International Journal of Emerging Trends & Technology in Computer Science (IJETTCS)
Web Site: www.ijettcs.org Email: editor@ijettcs.org
Volume 6, Issue 3, May- June 2017 ISSN 2278-6856

Analytical Study Of T-Beam Using ANSYS

T.Subramani[1], A.Mohammed Ali[2], R.Karthikeyan[3], E.Panner Selvan[4] , K.Periyasamy[5]

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

[2,3,4,5]UG Student, , Department of Civil Engineering, VMKV Engineering College, Vinayaka Missions University, Salem, India

Abstract
Recent trends of structural mechanics applications in finite element analysis demonstrate an increasing demand for efficient analysis tools. This paper presents finite element analysis for modeling T-beams structure used in building service system (mechanical, electrical, communications, and plumbing). The experimental program reported in this paper tested T-beams to failure effect on various beam behaviors. Using ANSYS, finite element models were developed to simulate beam deflection behavior. In reality, uncertainties exist in a system and environment that may make the application of deterministic design unreliable which causes the values of the variables that are acting on the system cannot be predicted with certainty. From the analysis results, it was observed that the changes in prestressing force, elastic modulus of prestressing steel, ultimate tensile strength of prestressing steel and beam width tend to be the most influencing parameters, which need to be tightly controlled.

Keywords: Analytical, Study, T-Beam, 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 behaviour of reinforced concrete structures. Due mainly to the complexity of the composite nature of the material, proper modelling of such structures is a challenging task. Despite the great advances achieved in the fields of plasticity, damage theory and fracture mechanics, among others, an unique and complete constitutive model for reinforced concrete is still lacking.
The shear failures in reinforced concrete (RC) structures are highly brittle when compared with the flexural failures. The addition of chopped steel fibers in the concrete matrix is effective in mitigating the brittle failures of RC structures. The addition of fibers in the matrix improves the strength and post cracking tensile stiffness of the concrete. The chopped fibers induce confinement effect in concrete matrix, which contributes to the increase in the strength characteristics of concrete. The toughening mechanisms, such as, fiber pullout, fiber bridging or fiber fracture at crack interface improves the post cracking tensile stiffness of the matrix. Thus, the presence of fibers increases the strength and results in a relatively ductile type of failure of RC beams. In the literature, the modeling of various effects due to the addition of fibers in RC structures has not been attempted extensively .The present study addresses this lacunae and reports the details of the finite element analysis of eleven shear critical partially prestressed concrete T-beams having steel fibers over partial or full depth. The finite element (FE) analysis of the T-beams has been carried out in the 'ANSYS' program. The predicted results, namely, loads, deflections and cracking behavior using the 'ANSYS' model have been compared with the corresponding test data

1.1 Objective
The purpose of this paper is to provide analytical data on the response of beams. The analytical program aimed at raising the strength of the shear deficient beams to that of the fully strengthened for beams. The analysis aimed at understanding the best wrapping style for retrofitting the deficient beams. The objective of this investigation is to study the effectiveness of increasing the flexural strength of concrete beams. The objective is achieved by conducting the following tasks:

- Flexural testing of concrete beams
- Calculating the effect of the flexural strength.
- Evaluating the failure modes
- Developing an analytical procedure to calculate the flexural strength of concrete beams.
- Comparing the two different experimental and analytical results.

In the scope of the present study to FEM modelling of the control beam under the static point loads have been analyzed using ANSYS software and the results so obtained have been compared to available result from the work done. Finally, comparison between the analytical results and experimental results the salient conclusion and recommendations of the present study

