Joint analysis of a high rise building frame considering seismic load using Ansys software: A Review

Rahul Halder[1], Hitesh Kodwani[2]
Department of Civil Engineering
[1] PG Scholar, SAM College, Bhopal M.P.
[2] Prof. & H.O.D, SAM College, Bhopal M.P.

Abstract- *As the development rate and population rate both are growing rapidly in our country, development of high rise building is becoming popular with the motive of high population settlement rate. In designing of high rise building designers are focusing on safety for which they are considering lateral forces and providing lateral load resisting members, and focusing mainly on the joints and connection which are causing failures. Analysis tools and advance technologies have been developed which are making such big tasks very easy for the designer. In this paper we are discussing the past researches done related to analysis of structural members where joint analysis and there failure reasons are studied using analysis tools.*

Keywords- Ansys, Review, Joints, Members, Building, lateral forces.

I. INTRODUCTION

In structure designing, analysis of joints is important to justify the flow of load from upper member to lower segments to transfer the load to the soil beneath. Analysis of these joint is quit easy using analysis tool which are advance and based on advance analysis methods and modules to provide approximate accurate results and help the designer to justify their designing and sectional properties. Anaysis is termed as the most advance analysis tool for mechanical and structure analysis. This software is developed by Ansys co. which is a U.S> based company.

In this paper we are presenting a review of past researches done in structural analysis field using analysis tools. This paper will provide us the latest updated work which has been done till date in the field of joint analysis.

Alma Narcy and Brian Broderick (2018) [An assessment of the utility of ANSYS structural modelling for the seismic retrofit of steel building frames] here the authors study simulated a real situation using the software and investigates the convenience of using ANSYS where the authors study covered two relevant aspects of different type of modelling. The first one was the creation of a parsimonious model, that was, the simplest model capturing the significant information. The second one was the different types of analysis required to model the seismic response of steel building frames. The author's paper assessed the ability of ANSYS to enable the selection of suitable brace members resisting specific inter-storey drifts and floor accelerations. ANSYS Mechanical APDL might be a very powerful tool to run complex analyses as a result of the variety of options in the\ analysis, models, and results available. As a matter of fact, all the parameters required by the project have either been obtained or shown good signs of their feasibility. For the ones that have been obtained, the pushover analysis, the interstorey drift, and the linear and non-linear models can be mentioned. and for the ones that show good signs of feasibility, the dynamic analysis can be mentioned. But ANSYS Mechanical APDL lacks various features, whose existence might be very convenient for this type of not so complex systems, as for instance, quick familiarisation of the tools available or accessibility of important features of the system (definition of materials and element types).

Rajkamal and Karthikeyan (2015) [simulation and analysis of beams] the primary objective of the authors was to study the deflection of the various beams on analysis of experimental value and the innovation of the project in order to bring all types in a single apparatus. The result also found the deflection of both the rectangular and circular cross sectional beams. The objective of the project was to provide a practical knowledge on how a beam gets deflected due to different types of loads acting transversely to the axis of the beam and analysis the experiment by using ANSYS software. Here the author performed analysis of the deflection on cantilever beams, simply supported beam and fixed beam. the system was performed in two strategies they were testing investigation and another strategy utilizing structure programming. In this way, the ANSYS programming is utilized for examination. The exploratory and examination esteem were discussed in the paper. The examined table qualities are nearly the equivalent however there is a slight variation in the qualities got. Subsequently, the surrenders on the bars are all around examined utilizing both the test technique and ANSYS programming.

Bora et. al. (2018) [Finite Element Analysis of Different Types of FRP on Beam-Column Joint] here the author study was primarily focused on analysis of the beam-column joint using ANSYS where the design was carried out on EtabS and as per the design analysis was done in ANSYS. The equivalent stresses and the total deformation generated in the beam-column joint is calculated. Due to the bad effects of environmental changes, pollution, carbonation, corrosion occurring in a structural element due to which the building is likely to collapse. The failure in the beam-column joint generally occurs due to these reasons and also due to lack of proper reinforcement in the joint region. While experiencing seismic activity with inadequate reinforcement the beam-column joint will fail. This project aimed in introducing FRP in the beam column joints which are likely to fail by retrofitting it. The failure can be identified by observing cracks or carrying out NDT tests to find out the region of cracks and the amount of reinforcement corroded. In this study different types of FRP is introduced to the beam-column joint in ANSYS. It is7analysedfand then the equivalent stresses and total deformations generated are compared with each other which ever will be more effective to be used in the field of construction.

From the results, it was concluded that upon using the different types of FRPs the equivalent stresses in the concrete got reduced. The reduction is basically due to the fact that the FRP is acting like an external reinforcement to the beam column joint and the stresses generated is taken by the FRPs. Different types of FRPs have different modulus of elasticity and different Poisson’s ratio. Therefore, the variation was observed. The maximum equivalent stress was generated at the junction of the FRP and the beam of the beam-column joint because of the fact that the load applied on the beam is counteracted by the FRP and at the end point of the FRP the stresses are concentrated.

