

Analysis of Stressed Retrofitted Reinforced Concrete Beams Strengthened with GFRP Laminates Using Finite Element Method

Dr.Ramesh Manoli ¹ , Basavaraja H S ² , Rashmi B A ³

Professor, Department of Civil Engineering, Global Academy of Technology and Management, Bangalore, Karnataka, India ¹

Assistant Professor Department of Civil Engineering, Jyothy Institute of Technology, Bangalore, Affiliated to VTU, Belagavi, Karnataka, India ²

Assistant Professor, Department of Science and Humanities, PES University Electronic City Campus, Bangalore South, Karnataka, India ³

ABSTRACT: This paper reports on an analytical investigation carried out to investigate the deflection behaviour of strengthened RC beams with glass fiber reinforced polymer (GFRP) sheets. Application of GFRP sheets on the flexural face of reinforced concrete enhances the structural performance in terms of, strength, deflection, shear and durability. Reinforced concrete structures undergo distress due to various causes such as faulty design, over loading, adverse environmental effects, earthquakes, vibrations, fire, lack of maintenance. The distress in reinforced concrete creates cracks, excessive deflections, spalling of concrete etc. indicating failure of structural elements. Hence the distressed concrete structures either has to be demolished or has to be strengthened depending upon the size of structure or age of the concrete structures. As demolition and rebuilding is uneconomical, the structural strengthening becomes more optimum in terms of economy, strength and time. The performance of strengthened beam can be analysed experimentally as well as analytically. Conducting experiments needs more time, energy and uneconomical for large scale structural elements Therefore a suitable method of modelling and analysis in software like finite element method leads to optimum results. Four reinforced concrete beams have been modelled using Ansys software. One beam is control beam and other three are stressed beams at different levels. The ultimate load for the control beam is found by analysing in Ansys. This ultimate load of control beam is used to stress the other beams at different levels and to strengthen them. These beams were strengthened at different level of stress of the control beam, and they were analysed by incremental load method to get ultimate deflections. These deflections were compared with the previous experimental results of Himanshu Singhal to validate the analysis.

KEYWORDS:- Reinforced concrete beam, Retrofitting, Finite element method, GFRP, Ansys

I. INTRODUCTION

Reinforced concrete structures get detrimental effects due to various inferences. In many of the instances detrimental may occur in the form of cracks, concrete spalling, and large deflection, etc. There are various causes which are responsible for these detrimental/damage/deteriorations, such as, due to low quality of materials, corrosion of steel, fire environmental effects and accidental loading, impact loads on the structure, increased load, earthquake, chloride attack, sulphate attack etc. Thus, repair and rehabilitation has become an important challenge for the reinforced concrete structures in modern years. It is necessary that repair techniques should be Reliable, widely Available, and Maintainable. Externally bonded fiber reinforced polymers (FRP) is a new structural strengthening technique, because of their high tensile strength, lightweight, resistance to corrosion, high durability, and ease of installation. The FRP laminates has shown to be applicable to the strengthening of structural members or repairing of damaged structures for RCC such as columns, beams, slabs, etc.

Repairing of Reinforced concrete structures by bonded Fiber Reinforced Polymer composites contained in adhering Fiber reinforced Polymer laminates at the tensile face of the beam. The major goals of the strengthening are to rehabilitate the structures for life safety. Hence, the existing detrimental structures be strengthened to improve their fulfilment in the event of any natural calamities and to avoid large scale damage to life and property.

Finite Element Analysis or FEA is the simulation of a physical phenomenon using a numerical mathematic technique referred to as the Finite Element Method, or FEM The simulations used in FEA are created using a mesh of millions of smaller elements that combine to create the shape of the structure that is being assessed. Each of these small elements is subjected to calculations; with these mesh refinements combining to produce the final result of the whole structure

II. LITERATURE REVIEW

Dong-suk yang, sung-namhong, and sun-kyupark[1] In this paper author discussed the failure behaviour of reinforced concrete beams retrofitted with composite materials based on experimental set-ups.

Lux.z.j.g. teng, l.p. ye and j.j. jiang [5] For the first time introduced meso-scale finite element model for simulating the debonding behaviour of FRP-to-concrete bonded joints in simple shear tests.

