Advanced Structural Analysis using ANSYS

Project 6:
Buckling Analysis

Group members

Kachhadiya, Devang Keshavlal (3127394)
Maduku Naga, Sasank (3129054)
Panchal, Nikunj Ashokbhai (3105056)
Yassin, Ahmed (3076314)

Submitted to:
Prof. Dr.-Ing. habil. Jörg Schröder
Fakult¨at fur Ingenieurwissenschaften
Abteilung Bauwissenschaften
Universität Duisburg-Essen
Essen, Germany

SEPTEMBER, 2022

Contents

1 Project description

Explore ANSYS's buckling simulation capabilities. The two main techniques offered by ANSYS to address such issues are eigenvalue buckling analysis and nonlinear buckling analysis, and these must be used to solve the straightforward buckling problem depicted in the above image.

The beam under consideration is meant to have a hard bottom constraint. Calculate the load that must be applied at the top-center of the beam to cause buckling.

- geometric data:
$$a = 0.1m, b = 0.12m, l = 1.2m$$
- material data:
$$E = 2.1 \times 10^5 MPa, v = 0.3, \rho = 7.9g/cm^3$$

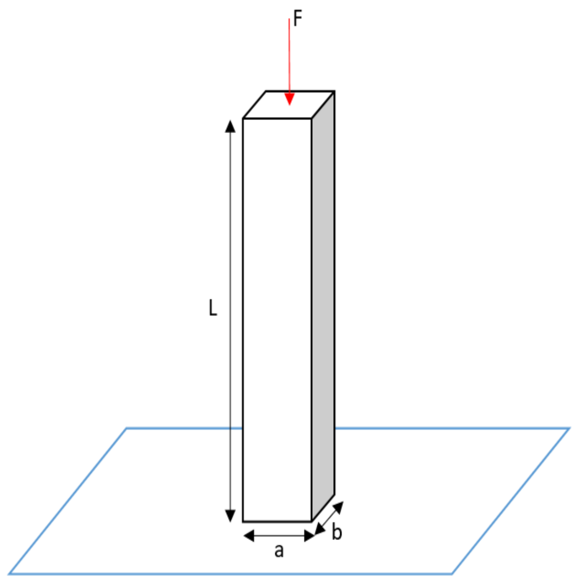


Figure 1: Problem Description

General Aspects

- choose the proper material model and finite element type.
- Convergence analysis to determine the required mesh density
- Result analysis
- the analytical answer is compared (if possible)
- Direct production of all charts and graphs using ANSYS' APDL is required (not necessary diagrams related to convergence study)
- Using the information provided, create a suitable boundary value issue in ANSYS. An APDL-script must be used for every ANSYS activity.

2 Theory

2.1 Buckling Analysis

Material failure and structural instability, also referred to as buckling, are the two main causes of the rapid breakdown of a mechanical component. Consider the yield stress for ductile materials and the ultimate stress for brittle materials when analyzing material failures. Axial tension and axial compression tests on short columns of the material are used to determine the material's characteristics. Such test specimens' geometry has been standardized. Thus, when defining material properties like yield stress, geometry is not expressly addressed. Because the stresses are calculated using analytical or numerical techniques, geometry only indirectly affects the problem of identifying material failure. It is possible to predict material failure using linear finite element analysis. Specifically, by resolving a system of linear equations for the unknown displacements, $K\delta = F$.

In contrast to material strength, a component's stiffness determines the load at which buckling takes place. Buckling, which is typically independent of material strength, is the loss of stability of a component. In the elastic region of the material, this loss of stability typically happens. Separate differential equations control the two phenomena. In contrast to the typical linear finite element analysis, buckling failure is largely characterized by a loss of structural stiffness and is modeled by a finite element eigenvalue–eigenvector solution,$|K - \lambda m\, K_F|\lambda m = 0$, where λm is the buckling load factor (BLF) for the m-th mode, K_F is the additional geometric stiffness due to the stresses caused by the loading, F, and δm is the associated buckling displacement shape for the m-th mode.

Although the load's spatial distribution is significant, its relative magnitude is not. A multiplier that scales the size of the load (up or down) to that necessary to cause buckling is provided by the buckling calculation.

