Experimental and Analytical Nonlinear Analysis of RC Flexural Members Strengthened with Exterior Bondedfiber Reinforcedpolymer

D Samuel Abraham, Aiswarya S, D Yogesh, S Janaki Raman Abishek R R, Blessanth Jeyaseelan J

ABSTRACT--- ***This paper illustrates the practical application of non-linear models in the analysis of reinforced concrete beams without and with wrapping using Fiber Reinforced Polymer (FRP). Finite element analysis is used to study the components. Non-linear Finite Element Analysis (FEA) method is used to deliberate the mechanical properties of the reinforced concrete beams and FRP sheets in terms of the crack load and ultimate bearing capacity. The finite element model is set up using ANSYS parametric design language (APDL) in this simulation process; this process also established the model of bond stress-slip relationship between concrete and steel bar and the model of bond between concrete and FRP sheets. The nonlinear FEA results showcase the behavior of beams under cracking load with the development of cracks in beams without wrapping and behavior of FRP sheets under loading with debonding of FRPsheets.***

Keywords: RC Beams, FRP, Wrapping, Finite Element Analysis, Non-Linear Analysis, Modeling.

I. INTRODUCTION

The most common testing method used to examine the elements individually and the impact of concrete strength when load is exerted,is experimental based testing. Though this testing method gives real time results, it takes long time and is of higher cost. Finiteelement analysis can also be used for this study. However, past attempts to accomplish this were not successful due to software as well as hardware constraints. Nowadays, FEAusage has enhancedbecause ofimprovements in software and hardware capabilities. At present finite element analysis method is most preferred for concrete structural components analysis.

Revised Manuscript Received on 14 February, 2019.

D Samuel Abraham, Assistant Professor, Department of Civil Engineering, Karunya Institute of Technology and Sciences, Coimbatore, Tamilnadu, India.(samuelabraham@karunya.edu)

Aiswarya S, Assistant Professor, Department of Civil Engineering, Karunya Institute of Technology and Sciences, Coimbatore, Tamilnadu, India.

D Yogesh, Structural Engineer, Kavin Engineering and services, Coimbatore, Tamilnadu, India.

S Janaki Raman, Assistant Professor, Department of Civil Engineering, Karunya Institute of Technology and Sciences, Coimbatore, Tamilnadu, India.

R R, Abishek, Post Graduate Student, Department of Civil Engineering, Karunya Institute of Technology and Sciences, Coimbatore, Tamilnadu, India.

Blessanth Jeyaseelan J, Post Graduate Student, Department of Civil Engineering, Karunya Institute of Technology and Sciences, Coimbatore, Tamilnadu, India.

II. REVIEW OF LITERATURE

Amir M.Malek and Hameed Saadatmanesh (1998) studied the shear and flexural capacity of beams reinforced with bonding fiber reinforced plastic (FRP) plates or fabrics to web-bonded reinforcement of concrete beams. Analytical modelsul were developed to compute the stress in reinforced concrete beams, and the shear force resisted by the composite plate before cracking and after development of flexural cracks. The model was grounded on the principle that there will be a seamless bond between concrete and FRP and between concrete and compatibility of strains in FRP and concrete beam.

Godat et al. (2011) established a finite element model to find the parameters that most influence the behavior of FRP shear-strengthened beams. The parameters studied were amount of steel stirrups, concrete compressive strength, stiffness of the FRP, amount of FRP, and span-to- effective depth ratio. Variation of axial strain in the FRP was also evaluated. The parameters influencing the initiation and propagation of deboning were also studied.

Mohammed A. Sakr et al. (2014) developed a Uniaxial Nonlinear Finite Element Model (UNFEM) to simulate the mechanical behavior of FRP- strengthened RC continuous beams using realistic nonlinear constitutive relations. The model developed was able to envisage the final load of FRP strengthened reinforced concrete beams.

