Finite Element Analysis of Beam-To-Column Joint In Steel Structure Subjected To Monotonic Load And Cyclic Load

Mohammadshoaib R. Shabhai

Master of Engineering(CAD/CAM),A.D.Patel Institute Of Technology,New V.V.Nagar, Anand,Gujrat

ABSTRACT

This paper is aimed to present a simple and accurate three-dimensional (3D) finite element model (FE) capable of simulating the actual behavior of beam-to-column joints in steel frames subjected to lateral loads. ANSYS is used to model the joint. The bolted extended-end-plate connection was chosen. It is important type of beam–column joints. The extended-end-plate connection is chosen for its complexity in the analysis and behavior due to the number of connection components and their inheritable non-linear behavior. Two experimental tests in the literature were chosen to verify the finite element model. And comparison of results of both the experimental and the proposed finite element carried out. One of these tests was monotonically loaded, while the second was cyclically loaded. In ANSYS instead of node-to-node contact element, surface-to-surface element is used as a contact element to enhance the model. The FE results show good correlation with the experimental one. An attempt to improve a new technique for modeling bolts is conducted. The results show that this technique is supposed to avoid the defects above, give much less elements number and less solution time than the other modeling techniques.

Keywords : *finite element analysis, beam-to-column joint, cyclic load, lateral load, steel frame*

1.INTRODUCTION

The behavior of beam-to-column joints in steel frames can be conveniently represented by its flexural behavior which is primarily shown by the moment–rotation (M–θ) relationship. This behavior is non-linear even at low load levels. In fact, moment–rotation curves represent the result of a very complex interaction among the elementary parts constituting the connection. The potential economic implication of connections on frame design is realized by code provisions [2,3]. As a result, special design guides for moment resisting connections have been developed [4,5]. Since the connection types are highly indeterminate, current design approaches cannot model three-dimensional (3D) systems which are governed by complex combined material and geometrical non-linearities, friction, slippage, contact, bolt–end plate interactions and, eventually, fractures. Hence, the finite element technique has been adopted as a rational supplement to the calibration of design models.

Krishnamurthy was the pioneer in the field of 3D modeling of connections, by adopting eight-node sub-parametric bricks in order to reproduce the behavior of bolted end plate connections [6]. The analyses carried out were linearly elastic but expensive, because contact was embodied artificially by attaching and releasing nodes at each loading step on the basis of the stress distribution. Bolt preloading phenomena were simulated also. Then, a correlation between two-dimensional (2D) and 3D finite element analyzes was established and a parametric study was conducted with 2D models, owing to the limited computer capabilities. A similar procedure was proposed by Kukreti et al. [7] in order to reproduce moment–rotation relationships of end plate connections. The results of these analyses were useful in the range for which such validations were performed. However, fundamental issues relating to the number of integration points, kinematic description, element type and discretization were not investigated. These models can be classified as hybrid since they encompassed solid elements for both plates and bolts and plane elements for both web and stiffeners. Satisfactory results were obtained using these models. Nevertheless, some basic issues like discretization and type of yield criterion were faced up. The main purpose of the proposed finite element (FE) model is to simulate the moment–rotation behavior of extended endplate connections subjected to either monotonic or cyclic loads. This model has the ability to be modified easily and then resolved repeatedly to take the different geometric and material parameters that affect the behavior of such joints into account.

2. BEAM-TO-COLUMN JOINT UNDER MONOTONIC LOAD

In order to analyze beam-to-column joint under lateral monotonic load, a finite element model is presented using the software package ANSYS. The model is utilized to investigate the results of both experimental and theoretical models made by Khalil [8] for steel beam-to-column connections.

2.1. Joint configuration

Fig. 1 shows the typical configuration of the joint which consisted of a rectangular end-plate welded to the beam cross-section and fixed to the column flange by three rows of bolts of diameter 22 mm, M22, and grade 10.9, two of them at the tension side of the connection (one above and the other below the tension beam flange) and the third above the compression beam flange. The dimensions of the used bolts and its nuts also shown.

