Numerical Study of Four Bolts End-Plate Joint Behaviour for Robustness Assessment

Bashir Saleh
b.Saleh@il.pw.edu.pl ; bashiak2003@gmail.com
Libyan Academy, Tripoli, Libya and High Institute of Engineering Technology, Ghryan, Libya

ABSTRACT

The paper presents new studies on numerical modeling (FEM) for beam-to-column behavior under sagging and hogging bending moment when the framework is exposed to service and unexpected loads that may cause a column loss scenario. This investigation is focused on four bolts end-plate joints with 10 mm thickness which are proven experimentally to have more ductile behavior than other end-plates joints (6, 8 bolts), spite of their weakness to transfer the unexpected loads from the initial state to a residual state of the stable equilibrium, that leads to a failure of limited floor area to adjacent joints when tested experimentally (Saleh, 2014). FEM technique used in this research is an extension of the previous technique and is characterized by the use of a more sophisticated technique than the previous, discovered from the result of continuous research and the use of all the options available in the new version of commercial ABAQUS/CAD software. The elements are designed using multiple layers of specific elements of a brick arranged in such a way that the mesh nodes of the tiles should coincide with certain layers with the top of the shear inlay and in line with reinforcement, not as the former study that used thick shell elements to model the reinforced concrete slab with total negligence of reinforcing steel and bolts between the slap and the beam. This investigation is very complex because of highly nonlinear effects associated with the prediction of joint performance, such as structural imperfections, huge displacements and large rotations, inelastic properties of steel and concrete, bonding effects between steel and concrete, friction between end-plate and column flange and focus on the slip between concrete and structural steel, among others which is the most difficult modelling application. The paper addresses all these problems in addition to an evaluation of the joint with four bolts end-plate and provides recommendations for new computer simulation techniques (FEM). With these satisfactory results, this technique can be used to solve many problems and difficulties facing the steel and steel-concrete composite frames due to extraordinary events (explosions and hurricanes).

Keywords: Flush end-plate; Robustness; Catenary action; Implicit; Explicit

1 INTRODUCTION

In the context of continuous scientific research on the establishment of static bases for numerical analysis using modeling, this paper presents a numerical investigation of the joint behavior in steel and steel-concrete, composite frameworks using end-plate joints with 10 mm thickness and four bolts connection (flush end-plate). However, codes do not cover the beam-to-column joints design when subjected to sagging bending moment with the effect of axial forces. When considering the structural robustness and the design requirements for cases of unexpected events. The most important issue is the structural ability to internally redistribute new forces from the initial state to the adjacent joints and in this way might stop damage propagation. The generation of so-called alternative loading paths is predominantly carried out through the joint ductility, i.e. by ensuring that joints possess a sufficient rotational capacity. In the analysis, the robustness of a structure is most often evaluated by considering a notional removal of the key structural elements and by checking whether the local damage may be absorbed by the deteriorated structural system. This requires investigations into the behavior of joint reactions when exposed to a negative and positive bending moment at the same time, linking these moments and comparing them to axial forces. Current modeling techniques are extended to the joints under a complex procedure that appears in the case of gradual collapse and validation through comparisons with experimental test results conducted before. (Saleh, 2014). The aim of this contribution is to enhance knowledge in the numerical research (FEM) of structures with unexpected loads resulting from the collapse of a part of the building (column lose) and to create the basis for the global assessment of the structural fragility of the disproportionate collapse through static and nonlinear analysis based on energy.