III.EXPERIMENTAL PROGRAM SPECIFICATION OF SPECIMEN

Beams of size 1000x150x250 mm were provided with main reinforcement of 10 mm dia. bars and stirrups of 8 mm dia. @ 150 mm spacing. Beam was externally wrapped with Glass FRP sheet of 2mm thickness

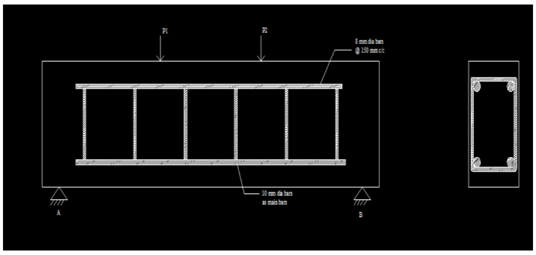

Fig 1: Beam specimen

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

IV. FINITE ELEMENT MODELLING IN ANSYS

Asthe cross-section of concrete beams is identical and loading of one quarter of the beam was modeled. Multiple tasks need to be performed to establish a working finite element model in ANSYS (12.0.) The upcoming section elaborates about the tasks and entries required to establish a working FE calibration model.

A. Types ofElements

The types of Elements used for this model are:

Solid65: Concrete is modeled using Solid65 elements which have eight nodes and each node in turn has three degrees of freedom– translations in the nodal x, y, and z directions. Deformation of plastic, crushing and cracking in three of the orthogonal directionscan be accomplished using Solid65elements.

Solid45: Steel plates for beam support were designedwith Solid45 elements. Solid45 element consists of eight nodes and each node in turn has three degrees of freedom – translations in the nodal x, y, and zdirections.

Link8: Steel reinforcement was designedwith Link8 elements. Link8 is a 3D spar element. Link8 consists of two nodes and each node has three degrees of freedom – translations in the nodal x, y, and z directions. Link8 exhibits the property of plasticdeformation.

Solid46 element was used for FRP wrapping. Solid46 allows maximum of 100 material layers with diverse orientations and orthotropic material properties in each layer. The element has three degrees of freedom at eachnode

B. RealConstants

S.No.	Element Type	Constants			
			Rebar 1	*Rebar 2*	*Rebar 3*
1	**Solid 65**	Count of Elements	0	0	0
		Volume proportion	0	0	0
		Orientation Angle	0	0	0
2	**Link 8**	Cross sectional area(mm^2)	201.50		
		Initial strain	0		
		Cross sectional area(mm^2)	50		
		Initial strain	0		

Real Constant 1stSetis defined to analyze the properties of Solid65 element. Assuming a smeared model, Real constants are required for re- bar. Values can be set for Count of elements, Volume Proportion, and Orientation Angles. The beam is modeled using discrete reinforcement for the purpose of this analysis. Therefore, all real constants were set to zero, to nullify the smeared reinforcement capability of Solid65 element. Real Constant 2nd and 3rdSets are defined for Link8 element. Real constants –initial strain and cross-sectional area were set as tabulated in Table1.

C. Properties of the Elements

Table 2 gives the list of parameters that are considered to define the model of elements used for this analysis.

Model 1 lists the parameters considered for Solid65 element. This element requires linear isotropic and multilinear isotropic properties to properly model concrete. The multilinear isotropic property determines the von Mises failure criterion. Model 2 lists the parameters considered for Solid45 element. This element is used in steel plates for beam support and at loading points. Therefore, Solid45 is designed as a linear isotropic element which has anelastic modulus for the steel and Poisson’s ratio. Model 3 lists the parameters considered for Link8 element. This element is modeled as bilinear isotropic element as it is used for the reinforcement of steel in the beam. Von Mises failure criteria is also considered in modelling this bilinear isotropic Link8 element.

