

Finite Element Modeling On Behaviour Of Reinforced Concrete Beam Column Joints Retrofitted With CFRP Sheets Using Ansys

T. Subramani¹, S.Krishnan², M.S.Saravanan³, Suboth Thomas⁴

¹Professor & Dean, Department Of Civil Engineering, VMKV Engineering College, Vinayaka Missions University, Salem, India.

²Associate Professor and Head, Department of Mechanical Engineering, Mahendra College of Engineering, Salem, India.

³Associate Professor, Department Of Civil Engineering, Annapoorana Engineering College, Salem, Tamilnadu, India.

⁴Professor & Director, Salem School of Architecture, Vinayaka Missions University, Salem, India.

ABSTRACT

Recent earthquakes have demonstrated that most of the reinforced concrete structures were severely damaged during earthquakes and they need major repair works. Beam column joints, being the lateral and vertical load resisting members in reinforced concrete structures are particularly vulnerable to failures during earthquakes. The existing reinforced concrete beam column joints which are not designed as per code IS13920:1993 must be strengthened since they do not meet the ductility requirements. The Finite element method (FEM) has become a staple for predicting and simulating the physical behaviour of complex engineering systems. The commercial finite element analysis (FEA) programs have gained common acceptance among engineers in industry and researchers. The details of the finite element analysis of beam column joints retrofitted with carbon fibre reinforced polymer sheets (CFRP) carried out using the package ANSYS are presented in this paper. Three exterior reinforced concrete beam column joint specimens were modelled using ANSYS package. The first specimen is the control specimen. This had reinforcement as per code IS 456:2000. The second specimen which is also the control specimen. This had reinforcement as per code IS 13920:1993. The third specimen had reinforcement as per code IS 456:2000 and was retrofitted with carbon fibre reinforced polymer (CFRP) sheets. During the analysis both the ends of column were hinged. Static load was applied at the free end of the cantilever beam up to a controlled load. The performance of the retrofitted beam column joint was compared with the control specimens and the results are presented in this paper.

KEYWORDS: Finite Element Modelling, Behaviour, RCC Beam Column Joints Retrofitted With CFRP Sheets, Ansys

I INTRODUCTION

The techniques of using fiber sheets for strengthen the beam-column joints have a number of favourable characteristics such as ease to install, immunity to corrosion and high strength. The simplest way to strengthen the joints is to wrap fibres sheets in the joint region in two orthogonal directions. Many fibre reinforced polymer sheets are available in the market for strengthening reinforced concrete members. Carbon fibre reinforced polymer sheets are commonly used for retrofitting the structural elements. During the present investigation, ANSYS modelling of reinforced concrete beam-column joints has been carried out to understand the behaviour of retrofitted reinforced concrete beam-column joint specimens retrofitted with carbon fibre wrapping sheets.

II INTRODUCTION TO ANSYS

The ANSYS program has many finite element analysis capabilities, ranging from a simple, linear, static analysis to a complex non – linear, transient dynamic analysis.

A typical ANSYS analysis has three distinct steps:

- Building the model
- Applying loads and obtains the solution
- Review the results.

2.1 BUILDING THE MODEL:

Building a finite element model requires a more of an ANSYS user's time than any other part of the analysis. First you specify the job name and analysis title. Then, define the element types, real constants, and material properties, and the model geometry.

2.2 DEFINING ELEMENT TYPES:

The analysis element library contains more than 100 different element types. Each element type has a unique number and a prefix that identifies the element category. Example: beam, pipe, plate, shell, solid.

2.3 DEFINING ELEMENT REAL CONSTANTS:

Element real constants are the properties that depend on the element type, such as cross sectional properties of a beam element. For example real constants for BEAM3, the 2-d beam element, or area, moment of inertia (IZZ), height, shear deflection constant (SHEAR Z), initial strain (ISTRN) different elements of same type may have different real constant values.

2.4 DEFINING MATERIAL PROPERTIES:

Most element types require material properties. Depending on the application, material properties may be:

- Linear or non linear
- Isotropic, Orthotropic, or an anisotropic
- Constant temperature or temperature – dependant

As with element type and real constant, each set of material properties has a material reference number. The table of material reference number versus material property set ids called material property table. Within, one analysis you may have multiple material properties set.

