iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 11 Issue: IV Month of publication: April 2023

DOI: https://doi.org/10.22214/ijraset.2023.50612

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 11 Issue IV Apr 2023- Available at www.ijraset.com

Effect of Lateral Reinforcement on Strength and Ductility of Reinforced Concrete Columns

Praful Lahase[1], Mahroof Ahmed[2]
[1]M.Tech Scholar, Department of Civil Engineering, Sushila Devi Bansal College of Engineering, Indore
[2]Assistant Professor, Department of Civil Engineering, Sushila Devi Bansal College of Engineering, Indore

Abstract: Concrete confined by stirrups has a greater ductility than unconfined concrete which is widely used in reinforced concrete (RC) structures and its behavior is a classic topic of discussion. Non-linear behavior of reinforced concrete is complex due to involvement of various heterogenic material properties and cracking behavior of concrete. It is very important to understand the response and failures of columns subjected to lateral loads in order to save lives and reduce the damage to the building components. ANSYS is widely used finite element analysis (FEA) software which has the tools to analyze structural elements and their non-linear behavior based on the finite element procedure The failure prediction of reinforced concrete elements is usually carried out using experimental testing, and the observations are recorded only at critical locations due to restriction at the cost of testing equipment and accessories. To avoid the destructive testing, cost-cutting of materials and labor, the behavior prediction of reinforced concrete elements is typically carried out using numerical methods like Finite Element Method which responds well to non-linear analysis as each component possesses different stress-strain behavior. In this study, reinforced concrete columns were modeled having different tranverse reinforcements and subjected to axial loads along with monotonic lateral loading, their behavior is studied in terms of deformation, stress, strain, crushing/cracking and force-displacement curves.

Keywords: Reinforced concrete columns, size effect, lateral loading, seismic analysis,ANSYS, finite element analysis

I. INTRODUCTION

A. Column

A column is a vertical member that supports the beam, slabs and transfers loads from itto the next floor below or up to the foundation, and ultimately load is transferred to theground. Basically, the column is a compression member because the load path is through its longitudinal axis. The bending moment may be induced in column about one or both of the perpendicular axes due to eccentricity, wind, earthquake or accidental loads. It is assumed to be the most significant structural member of a building as the safety of a building rest on the column strength. This is since the failure of the column would cause a progressive collapse in buildings whereas such an event would not occur when other members fail. The failure of horizontal members such as beams or columns will not be instant and will give people enough time for either moving out or even for repairing, however the failure of a columns may be sudden and lead to collapse of the building. Hence column needs to be designed and constructed with more care and proper supervision.

B. Types of Reinforcement

Concrete columns, especially those subjected to bending stresses, must always be reinforced with steel. In concrete columns, vertical reinforcement is the principal reinforcement. However, a loaded column shortens vertically and expands laterally; hence, lateral reinforcements in the form of lateral ties are used to restrain the expansion. Columns reinforced in this manner are called tied columns. If the restrainingreinforcement is a continuous winding spiral that encircles the core and longitudinal steel, the column is called a spiral column.

II. RESEARCH METHODOLOGY

A. General

The present work is divided into two parts. In the first phase literature review is done on effect of size on reinforced concrete columns subjected to seismic loading. The modelling of reinforced concrete buildings was done using ETABS software.
In the second phase finite element analysis of columns was done using ANSYS software.

B. Problem Formulation

1) Modelling of reinforced concrete building was done using ETABS software.
2) Study the procedure to conduct finite element analysis using ANSYS
3) Verify and compare the results obtained from ANSYS with various experimental studies.

Finite Element Analysis of columns with different reinforcement spacings is done using ANSYS software to see the behavior of columns under the effect of axial andlateral loading in terms of cracking, crushing and stresses in reinforcements

III. FINITE ELEMENT MODELLING & ANALYSIS

A. Finite Element Analysis

Ductility is calculated as the ratio of post-yield deformation to yield deformation of steel in reinforced concrete (RC) structures. Ductility is especially vital in structures because it ensures safety when structures are subjected to accidental, impact or seismicloads. In these cases, a ductile structure permits the stress redistribution and production of plastic hinges within the critical parts of the structure without severe loss ofresistance capacity. It eventually avoids the total collapse of the structure by issuing a warning of imminent failure.

B. Material Definition and Modelling

For any Finite Element Analysis, selecting a proper element type is an important criterion. The concrete parts were modeled using a special element available in the library particularly for Concrete, called SOLID65 element. The reinforcements were modeled using LINK180 element. SOLID185element can be used to model steel platesto be used for loading or supports. The details of both the elements used in the analysisare explained below briefly:

1) Concrete

ANSYS utilizes an eight-node element SOLID65, which is used as non-linear model for brittle material such as concrete. SOLID65 is used for the 3D modelling of solids with or without reinforcement bars (rebar).

