Non Linear Finite Element Analysis of SFRSCC and SFRNCC One Way Simply Supported Slabs In Flexure using ANSYS

Shivakumar V Poojar[1], T. Geetha Kumari[2],

[1] *Research Scholor, Civil Engg. Department, M S Ramaiah Institute of Technology, Bangalore, Karnataka, India*
[2] *Assistant Professor, Civil Engg. Department, M S Ramaiah Institute of Technology, Bangalore, Karnataka, India*

---***---

Abstract – *This paper presents a nonlinear finite element modeling and analysis of Steel Fibre Reinforced Self Compacting Concrete (SFRSCC) and Steel Fibre reinforced Normal Cement Concrete(SFRNCC)one way simply supported slabs subjected to four point bending load. In this study, the slabs were modeled using ANSYS V.14 nonlinear finite element software. The percentage of steel fibre was fixed at 1.0%. The concrete is modeled* ***using 'SOLID65'****- eight-node brick element, which is capable of simulating the cracking and crushing behaviour of brittle materials. The tension* ***reinforcement has been modeled discretely using 'LINK-180' –*** *3D spar element. A total ten slabs are analysed, out of which five were SFRSCC and other five were SFRNCC slabs and grade of concrete used was M70 for all the slabs. In these slabs the tensile reinforcement is varied and fibre volume percentages will remain constant for all slabs (i.e, Vf = 1.0%). The slab had an overall dimension of 1100×500×65 mm. The main reinforcement is of 8 mm and the distribution steel was of 6 mm diameter. The fibre contribution to the multiaxial concrete behaviour is considered by changing accordingly the default concrete parameters from ANSYS. The slabs are studied for the ultimate load, load-deflection and load-strain behaviour for each case and compared with the available experimental values.*
The above study indicates that finite element modeling is properly able to simulate the behaviour and strength of SFRC slabs under flexure. The Comparison study showed that the FEA predicts a 10% variation in the deflection studies, the ratio of FE model deflection to Experimental deflection being 1.11 and also the Ultimate load predicted by FE model is lessar than Experimental by a factor of 0.98 and a variation of 11%

Key Words: *SFRSCC, SFRNCC, FEM, ANSYS etc…*

1. INTRODUCTION

Self-Compacting Concrete (SCC) can be defined as a concrete that is able to flow in the interior of the formwork, filling it in a natural manner and passing through the reinforcing bars and other obstacles, flowing and consolidating under the action of its own weight (Okamura 1997). SCC was introduced in Japan in the late 1980's in order to overcome the congestion of steel reinforcement in case of heavily reinforced structures viz., seismic resistant structures. SCC meanwhile is spread all over the world with a steadily increasing number of applications. The use of SCC offers many benefits to the construction practice: the elimination of the compaction work results in reduced costs of placement, shortening of the construction time and therefore improved productivity. Since then several attempts have been made to study the properties of SCC.

Concrete structures are generally analyzed either by specifically developed finite element based computer programs, or by general purpose codes that provide some kind of material model intended to be employed in the analysis of these structures. Even though the latter includes finite elements dedicated to concrete, there is no dedicated SFRC element or material law. On the other hand, although having special finite elements and material laws to represent SFRC, the specific finite element codes are in general private codes which are not always available for the research and industry communities.

Numerous commercial FE analysis codes are available along with the advanced modules for complex analyses. The use of FEA has increased because of progressing knowledge and capability of computer package and hardware. Any attempts for engineering analyses can be done conveniently and fast using such versatile FE analysis packages. Nonlinear material models have been integrated in many of general purpose finite element codes, i.e., ABAQUS, ANSYS, STRAND7, or MSC.NASTRAN. Those nonlinear models play a vital role in nonlinear response analyses since each material component tends to possess the complicated stress-strain behaviour. Among those packages, ANSYS provides a three-dimensional element (SOLID65) with the nonlinear model of brittle materials similar to the concrete [8].

1.1 Experimental Study

T. Geetha Kumari et al (2013) [1], modelled five simply supported SFRSCC and five SFRNCC one way simply supported Slabs of dimension 1100mm x 500mm x 65mm and tested them under four-point bending load. The percentage of steel fibre was kept constant while varying the percentage of main reinforcement.

