Non-Linear Finite Element Analysis of FRP Strengthened RC Beams Using ANSYS

M. AMMAR MAHDI[1], CHEN ZHIJUN[2] & A. GHASSAN MUNDHER[3]

[1] PG Scholar, Dept. of Bridge and Tunnel Engineering, Huazhong University of Science & Technology. Email: ammar_mahdie@hust.edu.cn

[2] Professor, Dept. of Bridge and Tunnel Engineering, Huazhong University of Science & Technology.

[3] PG Scholar, Dept. of Bridge and Tunnel Engineering, Huazhong University of Science & Technology.

Abstract:

Most of the reinforced concrete (RC) structures get damaged during earthquakes and they need replacements. Presently several studies have been conducted to investigate the flexural strengths of reinforced concrete members with fiber reinforced composites. This paper presents the finite element analysis of control RC beam and strengthened beams with carbon fiber reinforced polymer (CFRP) composite sheets carried out using ANSYS software. In this study, a control beam was modelled, the results were analyzed, and then strengthened beams were modelled and analyzed. The results of the control and strengthened beams were compared with the experimental results. It was observed that the results of the strengthened beams are in close agreement with the experimental results.

Keywords—RC Reinforced Concrete, Strengthened beams, CFRP Carbon Fibre Reinforced Polymer, Numerical (ANSYS).

1. INTRODUCTION

In general FRP composite materials have good physical and mechanical properties like high strength, good damping capacity, lightweight and easy to fabricate and also high corrosion-resistant. Moreover, application-wise the composite materials have the potential of reducing costs in construction, operation and development while improving structural reliability and enhancing safety [1]. Because of these unique specifications, they are widely used in high technology structural applications [2-4].

Much fiber reinforced polymer sheets are available in the market for strengthening reinforced concrete Structures [5]. The carbon fiber composites are the frequently used system by many researchers and this retrofitting method finds wide field applications. Carbon fibers known for their high tensile strength, glass fibers for their relatively low modulus of elasticity and low-cost and the Kevlar fibers for their high damping capacity [6-7].

This paper presents the numerical study to simulate the behaviour of control R.C beams and CFRP strengthened reinforced concrete beams using ANSYS through nonlinear response and load up to failure.

The following are the main objectives of the present study:

a) To model the reinforced concrete beams called as control beams and the strengthened beams with different reinforcement using FEM.

b) To determine analytically the load deflection curve Of control beam and strengthened beams.

c) To compare the results of control beams and the strengthened beams and also compare the analytical results of finite element analysis from ANSYS with the experimental result available in literatures.

2. LITERATURE REVIEW

The use of FRP (Fiber Reinforced Plastics) plates for strengthening and repairing of RC structures represents an interesting alternative for steel plates. FRP materials are lighter than steel. They present a high strength to mass ratio. They are corrosion-resistant and are generally resistant to chemical attacks. This technique has been widely investigated, and several examples of existing structures retrofitted using epoxy-bonded composite materials can be found in the literature.

Bodin , et al., (2002)[8] , proposed a non-linear finite element (FE) analysis in order to complete the experimental analysis of the flexural behavior of the beams. Elasto-plastic behavior was assumed for reinforced concrete and interface elements were used to model the steel concrete bond and the adhesive. A numerical analysis also included simulations on pre-cracked beams.

Yang , et al., (2009)[9] , tested 13 FRP-strengthened reinforced concrete beams in flexure and analyzed using the finite element method. The various variables included bonding or no bonding of the FRP, the anchorage system. All the beams were subjected to three-point and four-point bending tests under deflection control, with loading, deflection and failure modes recorded to the point of failure. A nonlinear finite element analysis of the tested beams was also performed using the DIANA software; this analysis accounted for the nonlinear concrete material behavior, reinforcement, and an interfacial bond-slip model between the concrete and CFRP Plates.

Kenneth , et al., (2011)[10] , presented the non-linear finite element method of reinforced concrete members externally strengthened with FRP which includes the flexure and shear strengthening of beams and FRP strengthening of two-way slabs.

