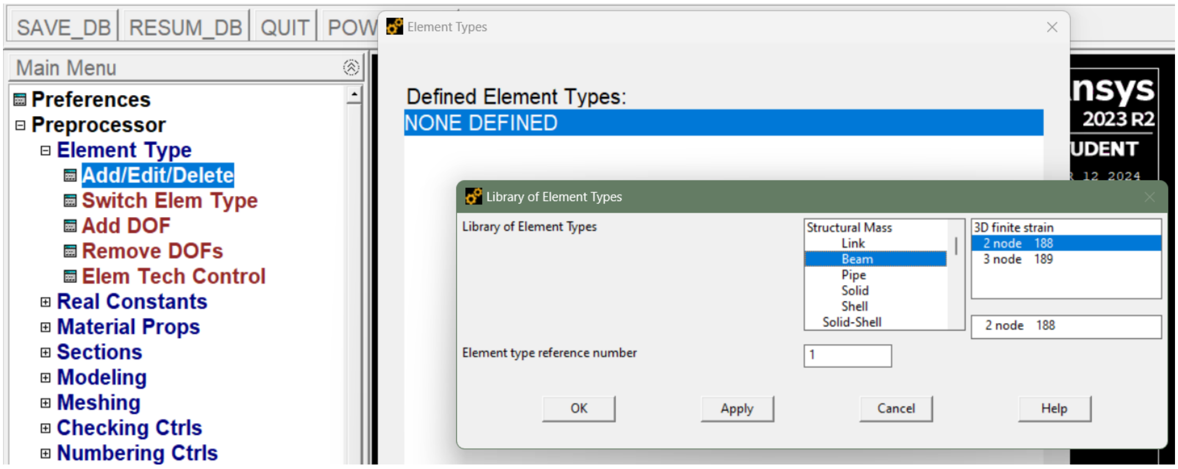

Fig. 7. Define the Type of Element.

Define Parameters:

Parameters > Scalar Parameters > Selection

In the 'Scalar Parameters' window, enter the following geometric properties:

i. Cross-sectional area AREA: 100

ii. Area moment of inertia IZZ: 833.333

iii. Total Beam Height HEIGHT: 10

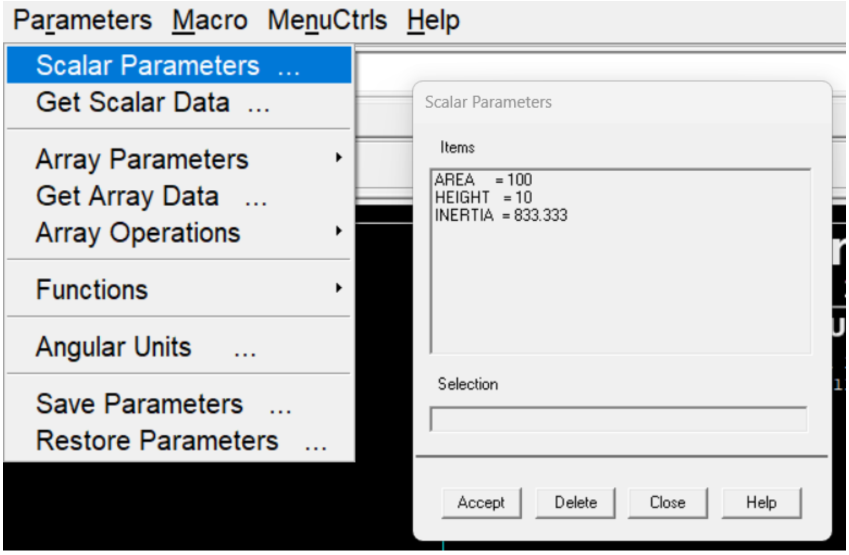

Fig. 8. Define Parameters.

This defines a beam with a height of 10 mm and a width of 10 mm.

<u>Define Element Material Properties:</u>

Preprocessor > Material Props > Material Models > Structural > Linear > Elastic > Isotropic

In the window that appears, enter the following geometric properties for steel:

i. Young's modulus EX: 200000

ii. Poisson's Ratio PRXY: 0.3

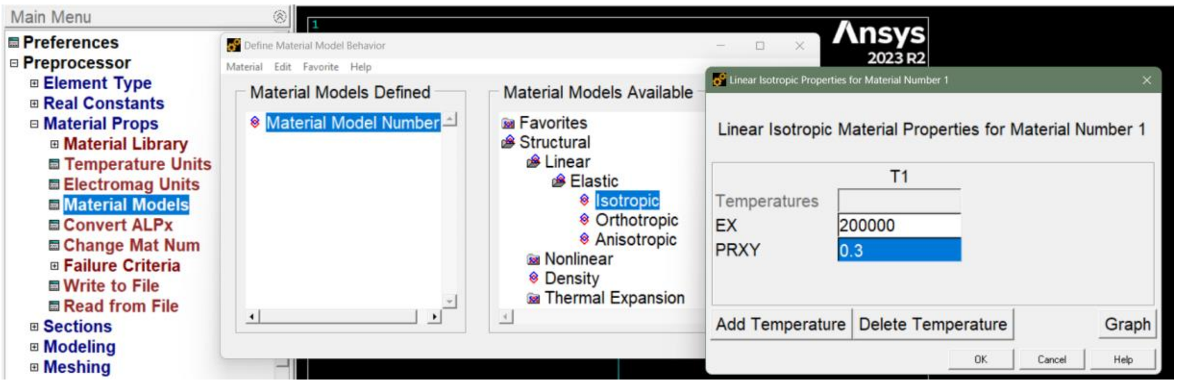

Fig. 9. Define Element Material Properties.

Define Mesh Size:

Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > All Lines...

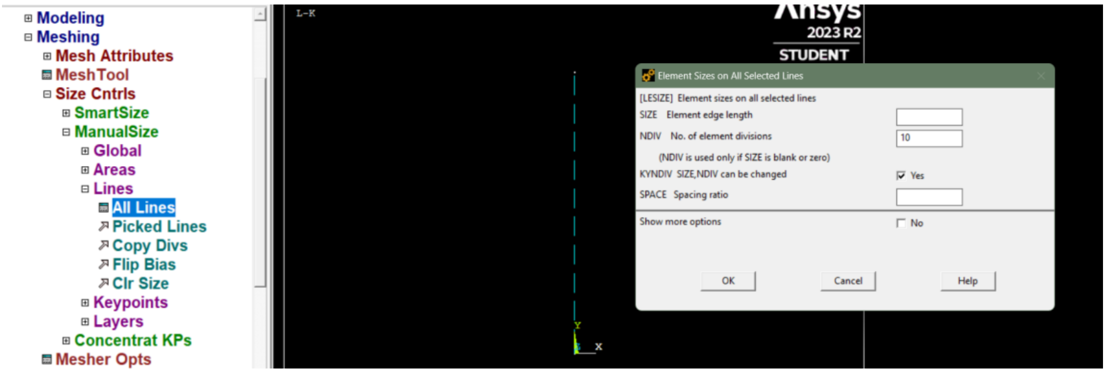

Fig. 10. Define Mesh Size.

For this project, we will specify an element edge length of 10 mm (10 element divisions along the line).

