Indian Journal of Research Foundation, ISSN : 2454 – 6585, Volume 4, Issue 2 (April 2023).

NUMERICAL ANALYSIS OF RCC BEAM WITH VARIOUS REINFORCEMENT USING ABAQUS

T.RAGIN[a*], N.AGNES FLORA[a]

[a] *Department of Civil Engineering, Stella Mary's College of Engineering, Aruthenganvilai*

[b] *Department of Civil Engineering, Stella Mary's College of Engineering, Aruthenganvilai*

ABSTRACT

The goal of this study is to use Abaqus to FEM Model and analyze the Flexural strength conventional concrete beam for static load with various reinforcement. In RCC beams, the reinforcement in tension zone and the provision of shear reinforcement altered by 8mm bar at compression as constant number and tension zone with three different combinations of 8, 10, 12mm bars are used in respective combination. By this combination set of experiment, shear reinforcement is altered with its spacing arrangement by providing shear reinforcement at 150mm and in another case shear reinforcement is placed at two ends and in mid span of beam. The beam of 150mm x 150mm with length of 1000mm is designed and analyzed. The support condition of beam is kept being hinged on both sides. The results of these RCC beams are evolved and compared the result which proves increase in tension reinforcement increases the strength of beams. Simultaneously, increase in shear reinforcement increases the strength of beams. Here M50 Grade of concrete is adopted.

Keywords: RCC beam in abaqus, FEM, Flexural Strength

1 INTRODUCTION

The RCC beams with various combination of reinforcement is analyzed which is a complicated material using ABAQUS. From the previous research work it is observed that FEM analysis and experiment-based testing ABAQUS has been widely used for analyzing individual elements [1-3]. The results from the FEM Analysis are quite reliable to the experimental work which is also cost efficient. Elastic models are commonly used in closed-form solutions for the analysis of forced structural members [4-6]. These are unable to handle situations involving gross

* For Correspondence: *t.ragin1988@gmail.com*

material and geometric non-linearity. When an RC beam with steel reinforcement undergoes non-linear changes, it is desirable to predict effects such as strain and stress fluctuations [5-7]. The modeling of concrete cracking is one of the most common approximations connected with non-linear concrete behavior. Concrete breaks in the tensions one when loaded, causing the stress path to become discontinuous and the load transfer to shift at the cracked portion. It is critical to use proper material criteria and concrete elements to simulate discrete concrete cracking. In this research, arrangement of reinforcement in RCC beams which is to be analyzed for the flexural strength with various size and orientation of bars are to be simulated and evolved. The deflection of each case is compared with the similar experimental work.

2 STEPS IN SIMULATION

The simulation in ABAQUS includes the following steps initially 3D model should be drafted. Followed by 3d modelling the pre-processing such as creation of model with respect to its dimensions given in the experimental research, selection of the element type and type of material like concrete, steel should be selected. For accurate analysis the meshing of element is made. Fine mesh gives accurate results, but it takes time for analysis. Hence optimum mesh blocks are impregnated. The surface and end condition of the element is adopted in the boundary condition. The application of load such as line load, concentrated load given in the experimental approach is adopted in the analysis. Step- The initial process which indicates the application rate of load and load increment option value should be assigned. After the analysis with corresponding the load increment the load and deflection of the element is resulted. The requirements of the stress, deformation, shear values with respect to its coordinates can be visualized using plot or graphical representation through ABAQUS. Finally, the required values of the RCC element plot using the graph report.

To mimic the behavior of the experimental beams, the ABAQUS finite element program was employed in this investigation. Because of its user-friendly interface and parametric modeling capabilities, ABAQUS/Standard was chosen for this simulation. The element type was added to the geometry of the concrete beam using the command prompt in ABAQUS/CAE. A 1D beam model with a defined cross section was generated in ABAQUS/CAE to reinforce a concrete beam.

3 ELEMENT TYPES

The Element in this case, two type of material is used. For concrete the elastic behavior and plastic behavior, Young’s Modulus values is assigned and named as CON. Similarly, for steel reinforcement the properties of the steel such as young’s modulus, Poisson ration are adopted and the material type named as Fe415.

