

Research Article

Finite Element Analysis of Different Types of Column for Different Load Position

Shwetha. S. M^a and Hemant. L. Sonawadekar^a^aDepartment of Civil Engineering, KLESCET, Belgaum

Abstract

Ansys is finite element software. In the present work finite element analysis of different types of column for different gravity load position is being done. The study is based on the fact that the stress and strain variation depends primarily on load position on column cross section. The present study is aimed to know the variation of node displacement, principal stress and principal strain i.e. the stress and strain at centroid of elements of column model. The objective of this work is to study the variation of principal stress, principal strain and node displacement for different load position.

Keywords: Concrete column model, the element (top corner element), Reinforced Concrete, stress analysis, ANSYS.

1. Introduction

The columns in the structure are mainly subjected to axial compression. The columns carrying axial compression only are the internal columns with beams in all four directions or beams in one plane having same spans and same loading.

Columns along the sides of a building, which carry beams either in three orthogonal directions or a single beam in one direction are subjected predominantly to axial load and uniaxial bending due to unbalanced moment transferred from a single beam on one side, while the moments from the other two beams in opposite directions balance each other provided their spans and loads on them are approximately equal.

The load carrying capacity of the column also depends on the slenderness ratio (L_{eff}/h) which is the ratio of effective length (L_{eff}) to the corresponding lateral dimension (h). If the slenderness ratio is greater than or equal to 12 is called long or slender column. If the slenderness ratio is less than 12 than it is short column.

The design of column necessitates determination of loads transferred from beam at different floor levels. Loads transferred from slab to beams and then to columns. Hence, slabs and beams are normally designed prior to the design of columns. This method is called as Exact method which enables one to assess the loads on columns more accurately and thereby the design of column becomes realistic and economical. The following types of columns are considered during analysis:

1. Rectangular column
2. Square column

3. Circular column

In the present work a Reinforced concrete column has been analyzed. The analysis of top corner element of column has been carried out for four cases.

Case I: Constant load is applied at top one corner node.

Case II: Constant load is divided in to two equal parts and applied at top two corner node.

Case III: Constant load is divided in to three equal parts and applied at top three corner node.

Case IV: Constant load is divided in to four equal parts and applied at top four corner node.

2. Modeling

The Ansys is finite element analysis software. The finite element method is a general technique for constructing solutions to boundary value problems. This method involves dividing the domain of interest into a finite number of simple sub domains, the finite elements, and using various concepts to construct an approximation of the solution over the collection of finite elements. In the present work finite element analysis of column for different axial load position has being done. The element considered is Solid45, mapped meshing has been done.

3. Types of Stresses

In the software being used, the element forces/stresses, strains, nodal displacement outputs are available at the following locations:

- Center point of the element.
- All corner nodes of the element.

*Corresponding author: **Shwetha. S. M** is a Masters in Engineering Student and **Hemant. L. Sonawadekar** is working as Asst. Prof

The following items are included in the analysis of element;

1. Principal stress at center point of element.
2. Principal strain at center point of element.
3. Nodal displacement of top corner node.

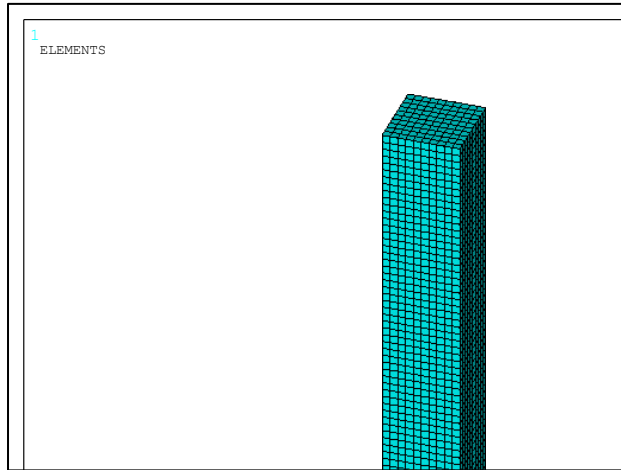


Figure1. Elements of Column Model

3.1 Generation of Data

The value of parameters depends upon load position. The parameters which are studied are nodal displacement, principal stress and principal strain. The model dimension for rectangular column is 300X750 mm, for square column 500X500 mm and for circular column 600 mm diameter is being taken. The value of Young’s modulus, Density and Poisson’s ratio are 22360 N/sqmm, 25kN/m³ and 0.17 respectively. The analysis is carried out for loads of 1700 kN, 1900 kN, 2100 kN, 2300 kN, 2500 kN