In concrete applications, the solid element may be used to model the concrete while the rebar is available for modelling the reinforcements. It is a solid iso-parametric element with three degrees of freedom at each node-translations in the nodelocations x, y, and z directions. The most important aspect of SOLID65 is the determination of nonlinear material properties. SOLID65 is capable of plastic deformation, directional integration point cracking in tension in three orthogonaldirections, and crushing in compression crushing including plastic and creep behavior.The reinforcement element has uniaxial stiffness only and is assumed to be smeared throughout the element.

2) SOLID65 Concrete Information

The data listed in Table4.1is entered in the data table with the commands. Data which are not input will be assumed as zero, excluding default constants. Eight constants maybe defined with the TBDATA commands following a temperature definition on the TBTEMP commands. The constants (C1-C9) entered on the TBDATA commands are:

Table 4.1 SOLID65 Material Constants

Constant	Meaning
1	Shear transfer coefficients for an open crack.
2	Shear transfer coefficients for a closed crack.
3	Uniaxial tensile cracking stress.
4	Uniaxial crushing stress (positive).
5	Biaxial crushing stress (positive).
6	Ambient hydrostatic stress state for use with constants 7 and 8.
7	Biaxial crushing stress (positive) under the ambient hydrostatic stress state (constant 6).
8	Uniaxial crushing stress (positive) under the ambient hydrostatic stress state (constant 6).
9	Stiffness multiplier for cracked tensile condition, used if KEYOPT (7) = 1 (defaults to 0.6).

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 11 Issue IV Apr 2023- Available at www.ijraset.com

C. Analysis

The analysis of the columns has been solved analytically by using the software ANSYS 16.0. It solves the engineering problems using Finite Element Method (FEM). The analysis using FEM is said to be Finite Element Analysis (FEA). FEA comprises threesteps to solve the problem.

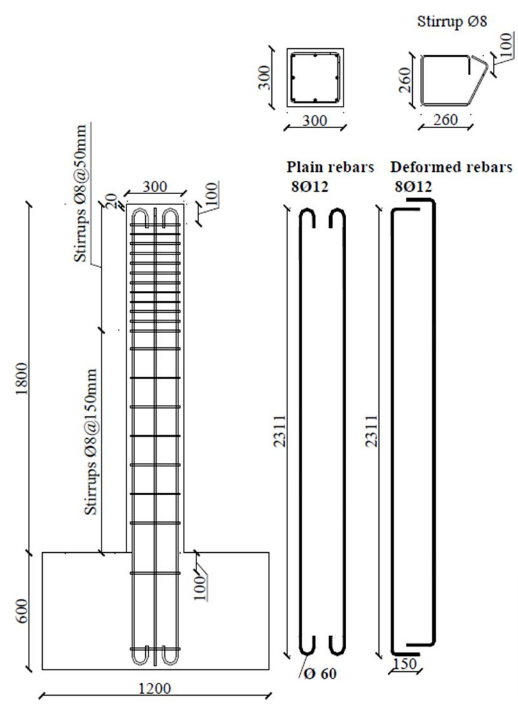
They are:

1) *Pre–Processing:* The results of pre - processing are to develop a suitable finite element mesh, against appropriate material properties, and to apply boundary conditions and loading conditions. The finite element mesh subdivides the geometry into smaller elements, upon which are found the nodes. These nodes are really just coordinating in space, and are generally located at the element corners and sometimes near midpoint of element.
2) *Solution:* The pre - processing and the post - processing stages of the finite element method are interactive for the analyst. The solution is often a batch process, and it requires computer resource. The equations are formed into a matrix and solved numerically. The equation production process depends not only on the type of analysis(e. g. static and dynamic), but also on the model's element types, material properties and boundary conditions.
3) *Post-Processing:* After completing and checking a finite element model, boundary conditions have been applied and the model is solved. Then, the results are studied. This output is known as the post-processing. The post - processing begins thorough a check for errors that may have occurred while arriving at the solution. Most solutions provide a log file, which should be examined for warning or errors.

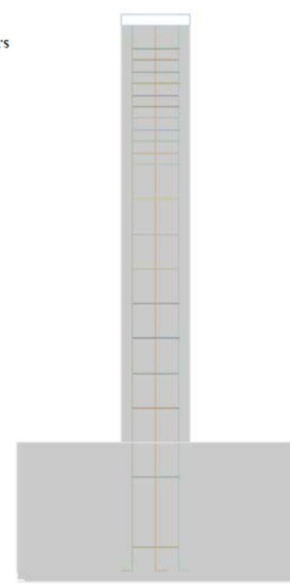
IV. ANSYS VALIDATION RESULTS

A. Columns Subjected to Axial and Monotonic Lateral Loading

The strength and deformation capacity of RC columns reinforced with plain and deformed rebars are investigated in this study. Two square columns having Plain and deformed bars were taken from paper entitled "CYCLIC BEHAVIOR OF NON- CONFORMING FULL SCALE RC COLUMNS" and subjected to monotonicloading and their experimental results were obtained. The same were modeled in ANSYS and the results were compared.



