International Journal of Application or Innovation in Engineering & Management (IJAIEM)
Web Site: www.ijaiem.org Email: editor@ijaiem.org
Volume 5, Issue 5, May 2016 **ISSN 2319 - 4847**

Finite Element Analysis Of Composite Element For FRP Reinforced Concrete Slab By Using ANSYS

T.Subramani[1], K.Balamurugan[2]

[1]Professor & Dean, Department of Civil Engineering, VMKV Engg. College, Vinayaka Missions University, Salem, Tamil Nadu, India.

[2]PG Student of Structural Engineering, Department of Civil Engineering, VMKV Engg. College, Vinayaka Missions University, Salem, Tamil Nadu, India.

ABSTRACT

In this paper, a nonlinear finite-element analyses have been carried out to investigate the behavior up to failure of simply supported composite steel-concrete beams with external pre stressing, in which a concrete slab is connected together with steel I-beam by means of headed stud shear connectors, subjected to symmetrically static loading. ANSYS computer program has been used to analyze the three dimensional model. This covers: load deflection behavior, strain in concrete, strain in steel beam and failure modes. The nonlinear material and geometrical analysis based on Incremental-Iterative load method, is adopted. Three models have been analyzed to verify its capability and efficiency. The results obtained by finite element solutions have shown good agreement with experimental results. The present study deals with the finite element modeling of control RC slabs and strengthened slabs with the help of ANSYS.ANSYS is software that is based on FEA method in which modeling of RC structures is done. In this study, a control slab was modeled and the results were analyzed and then strengthened slabs were modeled and analyzed. The results of the control and strengthened slabs were compared with the experimental results. It was observed that the results of the strengthened slabs are in close agreement with the experimental results.

Keywords*:* Finite Element Analysis, Composite Element, FRP Reinforced Concrete Slab, Ansys

1 INTRODUCTION

Reinforced concrete structures are largely employed in engineering practice in a variety of situations and applications. In most cases these structures are designed following simplified procedures based on experimental data. Although traditional empirical methods remain adequate for ordinary design of reinforced concrete members, the wide dissemination of computers and the development of the finite element method have provided means for analysis of much more complex systems in a much more realistic way. The main obstacle to finite element analysis of reinforced concrete structures is the difficulty in characterizing the material properties. Much effort has been spent in search of a realistic model to predict the behavior of reinforced concrete structures. Due mainly to the complexity of the composite nature of the material, proper modeling of such structures is a challenging task. Many times specifically developed computer programs are used in finite element analyses of reinforced concrete structures. However, many general purpose codes commercially available provide some kind of material model intended to be employed in the analysis of concrete structures. Once general purpose computer codes are supposed to be more likely used as a design tool, an investigation of the capabilities of such codes seems to be of much concern. The objective of this paper is to discuss the possibilities of different reinforced concrete models in practical use. It reports the results of some analyses performed using the reinforced concrete model of the general purpose finite element code Ansys. A series of analysis of the same structure has been performed, exploring different aspects of material modeling.

In recent years, numbers of studies have been made in designing the multi storey steel frames and the development in this type is still continued until now. Many researchers try to present the easier and simple way for example comes out with software such as ANSYS, Visual Basic and so on for determination of the section sizes which meet the principals and limitations as stated. Apart from that, several methods were introduced to fulfill the criteria which are Wind-Moment Method and Merchant-Rankin Method. However, the effectiveness of these methods is still in question because the second order effect due to sway does not taking into account though sway deflection give the major effect in the construction of frames. It is irrelevant to attempt final design that exceeds the sway limitation because sway does not affect the frame structure. Therefore, a simplified method which meets the sway deflection criterion is introduced and the method is called as Direct Design Method.

The use of external pre stressing as a means of strengthening or rehabilitating existing bridges has been used in many countries since the 1950s. It has been found to provide an efficient and economical solution for a wide range of bridge types and conditions. The technique is growing in popularity because of the speed of installation and the minimal disruption to traffic flow. The principle pre stressing, is the application of an axial load combined with a hogging bending moment to increase the flexural capacity of a beam and improve the cracking performance. It can also have a beneficial effect on shear capacity. Composite steel - concrete beams prestressed with high strength external tendons have demonstrated many advantages as compared with plain composite beams: Increase in ultimate moment capacity of structure, Enlarge the range of elastic behavior before yielding for the structure with the introduction of internal stresses. The stresses can then oppose the moment generated by the loading. The amount of structural steel used in construction, based on yield strength alone, can be significantly reduced by the use of high-strength tendons, thereby reducing the cost of construction. A composite beam can be prestressed, using a jack, by the tensioning high-strength tendons connected at both ends to brackets or anchorages that are fixed to the composite beam. Prestressing a composite beam can introduce internal stresses into the member cross sections that can be defined for different purposes.