Oral-büyüköztürk, tzu-yang yu[8] have highlighted that the Failures in FRP-strengthened reinforced concrete members may occur by flexural collapse of critical sections or by debonding of FRP plate from the reinforced concrete beams.

Perera and Recuero, [9] Has made a study on the adherence analysis of fiber reinforced polymer strengthened reinforced concrete beams.

Supaviriyakit et.al,[13] Has carried out analytical non-linear finite element analysis of reinforced concrete beam retrofitted with externally bonded Fiber Reinforced Polymer plates.

III. MATERIAL MODELS

A. Concrete Modelling

In the Ansys software, the material used in the finite element model for concrete predicts the failure of quasi-brittle materials. Concrete, a quasi-brittle material, is simulated with a simplified model in which the stress–strain graph decreases in the last part after the maximum stress value.

B. Geometry of the Concrete

The element used to model the concrete is as shown in the diagram Solid65, was used to model the concrete.

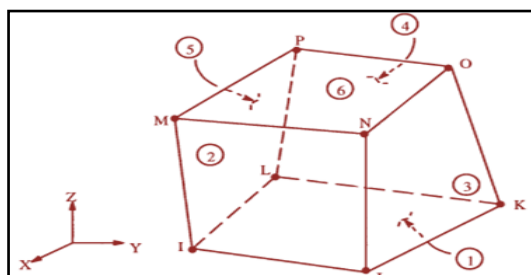


Figure. 1: Geometry of the Concrete

C. Element Properties

3 dimensional solid brick element having three degree of freedom at each node: translations in the nodal x, y and z directions. This is an isoperimetric elements integrated by Gauss integration at integration points. This element is capable of plastic deformation, cracking in three orthogonal directions, and crushing. The most important aspect of this element is the treatment of nonlinear material properties.

D. Element Interpolation function

3D solid brick element interpolation functions for all variants of the elements are given below:

$$N1 = (1/8) (1+r) (1+s) (1+t) \quad , N2 = (1/8) (1-r) (1+s) (1+t) \quad N3 = (1/8) (1-r) (1-s) (1+t) \quad N4 = (1/8) (1+r) (1-s) (1+t)$$

$$N5 = (1/8) (1+r) (1+s) (1-t) \quad , N6 = (1/8) (1-r) (1+s) (1-t) \quad N7 = (1/8) (1-r) (1-s) (1-t) \quad , N8 = (1/8) (1+r) (1-s) (1-t)$$

E. Reinforcement Modelling

Reinforcement modelling could be discrete or smeared. In this study, a discrete modelling of reinforcement has been considered. A Link8 element was used to model the steel reinforcement.

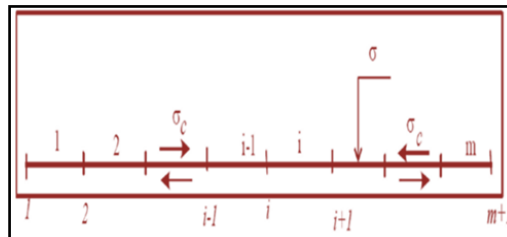


Figure 2 Reinforcement model

F. Fiber Reinforced Polymer modeling

Geometry of the FRP

The Fiber reinforced polymer can be modelled as a shell element. The shell element used the 20 nodes isoperimetric brick element is shown in fig

