Acta Technica Napocensis: Civil Engineering & Architecture Vol. 55, No.1 (2012)
***Journal homepage:* http://constructii.utcluj.ro/ActaCivilEng**

Numerical models of welded and bolted beam to column connections

Roxana Balc*[1]

[1,] *Technical University of Cluj-Napoca, Faculty of Civil Engineering. 15 C Daicoviciu Str., 400020, Cluj-Napoca, Romania*

Received 11 March 2012; Accepted 21 May 2012

Abstract

The significant progress achieved in the electronics technology and computer programming, made possible the application of Finite Element Method (FEM) for the analysis of steel frame joints. The goal of the FEA on steel joints was to obtain "bending moment - relative rotation" curves as close as possible to the ones experimentally obtained, in a reasonable time period and with reduced costs.

This paper presents the analysis with finite elements of two joint typologies: welded and bolted with end plate, using the ABAQUS programm. The results obtained after the numerical simulation were compared with the experimental data extracted by scientifical literature.

Rezumat

Progresele spectaculoase realizate în domeniul sistemelor electronice de calcul a făcut posibilă utilizarea metodei elementelor finite (MEF) la cercetarea şi proiectarea cadrelor metalice. În domeniul calculului structurilor metalice în cadre,Aplicarea analizei cu MEF nodurilor cadrelor metalice oferă posibilitatea simulării comportării reale a acestora, cu costuri mici, în comparaţie cu testele experimentale şi într-un timp relativ scurt.

Lucrarea de faţă prezintă analiza cu elemente finite a două tipologii de îmbinări: cu sudură şi cu placă de capăt extinsă şi şuruburi pretensionate, utilizând programul de calcul ABAQUS. Rezultatele obţinute în urma modelării au fost comparate cu date experimentale extrase din literatura de specialitate.

Keywords: Finite element analysis, welded connections, endplate connection

1. Introduction

The behavior of steel connections continues to be an issue of interest in the area of steel structures.
The non linear behavior of steel frames joints is experimentally proved even in the case of welded connections. The study presents in detail two illustrative examples created by using FEM for a welded connection and a connection with extended end plate. For both joints, a nonlinear static analysis was applied using ABAQUS commercial code, ver. 6.11 [1-4] and the obtained results were compared with experimental data from scientific literature [7].
The numerical models were created by using 3D finite elements of solid type C3D4R, considering the contact between components and the bolts pre-tensioning.
The analysis focused on the following aspects:

- The state of stress at each loading step

* Corresponding author: Tel./ Fax.:+40 264 594967
E-mail address: balcroxana@yahoo.com

- The faillure mode in comparison with the one experimentally observed
- The comparison of the bending moment–relative rotation curves with the ones experimentally obtained [7]

2. Static scheme and connection description

Two typologies of beam-to-column joints have been modeled using ABAQUS commercial code, ver. 6.11 and compared with corresponding experimental tests. The joints are double sided and the static scheme was perfectly symetrical (Fig.1).

