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

Finite element analysis of beam to column end plate bolted connection

Roxana Balc[*1] , Alexandru Chira[1], Nicolae Chira[1]

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

Received 24 June 2012; Accepted 22 July 2012

Abstract

The improvement of the electronic systems of computation made the use of the finite element method (FEM) possible in the current research and design activity. In the field of the design steel frames, the finite element analysis (FEA) of the joints offers the possibility to simulate their real behavior at low costs and in a relatively short period of time compared with the experimental tests. This paper presents the analysis with finite elements of a steel joint with end plate and prestressed bolts, using the ABAQUS finite element software code. The results obtained after the numerical simulation were compared with the experimental data in order to validate the model.

Rezumat

Perfecţionarea sistemelor electronice de calcul a făcut posibilă utilizarea metodei elementelor finite (MEF) în activitatea curentă de cercetare şi proiectare. În domeniul calculului structurilor metalice în cadre, analiza cu MEF a nodurilor 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 unui nod metalic cu placă de capăt şi şuruburi pretensionate, utilizând programul de calcul ABAQUS. Rezultatele obţinute în urma modelării au fost comparate cu date experimentale, obţinând o apropiere satisfăcătoare. Validarea modelului permite continuarea studiului în scopul determinării unei metode de cacul simplificate al acestui tip de prindere.

Keywords:, Finite element analysis, steel joint,prestressed bolts, endplate connection

1. Introduction

The behavior of steel bolted connections continues to be an issue of interest in the area of steel structures. The design of semi rigid steel frames depends on the joint feature, wich is the moment-rotation model. This work implies a great number of experimental tests, that could lead to a great cost of time and money. Starting from this point of view ,the finite element method represents an attractive alternative for predictioning the appropriate moment-rotation model.

In this study, a beam to column end plate bolted connection was analyzed using ABAQUS, a general purpose FE software. The results obtained were compared, for validation, with experimental results extracted from scientific literature [1].

[*] Corresponding author: Tel./ Fax.:
E-mail address: balcroxana@yahoo.com

2. Geometry of the model

The joint under observation consists of a pair of beams with end plates connected to a central column with six pairs of prestressed bolts. The geometry of the joint is presented in Figure1. Beam and column are I, respectively H profiles. High strength bolts M18/ 10.9 are used.

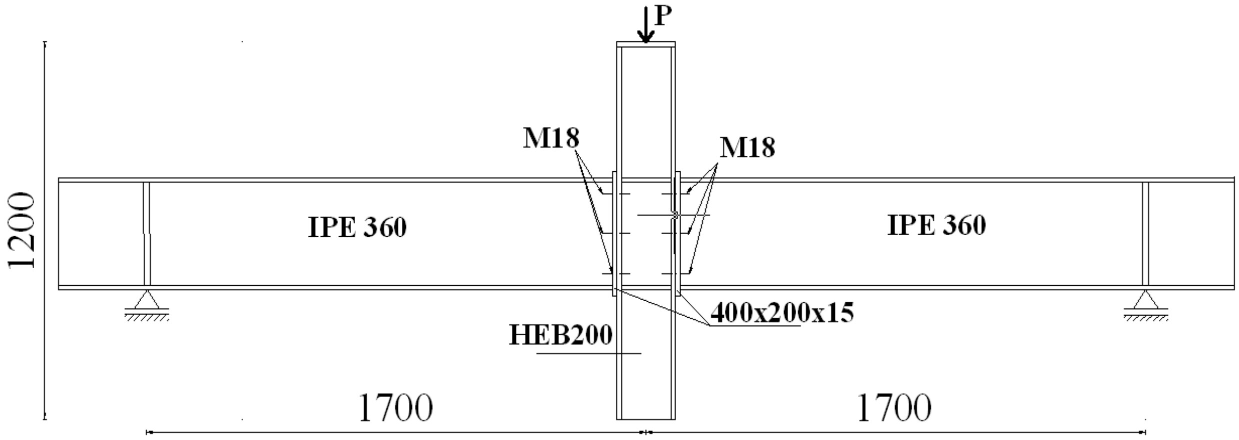

Figure 1. Geometry of the model

The structure is simply supported on the beam ends. A vertical force was applied on the top of the column.