2.METHODOLOGY

Figure.1 shows methodology adopted in this study

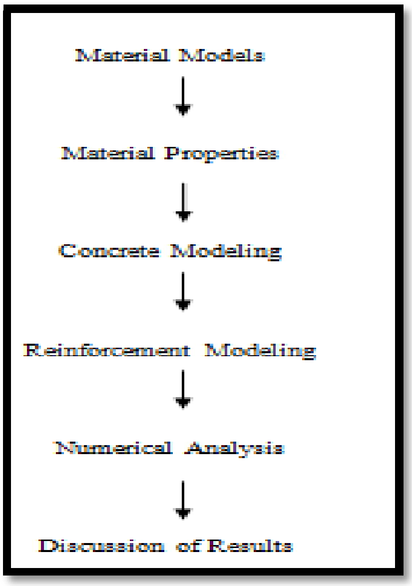

Figure.1 Methodology

3. ABOUT SOFTWARE

3.1.Finite Element Modelling

ANSYS computer program has been used for the finite element modeling. SOLID65 element is used to model the plain concrete material, since it has a capability of both cracking in tension and crushing in compression. The Finite Element Method (FEM) is a numerical analysis for obtaining approximate solutions to a wide variety of engineering problems. This has developed simultaneously with the increasing use of high-speed electronic digital computers and with the growing emphasis on numerical methods for engineering analysis.

3.1.1 General Description Of The FEM

The basic concept behind FEM is that a body or structure is divided into smaller elements of finite dimensions called „finite elements. The original structure is then considered as an assemblage of these elements at a finite number of joints called „nodes. The properties of the elements are formulated and combined to obtain the solution for the entire structure. The shape functions are chosen to approximate the variation of displacement within an element in terms of displacement at the nodes of the element. The strains and stresses within an element will also be expressed in terms of the nodal displacement. The principle of virtual displacement is used to derive the equations of equilibrium for the element and the nodal displacement will be the unknowns in the equations. The boundary conditions are imposed and the equations of equilibrium are solved for the nodal displacement. From the values of the nodal displacement for each element, the stresses and strains are evaluated using the element properties. Thus instead of solving the problem for the entire structure in one operation, in this Finite Element Method attention is mainly developed to the formulation of properties of the constituent elements.

3.2 Element Types

Selection of proper element types is another important part in Finite Element Analysis. For composite beams, the C-R steel sheet trough and braces were modeled by using SHELL 63 element in ANSYS. The Concrete portion was modeled by using a special element developed particularly for Concrete by ANSYS, SOLID 65 element. The stud connectors and reinforcement were modeled by using LINK 8 Element. SOLID 45 elements were used to model the steel plates provided at support and loading points.

SOLID65 element is defined by 8 nodes with three degrees of freedom at each node; translations in the nodal x, y, and z directions. The element material is assumed to be initially isotropic. The most important aspect of this element is the treatment of nonlinear material properties, where concrete is capable of directional cracking and crushing besides incorporating plastic and creep behavior. The LINK8 element used to model the reinforcing steel bar. It is a uniaxial tension-compression member that can include nonlinear material properties. The element comprises two nodes with three degree of freedom at each one. The elastic-perfectly plastic representation is assumed for the reinforcing steel bars. The SOLID46 layered structural solid element is used to model the CFRP materials. The element comprises 8 nodes with three degree of freedom at each node. The element material is assumed to be orthotropic and no slippage is assumed between the element layers (perfect interlaminate bond). Whereas, CFRP sheets are brittle materials, the stress-strain relationship is roughly linear up to failure. Consequently, in this study it is assumed that the stress-strain relationships for the CFRP laminates are linearly elastic.

3.3 Material Properties

From the experiments conducted, the following values are found out and listed.

- Compressive strength of concrete cubes
- Yield stress of C-R steel sheet
- Yield stress of reinforcing bars.

Coupon tests are done in order to determine the yield stress and modulus of elasticity of C-R sheet and reinforcing bars.

4.EXPERIMENTAL PROGRAMME

The experimental program was designed to investigate the failure behavior of T-beams under static loading conditions. Testing was intended to evaluate the flexural strength. Therefore, all beams were designed such that shear strength exceeded flexural strength. Accordingly, flexural failure was expected.The materials used, design and fabrication of T beams are described below. The experimental program was carried out at the Hume Concrete Product Research Centre (HCPRC) and Tenaga

Nasional Research Berhad (TNBR) for testing facilities and analysis assistance of prestressed beam with web openings.