Mesh the frame:

Preprocessor > Meshing > Mesh > Lines > click 'Pick All'

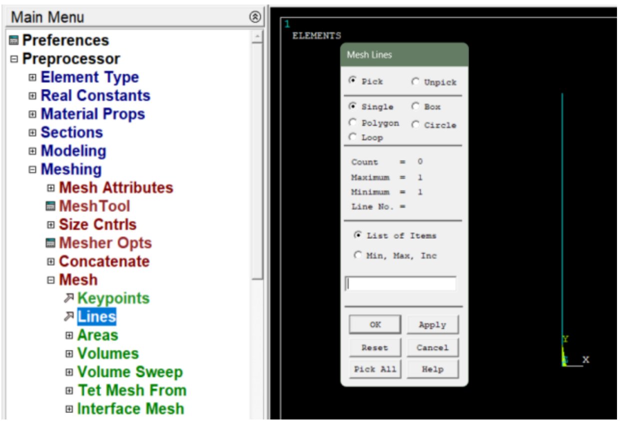

Fig. 11. Mesh the frame.

1.2. Solution Phase: Assigning Loads and Solving

Define Analysis Type:

Solution > Analysis Type > New Analysis > Static

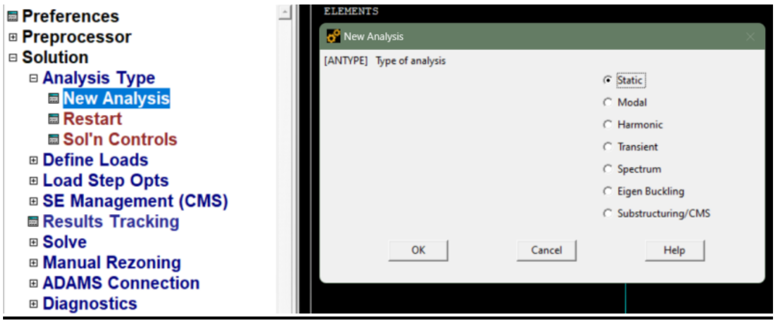

Fig. 12. Define analysis type.

<u>Apply Constraints:</u>

Solution > Define Loads > Apply > Structural > Displacement > On Keypoints

Fix Keypoint 1 (ie all DOF constrained).

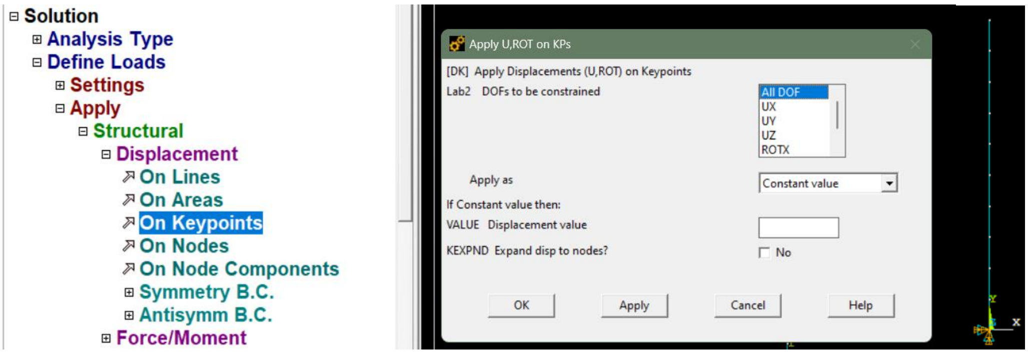

Fig. 13. Apply constaints.

Fix Keypoint 1 (ie all DOF constrained).

<u>Apply Loads:</u>

Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints

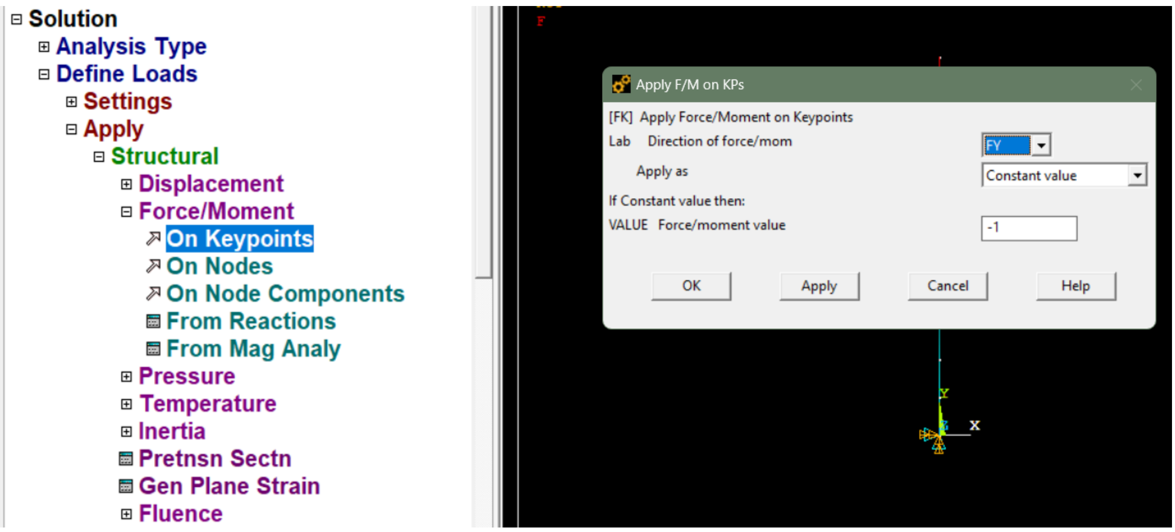

Fig. 14. Apply loads.

The eigenvalue solver uses a unit force to determine the necessary buckling load. Applying a load other than 1 will scale the answer by a factor of the load. Apply a vertical (FY) point load of -1 N to the top of the beam (keypoint 2).

The applied loads and constraints should now appear as shown in the figure below.

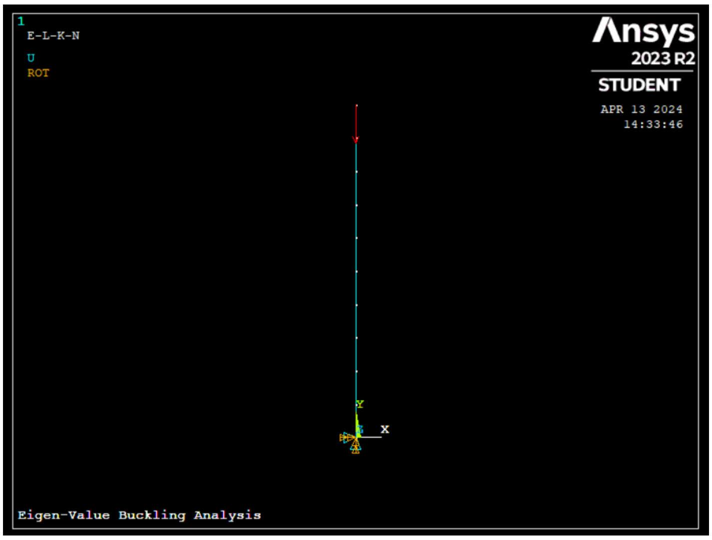

Fig. 15. Applied loads and constraints.

<u>Solve the System:</u>

Solution > Solve > Current LS

Fig. 16. Solve the system.

<u>Exit the Solution processor:</u>

Close the solution menu and click **FINISH** at the bottom of the Main Menu.

Normally at this point you enter the postprocessing phase. However, with a buckling analysis you must re-enter the solution phase and specify the buckling analysis. Be sure to close the solution menu and reenter it or the buckling analysis may not function properly.