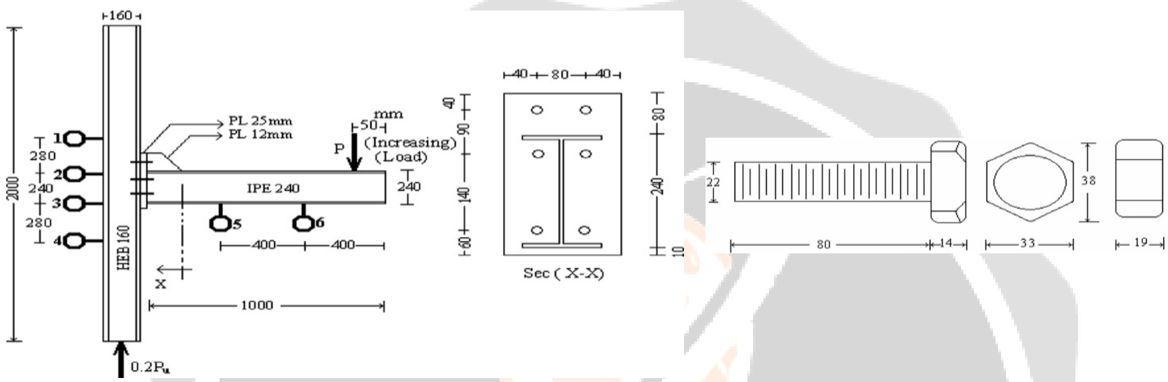


Figure 1 *:* Beam-to-column steel joint by Khalil [15]

2.2. Loading system

For the studied joint, two concentrated loads were used. The first acted axially on the bottom of the column (upward) to generate constant concentrated load equal to 0.2Pu [8]. The second load increasingly acts 50 mm away from the tip of the beam to generate an increasing bending moment during the loading, as shown in Fig. 1. Six dial gauges were used to measure both the vertical and horizontal displacements of the connection. Dial gauges from 1 to 4 were used for horizontal displacements, whereas gauges 5 and 6 were used for vertical displacements.

2.3. Finite element model

The details of the finite element model used in this part are very closely based on the models presented by Khalil [8] (Fig. 3). Node-to-node contact elements and hybrid bolts are used in the modeling. Due to symmetry about a plane passing through the beam and column webs, only one half of the connection is considered in the modeling.

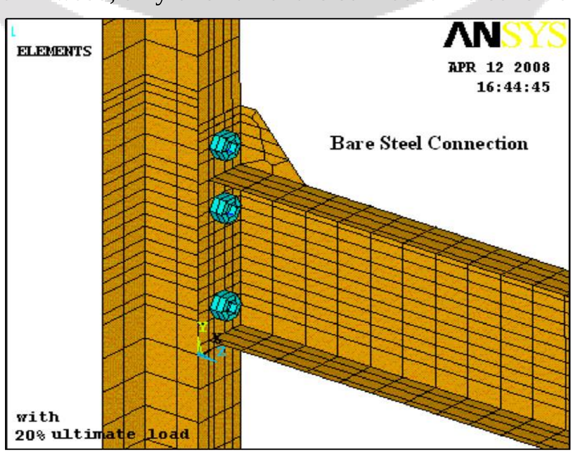


Figure 2 : General view of the studied connection under monotonic loading

2.3.1. Material properties

The stress–strain curves are taken as elastic-strain hardening. This is acceptable since strain hardening is paired with excessive yielding in large areas and a large deflection criterion governs the ultimate strength design. However, in end-plate connections excessive strain is mostly local and besides considerable shear stresses occur in the region between the top bolts and the beam tension flange which necessitates considering strain hardening. Stress–strain curves for HS (high strength) bolts, and steel sections, with the values of stresses and strains, are shown in Fig. 3a and b, respectively. bolts, and steel sections, with the values of stresses and strains, are shown in Fig. 3a and b, respectively.