3. FINITE ELEMENT METHOD & MODELING

The finite element method is a numerical analysis technique for obtaining approximate solutions to a wide variety of engineering problems. ANSYS is a general purpose finite element modelling package for numerically solving a wide variety of problems which include static/dynamic structural analysis (both linear and nonlinear), heat transfer and fluid problems, as well as acoustic and electro-magnetic problems[11-12]. The Rectangular RC beams with tensile reinforcement have been analyzed using a finite element (FE) model in ANSYS. Here, a nonlinear analysis is considered throughout the study by assuming that there is a perfect bonding between concrete and steel reinforcement, and reinforced concrete with CFRP sheets. The results from the numerical simulation are used for comparison with the results from the experiment to achieve the principle objective of studying the flexural capacity behaviour of reinforced beams strengthened with CFRP sheets subjected to distributed line loading. The experimental investigation (M. IMAM, A. Tahwia, A. Elagamy, and M. Yousef et al., 2004 [13]), including eight commercial material models was performed using this numerical simulation study. ANSYS (version 12.1) has been chosen for the purpose of analyzing RC beams with and without external CFRP strengthening in this study due to its flexibility in geometry and materials modelling. The details of control and strengthened beams models are presented. The beam is a simply supported beam with a total length of 2300 mm, and a clear span of 2000 mm. The beam has a rectangular cross section with 120 mm width and 250 mm height and a concrete cover of 25 mm was assumed. The beam was subjected to two concentrated static loads, spaced 500 mm. experimental groups consist of two groups as BF1 and BF2. Group BF1 consists of four beams with 2φ12 lower reinforcement, 2φ10 upper reinforcement, and 10φ6 /m` stirrups. Group BF2 consists of four beams with 2φ16 lower reinforcement, 2φ10 upper reinforcement, and 10φ6 /m` stirrups , High grade steel with yield strength of 400 MPa was used for longitudinal reinforcement while mild steel (fy=245 MPa) was used for stirrups for all beams. Fig. (1) showed the strengthening modes for these beams and summarized in Table (1) which may be distinguished as:

- BF1-1 was defined as control beam or reference beam without any strengthening system.
- BF1-2 was strengthened by external single flat layer of CFRP fabrics with cross section of 120×0.13 mm and 1500 mm total length. The CFRP sheet was applied on the bottom of the beam (tension face).
- BF1-3 was externally strengthened same as BF1-2 but the longitudinal CFRP sheet was double layers instead of single layer.
- BF1-4 was externally strengthened by single layer of longitudinal CFRP fabrics with cross section of 240×0.13 mm. The breadth of CFRP was selected to completely cover the beam bottom (tension face) in addition to 60 mm upward on each side of the beam.
- BF2-1 was defined as control beam or reference beam without any strengthening system.
- BF2-2 was strengthened by external double-flat layers of CFRP fabrics with cross section of 110×0.13 mm and 1500 mm total length. The CFRP sheet was applied on the bottom of the beam (tension face).
- BF2-3 was externally strengthened by single layer of longitudinal CFRP fabrics with cross section of 220×0.13 mm. The breadth of CFRP was selected to completely cover the beam bottom (tension face) in addition to 50 mm upward on each side of the beam.
- BF2-4 was externally strengthened same as BF 2-3 but the longitudinal CFRP sheet was double layers instead of single layer.