Define Analysis Type:

Solution > Analysis Type > New Analysis > Eigen Buckling

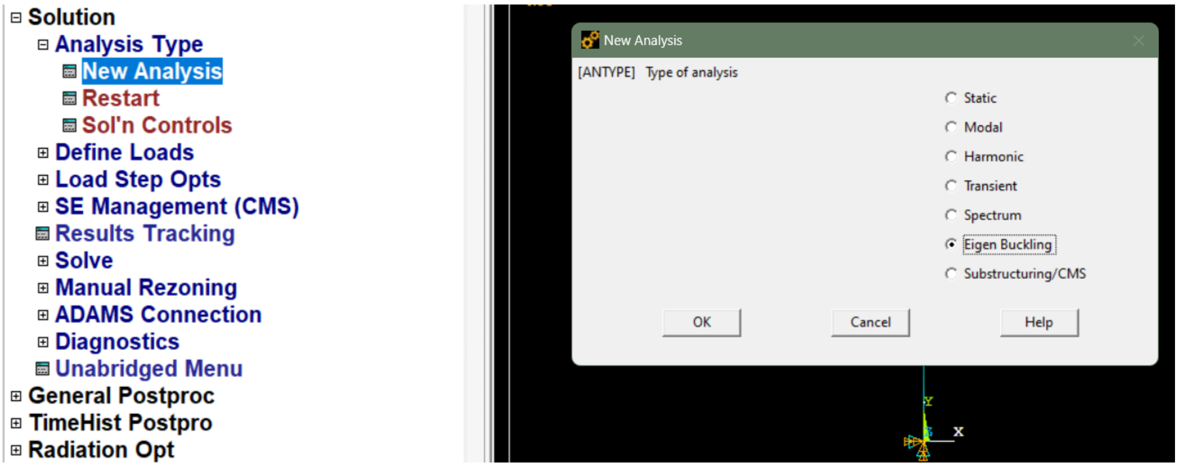

Fig. 17. Define analysis type.

Specify Buckling Analysis Options:

Select Solution > Analysis Type > Analysis Options

Complete the window that appears, as shown below. Select 'Block Lanczos' as an extraction method and extract 1 mode. The 'Block Lanczos' method is used for large symmetric eigenvalue problems and uses the sparse matrix solver. The 'Subspace' method could also be used, however it tends to converge slower as it is a more robust solver. In more complex analyses the Block Lanczos method may not be adequate and the Subspace method would have to be used.

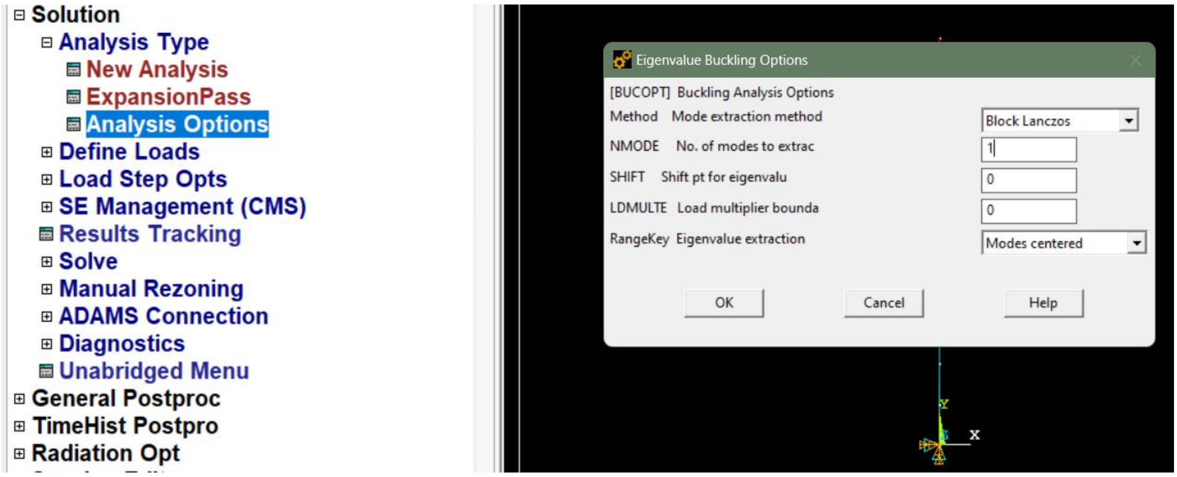

Fig. 18. Specify Buckling Analysis Options.

Solve the System:

Solution > Solve > Current LS

Expand the solution:

Select Solution > Analysis Type > Expansion Pass... and ensure that it is on. You may have to select the 'Unabridged Menu' again to make this option visible.

Select Solution > Load Step Opts > ExpansionPass > Single Expand > Expand Modes ...

Complete the following window as shown to expand the first mode.

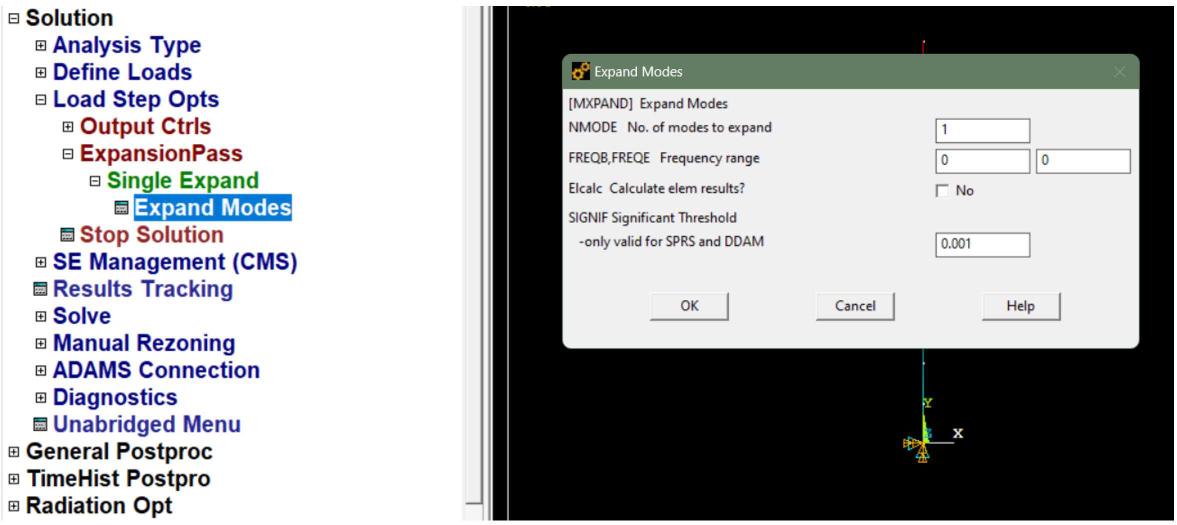

Fig. 19. Expand the solutions.

<u>Solve the System</u>

Solution > Solve > Current LS

1.3. Postprocessing: Viewing the Results

<u>View the Buckling Load:</u>

To display the minimum load required to buckle the beam select General Postproc > List Results > Detailed Summary.

The value listed under 'TIME/FREQ' is the load (41,123), which is in Newtons for this example. If more than one mode was selected in the steps above, the corresponding loads would be listed here as well.

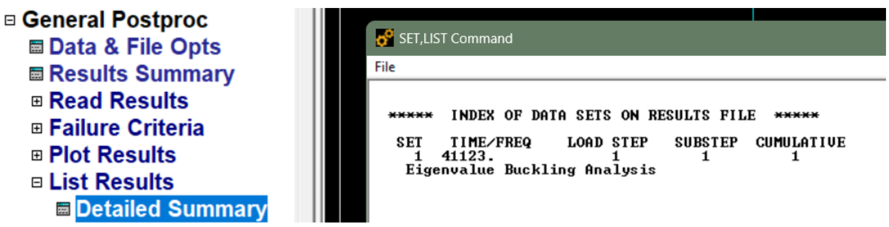

Fig. 20. View the buckling load.

<u>Display the Mode Shape:</u>

Select General Postproc > Read Results > Last Set to bring up the data for the last mode calculated.