2.2 Eigenvalue analysis

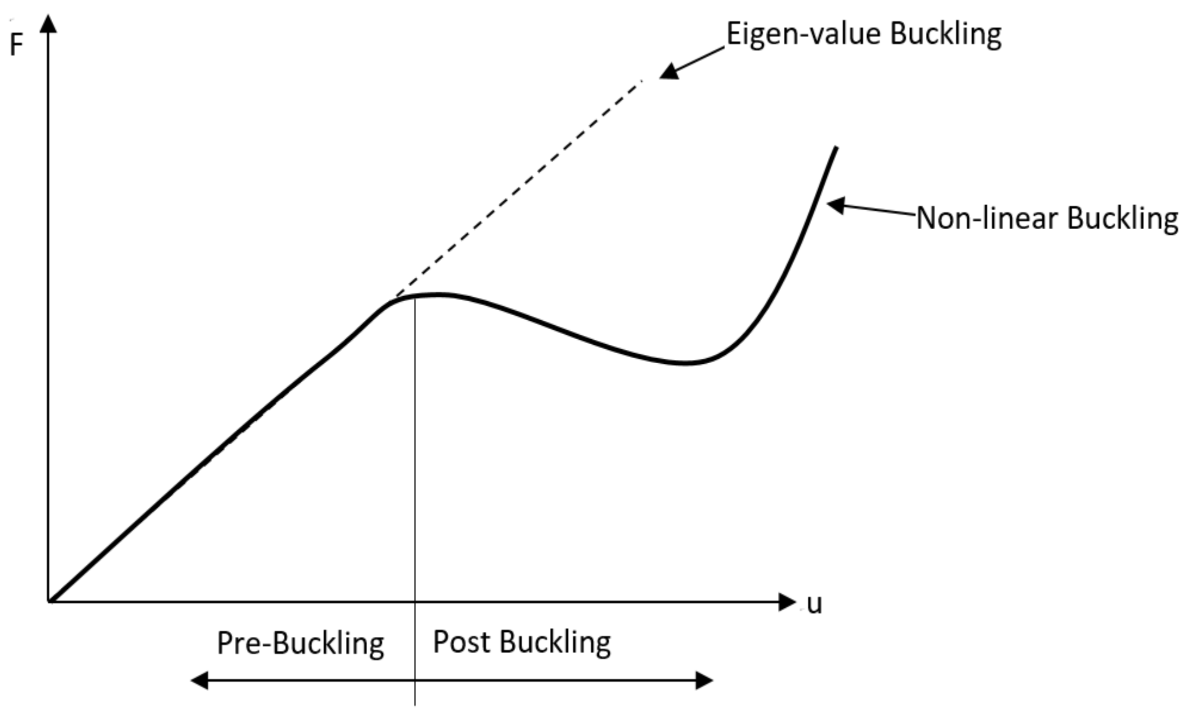
Theoretical buckling strength of an idealized elastic structure is predicted through eigenvalue analysis. Structural eigenvalues are calculated from restrictions and loading conditions for a simple structural structure. Then, buckling loads are generated, each of which is connected to a buckled mode shape that symbolizes the shape that a structure takes when it buckles. The system in a practical structure cannot reach this theoretical buckling strength due to flaws and nonlinear behavior, which causes Eigenvalue analysis to overestimate the buckling load. We recommend nonlinear buckling analysis as a result.

2.3 Nonlinear buckling analysis

Greater accuracy is offered by nonlinear buckling analysis than by elastic formulation. The amount of applied force is gradually increased until a little change in the load level results in a considerable change in displacement. A structure has become unstable if it is in this state. Nonlinear buckling analysis is a static approach that takes into consideration gaps, load disturbances, geometrical flaws, and material and geometric non-linearities. The initiation of a desirable buckling mode requires either a minor destabilizing force or an initial defect.

Comparision

Results from Nonlinear-Static Buckling Analysis are frequently more accurate than those from Linear Buckling Analysis because it takes material nonlinearity into account while generating buckling response. Results of the nonlinear-static analysis are displayed when a graph of the deformed configuration vs load applied. The plot shows the indication of buckling, which is softening.



