https://doi.org/10.48047/AFJBS.6.15.2024.4909-4917

AFJBS

African Journal of Biological Sciences

Journal homepage: http://www.afjbs.com

Research Paper Open Access

NONLINEAR ANALYSIS OF REINFORCED CONCRETE COLUMNS USING ANSYS

Wilfred Rohit Peters[1], Dr.J. Rex[2], Dr. C. Venkata Siva Rama Prasad[3]

[1]MTech Student, Department of Civil Engineering, Malla Reddy Engineering College, Secunderabad -500100, Telangana, India.
[2]Associate Professor, Department of Civil Engineering, Malla Reddy Engineering College, Secunderabad -500100, Telangana, India.
[3]Associate Professor, Department of Civil Engineering, Malla Reddy Engineering College, Secunderabad -500100, Telangana, India.

Volume 6, Issue 15, Sep 2024

Received: 15 July 2024

Accepted: 25 Aug 2024

Published: 05 Sep 2024

doi: 10.48047/AFJBS.6.15.2024.4909-4917

Abstract: ANSYS's nonlinear analysis is crucial for assessing concrete structures under load, capturing both material and geometric nonlinearities. It predicts structural issues, informs reinforcement, optimizes designs, and enhances seismic performance. Combined with experimental tests, ANSYS accurately models reinforced concrete behavior, ensuring safe and efficient designs.

Keywords: Nonlinear analysis, Reinforced concrete columns, ANSYS, Finite element analysis, Structural integrity.

1. Introduction

Experimental analysis of concrete strength provides detailed insights but is costly and time-consuming. Finite Element Analysis (FEA) offers a faster, cost-effective alternative for evaluating structural behavior. FEA, using tools like ANSYS v15, accurately predicts responses such as load, displacement, and crack development, and is useful for complex structures and materials. FEA has evolved into a key tool in computer-aided engineering, solving both simple and complex problems across various fields. It involves modeling geometry, material properties, and loading conditions to analyze structural responses. Columns, vital for load-bearing and stiffness, are studied under lateral loads to understand deflection and horizontal stiffness.

The finite element method (FEM) divides structures into finite elements to approximate solutions, accommodating nonlinear responses like cracking and crushing. This method is also applied to analyze tapered structures and reinforced concrete columns under torsion, where traditional methods fall short. Recent advancements include using carbon fiber reinforced plastics (CFRPs) for rehabilitating deteriorated concrete structures due to their strength, durability, and cost-effectiveness. While CFRP applications have been well-studied for circular columns, research on rectangular and large-scale columns is still developing, highlighting the need for further investigation into their effectiveness and application.

1.1 Objectives

The project has three primary objectives. First, it aims to develop a reliable ANSYS tool capable of analyzing large arrangements using advanced techniques. Second, it focuses on examining reinforced concrete structures while considering the effects of construction joints and cyclic loading. Third, the project seeks to validate the theoretical model by comparing results from the lumped dissipation model with experimental data from tests on reinforced concrete structures. This involves comparing numerical results from ANSYS with experimental outcomes, particularly under cyclic loading conditions. The goal is to refine the lumped dissipation models to achieve more accurate responses in ANSYS. Additionally, the study explores the relationship between the amount of tensile longitudinal reinforcement in columns and the non-dimensional γ coefficient, which indicates damage levels.

The need for concrete structure strengthening is significant due to factors such as increased loads, design and construction faults, and changes in structural systems. Strengthening techniques, like epoxy plate bonding with carbon fiber reinforced polymers (CFRPs), are competitive for enhancing the load-bearing capacity of existing structures. Addressing deterioration and damage is crucial for maintaining modern structures, including skyscrapers and bridges, which are costly and disrupt society during construction. Effective maintenance ensures long-term durability, load-bearing capacity, and aesthetic appeal. Strengthening existing structures is often more complex than building new ones, requiring careful consideration of existing materials, loads during reinforcement, and structural geometry. The process must evaluate all potential failure modes to avoid unintended issues such as shear failure. Moreover, the design must meet current code requirements, taking into account environmental and aesthetic factors. The research employs nonlinear finite element analysis to predict beam-column joint responses across various load ranges, design innovative retrofitting techniques, and understand the behavior of retrofitted joints using CFRP. The study aims to assess different CFRP wrapping techniques and provide practical recommendations for engineers.