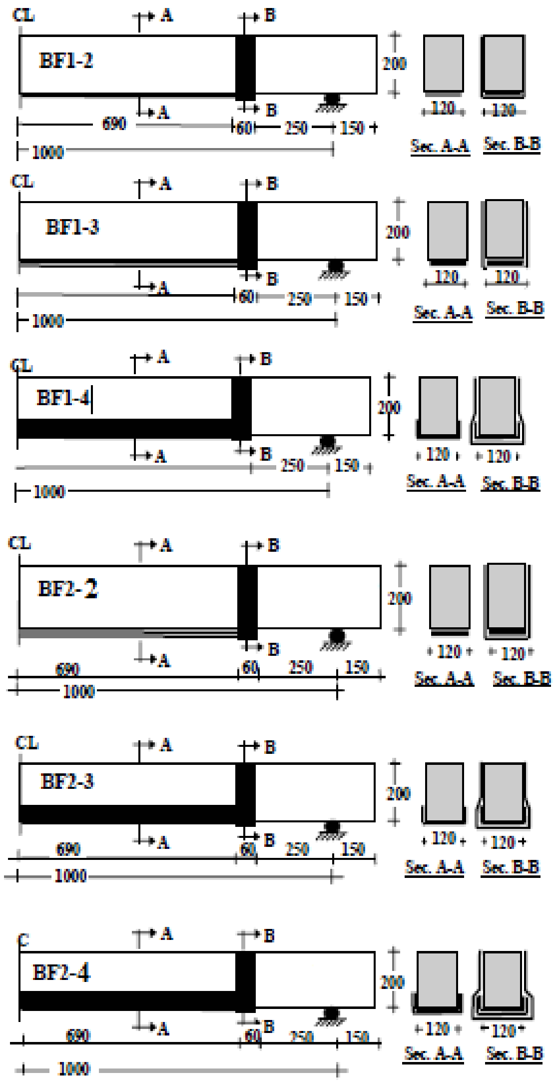

Fig (1) Strengthening Modes of Beams

Table (1) Variables of models

Group No.	Beam No.	Strengthening modes
1	BF1-1	Reference
	BF1-2	Single flat layer
	BF1-3	Double flat layers
	BF1-4	U-shape single layer
2	BF2-1	Reference
	BF2-2	Double flat layers
	BF2-3	U-shape single layer
	BF2-4	U-shape double layers

The following has been used for the materials idealization:

a) *Concrete Idealization*

SOLID65 was used to model the concrete. the geometry, node locations, and the coordinate system for this element are shown in figure (2).

b) *Steel Reinforcement*

A Link 8 element was used to model steel reinforcement. The element is a uniaxial tension-compression element with three degrees of freedom at each node. The geometry, node locations, and the coordinate system for this element are shown in figure (3).

c) *Steel Plate*

SOLID 45 is used for 3-D modelling of solid structures. It was used to model Steel plates were added at support and point of loading locations .The geometry and node location for this element type are shown in Figure (4).

d) *CFRP sheets*

SHELL41 A four node element was used to model CFRP strips which is a 3-D element having membrane (in-plane) stiffness but no bending (out-of-plane) stiffness. The geometry and node location of this element type are shown in Figure (5).

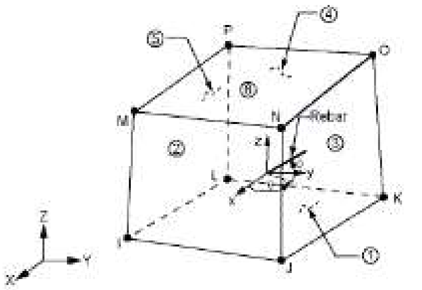

FIGURE 2: SOLID 65 GEOMETRY (ANSYS Help, 2012) [14]

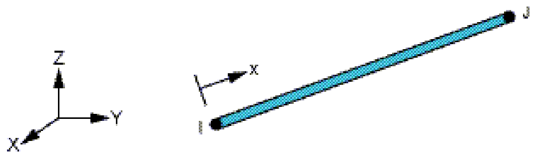

FIGURE 3: LINK 8 GEOMETRY

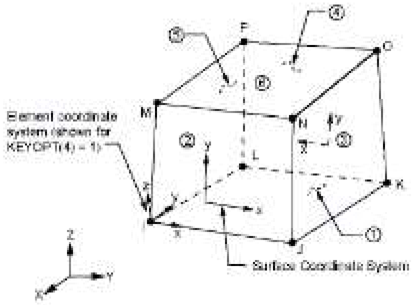

FIGURE 4: SOLID 45 GEOMETRY

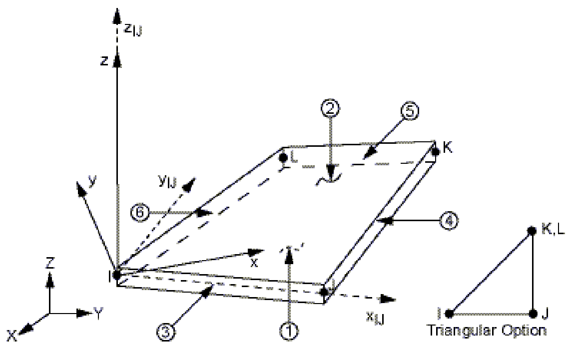

FIGURE 5: SHELL 41 GEOMETRY

4. RESULTS AND DISCUSSION

This chapter presents the results of Finite Element analysis of two control RC beams, sex RC beams