3 Analysis

3.1 Calculating the Load

Buckling Analysis on column

$$S = \frac{L_e}{k_{min}} = \frac{effective\ length}{minimum\ radius\ of\ gyration}$$

Given data

$$a = 0.1\ m = 100\ mm;$$

$$b = 0.12\ m = 120\ mm;$$

$$l = 1.2\ m = 1200\ mm;$$

$$E = 2.1 \times 10^5\ MPa;$$

$$v = 0.3;$$

Minimum Moment of inertia

$$I_{min} = \frac{b.d^3}{12} = \frac{120 \times 100^3}{12} = 10 \times 10^6\, mm^4$$

Maximum Moment of inertia

$$I_{max} = \frac{d.b^3}{12} = \frac{100 \times 120^3}{12} = 14.4 \times 10^6\, mm^4$$

For column with one end fixed and other end free,

Effective length, $L_e = 2L = 2 \times 1200 = 2400\, mm$

Slenderness Ratio,

$$S = \frac{L_e}{k_{min}} = \frac{L_e}{\sqrt{\frac{I_{min}}{A}}} = \frac{2400}{\sqrt{\frac{10 \times 10^6}{120 \times 100}}} = 83.138$$

Critical load

$$P_{critical} = \frac{\pi^2 E\, l}{L_e^2}$$

Buckling above the axis of minimum moment of inertia.(Euler's Equation)

$$F = P_{critical} = \frac{\pi^2 E\, l_{min}}{L_e^2} = \frac{\pi^2 \times 2.1 \times 10^5 \times 10 \times 10^6}{2400^2} = 3598.3\, KN$$