2. Literature Review

Barbosa et al. [1] thought about the viable use of nonlinear models in the examination of built up substantial designs and the outcomes of little changes in demonstrating. The work-hardening, elastoplastic, perfectly plastic models that achieved ultimate loads that were very close to the predicted values produced the best results.

Dahmani et al. [2] investigated the advantages of numerical simulation over experimental tests and the applicability of ANSYS software for crack pattern analysis and prediction in RC beams. From initial cracking to beam failure, various phases of the FE model's behavior of an RC beam were studied for this purpose.

In his thesis, Wolanski [3] used Finite Element Analysis (FEA) to investigate the load-deformation response of reinforced and pre-stressed concrete beams. The experimental data were compared to the results. The theoretical results and the characteristic points on the load-deformation curve that were predicted by FEA were then compared. The model's nonlinear analysis produced results that were comparable to the calculated values.

Using the ANSYS program (ANSYS 1998), Kachlakev et al. [4] developed finite element method (FEM) models to simulate the behavior of four full-size beams from linear to nonlinear response and failure. Plots of load-strain at specific points on the beams, load-deflection at mid-span, first cracking loads, loads at failure, and crack patterns at failure are all compared.

A unified approach to numerical modeling of concrete structures is presented in Bangash [5] (1989). It spans hypothesis and practice by connecting numerical models with limited component examination (FEA), taking into account cement's different properties like drag and breaking. Although it is valuable, it may not reflect the most recent software and material advancements.

J.G. MacGregor's [6] (1992) "Built up Substantial Mechanics and Configuration" gives a foundation to understanding the hypothesis and plan of supported substantial designs, it fundamentally centers around customary scientific strategies. New perspectives can be gained by combining these fundamentals with cutting-edge computational tools like Ansys. Ansys, a powerful software for finite element analysis (FEA), lets you model intricate structures and look at reinforced concrete under different loads. Engineers can use this synergy to learn more about structural performance, improve designs, and look into novel solutions. Consolidating MacGregor's central information with Ansys' capacities prompts a more thorough grasping, productive plan, and investigation of novel methodologies in the domain of supported substantial designs.

William and Warnke's [7] (1975) constitutive model, carried out in Ansys, offers a complex way to deal with reenacting the triaxial conduct (conduct under three-layered pressure) of typical cement inside the ordinary security system. Concrete yield and plastic flow can be predicted using this 5-parameter model, which takes hydrostatic pressure and confining effects into account. Even though it works, it may not be able to capture the complexity of damaged or high-strength concrete, and it needs to be calibrated carefully for specific applications.

Tavarez [8] (2001) uses express limited component techniques in LS-DYNA to reenact the way of behaving of composite matrix supported cement footers under four-point bowing. The review centers around foreseeing load-redirection qualities and investigates the pressure condition of the support. Outstandingly, it examines the achievability of involving this methodology for moderate plan expectations of the heap conveying limit.

As of late, Hao-Jan et al. [9] (2007) tentatively examined the impacts of angle proportion of the cross segment and the variety of volume proportion of cross over to longitudinal support on the breaking and extreme strength of built up rectangular substantial Sections under unadulterated twist. The impact of the range on torsional conduct of supported concrete. Column has not yet been examined. In this way, the current work is an endeavor to foresee the nonlinear reaction of cantilever supported concrete rectangular shaft under unadulterated twist utilizing the limited component. bundle ANSYS-VI 0 (2005) meaning to foresee the impact of the range of cantilever Segment on its torsional reaction and the wellbeing edge in the (ACI 318-05, 2005) code arrangements, feature the adequacy of the torsional support in the pre and post breaking phases of rectangular built up cement footers under unadulterated twist, anticipate variety of the burdens in the cross over and longitudinal torsional fortifications at various phases of stacking. The cross segment, longitudinal and cross over support are saved constants for all the Section, and this is the fundamental constraints of the current review.

Ansys is used to conduct a non-linear finite element (FE) analysis of reinforced concrete (RC) columns with openings that are subjected to combined axial and flexural loading, as demonstrated by Rezazadeh et al. [10] (2016). The effect of opening size and location on the columns' load-carrying capacity and deformation behavior is the subject of their investigation.

