# NUMERICAL MODELING ON BEHAVIOUR OF REINFORCED CONCRETE EXTERIOR BEAM-COLUMN JOINT RETROFITTE WITH EXTERNALLY BONDED FIBER REINFORCED POLYMERE (FRP)

**S.Vijaya[1], B.Shivakumaraswamy[2], Ravikiran K.V[3]**

*[1]Professor, Department of Civil Engineering, Dr. Ambedkar Institute of Technology, Bengaluru*
*[2]Professor & Head, Department of Civil Engineering, Dr. Ambedkar Institute of Technology, Bengaluru*
*[3]Assistant Professor Department of Civil Engineering, Sai Vidya Institute of Technology, Bengaluru*

## Abstract

*Retrofitting of existing structure is one of the major challenges that modern civil engineering Structures have demonstrated that most of them will need major repairs in the near future. Until early 1990s, concrete jacketing and steel were the two common methods adopted for strengthening the deficient RC Beam Column Joints. A new technique has emerged recently which uses fiber reinforced polymer sheet such as carbon fiber reinforced polymer (CFRP), glass fiber reinforced polymer (GFRP) and aramid fiber reinforced polymer (AFRP) sheets to strengthen the beam-column joint. Also, recent research has attempted to simulate the behavior of reinforced concrete structures strengthened with FRP composites using the finite element method (FEM). In the present study, finite element modeling of a RC exterior beam-column joint retrofitted with externally bonded FRP is carried out with the help of commercially available software ANSYS 13.0. First, the control specimen is analyzed and the results obtained are compared with an experimental study from the literature. Then, the specimen is retrofitted with externally bonded glass-fiber-reinforced polymer (GFRP) sheets with different wrapping schemes and analyzed. The results from the retrofitted specimens are then compared with the results of the control specimen. It is found that for the control specimen, the values of the yield load and ultimate load obtained in the ANSYS are very close to the values obtained from the experimental study. Comparison between the load-deflection results obtained from the ANSYS for the control and retrofitted specimen. This is accompanied by the limited deflections for the retrofitted specimen as compared to the control specimen. The deflection ductility ratio and energy absorption has also increased for the retrofitted specimen.*

***Keywords:*** *Beam-Column Joint, Retrofitting, CFRP sheet, GFRP sheet, AFRP sheet, ANSYS 13.0*

---------------------------------------------------------------------***---------------------------------------------------------------------

## 1. INTRODUCTION

Reinforced concrete frame buildings designed before the introduction of the modern seismic oriented codes in early 1980s, offer an inadequate response to lateral load typical of seismic events. In this work the attention is focused on the behavior of exterior beam-column joint, since it is recognized that they are most vulnerable part of moment resisting RC frames, due to their lack of a reliable joint shear transfer mechanism. This poor behavior is mainly due to: inadequate reinforcement detailing (plain round bars) and deficiencies in the anchorage details (bars with end-hooks). Despite the fact that many nominally ductile existing structures did survive previous low-to-moderate ground motion events, the level of damage attained in these structures deems them vulnerable to collapse in future earthquake events. Therefore, rehabilitation of such structures is essential and cannot b neglected.

During the last two decades, the studies and applications of composites in construction, more particularly in the strengthening of existing buildings, represented one the fastest growing new areas within structural engineering. Retrofit techniques based on the use of externally bonded fiber-reinforced polymers (FRP) systems for structural enhancement mitigates several disadvantages and is gaining preference over the traditional strengthening methods such as concrete jacketing, steel plate bonding and sprayed concrete.

The majority of the research effort regarding numerical studies has been focused on the Finite Element modeling of strengthened RC beams to address the bonding issue of FRP plates and sheets. More researchers had used commercially-available software package such as ANSYS, ABAQUS, DIANA, ATENA, or SBETA to carry out the FE analysis. Studies regarding the modeling of RC joints with FRP materials are relatively limited. The available finite element software package, ANSYS program (ANSYS, Release 13.0 2010), was used in the present study.

