iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 6 **Issue:** VI **Month of publication:** June 2018

DOI: http://doi.org/10.22214/ijraset.2018.6087

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue VI, June 2018- Available at www.ijraset.com

Finite Element Analysis of Beam Column Joint in RCC Structure

Vibha Pandey1, Dr. P.S. Bokare2
[1]*Post Graduate Student, Civil Engineering,*
[2]*Principal, RSR Rungta College of Engineering and Technology (C.G.)*

Abstract:* Finite element analysis is basically used in developing an analytical model for the study of behavior of Reinforced Concrete Member. This method is chosen to represent concrete and reinforcement at the end joint and the intermediate joint of continuous reinforced concrete T beam. The behavior of such joints is studied in several peer reviewed publications. The objectives, methodology, observation, results and discussions are represented in a concise manner. It is concluded that finite element analysis is widely used for analysis of reinforced cement concrete structures.
Keywords: FEM, Beam Column Joint, Reinforced Concrete Structure.

I. INTRODUCTION

The modern engineering demands very fine design of structures. We should combine the demand of the conservative use of materials and the safety requirements. The structure are needed to be designed having sufficient durability and safety as well as consuming minimum amount material. These requirements of a structural design are contradictory to each other and should be optimized to achieve a golden mean of these two.

The design of structures fulfilling above requirements is based on intricate knowledge of analysis methods available with the designer. Out of various methods of analysis, studies reveals that methods like strain energy method; matrix method, finite element analysis method etc. are some of the common methods which result in near accurate assessment of state of the stress (Fig. 1) at a particular point in a structure.

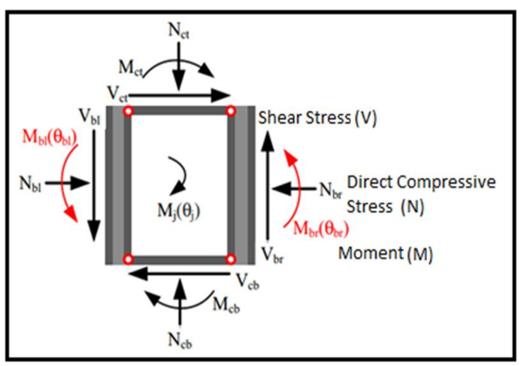

Fig. 1 State of stress at joint of RC beam and column

The joint in an RCC structure is subjected to various stresses as shown in Figure 1. The shear stresses (V), the Moment (M) and Direct Compressive Stress (N) produce compression and tension in the joint. In order to design a structure properly, the estimate of these stresses need to be done accurately. The literature shows that the Finite Element Method (FEM) is one such method (out of various available methods) which yields fairly accurate results. Hence, the objective of this paper to explore from various available resources whether FEM serves the desired purpose.

Finite Element analysis method introduced by Zeinkiewicz (1) analyses the structure fairly well and near accurate. This paper presents review of some publications on use of Finite element method, for the design of Reinforced concrete structural end and intermediate joints.

II. LITERATURE REVIEW

N. Bansal et al. 2013 (2) presented a study on the best reinforcement detailing of reinforced concrete corners were determined out of four different reinforcement details on the basis of load deflection behavior using finite element analysis software (ANSYS). This paper also discusses the parametric study including the effect of diameter of main steel bar, concrete grade and spacing of shear reinforcement on the load deformation behavior of corners under opening bending moments. Eight-nodded element, having translations in x, y and z directions, was used for the modeling of concrete. A discrete approach was adopted, using two-nodded, three-dimensional bar elements, adopted for the modeling of reinforcements. The beam column joint has been modeled in finite element analysis software preprocessor using 3D elements (Solid65). In the finite element analysis of concrete structures two principally different approaches were used for modeling of crack. Those were discrete crack modeling and smeared crack modeling. A concrete-based model was used for the consideration of tension stiffening to simulate the post-cracking behavior of concrete. The effect of detailing of steel on joints under opening bending moment studied with help of load deflection curves. The load deflection plots obtained by analysis were compared with corresponding experimental plots. The analytical results show good agreement with the experimental results. It was concluded that model having- shear reinforcement of spacing 75 mm centre to centre, Main diameter of bar as 12 mm, Concrete Grade M-40 gives the best result because it takes the maximum load with maximum deflection and having maximum ductility value.