Ansys is a powerful tool for engineers and researchers because it can model complex material behavior and capture nonlinear responses. Concrete structures that are both safer and more dependable can be designed with greater precision and efficiency thanks to this strategy. In any case, it is critical to recognize constraints like possible errors among models and true way of behaving, and the requirement for cautious alignment and approval of models with trial information.

3. Methodology

The text describes the finite element modeling (FEM) of concrete and steel reinforcement using ANSYS. Concrete is modeled using the Solid65 element, capable of plastic deformation, cracking, and crushing, while steel reinforcement is modeled with the Link180 element, which handles plastic deformation in 3D. Steel plates and supports use the Solid185 element, allowing for complex behaviors like plasticity and large strain. The reinforcement is modeled using a discrete approach, where bar elements share nodes with the concrete mesh.

Concrete's nonlinear behavior is captured using the modified Hognestad model, with stress-strain curves illustrating elastic and post-peak softening regions for both compression and tension. Cracking and crushing failures are considered, though crushing is disabled to prevent premature failure in the model. The Willam and Warnke material model is implemented with nine constants defining shear transfer, cracking stress, and crushing stress. Steel reinforcement follows an elastic-bilinear behavior with work hardening to stabilize the FEM.

4. Materials & Methods

4.1 Methodology of the Study

The review examines the primary way of behaving of supported substantial sections fortified with carbon fiber built up polymers (CFRP). 37 sections were investigated utilizing ANSYS, partitioned into four series. In order to contrast theoretical results with experimental tests, verification studies were carried out. The influence of key parameters on the behavior of CFRP-strengthened RC columns was then the subject of a parametric study.

4.2 Material Investigation

A reinforcing phase (fibers, sheets, or particles) and a matrix phase (metal, ceramic, or polymer) are the two components that make up composites. When compared to individual components, composites have superior properties like a high strength-to-weight ratio, creep resistance, and toughness. Carbon fibers are utilized extensively due to their low thermal expansion and high modulus of elasticity (200-800 GPa). They are treated with measuring and folios to further develop holding. While carbon fibers are resistant to fatigue, chemicals, and water, coming into direct contact with steel can result in galvanic corrosion.

4.3 Finite Element Analysis

Nonlinear behavior in concrete structures can be studied using finite element analysis (FEA). Concrete has material nonlinearity, and the elastic, inelastic, and ultimate load stages of FEA help analyze structures. SOLID65, PIPE16, and SHELL63 components are utilized in ANSYS for demonstrating concrete, building up bars, and CFRP composites, separately. Optimized structural design is made possible by nonlinear finite element analysis's insights into the distribution of stress and strain.

A. Elements Used for Discretization

- Concrete: SOLID65, capable of modeling cracking and crushing.

- Reinforcing Bars: PIPE16, a uniaxial element for tension, compression, and bending.

- Carbon Fibers: SHELL63, with bending and membrane capabilities.

B. Constitutive Modeling of Concrete

Concrete's non-homogeneity makes its behavior complicated. Concrete's constitutive laws take into account nonlinear properties like plastic deformation and cracking. Stress-strain relationships are included in the models, and factors like tension cracking and stiffening are taken into account.

C. Nonlinear Stress-Strain Relationship of Concrete

Concrete doesn't act impeccably flexibly and shows a mind boggling pressure strain relationship. Methods like tangent and secant moduli are used to determine the modulus of elasticity, which varies with stress levels. Substantial's way of behaving is impacted by its arrangement and the presence of miniature breaks.

D. Stress-Strain Relationship of Steel

Due to its ductility, steel can withstand significant deformation before breaking. Steel's stress-strain curve exhibits linear behavior up until the yield point, at which point it undergoes plastic deformation. Steel's flexibility is urgent for primary applications.

4.4 Mechanism of Concrete Column by Confinement

FRP coats or other restricting gadgets upgrade substantial sections' solidarity and pliability. Constrainment builds the substantial's compressive strength by making a triaxial stress state. Aloof constrainment, created by the substantial's parallel extension, is much of the time more viable than dynamic repression techniques.