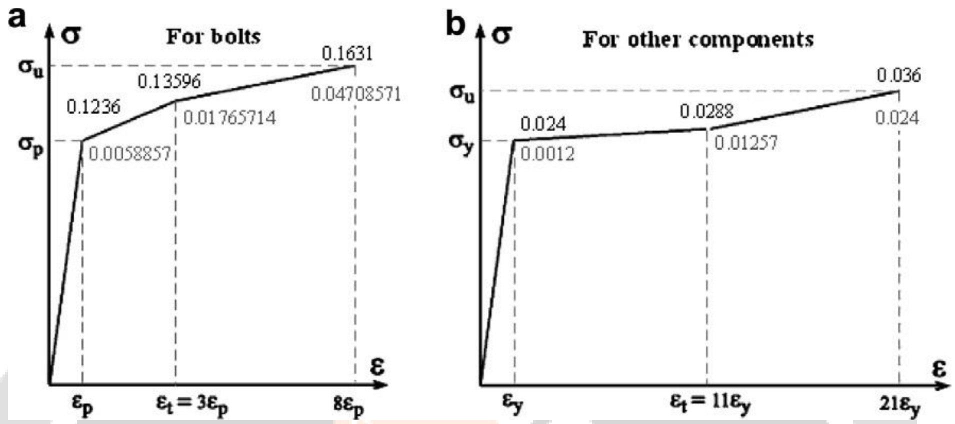


Figure 3 : Tri-linear stress–strain curve: (a) for high strength bolts; (b) for steel sections.

Table 1 : Axial forces in the bolts

Location of bolt	Axial force (ANSYS) (ton)	Axial force in Ref. [8] (ton)	Percentage of difference (%)
Upper bolt	6.02	6.49	7.2
Intermediate bolt	3.32	3.44	3.49
Lower bolt	0.487	0.32	52

2.3.2. Boundary conditions

Upper column end is a pinned support while the lower end is a roller support along the vertical axis (direction of the column axis), as shown in Fig. 4. Due to symmetry, only half of each connection is modeled. Symmetric displacement boundary conditions are defined for the nodes along the plane of symmetry.

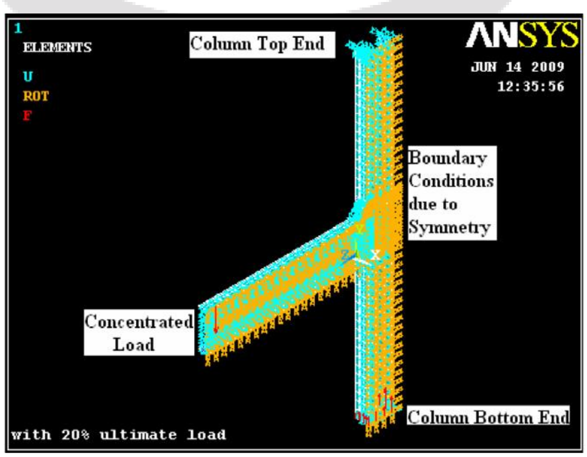


Figure 4 : Boundary conditions [load and constraints]

2.4. Results of the FE modeling

The main aim of this section is to declare the accuracy of the results obtained by the finite elements models established using ANSYS.

2.4.1. Connection bending moment

The connection bending moment is transferred by both axial tension in the bolts and compressive forces in the contact elements between the end-plate and the column flange.

2.4.1.1. Axial forces in bolts

Table 1 show the axial forces in the shanks of the bolts at load level 4.0 ton as an example. The difference in the lower bolt is large since the value itself is small. Also, the axial forces in the links of tension bolts near the axis of symmetry (beam web) are greater than those far away from the axis of symmetry.

2.4.1.2. Compression forces in contact elements.

There are distribution of the contact forces between the end-plate and the column flange. There are two main regions, the first is on the bottom part (near to the beam compression flange) and the second is on the top part (near the most stressed bolts on the upper outer side) resulting from the prying forces on the connection.