2. SCOPE OF THE INVESTIGATION

The finite element analysis was conducted using ANSYS, taking advantage of the wide range of element types and material models available in this computer program. The analysis was performed using available material models and element formulations included in the finite element software. The study was intended to provide a better understanding of the behaviour of SFRSCC and SFRNCC one way rectangular slabs under flexure using finite element tools. The finite element models were developed, with the intent of evaluating their load-deflection behaviour and ultimate loads. The cross-sectional geometric and material properties were different for all the ten slabs.

Simulation results were compared with experimental data and conventional analysis theory. Since the study was performed to compare the results with already available experimental data, only models having the exact characteristics of the experimental steel fibre reinforced concrete slab specimens were developed.

3. OBJECTIVES OF THE PRESENT STUDY

The following are the objectives of the study:

- The objective of this study was to understand the flexural behaviour of SFRSCC and SFRNCC slabs under four point bending load, and to simulate the experimental results calculated, load deformation of this type of section is agreeable or not.
- To study the behaviour of Steel Fibre Reinforced Concrete when loaded under the four point bending load and examine the strain and stress at corresponding loads.
- To compare the load carrying capacity of SFRSCC and SFRNCC slabs obtained from the FEA with experimental values.

4. PROBLEM CONSIDERED

Ten one way rectangular simply supported slabs considered for the analysis for which the experimental results are available. Out of the ten slabs, five are made up of Steel fibre reinforced Self compacting concrete (SFRSCC) and other five are made of Steel fibre reinforced Normal cement concrete (SFRNCC). In these slabs the tensile reinforcement is varied and fibre volume percentages will remain constant for all slabs (i.e, Vf = 1.0%). The slab had an overall dimension of 1100×500×65 mm. The main reinforcement is of 8 mm and the distribution steel was of 6 mm diameter. Table 1 gives the detailed dimensions of all the slabs along with spacing of the main reinforcement and percentage of steel used.

Table -1: Slabs Dimensions and their Reinforcement Specifications

Slab designation	Length, L mm	Breadth B mm	Thickness D mm	Spacing of steel mm	Percentage of Steel, ρ (%)
M70-NCF-75	1050	500	65	75	1.202
M70-NCF-100	1050	500	65	100	0.859
M70-NCF-125	1050	500	65	125	0.687
M70-NCF-200	1050	500	65	200	0.601
M70-NCF-215	1050	500	65	215	0.57
M70-SCF-75	1050	500	65	75	1.202
M70-SCF-100	1050	500	65	100	0.859
M70-SCF-125	1050	500	65	125	0.687
M70-SCF-200	1050	500	65	200	0.601
M70-SCF-215	1050	500	65	215	0.57

5. FINITE ELEMENT MODELING

The finite element method is a numerical technique of solving differential equations describing a physical phenomenon. It is a convenient way to find displacements and stresses of structures at definite physical coordinates called nodes. The structure to be analysed is discredited into finite elements connected to each other at their nodes. Elements are defined and equations are formed to express nodal forces in terms of the unknown nodal displacements, based on known material constitutive laws. Forces and initial displacements are prescribed as initial conditions and boundary conditions. A global matrix system is assembled by summing up all individual element stiffness matrices and the global vector of unknown nodal displacement values is solved for using current numerical

International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 -0056
Volume: 02 Issue: 01 | Jul-2015 www.irjet.net p-ISSN: 2395-0072

techniques. Many software programs are available in the market for the analysis of structures by this method. In the present study, the computer program ANSYS is used for the analyses performed.

5.1 Finite Element Modeling of Steel Reinforcement and Steel Fibre

According to Tavarez , three techniques exists to model steel reinforcement in a three-dimensional finite element model of a reinforced concrete structure: the discrete model, the embedded model, and the smeared model as shown in figure-1 [7]. In this study the steel reinforcement are modeled as discrete and embedded, and the steel fibre modeled as smeared model.