4 MATERIAL PROPERTIES

Table 1: Material models for concrete (CON)

E In X direction	31780Mpa	31278Mpa	32098 Mpa
Poisson Ratio in XY direction	0.20	0.20	0.20

5 PREPROCESSING IN ABAQUS

Table 2: Combination of reinforcement

<table>
<tr><th rowspan="2">Beam Specimens</th><th colspan="2">Compression Reinforcement</th><th>Tension reinforcement</th><th>Shear reinforcement</th><th colspan="2">Spacing between stirrups</th></tr>
<tr><th>Diameter(mm)</th><th>Nos</th><th>Diameter(mm)</th><th>Nos</th><th>Diameter(mm)</th><th>mm</th></tr>
<tr><td>B808S150</td><td>8</td><td>2</td><td>8</td><td>2</td><td>6</td><td>150</td></tr>
<tr><td>B810S150</td><td>8</td><td>2</td><td>10</td><td>2</td><td>6</td><td>150</td></tr>
<tr><td>B812S150</td><td>8</td><td>2</td><td>12</td><td>2</td><td>6</td><td>150</td></tr>
<tr><td>B808SA</td><td>8</td><td>2</td><td>8</td><td>2</td><td>6</td><td rowspan="3">At two support end, Mid-section</td></tr>
<tr><td>B810SA</td><td>8</td><td>2</td><td>10</td><td>2</td><td>6</td></tr>
<tr><td>B812SA</td><td>8</td><td>2</td><td>12</td><td>2</td><td>6</td></tr>
</table>

Figure 1 Modeling of RCC Elements

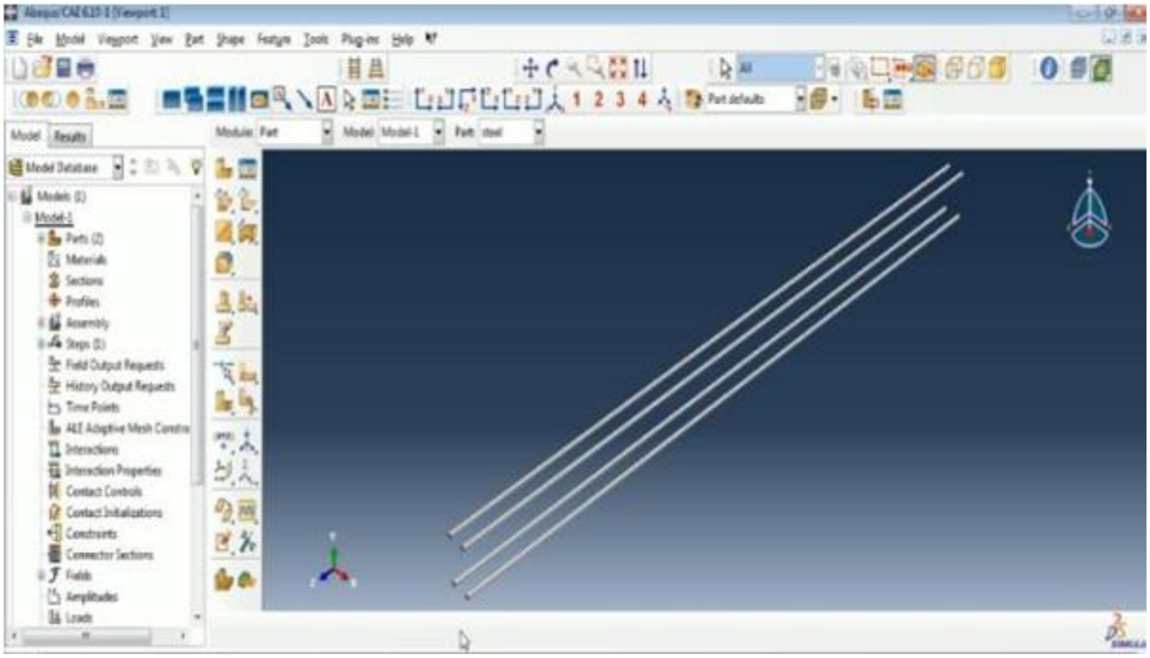


Fig 1.a Modelling – con (concrete)