Table 2 Material Property for model

Model No	Element Type	Element Properties		
1	**Solid 65**	**Linear Isotropic**	**EX**	**25000**
			PRXY	**0.23**
		Multiline ar Isotropic	**Strain**	**Stress**
			0.0003	**7.5**
			0.0007	**15.59**
			0.00098	**19.76**
			0.0014	**23.49**
			0.002	**25**

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

		Concrete		
		Shear transfer coefficient for open crack	**0.3**	
		Shear transfer coefficient for closed crack	**1**	
		Uniaxial tensile cracking stress	**3.5**	
		Uniaxial crushing stress(positive)	**-1**	
		Biaxial crushing stress(Positive)	**0**	
		Ambient hydrostatic stress state	**0**	
		Biaxial crushing stress(Positive) under the ambient hydrostatic stress	**0**	
		Uniaxial crushing stress(positive) under ambient hydrostatic stress	**0**	
		Stiffness multiplier for cracked tensile condition	**0**	
2	**Solid 45**	**Linear isotropic**	**EX**	**20000 0**
			Prxy	**0.3**
3	**Link 8**	**Linear isotropic**	**EX**	**20000 0**
			Prxy	**0.3**
		Bilinear isotropic	**Yield Stress**	**415**
			Tangen t Modulus	**20**

D. Modeling

The beam, supports and plates were designed as volumes. Since a quarter of the beam is being modeled, the model is 600 mm long, with a cross-section of 75 mm X 800 mm. Fig 1 represents the combined volumes of the plate, support and beam. Fig 3 represents FE mesh for the beam model. Flexural and shear reinforcements were created using Link8 model. Reinforcement exists at a plane of symmetry and in the beam. As per Figure 4, the rebar shares the same nodes at the points that it intersects the shear stirrups. Rectangular mesh is recommended by Damian Kachlakev (2001) to obtain good results from the Solid65elements.

Table 3 Coordinates Values for the rectangular mesh

Coordinates	Concrete	Steel Support	Steel Plate
X-Coordinate	(0,1000)	(25,75)& (925,975)	(300,400) & (600,900)
Y-Coordinate	(0,250)	(0,-25)	(250,275)
Z-Coordinate	(0,150)	(0,150)	(0,150)