Such induced stresses can then counteract the external loads applied on the structure. Prestressing can be carried out for simple-span or continuous-span composite beams. In the positive moment region, the steel beam is usually prestressed before the concrete is cast because the negative moment induced by prestressing may be used to counteract the positive moments caused by the concrete's self weight. In the negative moment region, the steel beam and concrete deck can also be prestressed either separately or jointly along the top flange before or after casting of the deck. Many different methods are suitable for repair and strengthening, such as additional reinforcement cover by concrete, external steel plate bonding, using of steel material, etc. Nowadays, steel frame system with beams and columns is become the conventional building structure in construction world. Structural design and structural analysis are both of the criteria needed to create a structure that safely accomplish its function in order to produce structures in a stability condition. In civil engineering field, steel is widely used in building construction. Its popularity may be due to the various sizes and the shape of steel sections to be used for various type of structures such as small and the simple buildings as well as complicated infrastructures construction. Generally, steel frame not is only design to sustain vertical loads but also able to resist lateral loads. In this research, Finite Element Method (FEM) models were used to stimulate the behavior of the steel frame structure's ability using ANSYS 12 program. The ANSYS was founded in 1970, develops and globally markets engineering simulation software and technologies widely used by engineers and designers across a broad spectrum of industries like civil and mechanical engineering. This program is capable of predicting deflection and stress in concrete concepts and also includes model's constitutive laws for concrete material, based on smeared crack concepts and for high strength composite materials.

2. OBJECTIVE AND SCOPE

2.1 Objective

The objectives of this research are following:

- To present software analysis with analytical results.
- Generate a new equation for frame structure by proving the parameters in ANSYS Parametric Design Language (APDL) using ANSYS program.
- To study the response and behavior of composite elements through a series of analysis under different load and dimension case.

2.2 Scope

These researches are mainly focused on the design of steel frame structures by using Indian Standard and use the data to generate the new equation by proving the parameters in ANSYS. In order to achieve the objectives of the researches, there are few researches scope is necessary to be followed. Study the types of steel design frames and the characteristics of the structure.

Explore the ANSYS program by learning how to use the programming by using tutorial from internet. Practicing of tutorials can helps to solve problem when running the real models. Model of columns are designed using ANSYS parametric design language (APDL). Finite Element Analysis is using to calculate the parameters which the parameters must success to get new equation as mention in objectives.

3. METHODOLOGY

The structures are analyzed using different software and different technologies available. But here in our project composite element structural member is analyzed using the software called ANSYS. Normally as in all other analysis software the structure is created and property are allotted to the structure that u had created. Then the load is applied to the structural member as required. In all analysis software the loads are applied on the top of the member of structure created or in the top of nodes that connect the R.C.C member you create where we get the result like bending moment and shear fore of the structure member created and it is the common result we get in all types of analysis software. Here in ANSYS software there is a type of analysis method called FEA (Finite Element Analysis) which is used to analysis the structural framed member you created. Finite element analysis (FEA) is the modeling of products and systems in a virtual environment, for the purpose of finding and solving potential (or existing) structural or performance issues. FEA is the practical application of the finite element method (FEM), which is used by engineers and scientist to mathematically model and numerically solve very complex structural, fluid, and metaphysics problems. FEA software can be utilized in a wide range of industries.

3.1 Meshing

The important requirement of the FEM is the need to split the solution domain (model geometry) into simply shaped sub domains called ‘finite elements’. This is a discretization process commonly called meshing and element are called finite because of their finite, rather than infinitesimally small size having infinite numbers of degree of freedom. Thus the continuous model with an infinite number of degrees of freedom (DOF) is approximately by a discredited FE model with a finite DOF. This allows the reasonably simple polynomial functions to be used to approximate the field variables in each element. Meshing the model geometry also discredited the original continuous.

4 ABOUT SOFTWARE (ANSYS)

Ansys is analysis software which is used mostly for analysis purpose of structure and objects. All over it is not only for the analysis process of building structure in civil it mostly used in the where area can be utilized in a wide range of industries, but is most commonly used in the aeronautical, biomechanical and automotive industries.
Using this ansys software different range of properties to be defined such as.

- Section areas
- Moments of inertia
- Tensional constant
- Plate thickness
- Bending stiffness
- Transverse shear

4.1 Types Of Analysis Include

- Linear statics: linear analysis with applied loads and constraints that are static
- Nonlinear statics and dynamics: effects due to contact (where one part of the model comes into contact with another), nonlinear material definitions (plasticity, elasticity, etc.) and large displacement (strains that exceed small displacement theory that limits a linear analysis approach)
- Normal modes: natural frequencies of vibration
- Dynamic response: loads or motions that vary with time and frequency
- Buckling: critical loads at which a structure becomes unstable
- Heat transfer: conduction, radiation and phase change

The Finite Element Analysis (FEA) is a numerical method for solving problems of engineering and mathematical physics. Useful for problems with complicated geometries, loadings, and material properties where analytical solutions cannot be obtained. Finite element analysis (FEA) has become commonplace in recent years, and is now the basis of a multibillion dollar per year industry. Numerical solutions to even very complicated stress problems can now be obtained routinely using FEA, and the method is so important that even introductory treatments of Mechanics of Materials such as these modules should outline its principal features. In spite of the great power of FEA, the disadvantages of computer solutions must be kept in mind when using this and similar methods: they do not necessarily reveal how the stresses are influenced by important problem variables such as materials properties and geometrical features, and errors in input data can produce wildly incorrect results that may be overlooked by the analyst. In mathematics, the finite element method (FEM) is a numerical technique for finding approximate solutions to boundary value problems for partial differential equations. It uses subdivision of a whole problem domain into simpler parts, called finite elements, and variation methods from the calculus of variations to solve the problem by minimizing an associated error function.