(a)



(b)

Figure 4.1 Concrete column (a) Experimental Setup (b) FEA Model

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 11 Issue IV Apr 2023- Available at www.ijraset.com

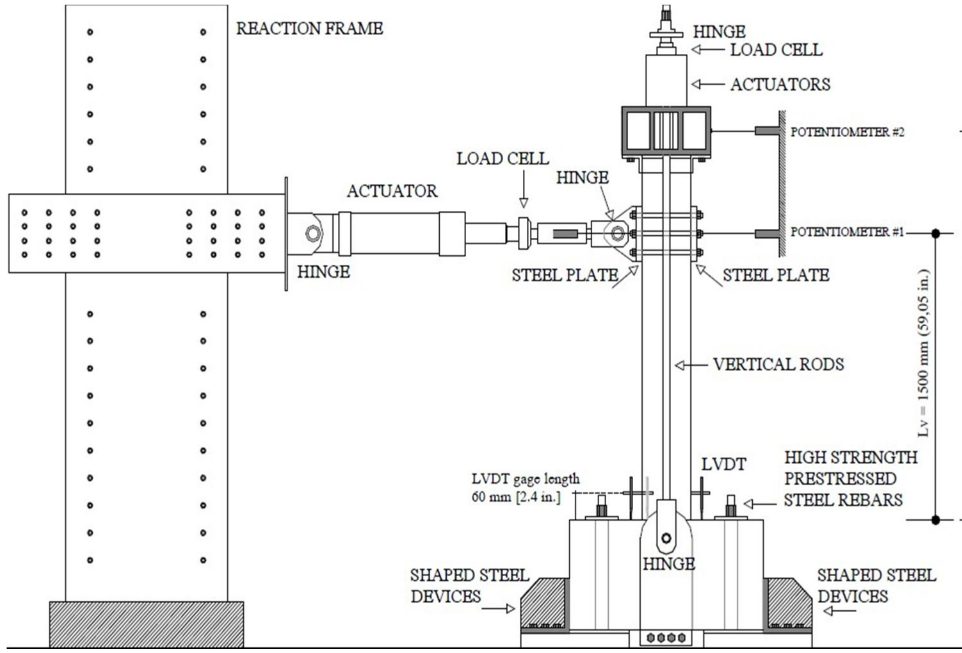


Figure 4.2 Experimental Setup Information

Cycle	*Displ. rate* [mm/sec.]	*Δ* [mm]	*Drift, θ* [%]
I	0.2	6.0	0.4
II	0.2	12.0	0.8
III	0.2	24.0	1.6
IV	1.0	36.0	2.4
V	1.0	48.0	3.2
VI	1.0	60.0	4.0
VII	1.0	72.0	4.8
VIII	1.0	84.0	5.6
IX	1.0	96.0	6.4
X	1.0	108.0	7.2
XI	1.0	120.0	8.0

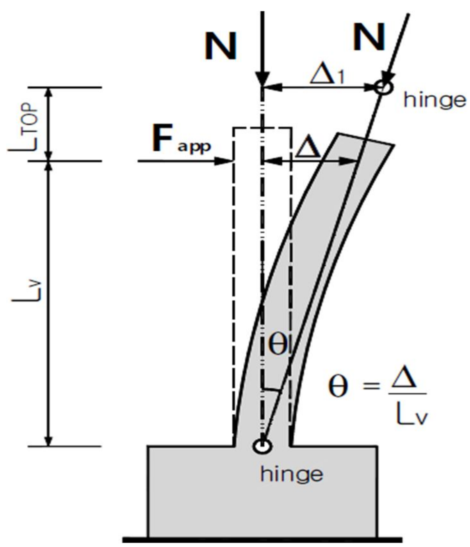


Figure 4.3 Loading Data

Table 4.1 Column Configuration and Material Properties

Name	Reinforcement	Length	Width	Depth	Concrete strength (fc’)	Steelyield stress (fy)
S-300P-m	8# 12mm + 8mm Stirrups	1800mm	300mm	300mm	22.71 MPa	330 MPa
S-300D-m	8# 12mm + 8mm Stirrups	1800mm	300mm	300mm	22.71 MPa	520 MPa