Select General Postproc > Plot Results > Deformed Shape

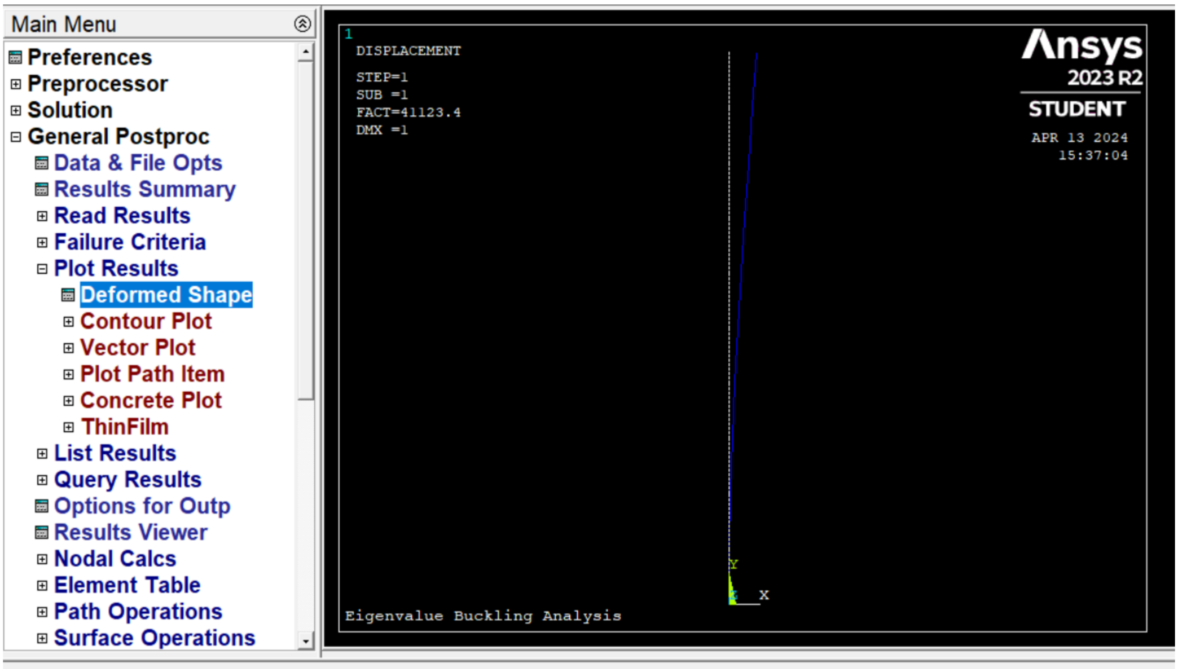

Fig. 21. Deformed shape.

1.4. Command File Mode of Solution

FINISH ! These two commands clear current data

/CLEAR

/TITLE,Eigenvalue Buckling Analysis

/PREP7 ! Enter the preprocessor

ET,1,BEAM3 ! Define the element of the beam to be buckled

R,1,100,833.333,10 ! Real Consts: type 1, area (mm^2), I (mm^4), height (mm)

MP,EX,1,200000 ! Young's modulus (in MPa)

MP,PRXY,1,0.3 ! Poisson's ratio

K,1,0,0 ! Define the geometry of beam (100 mm high)

```
K,2,0,100
L,1,2                   ! Draw the line
ESIZE,10                ! Set element size to 1 mm
LMESH,ALL,ALL           ! Mesh the line
FINISH
/SOLU                   ! Enter the solution mode
ANTYPE,STATIC           ! Before you can do a buckling analysis, ANSYS
                        ! needs the info from a static analysis
PSTRES,ON               ! Prestress can be accounted for - required
                        ! during buckling analysis
DK,1,ALL                ! Constrain the bottom of beam
FK,2,FY,-1              ! Load the top vertically with a unit load.
                        ! This is done so the eigenvalue calculated
                        ! will be the actual buckling load, since
                        ! all loads are scaled during the analysis.
SOLVE
FINISH
/SOLU                        ! Enter the solution mode again to solve buckling
ANTYPE,BUCKLE                ! Buckling analysis
BUCOPT,LANB,1                ! Buckling options - subspace, one mode
SOLVE
FINISH
/SOLU                        ! Re-enter solution mode to expand info -
necessary
```

EXPASS,ON ! An expantion pass will be performed

MXPAND,1 ! Specifies the number of modes to expand

SOLVE

FINISH

/POST1 ! Enter post-processor

SET,LIST ! List eigenvalue solution - Time/Freq listing is the

! force required for buckling (in N for this case).

SET,LAST ! Read in data for the desired mode

PLDISP ! Plots the deflected shape

2. NonLinear Buckling Analysis

2.1. Preprocessing: Definening the Problem

Open preprocessor menu:

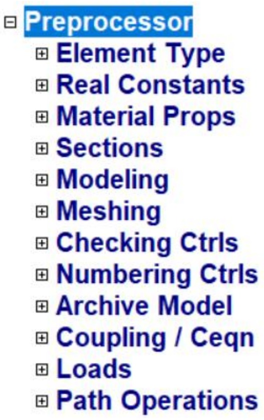

Fig. 22. Preprocessor menu.

Give example a Title:

Utility Menu >File > Change Title ...

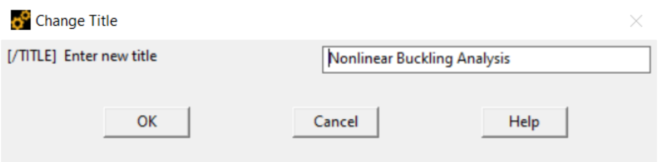

Fig. 23. New title.

Create Keypoints:

Preprocessor > Modeling > Create > Keypoints > In Active CS

We are going to define 2 keypoints (the beam vertices) for this structure to create a beam with a length of 100 millimeters:

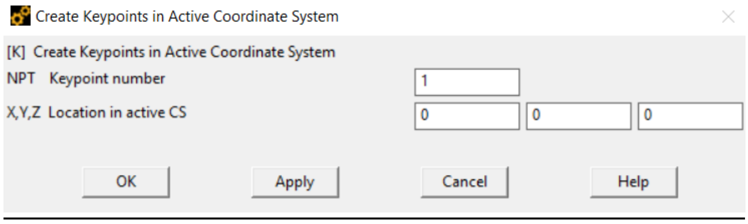

Fig. 24. Create keypoint 1 (0,0).

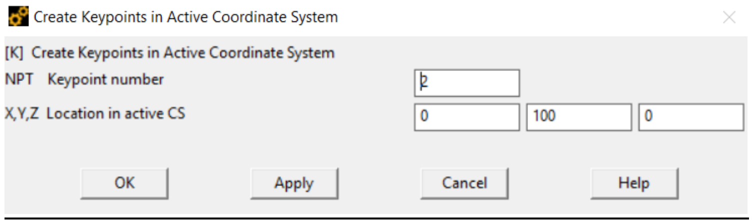

Fig. 25. Create keypoint 2 (0,100).

Define Lines:

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

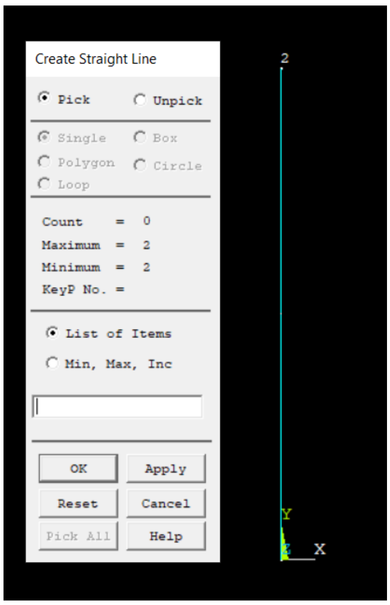

Fig. 26. Create a line between Keypoint 1 and Keypoint 2.