2.5 MATERIAL PROPERTY TEST:

Although you can define material properties separately for each element analysis, the ANSYS program enables you to store a material property set in an archival material library file, then retrieve the set and reuse it in multiple analysis. The material library files also enable several ANSYS users to share common used material property data.

III OVERVIEW OF MODEL GENERATION

The ultimate purpose of finite element analysis is to recreate mathematically the behaviour of an actual engineering system. In other words, the analysis must be an accurate mathematical model of a physical prototype. In the broadest sense, this model comprises all the nodes, elements, material properties, real constants, boundary conditions, and other features that are used to represent the physical system. In ANSYS terminology, the term model generation usually takes on the narrower meaning of generating the nodes and elements that represent the spatial volume and connectivity of actual system. Thus, model generation in this discussion will mean the process of defining the geometric configuration model's nodes and elements.

The ANSYS program offers you the following approaches to model generation:

- Creating a solid model within ANSYS.
- Using direct generation reporting a model created in CAD system.

3.1 MESHING YOUR SOLID MODEL:

The procedure for generating a mesh of nodes & elements consists of three main steps:

- Set the element attributes
- Set mesh controls
- Generate the mesh controls,

The second step, setting mesh controls, is not always necessary because the default mesh controls are appropriate for many models. If no controls are specified, the program will use the default setting on the mesh command to produce a free mesh. As an alternative, you can use the mesh size feature to produce a better quality free mesh.

3.2 FREE AREA MAPPED MESH:

Before meshing the model, and even before building the model, it is important to think about whether a free mesh or a mapped mesh is appropriate for the analysis. A free mesh has no restrictions in terms of element shapes, and has no specified pattern applied to it.

Compared to a free mesh, a mapped mesh is restricted in terms of the element shape it contains and the pattern of the mesh. A mapped area mesh contains either only quadrilateral or only triangular elements, while a mapped volume mesh contains only hexahedron elements. In addition, a mapped mesh typically has a regular pattern, with obvious rows of elements. If you want this type of mesh, you must build the geometry as series of fairly regular volumes and or areas that can accept a mapped mesh.

3.3 SETTING ELEMENT ATTRIBUTES:

Before you generate a mesh of nodes and elements, you must first define the appropriate element attributes. That is, you must specify the following:

- Element type
- Real constant set
- Material properties set
- Element co-ordinate system.

LOADING:

The main goal of finite element analysis is to examine how a structure or component response to certain loading condition. Specifying the proper loading conditions, therefore, a key stepping analysis. You can apply loads on the model in variety of ways in ANSYS program.

LOADS:

The word loads in ANSYS terminology includes boundary.

Conditions and externally or internally applied forcing functions. Example of loads in different disciplines is: Structural: displacement, forces, pressures, temperatures (for thermal strain), gravity
Thermal: temperatures, heat flow rate, convections, and internal heat generation, infinite surface
Magnetic: Magnetic Potentials, magnetic flux, and magnetic current segment
Electric: electric potentials, electric current, charges, charge densities, infinite
Fluid: Velocities and pressures

A DOF constraint fixes the degrees of freedom of a known value. Examples of constraints are specified displacement and symmetric boundary conditions in structural analysis, prescribed temperatures in thermal analysis, and flux parallel boundary conditions.

A force is concentrated load applied at a node in a model. Examples are forces and moments in structural analysis, heat flow rates in thermal analysis.

A surface load is distributed load applied over a surface. Examples are pressures in structural analysis and convections and heat fluxes in thermal analysis.

Coupled field loads are simple case of one of the above loads, where results from analysis are used as loads in another analysis. For examples you may apply magnetic forces calculated in magnetic field analysis are force loads in structural analysis.

3.4 HOW TO APPLY LOADS:

You can apply loads most loads either on the solid model (on key points, line, areas) or on the finite element model (on nodes and elements). For example, you can specify forces at a key point or a node. Similarly, you can specify convections (and other surface loads) on lines and areas or nodes and element faces.

No matter how you specify loads, the solver expects all loads to be in term of finite element model. Therefore if your specify loads on the solid model, the program automatically transfers them to the nodes and element at the beginning of the solution.