5 ANALYTICAL MODEL

In this paper, prestressed composite steel-concrete beams are analyzed; the dimensions of the beams are illustrated in Figure 1 to Figure 2. Three beams, VS-1, VS-2, and VS-3 were analyzed in this study are selected to investigate the behavior of externally prestressed composite steel-concrete beams, by develop a general analytical approach to predict the ultimate flexural response. The accuracy and validity of the finite element models is determined by ensuring that failure modes are correct, the ultimate load is reasonably predicted in comparison with the available experimental investigations. VS-1 is tested by Saadatmanesh et al. The beam is simply supported composite steel-concrete beam, having prestressing bars along the bottom flange and which is subjected to positive bending moment as shown in Figure.1. VS-2, and VS-3 were tested by Ayyub et al. VS-2 is simply supported consisted of a concrete slab, a steel beam, and two prestressing tendons as shown in Figure 2 . The prestressing bars were anchored at the two ends of the beam 30 mm above the bottom (tension) flange and were extended on both sides of the web along the full length of the beam

Beam VS-3 is similar to beam VS-2 and was prestressed with draped tendon profile, as shown in Figure 3. The tendons were anchored at both ends of the centroid axis of the composite section, 32 mm below the top flange and were positioned between the loading points 30 mm above the bottom flange.

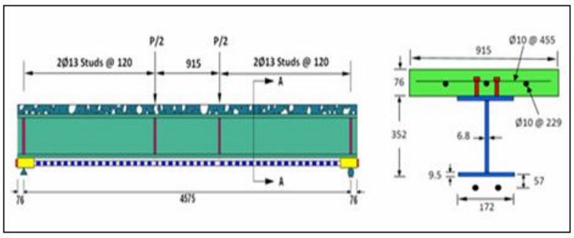

Figure.1 Details of Beam (VS-1)

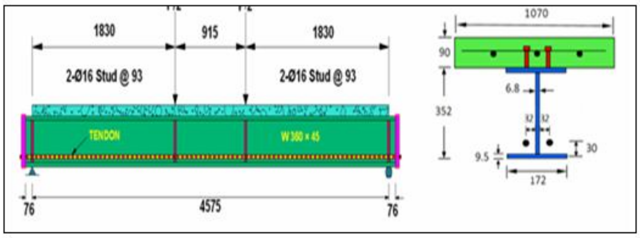

Figure.2 Details of Beam (VS-2)

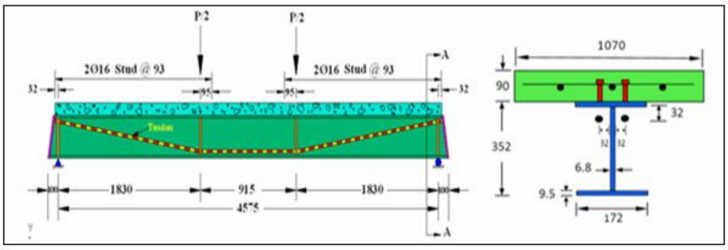

Figure. 3 Details of Beam (VS-3)

6 ANALYSIS RESULTS

6.1 Load-Deflection Curve

The load-midspan deflection curves of the prestressed composite steel-concrete beams obtained from the finite element analysis was compared with corresponding experimental data as shown in figures from (10.1) to (10.3). In general, it can be noted from the load deflection curves that the finite element analyses are agree well with the experimental results throughout the entire range of behavior.

6.2 Ultimate Load At Failure

The comparison between the ultimate loads of the experimental (tested) beams, (Pu) EXP., and the final loads from the finite element models, (Pu) FEM. The final loads for the finite element models are the last applied load steps before the solution starts to diverge due to numerous cracks and large deflections Since the ordinary steel bars and prestressing cable are slender, they can be assumed to transmit axial force only. Modeling of ordinary steel and prestressed steel in finite element is much simpler. The stress-strain relationship for ordinary reinforcing steel and prestressing tendons can be represented. The numerical model predicts ultimate loads of (656.7 kN), (723 kN) and (769.9kN) for beams (VS-1), (VS-2) and (VS-3) (Figure.4, Figure5. & Figure.6) respectively and captures well the nonlinear load deflection response of the beams up to failure as mentioned before. In comparison with the experimental values, the numerical models show (2.7%), (1.68%) and (0.66%) increase in ultimate loads for the beams (VS-1), (VS-2) and (VS-3) respectively. (Table.1)

International Journal of Application or Innovation in Engineering & Management (IJAIEM)
Web Site: www.ijaiem.org Email: editor@ijaiem.org
Volume 5, Issue 5, May 2016 ISSN 2319 - 4847

