ANALYTICAL STUDY ON STRESS-STRAIN BEHAVIOUR OF REINFORCED CONCRETE COLUMN

Ananya John¹, Unni Kartha G.², Prof. S.Usha³
¹, ²Civil Engineering, SNGCE Kadaviruppu, India
³Civil Engineering, FISAT Mookkanoor, India

ABSTRACT

Reinforced concrete columns are main load bearing members in any type of structure, which contribute lateral stiffness also. There are two types of reinforcements in a column ie. Longitudinal and transverse reinforcements. Longitudinal reinforcements are provided parallel to the longitudinal axis of column and transverse reinforcements may be hoop, ties or spirals. Stress-strain behaviour of column is important in order to find out the available ductility from a column by moment curvature analysis. This paper presents analytical study on stress-strain behaviour of reinforced concrete column by modelling concrete and steel part separately. A new finite element software Calculix and ANSYS software are used for the analysis. The results were validated using the available experimental data.

Keywords: ANSYS, Calculix, Column, Strain, Stress.

1. INTRODUCTION

Reinforced concrete columns are main load bearing members of any type of structure. It is necessary to design and detail the column member adequately since it support beams and slabs and transfer the load to the foundation. There are two types of reinforcements in a column element. They are; (1)Longitudinal reinforcements: To take care of the moments and forces in columns. Provided parallel to the longitudinal axis of the column. (2)Transverse reinforcements: It may be hoops, ties or spirals. It is provided to take care of local buckling, shear resistance and it confines the concrete also. So transverse reinforcements increases the strength and ductility of column member.

Stress-strain behaviour of column is important in order to find out the available ductility from a column element by moment curvature analysis. Many studies have been conducted in reinforced concrete columns. In such studies many of them are experimental studies and others numerical are numerical approaches. In that experimental program the maximum load carrying capacity, effect of confinement, ductility etc. were studied. The numerical approaches are proposed a number of mathematical models, from which the stress-strain behaviour can be obtained. The experimental studies are very costly and time consuming. The use of different finite element softwares overcome such problems. In this study the finite element softwares Calculix and ANSYS were used. Discrete modelling is adopted in ANSYS. In both the softwares concrete and steel parts are separately modelled so that its behavior is same as that of experimental program.
2. FINITE ELEMENT MODELLING

The finite element analysis softwares give a good approach to reinforced concrete members. In ANSYS modelling graphic user interface is used while in Calculix, it is based on commands. Table I shows different column specimens which is used in this analytical studies. To validate the results with experimental data, the same specimens used in the experimental programs[1-2]were used in this study. These specimens were modelled in both the softwares.

2.1 Calculix Software

Calculix is a open source finite element analysis application that uses a similar input format to Abaqus. It has an implicit and explicit solver (CCX) written by Guido Dhondt and a pre and post processor (CGX) written by Klaus Wittig. The solver is able to solve static, dynamic, buckling analysis, heat transfer etc. Material nonlinearities, as well as geometrical nonlinearities, can be introduced to solve more complex structural and mechanical problems. Calculix includes a complete element library for volumetric elements, as well as quadratic formulations for plane stress, plane strain, axi-symmetric, shells, and beam elements.

2.1.1 Calculix Input Deck

An input file is commonly called an “input deck”, the characters from the asterisk to the first comma are called a “keyword” and the keyword with associated data is called a “card”. An input deck defines the finite element analysis problem to be solved. In general, it contains a mesh definition, material, analysis type, boundary conditions and output requests.

2.1.2 Geometry

Columns are modelled in the same way which is used in the experimental program.

2.1.3 Material Properties

Material properties such as Young’s modulus and poisson’s ratios are specified in the calculix inputdeck.

The following concrete and steel properties are defined in Calculix inputdeck and it is assigned in ANSYS.

- Compressive strength of concrete (fc’)
- Modulus of Elasticity of concrete (Ec)
- Concrete density=25 kN/m3
- Poisson’s ratio of concrete(0.2)
- Elastic modulus of steel(Es)
- Poisson’s ratio of steel(0.3)

2.1.4 Elements Used

20 node brick element (he20 or C3D20) was used for the modelling of both concrete and steel. The C3D20 element is a general purpose quadratic brick element. It can be used for linear and nonlinear problems. Fig. 1 shows the he20 element.

![Fig 1: He20 element](image-url)
Table 1: Details of the Column Specimens

<table>
<thead>
<tr>
<th>Sl No</th>
<th>Size (mm)</th>
<th>fc’ (MPa)</th>
<th>Eccentricity (mm)</th>
<th>Reinforcements</th>
<th>Section</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>500 x 500 x 1500 (C1)</td>
<td>24.3</td>
<td>-</td>
<td>Long. Reinf. 16mm Φ Ties 10mm Φ @ 60mm c/c</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>500 x 500 x 1500 (C2)</td>
<td>24.3</td>
<td>-</td>
<td>Long. Reinf. 16mm Φ Ties 10mm Φ @ 40mm c/c</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>200 x 200 x 800 (C3)</td>
<td>48.3</td>
<td>-</td>
<td>Long. Reinf. 12mm Φ Ties 8mm Φ @ 50mm c/c</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>500 Φ (C4)</td>
<td>28.8</td>
<td>-</td>
<td>Long. Reinf. 16mm Φ Transverse Reinf. 10mm Φ @ 300mm c/c</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>500 Φ (C5)</td>
<td>28.8</td>
<td>-</td>
<td>Long Reinf. 16mm Φ Transverse Reinf. 10mm Φ @ 150mm c/c</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>200 x 200 x 800 (C6)</td>
<td>49</td>
<td>20</td>
<td>Long. Reinf. 12mm Φ Ties 8mm Φ @ 50mm c/c</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>200 x 200 x 800 (C7)</td>
<td>49</td>
<td>40</td>
<td>Long. Reinf. 12mm Φ Ties 8mm Φ @ 50mm c/c</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>200 x 200 x 800 (C8)</td>
<td>49</td>
<td>60</td>
<td>Long. Reinf. 12mm Φ Ties 8mm Φ @ 50mm c/c</td>
<td></td>
</tr>
</tbody>
</table>