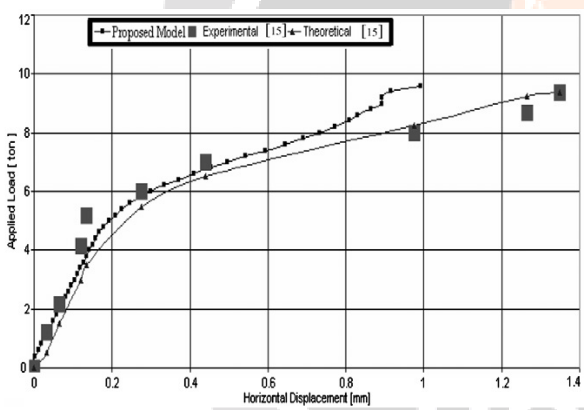


Figure 6 Results at the location of Dial (1)

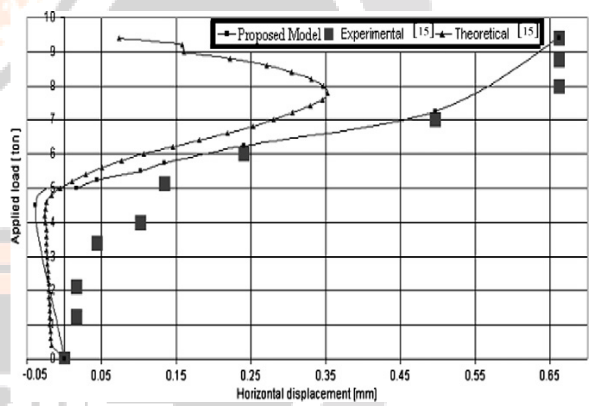


Figure 5 Results at the location of Dial (2)

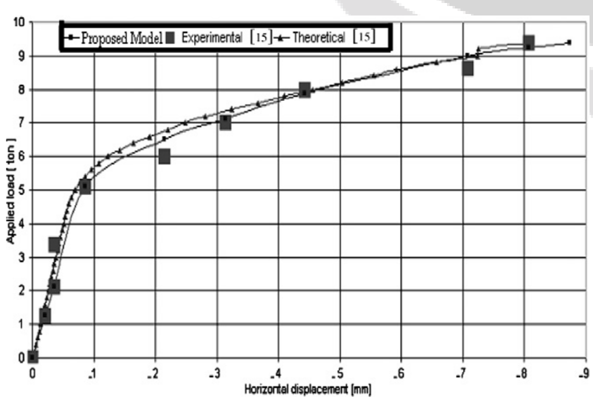


Figure 7 Results at the location of Dial (3)

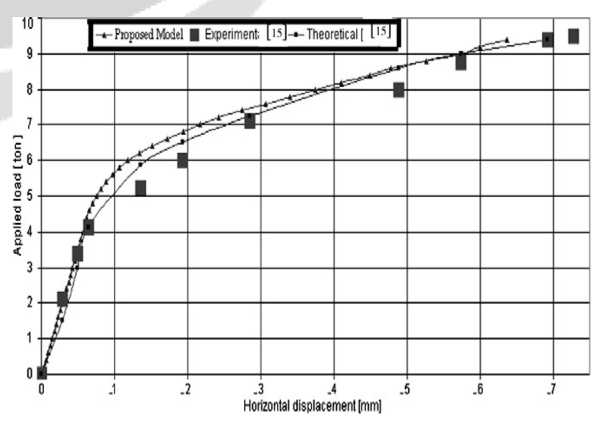


Figure 8 Results at the location of Dial (4)

2.4.2. Load–deflection relationship

The results of the horizontal displacements in the location of Dials 1–4 Fig. 1, obtained both experimentally and theoretically by Khalil [8] are compared with those obtained using finite element modeling. Fig. 9 shows the deformed shape of the connection at failure.