p-ISSN: 2348-6848
e-ISSN: 2348-795X
Volume 04 Issue 03
March 2017

strengthened with CFRP bonded at the side and bottom face. Finite element analysis (using ANSYS computer program version 12.1) of RC beams under the static incremental loads has been performed in the present work. Subsequently these results are compared with experimental results. The following comparisons are made with regards to deflection values as well as the detailed overall behaviour of the of the beams.

a) Loading And Boundary Conditions

In the experiment, the bearing plate of loading and support dimensions are (50mm x 500mm). A 25 mm thick steel plate, modelled using Solid 45 elements, is added at the support and point of loading locations in order to avoid stress concentration problems shown in figure (6). This provides a more even stress distribution over the support and point of loading areas. Figure (7) shows the distribution of the applied load at nodes.

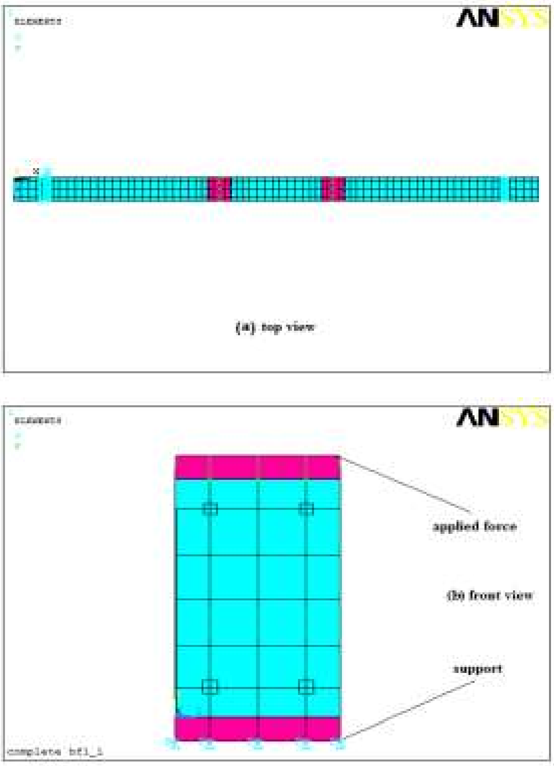

Figure 6: Applied load and Boundary conditions (a) Top view (b) Front view

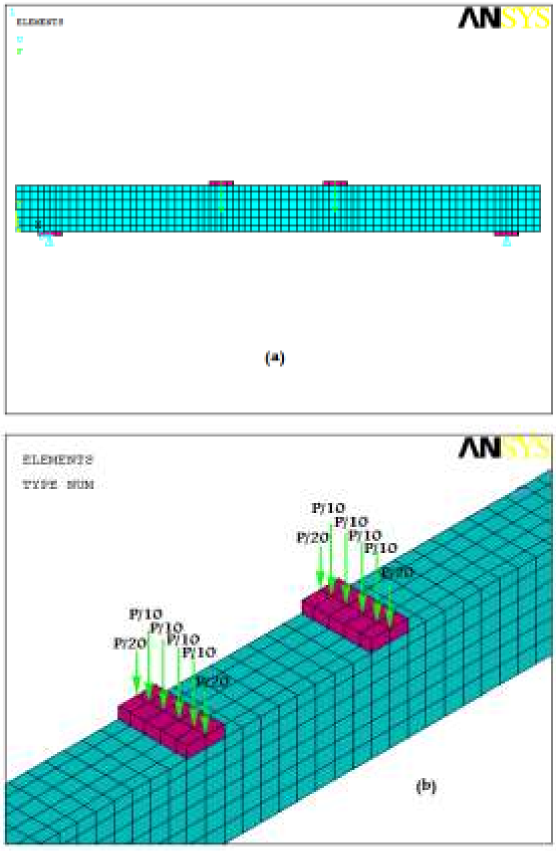

Figure 7: Distribution of Applied Load at Nodes (a) Side view (b) Isometric view

b) Results Of Finite Element Analysis

The numerical load-deflection curves are compared with the experimental results, shown in Figure (8) to (15). In general, the load deflection curves for the beams from the finite element analyses agree quite well with the experimental data. The finite element load deflection curves in the linear range are somewhat stiffer than the experimental plots. After first cracking, the stiffness of the finite element models is again higher than that of the experimental beams. Several factors may cause the higher stiffness in the finite element models. The bond between the concrete and steel reinforcing is assumed to be perfect (no slip) in the finite element analyses, but for the actual beams the assumption would not be true slip occurs, therefore the composite action between the concrete and steel reinforcing is lost in the actual beams. Also the micro-cracks produced by drying shrinkage and handling are present in the concrete to some degree. These would reduce the stiffness of the actual beams, while the finite element models do not include micro-cracks due to factors that are not incorporated into the models. Thus, the overall stiffness of the experimental beams is expected to be lower than for the finite element models.

The figures (16, 17, 18) shows the reinforcement for the groups BF1 of beams, Variation of deflection along BF1-1 and Strengthened Beam by U-shape of CFRP Sheets for beam BF1-4 respectively.