3.5 SOLUTION:

In the solution phase of the analysis, the computer takes over and solves the simultaneous equations that the finite element method generates. The result

Solution is: a nodal degree of freedom value, which form the primary solution, and b) derived values, which form the element solution. The element solution is usually calculated at the elements integration points. The ANSYS program writes the results to the database as well as to the result file. (Job name. rst , rth, rmg, or. rfl}.

Several methods of solving the simultaneous equations are available in the ANSYS program:

frontal solution, sparse direct solution, Jacobi Conjugate gradient (JCG solution, incomplete cholesky conjugate (ICCG) solution, preconditioned conjugate gradient (PCG) solution, automatic iterative solver option (ITER). The frontal solver is the default, but you can select a different solver.

3.6 POST PROCESSING:

After building the model and obtaining the solution, you will want answers to some critical question: will the design really work when put to use? How high are the stresses in this region? How does the temperature of this part vary with time? What is the heat loss across my model? How does the magnetic flow through this device? How does the placement of this object affect fluid flow? The post processors in the ANSYS program can help you find answer these questions and others. Post processing means reviewing the results of an analysis. it is probably the most important step in the analysis, because you are trying to understand how the applied loads affect your design, how good you finite element mesh is, and so on.

Two post processors are available review your results: Post 1, the general post processor, and post 26, the time history post processor. Post 1 allows you to review the results over the entire model at specific load steps and sub steps (or at specific time – points or frequencies). In a static structural analysis, for example, you can display the stress distribution for load step 3 or, in a transient thermal analysis; you can display the temperature distribution at time – 100 seconds.

3.7 THE RESULT FILES

The ANSYS solver writes results of an analysis to the results file during solution. The name of the results file depends on the analysis discipline: Job Name. rst for structural analysis.

3.8 THE GENERAL POST PROCESSOR

You use Post1, the general post processor, to review analysis results over the entire model, or selected portion of the model, of a specifically defined combination of loads at a single time (or frequency). Post 1 has many capabilities, ranging from simple graphics display and tabular listing to more complex data manipulation such as load case combinations.

DISPLAYING RESULTS GRAPHICALLY:

Graphics display is perhaps the most effective way to review results. You can display the following types of graphics in post1:

- Contour displays
- Deformed shape displays
- Vector displays
- Path plots
- Reaction force displays
- Particle flow traces.

IV INTRODUCTION TO STRUCTURAL ANALYSIS

Structural analysis is probably the most common application of the finite element method. The term structural (or structure) implies not only civil engineering structures such as bridges and buildings, but also naval, aeronautical and mechanical structures such as ship hulls, aircraft bodies and machine housings, as well as mechanical components such as pistons, machine parts and tools.

Types of structural analysis.

The seven types of structural analysis provided by ANSYS are given below.

1. Static analysis: used to determine displacement, stresses etc. under static loading conditions. Both linear and non-linear static analyses. Non – Linearity's can include plasticity, stress stiffening, large deflection, large strain, hyper elasticity, contact surfaces, and creep.
2. Modal analysis: used to calculate the natural frequencies and mode shapes of a structure. Different mode extraction methods are available.
3. Harmonic analysis: used to determine the response of a structure to harmonically time varying loads.
4. Transient dynamic analysis: used to determine the response of a structure to arbitrarily time varying loads. All non-linearity's mentioned under static analysis above are allowed.
5. Spectrum analysis: an extension of the modal analysis, used to calculate stress and strain due to response spectrum or a PSD input (random vibrations).
6. Buckling analysis: used to calculate the buckling load and determine the buckling mode shape. Both linear (eigenvalue) buckling and non –linear buckling analyses are possible.
7. Explicit dynamic analysis – ANSYS provides an interface to the LS-

Dynamic explicit finite element programs is used to calculate fast solution for large deformation dynamics and complex contact problems.

V MODELING

The beam column joint considered for analysis consists of a cantilever portion and column portion as shown in Figure 1.a and Figure 1.b. The column had a cross section of 200 mm x 200 mm with an overall length of 1500 mm and the beam had a cross section of 200 mm x 200 mm and the length of the cantilevered portion was 600 mm.