3.2 Flow Chart

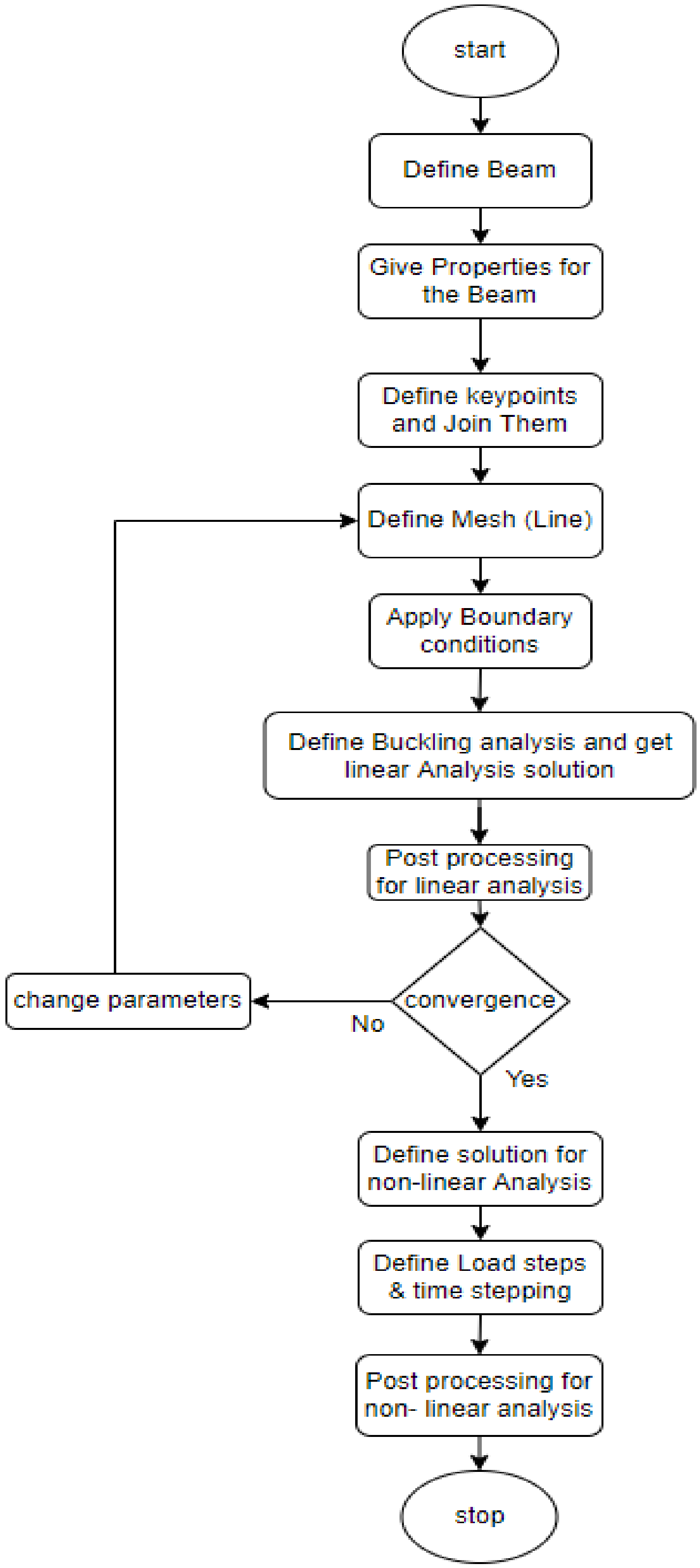


Figure 2: Flow Chart

3.2.1 Define Beam

we define beam using ET. Because beam188 is better suited for assessing thin to moderately thick beam structures, we use it for this case. In a study with significant deflection, the element by default contains terms for stress stiffness. The elements are able to examine flexural, lateral, and torsional stability problems thanks to the stress-stiffness terms that are included (using nonlinear stabilization, collapse analyses with arc length techniques, or eigenvalue buckling).

3.2.2 Give Properties for the Beam

We give the defined beam its young's modulus, poisson's ratio, and density.

3.2.3 Define Mesh (Line)

We use Line Mesh to generate Mesh since the beam is built using a line element. Convergence analysis is used. We obtain the optimal amount of elements for use in simulating our analysis.

3.2.4 Apply Boundary conditions

We must make some assumptions and provide some boundary conditions before we can perform any type of analysis. In our instance, the beam's bottom portion is fixed because it is positioned on a flat surface. Additionally, we apply load to the top of the beam for buckling analysis. We provide a tiny amount of horizontal displacement to start buckling in nonlinear analysis.

3.2.5 Define Buckling analysis and get linear analysis solution

We use ANTYPE to define the Buckling Analysis Solution Mode and switch the Buckling Options to a single mode. Get the eigenvalue solution data for the force needed for buckling and specify the expansion pass for the study.

3.2.6 Post processing for linear analysis

Read the data obtained and plot the deflected beam. so we can the analyze ow load is affecting te beam.

3.2.7 convergence

To select the appropriate mesh size for the Analysis, convergence is used. To get an accurate simulation, we must analyze the solution using elements of various sizes.

3.2.8 Define solution for non-linear Analysis

In this case, eigen value analysis will be used to find the solution. use nonlinear geometry solutions as well. The output is kept in OUTRES.

3.2.9 Define Load steps & time stepping