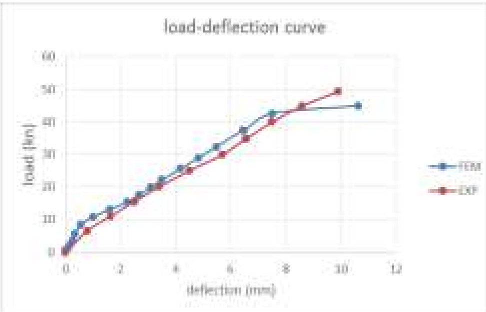

Figure 8: Load-deflection curve for control beam BF 1-1

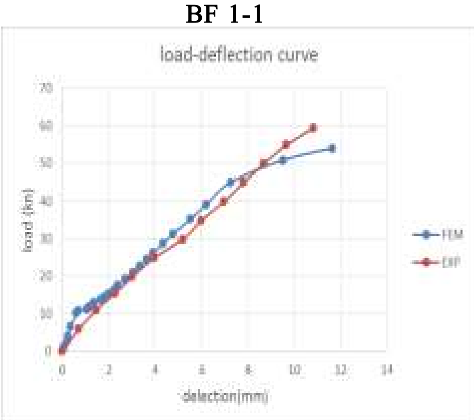

Figure 9 : Load-deflection curve for beam BF1-2

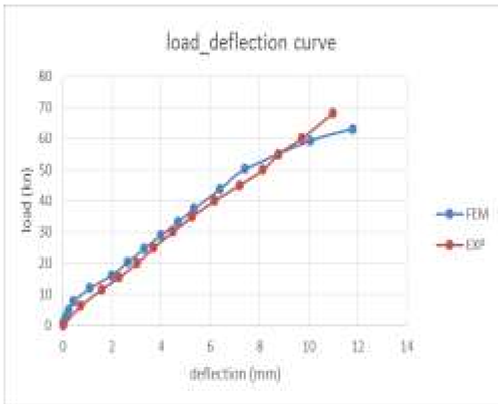

Figure 10: Load-deflection curve for beam BF 1-3

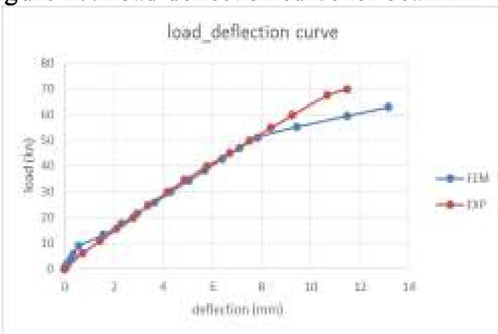

Figure 11: Load-deflection curve for beam BF 1-4

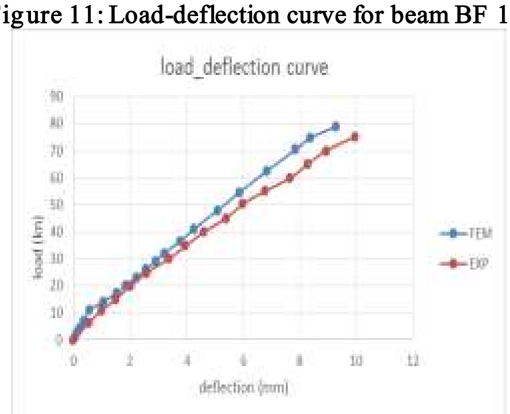

Figure 12: Load-deflection curve for beam BF 2-1

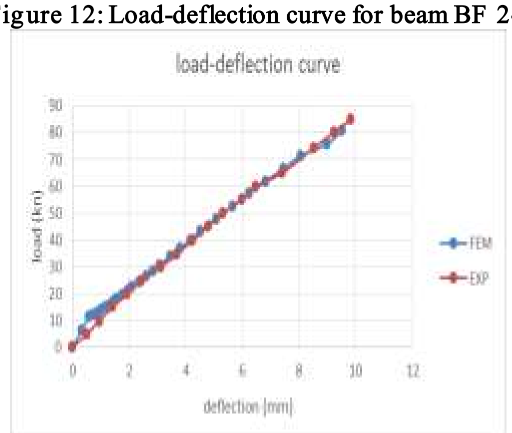

Figure 13: Load-deflection curve for beam BF 2-2

International Journal of Research
Available at https://edupediapublications.org/journals
p-ISSN: 2348-6848
e-ISSN: 2348-795X
Volume 04 Issue 03
March 2017