M. Barbato et al. 2009 (3) presented a study on finite element modeling of reinforced concrete beams retrofitted with fiber reinforced polymers to evaluate the load-carrying capacity of reinforced concrete beams flex rally strengthened with externally bonded fiber reinforced polymer strips and plates using an efficient two-dimensional frame finite element . FEDEAS Lab, a MATLAB tool was used for static & dynamic structural analysis. Carbon Fiber Reinforced Polymer & Glass Fiber Reinforced Polymer was used with stiffness from 11.7 G Pa to 270 G Pa. Element state determination was performed by a Gauss–Lobatto integration scheme. The reinforcing steel modeled using the one-dimensional Menegotto–Pinto (M–P) plasticity model. The Monti–Renzelli model used to describe de bonding of FRP strips/plates on un cracked concrete and linear anchorage system. The study carried out a comparison between experimentally recorded and numerically simulated applied load-mid span deflection response. Comparison between experimental measurement and FE simulation of the ultimate load-carrying capacity were also carried out. This paper presented a simple and efficient two-dimensional frame finite element, denoted as FRP–FB-beam, able to accurately simulate the response of RC beams flex rally strengthened with externally bonded FRP strips and/or plate's .The FRP–FB-beam were developed from a force-based formulation and considered distributed plasticity with layer dis-cretization of the cross-sections. The proposed FE was used to predict the ultimate load-carrying capacity of beams subjected to three- and four-point bending loading. The agreement between experimental results and numerical simulations was very good. In addition, the finite element results, obtained employing very coarse meshes built using the FRP–FB-beam element, provided information on the specific failure mode experienced by the considered structural system. The Recorded and simulated applied load-mid span deflection responses of the reference and fiber reinforced polymer -retrofitted beams and slabs were also compared, showing again a very good agreement. The major features of this frame FE was its simplicity and its efficiency in terms of mesh refinement.

Ibrahim G. et al. (4) 2018 presented a study on the seismic performance of exterior beam-column joints in building frames strengthened by Ferro-cement using nonlinear finite element analysis. The parametric studied variables were the level of axial loading on the column, compressive strength of specimens, percentage of longitudinal reinforcement in the beam, and orientation of expanded wire mesh in Ferro-cement layer, for specimens strengthened by different number of Ferro-cement layers. ANSYS was used for Non linear finite element analysis. While analyzing basic idealization were carried out. The test beam-to-column joint specimens were typically discretized using 3-D isoperimetric 8-node solid elements; Solid65. The element "Solid65" was adopted to model the concrete and Ferro-cement layers to simulate cracking in tension and crushing in compression. The comparison between experimental and numerical results carried out and the analysis indicated formation of flexural cracks in the test specimens at low levels of displacements ranging between 1.6mm and 3.0 mm. Symmetrical crack patterns occurred for both positive and negative loading directions. Load carrying capacity, Crack patterns, load displacement hysteresis loops, and stress distribution results for theoretically studied specimens were simulated accurately using ANSYS package. It was concluded that Strengthening of specimens by Ferro-cement reduced the effect of axial loading level and longitudinal steel ratio in the beam on the ultimate load of studied specimens. Changing the orientation angle of expanded wire mesh from 60° per Ferro-cement layer to 45° had a minor effect on the ultimate load but a significant effect on the ductility of studied specimens.

Yu. M. Pleskachevskii et al. (5) 2008 presented Finite element modeling of the interaction of the reinforcement with concrete Matrix to determine the stress distribution on cross sections of the rebar and the location of zones with cracks in concrete. The study was dedicated to the problem of refined determination of the stress-strain state in forced concrete beam, based on simulating the contact interaction between the metal rebar and concrete matrix with the use of finite element method (FEM). A finite-element model of a reinforced concrete beam with rebar modeled by a 3-D deformable body was developed. ANSYS was used for the analysis Work. The deformation and destruction of concrete were described using a mathematical model with an 8-node finite element with three degrees of freedom at each node. The reinforcement was modeled by using 20-node prismatic finite elements. The finite-element model developed has allowed us to determine that the stresses in the cross-sections of reinforcement vary considerably within the cross-sectional area of each rebar. It was found that the break of bond between the reinforcement and concrete goes outside the areas of intensely cracked concrete matrix. The greatest compressive stresses occur in the lower part of the fastening region, the stresses in the upper part of this region are close to zero, because exactly in that location the first cracks arise. Within the limits of cross-section of the rebar, the stresses differ by more than 30%, while the maximum stresses were observed not on the surface of the rebar, but inside it. A finite element model of interaction between a cylindrical metal rebar and concrete matrix was developed, whose particular feature is the representation of the rebar by 3D finite elements. By the Calculations of the stress-strain state of a RC beam it was concluded that the accuracy of the classical model currently used in designing RC structures was inadequate to describe the deformation in zones with cracks. In addition, the maximum stresses in the reinforcement not always occur in the cross-section with the maximum BM. By analyzing the location of cracks in concrete, it is found that breaks of bonds between a rebar and concrete are observed not only in the areas of intense cracking of the concrete matrix, they also go outside these areas.