To apply load progressively, use load step. The load step is first divided into smaller load segments, and the number of analysis iterations is determined. activate automatic time stepping.

3.2.10 Post processing

we analyze the data obtained in the solution and plot graph for Force and deflection.

3.3 Boundary Value Problem

- Bottom of the beam is fixed.
- Load is applied on the Top.
- Gravity is Neglected.
- The beam is assumed to be perfectly straight.
- Cross-section of the Beam is same throughout its length.
- the load is axial and applied along centerline
- Beam weight is ignored

3.4 Buckling Analysis in APDL

The Buckling Analysis for the given project is done using APDL as follows:

3.4.1 Eigen Value(Linear) Buckling Analysis

```
FINISH                          ! Clearing the current data
/CLEAR
/TITLE,Eigenvalue Buckling Analysis
/PREP7
ET,1,beam188,1,,2               ! Beam188 as element type for the quadrilateral
SECTYPE,1,BEAM,RECT             ! Rectangular cross section
SECDATA,100,120                 ! Breadth and width of the cross section
MP,EX,1,2.1e5                   ! Young's modulus
MP,PRXY,1,0.3                   ! Poisson's ratio
MP,DENS,1,7.9e-9                ! Density
K,1,0,0                         ! Coordinate for bottom of the beam
K,2,0,1200                      ! Coordinate for top of the beam
L,1,2                           ! Create a line between keypoint 1 and 2
LESIZE,ALL,,,15                 ! Line is divided into 15 elements
LMESH,all                       ! Mesh Line
FINISH
/SOLU                           ! Enter solution mode
ANTYPE,STATIC                   ! Before you can do a buckling analysis, ANSYS needs
                                ! the data from Static Analysis
PSTRES,ON                       ! Pre-stress can be accounted for - required during
                                ! buckling analysis
DK,1,all                        ! Fix the bottom of the 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,SUBSP,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,all                      ! 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
PLNSOL,U,Y,2
```