Table. 1 Ultimate loads from experimental test and Finite element analysis

Analytic and test beam	Ultimate load (kN)		$\frac{(Pu)_{FEM}}{(Pu)_{EXP}}$	Difference Ratio
	$(Pu)_{FEM}$	$(Pu)_{EXP.}$		
VS-1	656.7	641	1.02	2.4%
VS-2	723	711	1.01	1.68%
VS-3	769.9	775	0.993	0.66%

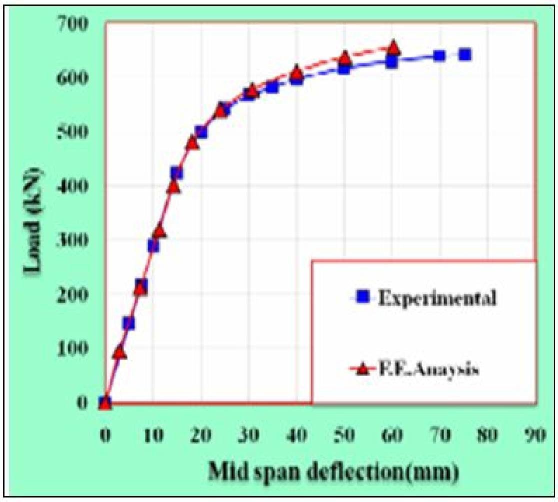

Figure.4 Load-Deflection Curve for Beam VS-1

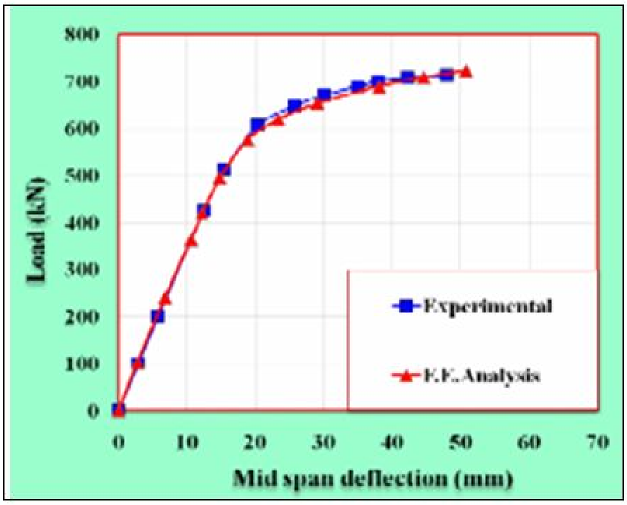

Figure.5. Load-Deflection Curve for Beam VS-2

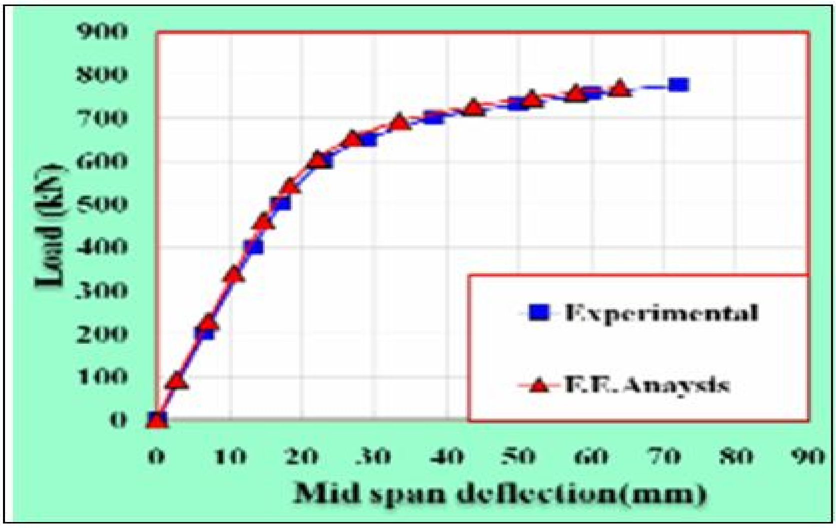

Figure.6 Load-Deflection Curve for Beam VS-3

6.3 Deflected Shape

Deflections (Vertical displacements) were measured at mid-span at the center of the bottom face of the beams, in y-direction (Uy). A deflected shape for two load steps, camber arising from the effect of prestressing force in the external tendon; and deflected shape due to externally applied loads for all beams are shown in Figures below(7 to 12).