2.1.5 Loads and Boundary Conditions

Displacement boundary conditions are applied to all column models. For concentric column models axial pressure is applied on the top of the specimen. For eccentric column specimen axial pressure on the top plus couple of forces at extreme edge on both sides of column faces were applied.

Fig.2 shows different column modelled in Calculix.
2.2 ANSYS Software

In ANSYS, there are 3 stages for analysing a structure. They are: (1) Pre-processing, (2) Analysis and (3) Post-processing. Discrete modeling is adopted for modeling column specimens.

2.2.1 Element Types

The Solid65 element was used to model the concrete. This element has eight nodes with three degree freedom at each node. A Link8 element was used to model steel reinforcement. This element is a 3D spar element and it has two nodes with three degrees of freedom.
2.2.2 Modelling

Modelling of all column specimens using ANSYS were done by creating a three dimensional solid (volume). Considering symmetry and to avoid complexity only quarter portion of the specimen is modelled. After creating solid volume meshing is done. Meshing of reinforcement is not needed. Because reinforcements were created in the modeling through the nodes created by mesh of solid volume.

The following Figs. shows different steps in column modelling and modelled concrete part and reinforcement part in ANSYS.
Fig 13: reinforcements of C1

Fig 14: reinforcements of C2

Fig 15: mesh of C3

Fig 16: reinforcements of C3

Fig 17: reinforcements of C4

Fig 18: reinforcements of C5
3. STATIC ANALYSIS

Static analysis was performed in Calculix and ANSYS softwares. The stress and strain diagram obtained from the softwares were shown in following figures. The peak stress and strain values were noted.

Fig 19: stress diagram of C1

Fig 20: strain diagram of C1

Fig 21: stress diagram of C4

Fig 22: strain diagram of C4
4. RESULTS AND DISCUSSIONS

The values obtained from the analysis is validated with the available experimental data. The peak stress, peak strain values were compared. Stress-strain curves were plotted.
Fig 27: stress-strain curve of C1

Fig 28: stress-strain curve of C2

Fig 29: stress-strain curve of C3

Fig 30: stress-strain curve of C4

Fig 31: stress-strain curve of C5

Fig 32: stress-strain curve of C6
Proceedings of the International Conference on Emerging Trends in Engineering and Management (ICETEM14)
30 – 31, December 2014, Ernakulam, India

Fig 33: stress-strain curve of C7

Table II shows that comparison of peak stress and peak strain value from experimental and analytical study (both Calculix and ANSYS). From the table, it is clear that Calculix gives more closer values to the experimental values than that obtained from ANSYS. So Calculix is an excellent finite element software for the analysis of reinforced concrete structures.

Table 2: Comparison between Experimental and FEA study

<table>
<thead>
<tr>
<th>Models</th>
<th>Experimental</th>
<th>Calculix</th>
<th>ANSYS 10</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Peak Stress MPa</td>
<td>Peak Strain</td>
<td>Peak Stress MPa</td>
</tr>
<tr>
<td>1</td>
<td>27.44</td>
<td>0.0042</td>
<td>27.9</td>
</tr>
<tr>
<td>2</td>
<td>32.11</td>
<td>0.0052</td>
<td>33.1</td>
</tr>
<tr>
<td>3</td>
<td>60.545</td>
<td>0.0027</td>
<td>61</td>
</tr>
<tr>
<td>4</td>
<td>32.53</td>
<td>0.0032</td>
<td>33.2</td>
</tr>
<tr>
<td>5</td>
<td>40.76</td>
<td>0.0042</td>
<td>40.9</td>
</tr>
<tr>
<td>6</td>
<td>68.34</td>
<td>0.0038</td>
<td>68.2</td>
</tr>
<tr>
<td>7</td>
<td>68.9</td>
<td>0.0037</td>
<td>69</td>
</tr>
<tr>
<td>8</td>
<td>76.99</td>
<td>0.0039</td>
<td>77</td>
</tr>
</tbody>
</table>

4. CONCLUSIONS

The following major conclusions are drawn based on the analytical studies carried out under this investigation.

- Modelling of confined RC column with Calculix and ANSYS is possible.
- Plotted the stress-strain curve of confined RC columns using the results obtained from softwares.
- Analysis results obtained in the present study by creating the model with concrete and reinforcement separately and assigning the properties is more closer to the experimental value than the results obtained with conventional model with properties assigning as RCC.
For future work,

- Reinforced concrete columns subjected to lateral loads can be studied.
- Different type of structures with different types of reinforcements (fibre reinforced etc.) can be studied.

REFERENCES