2 FE MODEL DEVELOPMENT

According to previous studies, it was recognized that nonlinear static and dynamic analysis yields the most realistic results for the numerical simulation of considered commercial finite element programs, ABAQUS/CAD. And for the accurate results, in the present study, the analysis is used for two different types of studies namely static and quasi-static and the finite element joint model adopted is different from previous study and defined in the following: the reinforced concrete slab of composite joint is modeled using several layers of brick finite elements arranged in such a way that the slab mesh nodes of certain layers must coincide with the top of shear studs and in line with the reinforcement. The use of FEM in this study simply means that an inelastic constitutive model for concrete is included in the structural analysis process. Therefore, as cited in the previous section, it means the use of two models of concrete: concrete smeared cracking and concrete damage plasticity process. Lately, the concrete damaged plasticity model has been the choice of researchers for doing various researches mainly due to its incredible preciseness and the fact that it captures the reaction of concrete remarkably well. Hence, it has been included in this study and thoroughly described below. Nevertheless, note, the main two failure mechanisms according to this model are tensile cracking and compressive crushing. Cracking is believed to take place when the stresses reach a failure surface and repeatedly, it reaches the point of crushing and collapse. The reinforcement of concrete slab is modeled with truss elements and located in the same place as in the tested specimens. This allows the real stud length to be modeled matching end nodes of the connected shell elements of the beam flange and brick elements of reinforced concrete slab. The effect of profiled sheeting is included. Sheeting is modeled with use of thin shell elements. 3D beam elements type B31, which is based on Timoshenko beam theory allowing for transverse shear strain, are employed to model bolts. The concrete ribs are modeled with use of brick finite elements in the form of cubes and other shapes. A circular cross section is used to model the bolt section profile. Small sliding contact interface is applied between the end-plate and the column flange. This refined numerical model of common behavior shows better accuracy than previous models that were created in the past. It was therefore used to represent the physical framework that was tested to simulate global behavior for the purpose of this research. It should be noted that; this investigation is an extension of previous studies on the possibility of developing simulation in such a way that the results of the analysis correspond to a large proportion of the results obtained from the experimental tests results. It is also pointed out that this study is unprecedented, in addition to its accuracy; therefore, very accurate results are expected that can be used to solve many construction-related problems.

3 TEST SCENARIOS

The most important thing of this study is to obtain highly consistent results between numerical analysis and experimental test results. The specimen with those recorded for full-scale experiments, which were carried out before (SALEH B. Thesis) on actual structural bare steel framework, for which the details are shown in Figure 1. Tested specimens had the same arrangement of steel part of beam-to-column joints that were equipped with flush end-plates of thickness 10 mm and four M20 bolts, one that is a bare steel specimen and the second one the specimen with the steel-concrete composite framework of effective non uniform width of the reinforced concrete slab and composite beam-to-column joints. Two types of loading programmed were applied. In the first case, the pushdown displacement is applied on the middle joint gradually until collapse. The second specimen composite test is different where it needs two stages of application loads: the application of loads on the slab to simulate the service loads in the first stage using concrete blocks 1.20x1.20x1.00 meters were steeply placed in two layers on the composite slab prepared specifically for the experimental test. In this stage, all the beam-to-columns composite joints were subjected to hogging bending. And the second step is to apply the pushdown displacement on the middle of the specimen vertical direction by suspending it to the actuator until the collapse. The idea of the gravity loads and the displacement application is applied as in the use of ABAQUS/ CAD documentations, with some precision in selecting the specific points in the position of the displacement and the gravity loads that are applied to the length of the slab according to what is contained in experimental test.

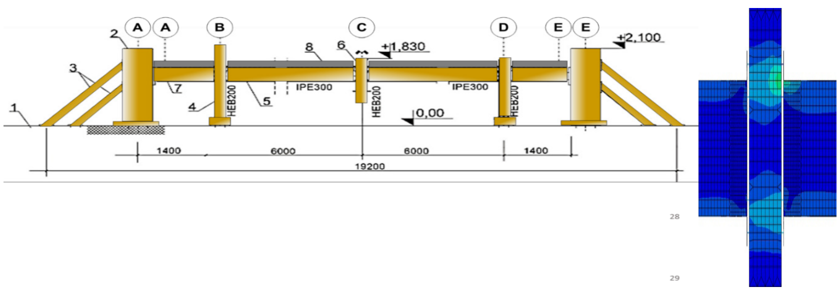

Figure 1: Frame specimen , 2 – stand resistance elements, 3- stand stabilizing elements, 4 –column, 5 –beam, 6 –middle column, 7 – pin joint between frame side and column, 8 - reinforced concrete slab

4 VALIDATION OF FE MODEL

4.1 Steel flush end-plate joints

The modified static general analysis is adopted for all subsequent numerical simulations. The results of the FE analysis are presented in terms of load-deflection curves corresponding to the internal column displacement and the applied load at the same point, and in terms of the characteristics corresponding to the joint rotation and the bending moment. The results of the FE simulation and experimental investigation of the analysis of steel flush end-plate joint are presented in Figure 2. The comparison of load-displacement curves shows compatible agreement for the initial elastic stage of the joint behavior as predicted numerically and measured experimentally, after that the FE curve separated gradually from the experimental curve when it reached the plastic phase and specifically when the force reached a value of 72.9 kN-385.2 mm, after which it began to rise until reaching the point of failure at 109.1 kN force and 621.9 mm displacement. The joint M-ϕ curves are presented in Figure 3. For the same test, the maximum value of applied load and the ultimate moment obtained from the tests and FE models are almost the same 41.93kNm – 93mm and 46 kNm - 90 mm in the range of moderate deformation then the FE results are slightly different from the results of experimental test until the point of collapse.