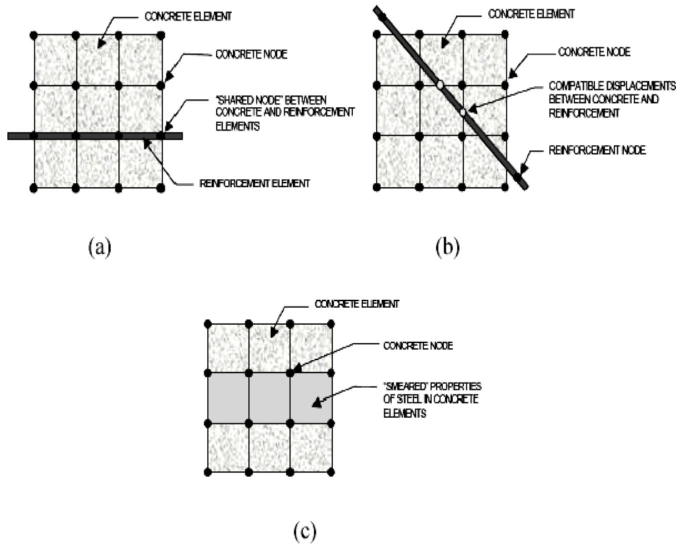


Fig -1: Models for Reinforcement in Reinforced Concrete (Tavarez 2001):
(a) discrete; (b) embedded; and (c) smeared

5.2 Element Types

The accuracy of the finite element analysis results highly depends on choosing the appropriate elements to predict the actual behavior of the structure. In this study, solid 65 and LINK180 were used to model the concrete and steel respectively.

5.3 Real Constants:

The real constants for this model is shown in Table 2.Individual elements contain different real constants. No real constant set exists for the Solid65 element.

Table -2: Material Real Constants

<table>
<tr><th>Real Constant Number</th><th>Element Type</th><th>Parameter</th><th>Value</th><th>Description</th></tr>
<tr><td>1</td><td>Solid 65</td><td colspan="2">No Real Constant input</td><td>Concrete</td></tr>
<tr><td rowspan="2">2</td><td rowspan="2">Link 180</td><td>Cross Sectional Area(mm^2)</td><td>50.26</td><td rowspan="2">Main steel</td></tr>
<tr><td>Initial Strain(mm/mm)</td><td>0</td></tr>
<tr><td rowspan="2">3</td><td rowspan="2">Link 180</td><td>Cross Sectional Area(mm^2)</td><td>28.27</td><td rowspan="2">Distribution steel</td></tr>
<tr><td>Initial Strain(mm/mm)</td><td>0</td></tr>
</table>

5.4 Material Properties

5.4.1 Concrete *:* The Solid element (Solid 65) has eight nodes with three degrees of freedom at each node and translations in the nodal x, y, and z directions. The element is capable of plastic deformation, cracking in three orthogonal directions, and crushing.

Smeared cracking approach has been used in modelling the concrete in the present study. The Solid65 element requires linear isotropic and multi-linear isotropic material properties to properly model concrete. The multi-linear isotropic material uses the von Mises failure criterion along with the Willam and Warnke (1975) [6] model to define the failure of the concrete. EX is the modulus of elasticity of the concrete (Ec),and PRXY is the Poisson's ratio (μ). The compressive uniaxial stress-strain relationship for the concrete model was obtained using the following equations to compute the multi-linear isotropic stress-strain [Saifullah, et al 2011].

$$f = \frac{E_c\, \varepsilon}{1+\left(\frac{\varepsilon}{\varepsilon_0}\right)^2} \quad \textbf{(1)}$$

$$\varepsilon_0 = \frac{2\, f_{ck}}{E_c} \quad \textbf{(2)}$$

$$E_c = \frac{f}{\varepsilon} \quad (3)$$

where:
Ec = elastic modulus of concrete, MPa
fck = ultimate compressive strength, MPa
The equations 1 and 2[8] are used along with Equation 3 to construct the uniaxial compressive stress-strain curve for concrete in this study.[FEA & MVOSD]

Poisson's ratio for concrete is assumed to be 0.2 and is used for all beams. The value of a shear transfer coefficient, representing conditions of the crack face, used in many studies of reinforced concrete structures varied between 0.05 and 0.25 [2], [8]. The shear transfer coefficient used in this study is equal to 0.2.

5.4.2 Reinforcement:

Link-180 element is a uniaxial tension-compression element with three degrees of freedom at each node: translations in the nodal x, y, and z directions. Elastic modulus and yield stress for the steel reinforcement used in this FEM study are taken from the material properties of the steel components used for the experimental tests. The steel for the finite element models is assumed to be an elastic-perfectly plastic material and identical in tension and compression. A Poisson's ratio of 0.3 is used for the steel reinforcement.