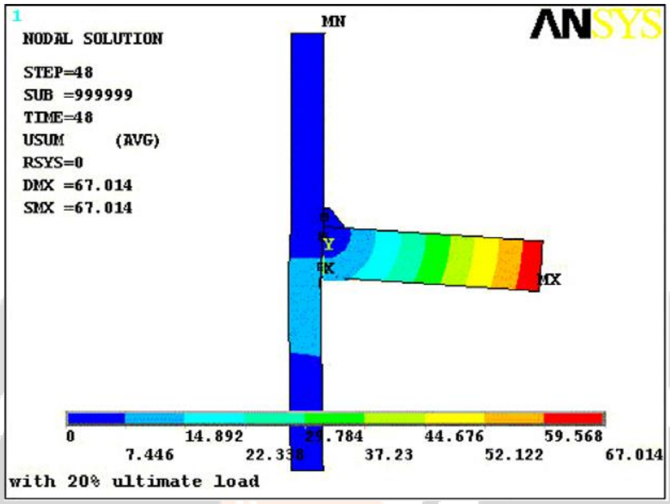


Figure 9 Deformed shape of the connection just before failure

The most obvious things are the deflection of the beam and the kinking of the panel zone. The results and comparisons of the deflection versus load history are shown in Figs. 5–8. Positive sign means that the horizontal displacement is in the right direction (Z-direction or direction of the beam web). The results of the FE model at the location of Dial gauge (2) are different from the experimental results, Fig. 6. Experimentally, Dial gauge (2) is in the region of panel zone kinking and tension bolts which may lead to errors in the very small readings of the dial gauge. Correlation coefficient is used to find the degrees of association that exists between the experimental deflections and the finite element results. Correlation coefficients calculated are 0.982167, 0.985492, 0.987595, 0.995761 for dials from 1 to 4, respectively,[1]. Failure load in FE model was 9.4 ton while it was 9.38 ton in the experimental model.

3. BEAM-TO-COLUMN JOINT UNDER CYCLIC LOAD

3.1. Joint configuration

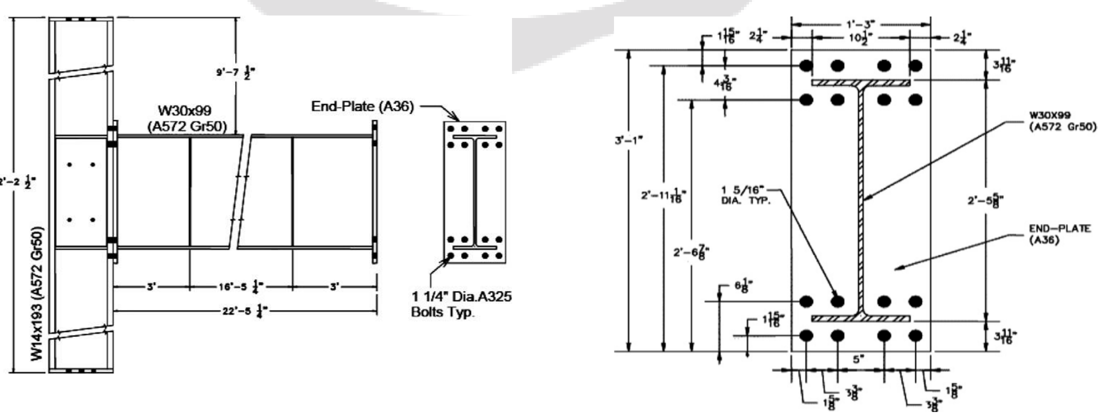


Figure 10 : Connection configuration and End-plate layout

The test specimen, chosen to verify the finite element model, consists of a W14x193 (A572 Gr.50) column with a single W30x99 (A572 Gr.50) beam attached to the flange. In this test, the connection was designed to develop 80% of the nominal plastic moment capacity of the beam. The joint configuration and its details are shown in Fig.10.

3.2. Loading protocol

The specimen was loaded cyclically according to the standard load history recommended by AISC [9]. In this protocol, the interstory drift angle, h, imposed on the test specimen is controlled as shown in Fig. 11.