Dahmani et. al. (2010) [Crack identification in reinforced concrete beams using ANSYS software] here the author developed Three-dimensional nonlinear finite element model of reinforced concrete beam for their study. The general purpose finite element package, ANSYS 8.0, was employed for the numerical analyses. Using SOLID65 solid elements, the compressive crushing of concrete was facilitated using plasticity algorithm while the concrete cracking in tension zone was accommodated by the nonlinear material model. Smeared reinforcement was used and introduced as a percentage of steel embedded in concrete. Comparison with hand calculated results was presented for the concrete beam. Convergence of analytical results was showed. The capability of the model to capture the critical crack regions, loads and deflections for various types of loadings in reinforced concrete beam was illustrated. Finite element models of 3.0 m ordinarily reinforced concrete beams, constructed in ANSYS V8.0 using the dedicated concrete elements have accurately captured the nonlinear flexural response of these systems up to failure. The dedicated element employs a smeared crack model to allow for concrete cracking with the option of modeling the reinforcement in a distributed or discrete ways. In terms of using finite element models to predict the strength of existing beams, the assignment of appropriate material properties was critical. The author stated various conclusions - The following conclusions can be stated based on the evaluation of the analyses of the reinforced concrete beam: the load applied to cause initial cracking of the reinforced concrete beam well correlates with hand calculations; flexural failure of the reinforced concrete beam was adequately modeled using a finite element package, and the load applied at failure was very close to hand-calculated results. The good results attained suggest that, in spite of the relative simplicity of the analyzed structure and of the employed models, satisfactory prediction of the response of reinforced concrete structures may be obtained. The failure model of concrete adopted by the commercial code ANSYS is adequate to determine the nonlinear behavior of reinforced concrete structures.

B. Venkatesan and R. Ilangovan (2016) [Finite Element Analysis (FEA) for the Beam-Column Joint Subjected to Cyclic Loading Was Performed Using ANSYS] the authors analyzed the seismic performance of exterior beam-column joints strengthened with unconventional reinforcement detailing. The beam-column joint specimens were tested with reverse cyclic loading applied at the beam end. The samples were divided into two groups based on the joint reinforcement detailing. The first group (Group A) of three non-ductility specimens had joint detailing in accordance with the construction code of practice in India IS456-2000, and the second group (Group B) of three ductility specimens had joint reinforcement detailed as per IS13920-1993, with similar axial load cases as the first group. The experimental studies are proven with the analytical studies carried out by finite element models using ANSYS. The results show that the hysteresis simulation is satisfactory for both un-strengthened and ferrocement strengthened specimens. Furthermore, when ferrocement strengthening is employed, the strengthened beam-column joints exhibit better structural performance than the un-strengthened specimens of about 31.56% and 38.98 for DD-T1 and DD-T2 respectively. The analytical shear strength predictions were in line with the test results reported in the literature, thus adding confidence to the validity of the proposed models. Here the author concluded from the results that the ferrocement for retrofitting increased the energy dissipation capacity and is more efficient for reinforced beam-column joints in seismic regions. In addition, the non-ductile reinforced beam-column joint can be vitalized by strengthening using ferrocement laminates. The non-ductile specimen ND-1, compared with the ferrocement-strengthened ND-T1 and ND-T2, showed increased strength, stiffness, and energy dissipation capacity by 33.11%, 45.40% and 23.8% and 37.5%, 64.39% and 74.07%, respectively. Analytical modelling gives a close prediction of the experimental behaviour of beam-column joints.

T. Subramani and A. Mohammed Ali (2017) [Analytical Study Of T-Beam Using ANSYS] here the authors paper presented finite element analysis for modeling T-beams structure used in building service system (mechanical, electrical, communications, and plumbing). The experimental program reported in this paper tested T-beams to failure effect on various beam behaviors. Using ANSYS, finite element models were developed to simulate beam deflection behavior. In reality, uncertainties exist in a system and environment that may make the application of deterministic design unreliable which causes the values of the variables that are acting on the system cannot be predicted with certainty. From the analysis results, it was observed that the changes in prestressing force, elastic modulus of prestressing steel, ultimate tensile strength of prestressing steel and beam width tend to be the most influencing parameters, which need to be tightly controlled. The analytical results of prestressed beams with the corresponding experimental data stated that the predicted load in of T-beams at various stages was found to be in good agreement with the test data, the proposed model predicted slightly softer results in post-cracking regime of the load deflection response of T-beams. This variation is due to the difference in the bond-slip model of reinforcement used in the analysis when compared with that present in the test. The 'ANSYS' model correctly predicted the diagonal tension failure and shear compression failure of prestressed concrete beams observed in the experiment. It is expected that the modeling strategy for the finite element analysis proposed in this study will be used for designing/ analyzing SFRC members.