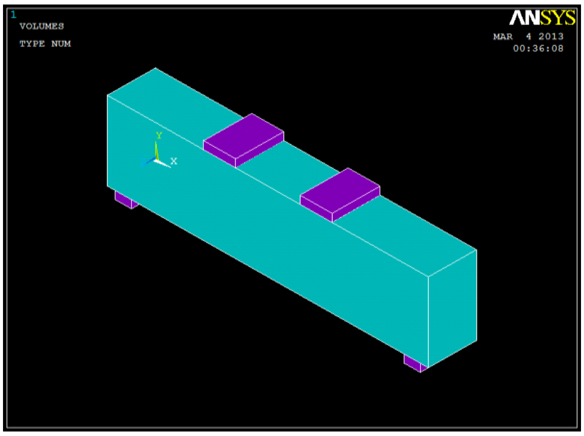

Fig 2 Combined volumes of plate, support and beam

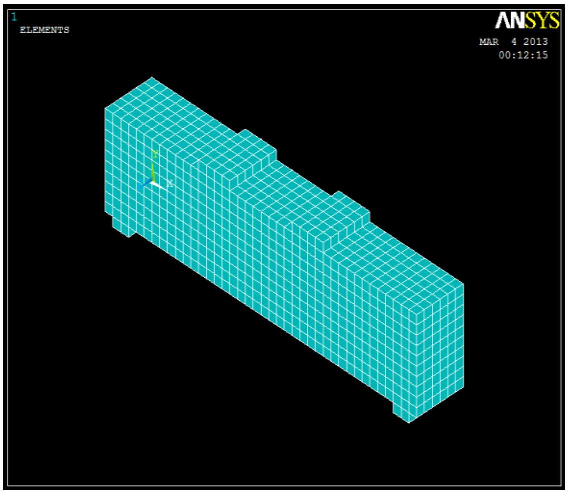

Fig 3 Meshed Beam model

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

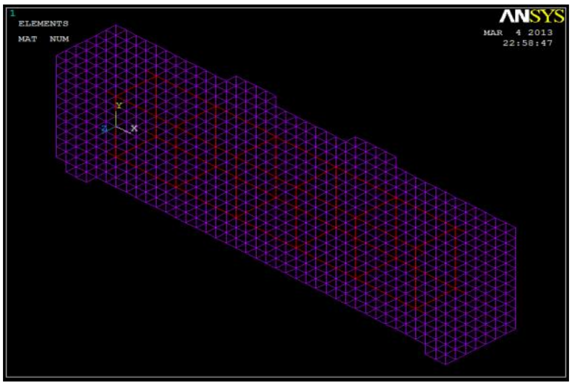

Fig 4 Reinforcement Model

E. NumberingControls

The command "merge item", merges separate entities that have the same location. Order in which merging occurs is significant and caution must be taken when merging entities in a model.

Merging key points before nodes can result in some of the nodes becoming "orphaned"; that is, the nodes lose their association with the solid model. The orphaned nodes can cause certain operations to fail, such as boundary condition transfers, surface load transfers, and so on. Care must be taken to always merge in the order that the entities appear. It was ensured that all entities were merged in the proper order. Also, the lowest number was retained duringmerging.

F. Boundary and LoadConditions

To get unique solution from the model, Displacement boundary conditions are required. Conditions need to be applied at point of symmetry, and where the supports and loadings exist in order to ensure that the model acts in a similar way as the experimental beam boundary. First, the symmetry boundary conditions were set. A hinged support was created in the model under experimentation. A single line of nodes on the plate were given constraint in the UX, UY and UZ directions, applied as constant values of 0. Fig. 5 and Fig. 6 represent the plate and appliedloading

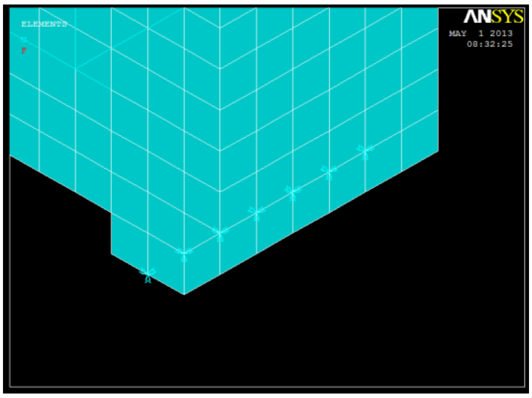

Fig 5 Support Condition

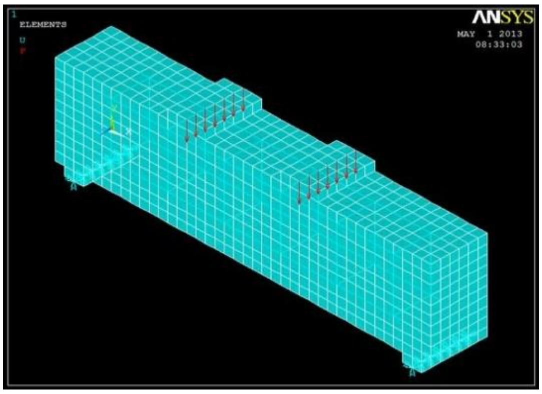

Fig 6 Beam after loading

G. Type of Analysis

The finite element model considered for this analysis is a deep beam under transverse loading. Static analysis type is utilized to analyze this model. The Solution Controls command indicates the use of a linear or non-linear solution for the finite element model. Default values are set for the convergence criteria except for the tolerances. The tolerances for displacement are set as 5 times the default values.

V. RESULTS AND DISCUSSIONS

Successful modeling of Deep Beam is achieved through suitable selection of elements, material models, real constants and convergence criteria.

Figure 7 shows the deformed shape of the deep beam for load step value =4kN.