## 2. MODELING

The beam-column joint considered for analysis consist of a exterior joint as shown in Figure 1(a) and Figure 1(b). The joint detail, material properties and the loading conditions are taken as in the experimental study conducted by Bhandari [1]. The column had a cross section of 100 mm × 100 mm with an overall length of 1100 mm and the beam had a cross section of 100 mm × 100 mm and the length of the cantilevered portion was 500 mm. The control specimens were designed as CS. Both beams and columns were reinforced with 4 bars of 8 mm diameter Fe500 bars and beam reinforcement is anchored in column up to a length of 500 mm. the ties consist of square hoops of 6 mm diameter Fe250 bars of size 60 mm × 60 mm placed 100 mm c/c in the column portion as well in the beam portion. M20 grade of concrete was adopted. The typical views of the ANSYS model of the specimens are shown in Figure 1(a) and Figure 1 (b).

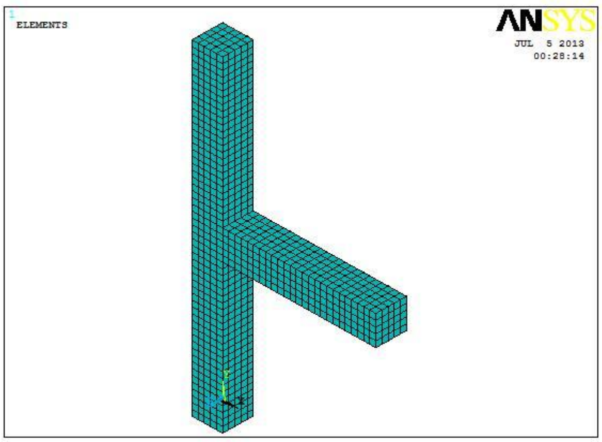



**Fig 1(a):** Typical View of ANSYS Model

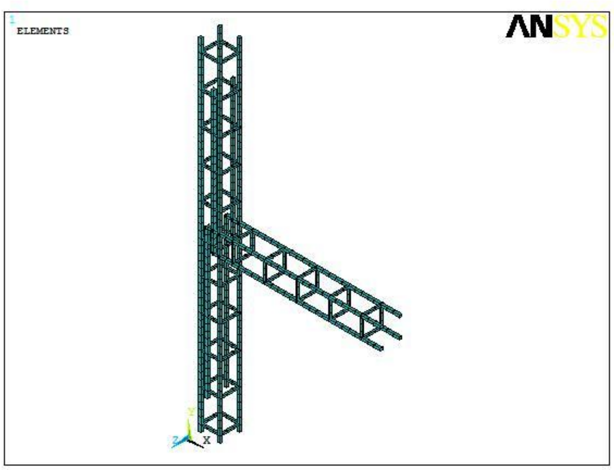
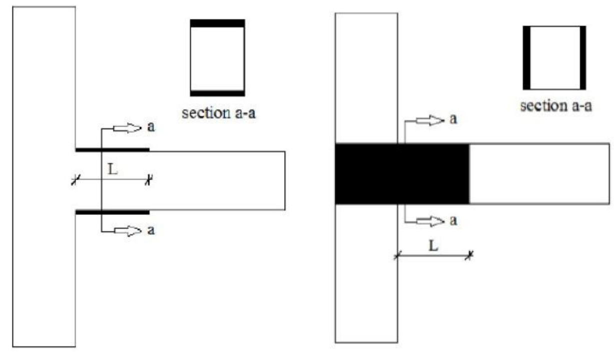