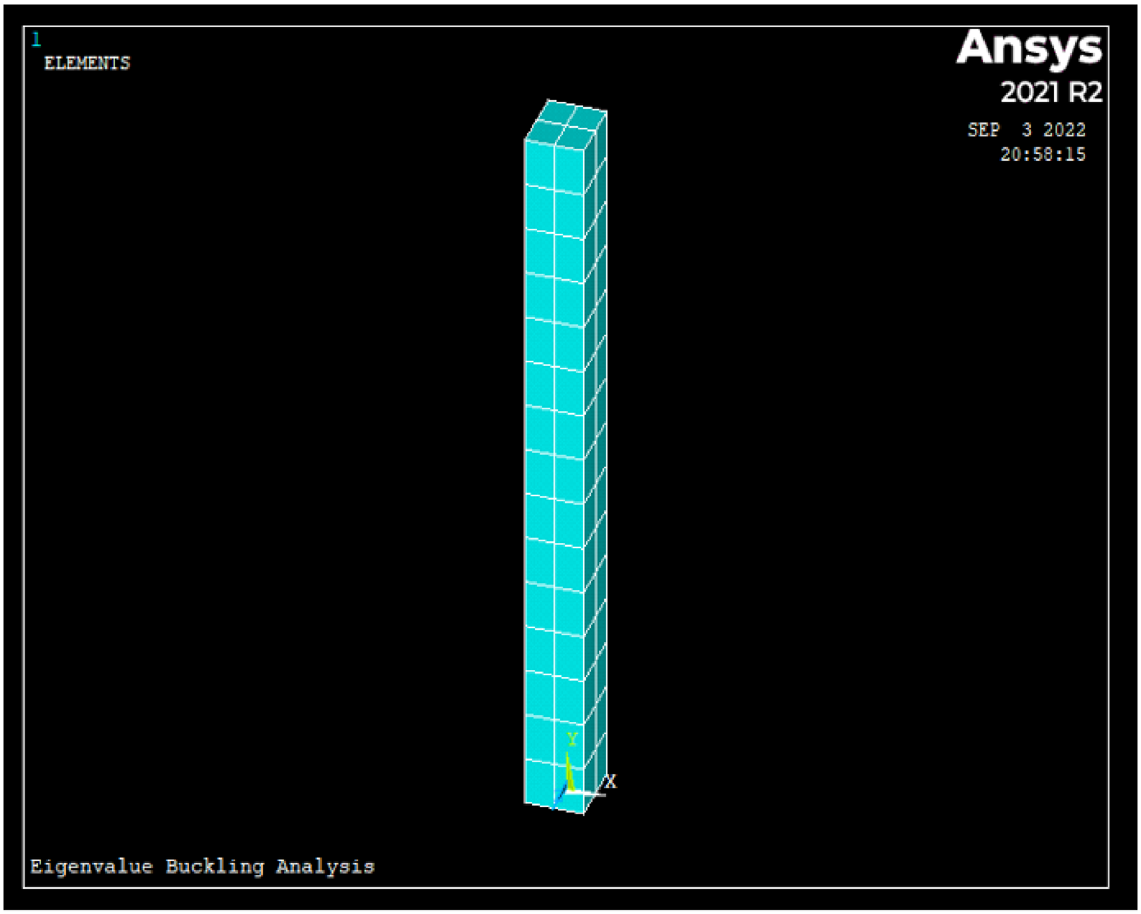


Figure 3: Mesh

3.4.2 Non-Linear Buckling Analysis

```
FINISH                          ! These two commands clear current data
/CLEAR
/TITLE,Non-Linear Buckling Analysis
/PREP7
ET,1,beam188,1,,2               ! Beam188 as element type for the quadrilateral
SECTYPE,1,BEAM,RECT             ! Rectangular cross section
SECDATA,100,120                 ! Breadth and width of the cross section
MP,EX,1,2.1e5                   ! Young's modulus
MP,PRXY,1,0.3                   ! Poisson's ratio
MP,DENS,1,7.9e-9                ! Density
K,1,0,0                         ! Coordinate for bottom of the beam
K,2,0,1200                      ! Coordinate for top of the beam
L,1,2                           ! Create a line between keypoint 1 and 2
LESIZE,ALL,,,15                 ! Line is divided into 15 elements
LMESH,all                       ! Mesh Line
FINISH

/PREP7
UPGEOM,1,1,1,'Z:&Ansys&Trial&file',rst    ! Using the Solution of previous Eigen Value
                                Buckling Analysis
/SOLU
ANTYPE,STATIC                   ! Static analysis
NLGEOM,ON                       ! Non-linear geometry solution supported
OUTRES,ALL,ALL                  ! Stores bunches of output
NSUBST,100,200,80               ! Load broken into small loadsteps
NEQIT,1000                      ! Define the maximum iterations to find solution
AUTOTS,ON                       ! Auto time stepping
LNSRCH,ON
/ESHAPE,1                       ! Plots the beam as a volume rather than line
Dk,1,ALL,0                      ! Constrain bottom
Fk,2,FY,-3600000                ! Apply load slightly greater than predicted
                                ! Required buckling load to upper node
Fk,2,FX, 4000                   ! Add a horizontal load (0.5 % of 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,X                    ! Reads y-deflection data into var 3
XVAR,3                          ! Make variable 2 the x-axis
PLVAR,2                         ! Plots variable 3 on y-axis
/AXLAB,Y, Load (N)              ! Changes Y label
```

```
/AXLAB,X, UX (mm)               ! Changes X label

/POST1                          ! Select the post processor
SET,,,,,1                       ! Define the dataset to be read
NSEL,s,node,,2                  ! Select the subset of node for the dataset
PRNSOL,u,x                      ! Print the nodal solution result

PLDISP,1                        ! Shows the deformed geometry
ANMODE,10,0.1,,0                ! Produce the Animated sequence
*use,modeshap.mac,1,2,3,4,1
SET,FIRST                       ! Read the first dataset
SET,NEXT                        ! Read the last dataset
```

4 Convergence Study

Convergence testing is a way of varying certain setup parameters to ensure accurate simulation results. here we are increasing the number of elements until we get the buckling load converges to a specific work. In our analysis we are getting the value of 3582.5 kN.

Number of Elements	Buckling Load
1	3636.2
2	3586.1
3	3583.2
4	3582.7
5	3582.6
6	3582.5
7	3582.5
8	3582.5