Tine Tysmansa et al. 2015 (6) presented Finite element modeling of the biaxial behavior of high-performance fiber-reinforced cement composites (HPFRCC). Concrete Damaged Plasticity (CDP) model was adopted to facilitate the numerical analysis and design of more complex structures model for biaxial stress states, and for cement composites with a large strain hardening capacity (ratio of failure stress to linear stress limit more than 8). The finite element analysis in the commercial software ABAQUS was performed in the three-dimensional space in both Standard implicit and explicit mode. Cruciform specimen was subjected to in equal-biaxial tension (load ratio1/1), the maximum applied displacement was 1/400 of the total length of the sample or 0.7 mm .In the second load case (load ratio 1/2), the applied displacements were equal to 0.175 mm and 0.7 mm respectively. The adapted CDP model was able to simulate the nonlinear constitutive behavior of highly reinforced cement composites, more specifically their significant strain hardening capacity after crack initiation. The good agreement between the numerical simulation and the biaxial laboratory tests proved that the identified model parameters were either correct, or in sensitive for the biaxial tension stress state and thus, the load bearing behavior and failure load of in-plane loaded structures can be the adapted predicted closely by the adopted CDP model.

S. Kesteloot et al. 2005 (7) presented a study on Finite-element modeling of concrete prisms reinforced with composites plates. The study was based on Theory of cracking for the repair and reinforcement work of sewage structures using composite (carbon fiber-based) plates. Finite-element analysis software URUS Version 8.2was used for analysis. Tear-off tests involving direct tension and Four-point bending tests were performed during the experimental study in order to characterize the reinforcement (composite) and the adhesive (glue). The results of numerical calculations were then compared with the results of experimental tests. It was concluded that the fracture loads recorded for the reinforced prisms were high as compared to the reference prisms. After eight days of water saturation the fracture loads recorded for the reference prisms and reinforced prism shows the improvement of 215%. The results of modeling shows that the first cracks occurred at the supports. These cracks were inclined at 450 and spread through the concrete support. Bonding carbon-fiber-based composite plates on concrete samples subjected to different environments proved to be a means of reinforcement and repair in a wet medium because of ease in application. it was seen that there was a loss of gain in ultimate load and stiffness when the plates were glued to a wet support. A safety coefficient must be taken into account in design calculations.

Sheng-En Fang et al. (8) 2008 presented a sensitivity-based updating method to identify the damage in a tested reinforced concrete (RC) frame modeled with a two-dimensional planar finite element (FE) by minimizing the discrepancies of modal frequencies and mode shapes. In order to reduce the number of unknown variables, a two dimensional damage (element) function was proposed, resulting in a considerable improvement of the optimization performance. For damage identification, a reference FE model of the undamaged frame divided into a few damage functions was firstly obtained and then a rough identification was carried out to detect possible damage locations, which were subsequently refined with new damage functions to accurately identify the damage. From a design point of view, it was found that in a simplified way, the remaining bending stiffness of cracked beam sections or segments could be evaluated. Hence, an RC damage model based on a static mechanism was proposed to estimate the remnant stiffness of a cracked RC beam segment. The proposed damage model was based on the assumption that the damage effect spreads over a region and the stiffness in the segment changes linearly. The stiffness reduction evaluated using this damage model was compared with the FE updating result. It is shown that the proposed two dimensional damage function was useful in producing a well-conditioned optimization problem and the aforementioned damage model can be used for an approximate stiffness estimation of a cracked beam segment.