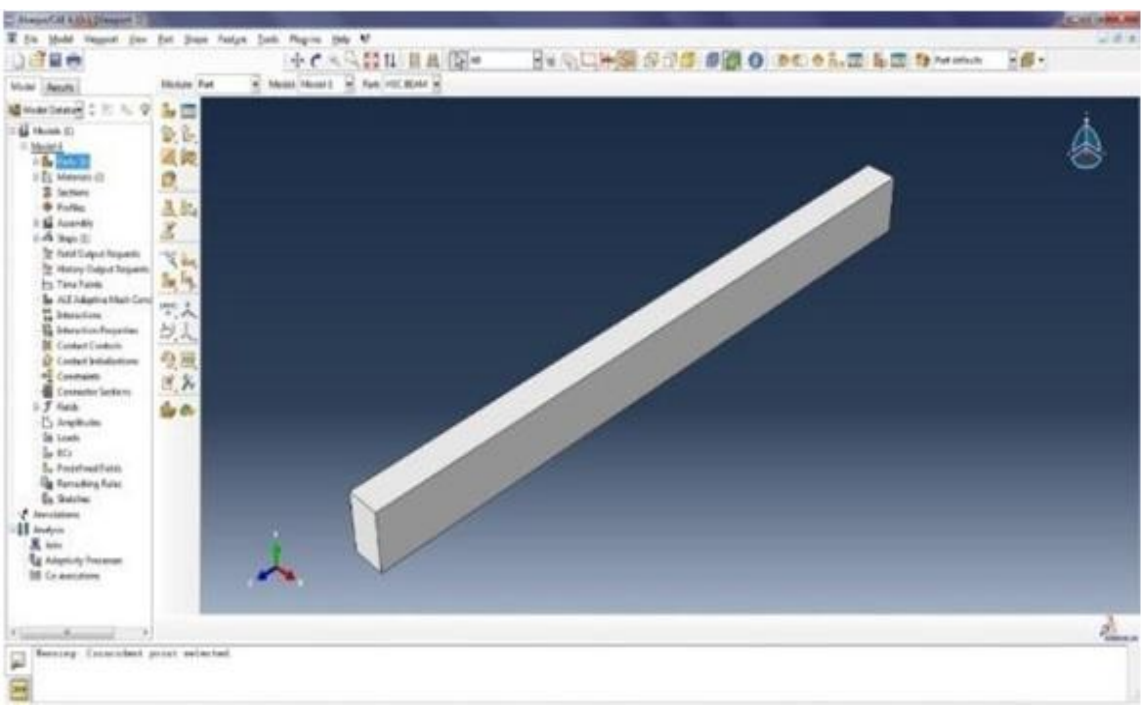


Fig 1.b Modeling of Fe415

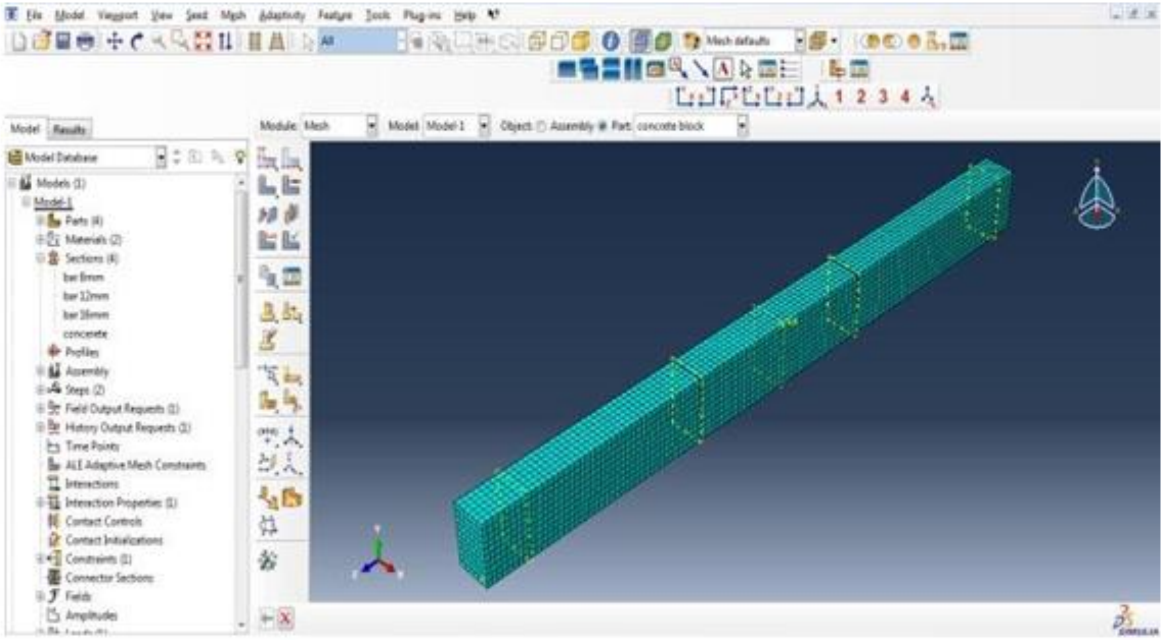


Fig 1.c Meshing con (concrete)