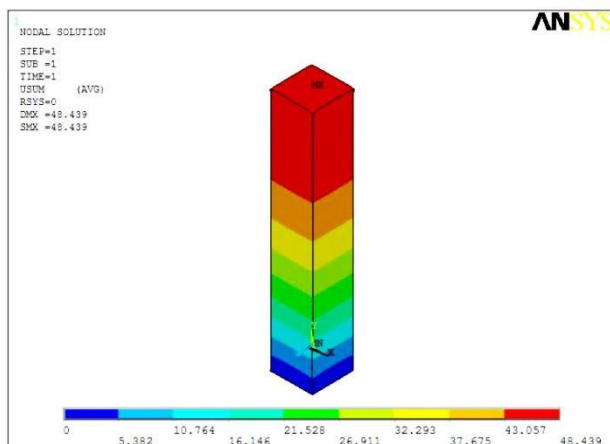


Figure2. Deformation of Column for Load Case IV

4. Results and Discussion

In the analysis variation of principal stress, the element has been studied. Similarly, variation of principal strain, has been studied and the top corner node displacement is

also analyzed. For Rectangular column the load case IV the value of principal stress, principal strain and node displacement is minimum compared to the remaining other three cases. For case I the value of principal stress, principal strain and node displacement is maximum and for case II and case III the variation of parameters are nearly same. The same trend follows for Square and Circular Column. The variation of principle stress, strain and nodal displacement are shown in the following figures.