Table 3: material properties

Designation	M70-NCF-75	
Criterion	concrete	
Compressive strength	91.92 MPa	
Elastic modulus	47937.46 MPa	
Poissons ratio	0.2	
Open shear coefficient	0.2	
Closed shear co-efficient	0.8	
Uni-axial cracking stress	6.71 MPa	
Multi linear elastic stress-strain curve	Strain	Stress, MPa
	0.00009	4.31
	0.00018	8.60
	0.00036	17.10
	0.00072	33.33
	0.00144	60.49
	0.0019	73.13
	0.0038	91.91
Criterion	Reinforcement Steel	
Elastic modulus	2.00E+05 MPa	
Poissons ratio	0.3	
Yield Strength	590 MPa	

5.5 Modelling:

The slabs had an overall dimension of 1100 × 500×65 mm including an overhang of 25 mm beyond the support. The main reinforcement is of 8 mm diameter bars and the distribution steel was of 6 mm diameter as shown in figures 2 and 3.

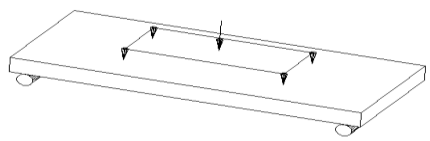


Fig -2: 3-D View of slab with Location of Loading Points

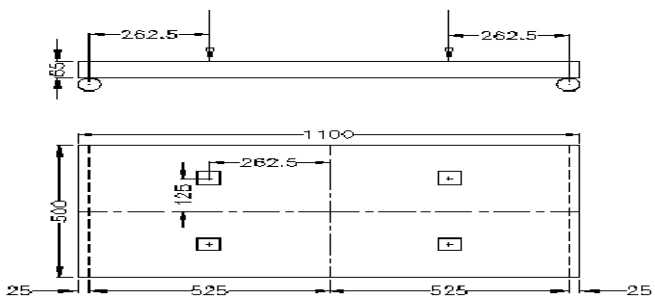


Figure -3: Plan and Cross Section of the Location of Loading Points

5.6 Meshing *:* A free mesh technique can be used for meshing but it would increase both the number of elements and the computational time. Instead, the model was meshed with two objectives: to create a sufficiently fine mesh to model the essential feature of the deformed shape, and to minimize the number of elements to reduce computation time.

The same numbers of element divisions are considered for both concrete and steel and volumes are divided in such a format, so that the two materials share the same nodes with merging or with gluing of the volumes. The end displacements of the steel element are assumed to be compatible with the displacements of the concrete element, so that perfect bond is implied. Ideally, the bond strength between the concrete and reinforced steel should be considered. However, in this study, perfect bond between materials is assumed. Figures 4 and 5 show the meshing of Concrete and Steel Respectively. [5]

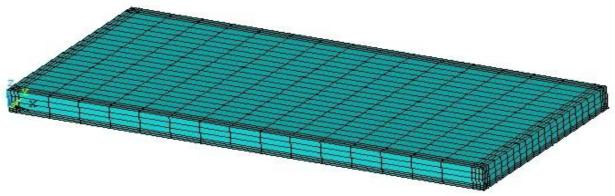


Fig -4: Meshing of the concrete slab with reinforcement.

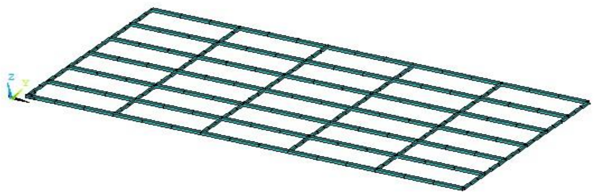


Fig -5: Meshing of reinforcement.

5.7 Boundary Conditions And Application Of Loads*:* Full section modeling is done of slab with appropriate boundary condition to have the better results with the experimental. The slabs were tested in four point bending load. The finite element models were loaded at the same locations as the full-size beams as shown in Figure 6. [5]

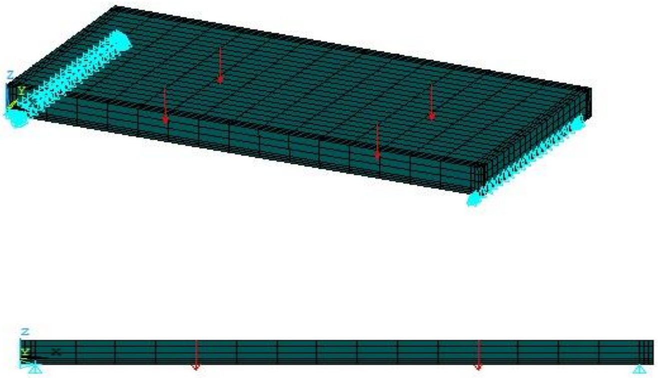


Fig -6: Loading and restrained condition.