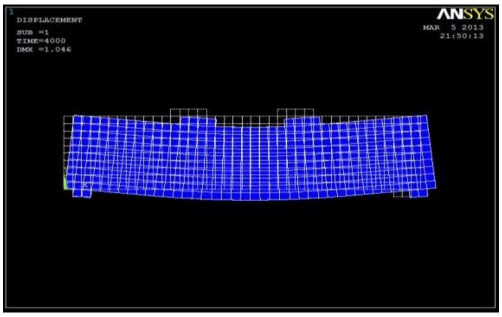

Fig 7: Deformed Shape Plot

Figure 8 shows the nodal displacement plot in y direction i.e. UY Plot.

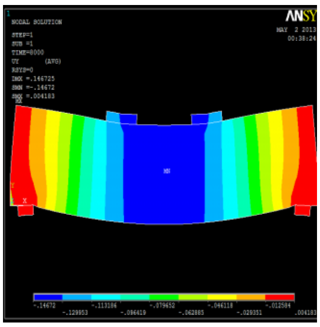

Fig 8 UY Displacement Plot

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

The Crack/Crushing plot option in ANSYS is used obtain the cracking patterns in the beam. In the non-linear region of the response, subsequent cracking occurs as more loads are applied to the beam. The Concrete Crush Plot is represented in Fig. 9

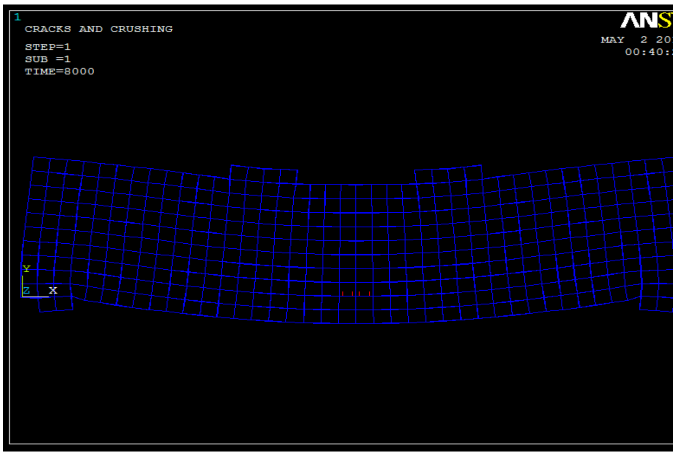

Fig 9 Concrete Crush Plot

Figure 10 shows the shear stress plot of XY plane. It clearly represents the concentration of stresses inthe shear span as according to the testbeam

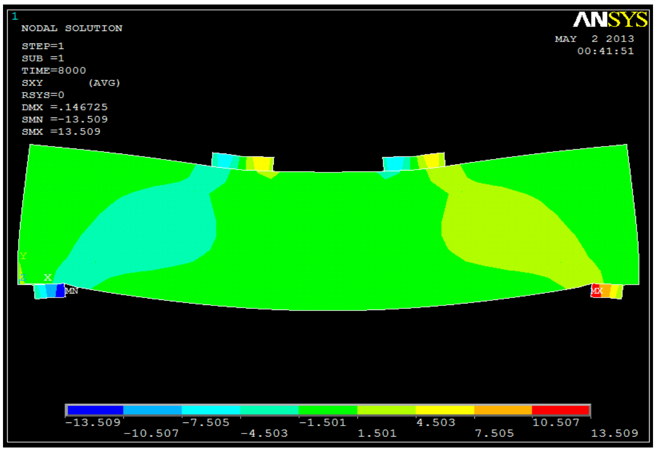

Fig 10 Shear Stress Plot in XY direction

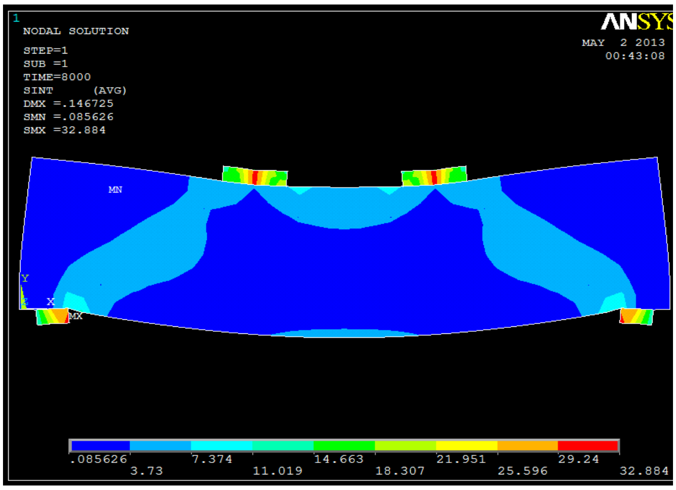

Fig 11 shows the stress intensity plot of XY plane.