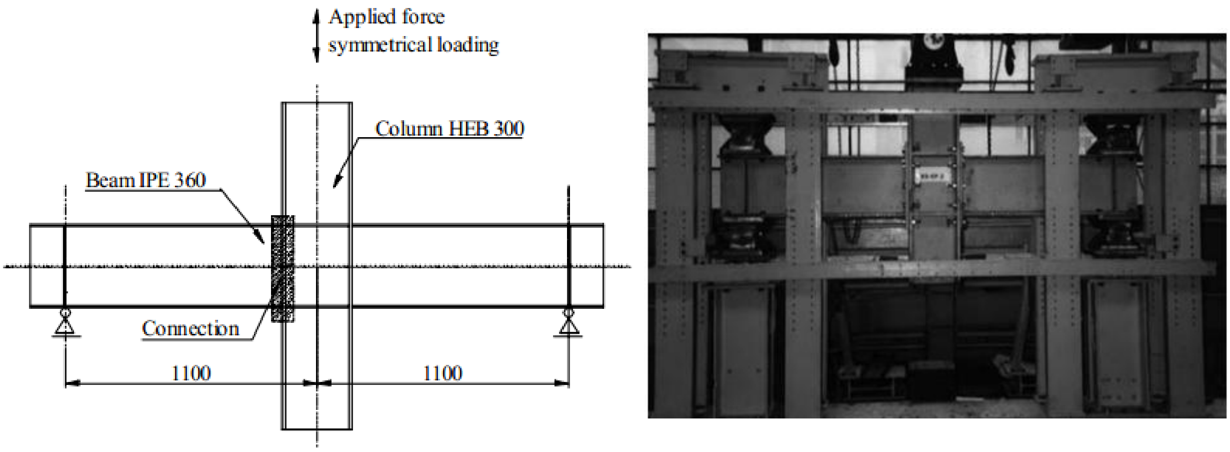
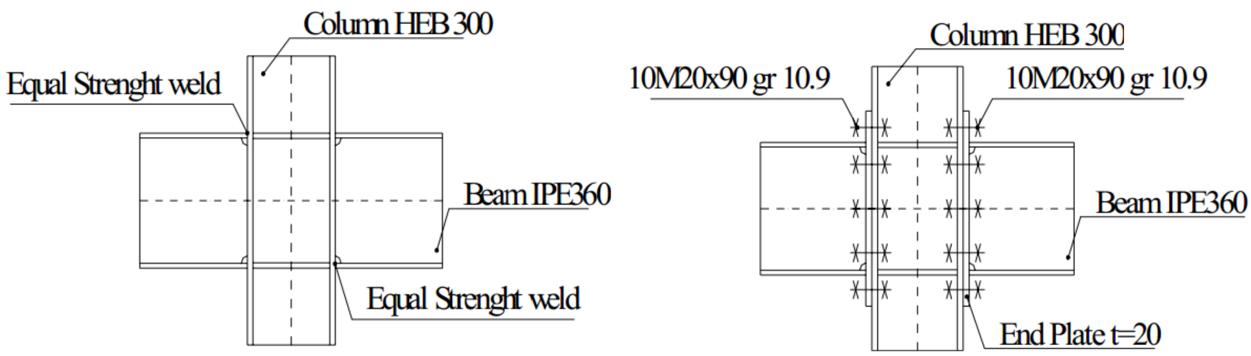


Figure 1. Static scheme

The first type was a welded connection (Fig.2.a.) and the second type was a bolted end plate connection (Fig.2.b.). Beam and column are I, respectively H profiles. High strength bolts M20/ 10.9 are used, for the extended end plate solution.



a. welded connection b. extended end plate connection

Figure 2. Connection configurations [7]

The mechanical and the measured geometrical characteristics of elements were taken from the experimental tests [7].
The structure is simply supported on the beam ends. A vertical cyclic alternated force was applied on the top of the column.

3. Discretisation of the models

The density of the mesh was made higher in areas where the stress concentrations were expected, respectively in the joint zone and in the supports regions, then in the remaining areas (Fig. 3.).
In order to respect the actual geometry of the profiles, all the components of the joints were modeled using the element C3D4 which refers to a 4 node linear tetrahedron [1].