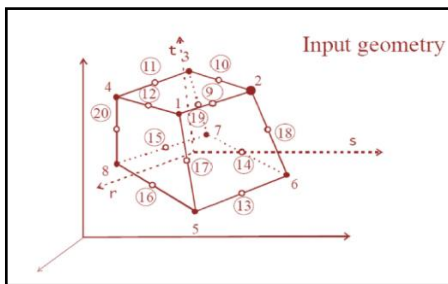


Figure 3:Input geometry of FRP

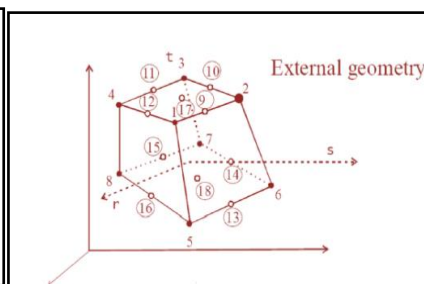


Figure 4: External geometry of FRP

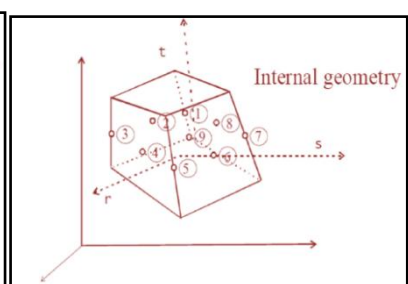


Figure 5:Internal geometry of FRP

G. Modelling Cracking of Concrete

i. Fixed Crack Model

In the fixed crack model (CERVENKA 1985, DARWIN 1974) the crack direction is given by the principal stress direction at the moment of the crack initiation. During further loading this direction is fixed and represents the material axis of the orthotropy.

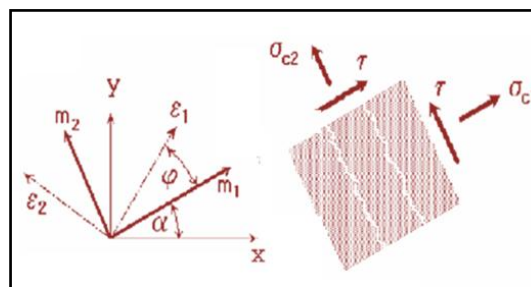


Figure 6: Fixed cracked model

ii. Rotated Crack Model

In the rotated crack model), the direction of the principal stress coincides with the direction of the principal strain. Thus, no shear strain occurs on the crack plane and only two normal stress components must be defined, as shown in Fig

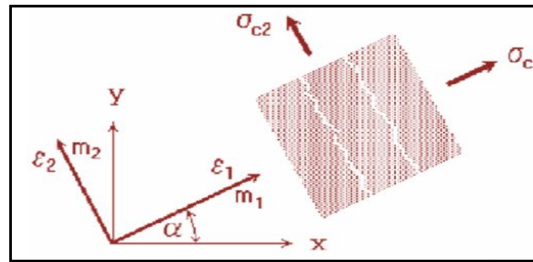


Figure. 7: Rotated Crack Model

IV. MATERIAL PROPERTIES

Concrete, reinforcement steel, steel plates, and fiber plates are used to model the RCC beam and fiber reinforced concrete beam. The specification and the properties of these materials are as under.

i. Concrete

In ANSYS, concrete material is taken as a 3D nonlinear Cementitious. The physical properties required for this material are given in table .

As per IS 456 2000 Elastic modulus of concrete $E_c=5000\sqrt{f_c}ckMPa$ and tensile strength $f_{cr}=0.7\sqrt{f_c}ckMPa$

Elastic Modulus	: 22360. 67 MPa	Coefficient of Thermal Expansion	:1E-05 1/K
Poisson Ratio	: 0.2	Reduction of Compressive Strength	:0.8
Tensile Strength	: 3.73 MPa	Close shear transfer coefficient	:0.9
Compressive Strength	:20 MPa	Open shear transfer co efficient	:0.1
Specific Material weight	:2400 Kn/cu.m		

ii. Reinforcement Bars

HYSD steel of grade Fe-415 of 10mm, 8mm and 6mm diameter were used as longitudinal steel. 10mm diameter bars are used as tension reinforcement and 8mm diameter bars are used as compression steel and 6mm diameter bars are used as shear reinforcement.

Elastic modulus : 200000 MPa

Yield Strength

10mm diameter bar : 445.55 MPa

8mm diameter bar :559.50MPa

Specific Material weight :0.0785 MN/mE+3

Coefficient of Thermal Expansion: 1.2E-05 1/

6mm diameter bar : 442.42 MPa

iii. Steel Plate

The function of the steel plate in the ANSYS is for support and for loading. Here, the property of steel plate is same as the reinforcement bar except its yield strength. The HYSD steel of gradeFe-415 was used for steel plate.

iv. Epoxy

Embracesaturant was used as a epoxy in the analysis. The material properties are taken from the “Watson Bowman Acme corp.” company paper. The material properties of epoxy are shown