3. Discretisation of the model

The mesh refinement must satisfy the need for a fine mesh to give an accurate stress distribution in a reasonable analysis time. The optimal solution is to use a finer mesh in areas of high stress, respectively in the contact zones of the joint and in the supports regions, then in the remaining areas (Fig. 2.).

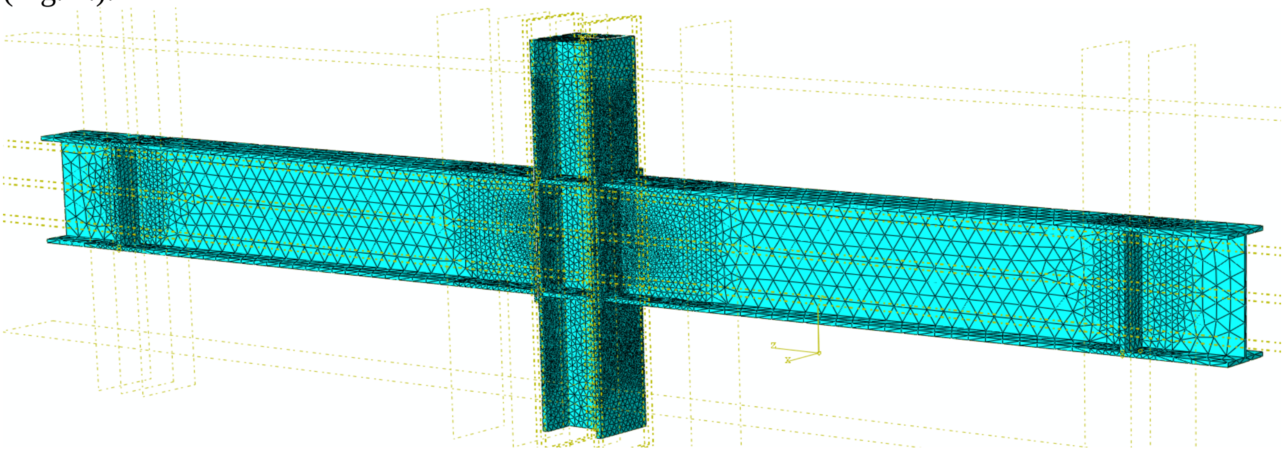

Figure 2. Mesh discretisation of the model

In order to simplify the model, hexagonal bolt head and nuts are idealized as circular and washers are not modeled. Bolts holes are assumed to be equal with bolt size.
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. Because the welding is in general not the cause of failure, the beam, the end plate and the support beam stiffener were considered to be one body, as well as the column and his stiffeners.

In order to respect the actual geometry of the profiles, all the components of the joint were modeled using the element C3D4 which refers to a 4 node linear tetrahedron.

4. Material model

Two material models was used in this 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 265 MPa and 370 MPa respectively. The ultimate strain was 0,21. High strength friction grip bolts (10.9) are considered in the present study. 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

The pretension of high strength bolts and the friction between the connection components are the essential parameters which define the joint behavior. The pretension force applied to the bolt creates a hard contact between the connected elements. 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 [2,4,5]. The tangential contact between the end plate and the column flange was considered as frictional contact ($\mu = 0{,}3$), using penalty stiffness formulation. Hard contact using augmented Lagrange formulation was considered for the normal contact between the same components. The bolt head / nut were tied constrained to the end plate / column flange. The tangential contact between the bolt hole and the bolt shank 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 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 [2,3]. 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.
The beams are simply supported at the ends. The roller supports were introduced at the beam ends. Supplementary, boundary conditions were applied on the top of the column in order to permit only vertical displacements for simulating the experiment situation.

7. Results interpretation

The action of concentrated force is transmitted by the rigid plate to the node zone (fig.3). 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 summitted to tension.