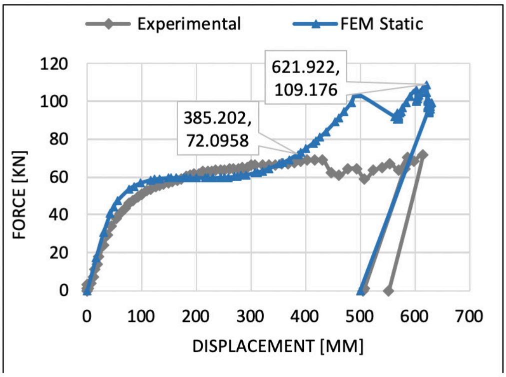

Figure 2: Force – displacement curves from FE and test for flush end-plate steel joints

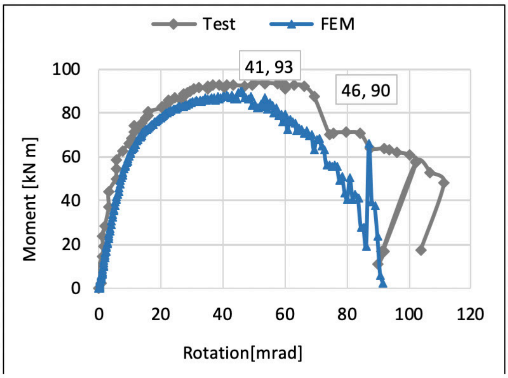

Figure 3: Moment-rotation curves from FE and test for flush end-plate steel joints

This is clearly observed in both curves of load-displacement and the moment-rotation. Because of the specimen symmetry, the sagging and hugging bending moment-rotation characteristics are almost the same. Figures 4 and 5 show a comparison between the deformed shapes of the flush end-plates obtained experimentally and numerically. The shape of flush end-plates is such that the most deformed part is located between the most distant bottom bolt rows of the inner joint at the adjacent of end-plate welded connection to the bottom flange of the beam. This is seen clearly in the joint that was subjected to sagging bending moment. For the external joint subjected to the hogging bending moment, the flush end-plate deformed shape is located above the most distant top bolt rows of the joint, at the adjacent of end-plate welded connection to the top flange of the beam.

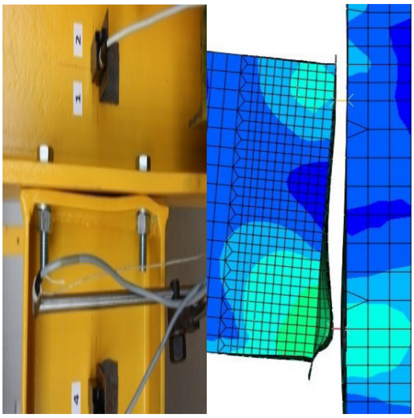

Figure 4: Deformed shape of steel joints with flush end-plates from test and FE–joint, sagging bending

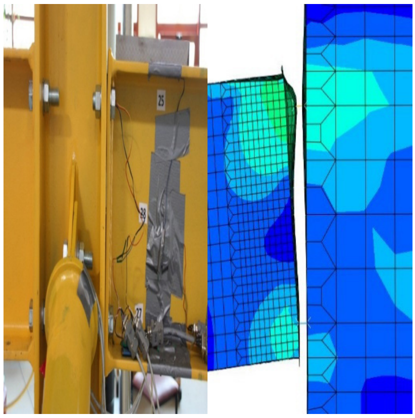

Figure 5: Deformed shape of steel joints with flush end-plates from test and FE joint, hogging moment