The geometry of FEM models created in ANSYS using the data provided in reference paper; the boundary conditions, loading, displacements, etc were identical to the information provided in the research paper. The material properties and test data are available in Table

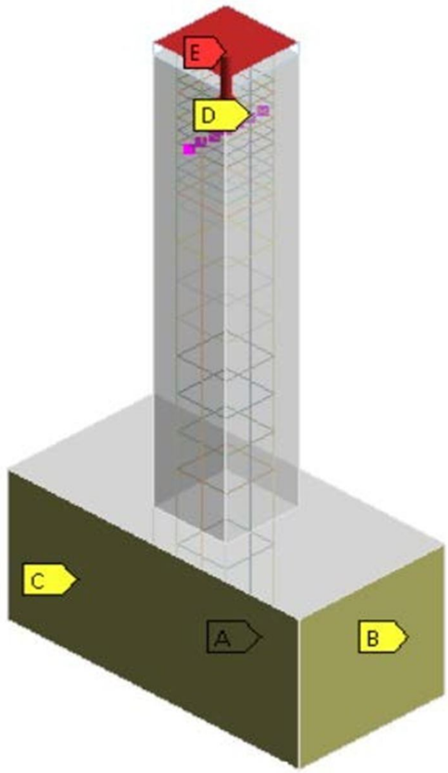


Figure 4.4 Finite Element Model with Boundary Conditions

1) Column with Plain Reinforcement Bars

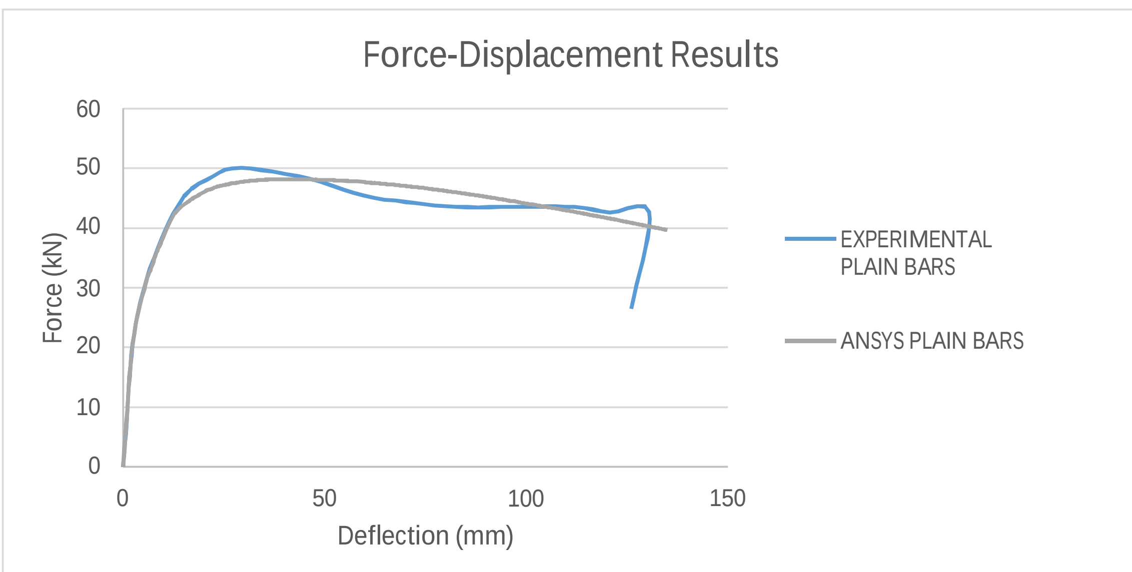


Figure 4.5 F-D Curve for Column with Plain Bars

From the F-D curve, we observed that a good relationship can be obtained using FEM software to determine the peak loads with reasonable accuracy. The results obtained from ANSYS show good agreement with the test data.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 11 Issue IV Apr 2023- Available at www.ijraset.com