Elastic Modulus	3035 Mpa	Hardness Modulus	0
Poisson Ratio	0.4	Specific Material weight	0.00983MN/Cu.m
Yield strength	54 Mpa	Co efficient of Thermal Expansion	5.75E ⁵ 1/K

v. Glass Fiber Reinforced Polymer

MbraceG sheet EU-900 unidirectional Glass Fiber sheet is used in retrofitting .The properties which are used in the modelling are taken from Mbrace product data as shown

Stress, strain (0,0),(0.005,390),(0.016,690) Mpa Specific Material weight 0.026 MN/cu.m
Co efficient of Thermal Expansion 5E-⁶ 1/K

V. FE MODELLING OF RCC BEAM IN ANSYS

According to the design, the dimensions of the beam are 4.1m X 0.127m X 0.227 m depth. Due to the symmetric of the beam, half of the beam is modelled, analysed and applied to the whole beam. Four beams were modelled in Ansys. One is the control beam and the remaining three are stressed retrofitted beams with use of GFRP at 60%,75%,and 90% of the ultimate load of the control beam. Firstly non linear FE modelling of the control beam was modelled in Ansys and analysed it to get the results. From the help of control beam results, other beams were modelled and analysed.

VI. RESULTS AND DISCUSSION

The analytical results of control RCC beams and the stressed retrofitted RCC beams to 60%, 75% & 90% of the ultimate load with GFRP under the static two point loads have been analyzed using ANSYS. These results are further compared with experimental results and analytical results.

A. FEM Analysis of the Control Beam

In the analysis, half of the beam is modelled by finite element method using ANSYS software as shown in fig 8. And results are obtained for full length of the beam. In ANSYS, load can be applied in terms of prescribed deformation and results can be seen by post processing. The beam size and the material properties which are used in the modelling of the control beam are same as of Himanshu Singhal thesis report. The characteristic of the control beam is studied at each step. The ultimate load of controlled beam was found out to be 16.5 kN and the ultimate deflection was found out to be 46.9mm.The results are tabulated as shown in table 1and compared with graph 1.

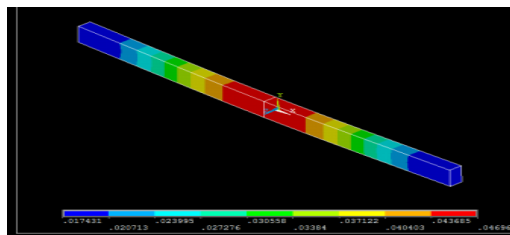


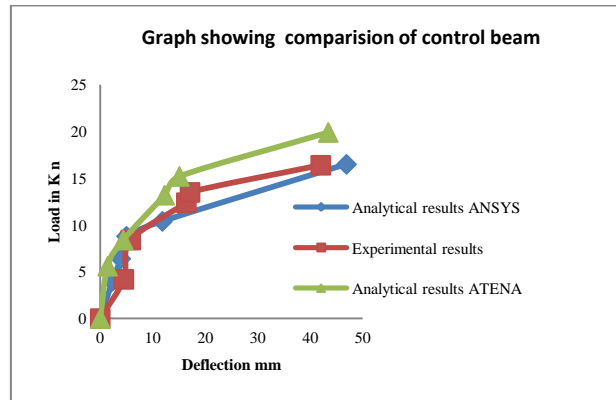
Figure .8: Model of the beam in Ansys

Table . 1: Results for control beam

SI no	Analytical results ANSYS		Experimental results		Analytical results ATENA	
	Load (kN)	Deflection mm	Load (kN)	Deflection mm	Load (kN)	Deflection mm
1	0.00	0.00	0.00	0.00	0.00	0.00
2	4.00	2.34	4.20	4.50	5.60	1.42
3	6.40	3.91	8.40	5.80	8.40	4.40
4	8.80	5.00	12.40	16.40	13.20	12.26
5	10.40	11.84	13.50	17.10	15.20	15.08
6	16.50	46.90	16.40	42.00	19.90	43.43

B. Analysis of Control beam and stressed beam

As did in the experiment, the three beams were stressed at different percentage of stress i.e. 60%, 75%, and 90%of the ultimate load carried by the control beams and then these beams are strengthened by using GFRP sheets. The analytical results obtained from these stressed retrofitted beam, are then compared with experimental results available and analysed