4.2 Composite flush end-plate joints of inner and outer columns

The test for the steel-composite specimen was conducted in two stages. In the first stage of numerical analysis, the gravity loading is applied including the application of uniform loads on a regular full slab (similar to the concrete cubes in the experimental test) in terms of weight, with the prevention of vertical movement by using the middle column as a support. This caused a vertical reaction in this column equal to an absolute value of 136.0 kN, then immediately the second phase began which was the application of displacement on a regular basis to the middle column. During this phase, the support was removed by lowering the middle column and leaving the specimen free in the vertical direction. This was done automatically until the end of the analysis when the specimen reached the point of failure. The data from the history output was collected, which is referred to displacement, rotation and stress data. This was then analyzed and kept in order to calculate the load-disablement, and the moment-rotation relationship of the data related to the joints that were later used to assess the global behavior of the framework and for the evaluation of local joints in terms of their degree of robustness. In the initial curve phase, there is a large consensus between FE and the test, both in the relationship between the loads-displacement curves as shown in Figure 6 and in the relationship between the moments-rotation curves as shown in Figure 7. At this stage, all the joints were subjected to positive bending moment. Compatible agreement between the experimental and numerical results for the first stage was reached, then compatibility continued to be shown between the curves (FEM and the test) at the elastic stage, and perhaps a little later, in the plastic phase, the FE curve starts to begin a gradual downward separation from the experimental curve and continues in this manner until the end of the analysis. In all the stages, agreement is observed. The only discrepancy is that these ultimate values are reached at different displacements of the load-displacement curve. There was no sign of failure in the first stage. It can be seen from the results using steel and composite joints that there are similar deformation mechanisms of the end-plates in the first stage of loading, and there is a good correlation between experimental and numerical characteristics. In the second stage, it was observed that there is a faster deterioration of rigidity with successive displacement application in the numerical simulation than in the test. This can be attributed to the fact that the effect of end-plate material fracture could not be represented in numerical simulations as precisely as in the experimental test. This also caused a discrepancy in the way load-displacement curves approach the zero level of loading. Moreover, in some of the investigations into the mesh shape configuration and its impact on results in the search, it turns out that fine mesh may also create these differences when FE analysis is carried out with the use of ABAQUS/CAD software.

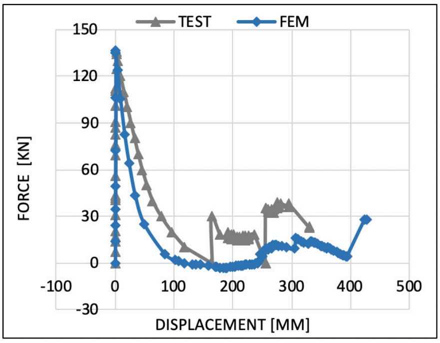

Figure 6: Force [kN]–displacement [mm] curves from FE and test for flush end-plate specimens

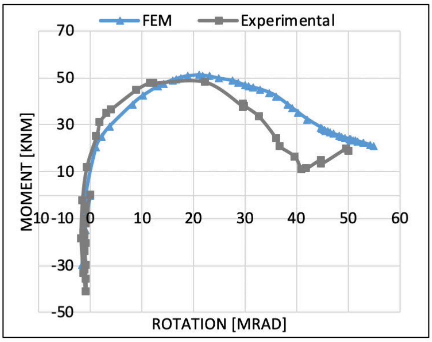

Figure 7: Moment-rotation curves from FE and test for flush end-plate composite specimens

Figure 8 shows a similarity between the end-plate deformation shape in the experimental test and that of the FEM simulations in the outer joint when this joint is subjected to hogging bending moment. Figure 9 shows the comparison of experimental and the FE results for the end-plate deformation of the inner joint when the joint is subjected to sagging bending moment. It can be clearly seen that there is considerable similarity between the end-plate deformations in the shapes that have been obtained experimentally and numerically.