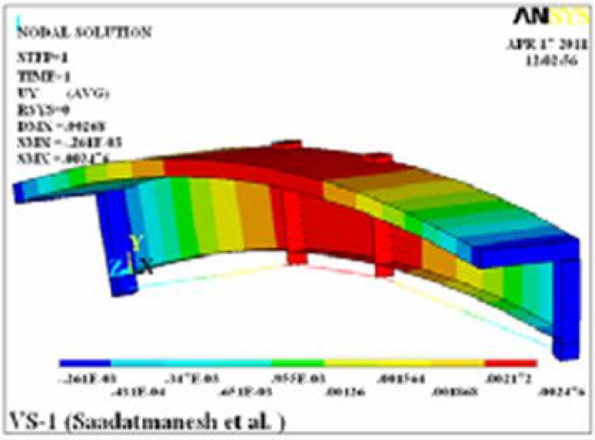

Figure.7 Initial Camber of the Beam VS-1 With no Applied Concentrated Load

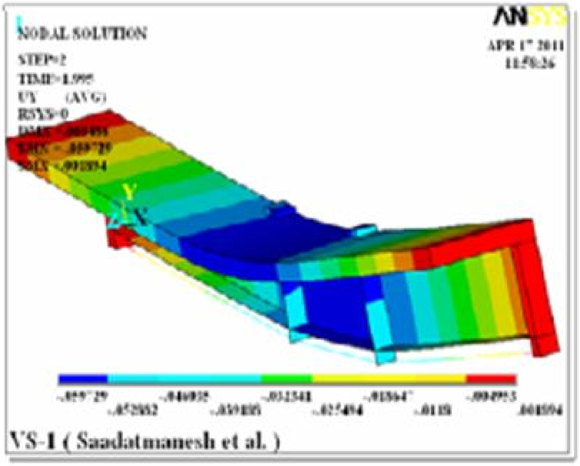

Figure.8 Deflected Shapes for Beam VS-1 at Load

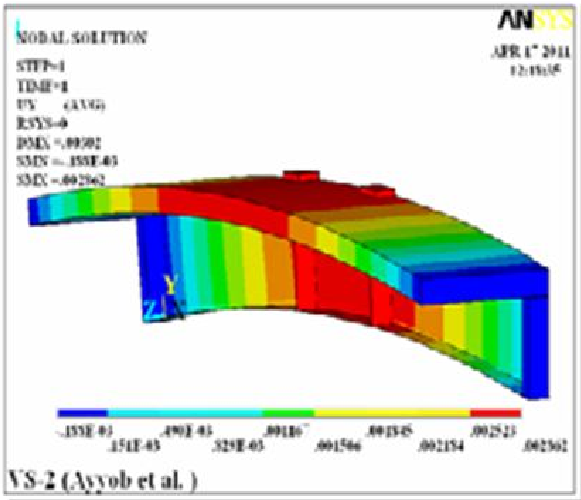

Figure.9 Initial Camber of the Beam VS-2 With no Applied Concentrated Load

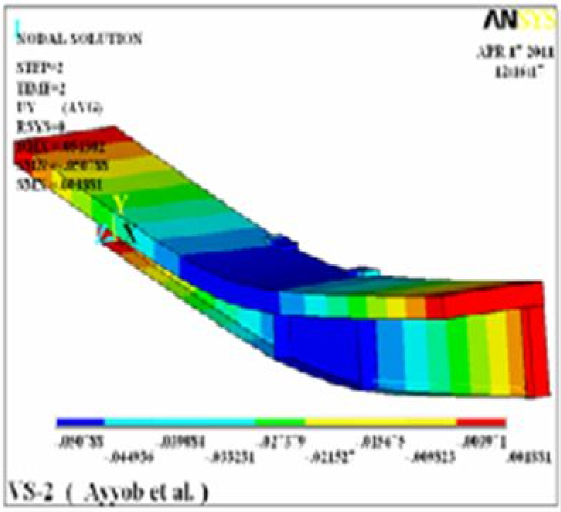

Figure.10 Deflected Shape for Beam VS-2 at Load

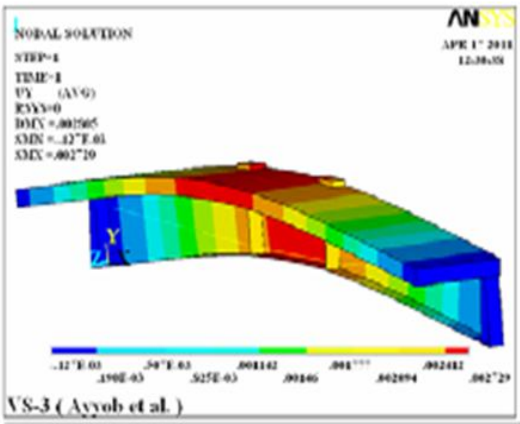

Figure.11 Initial Camber of the Beam VS-3 with no Applied Concentrated Load

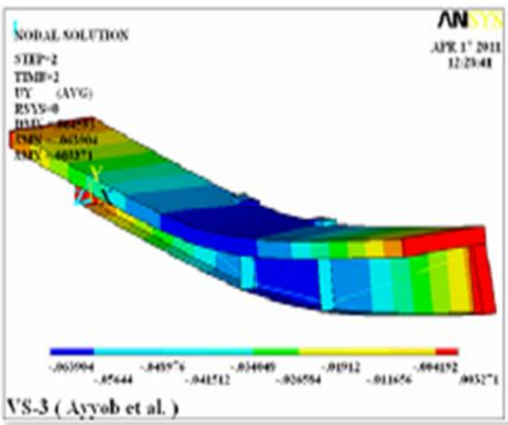

Figure. 12 Deflected Shape for Beam VS-3 at Load

6.4 Loads-Strain Curves

Figur.13shows the load versus extreme compression fiber strain response of the concrete slab observed from experimental and analytical study (computer program) of (VS-1) beam. It is shown that the analytical compressive

International Journal of Application or Innovation in Engineering & Management (IJAIEM)
Web Site: www.ijaiem.org Email: editor@ijaiem.org
Volume 5, Issue 5, May 2016 ISSN 2319 - 4847

strains and corresponding experimental strains are very close throughout the loading up to the yielding then the analytical strain become slightly higher than the experimental strain up to ultimate load.