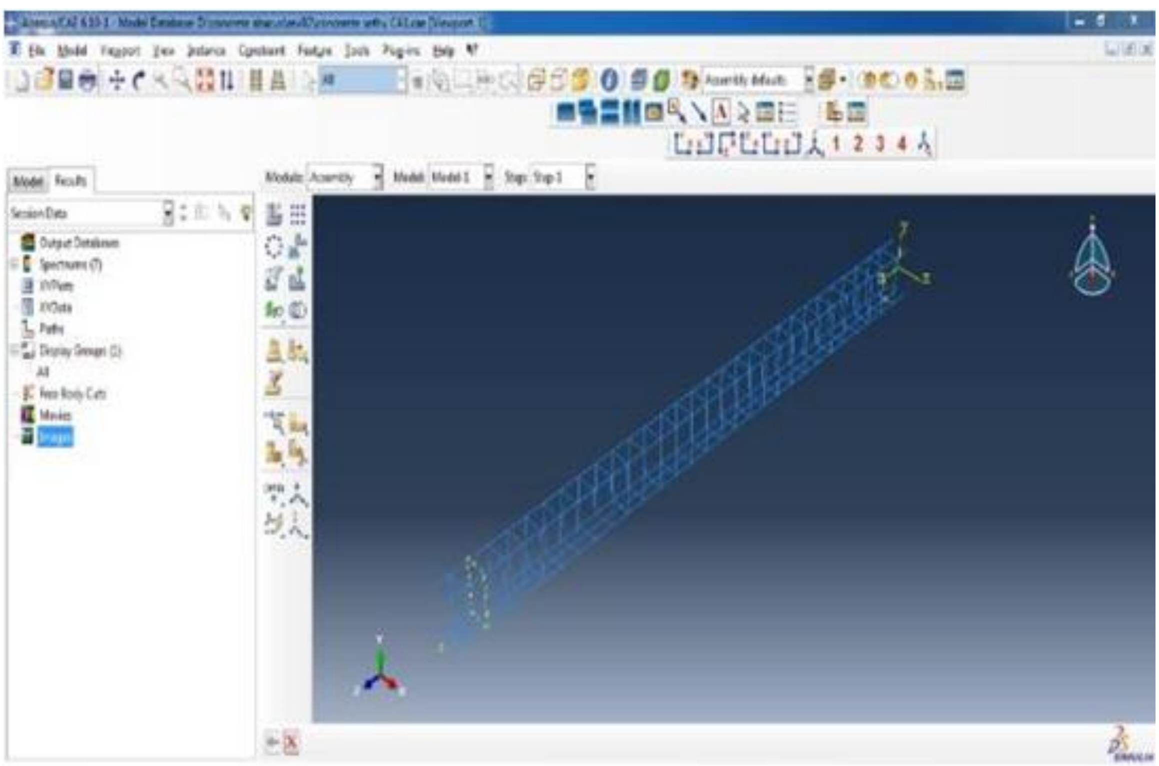


Fig 1.d Meshing Fe415 reinforcement

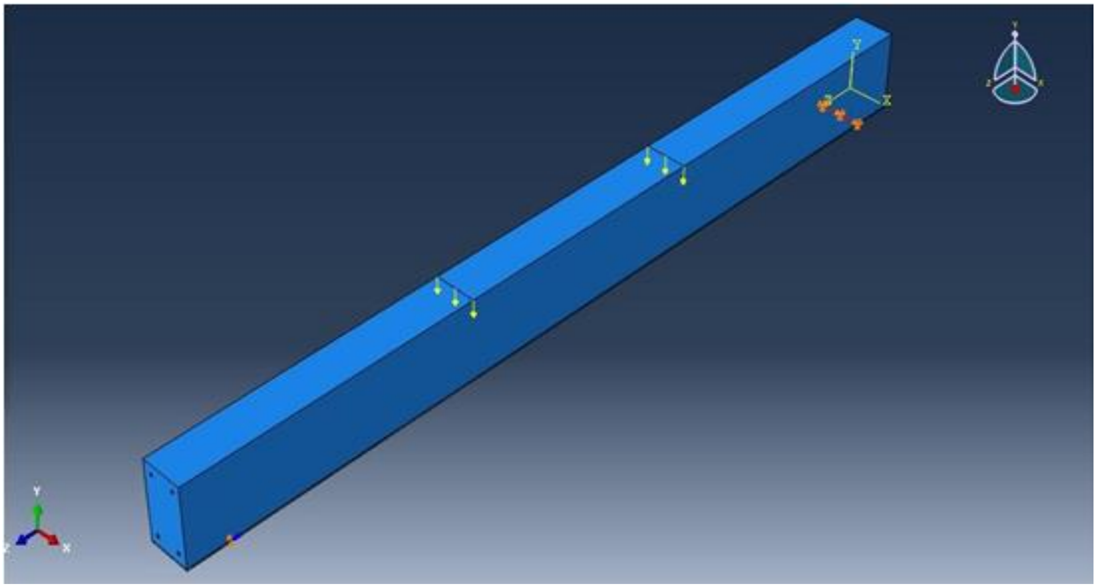
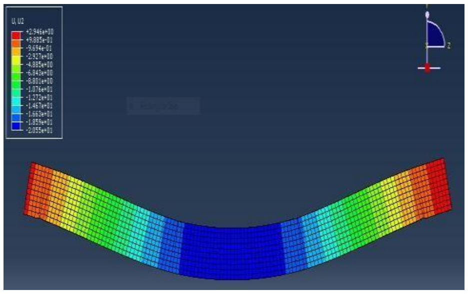
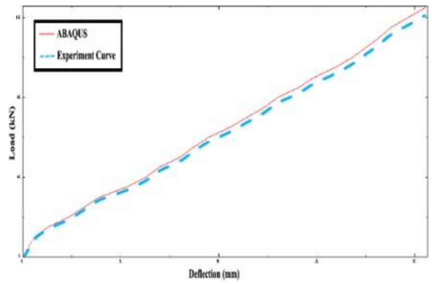