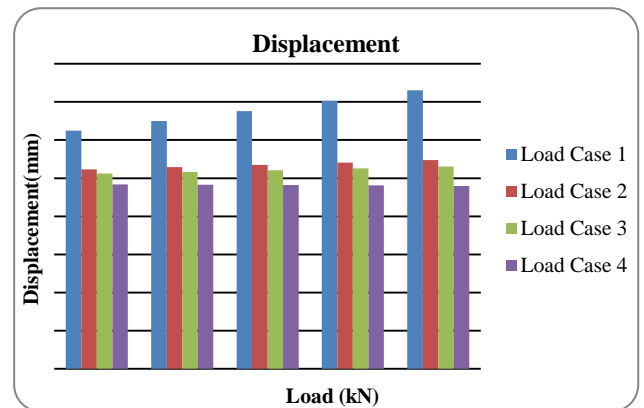


Figure 4.1 Maximum displacement in Rectangular column for different Load Cases

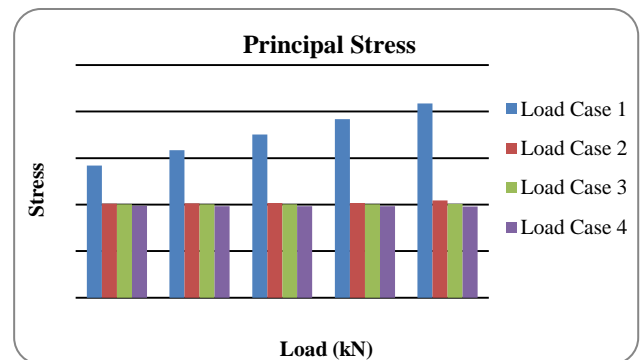


Figure 4.2 Principal Stress in Rectangular column for different Load Cases

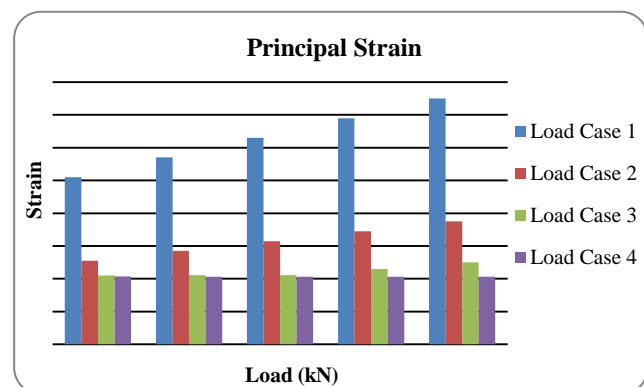


Figure 4.3 Principal Strain in Rectangular column for different Load Cases

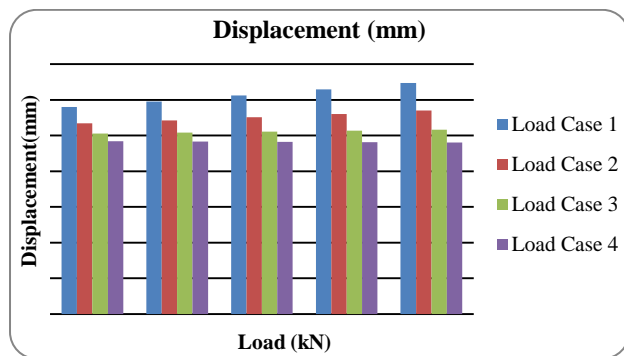


Figure 4.4 Maximum displacement in Square column for different Load Cases

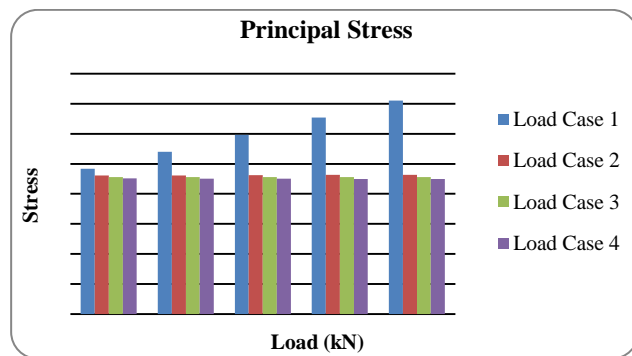


Figure 4.8 Principal Stress in Circular column for different Load Cases

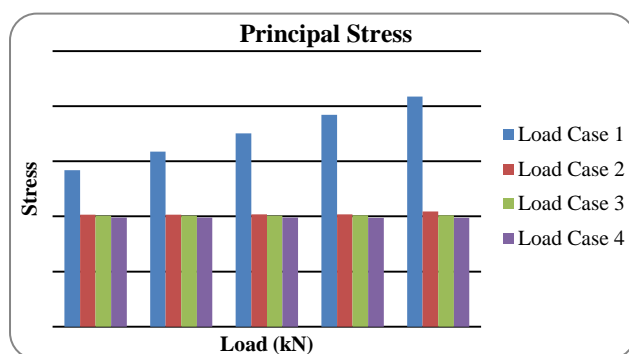


Figure 4.5 Principal Stress in Square column for different Load Cases

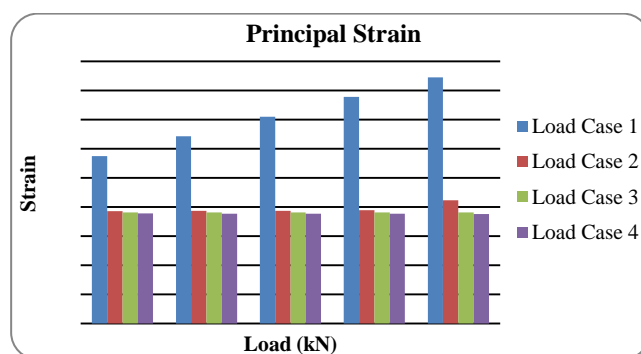


Figure 4.9 Principal Strain in Circular column for different Load Cases

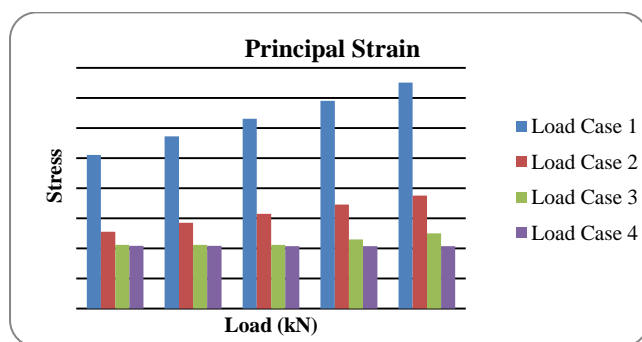


Figure 4.6 Principal Strain in Square column for different Load Cases

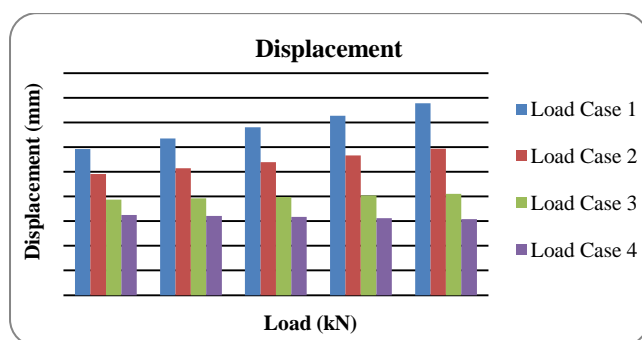


Figure 4.7 Maximum displacement in Circular column for different Load Cases

Conclusion

On the basis of study carried out following conclusions can be drawn.

1. *Rectangular Column*
 - a. The value of node displacement is found to be lowest in the case IV of all the cases considered. The value of nodal displacement increases in case III, II and I by 9.51%, 14.06% and 52.26% respectively.
 - b. The value of principal stress is found to be lowest in the case IV of all the cases considered. The value of principal stress increases in case III, II and I by 2.62%, 6.49% and 112.89% respectively.
 - c. The value of principal strain is found to be lowest in the case IV of all the cases considered. The value of principal strain increases in case III, II and I by 21.47%, 82.16% and 264.39% respectively.
2. *Square Column*
 - a. The value of node displacement is found to be lowest in the case IV of all the cases considered. The value of nodal displacement increases in case III, II and I by 7.53%, 18.57% and 34.70% respectively.