Figure.14 shows comparison between experimental results and numerical results that are obtained by ANSYS computer program for load versus strain in the prestressed bars. The strain in the bar was recorded up to yielding. The curves were offset from the origin of the coordinate system by the initial prestressing strains. This Figure shows the increase in strain in the prestressing tendons with loading. Both the tendons were prestrained to an initial strain of 2700 micro-strain reaching a load of 98 kN in each of the tendons.

Figures 15 and 16 shows the load versus strain curves for the bottom and top steel flange observed from experimental and analytical study. All the curves were offset from the origin by the strain induced during the initial prestressing. The strain in the bottom flange was recorded up to yielding. Higher strain values were recorded in the bottom flange than the top flange.

Both top and bottom steel flanges curves started in compression and moved towards the right side of the curve as the external load started to neutralize their initial pre compression due to prestressing. At a load of about 98 kN, the initial pre compression was reduced to almost zero as the load was increased. It was observed that the behavior still linear up to a load of 455 kN where the bottom flange of steel beam is yielded.

Figures from 17 to 20 shows comparison between experimental results and numerical results for load versus strain in extreme fiber of the top and bottom concrete slab, top and bottom flange of steel beam respectively for (VS-2) beam. It is shown that the analytical compressive strains and corresponding experimental strains are very close throughout the loading up to ultimate load. Higher strain values were recorded in the bottom flange than the top flange.

Figure 21 shows the load versus extreme compression fiber strain response of the concrete slab of (VS-3) beam. It is shown that the analytical compressive strains and corresponding experimental strains are very close. In Figures.22 to 24 shows the midspan load versus strain for bottom concrete, top, and bottom flange of the (VS-3) beam. Higher strain values were recorded in the bottom flange than the top flange.

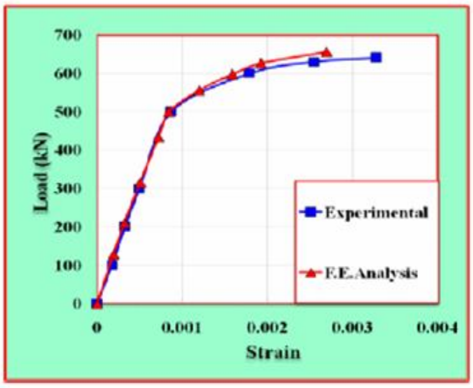

Figure.13 Load-strain curve for extreme compression concrete fiber of vs-1

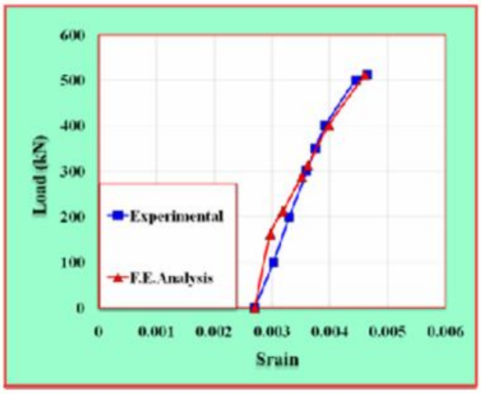

Figure.14 Load-strain curve for prestressed bars for VS-1

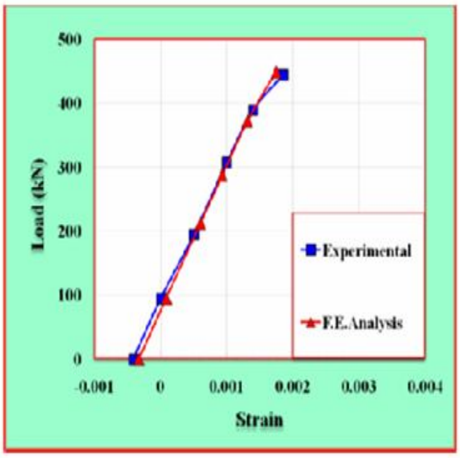

Figure.15 Load-strain curve for bottom flange for VS-1

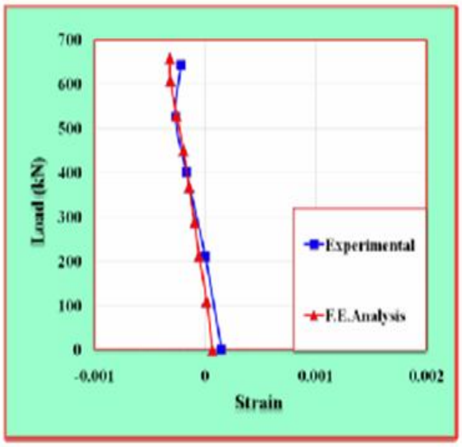

Figure.16 Load-strain curve for top flange for VS-1

International Journal of Application or Innovation in Engineering & Management (IJAIEM)
Web Site: www.ijaiem.org Email: editor@ijaiem.org
Volume 5, Issue 5, May 2016 **ISSN 2319 - 4847**