T. Subramani et. al. (2014) [Finite Element Modeling On Behaviour Of Reinforced Concrete Beam Column Joints Retrofitted With CFRP Sheets Using Ansys] here the author carried out 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. 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. The author derived the conclusions based on the ANSYS modeling and analysis carried out on the control and retrofitted beam column joint specimens using CFRP sheets, the deflection of the beam column joint specimen detailed as per code IS139201993 was found to be 23.53 % lower than that of the specimen detailed per code IS 4562000, the deflection of the beamcolumn joint specimen retrofitted with GFRP sheet reduced the deflection about 75.29 %.when compared with the deflection of specimen detailed as per code IS 4562000, the energy absorption capacity of the beamcolumn joint specimen detailed as per code IS 139201993 was found to be 42.89 % higher than that of the specimen detailed per code IS 4562000 and the energy absorption capacity of the beamcolumn joint specimen retrofitted with CFRP sheet increased about 114.29 %.when compared with the energy absorption capacity of specimen detailed as per code IS 4562000.

Lokhande et. al. (2018) [Project Report on Deep Beam Analysis using ANSYS] here the author considered a model of cantilever rectangular deep beam which can dissect the model with fem program by Dr P N Godbole and ANSYS 2D with Plane 82. Ansys was more precise software program when contrasted with staad genius, Etabs. Analyze Cantilever Rectangular deep beam with 4nodes Element by Ansys 2d with plane 182 and fem program. Break down Ansys 2d with plane 183 of 8 nodes component and fem program. Deep beam was discovered when L/D was not as much as equivalent to 2 was called as deep beam. Continuous beam were considered as deep when the proportion L/D was under 2.5. The successful traverse was characterized as the inside to focus remove between the backings or 1.15 times the unmistakable traverse whichever is less.For displacement in regular and fine work, plane 182 and plane 183 and Eulers Bernoullis Theory (EBT) comes about are shut to diagnostic solution, for utilize, so for this situation FEM is smarter to use. By an indistinguishable reason from FEM was\s proposed for Normal and Shear stress. FEM was prescribed and better for use to solve the problems. Deflection esteem given by EBT are considerably less than the esteem given by Fem program and Ansys program if there should be an occurrence of rectangular deep beam. From the outcomes given by fem program, Ansys it is discovered that the flexural stress distribution in deep beam isn't straight as in the event of slim beam, But EBT gives the flexural stress distribution as direct for deep beam. EBT gives the greatest shear stress at the middle line of the beam, however in deep beam the most extreme shear stress it was beneath the inside line of the beam.

II. CONCLUSION

It can be said that in past no detailed study on joint analysis has been done using rigid frames. In past researches were conducted on different loading conditions including live load, wind load, thermal load etc. however information on technoeconomic feasibility of loads to be used in buildings is lacking.

- In all of the previous work generally 2-d analysis is done.
- In previous studies no comparison was done on the effects of diaphragm with considering different types of slab.
- In previous works no one implemented analysis of structure using I.S. specifications whereas in this study we will provide research work as per Indian standard specifications.

REFERENCES

[1] Bindhu, K.R. and Jaya, K.P. (2012) Strength and Behavior of Exterior Beam Column Joints with Diagonal cross Bracing Bars. Asian Journal of Civil Engineering (Building and Housing), 11,397-410.

[2] Dogan, E. and Neven, K. (2003) Seismic Retrofit with Continuous Slurry Infiltrated Mat Concrete Jackets. ACI Structural Journal, 100, 713-722.

[3] Maria, J.F. and Chris, G.K. (2014) Influence of Pinching Effect of Exterior Joints on the Seismic Behaviour of RC Frames. Earthquake and Structure, 6, 89-110.

[4] Matthew, J.F. and Halil, S. (2011) Behavior of Exterior Reinforced Concrete Beam-Column Joints Including a New Reinforcement. Structural Engineering and Mechanics, 40, 867-883.

[5] Ramakrishna, R.V.S. and Ravindra, V. (2012) Experimental Investigation on Rehabilitation of Reinforced Cement Concrete Interior Beam-Column Joints Using CFRP and GFRP Sheets. International Journal of Engineering Science and Technology, 4, 874-881.

[6] Hamid, N.H., Hadi, N.D. and Ghani, K.D. (2013) Retrofitting of Beam-Column Joint Using CFRP and Steel Plate. International Journal of Civil, Architectural Science and Engineering, 7, 300-305.

[7] Robert, R.S. and Arulraj, P.G. (2010) Experimental Investigation on Behavior of Reinforced Concrete Beam Column Joints Retrofitted with GFRP-AFRP Hybrid Wrapping. International Journal of Civil and Structural Engineering, 1, 245-253.

[8] Umut, A. and Stefano, P. (2002) Assessment and Design Procedure for the Seismic Retrofit of Reinforced Concrete Beam-Column Joints Using FRP Composite B Structures Subjected to Seismic Forces. Journal of Composites for Construction, 16, 21-34.