6.0 Results and Comparisons

The load-deflection curve of the Experimental carried out by T. Geetha Kumari et al (2013) [1] and the ANSYS is illustrated below.

6.1 M70-SCF-75

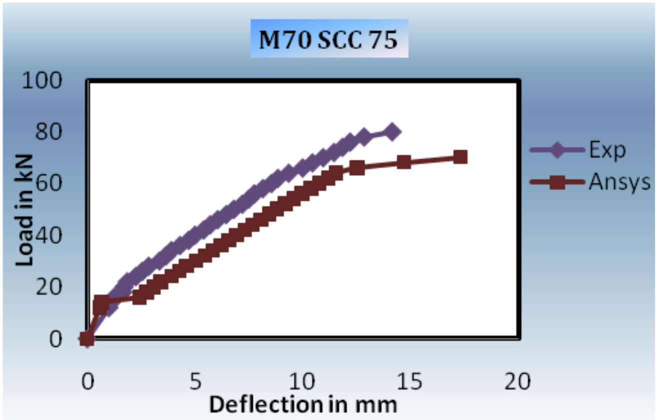


CHART -1: load-deflection behavior for M70 SCC 75

From chart 1, it is observed that load-deflection behaviour of the FE model is stiffer than the corresponding experimental model and the ultimate load is higher in the case of FE model. The ultimate load predicted by FEA is 14.28% lesser than experimental value. The maximum compressive stress in the slab is 46.68 N/mm^2 and the maximum strain is 0.013899

Figures 8 & 9 shows the shear Strain in the steel and 1st principal stress, respectively, at ultimate load for the slab. It is observed that, the maximum compressive stress in the slab is 39.05 N/mm^2 and the maximum strain is 0.013874.