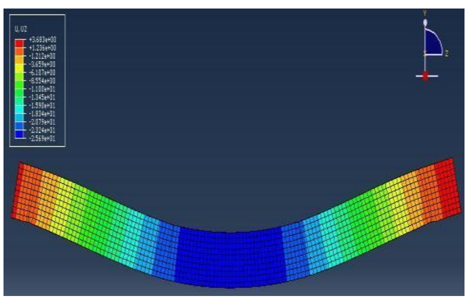
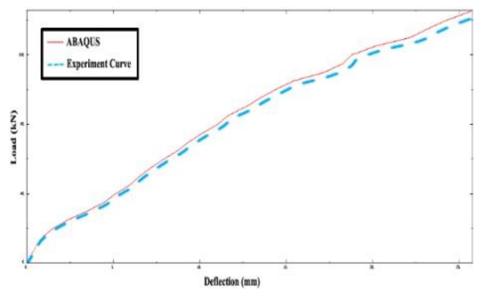
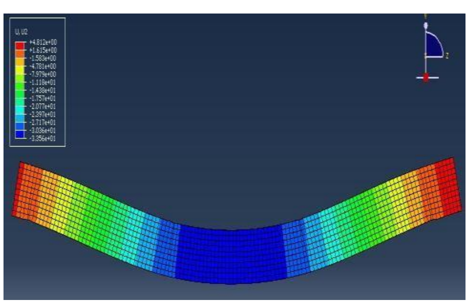
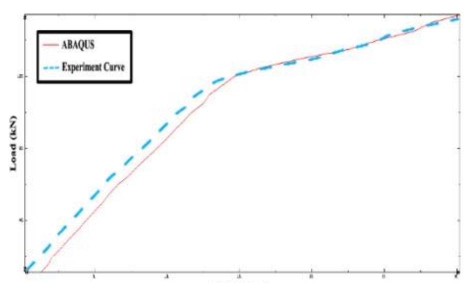