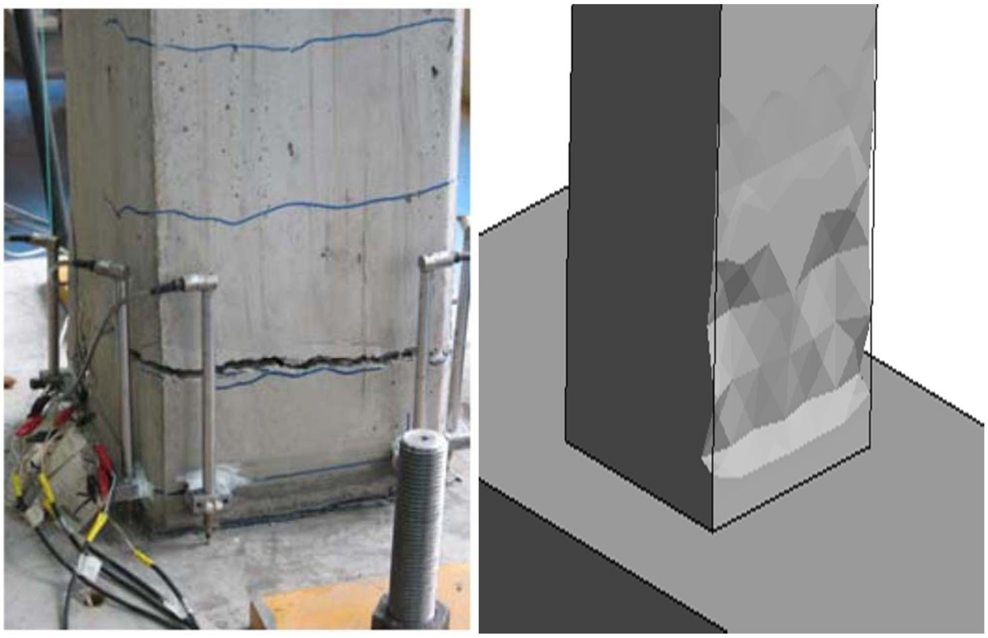
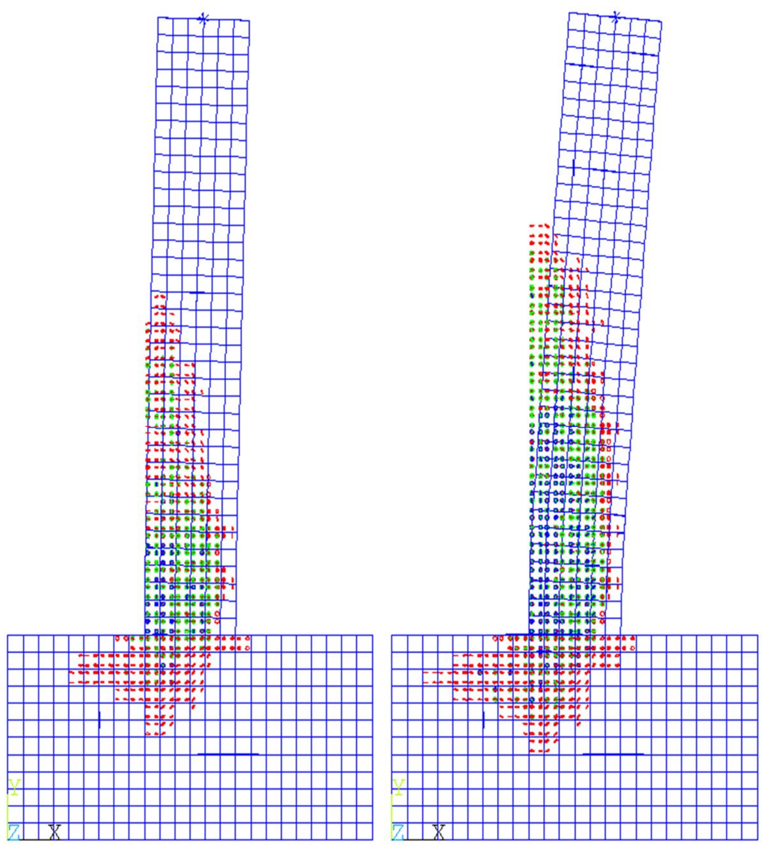


Figure 4.5 Compression Damage in Section

In addition to force displacement curves, we can also obtain compression damageusing FEM analysis.



(a) (b)

Figure 4.6 Tensile and Flexural Cracking At (a) Peak Loads And (b) Peak Displacements

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 11 Issue IV Apr 2023- Available at www.ijraset.com