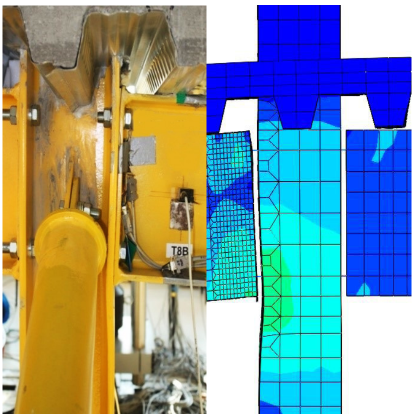

Figure 8: Deformed shape of experimental and FEM of outer composite joint

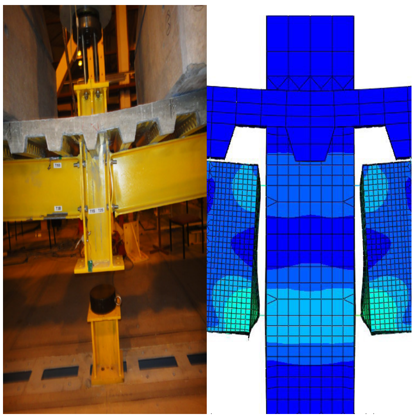

Figure 9: Deformed shape of experimental and FEM of inner composite joint

5 CONCLUSION

In the process of computer simulation (FEM), investigating the joint moment-rotation characteristic, the results of tests conducted elsewhere on joints were considered. It has been obvious that for the symmetric arrangement of bolts, like for flush end-plates, the moment-rotation characteristics of steel joints are the same in cases of both hogging and sagging bending. The results obtained for composite joints are more complex. The moment-rotation characteristics are different for hogging and sagging bending moments for flush end-plate joint with symmetric bolt arrangement on both sides according to the results obtained numerically.

The most important property for the robustness of a structure is its resistance to progressive collapse scenarios. For robust structural systems, both tests and FE modeling simulations should prove that the structure could sustain the applied load in case of a deteriorated static scheme after the removal of its critical structural element as a result of the expected local damage simulation.

Robust joints have to have a sufficient balance between the rotation capacity and strength. FEM simulation and tests on steel and steel-composite frameworks have shown that end-plate joints with four bolts are not robust since their low strength, despite good ductility proven in tests and FE modeling simulation on joints does not allow for the sufficient development of catenary action in the beam when tested in the framework.

Tests relating to the global behavior of frameworks conducted in previous studies and in this study should be repeated with respect to simulation of fixed column removal taking into account the dynamic effects associated with column removal. This would need more sophisticated measurement devices, more of a high-speed visual recording nature that of the mechanical nature, including the fast speed video camera and laser technology.

There is a need for continued research on Finite Element Modeling (FEM) of the local and global behavior of frameworks, especially with reference to dynamic situations. Initial investigations presented in this study on the dynamic simulations of static behavior of joints have shown that there are a number of difficulties in terms of numerical stability, this causes the non-continuity of the analysis and early termination of numerical analysis as a result of severe nonlinearities appearing in the finite element formulation of physical phenomena, especially related to modeling of the inelastic behavior and cracking phase of concrete.

REFERENCES

Eurocode 3: EN 1993-1-8: Design of steel structures (2005). Part 1-8: Design of joints. CEN, Brussels.

Gizejowski, M. A., Kwasniewski, L., Saleh, B. & Balcerzak, M. (2012). Numerical Study of Joint Behavior for Robustness Assessment, Applied Mechanics and Materials, Yantai, Peoples R CHINA, vol. 166-169, pp. 3114-3117.

Gizejowski, M., Saleh, B., Kozlowski, A., Pisarek, Z. & Sleczka, L. (2012). Experimental investigations of frame behavior subjected to exceptional actions. *85th Annual Conference on Scientific Problems of Civil Engineering*, Krynica- Rzeszow, Poland.

Hibbit, Karlsson, and Sorenson Inc (2006). ABAQUS/ CAD, User's Manual, Version 6.10, Providence, RI.

ABAQUS, User's Manual, version 6.10, Hibbit, Karlsson and Sorenson Inc., Providence, RI, (2006).

Saleh, B. (2015). Moment-rotation characteristic of joints of steel-concrete composite frame under exceptional events. *IABSE, Nara*, Japan, pp. 504-505.

Saleh, B. (2014). *Modelling of beam-to-column joint of steel-concrete composite frames subjected to standard and extreme load combinations* (Warsaw University of Technology, Warsaw, Poland).

Saleh, B. (2016). Experimental investigation for the steel-concrete composite joints subjected to in exceptional load. *6th Conference of Building Materials and Construction*, Ghryan, Libya.

Cite this article as: Saleh B., "Numerical Study of Four Bolts End-Plate Joint Behaviour for Robustness Assessment", *International Conference on Civil Infrastructure and Construction (CIC 2020)*, Doha, Qatar, 2-5 February 2020, DOI: https://doi.org/10.29117/cic.2020.0132