Define Element Types:

Preprocessor > Element Type > Add/Edit/Delete...

For this problem we will use the BEAM188 (Beam 2D elastic) element.

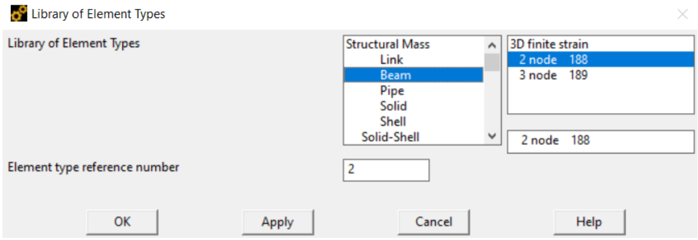

Fig. 27. Choose Beam 188.

<u>Define Parameters:</u>

Utility Menu > Scalar Parameters... > Add...

In the Scalar Parameters window, enter the following geometric properties:

i. Cross-sectional area AREA: 100

ii. ii. Area Moment of Inertia IZZ: 833.333

iii. iii. Total beam height HEIGHT: 10

This defines an element with a solid rectangular cross section 10 x 10 millimeters.

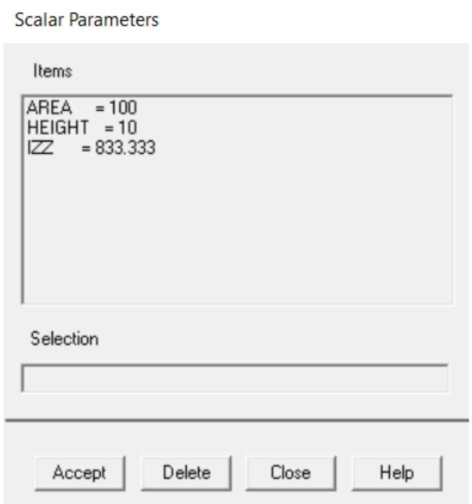

Fig. 28. Enter geometric properties.

<u>Define Element Material Properties:</u>

Preprocessor > Material Props > Material Models > Structural > Linear > Elastic > Isotropic

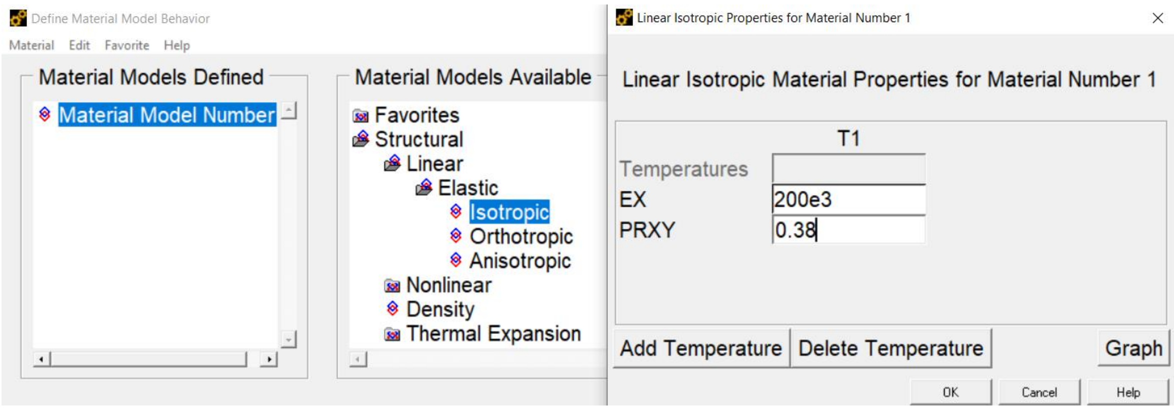

Fig. 29. Define material properties.

In the window that appears, enter the following geometric properties for steel:

i. Young's modulus: 200e3

ii. ii. Poisson's Ratio: 0.38.

<u>Define Mesh Size:</u>

Preprocessor > Meshing > Manual Size > Size Cntrls > Lines > All Lines...

For this project we will specify an element edge length of 1 mm (100 element divisions along the line).

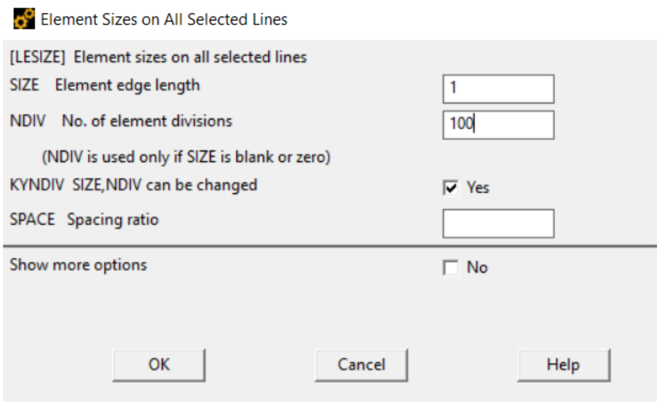

Fig. 30. Mesh the line.

<u>Mesh the frame:</u>

Preprocessor > Meshing > Mesh > Lines > click 'Pick All'

2.2. Solution: Assigning Loads and Solving

<u>Define Analysis Type:</u>

Solution > New Analysis > Static

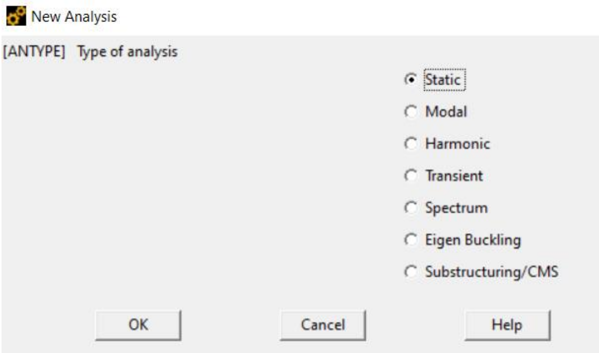

Fig. 31. Choose static analysis.

<u>Set Solution Controls:</u>

Select Solution > Analysis Type > Sol'n Control...

The following image will appear:

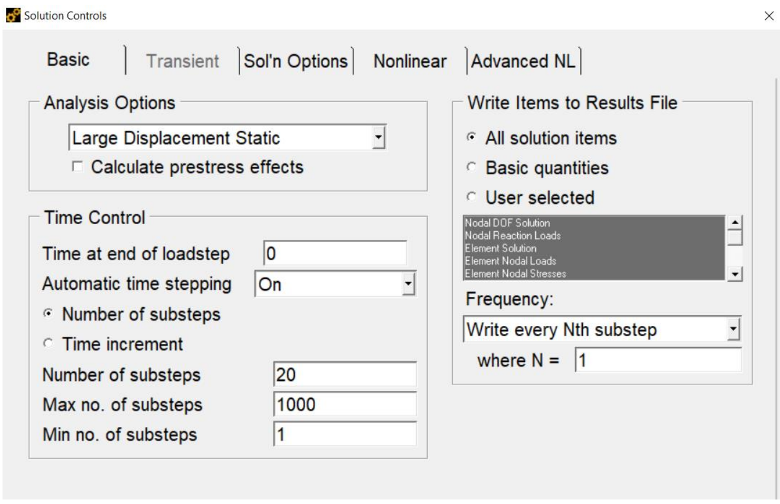

Fig. 32. Sol’n control UI (Basic tab).