Varinder Singh et al. 2014 (9) presented a study on Finite Element Modeling of the non linear behavior of RC beam-column joints under the static load to study the response and load carrying capacity of exterior RC beam-column joints. The analysis of FE model for the two layer of carbon fiber reinforced polymer (CFRP) retrofitted RC beam-column joint was carried out by using non-linear finite element analysis software ATENA-3D. Reinforced concrete non retrofitted and retrofitted BC joints were analyzed with all aspects of materials in tension and compression. The Stress-Strain relationship used was as per I.S.456:2000. Solid brick element having minimum 8 and maximum 20 nodes was taken for modeling of concrete element having three degree of freedom (X, Y, & Z) directions at each node. Bar elements were used in ATENA for modeling of steel. The CFRP element was provided in L-shape and at 45 degree orientation to the joint in two layer finite element models of CFRP retrofitted joint shows increase of 18.78% in ultimate load carrying capacity as compared to 9.47%, as shown by experimental study. It was concluded that the increase in yield load was 14.75% in the FEM analysis, while 15% in the experimental study and also concluded that the increase in stiffness of CFRP jacketed joints in FEM was 43.29% as compared to 15.46% in experimental study. In the comparison of the results obtained from FEM with ATENA and that from the experimental analysis shows that the FEM with ATENA results are approximately similar to the experimental results for the carbon fiber reinforced polymers. The results showed a significant improvement in the ultimate load carrying capacity percent along with an increase of percent in yield load and percent increase in stiffness of the CFRP retrofitted FE beam model, when compared to control FE beam model of such exterior beam column joints.

Robert Ravi. S et al. 2010 (10) presented a study on The details of the finite element analysis of beam column joints retrofitted with carbon fiber reinforced polymer sheets (CFRP) carried out. Three exterior RC beam column joint specimens were modeled using ANSYS package. The first specimen was the control specimen. This had reinforcement as per code IS 456:2000. The second one was also the control specimen had reinforcement as per IS 13920:1993. The third specimen had reinforcement as per code IS 456:2000 and was retrofitted with (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. It was found that the deflection of beam column joint specimen detailed as per code IS 139201993 was 23.53 % less than that of specimen detailed as per IS 4562000 and deflection of the beam column joint specimen retrofitted with CRP sheet was 75.29 % less than that of the specimen detailed as IS 4562000. it was also observed that The energy absorption capacity of the specimen detailed as per code IS 139201993 was 42.86 % more than that of the specimen detailed as per code IS 4562000 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 4562000. Based on the ANSYS modeling and analysis carried out on the control and retrofitted beam column joint specimens using CFRP sheets, it was concluded that The deflection of the beam column joint specimen detailed as per code IS 139201993 was found to be 23.53 % lower than that of the specimen detailed as per code IS 4562000.

Syed Sohailuddin S.et al. 2013 (11) presented a study on finite element modeling of reinforced concrete beam column joint. Finite element modeling of four types of exterior beam-column joint specimens was carried out. The first specimen was confirmed to the guide lines of IS 13920: 1993 for seismic resistant design. Second one was detailed with additional diagonal cross bracing bars at joints and beam reinforcements. Third one having cross bars in beam region of 6 mm instead of cross bars in joint. Fourth specimen had cross bars of 8 mm instead of 6 mm in beam region. The beams were 225 mm deep by 125 mm wide and columns were 225 mm deep by 125 mm wide. The BC joint was modeled in ANSYS 11.0 (1995) with Solid 65, Solid 45 and Link 8 elements. The Solid 65 element was used to model the concrete and Solid 45 element was used to model hinge support at base. These elements have 8 nodes with 3 degrees of freedom at each node translations in the nodal x, y and z directions. The Link 8 element was used to model the reinforcement. This three dimensional spar element has two nodes with 3 DOF at each node having translations in the nodal x, y and z directions. The experimental results were validated with the analytical model developed using FE software package ANSYS11.0. The load taking capacity of the specimen with the cross bars of 8 mm in the beam region found to be more confined than the other three types of the detailing arrangements. Also the defection capacity was improved in that type; moreover the cracks were reduced in fourth specimen. The ANSYS model found to be stiffer than the experimental results. The displacement ductility was enhanced for cross bars of 8 mm in beam region specimens than that of other three specimens. The displacement ductility for the specimen with cross bars of 6 mm in beam region was increased by 17.62% as compared with the cross bracing bars in the joint region and was further increased in case of cross bars of 8 mm in beam region by 27.79% as that of the cross bars in joint region. The entire load-deformation response of the model produced compared well with the response from experimental result. The test specimens with diagonal confining bars of 8 mm in the beam region showed better performance, exhibiting higher strength with minimum cracks in the joint. The joint region of specimens of cross bars was free from cracks except some hair line cracks which show the joints had adequate shear resisting capacity. It was also observed that the provision of cross diagonal reinforcement in beam region increased the ultimate load carrying capacity and ductility of joints in the both upward and downward loading conditions. It was concluded that the increase in reinforcing bar cross section had a significant effect on the flexural strength.