Comparison of experimental and studies carried out is presented in Table 4.

Table 4 Comparison of Results

Beam	Without wrapping		With wrapping	
	First crack load (kN)	**Deflection (mm)**	**First crack load (kN)**	**Deflection (mm)**
Experimental Result	30	4.73	40	4.92
ANSYS model	56	0.15	70	0.18

VI. CONCLUSION

The objective of testing and validation of results through a FEA package (ANSYS) is ensured and the results are tabulated below (Table 4). The experimental testing value on beam without wrapping resulted a maximum mid span deflection of 4.73 mm and beam with GFRP wrapping resulted a maximum mid span deflection of 4.92 mm whereas the FE model in ANSYS resulted as 0.15mm and 0.18mm respectively. The first crack load obtained in experimental testing is 30 kN for beam without wrapping and 40kN for beam with GFRP wrapping whereas in FE model in ANSYS is resulted as 56kN and 70kN respectively. These uncertainties in the values may be due to equation solver used in nonlinear analysis inANSYS.

The following conclusions were arrived at based on the evaluation of the analysis of the beam model:

1. Deflection in the centerline inline with the initial cracking of concrete is obtained andplotted.
2. FEA package is used to model the failure mechanism of RC beam.
3. Experimental values were validated along with the analyticalvalues.

REFERENCES

1. Anthony J. Wolanski, B.S. (2004) “Flexural Behaviour of Reinforced and Prestressed Concrete Beams Using Finite Element Analysis”, Master’s Thesis, Marquette University, Milwaukee, Wisconsin
2. Federico Tavarez (2001) “Simulation of Behavior of Composite Grid Reinforced Concrete Beams Using Explicit Finite Element Methods” Master’s Thesis, University of Wisconsin,Madison.
3. Damian Kachlakev (2001)“Finite Element Modeling of ReinforcedConcrete Structures Strengthened With Frp Laminates”, Report, Oregon Department of Transportation Research Group.
4. S.Ahmad,A.Shah,N.Zaman,K.Salimullah,“ Design and evaluation of the shear strength of deep beams by strut and Tie model”,IJSt,Trasactions of civil and Environmental Engineering,Vol.35,pp1-13 (2011)
5. Madenci.E(2006),“The Finite Element Methods and Application in Engineering”,Springerlink, 686P.463
6. “Structural Analysis Guide” (2006), Release 12.0, ANSYSInc
7. Malek A, Saadatmanesh H. Analytical study of reinforced concrete beams strengthened with

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

web-bonded fiber reinforced plastic plates or fabrics.ACI Struct J1998;9:343–52.

8. Malek A, Saadatmanesh H. Ultimate shear capacity of reinforced concrete beams strengthened with web-bonded fiber- reinforced plastic plates. ACI Struct J 1998;95:391–9.
9. Kachlakev D, Miller T, Yim S, ChansawatK, Postisuk T. Finite element modeling of reinforced concrete structures strengthened with FRP laminates. Final Report SPR316, Orgon Department of Transportation Research Group; 2001. p.113.
10. Mohammed A. Sakr, Tarek M.Khalifa, Walid N.Mansour, External Strengthening of RC Continuous Beams Using FRP Plates: Finite Element Model, Proceedings of the Second International. Conference on Advances in Civil, Structural and Mechanical Engineering- CSM 2014. ISBN: 978-1-63248-054-5. DOI: 10.15224/978-1-63248-054-5-101.