The control

Specimens were designated as C1 and C2. C1 had reinforcement as per code IS 456-2000 And C2 had reinforcement as per code IS 13920-1993. The specimen retrofitted with Carbon reinforced polymer sheet was designated as C3 which had reinforcement as per code IS 456-2000. The column portion was reinforced with 4 numbers of 12mm diameter Fe 415 rods and the beam portion was reinforced with 2 numbers of 16 mm diameter Fe415 rods each in the tension and compression zones. The lateral ties in the columns of the specimens C1 and C3 were 6 mm diameter Fe 250 bars with the spacing of 180 mm c/c as per code IS 456:2000, clause 26.5.3.2(c). Beam had vertical stirrups of 6 mm diameter Fe 250 bar at 120 mm c/c as per code IS 456:2000, clause 26.5.1.6. The development length of the tension and compression rods in beam were also provided as per clause 26.2.1 of code IS 456:2000. For the specimen C2, the lateral ties in the columns consisted of 8 mm diameter Fe 415 bar at 75 mm c/c for the central distance of 1100 mm as per code IS 13920:1993, clause 7.4.6 and 6mm diameter Fe250 bars at 100 mm c/c for the remaining length of the column. Beams had vertical stirrups of 6 mm diameter Fe 250 bar at 40 mm c/c. up to a distance of 340 mm from the face of the column as per code IS13920:1993, clause 6.3.5 and 6 mm diameter Fe250 bar at 80 mm c/c for remaining length of the beam.

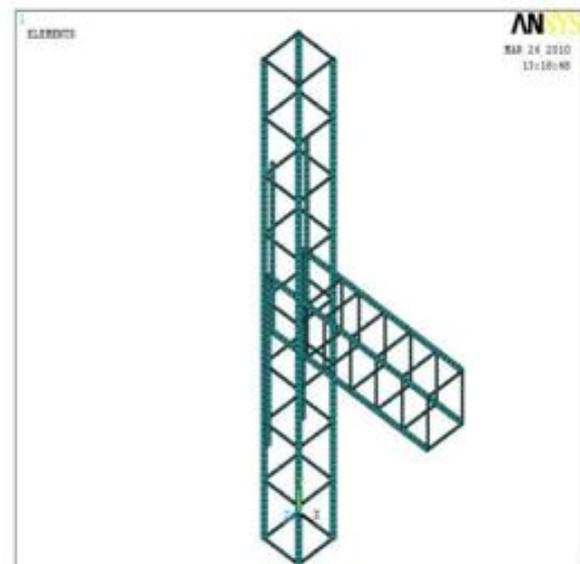


Figure 5.1. Typical View of ANSYS Model
Detailed as per code IS 456:2000

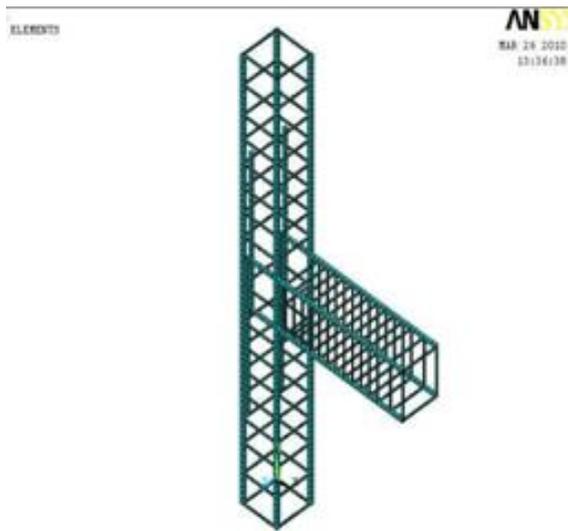


Figure 5.2. Typical View of ANSYS Model

Detailed as per code IS 13920:1993

The development length of the beam rods were also provided as per code IS 13920:1993, clause 6.2.5. M20 grade of concrete was adopted. The typical views of the ANSYS model of the specimens are shown in Figure.5.1 and Figure.5.2 Meshing was done for both control and retrofitted specimens using ANSYS. Both ends of the column were hinged. The concrete was modelled using Solid 65 element. The reinforcement was modelled using Link 8 element. The wrapping was modelled using Solid 45 element. The static load was applied at the free end of the cantilever beam at a regular load interval of 5 kN for the control and retrofitted reinforced concrete beam- column joint models. The performance of the retrofitted beam-column joint specimen was compared with the control beam-column joint specimens. Figure.5.3 and Figure. 5.4 show the meshing for the control and retrofitted specimen.