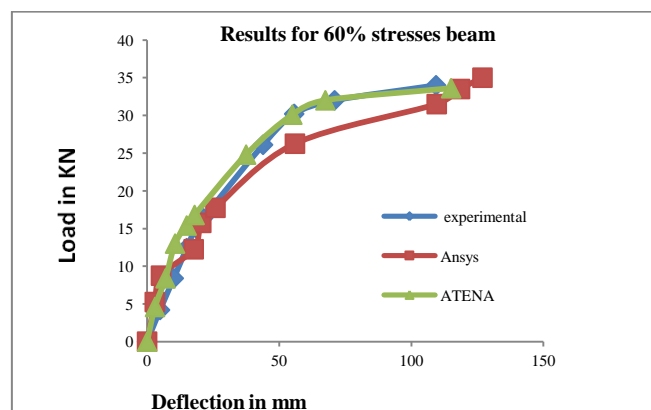
Graph. 1: Results for control beam

C. 60% Stressed Retrofitted Beam

Graph 2 and table 2 shows the comparison between the experimental results and the analytical results for 60% stressed retrofitted beam. From the table and graph, the results of the experimental and analytical are close to each other. The ultimate load and the ultimate deflection are 35 kN and 126.97 mm

Table. 2: Results for 60% stresses beam

S I N O	Analytical results ANSYS		Experimental results		Analytical results ATENA	
	Load (kN)	Deflection mm	Load (kN)	Deflection mm	Load (kN)	Deflection mm
1	0.00	0.00	0.00	0.00	0.00	0.00
2	5.25	2.93	4.20	4.90	4.60	3.00
3	8.75	5.27	8.40	10.10	8.40	7.00
4	12.25	17.64	12.40	15.00	13.00	10.60
5	15.75	20.50	15.10	18.40	15.40	15.00
6	17.75	25.89	16.40	21.25	16.80	18.00
7	26.25	55.98	26.10	43.90	24.80	37.50
8	31.50	109.65	30.20	55.70	30.10	55.00
9	33.50	118.47	32.00	71.00	32.00	67.50
10	35.00	126.97	34.00	109.40	33.60	115.10



Graph. 2: Results for 60% stresses beam

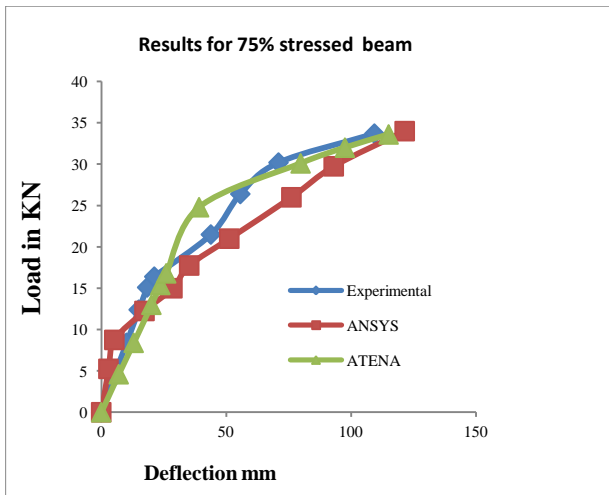


D. 75% Stressed Retrofitted Beam

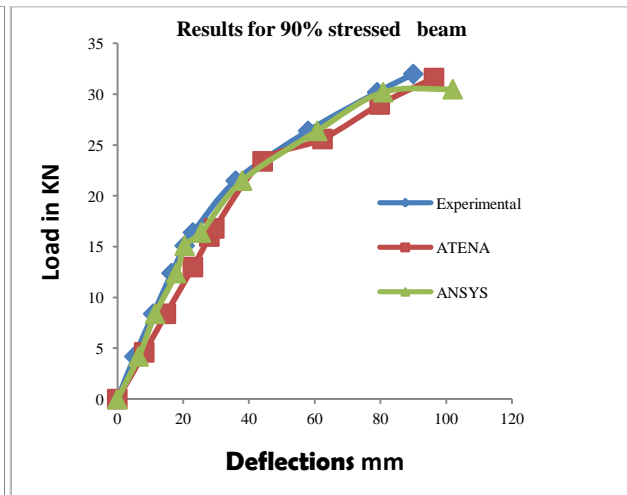
Graph 3 and table 3 shows the comparison between the experimental results and the analytical results. From the above table and graph, the analytical results and experimental results are almost same. The ultimate load and the ultimate deflection from the analytical result for Ansys are 34 kN and 121.54 mm respectively whereas the ultimate load and the ultimate deflection from the experimental result was 33.7 kN and 109.4 mm respectively.