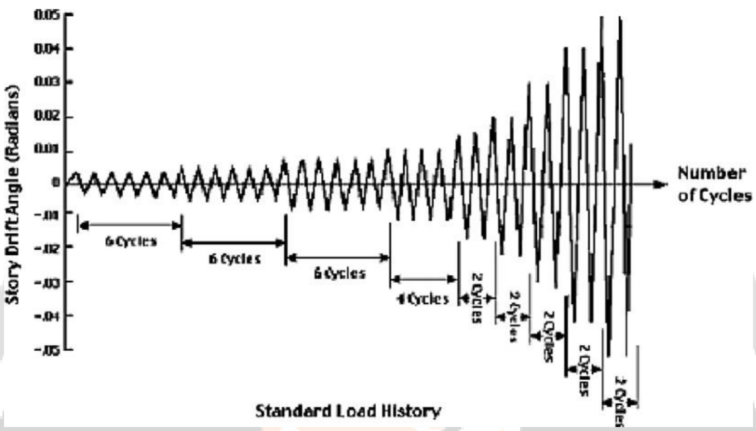
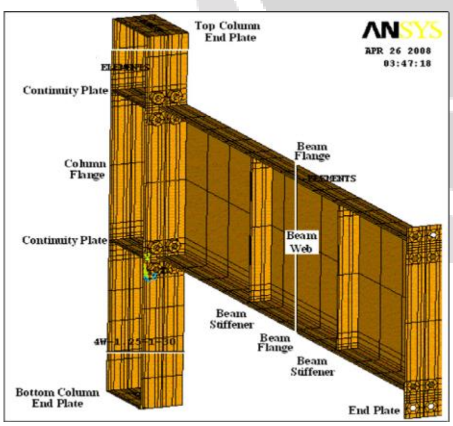


Figure 11 : Standard load history recommended by AISC [9]

3.3. Finite element model

Due to symmetry about a plane passing through the beam and column webs, only one half of the connection is considered in the modeling. Finite element types used in the modeling of beams, columns, end-plates, and bolts are the same as for the previous model. Young's modulus, Poisson's ratio, and Friction coefficient are equal to 29,870 ksi, 0.3, and 0.5, respectively.



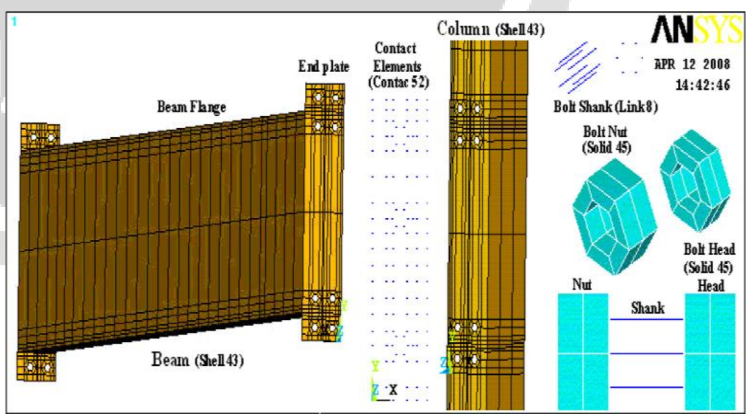


Figure 12 : General view of the connection **Figure 13** : Main components for the modeling of the connection

Figs.12 and 13 show general view and main components of the analyzed connections, respectively. Both column ends are hinged supports. Six lateral supports for beam flanges constrained in X-direction are provided at a distance of 4 ft. 1 in., 12 ft. 1 in. and 18 ft. 9 in. from the centerline of the column. The free end of the beam is considered as a roller support in the vertical direction. The loading (displacement control) was applied to the centerline of the beam at a node located on the upper flange at a distance of 20 ft. 1 1/4 in. from the centerline of the column.

3.4. Results of the FE model

Table 2 shows a comparison between the experimental results and the FE results.

Table 2 : Results of both models

Item	Experimental results [10]	FE results	Percentage of error (%)
Maximum applied moment (kips in)	18,521	17,606.425	4.9
Corresponding peak applied load (kips)	76.77	72.98	4.9
Maximum inelastic story drift (rad)	0.021	0.028	33.33
Moment corresponding to the maximum inelastic story drift (kips in)	18,120	17,233.1	4.9