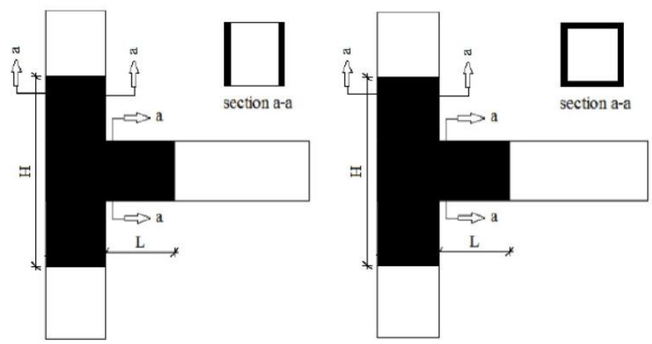

**Fig 1(b):** Typical View of ANSYS Model Detailed as experimental study

## 2.1 Retrofitted Specimens

For the purpose of study the behavior of the beam-column joint retrofitted with different FRP wrapping schemes, four specimens retrofitted with GFRP were modeled. Figure 2 shows the different wrapping arrangements of GFRP composites. The characteristics of these specimens are described in Table 1.




(a) RG1 (b) RG2




(c) RG3 (d) RG4

**Fig 2:** Different wrapping arrangements of GFRP composites.

## 2.2 Material Properties

Solid65 element was used to model the concrete. The modulus of elasticity of the concrete ($E_c$) and the Poisson's ratio (v) are mandatory information for the material definition. In ANSYS EX is the modulus of elasticity of the concrete ($E_c$), and PRXY is the Poisson's ratio (v). The modulus is based on the equation (as per Cl. 6.2.3.1 of IS 456: 2000 [2]),

$$E_c = 5000\sqrt{f_{ck}} \quad \ldots\ldots\ldots(1)$$

With a value of $f_{ck}$ equal to 20 Mpa. Poison's ratio was assumed to be 0.2. The ANSYS program requires the uniaxial stress-strain relationship for concrete in compression. Numerical expressions, Equations 2, 3 and 4, to construct the uniaxial compressive stress-strain curve for concrete in this study.

$$f = E_c / [1 + (\varepsilon / \varepsilon_o)^2]$$

$$\varepsilon o = 2 f_{ck} / E_c$$

$$E_c = f / \varepsilon$$

Where:

$f$ = stress at any strain $\varepsilon$, MPa

$\varepsilon$ = strain at stress $f$

$\varepsilon_o$ = strain at the ultimate compressive strength $f_{ck}$

**Table 1:** Characteristics of strengthen specimens.

| Specimen Name | Length in Beam (mm) | Height in column (mm) |
|---|---|---|
| RG1 | 135 | - |
| RG2 | 135 | - |
| RG3 | 135 | 400 |
| RG4 | 135 | 400 |

An example is included here to demonstrate a calculation of the five points (1-5). The model has a concrete elastic modulus of 22,360.67978 MPa. The value of $f_{ck}$ is equal to 20 MPa. Point No. 1, strain at a stress of 6 MPa (0.3 $f_{ck}$) is obtained for a linear stress-strain relationship for concrete (Equation 4), and is 0.00026832. Strain at the ultimate compressive strength, $\varepsilon_o$, is calculated by Equation 3, and equals 0.00179 mm/mm. Point Nos. 2, 3, and 4 are calculated from Equation 2, which gives the strain of 0.0006485, 0.0010286 and 0.0014087 mm/mm, corresponding to stress of 13, 17 and 19 MPa, respectively. Finally, Point No. 5 is at the ultimate strength, $f_{ck}$ of 20 MPa and $\varepsilon_o$ of 0.00179 mm/mm. Figure 3 shows the simplified compressive uniaxial stress-strain relationship that was used in this study.

For concrete, ANSYS requires four mandatory input data for the material property to be defined; open shear transfer coefficient, closed shear shear transfer coefficient, uni-axial cracking stress and uni-axial crushing stress. The shear transfer coefficient for open and closed cracks represent the condition at the crack face while it is open (loaded) or closed (reversed load), respectively. The value of these coefficient ranges from 0.0 to 1.0, with 0.0 representing a smooth crack (complete loss of shear transfer) and 1.0 representing a rough crack (no loss of shear transfer) (ANSYS, Release 13.0 [3]). Convergence problem occurs when the shear transfer coefficient for the open crack drops below 0.2. No deviation of response occurs with the change of coefficient. Therefore, the coefficient for the open crack is set to 0.3 (Kachlakev etal., [4]). The uniaxial cracking stress is based upon the modulus of rupture. This value is determined using the following equation (as per Cl.6.2.2 of IS 456; 2000[2]).