2) Column with Deformed Bars

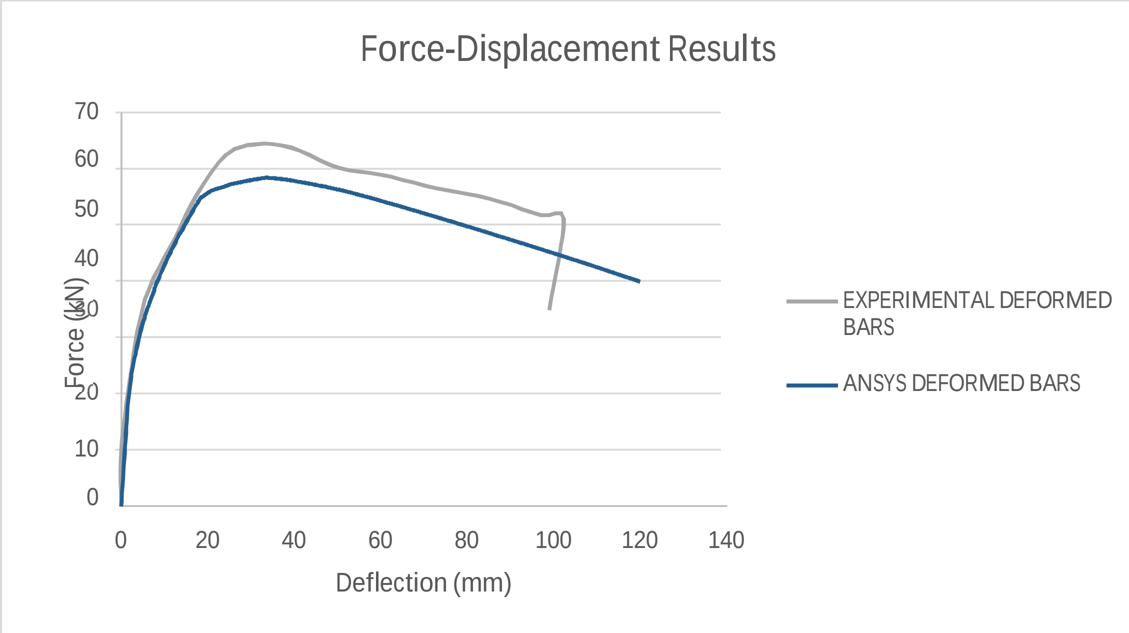


Figure 4.7 Force-Displacement Curves for Deformed Bars

Similarly, for the column with deformed bars, we obtained a fairly good relationship for force-displacement curves, the peak loads had a difference of about 10 percent which is great in accordance with structural analysis. We can also see that ANSYS is able to determine the falling slope of force displacement curves.

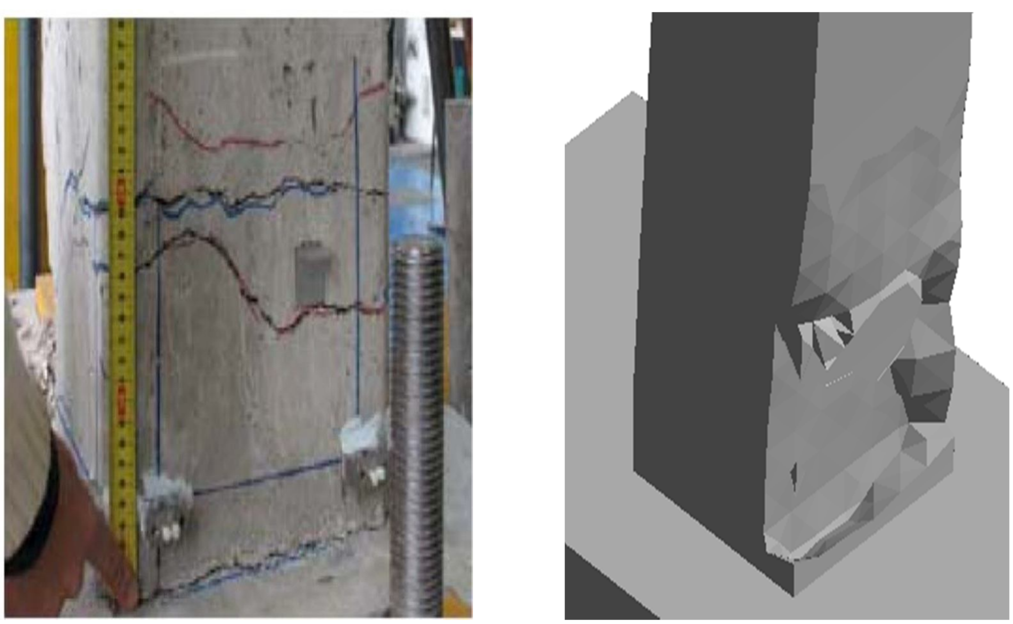


Figure 5.23 Compression Damage in Concrete Column with Deformed Bars

The compression damage in the column having deformed bars increased as it can be seen in the damage from experimental model, this can be explained by the fact that deformed bars take additional stresses which are transferred to the concrete and thus leading to higher crushing in the concrete. Similarly, if we obtain the cracking due to tension and flexural bending of column, higher cracks in the columns with plain steel bars whereas there is a reduction in cracks formed in the column with deformed bars.