 - b. The value of principal stress is found to be lowest in the case IV of all the cases considered. The value of principal stress increases in case III, II and I by 2.41%, 5.85% and 111.71% respectively.
 - c. The value of principal strain is found to be lowest in the case IV of all the cases considered. The value of

principal strain increases in case III, II and I by 20.73%, 81.07% and 262.15% respectively.

3. Circular Column

- a. The value of node displacement is found to be lowest in the case IV of all the cases considered. The value of nodal displacement increases in case III, II and I by 4.25%, 11.86% and 19.48% respectively.
- b. The value of principal stress is found to be lowest in the case IV of all the cases considered. The value of principal stress increases in case III, II and I by 1.38%, 3.15% and 57.94 % respectively.
- c. The value of principal strain is found to be lowest in the case IV of all the cases considered. The value of principal strain increases in case III, II and I by 1.38%, 12.28% and 124.70% respectively.

References

Ying Wang, Yuan Xuan, (2010), Nonlinear Finite Element

- Analysis of the Steelconcrete Composite Beam to Concretefilled Steel Tubular Column Joints *International Journal of Nonlinear Science* ,Vol.9 (2010) No.3, pp341348.
- P. Fe Athanasios I. Karabinis,(Sept/Oct2008) 3D Finite Element Analysis of Substandard RC Columns Strengthened by FiberReinforced Polymer Sheets *Journal of Composites for Construction*, Vol. 12, No. 5, pp 531540
- M.E.Final year student (July 2010), Finite Element Analysis of Column for Different Gravity Load Position, *Department of Civil Engineering, UEC, Ujjain (M.P.)*
- Kishen, Chandra JM and Kumar, Avinash (2004),Finite element analysis for fracture behavior of cracked beamcolumns, *Finite Elements in Analysis and Design*, 4013pp 1773178
- Anupam Chakrabarti and Amit Chandra (Aug 2008) Finite Element Analysis of Concrete Columns Confined with FRP Sheets *Journal of Reinforced Plastics and Composites*, vol. 27 no. 12 ,pp13491373.
- (March 2012), Finite Element Modeling of Reinforced Concrete Column under Monotonic Lateral Loads. *International Journal of Computer Applications* (0975 – 8887) Volume 42– No.3,