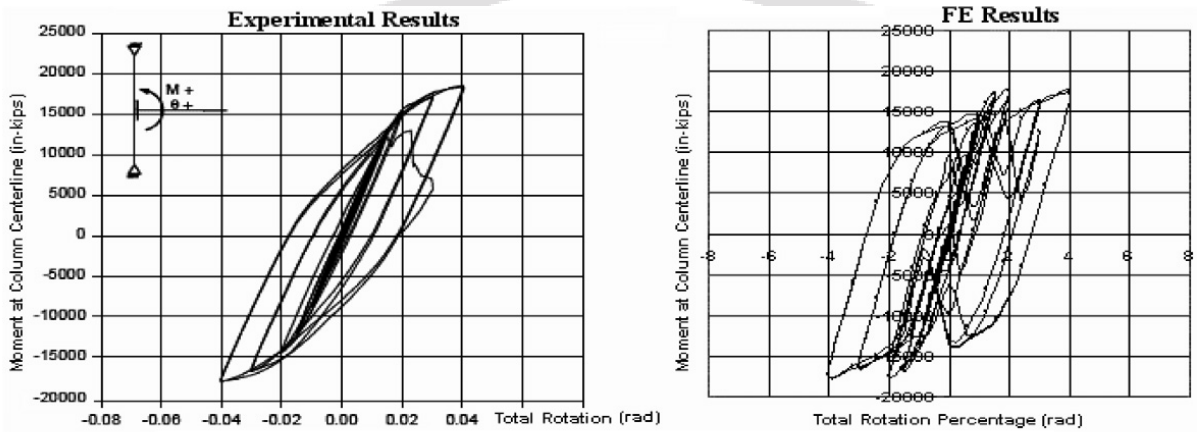


Figure 14 *:* Relation between total rotation and moment at column centerline.

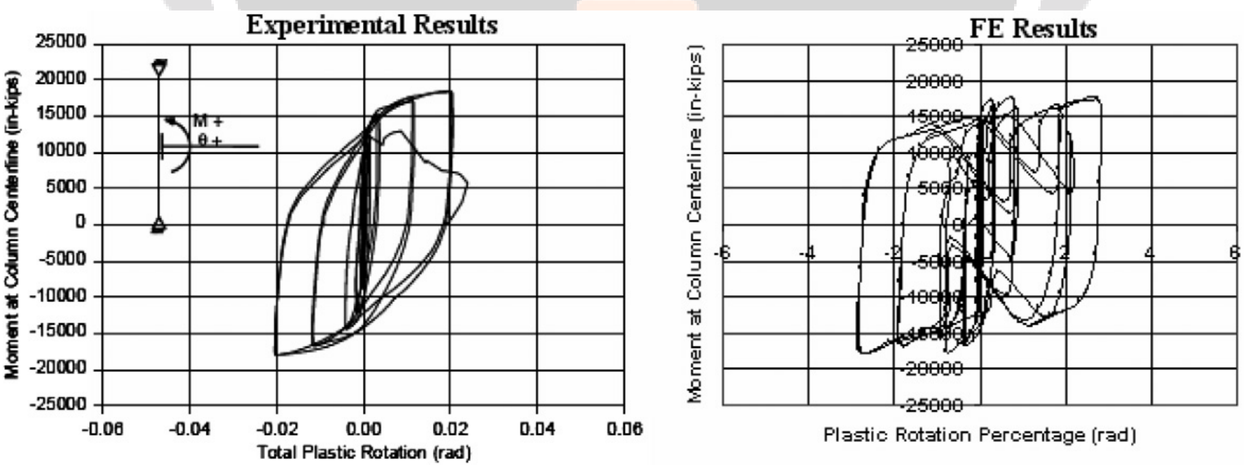


Figure 15 Relation between plastic rotation and moment at column centerline.

Figs.14 and 15 show the relations between the moment at the column centerline and both the total and plastic rotations, respectively; for both experimental and FE models. Figs. 14 and 15 indicate that there is an increase in softening and stiffening for both loading and unloading stages. This is mainly due to using of node-to-node contact elements CONTAC52. These elements lead to both stiffening and softening according to their being compressed or tensioned, respectively. When the element force is compression, the interface remains in contact and responds as a linear spring leading to increasing of the structure stiffness. As the normal force becomes tension, contact is broken and no force is transmitted leading to decreasing of the structure stiffness.