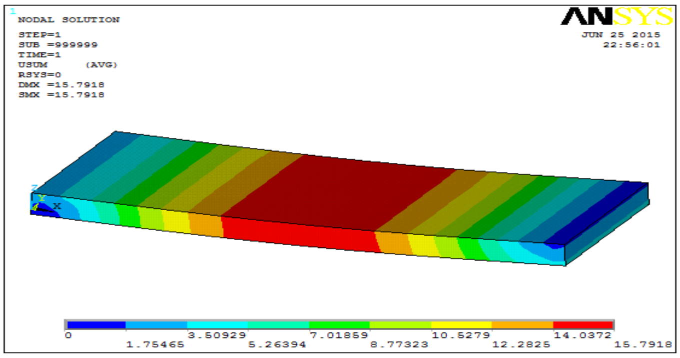


Fig -7: Ultimate deflection for M70 SCC 75

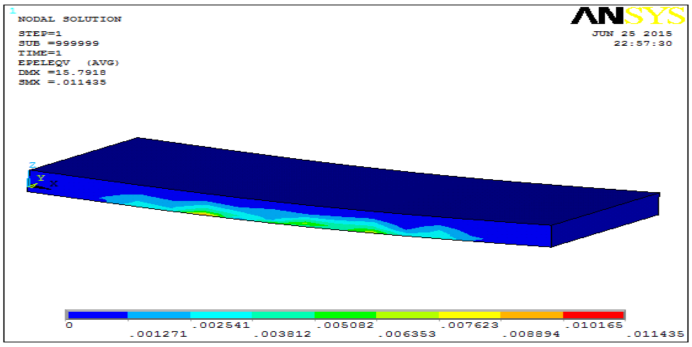


Fig -8: Ultimate strain for M70 SCC 75

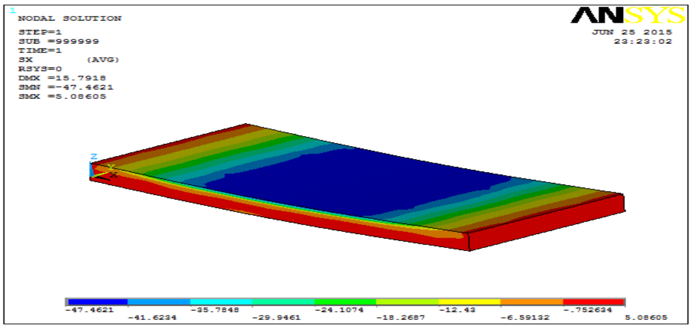


Fig -9: Ultimate stress for M70 SCC 75

6.2 M70-NCF-75

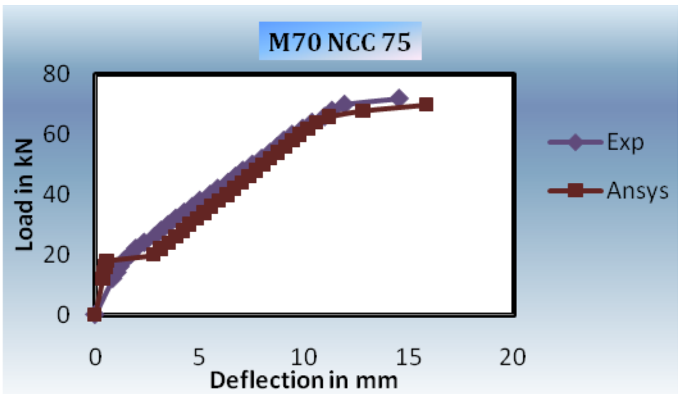


CHART -2: load-deflection behavior for M70 SCC 75

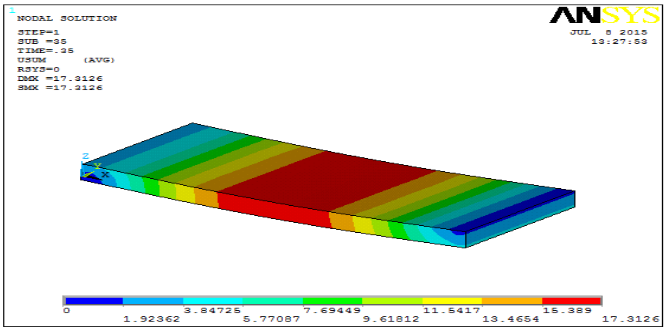


Fig -10: Ultimate deflection for M70 NCC 75

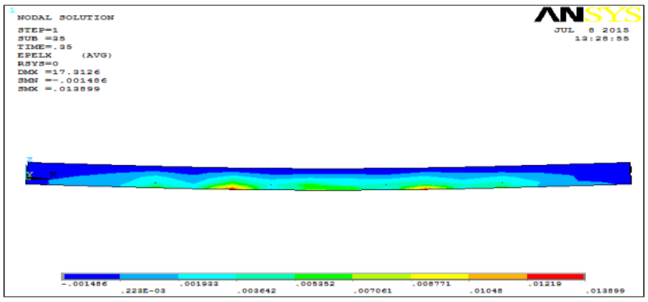


Fig -11: Ultimate srain for M70 NCC 75

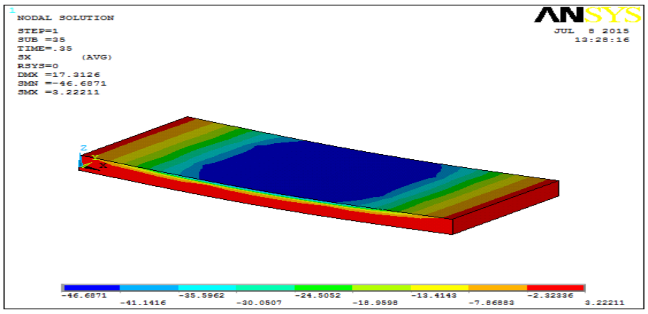


Fig -12: Ultimate stess for M70 NCC 75

From chart 2,load-deflection behaviour, it is observed that the finite element model shows stiffer behaviour than the experimental model. The finite element analysis has been performed corresponding to the experimental ultimate load and it has been observed that the FE analysis gives comparatively lesser deflection at the load intervals considered. From this, it can be said that the ultimate load with respect to finite element analysis is higher than the experimental value. The ultimate load predicted by FEA is 2.77% lesser than experimental value. The maximum compressive stress in the slab is 47.46 N/mm² and the maximum strain is 0.01143.

6.1: Ratio of experimental to FEA ultimate load and deflection.