Aseena N et al. 2016 (12) presented finite element modeling of an exterior RC beam column joint retrofitted with externally bonded carbon fiber reinforced polymer and aluminum plate. The analysis was carried out with the help of commercially available software ANSYS. Nonlinear static analysis was carried out to evaluate the performance of original and retrofitted models using commercially available software ANSYS. Comparative study was done for obtaining better retrofitting material by analyzing controlled specimen, CFRP retrofitted specimen, and aluminum plate retrofitted specimen. Designing of beam column joint was done by ACI and IS 456:2000. The designed structure had column with a cross section of 200 X 200 mm with an overall length of 1500 mm and beams with a cross section of 200 X 200 mm with an overall length of 600 mm. An eight nodded SOLID65 was used to model the concrete that was capable of plastic deformation, cracking in orthogonal directions, crushing. Link 180 element capable of plastic deformation was used to model the steel reinforcement having two nodes with two degrees of freedom for each node. SOLID185 with eight nodes having three degrees of freedom was used to model retrofitting by CFRP and aluminum plate. A comparison between load carrying capacity for the original and retrofitted specimen was done. The analysis shows significant improvements when applying the retrofitting material. After performing the analysis on beam column joint using aluminum plate, significant improvements were obtained by increasing the plate thickness. It was concluded that ANSYS was able to model different components of load deflection curve such as linear region, initial cracking, the nonlinear region, yielding of steel and failure. Comparison between load deflection results obtained from ANSYS for control specimen shows that the deflection values are less for retrofitted specimen. While comparing retrofitted specimens, CFRP shows lesser deflection compared to aluminum pate retrofitted specimen and the crack pattern obtained was less compared to control specimen.

A.V Singh et al. 2003 (13) presented a study on finite element analysis of beams with random material properties. The linear elastic response of beams with random material properties and deterministic static loads was examined using the displacement based finite element method. A procedure for the linear finite element analysis of a prismatic beam having random material property has been described. Due to the fact that the material property was described by a random function along the length of the beam and the direct functional form of its variation was unknown, the integration of the strain energy expression might not be possible. This makes the problem very complex and the exact form of the stiffness matrix cannot be obtained. To analyze such structures, the stiffness matrix was derived in two parts. The first part is deterministic and the second part contains spectral moments of the power spectral density function of the random variable in addition to the geometric and material properties of the beam. . Additional numerical results for beams with both ends fixed and both ends simply supported were also presented. The loading cases included both the uniformly distributed and concentrated loads. The solution was obtained using the Taylor series expansion. A numerical procedure to include inter-element correlation was also described in detail. The numerical results involving the variability of the response of beams under different deterministic loads and boundary conditions were obtained and found to agree well with the with the other literature.

Prabhu N. et al. 2015 (14) presented a study on Behavior of RCC Beam Column Joint with Special Confinement Subjected to Static Loading Numerical Investigation. Ductility, shear strength of the steel, the variation in shear strength of steel and deformations with the different provision of transverse reinforcement were presented. Finite element method, software called ANSYS was used to carry out the linear elastic analysis. The specimen consisted of beam column joint at the exterior and the corner was used. Stirrups and ties spacing was reduced on the joint portion in second specimen. Static loading was applied from 5 k N to 30 k N at the end of the beam. Deformations and shear stresses were determined for both the specimens. Maximum deformation was 0.78 mm and max shear stress was 2.18 N/mm2 for the first specimen at corner joint while at Exterior joint max deformation was 0.82 mm and max shear stress was 11.16 N/mm2. For second specimen, max deformation was 0.53 mm and max shear stress was 0.71 N/mm2 at corner joint as well as 0.63 mm & 7.3 N/mm2 respectively at exterior joint. ANSYS 14.0 was used to determine the equivalent stress, shear stress and deformation for all the specimens. As per the numerical analysis, it was concluded that the deformation was lesser for the specimen having more transverse reinforcement at corner and exterior joints. The maximum shear was more for the specimen having the less transverse reinforcement. Hence the shear strength of the joint will be high for the specimens with special confinement.