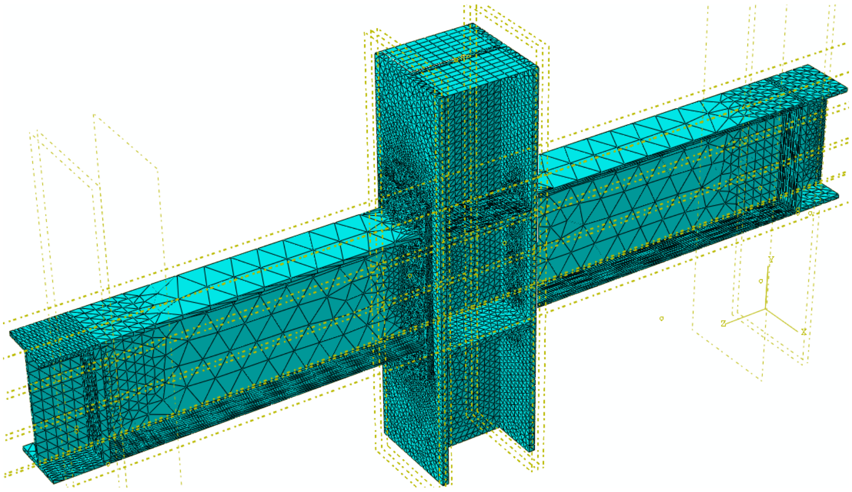


Figure 3. Mesh discretisation of the end plate connection model

The column web panel is stiffened by two plates aligned with the beams flanges. The stiffeners are welded to the beam in the support zones. Welds are modeled using tie constraint option.
In order to simplify the bolted joint model, hexagonal bolt head and nuts are idealized as circular and washers are not modeled. Slip between end plate and bolt head was neglected. Bolt holes are assumed to be 1mm greater then the bolt size.

4. Material model

For the welded connection, the yield and ultimate tensile were 313,8MPa and 449,8MPa respectively and the ultimate strain was 0,23.
Two material models was used in the bolted joint model, one for the joint elements, and the other for the bolts.
The yield and ultimate tensile stresses considered for the beam, column and end plate were 248.3MPa and 416 MPa respectively. The ultimate strain was 0,23.
High strength friction grip bolts (10.9) are considered in the bolted joint. The yield and ultimate stress for the bolts were considered as 900 MPa and 1000 MPa respectively and the ultimate bolt strain considered was 0,09.

5. Contact modeling

In the case of the joint with end plate and prestressed high strength bolts the transfer of the forces is realized through friction due to the clamping action between the connection elements.
Small sliding surface to surface contact was applied to all the surfaces which have small relative sliding [1-4, 9]. Between the end plate and the column flange was considered a tangential frictional contact ($\mu = 0{,}3$), using penalty stiffness formulation and a hard normal contact using augmented Lagrange formulation. The bolt head / nut were tied constrained to the end plate / column flange. Between the bolt hole and the bolt shank the tangential contact was considered as frictionless. Hard constraint was considered between the rigid plate and the top column section.

6. Load application and boundary conditions

The load was applied, for the extended end plate connection model, in two steps. In the first step a pretension force, about 0,7 time of the yield bolt stress, was applied to all the bolts, at the center section of the bolts, using the pretension option of ABAQUS [1-4, 9]. In the second step, a concentrated vertical force was applied on the top of the column. A rigid plate tie constrained on the top of the column effected the transmission of the concentrated force to the joint zone. In the two numerical models, the vertical force were applied statically.

7. Results interpretation

7.1.Welded connection

The action of concentrated force is transmitted by the rigid plate to the node zone (fig.4).
Joint faillure was produced by local buckling of the upper beam flange associated with beam web and bottom flanges plastification (Fig.5.a). A similar failure mode has been identified in the experimetal test (Fig.5.b).

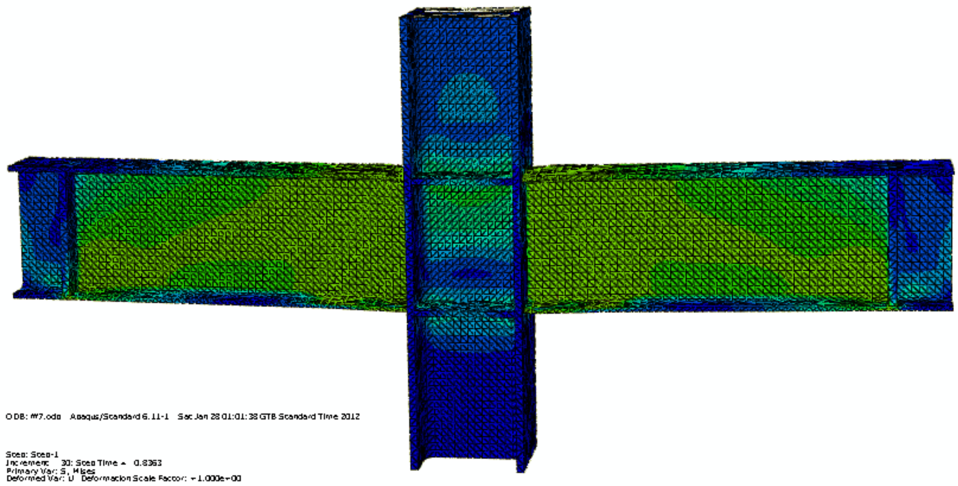
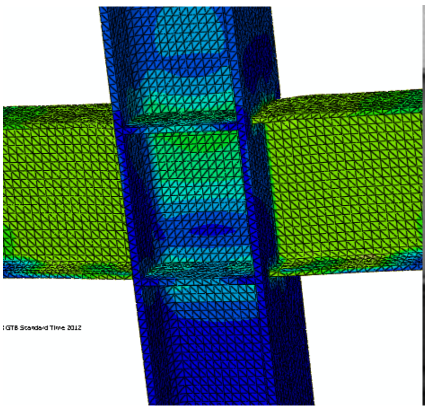