4.5 Finite Element of RC Columns

Limited component techniques have progressed the investigation of built up substantial segments. With SOLID65, SOLID185, LINK180, and SHELL41 elements, nonlinear FEA using ANSYS (Version 16.0) models concrete, steel plates, and CFRP composites. This method improves design and performance evaluation by providing a realistic analysis of complex systems.

Nonlinear Solution Procedure:

- Newton-Raphson Strategy: ANSYS involves this technique for tackling nonlinear issues by separating the heap into increases and applying them more than a few burden steps. The harmony condition is refreshed iteratively. On the off chance that assembly isn't accomplished, highlights like line search and programmed load venturing are initiated. The arc-length method is used to avoid divergence if issues persist, particularly in cases of nonlinear buckling.

Material Characteristics:

- CFRP Bound Sections: ANSYS models substantial segments with CFRP sheets by characterizing material properties like Youthful's modulus and Poisson's proportion. Concrete is demonstrated with versatile way of behaving, while CFRP is treated as a straight flexible orthotropic material.

Experimental Procedure:

- Compressive Strength Test: Conducted on 150x150x450 mm columns at 7 and 28 days. Concrete's compressive strength is determined by applying load and recording failure.

- Split Tensile Strength Test: Measures tensile strength using cylindrical specimens, following ASTM C496 or IS 5816. Tests involve applying load until failure, with curing and specimen preparation adhering to specific guidelines.

Analysis of RC Columns Using ANSYS:

- Displaying and Lattice: The section is demonstrated utilizing Solid65 components with rectangular lattice. Limit conditions incorporate pivot and roller supports, and burden is applied across nodal lines.

- Nonlinear Analysis: The stiffness matrix is updated iteratively using the Newton-Raphson method as load is applied in small increments to guarantee convergence.

- Load Venturing and Disappointment: Programmed time venturing predicts load step sizes. Non-convergence in small load increments, which indicates significant deflection, is what defines failure.

Finite Element Analysis of RC Columns:

- Problem Statement: Analyzed reinforced concrete columns (250x250 mm, height 1250 mm) with various slenderness ratios, using experimental data. The study includes columns with FRP wraps and examines behavior under axial and eccentric loads in ANSYS.

5. Results and Discussion

5.1 Design of Column:

FE Models: Columns are examined under 50mm and 100mm axial and eccentric loadings. Upholds are demonstrated as pivoted (for hub stacking) and fixed (for unpredictable stacking). Each load is applied one at a time.

5.2 Results and Discussion:

Load versus Mid-Span Deflection: ANSYS FEA results and experimental data are compared in charts.

Vertical Avoidance: Under pivotal stacking, vertical redirection is more huge at the free end. Expanded erraticism moves and enhances diversion because of bowing impacts.

Horizontal Deflection: Under axial loading, shear failure results in minimal horizontal deflection. Due to slenderness and buckling effects, eccentric loading causes long columns to exhibit greater horizontal deflection than short columns.

Concrete Cracks: Under axial loading, concrete columns exhibit diagonal tensile cracks caused by shear failure. With increased tensile cracks on the opposite side, eccentric loading causes cracks to shift toward the load. Long segments show decreased shear disappointment with additional pliable breaks contrasted with short sections.

6 Results

6.1 Ultimate Load vs. Axial Strain:

Table 1: Experimental vs. Numerical Results:

Column	Ultimate Load [KN]	Axial Strain
C1	0.93	0.97
C2	0.96	0.95
C3	0.92	1.03
C4	0.98	0.98
C5	0.95	0.98
C6	0.97	1.07

