8th International Conference on Seismology & Earthquake Engineering
11-13 Nov. 2019, Tehran, Iran

MODELING THE BEHAVIOR OF REINFORCED CONCRETE BEAM-COLUMN EXTERIOR JOINT USING ANSYS WITH THE COMPARISON OF RESULTS OF ANALYTICALLY AND EXPERIMENTALLY

Somayeh GHENAATIANJOBZARI
M.Sc. Graduate, Kharazmi University, Tehran, Iran
somayeh.ghenaatian@gmail.com
Gholamreza NOURI
Professor of Structural Engineering, Kharazmi University, Tehran, Iran
r.nouri@khu.ac.ir

Keywords: Beam-column connection, Ansys, Nonlinear analysis, Finite element analysis

The beam-column joint is the most affected area in a reinforced concrete moment resisting frame. It is unprotected to large forces during severe ground shaking and its behavior has a significant influence on the retort of the structure. Joints are a crucial part of the transfer of force reactions and moments effectively between the connecting elements like beams and columns. In this paper, the main aim of the study is modeling the behavior of reinforced concrete beam-column exterior joint using ANSYS with the comparison of results of analytically and experimentally.

El-Amoury and Ghobarah performed cyclic tests on exterior beam-column joints that represented the joints built following pre-1970's codes. Sharma model that simulated the shear behavior of reinforced concrete connections in structures subjected to seismic loads was proposed. The model used limiting principal tensile stress in the joint as the failure criteria so that due consideration was given to the axial load on the column. The spring characteristics were based on the actual deformations taking place in the sub-assemblage due to joint shear distortion.

In this paper, a finite element model of the beam-column joint is done in ANSYS. Modeling is an important feature in Finite Element Analysis. Improper modeling of the structures leads to unexpected errors in the solution. Hence, proper care should be taken to modeling the structures. Finite Element modeling of beam-column joints in ANSYS consists of three stages, which are listed below. I) Selection of element type, II) Assigning material properties, III) Modeling and meshing the geometry.

The external beam-column joint is hinged at the top and bottom column ends and subjected to a cyclic load applied at the beam tip. A constant axial gravity load of 600 KN is applied to the column. After going through literature and after several initial trials, the elements for modeling various materials are finalized and element type opted for modeling concrete member is solid65 and for rebar element link8 is adopted. The typical views of the reinforcements detailed generate by the Ansys program are shown in Figure 1.

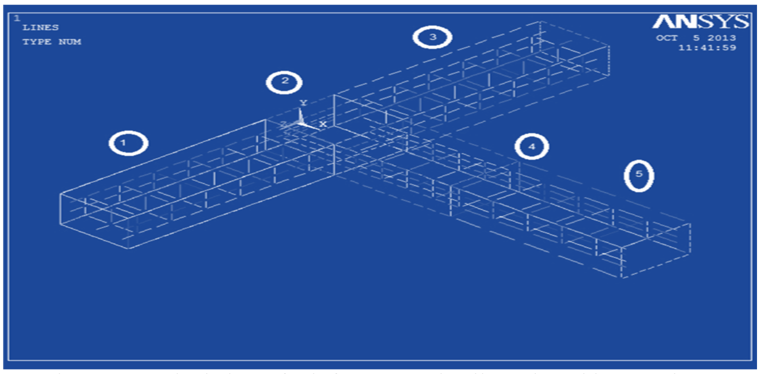

Figure 1. Typical view of reinforcement details and meshing specimen.

The comparison between the displacement of the beam in different cycles loading in finite element and analytical methods with experimental results as well as there have been comparisons between cracks created in different cycles of loading in finite element modeling and experimental results. The results show that static nonlinear analysis for the load-displacement behavior of the connection, with the behavior of the hysteresis rule in the laboratory modeling are very close. Also, the ultimate strength and yields obtained in finite element results in comparison with methods analytically methods are closer to the laboratory results. They are shown in Table 1 and Figure 2. In conclusion, this study also proves computational study is showing almost similar results from the experimental and analytical.

Table 2. Comparison of laboratory, analytical and finite element results.

Specimen reference	Experimentally observed resistance at yield (initial joint cracking)	Analytically obtained resistance using joint model corresponding to yield (KN)	Computational obtained resistance using finite element model corresponding to yield (KN)	Experimentally observed ultimate resistance (KN)	Analytically obtained ultimate resistance using joint model (KN)	Computational obtained ultimate resistance using finite element model (KN)
down	73.8	79.5	75.6	86.0	104.0	89.5
up	48.2	50.8	48.8	60.0	65.0	61.2

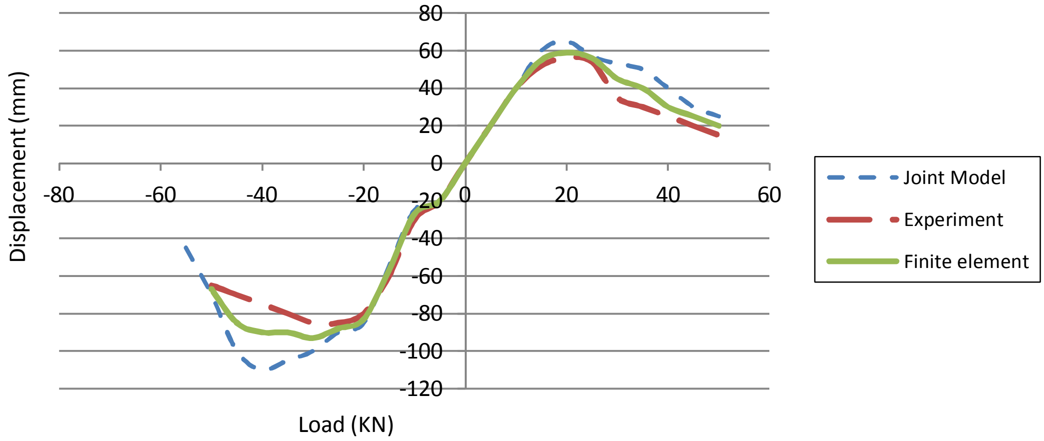

Figure 2. Load-displacement diagram for different methods of connection modeling.

REFERENCES

Bsisu, K.A.-D. and Hiari, B.O. (2015). Finite element analysis of retrofitting techniques for reinforced concrete beam-column joint. *J. Am. Sci.*, *11*(8), 48-56, ISSN: 1545-1003.

El-Amoury, T. and Ghobarah, A. (2002). Seismic rehabilitation of beam–column joint using GFRP sheets, 1397-1407.

Sharma, A., Eligehausen, R., and Reddy, G.R. (2011). A new model to simulate joint shear behavior of poorly detailed beam–column connections in RC structures under seismic loads, Part I: Exterior joints.