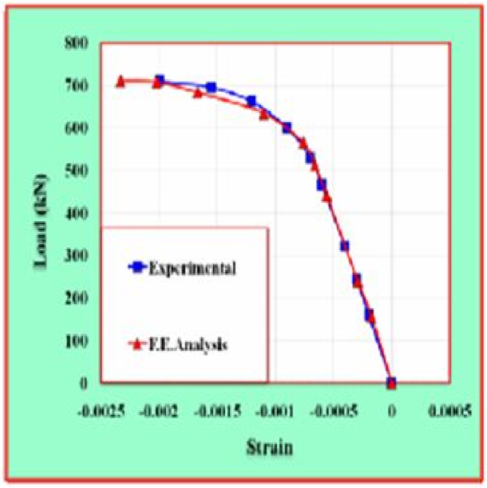

Figure 17 Load-strain curve for extreme compression concrete fiber of VS-2.

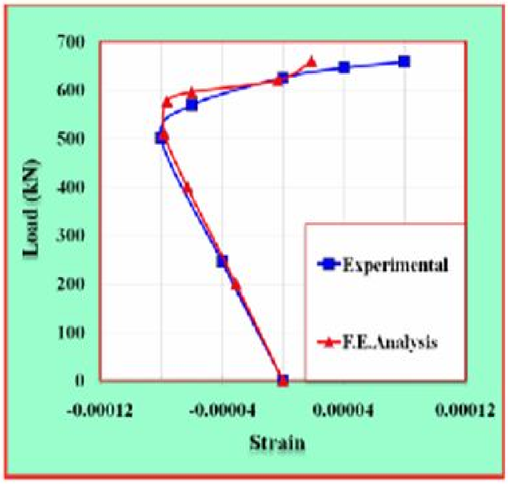

Figure 18 Load-strain curve for bottom concrete fiber for VS-2.