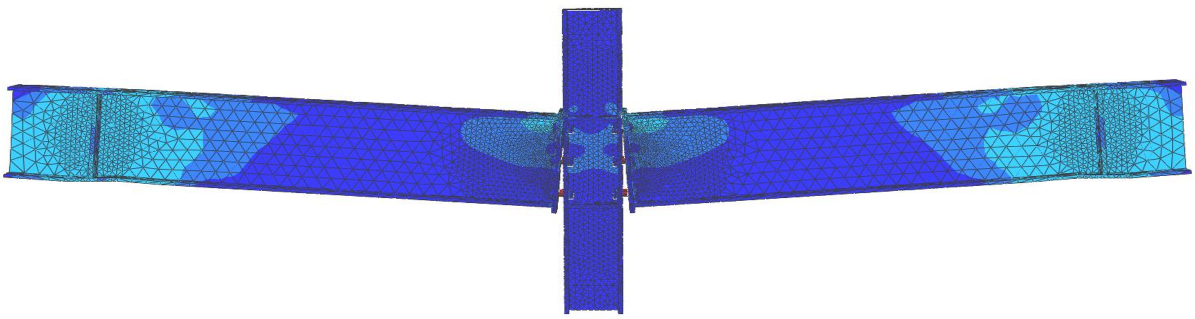

Figure 3. Deformated shape of the structure

The same failure mode was obtained after its experimental testing. In the figure 4., is presented the distribution of the stresses in the most strained bolt.

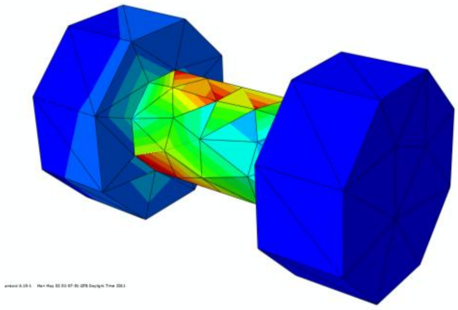

Figure 4 Stress distribution in the last row bolt

The friction between the connected elements, end plate and column flange, caused by the bolts prestress, made the force transmission. In the figure 5 is presented the global stress distribution after the first loading step.

All the head/nut bolts of the joint were sumitted to the compression caused by the pretension effect. The remaining areas of the joint rest undefformed.

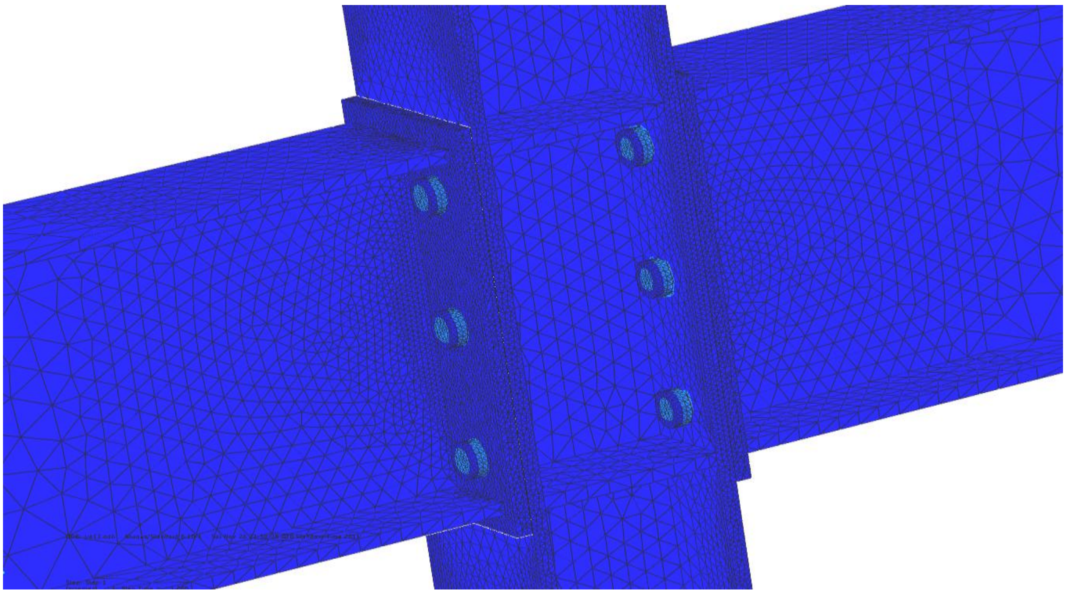

Figure 5 Stress distribution after the first loading step

Stress distributions in the connected elements after the prestress step was presented in the figure 6.