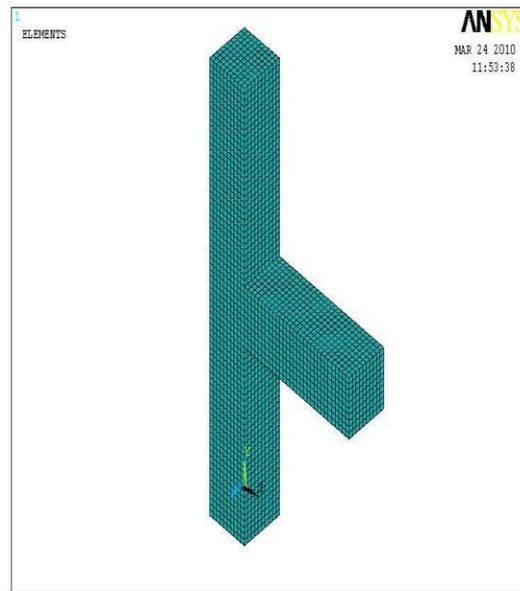


Figure 5.3: Typical meshed Control Specimen

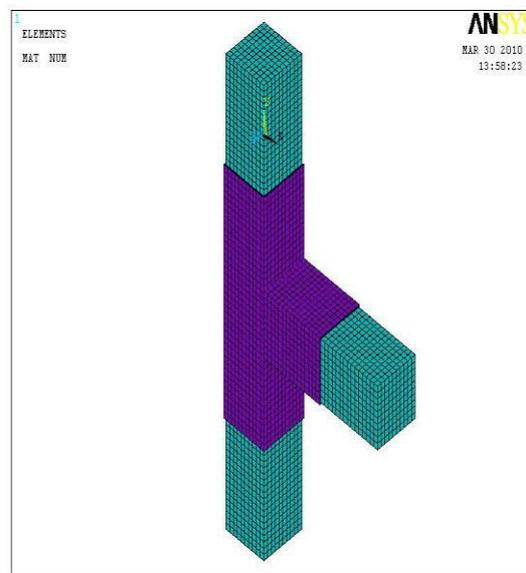


Figure.5.4 Typical meshed Retrofitted specimen

VI NON LINEAR MODELLING OF THE BEAM COLUMN JOINTS

Non-linear analysis was done for three beam column specimens using the software ANSYS. A transverse static was applied at the free end of the beam to develop a bending moment at the joint. The load was increased in steps till a controlled load of 20 kN. The deflection at the free end of the cantilever beam was noted. The deflections of the specimen C1 were found to be 3.5 mm for the load of 5 kN, 5 mm for the load of 10 kN, 15.5 mm for the load of 15 kN and 42.5 mm for the load of 20 kN. The same

procedure was repeated for the specimen detailed as per code IS 13920-1993 and for the retrofitted specimen. .

Table 6.1 Deflections of Control and Retrofitted Specimens.

Specimen	Deflection in mm for the load of			
	5 kN	10 kN	15 kN	20 kN
C1	3.5	5.0	15.5	42.5
C2	2.5	3.5	10.0	32.5
C3	1.0	2.0	8.0	10.5

Table 6.2 Comparison between Deflection and Energy Absorption Capacity

Specimen I D	Def. in mm for the loa	% reduction in	E.A.C up to a def. of 10 kN.mm	% increase in E.A.C
C 1	4 2	-	61.25	-
C 2	3 2	23.53	87.50	42.86
C 3	1 0	75.29	131.25	114.89