S.J. Hamil et al. 1999 (15) presented a study on finite element modeling of reinforced concrete beam-column connections. Non-linear finite element techniques were used for the modeling of reinforced concrete beam-column connection specimens. SBETA, a non-linear finite element package was used for the entire specimen modeling. SBETA had a mesh generating pre-processor which allows pre-defined concrete and steel material properties to be used. First order quadrilateral elements, formed from a pair of triangular elements with nodes at each corner, were used. One of the most important aspects of finite element modeling was the mesh design. The testing of the model followed the loading method used in the experimental investigation. The applied column load was represented by a gravity load spread over the elements above the connection zone. This was believed to reduce any stress concentrations in the elements at the top of the column. The beam load was applied in 1 k N steps, until specimen failure occurred. The model presented different types of failure mechanisms, including beam and column flexural failure, joint shear and anchorage failure; depending on the concrete strength and steel reinforcement used. Different material properties of normal and high strength concrete were matched using data from the experimental results. It was concluded that Reinforced concrete beam-column connections were modeled successfully using non-linear finite element methods. The presented model was sensitive to variations in concrete strength, the detailing arrangements of the beam tension steel and the presence (or absence) of joint ties. Throughout the modeling high levels of correlation, of good agreement, were observed. The results indicated that the strength of a joint without column ties was proportional to the square root of the concrete's compressive strength.

Yuan-Gao Zhang et al. 1994 (16) presented a Finite element modeling of reinforced concrete structure to develop a reinforced concrete model for nonlinear finite element analysis. The model performs the nonlinear behaviors of both concrete and reinforcement steel in each element. The Modeling of concrete included Failure criterion for plain concrete, Concrete behavior before failure as well as constitutive modeling of cracked concrete. Modeling of reinforcement steel was carried out by a linear elastic isotropic-hardening model to describe the material behavior of reinforcement steel. In the study Reinforced concrete panels were subjected to in-plane shear and normal stresses.

First Panel was subjected to pure shear stress and second panel was subjected to combined tension and shear. Concrete restrained beam was subjected to concentrated loads and the load deflection curve shows 3.9mm deflection at 200KN concentrated load. An improved smeared/layered reinforced concreted model was introduced which can efficiently describe the non linear behaviors of both concrete and reinforcement steel. The effects of interaction between cracked concrete and reinforcement steel, such as tension stiffening and dowel action, were properly simulated by the present model.

Aditya Kumar Tiwary et al. 2015 (17) Presented study on Strengthening of Exterior Beam Column Joint with Modified Reinforcement Technique. The finite-element program ANSYS Workbench Version 12 was used for the analysis purpose. The element details of each material were presented subsequently. The modeling of concrete was carried out using an eight-node solid (Hamil, 1999) (Tiwary1, 2015) (S, 2015) element, Solid65, was used having eight nodes with three degrees of freedom at each node with translations in the nodal x, y, and z directions. Steel reinforcement in the experimental beam-column joint was constructed with typical Grade 60 (fy-500 M Pa) steel reinforcing bars.

The steel for the finite-element model was assumed to be an elastic–perfectly plastic material with identical properties in tension and compression. A Link8 element was used to model the steel reinforcement. From the analytical study it is observed that the provision of cross diagonal reinforcement increased the ultimate load carrying capacity, stiffness and ductility of joints in the both upward and downward loading conditions.

The present study is confined to static load only. External beam-column joint reinforcement modified with crossed inclined bars modeled in ANSYS Workbench showed high strength under static applied load. From experimental data and the analytical it was concluded that the specimen with diagonal cross bar at joint shows better performance under the static loading and it was a feasible solution for increasing the shear capacity of exterior and interior beam-column joints. It was also concluded that in order to meet the requirement of strength, stiffness and ductility, high percentages of transverse hoops in the core of joints should be provided. The provision of cross diagonal reinforcement increased the ultimate load carrying capacity, stiffness and ductility of joints in the both upward and downward loading conditions.