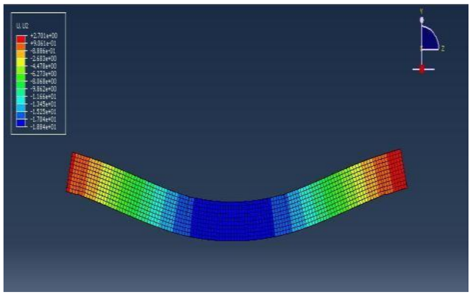
Fig 1.e Line load setup

6 RESULTS AND DISCUSSION OF FINITE ELEMENT ANALYSIS

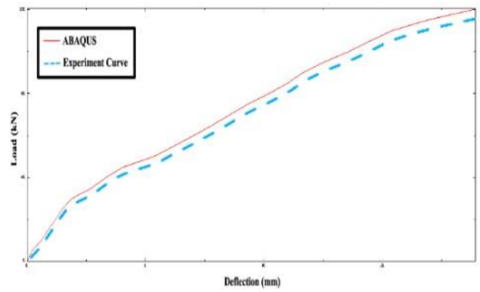
Table 2 Static Load FEA Test Results

Beam ID	Ultimate Load	Ultimate Deflection			Stiffness		
		Expt.	FEA	Error	Expt.	FEA	Error
	kN	mm	mm	%	mm	mm	%
B808S150	91	18	17.21	4.39	5.06	5.29	**4.59**
B810S150	110	23	22.13	3.78	4.78	4.97	**3.93**
B812S150	130	29	27.91	3.76	4.48	4.66	**3.91**
B808SA	84	19	18.44	2.95	4.42	4.56	**3.04**
B810SA	107	26	25.64	1.38	4.12	4.17	**1.40**
B812SA	**124**	**31**	**30.27**	**2.35**	**4.00**	**4.10**	**2.41**