4.1 Materials

The average 28 days concrete cube strength in compression was 55 MPa for all four beams, as evaluated by tests on three cubes specimens for each beam. The prestressing steel and shear reinforcement location in the prestressed concrete beams. Straight, 3 show the prestressing steel and shear reinforcement location in the prestressed concrete beams. Straight, bonded, seven-wired super high tensile strand with 12.9 mm diameter were used as prestressing tendons, with ultimate strength of 1,860 MPa. The stirrups for shear reinforcement were made from 10 mm rebars with minimum specified yield strength of 250 N/mm2.The elastic modulus of the prestressing steel is taken as 195 103 N/mm2. Figure 4 shows the fabricated prestressed beams used in the test program.

4.1.1 Testing Set-Up

All tests were conducted with a close loop hydraulic servo-controlled MTS testing system. The 360 KN jack was capable of both displacement and load control for monotonic or cyclic loading. A four-point loading scheme, with an effective span of 4,000 mm and a distance of 1,200 mm between the loads points was used to limit the presence of shear stress in the mid-span zone.

4.1.2 Finite Element Models:

In this study, the finite element analysis of the model was set up to examine three different behaviors: initial cracking of the beam, yielding of the steel reinforcement, and the strength limit state of the beam. The Newton-Raphson method of analysis was used to compute the nonlinear response. The application of the loads up to failure was done incrementally as required by the Newton-Raphson procedure. After each load increment was applied, the restart option was used to go to the next step after convergence. The two convergence criteria used for the analysis were force and displacement.

4.1.3 Element Types

Concrete part was modeled using a three dimensional solid element, SOLID65, which has the material model to predict the failure of brittle elements. SOLID65 is defined with eight nodes each with three degrees of freedom: translations in nodal x, y and z directions. This element is capable of cracking in tension and crushing in compression. Plastic deformation and creep can also be captured. The cracking is determined by the criterion of maximum tensile stress, called ***`tension cutoff*** `. Concrete crushes when the compressive principal stress (*Von Mises* stress) on the failure surface surpasses the *Willam-Warnke* failure criterion dependent on five materials parameters . The SOLID 45 element was used for the supports for the beam. This element has eight nodes with three degrees of freedom at each node with translations in the nodal x, y and z directions. To simulate the behaviors of prestressing steel, a truss element, LINK8, were used to withstand the initial strain attributed to prestressing forces, by assuming perfect bond between these elements and concrete. LINK8 requires users to input `real constants` to define reinforcement geometry, material behavior, and prestressing strain. Note that this truss element cannot resist neither bending moments nor shear forces. The descriptions for each element type were laid out in the ANSYS element library

4.2 Material Properties

4.2.1concrete

The SOLID65 element requires linear isotropic and multi-linear isotropic materials properties to properly model concrete. Concrete is a quasi-brittle material and has very different behaviors in compression and tension. The tensile strength of concrete is typically 8-15% of the compressive strength.

In tension, the stress-strain curve for concrete is assumed to be linearly elastic up to the ultimate tensile Poisson`s ration for concrete was assumed to be 0.3 and was 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.The shear transfer coefficient used in this study is equal to 0.2. Steel reinforcement and prestrssing steel:

Steel reinforcement in the experiment beams was constructed with typical steel reinforcing bars fy = 1000 MPa.

Elastic modulus and yield stress for the steel reinforcement used in this FEM study follow the design material properties used for the experimental investigation. 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.

The SOLID 45 element is being used for supports on the beam. Therefore, this element is modeled as linear isotropic element with a modulus of elasticity for the steel Es = 1000 MPa and Poisson`s ratio v = 0.3. The LINK8 element is being used for all the steel reinforcement in the beam and it is assumed to be bilinear isotropic. Bilinear isotropic material was also based on the *Von Mises* failure criteria.

5.RESULTS

5.1 FE Analysis of the T-beams using the 'ANSYS'

The partially prestressed concrete T-beams have been analyzed using the ANSYS. The 'ANSYS' model accounts for the nonlinearity, such as, bond-slip of longitudinal reinforcement, post-cracking tensile stiffness of the concrete, stress transfer across the cracked blocks of the concrete, load sustenance through the bridging action of steel fibers at crack interface and yielding of reinforcement. The analysis was carried out in stages using Newton-Raphson technique.