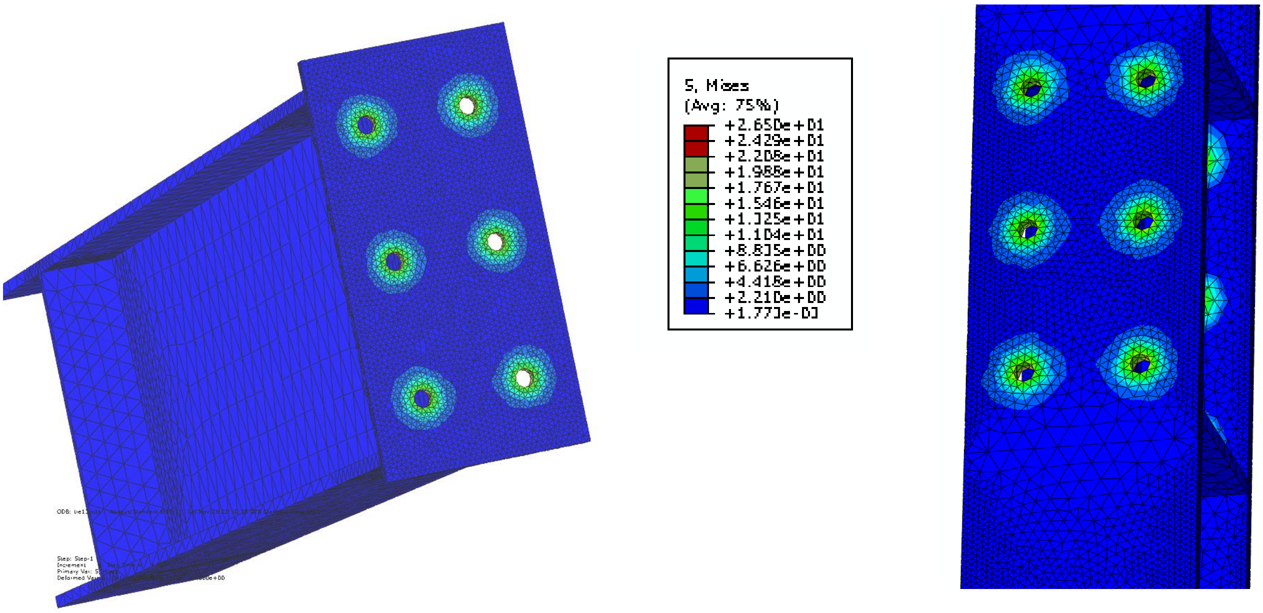

Figure 6 Stress distribution in the connected elements after the first load step

The moment-rotation curve obtained after the numerical simulation was presented in the figure 7. The curve is linear up to 100 kNm, and the ductility of the connection starts beyond that load.