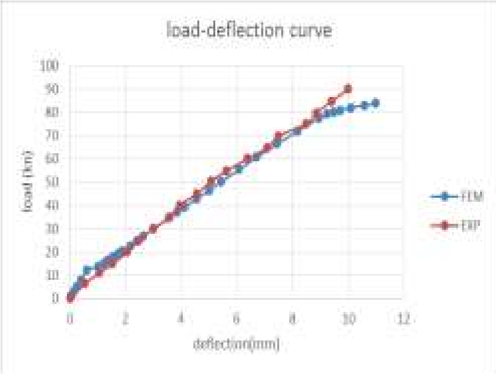

Figure 14: Load-deflection curve for beam BF 2-3

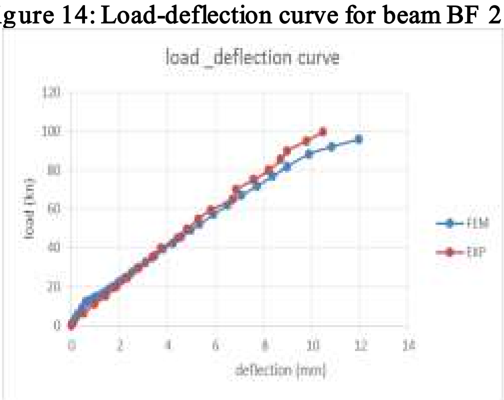

Figure 15: Load-deflection curve for beam BF 2-4

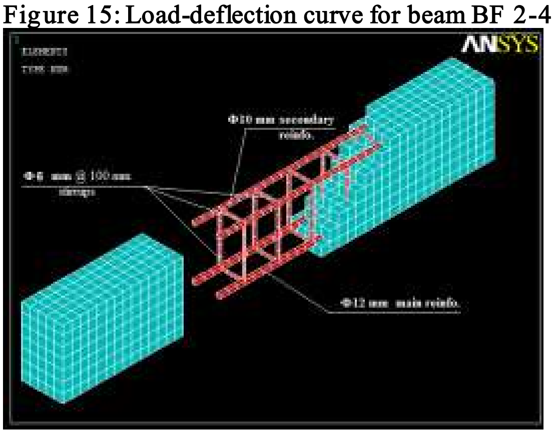

Figure (16): Distribution of reinforcement for all beams in group BF1

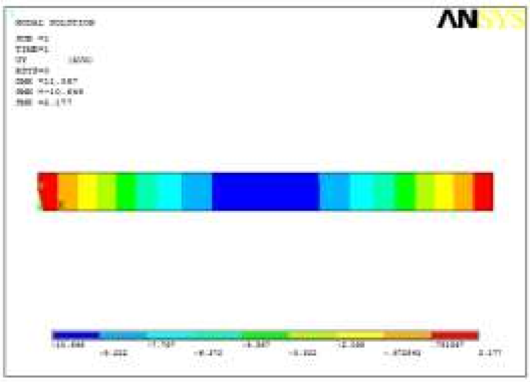

Figure (17):Variation of deflection along control beam BF1-1 at ultimate load

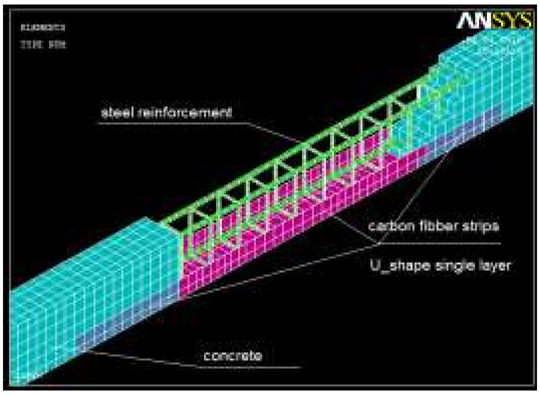

Figure 18: Strengthened Beam by Single Layer U-shape of CFRP Sheets for beam BF1-4

c) *Crack Pattern*

The ANSYS program records a crack pattern at each applied load step. Figure19 shows evolutions of crack patterns developing for control beam at the last loading step. ANSYS program displays circles at locations of cracking or crushing in concrete elements. Cracking is shown with a circle outline in the plane of the crack, and crushing is shown with an octahedron outline. The first crack at an integration point is shown with a red circle outline, the second crack with a green outline, and the third crack with a blue outline [14].