Ensure the following selections are made under the 'Basic' tab (as shown above)

A. Ensure Large Static Displacements are permitted (this will include the effects of large deflection in the results)

B. Ensure Automatic time stepping is on. Automatic time stepping allows ANSYS to determine appropriate sizes to break the load steps into. Decreasing the step size usually ensures better accuracy, however, this takes time. The Automatic Time Step feature will determine an appropriate balance. This feature also activates the ANSYS bisection feature which will allow recovery if convergence fails.

C. Enter 20 as the number of substeps. This will set the initial substep to 1/20 th of the total load.

D. Enter a maximum number of substeps of 1000. This stops the program if the solution does not converge after 1000 steps.

E. Enter a minimum number of substeps of 1.

F. Ensure all solution items are written to a results file.

Ensure the following selection is made under the 'Nonlinear' tab (as shown below)

A. Ensure Line Search is 'On'. This option is used to help the Newton-Raphson solver converge.

B. Ensure Maximum Number of Iterations is set to 1000.

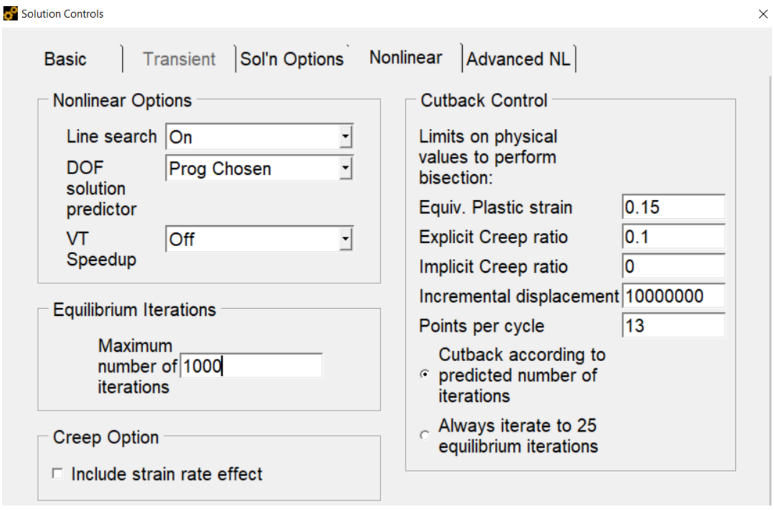

Fig. 33. Sol’n control UI (Nonlinear tab).

NOTE There are several options which have not been changed from their default values. For more information about these commands, type help followed by the command into the command line.

<u>Apply Constraints:</u>

Solution > Define Loads > Apply > Structural > Displacement > On Keypoints.

Fix Keypoint 1 (ie all DOFs constrained).

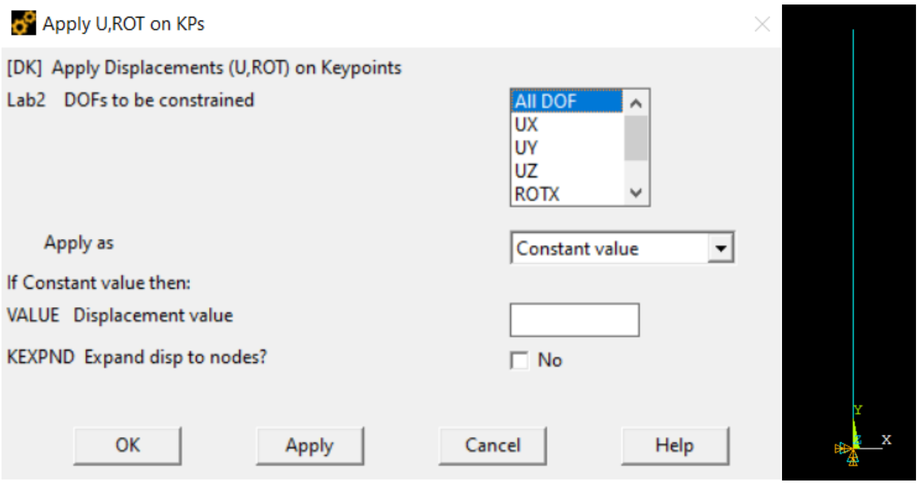

Fig. 34. Fix Keypoint 1.

Apply Loads Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints

Place a -50,000 N load in the FY direction on the top of the beam (Keypoint 2). Also apply a -250 N load in the FX direction on Keypoint 2. This horizontal load will persuade the beam to buckle at the minimum buckling load.

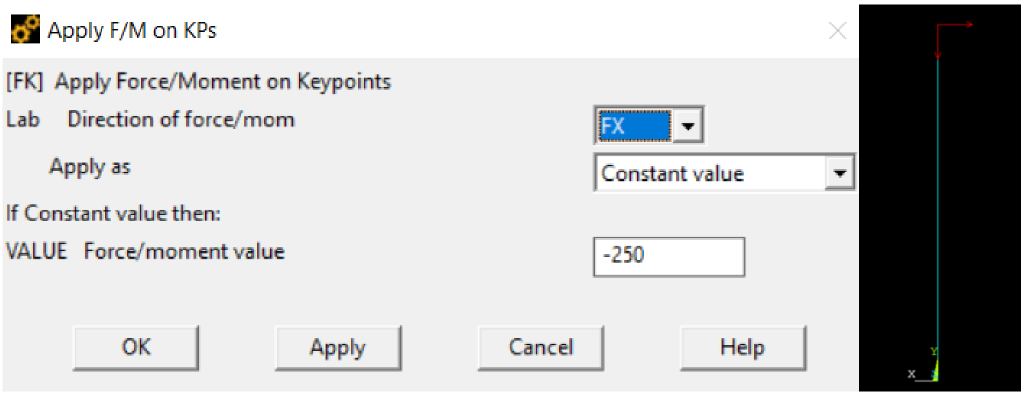

Fig. 35. Apply Forces on Keypoint 2.

<u>Solve the System:</u>

Solution > Solve > Current

The following will appear on your screen for NonLinear Analyses

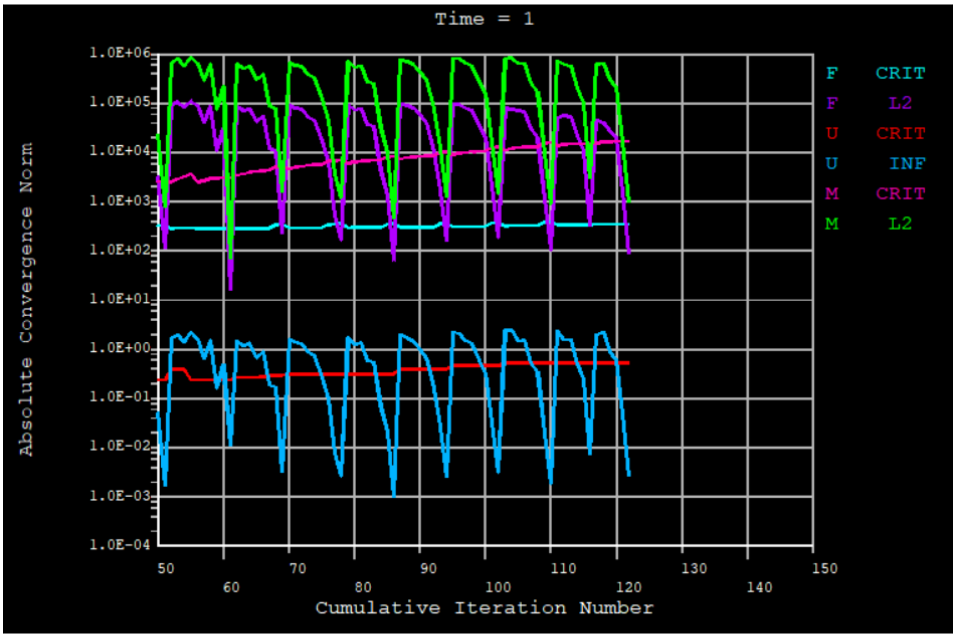

Fig. 36. The convergence of the solution.