Table 3 Results for 75% Stressed Beam

S I N O	Analytical results ANSYS		Experimental results		Analytical results ATENA	
	Load (kN)	Deflection mm	Load (kN)	Deflection mm	Load (kN)	Deflection mm
1	0.00	0.00	0.00	0.00	0.00	0.00
2	5.25	2.89	4.20	4.90	4.60	7.00
3	8.75	5.25	8.40	10.10	8.40	13.00
4	12.25	17.33	12.40	15.00	13.00	20.10
5	15.00	28.50	15.10	18.40	15.40	23.66
6	17.75	35.23	16.40	21.25	16.80	26.02
7	21.00	51.28	21.50	43.90	24.80	39.19
8	26.00	76.17	26.40	55.70	30.10	79.85
9	29.75	93.11	30.20	71.00	32.00	97.60
10	34.00	121.54	33.70	109.40	33.60	114.10



Graph. 3: Results for 75% stressed beam



Graph. 4: Results for 90% stresses beam

E. 90% Stressed Retrofitted Beam

Graph 4 and table 4 shows the comparison between the experimental results and the analytical results. From the above data it's observed that the ultimate load for analytical and experiment beam are almost same. The ultimate load and the ultimate deflection from the analytical result is 31 kN and 83.95 mm respectively whereas the ultimate load and the ultimate deflection from the experimental result was 32 kN and 90 mm respectively.

Table 4 Results for the 90% Stressed Beam

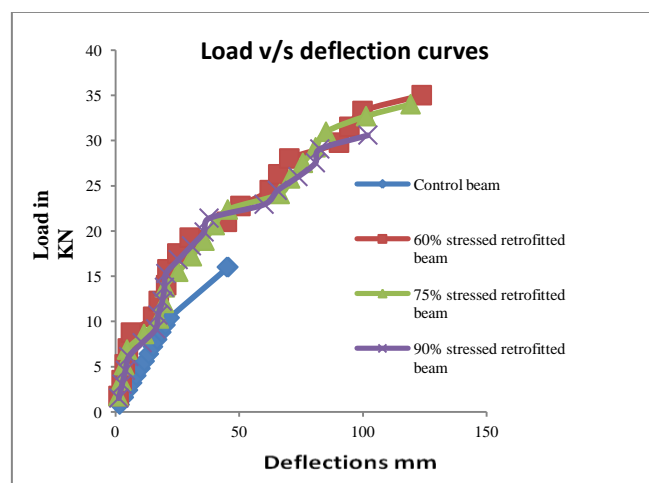
SI NO	Analytical results ANSYS		Experimental results		Analytical results ATENA	
	Load (kN)	Deflection mm	Load (kN)	Deflection mm	Load (kN)	Deflection mm
1	0.00	0.00	0.00	0.00	0.00	0.00
2	3.10	1.72	4.20	5.20	4.60	8.20
3	9.30	5.66	8.40	10.90	8.40	14.77
4	12.40	17.89	12.40	16.50	13.00	23.00
5	15.50	20.59	15.10	20.50	16.00	28.00
6	16.40	25.47	16.40	23.00	16.80	29.50
7	21.70	42.54	21.50	36.00	23.40	44.20
8	24.80	56.05	26.40	58.00	25.60	62.40
9	27.90	69.81	30.20	79.00	29.00	79.80
10	31.00	83.95	32.00	90.00	31.60	96.30

F. Final Analysis Retrofitted Beam at 60%, 75% and 90%.

Load deflection curve for control beam and the stressed retrofitted beam has been shown below in graph 5 Control beam fails at a load of 16.5 kN with an ultimate deflection of 46.9mm whereas retrofitted stressed beam at 60% fails at a load of 35 kN with an ultimate deflection of 126.94mm at the centre showing that there is considerable increase in load carrying capacity.