$$f_{cr} = 0.7\sqrt{f_{ck}}$$

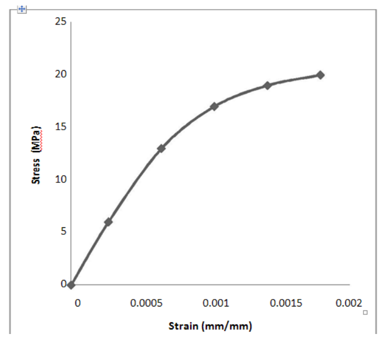



**Fig 3:** Simplified stress-strain curve for concrete used in FE model

The geometry of the beam-column connection has a significant influence on the model. The existence of corners at the interface between beam and column is resulting in stress concentration which in turn leads not only to convergence but also to a pre-mature failure for the finite element model. Therefore, in this study, the concrete crushing capability was turned off to avoid such problems. It was entered as -1 to turn off the crushing capability of the concrete element as suggested by past researchers (Kachlakev et al. [4]; Wolanski [5]; Mostofinejad and Talaeitaba [6]; Gorji [7]; Büyükkaragöz [8]).

Link 180 elements were used to model the steel reinforcement. The Link180 element requires linear isotropic and bi-linear isotropic material properties to properly model steel reinforcement. Elastic modulus (EX) is defined as 2,00,000 MPa and Poisson's ratio (PRXY) as 0.3. The bilinear model requires the yield stress ($f$y), as well as the hardening modulus of steel to be defined. The yield stress is defined as 500 MPa for 8 mm diameter bars while it is 250 MPa for 6 mm diameter bars.

The layered Solid185 element is used for the modeling Fiber Reinforced Polymer (FRP). Material properties for GFRP as specified by Kachlakev et al. [4] are taken in the present study and shown in Table 2.

| FRP Composite | Elastic Modulus (MPa) | Major Poisson's ratio | Tensile strength (MPa) | Shear modulus (MPa) | Thickness of laminate (mm) |
|---|---|---|---|---|---|
| GFRP | $E_x$ = 21,000 | $v_{xy}$ = 0.26 | 600 | $G_{xy}$ = 1520 | 1.3 |
| | $E_y$ = 7000 | $v_{xz}$ = 0.26 | | $G_{xz}$ = 1520 | |
| | $E_z$ = 7000 | $v_{yz}$ =0.30 | | $G_{yz}$ = 2650 | |

## 2.3 Real Constants

Solid65 element requires real constants for rebar assuming a smeared model. Values can be entered for Material Number, Volume Ratio, and Orientation Angles. In the present study the joint is modeled using discrete reinforcement as suggested by Fanning [9]. Therefore, a value of zero was entered for all real constants which turned the smeared reinforcement capability of the Solid65 element off as suggested by past researches (Ibrahim and Mahmood, [10]; Wolanski, [5]; Kachlakev et al., [4]).

Link180 element requires values for cross sectional area and initial strain are entered. Cross-sectional areas for the reinforcement of 8 mm and 6 mm diameter bars are 50.3 $mm^2$ and 28.3 $mm^2$ respectively. No real constants exist for the Solid185 element.

## 2.4 Loading and Boundary Conditions

To simulate the test conditions, column ends were fixed at the bottom as well as the top of the column. To achieve this, the translations at the nodes (UX, UY and UZ) are given constant values of 0. Then, the finite element model is loaded at a distance of 275 mm from the column face. For the both Control Specimen and Retrofitted Specimens the loading and boundary conditions are same as above.