Shabana T S et al. 2015 (18) presented a study on Finite Element Analysis of Beam Column Joint with GFRP under Dynamic Loading. The details of finite element analysis of beam column joints wrapped with glass fiber reinforced polymer sheets (GFRP) carried out using the ANSYS package. The structural modeling of an office building was done using ETABS. Beam column joints were manually designed on the basis of both IS456:2000 and IS13920:1993 by using structural data available from ETABS. Four exterior reinforced concrete beam column joint specimens were modeled using ANSYS package.

The first specimen had reinforcement as per code IS 456:2000. The second specimen had reinforcement as per code IS 13920:1993. The third Specimen had reinforcement as per code IS 456:2000 and was wrapped with GFRP sheets. The fourth specimen had reinforcement as per code IS 13920:1993 and was wrapped with GFRP 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 efficiency of confining the reinforced beam column joints with GFRP sheet wrapped at the beam column joint under dynamic loading and the results are presented in this paper. The percentage of increase in efficiency of wrapped over unwrapped was found to be 37% for beam column joint designed as per IS 456:2000 and 20% for designed as per IS 13920:1993.

III.CONCLUSION

The above referred literature clearly indicates that the use of FEM in analyzing the RCC structures is wide spread and that the method results in fairly accurate results.

REFERENCES

[1] Zienkiewic. (n.d.). finite element analysis.

[2] Bansal, N. (2013). finite element modeling of reinforced concrete corners under opening bending moment. springer , 23-33.

[3] Barbato, M. (2008). Efficient finite element modelling of reinforced concrete beams retrofitted with fibre reinforced polymers. Elsevier Ltd. , 167-176.

[4] Shaaban, I. G. (2018). Finite element modeling of exterior beam-column joints strengthened by ferrocement under cyclic loading. Elsevier Ltd , 333-346.

[5] Pleskachevskii, Y. M. (2008). FINITE-ELEMENT MOD ELING OF THE INTER ACTION OF RE IN FORCE MENT WITH CON CRETE MATRIX. Springer Sci ence+Busi ness Me dia, Inc. , 309-316.

[6] Tysmans, T. (2015). Finiteelementmodellingofthebiaxialbehaviourofhigh-performance fibre-reinforcedcementcomposites(HPFRCC)usingConcreteDamaged Plasticity. ElsevierB.V , 47-53.

[7] Kesteloot, S. (2005). Finite-element modelling of concrete prisms reinforced with composites plates- Theory of cracking. RILEM , 219-227.

[8] Fang, S.-E. (2008). Damage identification of a reinforced concrete frame by finite element model updating using damage parameterization. Elsevier Ltd , 544-559.

[9] Singh, V. (2014). Finite Element Modeling of CFRP Retrofitted RC Beam-Column Joints. International Journal on Emerging Technologies 5(2) , 31-39.

[10] Ravi.S, R. (2010). Finite Element Modeling on behavior of Reinforced Concrete Beam-Column Joints Retrofitted with Carbon Fiber Reinforced Polymer Sheets. INTERNATIONAL JOURNAL OF CIVIL AND STRUCTURAL ENGINEERING , 576-582.

[11] Sohailuddin, S. (2013). FINITE ELEMENT MODELING OF REINFORCED CONCRETE BEAM COLUMN JOINT USING ANSYS. Int. J. Struct. & Civil Engg. Res , 22-31.

[12] Aseena N. (2016). Finite Element Analysis of Retrofitted Exterior Beam Column Joint. International Journal of Innovative Research in Science , 153787-15392.

[13] A.V.Singh. (2003). finite On element analysis of beams with random material properties. Elsevier , 273-278.

[14] Prabhu N. (2015). Study on Behavior of RCC Beam Column Joint With Special Confinement Subjected To Static Loading Numerical Investigation. International Journal of Innovative Research in Engineering & Management (IJIREM) , 187-190.

[15] S. J. Hamil. (1999). Finite Element Modeling of Reinforce Concrete Beam-Column Connections. University of Durham .

[16] Zhang, Y.-G. (1994). Finite element modeling of reinforced concrete structures. Elsevier Science B.V. , 51-58.

[17] Tiwary, A. K. (2015). Strengthening Of Exterior Beam Column Joint With Modified Reinforcement Technique. IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) , 32-37.

[18] Shabana S. T. (2015). Finite Element Analysis of Beam Column Joint with GFRP under Dynamic Loading. International Journal of Engineering Trends and Technology (IJETT) , 374-379.