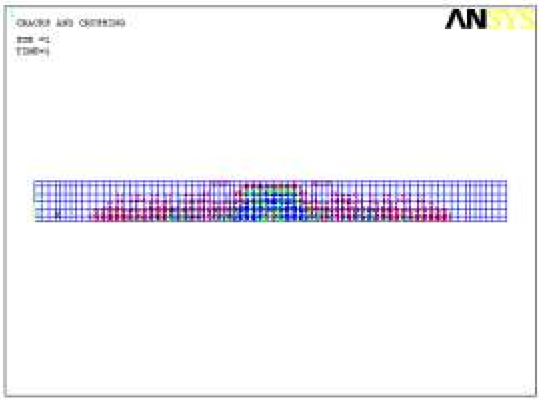

Figure 19: Crack Patterns of Control Beam

d) *Ultimate Loads And Ultimate Deflection*

A comparison between the ultimate loads of the tested beams and ultimate loads from finite element analysis is shown in Table (2), and comparison between the ultimate deflections of the tested beams and ultimate deflections from finite element analysis is shown in Table (3).

Table (2) Comparison between Experimental and Finite Element Ultimate load

beam symbol	(Pu)exp. (kN)	(Pu) FE. (kN)	Pu FE / Pu Exp
BF1-1	50	45.12	0.9
BF1-2	60	54.09	0.9
BF1-3	68	63.23	0.93
BF1-4	70	63.3	0.9
BF2-1	76	78.93	1.04
BF2-2	85	82.4	0.969
BF2-3	88	84.52	0.96
BF2-4	100	96	0.96

Table (3): Comparison between Experimental and Finite Element Ultimate Deflection at mid span

beam symbol	(Δu) exp. (mm)	(Δu) FE. (mm)	(Δu) FE / (Δu) Exp
BF1-1	9.61	10.63	1.106
BF1-2	10.56	11.68	1.106
BF1-3	11	11.81	1.07
BF1-4	11.1	13.2	1.189
BF2-1	9.97	9	0.9
BF2-2	9.85	9.54	0.968
BF2-3	10	11	1.1
BF2-4	10.4	11.96	1.15