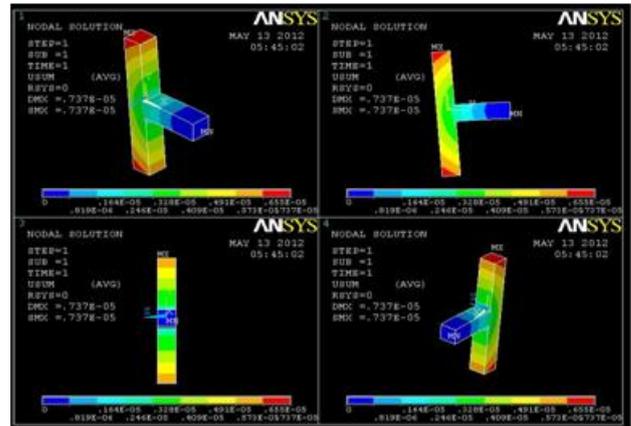


Figure 6.2 Typical View of Deflected Control Specimen (IS 13920-1993)

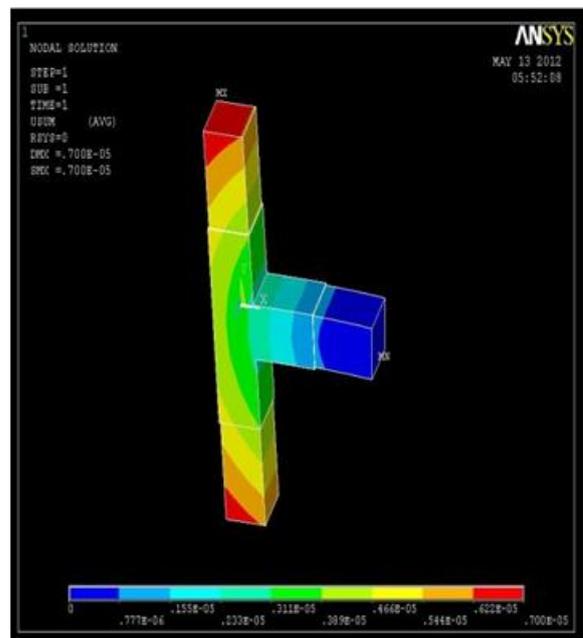


Figure 6.3 Typical View of Deflected Retrofitted Specimen

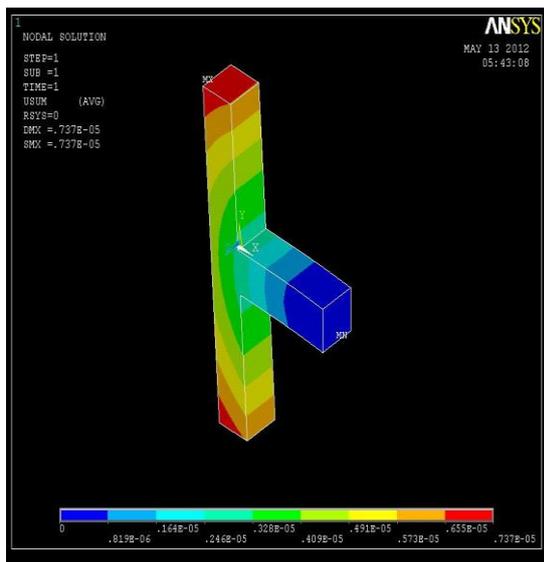


Figure 6.1 Typical View of Deflected Control Specimen (IS 456-2000)

Figure 6.1, Figure 6.2, and Figure 6.3 show the typical views of the deflected control and retrofitted specimens. Figure 6.4. shows the load deflection curve for the control and retrofitted specimens. Table 6.1.shows the deflections at various interval Of load of the control and retrofitted specimens. Table.6.2 shows comparison between the Deflections and energy absorption capacity of the control and retrofitted specimens.