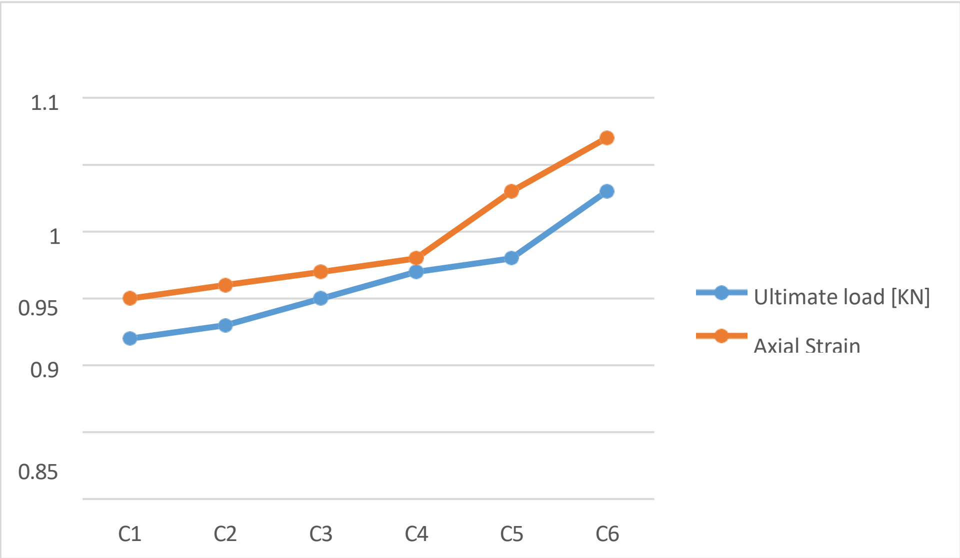

Figure 1: Ultimate Load vs. Axial Strain.

6.2 Ultimate Load vs. Axial Displacement:

Table 2: Experimental vs. Numerical Results:

Column	Ultimate Load [KN]	Axial Displacement
C1	0.92	0.99
C2	0.94	1.01
C3	0.96	1.03
C4	0.98	1.05
C5	1.01	1.06
C6	1.05	1.10

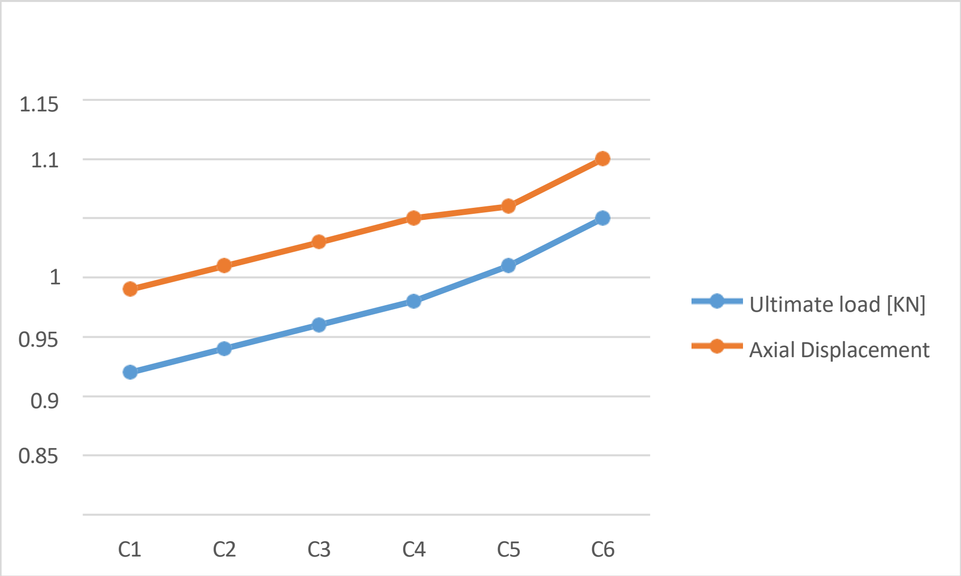

Figure 2: Experimental and Numerical Results of Ultimate Load vs. Axial Displacement.

6.3 Axial Stress vs. Axial Strain Curves:

Table 3: Experimental vs. Numerical Results:

Column	Axial Stress Curve [KN]	Axial Strain Curves
C1	0.89	0.91
C2	0.92	0.94
C3	0.95	0.96
C4	0.98	0.98
C5	1.01	1.06
C6	1.05	1.03