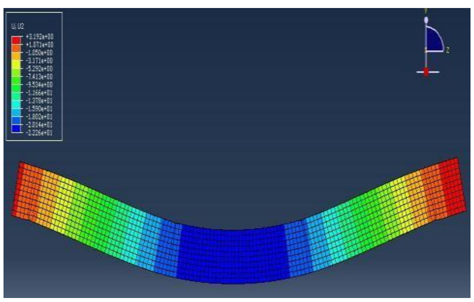
	
STRESS XY of B808S150	Load Deflection of Beam B808S150
	
STRESS XY ofB810S150	Load Deflection of Beam B810S150
	
STRESS XY ofB812S150	Load Deflection of Beam S- B812S150



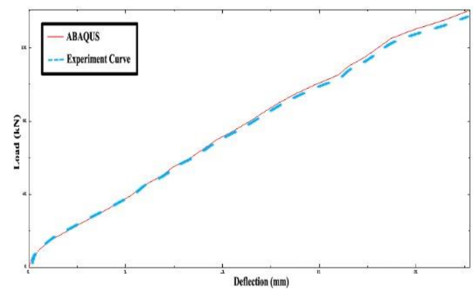
STRESS XY of B808SA



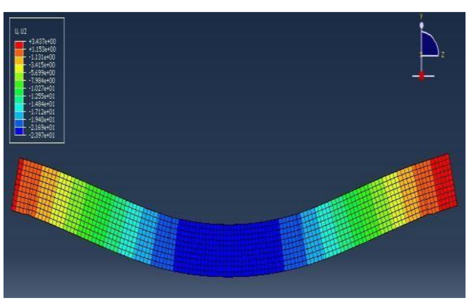
Load Deflection of Beam S- B808SA



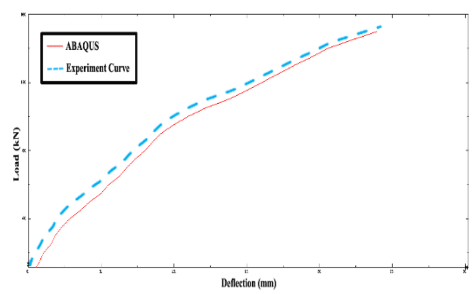
STRESS XY of B810SA



Load Deflection of Beam S- B810SA



STRESS XY ofB812SA



Load Deflection of Beam S- B812SA

Figure 2: Load-Deflection values

The ABAQUS Standard utilized to identify the stress and strain of the RCC members with various reinforcement provisions. Here the beam with shear reinforcement throughout B812S150 failed at an Ultimate load of 130kN by the deflection of 27.91mm. On the other side shear reinforcement at supports and mid-section resulted that the beam B812SA attains 124kN ultimate load with the deflection of 30.27 mm. Fig 1 and 2 shows the entire computerized analysis results.

7 CONCLUSION

From this investigation it is observed that increasing in Area of flexural and shear reinforcement improves the strength of the member. The deflection is satisfactory and within the permissible limits when compared to Numerical Analysis and experimental measurements.

REFERENCE

(1) Yang, Z. J.; Yao, F.; Huang, Y. J. Development of ABAQUS UEL/VUEL Subroutines for Scaled Boundary Finite Element Method for General Static and Dynamic Stress Analyses. *Eng. Anal. Bound. Elem.***2020**, *114* (January), 58–73. https://doi.org/10.1016/j.enganabound.2020.02.004.

(2) Genikomsou, A. S.; Polak, M. A. Finite Element Analysis of Punching Shear of Concrete Slabs Using Damaged Plasticity Model in ABAQUS. *Eng. Struct.***2015**, *98*, 38–48. https://doi.org/10.1016/j.engstruct.2015.04.016.

(3) Cao, X.; Wu, L.; Li, Z. Behaviour of Steel-Reinforced Concrete Columns under Combined Torsion Based on ABAQUS FEA. *Eng. Struct.***2020**, *209* (January), 109980. https://doi.org/10.1016/j.engstruct.2019.109980.

(4) Kartheek, T.; Das, T. V. 3D Modelling and Analysis of Encased Steel-Concrete Composite Column Using Abaqus. *Mater. Today Proc.***2020**, *27*, 1545–1554. https://doi.org/10.1016/j.matpr.2020.03.200.

(5) Balineni, H.; Jagarapu, D. C. K.; Eluru, A. Analysis of Dry and Wet Connections in Precast Beam-Column Joint Using ABAQUS Software. *Mater. Today Proc.***2020**, *33*,

287–295. https://doi.org/10.1016/j.matpr.2020.04.073.

(6) da Silva, G. S.; Kosteski, L. E.; Iturrioz, I. Analysis of the Failure Process by Using the Lattice Discrete Element Method in the Abaqus Environment. *Theor. Appl. Fract. Mech.* **2020**, *107*, 102563. https://doi.org/10.1016/j.tafmec.2020.102563.

(7) Ya, S.; Eisenträger, S.; Song, C.; Li, J. An Open-Source ABAQUS Implementation of the Scaled Boundary Finite Element Method to Study Interfacial Problems Using Polyhedral Meshes. *Comput. Methods Appl. Mech. Eng.* **2021**, *381*, 113766. https://doi.org/10.1016/j.cma.2021.113766.

Received 22 Dec 2022
Revised 4 Feb 2023
Accepted 19 April 2023
Published Online 27 April 2023