2.4. General Postprocessing: Viewing the Results

View the deformed shape:

To view the element in 2D rather than a line: Utility Menu > PlotCtrls > Style > Size and Shape and turn 'Display of element' ON (as shown below).

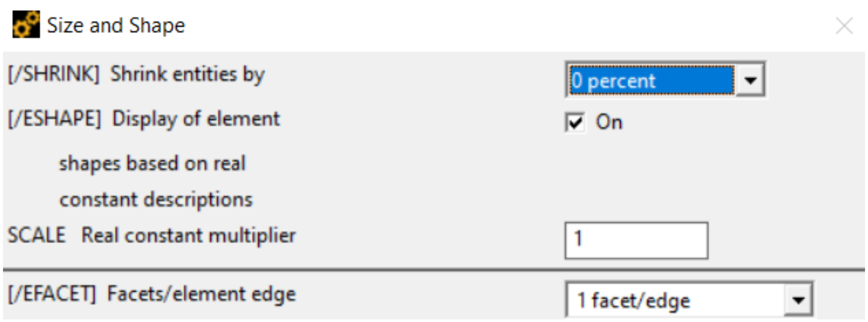

Fig. 37. Turn on display of element.

General Postproc > Read Result > First set > Plot Results > Deformed Shape... > Def + undeformed

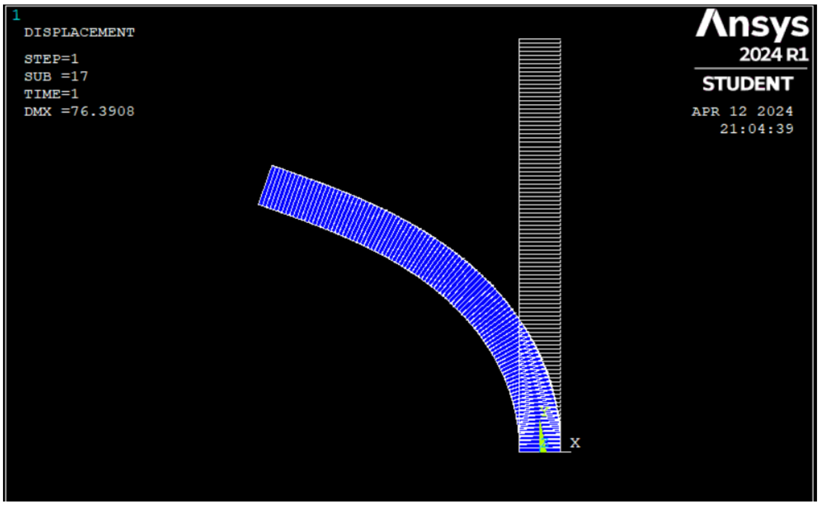

Fig. 38. Deformed shape of the beam.

<u>View the deflection contour plot:</u>

General Postproc > Plot Results > Contour Plot > Nodal Solu... > DOF solution, UY

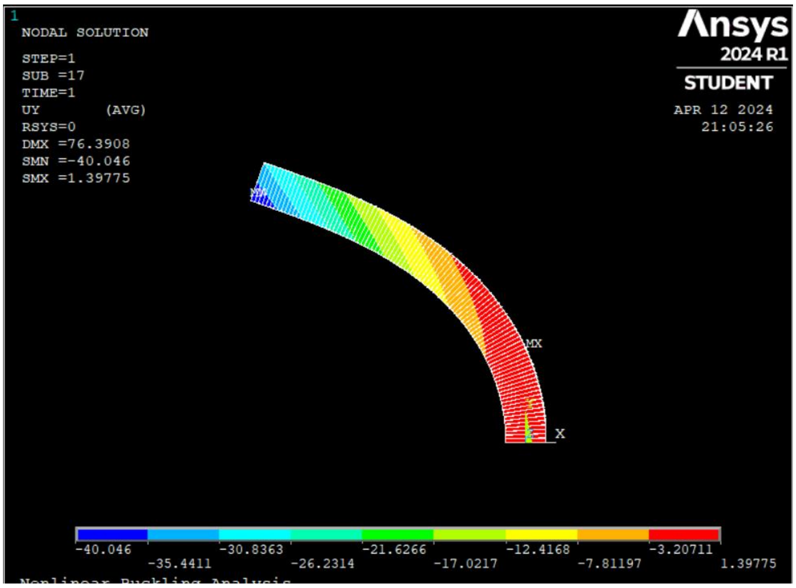

Fig. 39. UY stress plot.

2.4. Time History Postprocessing: Viewing the Results

<u>Define Variables:</u>

Select: Main Menu > TimeHist Postpro.

The following window should open automatically.

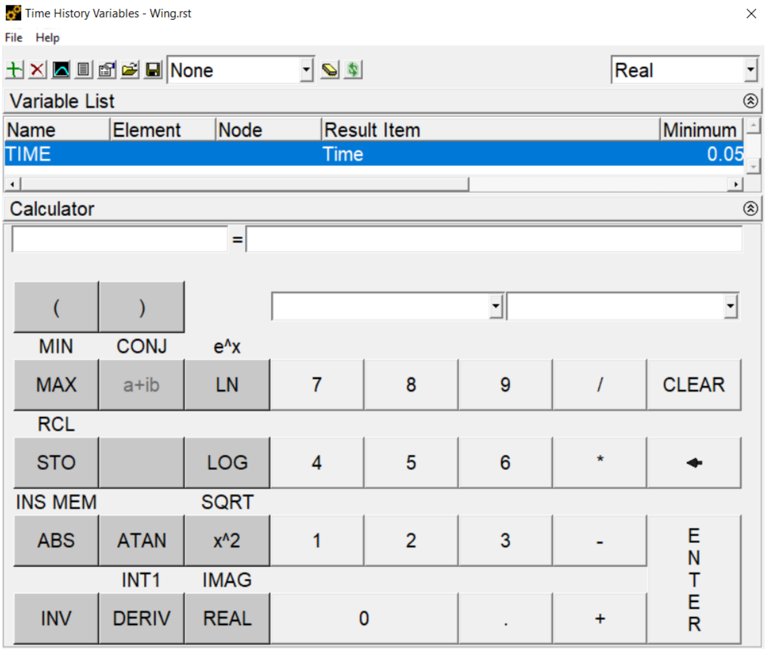

Fig. 40. TimeHist Postpro.

If it does not open automatically, select Main Menu > TimeHist Postpro > Variable Viewer.

Click the add button in the upper left corner of the window to add a variable.

Double-click Nodal Solution > DOF Solution > Y-Component of displacement (as shown below) and click OK. Pick the uppermost node on the beam and click OK in the 'Node for Data' window.

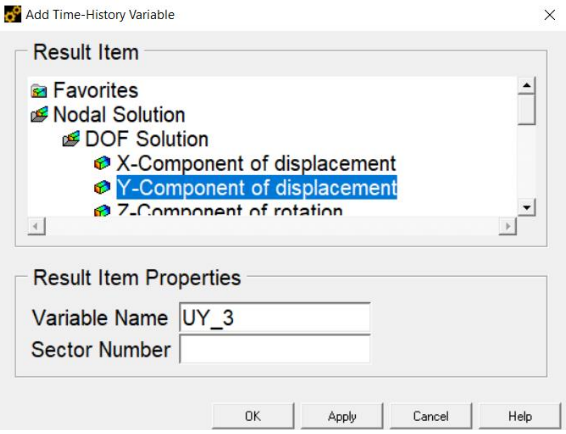

Fig. 41. Add Time-History Variable.