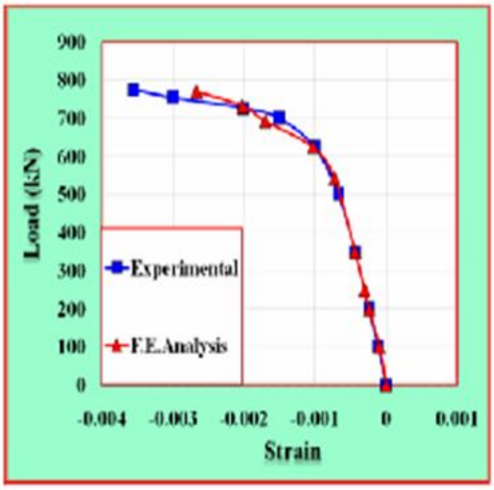

Figure.21 Load-strain curve for extreme compression concrete fiber for VS-3

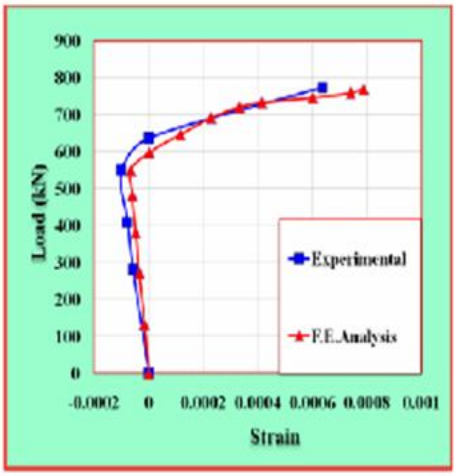

Figure.22 Load-strain curve for bottom concrete for VS-3

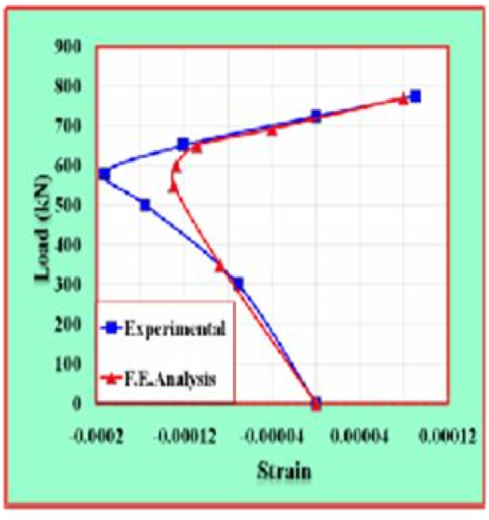

Figure.23 Load-strain curve for top flange for VS-3

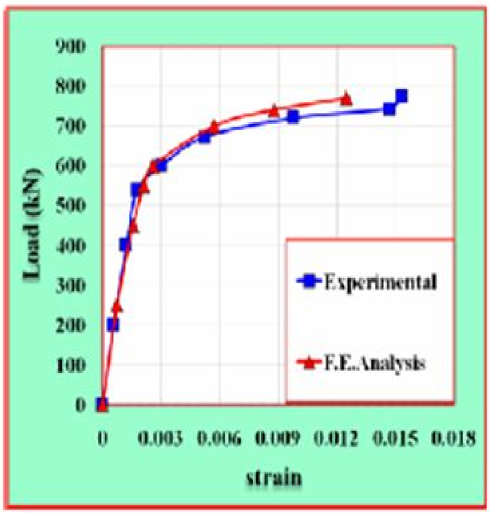

Figure. 24 Load-strain curve for bottom flange for VS-3

7 CONCLUSION

This study intended to investigate the possibilities of performing nonlinear finite element analysis of reinforced concrete structures using Ansys concrete model. The behavior of slab represented by the load-deflection curves in ANSYS show close agreement with the experimental data from the full-scale RC slab tests. The analytical tests carried out for the different cases studied (straight and draped) indicated that the load-deflection behavior and the ultimate loads are in good agreement with the published experimental results. The maximum difference between the predicted numerical ultimate load to the experimental ultimate load for all cases has value of (2.4%) for prestressed composite steel-concrete beams. The difference between numerical and experimental values for loads and deflections gives negative values for the two control slabs and positive values for the other strengthened slabs and the reason for that is the existing of with strengthened models in ANSYS program that makes the program solving for long time. The results also showed the load strain behavior in concrete slab and steel beam are in good agreement with the experimental results.

REFERENCES

[1] T.Subramani, M.S.Saravanan, “Analysis Of Non Linear Reinforced And Post Tensioned Concrete Beams Using ANSYS”, International Journal of Applied Engineering Research (IJAER) International Journal of Applied Engineering
[2] T.Subramani, T.Krishnan, M.S.Saravanan , Suboth Thomas, “Finite Element Modeling On Behaviour Of Reinforced Concrete Beam Column Joints Retrofitted With CFRP Sheets Using Ansys” International Journal of Engineering Research and Applications Vol. 4, Issue 12(Version 5), pp.69 -76, 2014
[3] T.Subramani.T, M.Subramani, K.Prasath.,"Analysis Of Three Dimensional Horizontal Reinforced Concrete Curved Beam Using Ansys" International Journal of Engineering Research and Applications, Volume. 4, Issue. 6 (Version 6), pp 156 - 161, 2014.
[4] T.Subramani, J.Jothi, M.Kavitha "Earthquake Analysis Of Structure By Base Isolation Technique In SAP", International Journal of Engineering Research and Applications, Volume. 4, Issue. 6 (Version 5), pp 296 - 305, 2014.
[5] T.Subramani.T, R.Manivannan, M.Kavitha, "Crack Identification In Reinforced Concrete Beams Using Ansys Software" ,International Journal of Engineering Research and Applications, Volume. 4, Issue. 6 (Version 6), pp 133 - 141, 2014.
[6] T.Subramani , A.Kumaresan, “ Advanced Cable Stayed Bridge Construction Process Analysis with ANSYS”, International Journal of Modern Engineering Research, Volume. 4, Issue.6 (Version 1), pp 28-33, 2014.
[7] T.Subramani, J.Jayalakshmi , " Analytical Investigation Of Bonded Glass Fibre Reinforced Polymer Sheets With Reinforced Concrete Beam Using Ansys" , International Journal of Application or Innovation in Engineering & Management (IJAIEM) , Volume 4, Issue 5, pp. 105-112 , 2015
[8] T.Subramani, K.Bharathi Devi, M.S.Saravanan. Suboth Thomas, Analysis Of RC Structures Subject To Vibration By Using Ansys,” International Journal of Engineering Research and Applications Vol. 4, Issue 12(Version 5), pp.45-54, 2014.

AUTHORS

Prof. Dr.T.Subramani Working as a Professor and Dean of Civil Engineering in VMKV Engineering. College, Vinayaka Missions University, Salem, Tamil Nadu, India. Having more than 25 years of Teaching experience in Various Engineering Colleges. He is a Chartered Civil Engineer and Approved Valuer for many banks. Chairman and Member in Board of Studies of Civil Engineering branch. Question paper setter and Valuer for UG and PG Courses of Civil Engineering in number of Universities. Life Fellow in Institution of Engineers (India) and Institution of Valuers. Life member in number of Technical Societies and Educational bodies. Guided more than 400 students in UG projects and 220 students in PG projects. He is a reviewer for number of International Journals and published 136 International Journal Publications and presented more than 30 papers in International Conferences

Er. K. Balamurugan,: Completed Bachelor of Civil Engineering in VMKV Engg. College, Vinayaka Missions University, Salem, Tamil Nadu, India. He is a Consulting Civil Engineer, Contractor and Managing Director of Balu Builders. He is member in Salem Civil Engineers Association. Doing final year M.E (Structural Engineering) in VMKV Engineering College, Vinayaka Missions University, Salem, Tamil Nadu, India.