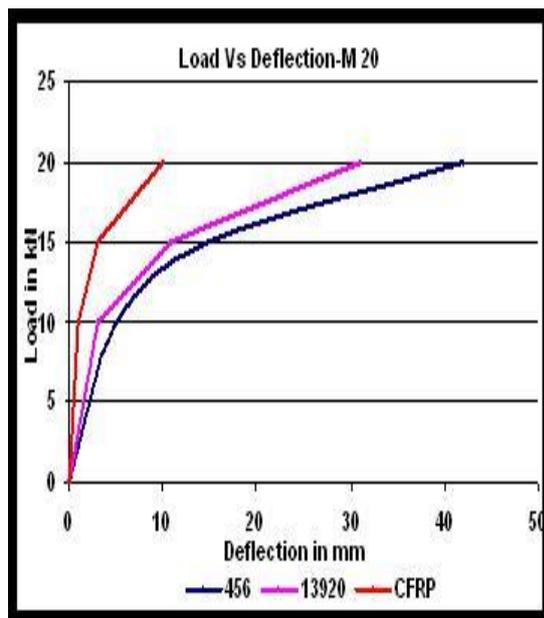


Figure 6.4. Load Deflection Curve for the Control and Retrofitted Specimen

VII DISCUSSION OF THE RESULTS

It can be found from the Table 2. that the deflection of the beam-column joint specimen detailed as per code IS 13920-1993 is 23.53 % less than that of the specimen detailed as per code IS 456-2000 and deflection of the beam-column joint specimen retrofitted with carbon reinforced polymer sheet was 75.29 % less than that of the specimen detailed as per code IS 456-2000. The energy absorption capacity of the specimen beam-column joint specimen detailed as per code IS 13920-1993 is 42.86 % more than that of the specimen detailed as per code IS 456-2000 and energy absorption capacity of the beam-column joint specimen retrofitted with carbon reinforced polymer sheet was 114.29 % more than that of the specimen detailed as per code IS 456-2000.

VIII CONCLUSIONS

Based on the ANSYS modeling and analysis carried out on the control and retrofitted beam-column joint specimens using CFRP sheets, the following conclusions were drawn.

- The deflection of the beam-column joint specimen detailed as per code IS13920-1993 was found to be 23.53 % lower than that of the specimen detailed per code IS 456-2000.
- The deflection of the beam-column joint specimen retrofitted with GFRP sheet reduced the deflection about 75.29 %.when compared with the deflection of specimen detailed as per code IS 456-2000.

- The energy absorption capacity of the beam-column joint specimen detailed as per code IS 13920-1993 was found to be 42.89 % higher than that of the specimen detailed per code IS 456-2000.
- The energy absorption capacity of the beam-column joint specimen retrofitted with CFRP sheet increased about 114.29 %.when compared with the energy absorption capacity of specimen detailed as per code IS 456-2000.

REFERENCES

- [1]. Bindu K.R.,Jaya K.P ., 2008. Performance of Exterior Beam Column Joints with Cross Inclined Bars Under Seismic Type Loading. Journal of engineering and applied science. vol.7,pp 591-597
- [2]. Alexander G. Tsonos ., 2008. Effectiveness of CFRP Jackets and RC Jackets in Post Earthquake and Pre Earthquake Retrofitting of Beam Column Sub Assemblages, Journal of engineering structures vol. 30,pp 777-793.
- [3]. Lakshmi G.A, Anjan Dutta, Deb S.K Grim, R.E., 2008. Numerical Study of Strengthening of Beam Column Joints Under Cyclic Excitation Using FRP Composites, Journal of structural engineering, vol 35,pp 59-65.
- [4]. Yousef A. Al-Salloum , Tarek H.Almusallam , 2007. Seismic Response of Interior RC Beam Column Joints Upgrade with FRP Sheets. I: Experimental Study Journal of composite for construction vol. 11,pp 575-589.
- [5]. Yousef A. Al-Salloum , Tarek H.Almusallam , 2007. Seismic Response of Interior RC Beam Column Joints Upgrade with FRP Sheets. II: Analysis And Parametric Study , Journal of composite for construction vol. 11,pp 590-599.
- [6]. Devados Menon ,Pradip Sarkar , Rajesh Agrawal , 2007, Design of RC Beam Column Joints Under Seismic Loading – A Review , Journal of structural engineering, vol. 33,pp 449-457.
- [7]. Subramani,T , Athulya Sugathan, “Finite Element Analysis of Thin Walled- Shell Structures by ANSYS and LS-DYNA”, International Journal of Modern Engineering Research,Vol.2, No.4, pp 1576-1587,2012.
- [8]. Subramani.T , Senthil Kumar.R, “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.
- [9]. Subramani.T, Manivannan.R, Kavitha.M, "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.

- [10]. Subramani.T, Subramani.M, Prasath.K," 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.