Figure 4. Deformated shape of welded joint



a. Numerical model　　　　b. Experimental test

Figure 5. Failure of welded connection

The moment-rotation curve obtained after the numerical simulation was presented in the figure 6, compared with the envelope moment-rotation experimental curve.

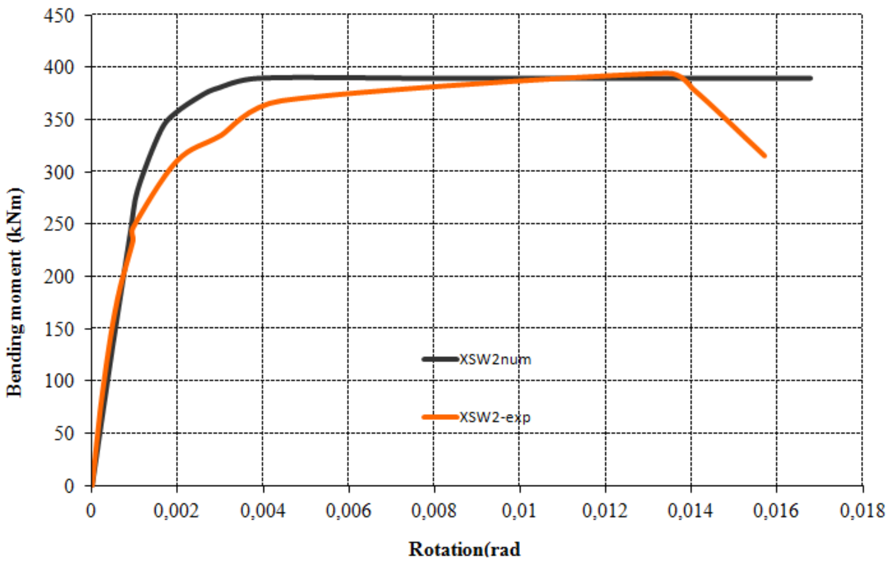


Figure 6. Moment-Rotation curve of the welded joint

The initial stiffnes of the models are similar as it can be observed from figure 6. A difference of 0.012% was obtained for the maximum bending moment.

7.2.Extended end plate connection

The highest stress areas of the structure were the bolts of the last row, submitted to tension, the beam flanges and web in compression, the column web in compression, the areas around the supports.

The failure of the joins was caused by the ruin of the bolts in the last row submitted to tension.

The same failure mode was obtained after its experimental testing.