5.2 With Reinforcement

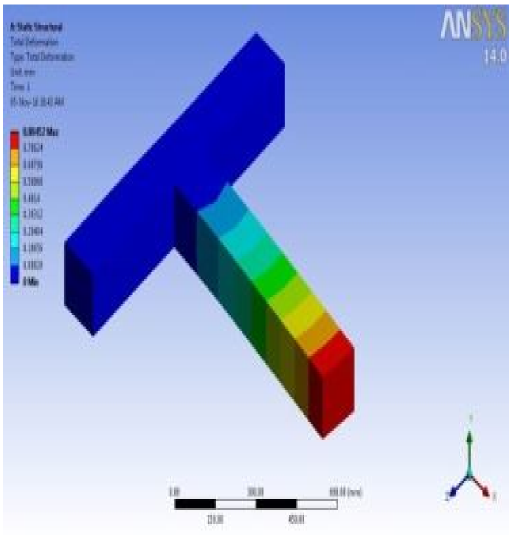

Figure.2 Deformation

Figure.2 shows Deformation. Figure.3. shows Meshed. Figure.4. shows Imported. Figure.5. shows Strain. Figure.6. shows Stress.

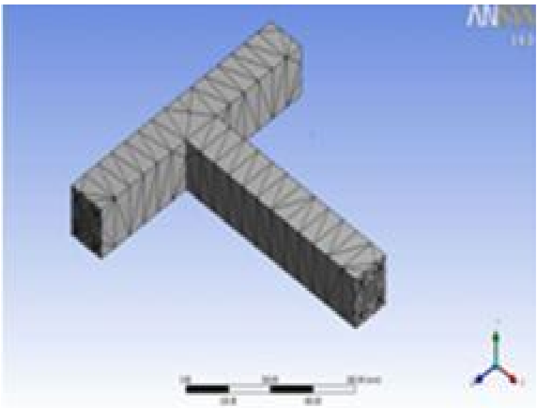

Figure.3. Meshed

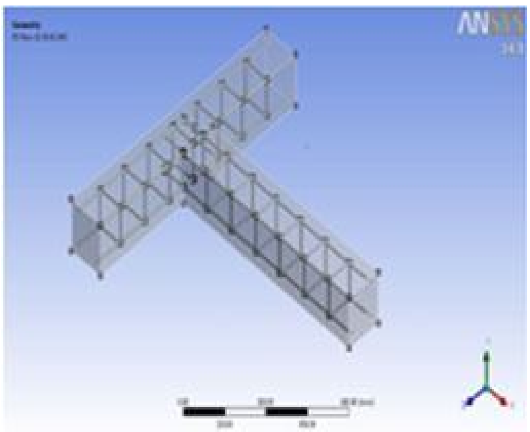

Figure.4. Imported

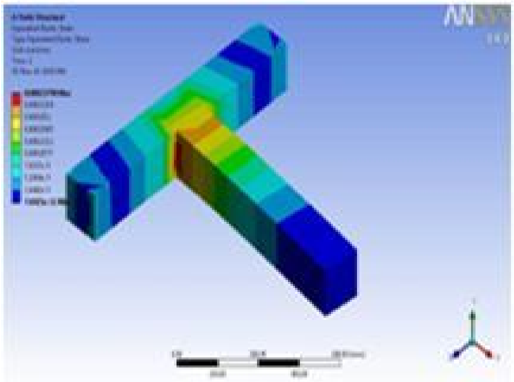

Figure.5. Strain

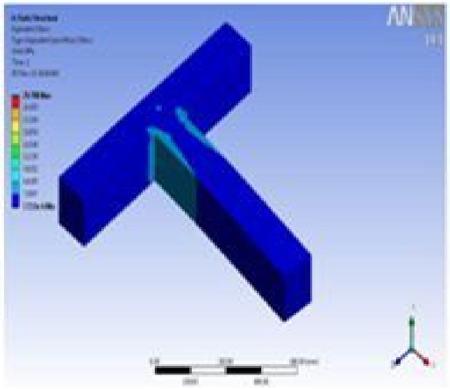

Figure.6. Stress

5.3 ANSYS Report

Contents

Units

TABLE 1

Unit System	Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius
Angle	Degrees
Rotational Velocity	rad/s
Temperature	Celsius