Table 1: Convergence Study

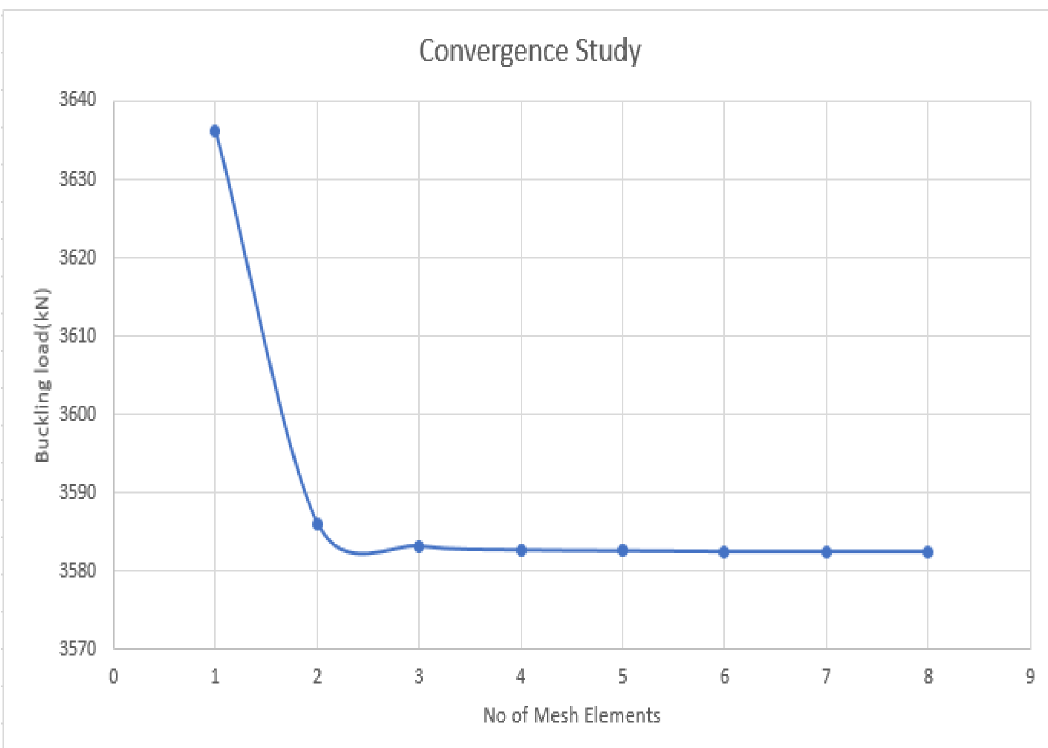


Figure 4: Convergence Study

5 Results

By calculation we got the force that is required to initiate the buckling is 3598.3 kN and by the analysing the problem in ANSYS we are getting the load of 3582.5 kN.

It can also be seen in the convergence study that the convergence occurs at 6 mesh elements. After that critical buckling load doesn't differ.

The Load-Deflection plot displays the softening behavior which indicates the onset of buckling. It can be seen from the graph that the column starts to buckle around 3000 kN of load and then it carries the load for a large amount of deflection. The graph also tells us about how much the beam deflects with the progression of load.Applying a load of 3600 kN results in a deflection of approximately 184.488 mm.