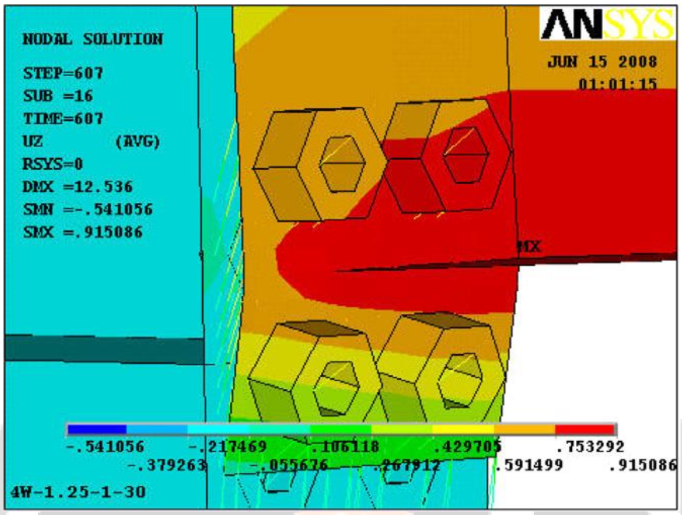


Figure 16 Rupture of the inner two bolts inside the beam

Experimentally, rupture of the two inner bolts inside the bottom flange of the beam and tearing through the thickness of the end-plate between the two inner bolts were observed. In the FE model, rupture in the two inner bolts inside the bottom flange of the beam happened as in the experimental model (Fig.16). Rupture in the bolt is, instead, realized when the location of the nut is out of the length of the shank, nut is moving away from the end-plate a distance >(L_{Shank} Clear $=L_{Shank} - t_{cf} - t_{ep}$) where L_{Shank} is the shank length, t_{cf} the column flange thickness and t_{ep} is the end-plate thickness.

4. CONCLUSIONS

1. Comparison of FE results and the experimental results done to examine the validity and the predictability of the proposed model. We found that FE results have good agreement with the experimental one at different stages of loading.
2. The FE model has variety of results at any location within the model. Observation of the full fields of stresses and strains are possible in the FE model. This gives a great advantage in monitoring the components of the connection.
3. Although its of great advantages, it is shown that modeling a beam-to-column connection loaded cyclically is expensive and time consuming in both building and solving the model. So, there is a great need to model the connection more simply and at the same time with an acceptable accuracy.

5.REFERENCES

[1] Elsayed Mashaly, Mohamed El-Heweity, Hamdy Abou-Elfath, Mohamed Osman "Finite element analysis of beam-to-column joints
in steel frames under cyclic loading", "Alexandria Engineering journal", May 2011
[2] American Institute of Steel Construction, Manual of Steel Construction – Load and Resistance Factor Design, first ed.American Institute of Steel Construction, Chicago, 1986
[3] Eurocode 3, Design of Steel Structures Part 1.1: General Rules and Rules for Buildings, ENV 1993-1-1, Brussels, 1992.
[4] American Institute of Steel Construction, Extended End-Plate Moment Connections, Steel Design Guide S, vol. 4, American Institute of Steel Construction, Chicago, 1990.
[5] Eurocode 3, New Revised Annex J: Joints in Building Frames,ENV 1993-1-1/pr A2, Brussels, 1994.
[6] N. Krishnamurthy, D. Graddy," Correlation between 2 and 3-dimensional finite element analysis of steel bolted end-plate connections", Journal of Computer and Structures 6 (1976) 381–389.
[7] A.R. Kukreti, T.M. Murray, A. Abolmaali, "End plateconnection moment–rotation relationship", Journal of Constructional Steel research 8 (1987) 137–157.

[8] N.N. Khalil, Composite Frames with Semi-Rigid Joints, Ph.D.Thesis, Faculty of Engineering, Tanta University, Tanta, Egypt,2005.
[9] American Institute of Steel Construction Inc. (AISC),Manual of Steel Construction – Load and Resistance Factor Design Specification for Structural Steel Buildings, second ed.1994.
[10] FEMA-355D, 2000i, State of the Art Report on Connection Performance, Prepared by the SAC Joint Venture for the Federal Emergency Management Agency, Washington, DC,2000.