Model (A4)

Geometry

TABLE 2
Model (A4) > Geometry

Object Name	Geometry
State	Fully Defined
Definition	
Source	D:\Michael\Projects\Older\Civil Beam (05-11-16)\Solidworks\Assem1.SLDASM

Figure.7. Ansys report

Figure.7. shows Ansys report. Figure.8.shows Ansys report

TABLE 14
Model (A4) > Static Structural (A5) > Solution (A6) > Results

Object Name	Total Deformation	Equivalent Elastic Stain	Equivalent Stress
State	Solved		
Scope			
Scoping Method	Geometry Selection		
Geometry	All Bodies		
Definition			
Type	Total Deformation	Equivalent Elastic Stain	Equivalent (von-Mises) Stess
By	Time		
Display Time	Last		
Calculate Time Histoy	Yes		
Identifier			
Suppressed	No		
Results			
Minimum	0. mm	7.8415e-011 mm/mm	7.713e-006 MPa
Maximum	0.88452 mm	2.3798e-004 mm/mm	29.788 MPa
Minimum Occurs On	CONCRETE-1	BEAM STRUCTURE-1	
Maximum Occurs On	CONCRETE-1		BEAM STRUCTURE-1
Information			
Time	1. s		

Figure.8. Ansys report

5.4 Without Reinforcement

Figure.9 shows Deformation. Figure.10 shows Imported Figure.11 shows Meshed. Figure.12 shows Strain Figure.13 shows Stress