The experimental ultimate load is 1% higher than the FEA ultimate load with a CV of 12% shown in table 4. The experimental ultimate deflection is 2% lesser than the FEA ultimate deflection with a CV of 10% shown in table 5.

Table -4: Ratio of experimental to FEA ultimate load

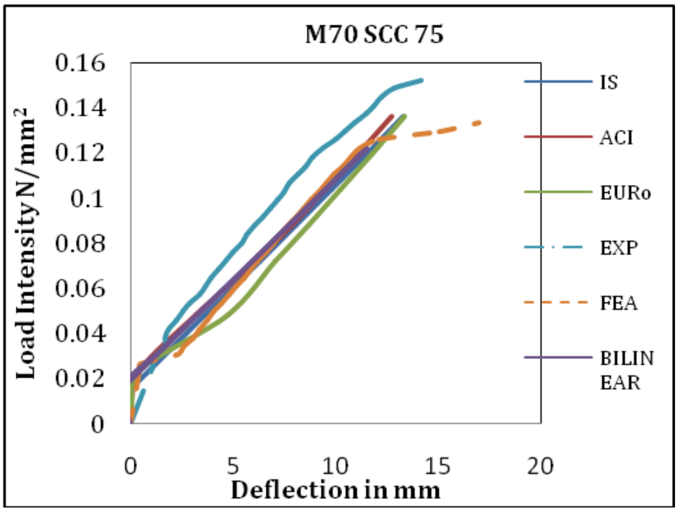
Slab No	Pu,EXP	Pu,FEA	Pu,EXP/Pu,FEA
M70-NCF-75	72	70	0.97
M70-NCF-100	52	58	1.12
M70-NCF-150	40	46	1.15
M70-NCF-200	36	40	1.11
M70-NCF-215	38	32	0.84
M70-SCF-75	80	70	0.85
M70-SCF-100	56	60	1.07
M70-SCF-150	50	50	1.00
M70-SCF-200	36	40	1.11
M70-SCF-215	38	32	0.84
		MEAN	1.01
		SD	0.12
		CV	0.12

Table -5: Ratio of experimental to FEA ultimate deflection

Slab No	δu,exp	δu,fea	δu,exp/δu,fea
M70-NCF-75	14.5	15.79	0.92
M70-NCF-100	12.25	11.03	1.11
M70-NCF-150	10.3	9.66	1.07
M70-NCF-200	10.2	9.94	1.03
M70-NCF-215	10.25	11.07	0.93
M70-SCF-75	14.15	17.31	0.82
M70-SCF-100	16.1	14.08	1.14
M70-SCF-150	13.4	14.28	0.94
M70-SCF-200	10.65	11.28	0.94
M70-SCF-215	8.55	9.95	0.86
		MEAN	0.98
		SD	0.10
		CV	0.10

6.2 Comparison of Deflections of SFR (SCC&NCC) Slabs Calculated by various Codal Equations.

The following chart 3 and 4 shows the load intensity-deflection with various codes viz. IS456, ACI318 , EN 1992, BILINEAR. FEA predicts a higher deflection than the codes. In chart 4, experimental ultimate load higher than the codes and FEA load.



HART -3: load intensity – deflection for M70 SCF-75

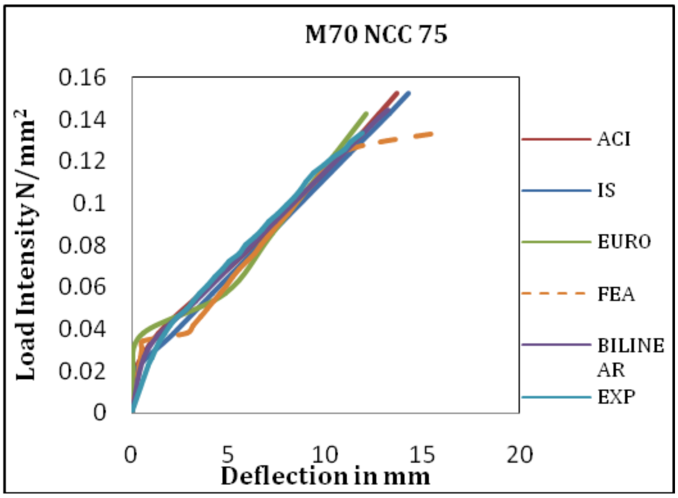


CHART -4: load intensity – deflection for M70-NCF-75

8.0 SUMMARY AND CONCLUSIONS

- The ANSYS (version 14) Finite Element Analysis has been used to understand the flexural behavior of five M70 grade concrete SFRSCC and five M70 grade concrete SFRNCC one way rectangular simply supported slabs under four point loading, and to simulate the experimental results calculated.