## 3. ANSYS SOLUTION CONTROL

In nonlinear analysis, the total load applied to a fine element model is divided into a series of load increments called load steps. Each load increment is assigned a specific amount of load in the specified direction. At the completion of each load increment, the stiffness matrix of the model is adjusted to reflect the non-linear changes in structural stiffness before proceeding to the next load increment. The ANSYS program (ANSYS Release 13.0, [3]) uses Newton-Raphson equilibrium iterations technique was selected for updating the model stiffness. In the present study, for the reinforced concrete solid elements, convergence criteria were based on force and displacement, and the convergence tolerance limits were initially selected by the ANSYS program. Note that convergence of solutions for the models was difficult to achieve due to the nonlinear behavior of reinforced concrete. For the nonlinear analysis, automatic time stepping in the ANSYS program predicts and controls load step sizes. The maximum and minimum load step sizes are required for the automatic time stepping. The time at the end of each load step corresponds to the loading applied. Failure for the model is defined when the solution for a 1 N load increment still does not converge. The program then gives a message specifying that the model has a significantly large deflection, exceeding the displacement limitation of the ANSYS program. ANSYS gives deflection and crack patterns at various load increments which are presented here.

## 4. DISCUSSION OF RESULTS

The following comparisons are made: load-deflection plots; first cracking loads; loads at failure; crack patterns at failure; ductility and energy absorption.

## 4.1 Model Verification

Comparison between the load-displacement plot obtained from finite element analysis and the experimental study by Bhandari is shown in Figure 4. The yield load obtained from finite element analysis at 6.0 KN is 5% more than the yield load of 5.7 KN obtained from the experimental study. Similarly, the ultimate load obtained from finite element analysis at 7.8 KN is 20.5% more than the ultimate load of 6.2 KN from the experimental study. At yield load, the displacement obtained in finite element analysis is 10.38 mm at beam end as compared to 10.6 mm which is 2.07% less than the experimental study. Similarly, at ultimate load, the displacement obtained infinite element analysis is 22.12 mm which is 10% lesser than the 24.58 mm of the experimental study. The deflected shape and the crack at the first load and the ultimate load are shown in Figure 5 and Figure 6.