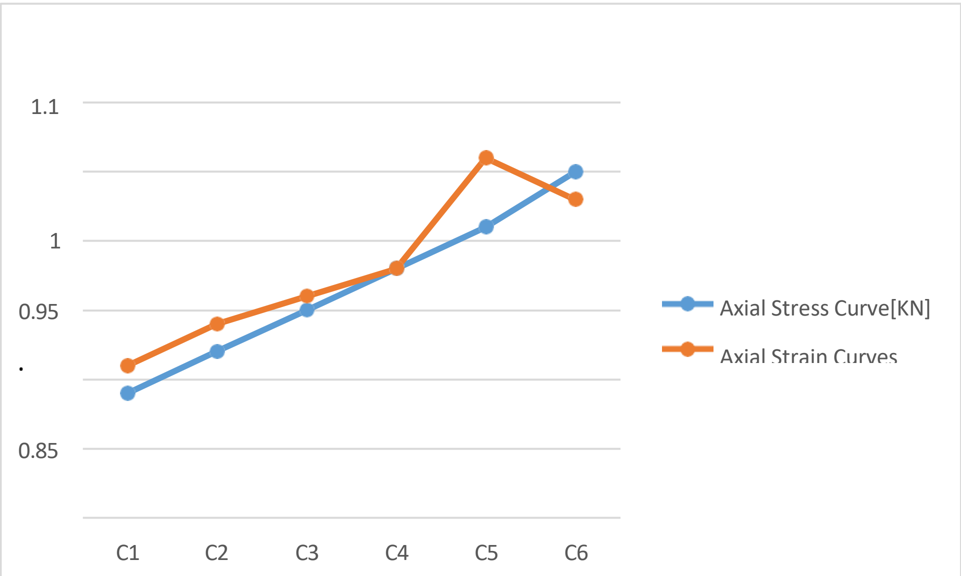

Figure 3: Experimental and Numerical Results of Axial Stress vs. Axial Strain.

7. Conclusion

The study evaluated the use of ANSYS software for nonlinear finite element analysis (FEA) of reinforced concrete columns, comparing it with experimental tests. The analysis showed that ANSYS effectively models failure mechanisms for both axial and eccentric loading conditions, capturing nonlinear responses accurately up to failure.

- ANSYS Accuracy: Finite element models in ANSYS 16, using specialized concrete elements, closely matched experimental results and provided valuable insights into the nonlinear behavior of RC columns.

- Sensitivity Factors: The results were sensitive to factors like mesh size, material properties, and load increments.

- Advantages: ANSYS offers comprehensive analysis, high accuracy, efficiency, design optimization, and dynamic/seismic analysis capabilities.

- Requirements: Effective use of ANSYS demands expertise in structural engineering, software proficiency, and rigorous validation against experimental data.

References

1. Islam, D.S. 2004, Non Linear Finite Element Analysis of Reinforced Concrete Plates in Punching Shear, M. Sc Engineering Thesis, Bangladesh University of Engineering and Technology.
2. Ahsan, M. S. 2004, Non Linear Finite Element Analysis of Reinforced Concrete Beams with Rectangular Opening', M. Sc. Engineering Thesis, Bangladesh University of Engineering and Technology.
3. Ehab M. Lotfy, (2010) "Behavior of reinforced concrete short columns with Fiber Reinforced polymers bars" International Journal of Civil and Structural Engineering Volume 1, No 3, pp 545-557
4. Mukhopadhyay M, "Mechanics of Composite Materials and Structures", University press India Pvt. Ltd.Publications, 2004
5. Murtada, A. I., 2008. Reinforced concrete columns strengthened with carbon fiber reinforced polymer. University of Mustansiriya. M.Sc. Thesis.
6. Pan J. L., T. Xu, and Z. J. Hu., 2007, "Experimental investigation of load carrying capacity of the slender reinforced concrete columns wrapped with FRP." Construction and Building Materials 21.11: 1991-1996.
7. Pessiki, Stephen et.al., ,2001,"Axial behavior of reinforced concrete columns confined with FRP jackets." Journal of Composites for Construction 5.4: 237- 245.
8. D. Kachlakev, T. Miller, S. Yim, "Finite Element Modeling of Rein-forced Concrete Structures Strengthened with FRP Laminates", Final Report SPR 316, Oregon Department of Transportation, 2001.
9. M.Y.H. Bangash, "Concrete and Concrete Structures: Numerical Modeling and Applications", Elsevier Science Publishers Ltd., London, England. 1989.
10. J.G. MacGregor, "Reinforced Concrete Mechanics and Design", Prentice-Hall, Inc., Englewood Cliffs, NJ., 1992.