- The structural characteristics studied in the program are load-deflection behavior, ultimate load, ultimate stress, short term deflection and strain in the slab at working load.
- The Ultimate loads predicted by all the codes are lesser than the experimental ultimate load. The predictions are very consistent which is shown by the CV being 7.5%.
- The Ultimate load predicted by the FE model is lesser than the experimental ultimate load. The ratio of FEA ultimate load to experimental ultimate load being 0.98 and a variation of 12%.
- The Finite Element model predicts a 10% variation in the ultimate deflection studies. The ratio of deflection to experimental deflection at working load being 1.22 for IS 456:2000, 0.93 for ACI 318, 1.26 for EN 1992:2002 codes, 1.10 for Bilinear method and 1.12 for the FEA.
- Except for the ACI 318 code, all the other codes predict a higher deflection than the experimental deflection.
- It is observed that strain in steel by analytical method is almost higher by three times than the experimental strain obtained by installing strain gauge in the main steel reinforcement.
- Load increment plays a significant role in the convergence of solution. Displacement convergence method proved efficient with respect to analysis time and storage space.
- The crack patterns at the final loads from the finite element models correspond well with the observed failure modes of the experimental slabs for most of the slabs, showing that the slab fails in flexure.
- The general conclusion is that the 3D ANSYS model is able to properly simulate the non linear behavior of the steel fibre reinforced concrete slabs under flexure. The general behavior of the finite element models shows good agreement with observations and data from the experimental tests.

REFERENCES

[1] T. Geetha Kumari, C.G Puttappa, C. Shashidhar, K.U Muthu, Flexural Characteristics of SFRSCC and SFRNC one way slabs, IJERT, Eissn: 2319-1163, vol. 02 Jul-2013

[2] Amer M. Ibrahim, Nazir K. Ali & Wissam D. Salman, " Finite Element Analysis Of Reinforced Concrete Slabs With Spherical Voids ", Diyala Journal of Engineering Sciences Vol. 06, No. 04, pp. 11- 73, December 2013.

[3] S.K. Padmarajaiah, Ananth Ramaswamy, "Flexural Strength Predictions of Steel Fibre Reinforced High-Strength Concrete in Fully/Partially Pre-Stressed Beam Specimens", Cement & Concrete Composites 26 (2004), pp.275–290

[4] XU Lihua, CHI Yin, SU Jie, XIA Dongtao, "Nonlinear Finite Element Analysis of Steel Fibre Reinforced Concrete Deep Beams", Journal of Natural Sciences, Wuhan University 2008, Vol.13 No.2, 201-206.

[5] T. Geetha Kumari & Ashwin Kumar M P, Finite Element Analysis on Flexural Characteristics of SFRNC and SFRSCC one way Restrained Slabs Using ANSYS, Journal of Civil Engineering Technology and ResearchVolume 2, Number 1 (2014), pp.267-276

[6] Willam,K. J. and Warnke, E. P. (1975), "Constitutive models for the triaxial behavior of concrete", Proceedings of the International Assoc. for Bridge and Structural Engineering , vol 19, pp. 1- 30.

[7] Tavarez, F.A., (2001), "Simulation of Behavior of Composite Grid Reinforced Concrete Beams Using Explicit Finite Element Methods," Master"s Thesis, University of Wisconsin-Madison, Madison, Wisconsin

[8] I. Saifullah, M.A. Hossain, S.M.K.Uddin, M.R.A. Khan and M.A. Amin., Nonlinear Analysis of RC Beam for Different Shear Reinforcement Patterns by Finite Element Analysis, International Journal of Civil & Environmental Engineering IJCEE-IJENS Vol: 11 No: 01-2011

BIOGRAPHIES

Shivakumar V Poojar is pursuing Final year M.Tech in Structural Engineering from M S Ramaiah Institute of Technology, Bangalore.

T. Geetha Kumari is assistant professor in civil engineering Dept, M S Ramaiah institute of techmology,Bangalore. Area of interest is self compacting concrete.