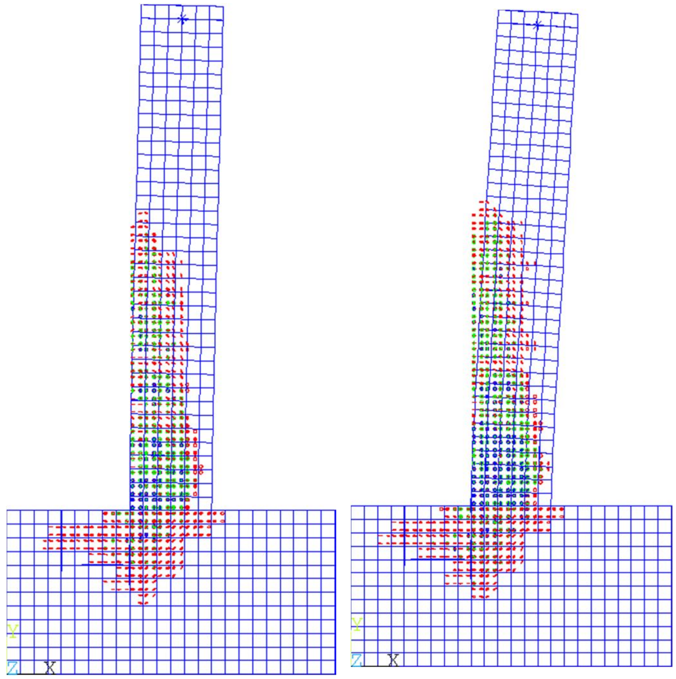


Figure 5.24 Tensile and Flexural Cracking at (a) Peak Loads and (b) Peak Displacements