5. CONCLUSION

In this paper, nonlinear finite element analyses of rectangular reinforced concrete beams strengthened by CFRP are performed. Based on the numerical results, the following conclusions may be drawn:

- The ultimate load carrying capacity of all the strengthened beams is higher when compared to the control beams.
- CFRP fabric properly bonded to the tension face of RC beams can enhance the flexural strength substantially. The strengthened beam BF1-2 exhibit an increase in flexural strength of 20 percent for single layer.
- The load carrying capacity of the strengthened beam BF1-4 which was strengthened using a single layer U-wrap CFRP was found to be higher when compared to BF1-2 beam which was strengthened using a CFRP sheets at the tension face of beam.
- The load carrying capacity of the strengthened beam BF2-4 which was strengthened using a double layer U-wrap CFRP was found to be higher when compared to strengthened beam BF2-3 which was strengthened using a single layer.
- The numerical solution was adopted to evaluate the ultimate strength of the reinforced concrete beams reinforced with FRP laminates in simple, cheap and rapid way compared with experimental full scale test.
- The general behaviors of the finite element models show good agreement with observations and data from the experimental full-scale beam tests. Therefore, the simulation of FRP beams can be done using ANSYS.

REFERENCES

[1] Hermann, Thomas M., 2006. Fabrication, Testing and Analysis of Anisotropic Carbon/ Glass Hybrid Composites. Technical Report, Sandia National Laboratories, 1: 12-24.

[2] Madenci, E. and I. Guven, 2006. The Finite Element Method and Applications in Engineering Using Ansys. Springer, pp: 397-403.

[3] Reddy, R., B.S. Reddy, N. Reddy and S. Surisetty,2012. Prediction of natural frequency of laminated composite plates using artificial neural networks, Engineering, 4(6): 329-337.

[4] Schubbe, J.J. and S. Mall, 1999. Investigation of a cracked thick aluminum panel repaired with a bonded composite patch. Engineering Fracture Mechanics, 63: 305-323.

[5] Choudhary, S.S. and V.B. Tungikar, 2011. A simple finite element for nonlinear analysis of composite plates, International Journal of Engineering Science and Technology, 3(6): 4897-4907.

[6] Saadatmanesh, H. and M.R. Ehsani, 1991. RC beams strengthened with GRFP plates, I: Experimental study, Journal of Structural Engineering, 117(11): 3417-3333.

[7] Saadatmanesh, H., M.R Ehsani and L. Jin, 1997. RC beams strengthened with FRP Plates II: analysis and parametric study, Journal of Structural Engineering, 117(11): 3434-3455.

[8] Bodin F.B, David E. and Ragneau E. “Finite element modelling of flexural behavior of externally bonded CFRP reinforced concrete structures”, Engineering Structures 24, (2002) pp. 1423–1429.

[9] Yang D., Park S.and Neale K.W. “Flexural Behaviour of Reinforced Concrete Beams Strengthened with Pre-stressed Carbon Composites”, Composite Structures, 88 (2009),pp. 497–508.

[10] Kenneth,W. N., Ahmed, G., Hussien, M. A. B.,walid , E. E., and Usama, A. E. (2011) "Approaches for finite element simulations of FRP-strengthened concrete beams and slabs." Architecture, 4.4: 59-72.

[11] Hii, Adrian KY, and Riadh Al-Mahaidi. "An experimental and numerical investigation on torsional strengthening of solid and box-section RC beams using CFRP laminates." Composite structures 75.1 (2006): 213-221.

[12] De Witte, F. C., and W. P. Kikstra. "DIANA Finite Element Analysis, User's Manual: Release 8.1." TNO Building and Construction Research, Delft, the Netherlands (2002).

[13] IMAM, M., A. Tahwia, A. Elagamy, and M. Yousef. "Behavior of Reinforced Concrete Beams Strengthened With Carbon Fiber Strips."

[14] ANSYS, CFX. "Help Manual 12.1."