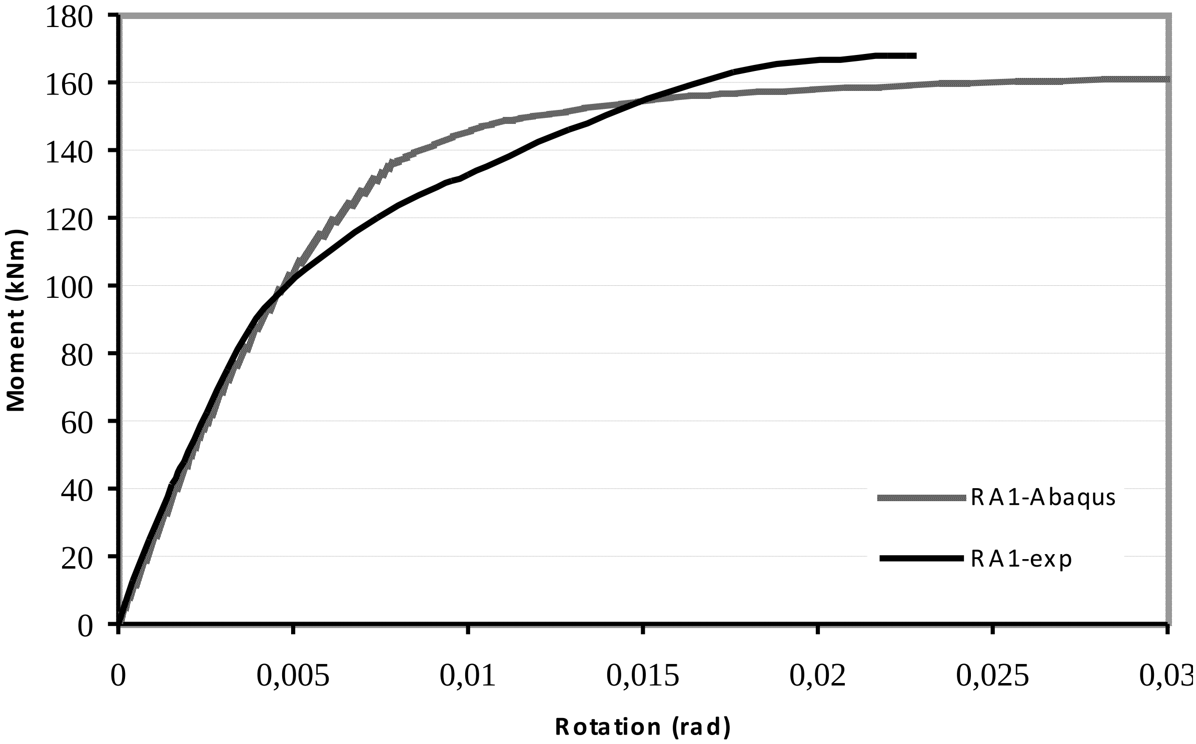

Figure 7. Moment-Rotation curve of the joint

The initial stiffness correlate well with the test results. The differences between the moment-rotation curves were caused by the material model, which is not perfectly known.

The differences between the stiffnes and the failure moments in the inelastic field could be caused by the sliding of the bolts in the holes, a fact that was included in the finite element model because the diameter of the bolt was considered equal to the hole diameter. Also the lack of information regarding the material inelastic law and the exact pretensioning forces used for the experiment lead to all the differences seen in figure 7. The displacements in the failure phase are bigger for the fem model.

8. Conclusions

FEA of the joints offers the possibility to simulate their actual behavior at low costs and in a relatively short period of time compared with the experimental tests. ABAQUS program presents a facile and attractive way to study the behavior of endplate connections. The model, validated by experimental tests can be used for other investigations.

The numerical analysis of the joint shows a good behaviour of the pretensioned bolts in accordance with the theoretical and experimental results.

The moment-rotation feature of the modeled joint as it is shown in figure 7 it's similar with the experiment for the values of the initial stiffnes,yielding moment, a slight underestimation of the failure moment and a slight overestimation for the failure displacement.

9. References

[1] Aribert JM, Lachal A, Dinga O. Modelisation du comportement d'assemblage metalliques semi-rigides de type poutre-poteau boulonnée par platine d'extrémité. *Construction Metallique* nr.1, pp. 26-41 1999

[2] 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

[3] ABAQUS user Manual, Version 6.6.

[4] Maggi Y. I., Gonclaves R M., Leon R.T. and Rebeiro L.F.L., "Parametric Analysis of Steel Bolted End-Plate Connections Using Finite Element Modeling", Journal of Constructional Steel Research, , Vol. 61. pp. 689-708 2005.

[5] Abolmaali Ali. John,. Matthys H, Mohammed Farooqi and Yeol Choi, "Development of Moment-Rotation Model Equations for Flush End Plate Connections", Journal of Constructional Steel Research, , Vol. 61. Pp. 1595-1612 2005

[6] Gang Shi, Yongjiu Shi,Yuanqing Wang, Bradford Mark A,"Numerical Simulation of Pretensioned bolted End-Plate Connections of different types and details", Engineering Structures, 30, pp.2677-2686 2008.