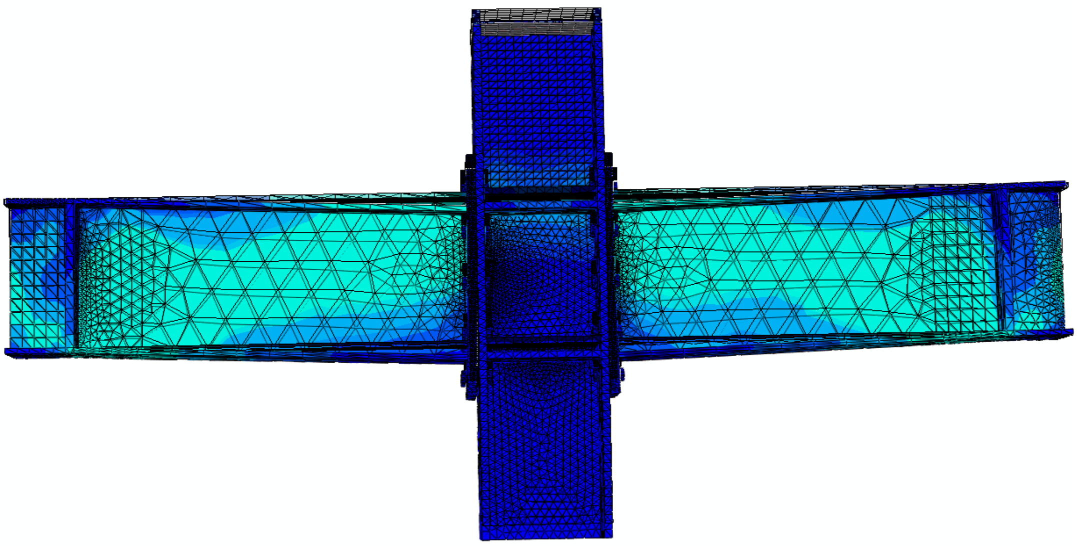


Figure 7. Deformated shape of bolted joint

In the figure 8., is presented the distribution of the stresses in the connection elements and in the most strained bolt.