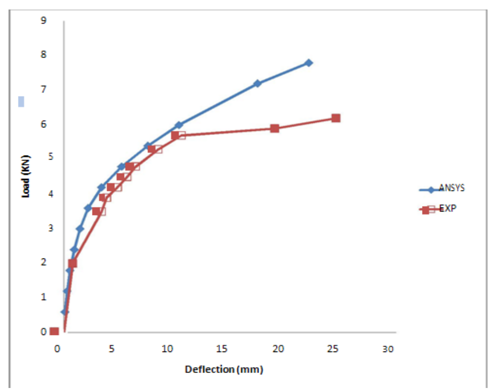



**Fig 4:** Comparison between the load- Displacement plot obtained from ANSYS and the experimental study.

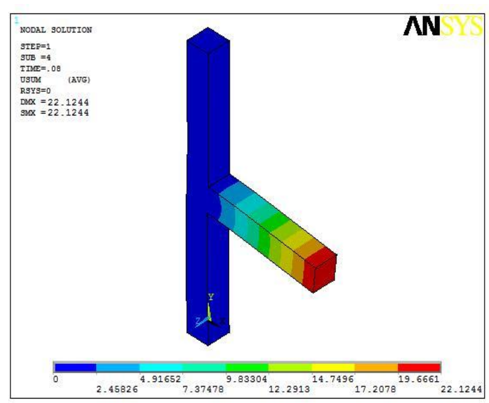



**Fig 5:** Deflected shape of the control specimen.

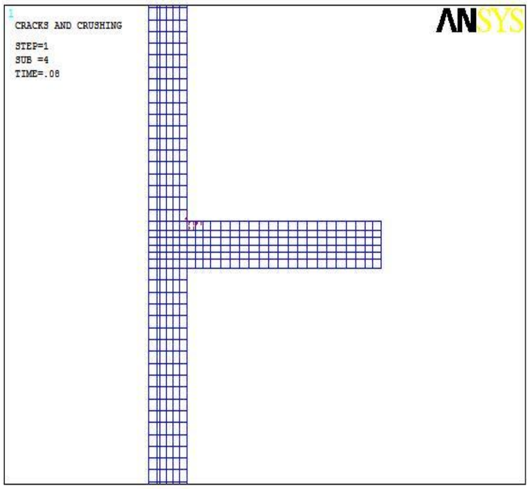



**Fig 6(a):** First crack at Load 2.4 KN.

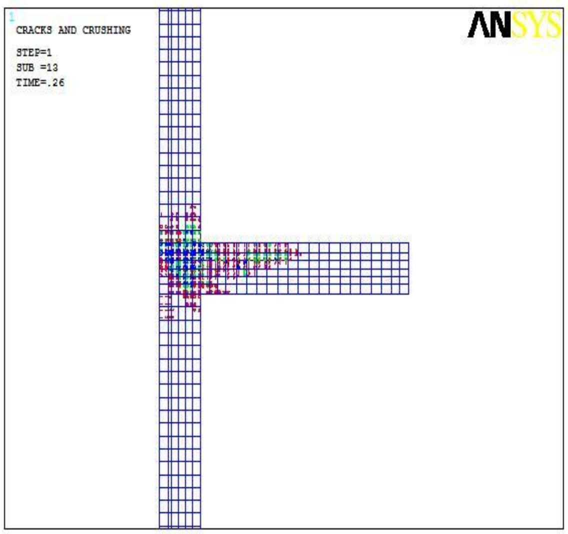



**Fig 6(b):** Crack pattern at ultimate load 7.8 KN.

This ensures that the element, material properties, real constants and convergence criteria are adequate to model the response of the model. This gives confidence in the use of ANSYS 13.0 Release 2010 and the model developed. This approach is then used to analyze the retrofitted beam-column joint.

### 4.2 Load- Deflection Plot

The load-deflection results of the GFRP retrofitted specimen are now compare with the control specimen as shown in Figure 7.

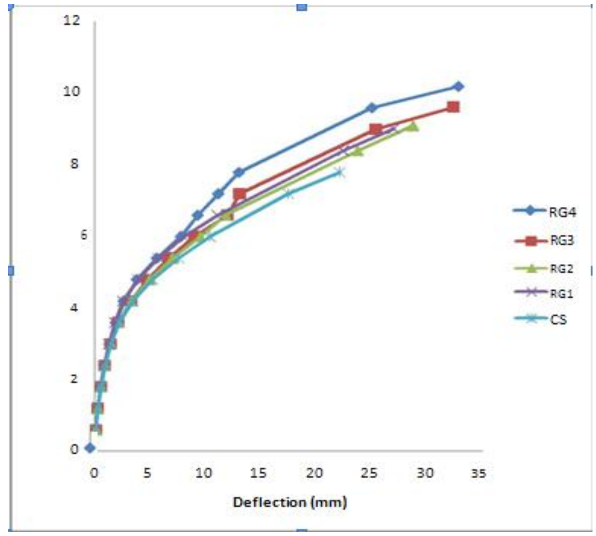



**Fig 7:** Load-deflection comparison of control specimen and GFRP retrofitted specimens

The yield loads for the retrofitted specimen RG1, RG2, RG3 and RG4 are observed at 6.6 KN, 7.2 KN and 7.8 KN which represents an increase of 10%, 10%, 16.67% and 23.07% from the yield load value of 5.7 KN for the control specimen. The ultimate loads for the retrofitted specimen increases by 31.26%, 31.86%, 35.55% and 39.21% from 6.2 KN to 9.02 KN, 9.10 KN, 9.62 KN and 10.2 KN when compared with the control specimen. The higher value of yield and ultimate load for the retrofitted specimens is associated with lower deflection as compared to the control specimen.

### 4.3 Ductility

Ductility is generally measured in terms of displacement and it is called as displacement ductility, which is the ratio of the maximum deformation that a structure or an element can undergo without significant loss of initial yielding resistance to the initial yielding deformation. The displacement ductility for all specimens is presented in Table 3. It is observed that ductility factor increases with the addition of FRP layers to the structural elements compared to control specimens.