Control beam fails at a load of 16.5 kN with an ultimate deflection of 46.9mm whereas retrofitted stressed beam at 75% fails at a load of 34 kN with an ultimate deflection of 120.19 mm at the centre showing that there is considerable increase in load carrying capacity.

1. Control beam fails at a load of 16.5 kN with an ultimate deflection of 46.9mm whereas retrofitted stressed beam at 90% fails at a load of 31 kN with an ultimate deflection of 83.95mm at the centre showing that there is considerable increase in load carrying capacity.
2. In comparison to control beam, the beam stressed at 90% of the ultimate load shows linearly variation up to a higher level. This implies that at higher load levels, the load is entirely taken by GFRP



Graph. 5: Load versus deflection curve

VI. CONCLUSIONS

The main conclusions drawn are summarized below:

1. The general behavior of the finite element models represented by the load-deflection curves show good agreement with the experimental data from the full-scale beam tests.
2. Finite element analysis using non-linear models of cracked concrete, steel bars and FRP is used to predict the behavior of FRP retrofitted beam. It is verified that the finite element analysis can accurately predict the load deformation behavior process similar to the experiment
3. The Analytical result of the control beam was same as of the Experimental results.
4. The Analytical results of the stressed retrofitted beam at 60%, 75% & 90% of the ultimate load of the control beam show good agreement as the Experimental results.
5. In the analytical results, the Stressed Retrofitted beam at 60%, 75% and 90% of the ultimate load of the control beam was increased by 118.75, 106.06% and 87.8% respectively as compared to the control beam.

REFERENCES

1. Dong-suk yang, sung-namhong, and sun-kyu park (2006), Experimental Observation on Bond- Slip Behavior between Concrete and CFRP Plate.
2. DatDuthinh and Monica Starnes (2001), Strengthening of Reinforced Concrete Beams with Carbon FRP. Journal of Composites in Constructions, pg 493-498.
3. ErdoganMadenci Ibrahim Guven(2005),The Finite Element Method and Applications in Engineering using ansys
4. Ferracuti B, M. Savoia, C. Mazzotti (2006),A Numerical Model for FRP Concrete Delamination. Journal of Composites, Vol part b 37, pg 356-364.
5. Lu x.z , j.g. Teng, L.P. ye and J J. Jiang (2004),Bond-Slip Models For FRP Sheet/Plate-to-Concrete Interfaces
6. Mostofinejad D and S. B. Talaeitaba (2006),Finite Element Modeling of RC Connections Strengthened with FRP Laminates. Iranian Journal of Science and Technology, Transaction B, Engineering, vol. 30, pg 21-30.
7. Giuseppe Simonelli (2005),Finite Element Analysis of RC Beams Retrofitted with Fibre Reinforced Polymers.
8. Rral Büyüköztürk, tzu-yang yu (2006), Understanding and Assessment of Debonding Failures in FRP Concrete Systems.
9. Perera r, a. recuero, a. de diego, c. lopez (2004),Adherence Analysis of Fiber reinforced Polymer Strengthened RC Beams. Journal of Computers and Structures, vol 82, pg 1865-1873.
10. Robert D .Cook, Davids.malkus ,Michael e. Plesha ,robert j. witt (2001),Concept and Applications of Finite Element Analysis
11. Santhakumar R. E. Chandrasekaran and R. Rhanaraj (2004),Analysis of Retrofitted Reinforced Concrete Shear Beams using Carbon Fiber Composites. Electronic Journal of Structural Engineering, vol 4, pg 66-74.
12. Songklanakarin J (2007),Finite Element Analysis of FRP Strengthened RC Beams. Science Technology, vol 26(4), pg 497-507
13. Supaviriyakit T, Pornpongsaroj P, and Pimanmas a (2004), Finite Element analysis of FRP Strengthened RC Beams. . Science technology, vol 26(4), pg 497-507.
14. Yang Z.J, J.F. Chen and D. Proverbs (2003), Finite Element Modeling of Concrete Cover Separation Failure in FRP Plated RC Beams.
15. Bhavikatti.S,Finite Element Analysis
16. Darlyl.logon, Finite Element Method
17. IS456 : 2000- Plain and Reinforced Concrete -Code of Practice