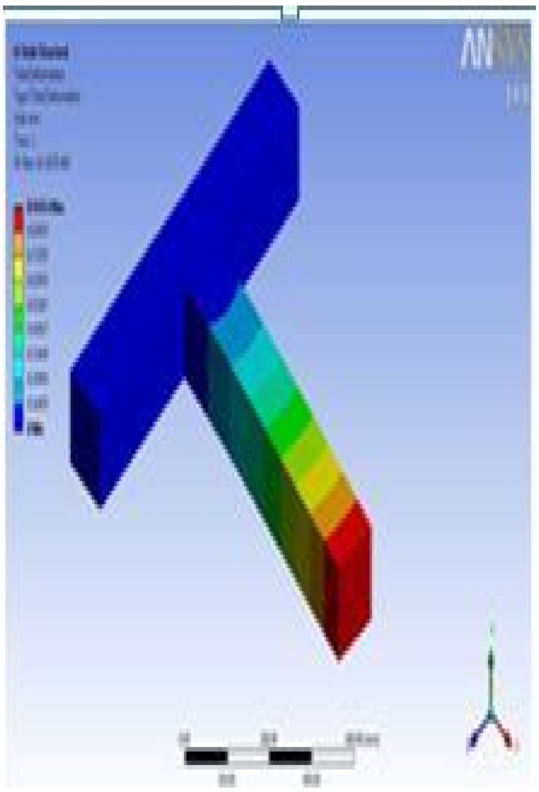

Figure.9 Deformation

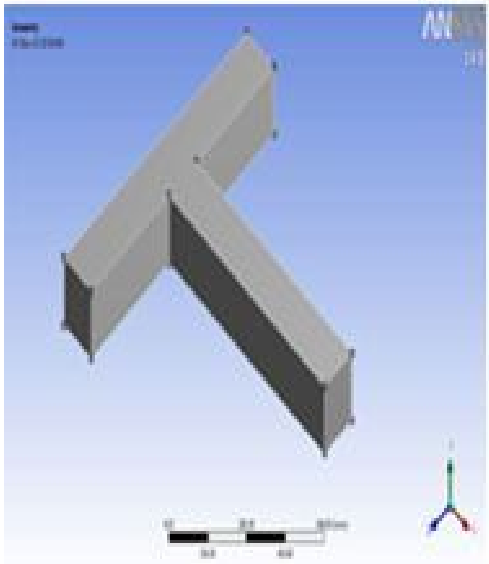

Figure.10 Imported

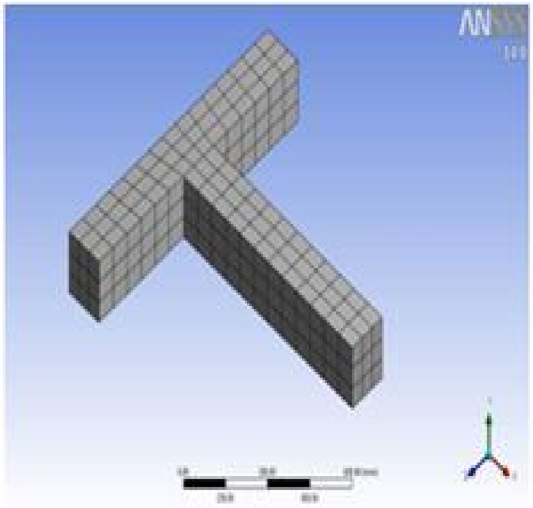

Figure.11 Meshed

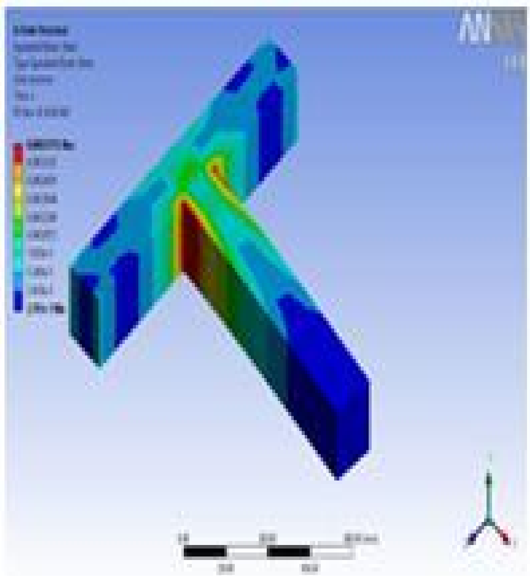

Figure.12 Strain

International Journal of Emerging Trends & Technology in Computer Science (IJETTCS)
Web Site: www.ijettcs.org Email: editor@ijettcs.org
Volume 6, Issue 3, May- June 2017 ISSN 2278-6856

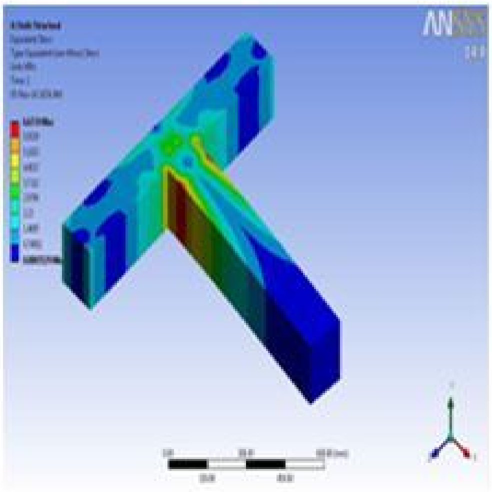

Figure.13 Stress

5.5Ansys report

Ansys result shown in Figure.14 & Figure.15

Contents

- Units
- Model (A4)
  - Geometry
    - Parts
  - Coordinate Systems
  - Connections
    - Contacts
      - Contact Region
  - Mesh
  - Static Structural (A5)
    - Analysis Settings
    - Loads
    - Solution (A6)
      - Solution Information
      - Results
- Material Data
  - Concrete
  - Structural Steel

Units

TABLE 1

Unit System	Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius
Angle	Degrees
Rotational Velocity	rad/s
Temperature	Celsius

Model (A4)