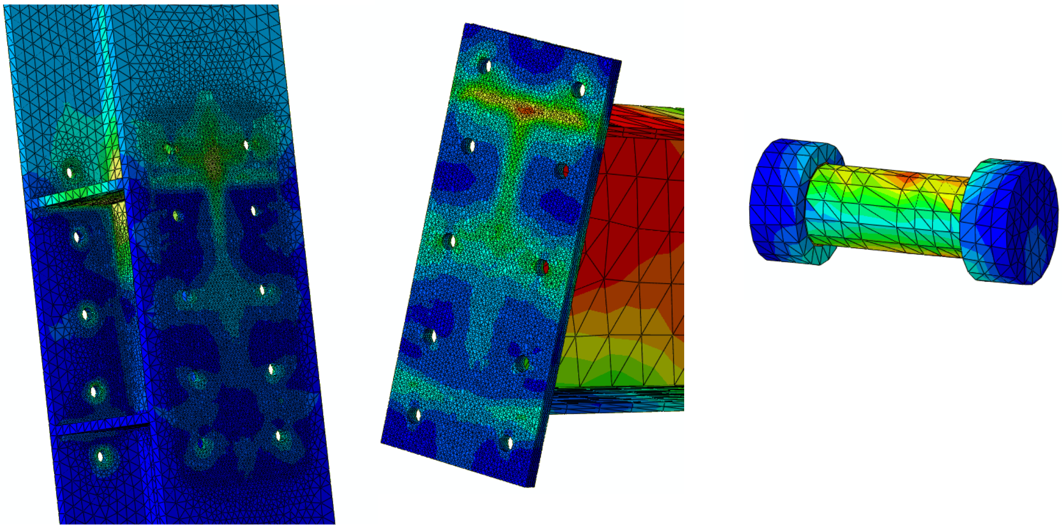


Figure 8. Stress distribution in the joint elements and in the last row bolt

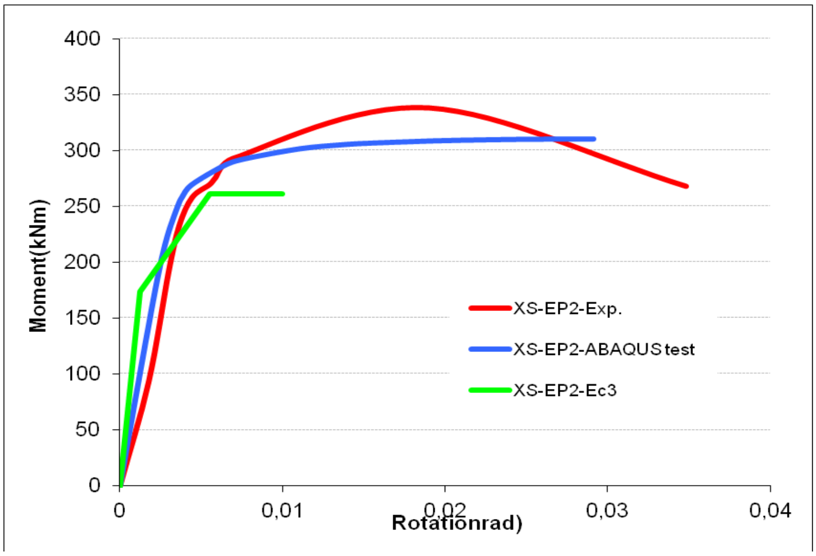


Figure 9. Moment-Rotation curve of the extended end plate connection

In the numerical model the evolution of the curve is linear till 570kN (concentrated load / force), after which the flexibility of the joint increases, at the end the value of the bending moment reaches 317kNm while the last rotation is 0.029rad.
The moment-rotation curve of the numerical model underestimated with 2,88% the yield moment and overestimated with 14,75% the initial stiffness.
The values of maximum moment for the two models presents an approach to about ten percent.
The difference between the initial stiffness values obtained for numerical and experimental models can come from various sources, most often as a direct consequence of the simplifications introduced in the numerical model. Other factors which explain the differences between experimental and numerical models may refer to:

- Analysis type (statical in the numerical model and cyclic alternated in the experimental tests).
- Constitutive material laws used in the FE symulation.
- Differences between prestress force values introduced in the models.

8. Conclusions

The simple geometry of the welded joints and the clear modality of efforts transmition facilitate the numerical modeling of this connections.
Numerical model developed for the welded joint was used to validate the statical scheme for the connections with extended end plate.
Finite element method analysis of joints with end plate and high strength prestressed bolts with the, ABAQUS softwere, has provided results close to those obtained through experimental tests.
A special attention was accorded to :

-contact modeling of interacting surffaces ,
-prestress bolts modeling,
-material model.

9. References

[1] ABAQUS Cae, User's Manual

[2] ABAQUS Cae, Theory Manual

[3] ABAQUS Cae, Analysis Manual

[4] ABAQUS Cae, Interactive edition

[5] Bursi,O.S., and Jaspart J. P., "Computer Modeling Of Extended End-Plate Bolted Connection" Computers and Structures, 52(5), pp. 879-893, 1994

[6] Bursi, O.S., and Jaspart, J. P., "Basic Issues in the Finite Element Simulation of Extended End-Plate Connections" Computers and Structures, 69, 361-382,1998.

[7] Dubina D. Grecea D., Ciutina A., Stratan A. : Moment Resistant Connections of Steel Frames in Seismic Areas. Design and Reliability (Editor F.M. Mazzolani) , E&FN SPON, London , 2000, ISBN 0-415-23577-4 Chapter 3. Cyclic behaviour of beam-to column bare steel connections 3.2 Influence of connections typology and loading asymmetry, p 217-244 ;

[8] Gira˜o Coelho AM, Simoes da Silva L. „Numerical evaluation of the ductility of a bolted T-stub connection." In: Chan SL, Teng JG, Chung KF, editors. Advances in steel structures. Proceedings ofthe Third International Conference (ICASS'02) Hong Kong. 2002, p. 277–84.

[9] Jayachandran, S., Marimuthu V, Prabha P, Seetharaman S, Pandian N. Investigations on the behaviour of semi-rigid endplate connections. Advanced Steel Construction, Vol.5, pp. 432-451, 2009

[10] Maggi,Y.I., Goncalves,R.M.,Leon,R.T., Ribeiro,L.F.L., „Parametric analysis of steel bolted end plate connections using finte element modeling", Journal of Constructional Steel Research, 61,pp. 689-708, 2005

[11] Shi G, Shi YJ, Wang YQ, Li S, Chen H. „Finite element analysis and tests on bolted end-plate connections in steel portal frames." Advances in Structural Engineering 2004;7(3):245–56.

[12] Shi YJ, Shi G, Wang YQ., Bradford M. A. (2008). „Numerical simulation of steel pretensioned bolted end-plate connections of different types and details", Engineering Structures.