**Table 3:** Ductility factor of GFRP specimens

| Joint Name | Ductility Factor (mm/mm) |
|---|---|
| CS | 2.13 |
| RG1 | 2.33 |
| RG2 | 2.40 |
| RG3 | 2.49 |
| RG4 | 2.52 |

**Table 4:** Energy absorption values of GFRP Specimens

| Joint Name | Energy Absorption Value (KN-mm) |
|---|---|
| CS | 127.845 |
| RG1 | 179.993 |
| RG2 | 189.628 |
| RG3 | 230.784 |
| RG4 | 252.481 |

### 4.4 Energy Absorption

The values of energy absorption (given by the area under the load-deflection plots up to ultimate load) are also compared for the control and retrofitted specimens in Table 4. All strengthened specimens dissipated more energy than the control specimen because of the improvement by addition of FRP wrapping.

## 5 CONCLUSIONS

The main observations and conclusions drawn are summarized below:

- Realistic non-linear analysis of RC beam column joint with FRP overlay could be performed using available software.
- Comparison between the load deflection results obtained from finite element analysis and that from the experimental shows that the finite element analysis results are more than the experimental results. The yield load and ultimate load are 5% and 20.5% more than the experimental results.
- Comparison between the load-deflection results obtained from finite element analysis for control and retrofitted specimens shows that the yield load and the ultimate load has significantly increased for the retrofitted specimen. The yield load of GFRP specimens RG1, RG2, RG3 and RG4 are 10%, 10%, 16.67% and 23.07% more than the control specimen.
- The different configurations of GFRP considered or the specimens were by attaching to the top, bottom and lateral sides of beams. The results show that respectable ductility and strength enhancement could be attained by engaging configured GFRP laminates correctly.
- Increase in cumulative energy shows that the specimens RG2 and RG4 are very good strengthening strategies for strength and ductility enhancement in the RC joints

## REFERENCES

[1] **Bhandari, P. (2010).** “Effect of stress level on retrofitting of exterior beam column joints,” Master’s Thesis, Thapar University, Patiala, Punjab, India.

[2] **IS 456: 2000 (Fourth Revision),** “Indian Standard: Plain and Reinforced Concrete- Code of Practice”, Bureau of Indian Standards, New Delhi, 2005.

[3] **ANSYS User’s Mannual,** Release 13.0. (2010). ANSYS, INC., Canonsburg, Pennsylvania.

[4] **Kachlakev, D., Miller, T.P., Yim, S., Chansawat, K., and Potisuk, T. (2001).** Finite element Modeling of Reinforced Concrete Structures Strengthened with FRP Laminations, Oregon Department of Transportation, Research Group, Salem, Oregon.

[5] **Wolanski, A.J.B.S. (2004).** Flexural Behavior of Reinforced and Prestressed Concrete Beams Using Finite Element Analysis, Master’s Thesis, Marquette University, Milwaukee, Wisconsin.

[6] **Mostofinejad, D. and Talaeitaba, S. B. (2006).** “ Finite Element Modeling of RC Connections Strengthened with FRP Laminations,” Iranian Journal of Science & Technology, V. 30, No. Bl, pp. 21-30.

[7] **Büyükkaragöz, A. (2010).** “ Finite Element Analysis of the Beam Strengthened with Prefabricated Reinforced Concrete Plate,” Scientific Research and Essays, V. 5. No. 6, pp. 533-544.

[8] **Fanning P. (2001).** Nonlinear Models of Reinforced and Post-tensioned Concrete Beams. Electronic Journal of Structural Engineering, 2, 111-119.

[9] **Ibrahim, A.M., and Mahmood, M.S. (2009).** Finite Element Modeling of Reinforced Concrete Beams Strengthened with FRP Laminations. European Journal of Scientific Research, 30(4), 526-541.