Geometry

TABLE 2
Model (A4) > Geometry

Object Name	Geometry
State	Fully Defined
Definition	

Figure.14. Ansys result

Material	
Assignment	Concrete
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Length X	1000. mm
Length Y	150. mm
Length Z	1000. mm
Properties	
Volume	4.1625e+007 mm³
Mass	95.737 kg
Centroid X	229.73 mm
Centroid Y	4.3411e-016 mm
Centroid Z	-3.907e-015 mm
Moment of Inertia Ip1	4.5745e+006 kg·mm²
Moment of Inertia Ip2	1.3085e+007 kg·mm²
Moment of Inertia Ip3	8.8692e+006 kg·mm²
Statistics	
Nodes	1552
Elements	243
Mesh Metric	None

Figure.15. Ansys result

6.CONCLUSION

Based on results of prestressed beams with the corresponding experimental data, following conclusions were drawn.

- The predicted load in of T-beams at various stages was found to be in good agreement with the test data.
- The proposed model predicted slightly softer results in post-cracking regime of the load-deflection response of T-beams. This variation is due to the difference in the bond-slip model of reinforcement used in the analysis when compared with that present in the test.
- The 'ANSYS' model correctly predicted the diagonal tension failure and shear compression failure of prestressed concrete beams observed in the experiment.
- It is expected that the modeling strategy for the finite element analysis proposed in this study will be used for designing/ analyzing SFRC members.

References

[1] T.Subramani. , S.Vishnupriya, "Finite Element Analysis of a Natural Fiber (Maize) Composite Beam", International Journal of Modern Engineering Research, Volume. 4, Issue. 6 (Version 1), pp 1 – 7, 2014,

[2] T.Subramani., R.Senthil Kumar, "Modelling and Analysis of Hybrid Composite Joint Using Fem in ANSYS", International Journal of Modern Engineering Research, Volume 4, Issue 6 (Version 1), pp 41- 46, 2014.

[3] T.Subramani, S.Sharmila, "Prediction of Deflection and Stresses of Laminated Composite Plate with Artificial Neural Network Aid", International Journal of Modern Engineering Research, Volume 4, Issue 6 (Version 1), pp 51 -58, 2014.

[4] T.Subramani., S.Sundar, M.Senthilkumar, "Investigation of the Behaviour for Reinforced Concrete Beam Using Non Linear Three Dimensional Finite Elements", International Journal of Modern Engineering Research, Volume. 4, Issue. 6 (Version 2), pp 13 -18, 2014,

[5] T.Subramani, A.Arul, "Design And Analysis Of Hybrid Composite Lap Joint Using Fem" International Journal of Engineering Research and Applications, Volume. 4, Issue. 6 (Version 5), pp 289- 295, 2014.

[6] 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.

[7] T.Subramani., R.Manivannan.R, 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.

[8] T.Subramani., Reni Kuruvilla, J.Jayalakshmi., "Nonlinear Analysis Of Reinforced Concrete Column With Fiber Reinforced Polymer Bars" International Journal of Engineering Research and Applications Volume. 4, Issue. 6 (Version 5), pp 306- 316, 2014.

[9] T.Subramani, D.Sakthi Kumar, S.Badrinarayanan. "Fem Modelling And Analysis Of Reinforced Concrete Section With Light Weight Blocks Infill " International Journal of Engineering Research and Applications, Volume. 4, Issue. 6 (Version 6), pp 142 - 149, 2014.

[10] T.Subramani, B.Saravanan., J.Jayalakshmi., "Dynamic Analysis Of Flanged Shear Wall Using Staad Pro", International Journal of Engineering Research and Applications, Volume. 4, Issue. 6 (Version 6), pp 150 - 155, 2014.

[11] T.Subramani, 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.

[12] T.Subramani., K.Bharathi Devi., M.S.Saravanan. , Suboth , 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.

[13] T.Subramani., K.Bharathi Devi., M.S.Saravanan., Suboth Thomas, "Analysis Of Seismic Performance Of Rock Block Structures With STAAD Pro International Journal of Engineering Research and Applications Vol. 4, Issue 12(Version 5), pp.55- 68, 2014.

[14] T.Subramani., T.Krishnan., M.S.Saravanan.M , 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

[15] T.Subramani., S.Krishnan., M.S.Saravanan.M, Suboth Thomas "Analysis Of Retrofitting Non-Linear Finite Element Of RCC Beam And Column Using Ansys" International Journal of Engineering Research and Applications ,Vol. 4, Issue 12(Version 5), pp.77-87, 2014.