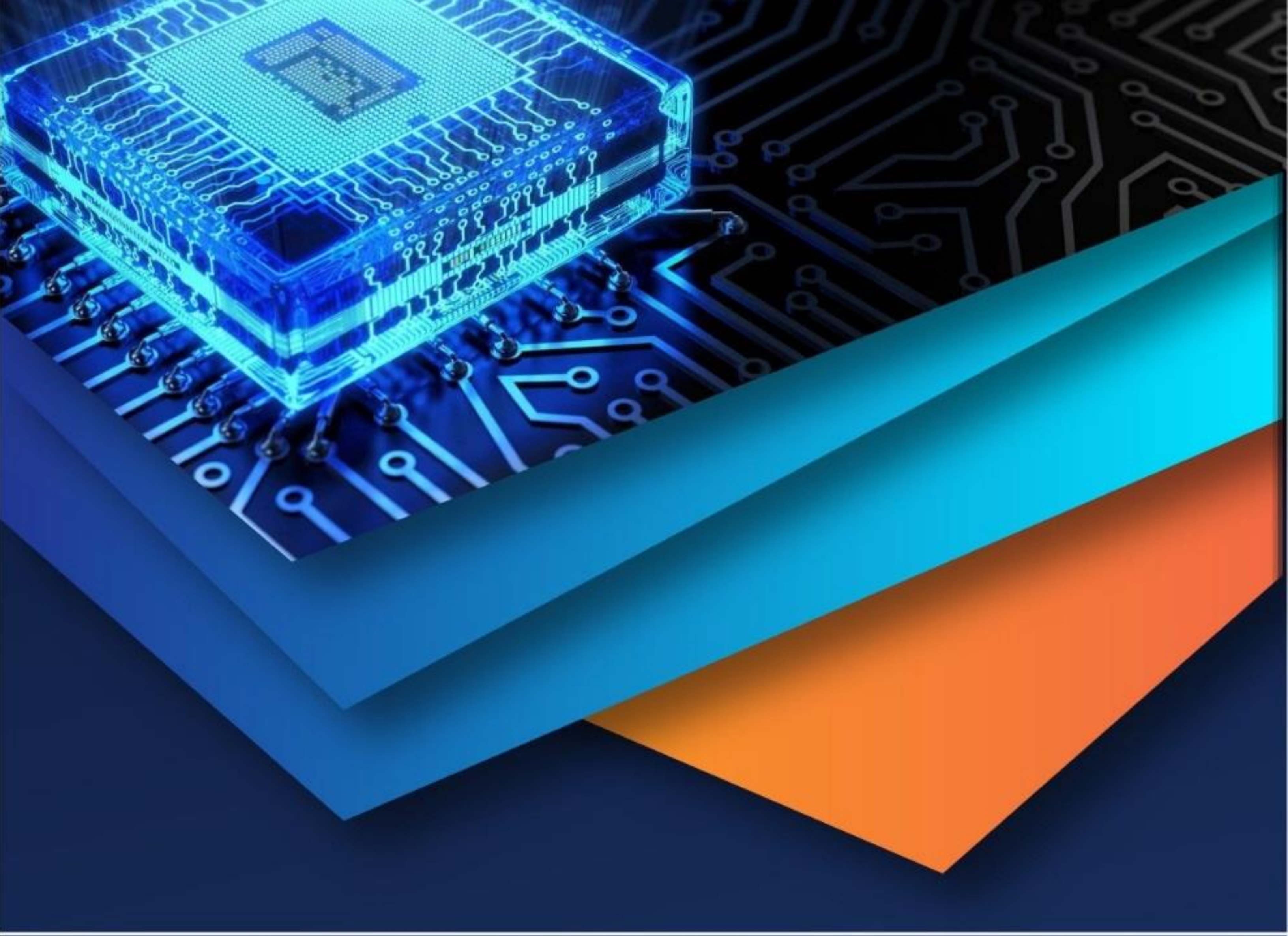
V. CONCULSION

From the above-mentioned results, it can be concluded that the simulations ran on ANSYS FEM software produces results with good accuracy which can be seen and compared with experimental results. Using ANSYS, we can also obtain results which are hard to experimentally evaluate or calculate, such as stresses, strains in reinforcement bars, tensile stresses in concrete at various sections, etc. From the inferences obtained through the testing of columns and beams, it is clear that ANSYS can be used for nonlinear finite element analysis of reinforced concrete material and the simulations provide large amount of data which can be used for deep analysis, this is not possible in physical experimental testing because of limited equipment availability, high costs and calibration problems. Hence finite element analysis is a very user-friendly tool to conduct analysis of reinforced concrete short amount oftime, without the need to create physical models and less equipment.

REFERENCES

[1] ANSYS Academic Research Mechanical, Release 20.2, ANSYS, Inc.
[2] Abd-Elhamed M.K. and Owida M.E. (2019). "Effect of stirrups densification on ultimate capacity of rectangular reinforced concrete columns." Structures 20 (2019)728–764.
[3] Abdesselama H., Kassoula A., Bouzidb H., (2019). "New model for confinement of reinforced concrete columns with an ultrahigh strength close to 200 MPa." Engineering Structures 199(2019) 109594.
[4] Ahmed F.S., Khan M.A. (2013). "Finite element analysis of reinforced concrete column under lateral load." International Journal of Engineering Research and Applications (IJERA), Vol. 3, Issue 4, Jul-Aug 2013, pp. 228-231.
[5] Badiger N.S., Malipatil K.M. (2014). "Parametric study on reinforced concrete beam using ANSYS" Civil and Environmental Research, ISSN 2224-5790 (Paper)ISSN 2225-0514 (Online), Vol.6, No.8, 2014.
[6] Bai Z. Z. and Au F. T. K. (2009). "Ductility of symmetrically reinforced concrete columns." Magazine of Concrete Research, 2009, 61, No. 5, June, 345–357.
[7] Barbosa A.F. and Ribeiro G.O. (1998). "Analysis of reinforced concrete structures using ANSYS nonlinear concrete model."COMPUTATIONAL MECHANICS, NewTrends and Applications, S. Idelsohn, E. Oñate and E. Dvorkin (Eds.).
[8] Candappa D.P., Setunge S., Sanjayan J.G. (1999). "Stress versus strain relationship of high strength concrete under high lateral confinement." Cement and Concrete Research 29 (1999) 1977–1982.
[9] Chung H.S., Yang K.H., Lee Y.H., Eun H.C. (2002). "Strength and ductility of laterally confined concrete columns." Canad J. Civil engineering, Vol. 29, 2002, pp. 820-830.
[10] Code of Practice for Plain and Reinforced Concrete, IS 456:2000, Bureau of IndianStandards, New Delhi.
[11] Code of Practice for Ductile Design and Detailing of Reinforced Concrete Structures Subjected to Seismic Forces, IS 13920:2016, Bureau of Indian Standards, New Delhi.

[12] Criteria for Earthquake Resistance Design of Structures, IS 1893:2016(1), Bureauof Indian Standards, New Delhi.
[13] Dahmani L., Khennane A., and Kaci S. (2010). "Crack identification in reinforcedconcrete beams using ANSYS software." Strength of Materials, Vol. 42, No. 2, Springer Science + Business Media, Inc.
[14] EL-Attar M., El-Karmoty H.Z. & EL-Moneim A.A. (2016). "The behavior of ultra-high-strength reinforced concrete columns under axial and cyclic lateral loads." HBRC Journal, 12:3, 284-295.
[15] ETABS v18 "Building Analysis and Design software" CSI, Inc.
[16] Gramblička S., Veróny P. (2013). "Transverse reinforcement in reinforced concretecolumns." SSP - JOURNAL OF CIVIL ENGINEERING Vol. 8, Issue 2, 2013
[17] Hasan W.M., El-Khatieb M. and Al-Ani H. (2011). "Steel reinforcement ratio dependency of plastic rotational capacity of reinforced concrete beams." Jordan Journal of Civil Engineering, Volume 5, No. 4, 2011.



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY