IJIRSET

ISSN (Online): 2319-8753
ISSN(Print) : 2347 - 6710

International Journal of Innovative Research in Science, Engineering and Technology

An ISO 3297: 2007 Certified Organization *Volume 4, Special Issue 5, April 2015*

International Conference On Emerging Trends in Engineering and Technology (ICETET'15)

On 13th & 14th March 2015

Organized by

Pandian Saraswathi Yadav Engineering College, Arasanoor, Sivagangai, Tamilnadu, India

Analytical Studies on Hybrid Self Compacting Concrete Deep Beam Using Fem Software

Anjitha Murali M.S[1], K.Rajesh Kumar[2]

PG Student, Department of Civil Engineering, PSNA CET, Dindigul, TamilNadu, India [1].

Assistant Professor, Department of Civil Engineering, PSNA CET, Dindigul, TamilNadu, India [2].

ABSTRACT: Analysis of deep beam using hybrid self-compacting concrete for the determination of physical properties such as deflection, strain distribution under two point loads and crack pattern for various loading values. For analysis purpose ANSYS V12.0 package is used as finite element analysis tool. Using ANSYS software flexural strains and deflection were determined at mid span and at one third distance from the support. The beam is designed using Strut and Tie method as per ACI 318-05(Appendix-A). The deep beam is modeled as simply supported beam with two point loads at its top at equal distance from the mid span. Material assumed to be homogeneous, isotropic and stress-strain relations are non-linear. Non-linear material properties were defined for both elements.It was found that by adding fibers in self-compacting concrete deep beam models improves the physical properties, especially the first crack strength and deflection at mid span.

KEYWORDS:FEA, Deep beam, Strut and Tie method, ANSYS, Crack and Deflection analysis.

I. INTRODUCTION

FEM has proven to be a versatile tool for studying the nonlinear behaviour of reinforced concrete structures.The finite element analysis calibration study included modelling a concrete beam with the dimensions and properties. Since the experimental work is uneconomicaland takes more time, it is advantageous to use finite elementanalysis to study the behaviour of reinforced concretemembers. Recentlythe existenceofcommercial computer software that can model the behaviour of reinforced concrete structural elements untilfailure made computer numerical modelling more attractive. Data obtainedfrom a finite element analysis package is not useful unlessthe necessary steps are taken to understand what ishappening within the model that is created using thesoftware. Four constant straintriangular elements were combined to form a quadrilateralelement by condensing out the central node. Cracking wasaccounted for by stopping the solution when an elementreached the tensile strength. The beams were subjected to two points loading.The objective of this paper is to perform anonlinear finite element analysis on a simply supported hybrid fiber reinforced self-compacting concrete deep beam using ANSYS software. The workinvolves studying the crack development as the appliedload is increased and also deriving the loaddeflections for SCC and hybrid fiber SCC deep beams. The outcomes of the analysis using ANSYS is compared with experimental values carried out in the laboratory.

II. DEEP BEAMS

Deep beam can be defined as a beam having a ratio of span to depth of about 2 or less. The applications of deep beams can see in transfer girder, pile cap, raft beam, wall of rectangular tank, hopper, shear wall etc.. In IS-456 (2000) Clause 29, a simply supported beam is classified as deep when the ratio of its effective span L to overall depth D is less than 2. Continuous beams are considered as deep when the ratio L/D is less than 2.5. The effective span is defined as the centre-to-centre distance between the supports or 1.15 times the clear span whichever is less.
The followings are the major differences of deep beam and beam with normal proportion based on the design assumption, as follows:

1) Two-Dimensional action i.e. deep beam is act as a plate subjected to heavy loads in its own plane.

2) Plane Section Do Not Remain Plane, this assumption of plane section remain plane, cannot be used in the design of deep beam. Thus strain distribution is no longer linear.

3) The shear deformation cannot be neglected as in the ordinary beam.

The behavior of deep beams is significantly different from that of shallow beams. The strain and stress distribution across the depth is not linear, and the variation is mainly dependent on the aspect ratio of the beam. Stresses in deep beams can be studied using the Finite Element analysis.

III. OBJECTIVES

The main objective of this work is to determine the improvement in the physical properties of RSCC deep beams when hybrid fibers are adding into the normal SCC mix. In This analysis taken two types of fibers those are glass fibers and polypropylenefibers. The fibers were added to the SCC with respect to cementations ratios. Glass fiber 0.06 % and polypropylene 0.2% has been added to the SCC mix. The goal of this work is to introduce a new model for the nonlinear FE methodusing high performance iso-parametric degenerated layered element to represent all the compositematerials such as SCC, reinforcement, and hybrid fibers. The results of the numerical program will be verified by theexperimental results.

IV. STRUT-AND-TIE METHOD DESIGN

The steps are described below (for asimply supported deep beam).

1) Define the boundaries of the D-Region and determine the boundary forces from the imposed and sectional forces. Boundary forces include the concentrated and distributed forces acting on the D-Region boundaries. Boundary forces can also come from sectional forces (moment, shear, and axial load) at the interface of D- and B-Regions.

2) Sketch a Strut-and-Tie Model and solve for the truss member forces.

3) evaluate the dimensions of the Struts and Nodes such that the capacity of all Struts and Nodes is sufficient to carry the truss member forces.

4) Provide distributed reinforcement to ensure ductile behavior of the D-Region. Since equilibrium of the truss with the boundary forces must be satisfied (step 2) and stresses everywhere must be below the limits (step 3 and 4), one can see that the Strut-and-Tie Method is a lower-bound (static or equilibrium)method of limit analysis.

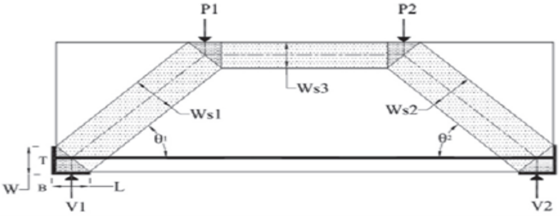

Figure1Schematic representation of STM

V. NON LINEAR FINITE ELEMENT ANALYSIS

The finite element analysis calibration study included modeling a concrete beam with the dimensions and properties. To create the finite element model in ANSYS 12.0 there are multiple tasks that have to be completed for the model to run properly. Models can be created using command prompt line input or the Graphical User Interface. For this model, the graphical user interface was utilized to create the model. This section describes the different tasks and entries to be used to create the finite element calibration model.

5.1. Element Types

The element type for this model is shown in Table 1.

Table1. Element Types for Working Model

Material Type Element	ANSYS
Concrete	Solid 65
Steel reinforcement	Link8

A Solid65 element was used to model the concrete. This element has eight nodes with three degrees of freedom at each node translations in the nodal x, y, and z directions. The element is capable of plastic deformation, cracking in three orthogonal directions, and crushing. A schematic of the element was shown in Fig.2.

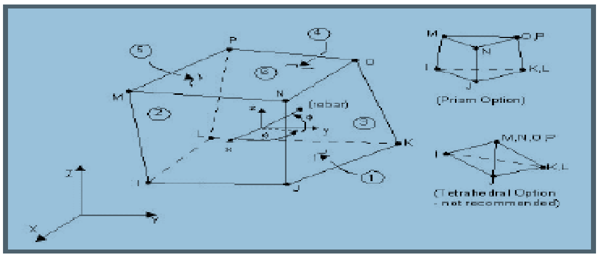

Figure2. Solid 65 elements

A Link8 element used to model RC. This element is a 3D spar element and it has two nodes with three degrees of freedom in the nodal x, y, and z directions. This element is capable of plastic deformation and shown in the Fig.3.

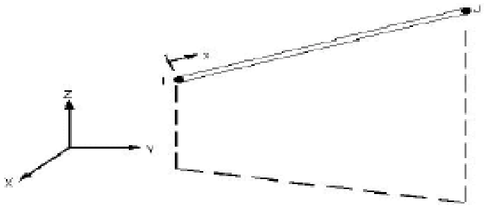

Figure 3 Link 8 elements

5.2. Real Constants

Real Constant Set 1 was used for the Solid65 element. It requires real constants for rebar assuming a smeared model. Values can be entered for Material Number, Volume Ratio, and Orientation Angles. The volume ratio refers to the ratio of steel to concrete in the element. The reinforcement has uniaxial stiffness and the directional orientations were defined by the user. In the present study the beam was modeled using discrete reinforcement. Therefore, a value of zero was entered for all real constants,

Table 2. Real Constants

Real Con-stant	Elem-ent Type	Real Const-ant for Rebar 1	Real Const-ant for Rebar 1	Real Const-ant for Rebar 1
1	Solid 65	0	0	0
2	Link 8	112	0	0
3	Link 8	78.5	0	0

1.3. Material Properties

5.3.1 Concrete

Concrete is a brittle material and has different behavior in compression and tension. Incompression, the stress – strain curve relationship for concrete is described by multi- linearisotropic curve, linear elastic up to about 40 percent of the maximum compressive strength(fc'). In flexure, the stress – strain curve for concrete was assumed to belinearly elastic up to the ultimate tensile strength (ft = 0.1 fc').

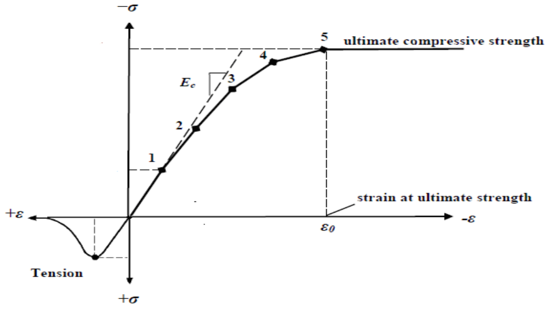

Figure4 Simplified Uniaxial Compressive and Tensile Stress-Strain Curves for Concrete.

5.3.2. FEM Input Data

For SCC, ANSYS requires input data for material properties as follows:
1)Elastic modulus (Ec = 5431.24 fc').
2) Ultimate uniaxial compressive strength (fc').
3) Ultimate uniaxial tensile strength (ft, assumed to be ft =0.1 fc') .
4) Poisson's ratio (ν) assumed to be ν =0.24.
5) Shear transfer coefficient (β0) for open cracks and (βc) for closed cracks, representing conditions of crack face for determining the amount of shear transfer across the crack were used. The range from (0, 0) to (1.0). In present study, (β0) was assumed to be (0.2) while (βc)was (0.6).
6) Compressive uniaxial stress – strain relationship for concrete.
7) Concrete density = 2400 kg/m3.
The steel reinforcement added both top and bottom for all the beams in ANSYS to provide even stress distribution over the top and bottom areas.
An elastic modulus equal to 200,000 MPa and Poisson's ratio of 0.3 were used for the steel bars.

1.4. Meshing

To obtain good results from the Solid65 element, the use of a rectangular mesh was recommended. Therefore, the mesh was set up such that square or rectangular elements were created. The meshing of the reinforcement was a special case compared to the volumes. The meshing and reinforcement configuration of the beam were shown in Fig.5

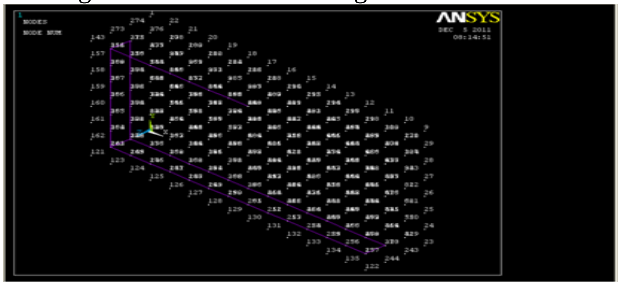

Figure 5 Mesh and Rein forcement Configuration

1.5. Modeling

The model is 1000 mm long with a cross section of 100 mm X 500 mm. The Finite Element beam model is shown in Fig.6. The dimensions for the concrete volume are shown in Table.3.

Table 3 Dimensions for Concrete

ANSYS	**Concrete(mm)**
X1,X2,Xcoordinates	0, 1000
Y1,Y2,Y-coordinates	0, 500
Z1,Z2,Z-coordinates	0, 100

5.6. Loads and Boundary Conditions

Displacement boundary conditions were needed to constraint the model to get a unique solution. To ensure that the model acts the same way as the experimental beam boundary conditions need to be applied at points of symmetry, and where the supports and loading exist. The support was modeled as a simply support at both ends. Nodes on the plate were given constraint in all directions, applied as constant values of zero. The loading and boundary conditions of the beam were shown in Fig.7.

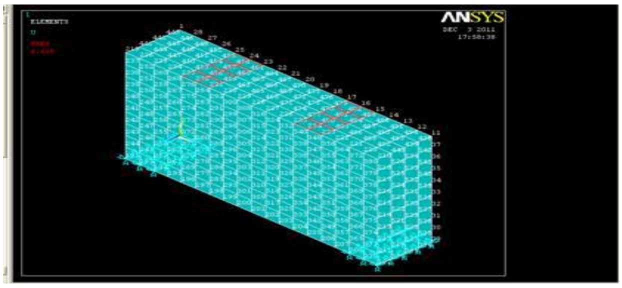

Figure 6 Loading and boundary conditions

1.6. Crack Patterns

The crack patterns of different beams using ANSYS 12.0 Software were shown in Fig.6 (i) to Fig.6 (iii).

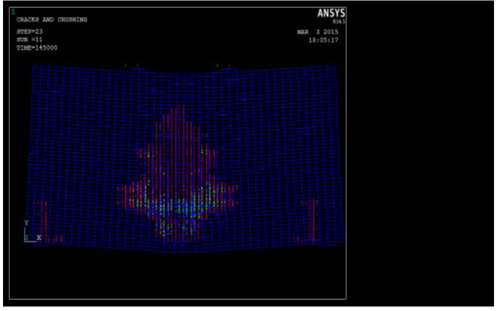

Figure 6 (i) forSCC**Figure**

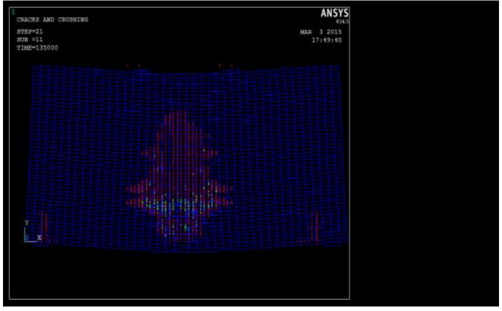

Figure 6 (ii) for SCC+ Glass fiber (0.06%)

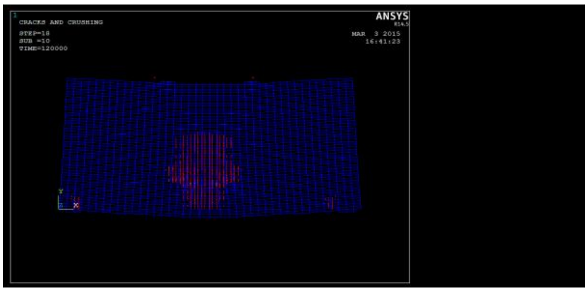

Figure 6(iii)forSCC+Glassfiber (0.06%) +Polypropylene (0.2%)