[16] 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

[17] T.Subramani and M.Kavitha, "Analysis Of Reliability Of Steel Frame Systems With Semi-Rigid Connections Using Numerical Method And Finite Element Analysis", International Journal of Applied Engineering Research (IJAER), Volume 10, Number 38,Special Issues, pp.28240-28246, 2015.

[18] 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 Research (IJAER), Volume 10, Number 38 Special Issues, pp.28247-28252, 2015

[19] T.Subramani, K.Balamurugan , " Finite Element Anaylsis Of Composite Element For FRP Reinforced Concrete Slab By Using ANSYS" , International Journal of Application or Innovation in Engineering & Management (IJAIEM) , Volume 5, Issue 5, pp. 076-084 , 2016 .

[20] T.Subramani, V.Kanian Poonkundran , " Prefabricated Multistory Structure Exposure To Engineering Seismicity By Using SAP" , International Journal of Application or Innovation in Engineering & Management (IJAIEM) , Volume 5, Issue 5, pp. 123-131 , 2016 .

[21] T.Subramani, A.Kumaravel , " Analysis Of Polymer Fibre Reinforced Concrete Pavements By Using ANSYS" , International Journal of Application or Innovation in Engineering & Management (IJAIEM) , Volume 5, Issue 5, pp. 132-139 , 2016 .

[22] T.Subramani, R.Praburaj , " Pushover Anaylsis Of Retrofitted Reinforced Concrete Buildings By Using SAP" , International Journal of Application or

Innovation in Engineering & Management (IJAIEM) , Volume 5, Issue 5, pp. 140-147 , 2016 .

[23] T.Subramani, M.Senthilkumar , " Finite Element Anaylsis Of RC Beams With Externally Bonded Simcon Laminates By Using ANSYS" , International Journal of Application or Innovation in Engineering & Management (IJAIEM) , Volume 5, Issue 5, pp. 148-155 , 2016

[24] T.Subramani, R.Vasanthi , " Earth Quake Resistant Building Using SAP" , International Journal of Application or Innovation in Engineering & Management (IJAIEM) , Volume 5, Issue 5, pp. 173-181 , 2016 .

[25] T.Subramani, A.Selvam , " Studies On Economical Configuration Of RCC And Prestressed Shell Roofs By Using ANSYS " , International Journal of Application or Innovation in Engineering & Management (IJAIEM) , Volume 5, Issue 5, pp. 182-191 , 2016 .

AUTHOR

Prof.Dr.T.Subramani Working as a Professor and Dean of Civil Engineering in VMKV Engineering College, Vinayaka Missions University, Salem, TamilNadu, India. Having more than 27 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 300 students in PG projects. He is a reviewer for number of International Journals and published 174 International Journal Publications and presented more than 25 papers in International Conferences.

A.Mohammed Ali, completed his D.C.E in the VMKVSS Polytechnic College in Salem Now working as an Deputy Manager, DHFL, Salem. Currently he is doing his B.E in the branch of civil Engineering in Vinayaga Mission Kirupananda Variyar Engineering College at Salem.

R.Karthikeyan, completed his D.C.E in the Rajaji Institute of Technology. Now working in Aditya Birla. Currently he is doing his B.E in the branch of civil Engineering in Vinayaga Mission Kirupananda Variyar Engineering College at Salem.

E.Panner Selvan, completed his D.C.E in the Lakshmi Narayanan Polytechnic College in Dharmapuri. Now he is a Propreitor of KSP & Co. Civil Engineering Contractor in Salem. Currently he is doing his B.E in the branch of civil Engineering in Vinayaga Mission Kirupananda Variyar Engineering College at Salem.

K.Periyasamy, completed his D.C.E in the Paavendhar Polytechnic College in Attur. Now working in Ranesh Construction Covai. Currently he is doing his B.E in the branch of civil Engineering in Vinayaga Mission Kirupananda Variyar Engineering College at Salem.