To add another variable, click the add button again. This time select Reaction Forces > Structural Forces > Y-Component of Force. Pick the lowermost node on the beam and click OK.

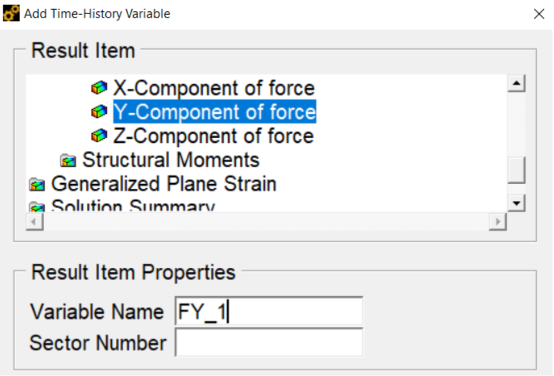

Fig. 42. Add Y Force.

<u>Graph Results over Time:</u>

Click on UY_1 in the Time History Variables window.

Click the graphing button in the Time History Variables window.

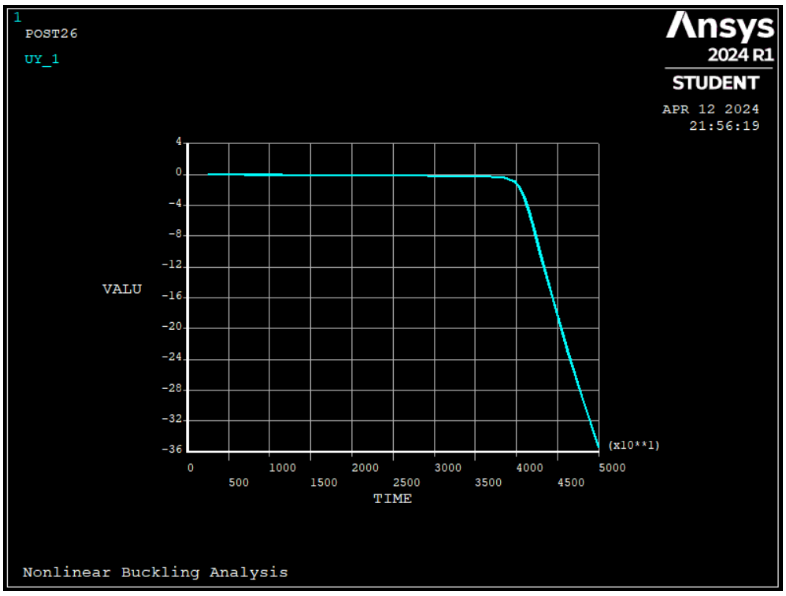

Fig. 43. UY stress over-time graph.

The labels on the plot are not updated by ANSYS, so you must change them manually.

Select Utility Menu > Plot Ctrls > Style > Graphs > Modify Axes and re-label the X and Y-axis appropriately.

Fig. 44. Change the graph labels.

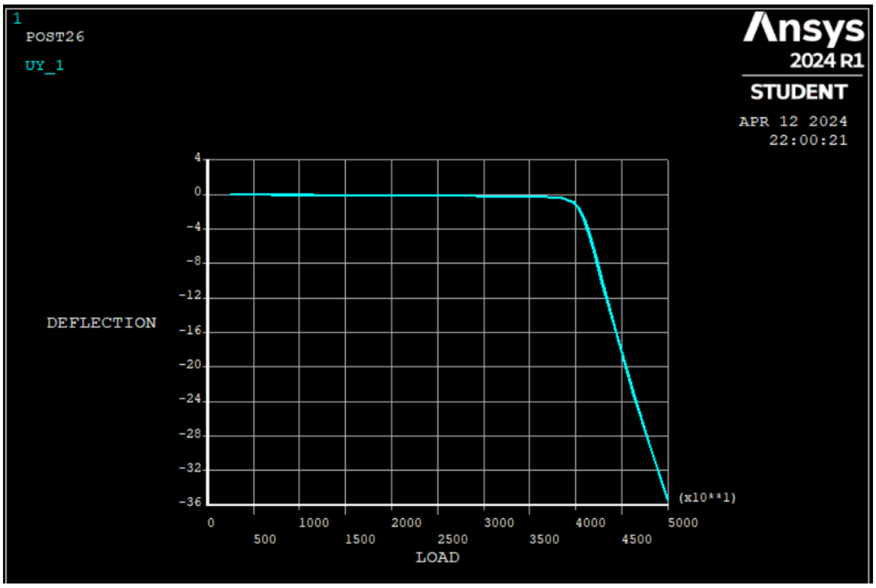

Fig. 45. Final result plot.

The plot shows how the beam became unstable and buckled with a load of approximately 40,000 N, the point where a large deflection occured due to a small increase in force. This is slightly less than the eigenvalue solution of 41,123 N, which was expected due to non-linear geometry issues discussed above.

Full APDL code:

```
FINISH                                ! These two commands clear current data
/CLEAR
/TITLE, Nonlinear Buckling Analysis
/PREP7                                ! Enter the preprocessor
ET,1,BEAM188                          ! Define element
MP,EX,1,200000                        ! Young's modulus (in Pa)
```

```
MP,PRXY,1,0.3                 ! Poisson's ratio
R,1,100,833.333,10            ! area, I, height
K,1,0,0,0                     ! Lower node
K,2,0,100,0                   ! Upper node (100 mm high)
L,1,2                         ! Draws line
ESIZE,1                       ! Sets element size to 1 mm
LMESH,ALL                     ! Mesh line
FINISH
 /SOLU ANTYPE,STATIC          ! Static analysis (not buckling)
NLGEOM,ON                     ! Non-linear geometry solution supported
OUTRES,ALL,ALL                ! Stores bunches of output
NSUBST,20                     ! Load broken into 5 load steps
NEQIT,1000                    ! Use 20 load steps to find solution
AUTOTS,ON                     ! Auto time stepping
LNSRCH,ON
/ESHAPE,1                     ! Plots the beam as volume
DK,1,ALL,0                    ! Constrain bottom
FK,2,FY,-50000                ! Apply load slightly greater than predicted
                              ! required buckling load to upper node
FK,2,FX,-250                  ! Add a horizontal load (0.5% FY) to
                              initiate buckling
SOLVE FINISH /POST26          ! Time history post processor
RFORCE,2,1,F,Y                ! Reads force data in variable 2
```

```
NSOL,3,2,U,Y                 ! Reads y-deflection data into var 3
XVAR,2                       ! Make variable 2 the x-axis
PLVAR,3                      ! Plots variable 3 on y-axis
/AXLAB,Y,DEFLECTION          ! Changes y label
/AXLAB,X,LOAD                ! Changes X label /REPLOT
```

III. REFERENCES

1. J.J. Lugthart, Buckling of a Beam: The force-engineering strain curve. Leiden University (2013).

2. https://en.wikipedia.org/wiki/Buckling, last accessed 2024/4/13.

3. https://mechanicalc.com/reference/column-buckling, last accessed 2024/4/13.

4. https://fr.scribd.com/document/622062208/Deflection-Buckling, last accessed 2024/4/13.

5. https://www.bu.edu/moss/mechanics-of-materials-beam-buckling/, last accessed 2024/4/13.