5.8. Deflections at the Mid-section

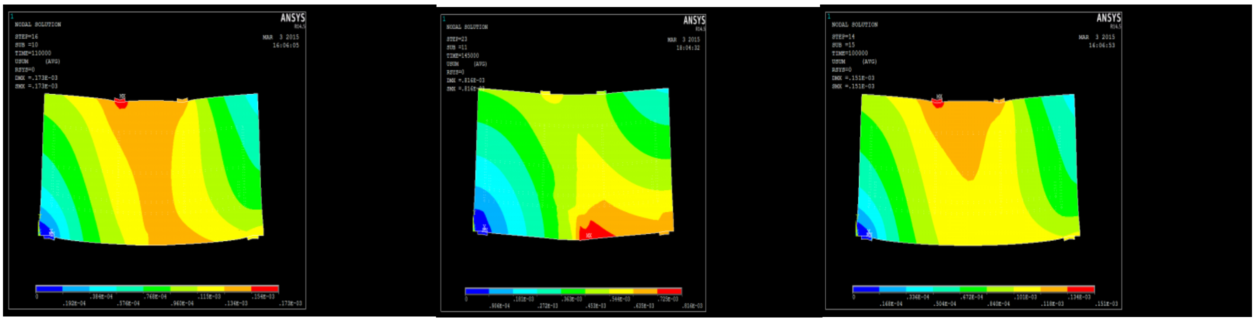

Figure 7(i)for control SCC ,**(ii)** For SCC+ Glass fibers (0.06%), **(iii)**For SCC+ Glass fiber (0.06%) +Polypropylene (0.2%)

II. RESULTS AND DISCUSSION

Deep beams with control SCC and HFRSCC wereanalyzed and crack pattern and deflections were measured at mid-section of the beam by using FEM software ANSYS. The fibers used for the study were Glass fiber and polypropylene fibers. The beams were designed for two points loading of 200 kN each at 1/3 of span. At a load increment of 30kN, cracks at midsection and deflections were measured. The average initial cracking load for the beams was found to be 11 tone, 18 tone, 21 tonne respectively. At these loads it was observed that minor cracks were developed in shear span region. Also minor flexural cracks at mid span were observed. Loading was continued beyond this point. The failure loads observed were 190 kN, 200 kN, 260 kN respectively.

III. CONCLUSIONS

Based on finite element analysis and discussion above, the following conclusions can be drawn:

1) ANSYS software is capable of predicting the actual crack pattern, deflections and stress- strain relationships of both normal and hybrid fiber SCC simply supported deep beam specimens subjected to two points loading.

2) The use of fibers improves the mechanical properties of concrete, especially the first crack strength and it reduces the deflection at mid span.

3)From the above results it was found that (0.06 % Glass fiber + 0.1%PP) HFSCC deep beam reduced the strain in concrete as well as in steel considerably when compared to other mixes.

4) FEA tool ANSYS can be used for analysis of a beam subjected to complex load such as torsional moments, combined loading etc also with complex problems with Nonlinear Geometry and Support conditions effectively compared to analytical methods.

REFERENCES

[1] Mr.Amit A. Kusanale, Prof. S. B. Kadam, "Analysis and Design of R.C. Deep Beam by Finite Element Method" ISSN 2349-4395 (Print) & ISSN 2349-4409

[2] Kale Shrikant M. Prof.Patil.S.S. Dr.Niranjan B.R. "Analysis of Deep Beam Using Cast Software and Compression of Analytical Strain with Experimental Strain Results", International Journal Of Computational Engineering Research (ijceronline.com) Vol. 2 Issue. 8

[3] NiranjanB.R ,Patil S.S, "Analysis of R.C Deep Beam by Finite Element Method", Vol. 2, Issue. 6, Nov.-Dec. 2012.

[4] NiranjanB.R ,Patil S.S, "Analysis and Design of deep beam by using Strut and Tie Method", ISSN: 2278-1684 Volume 3, Issue 4 (Sep-Oct. 2012).

[5] M. A. Musmar1, M. I. Rjoub2 and M. A. Abdel, "Nonlinear Finite Element Analysis OF Shallow ReinforcedConcrete Beams Using Solid65 Element" ARPN Journal of Engineering and Applied Sciences, ISSN 1819-6608

[6] Mohammed Sh. Mahmood and Amer M. Ibrahim, "Finite Element Modeling of Reinforced Concrete Beams Strengthened with FRP Laminates", European Journal of Scientific Research ISSN 1450-216X Vol.30 No.4 (2009), pp.526-541.

[7] Muhammad Abdur Rashid And AhsanulKabir, "Behaviour Of Reinforced Concrete Deep Beam Under Uniform Loading", Journal Of Civil Engineering, The Institution Of Engineers, Bangladesh, Vol CE 24,No-2,1996.

[8] M Chandrasekhar1, M V Seshagiri Rao2, Maganti Janardhana3. "Analytical modelling on the Stress-Strain BehaviorOF Hybrid Fibre Reinforced Self Compacting" International Journal of Mathematical Sciences, Technology and Humanities 2 (2011) 19 – 24