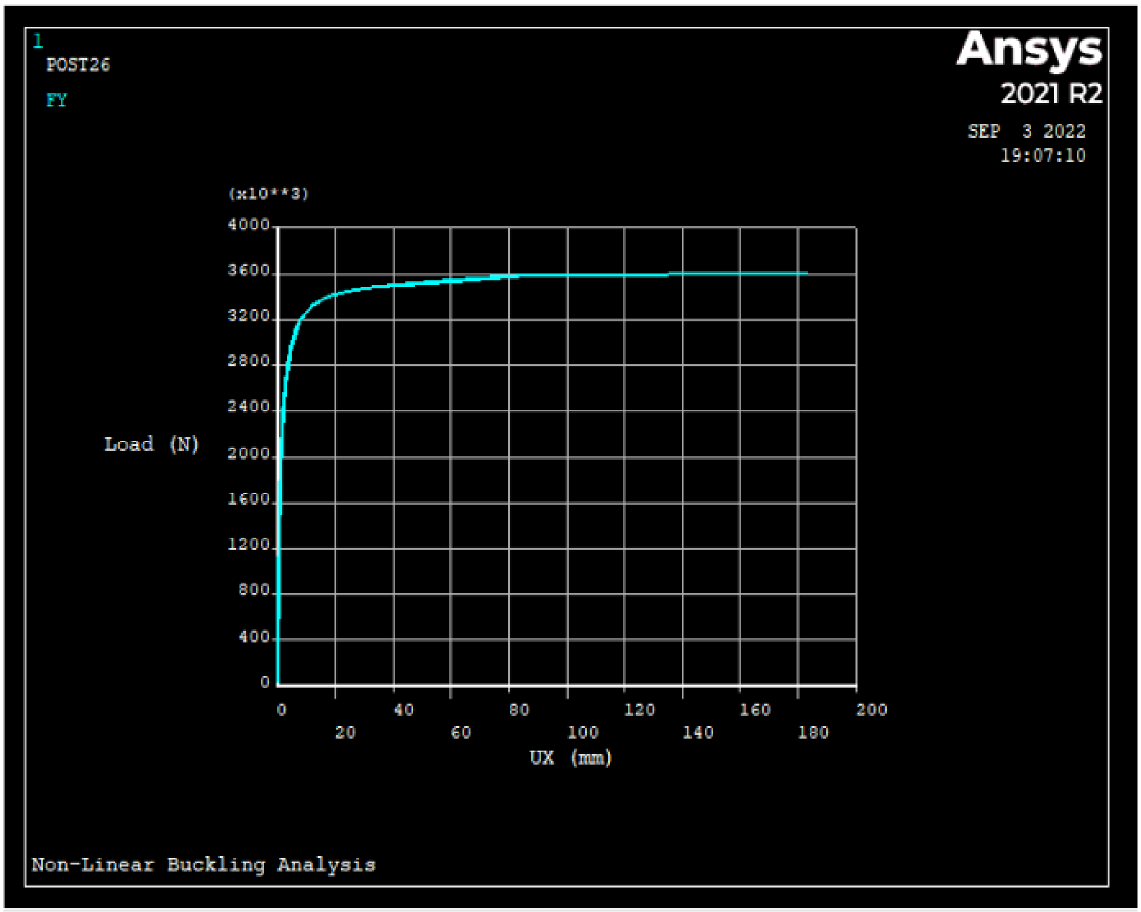


Figure 5: Force - Deflection Graph

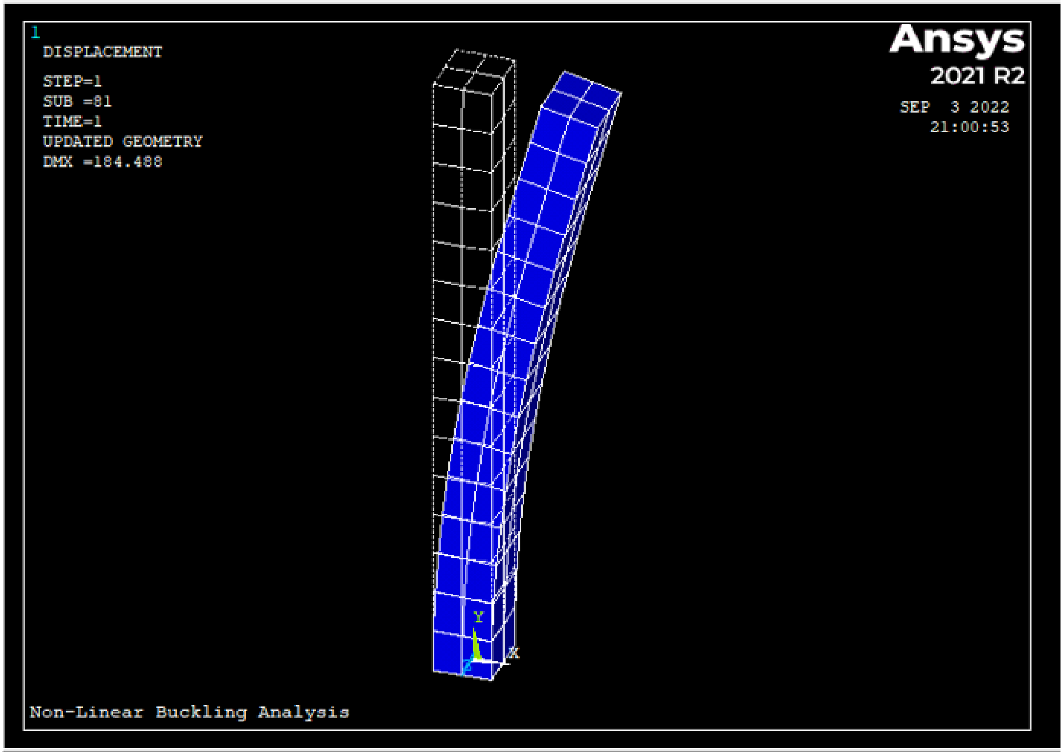


Figure 6: Displacment

6 Conclusion

- If loads and boundary conditions are carefully considered, linear solutions may be suitable for simple structures. However, a complete nonlinear analysis is necessary for the majority of unstable structures.
- Because linear buckling overestimates the model's capacity, the results of eigen value buckling and non-linear buckling may differ.
- A linear buckling analysis might be suitable in some circumstances to fulfill checks against buckling, but in others, it might only serve as a decent beginning point for a comprehensive nonlinear buckling analysis.
- An in-depth evaluation of buckling is provided by nonlinear analysis, which can take boundary, material, and geometric influences into account.

List of Figures

7 References

1. ANSYS Mechanical APDL Technology Demonstration Guide, ANSYS Mechanical APDL 2 - ANSYS, Inc; Release 14.0, November 2011