Nonlinear FEM analysis of steel beam-to-column connections with extended end plate

M Stașcov¹, V M Venghiac¹ and S G Luca¹
¹The Faculty of Civil Engineering and Building Services, “Gheorghe Asachi” Technical University of Iași, Romania.

E-mail: stascov.mihail@gmail.com

Abstract. Even if it has been shown that steel beam-to-column connections with extended end plate has a semi-continuous behaviour, many structural analyses consider these joints either fixed or pinned. For advanced structural analysis of connections and frames it is necessary to know the behaviour of the joint. The characteristics of the joint can be identified on the moment-rotation curve. This paper presents a comparative study of the moment-rotation curve for a standardized node calculated according Eurocode 3, and obtained by FEM analysis using IDEASTATICA and ANSYS finite element analysis program. The analyses were performed for the models with non pre-tensioned bolts. Also, an additional study was performed in ANSYS to identify the influence of the thickness of the extended end plate.

1. Introduction
The distribution of forces acting on the structure are influenced by the strength and rigidity of the entire structure. The characteristics of the structure are influenced by the characteristics of the joints. Thus, the characteristics of the joints, such as strength, stiffness and plastic deformability (rotation) directly influences the behaviour of the entire structure, figure 1.

![Figure 1. The interdependent relation bending moment-rotation.](image)

The failure mode of steel structures subjected to cyclic actions such as earthquakes, creates the most problems. Conventional Pre-Northridge joints, which involve direct welding of the beam to the column flange, have proved to be ineffective due to the brittle failure before the plastic hinge can be formed. In that case, a small amount of energy is absorbed, which leads to a reduced plastic deformation capacity [1].
The traditional approach in designing of steel frames structures implies that joints were either pinned of perfectly rigid. Research performed after the Northridge (1994), and Kobe (1995) earthquakes showed that joints considered perfectly rigid had a semi-rigid behaviour, and some joints considered pinned had capacity to transfer the bending moment, thus was introduced the notion of semi-rigid joint, shown in figure 2c.

![Figure 2](image-url)

At the same time, the design rules for steel structures design, such as EC3 [2, 3, 4] require respecting the “strong column – weak beam” principle in designing of steel frame structures (figure 3).

![Figure 3](image-url)

In order to satisfy this requirement joints were proposed with reduced section beams, reduced beam flanges sections RBS [5, 6], and reduced web sections RWS [7]. Additional stiffening of the end of the beam or additional stiffeners on the column flanges were also proposed.

Design of the joints implies the assessment of the capability of each element (component method) [2]. The evaluation of all components is quite complicated due to the geometrical and material nonlinearity and due to the effects of local imperfections and residual forces [8, 9]. Despite these difficulties in the past two decades, a large number of advanced FE studies have been performed for the study of end plate joints with a large variety of geometries, to obtain values for resistance and ductility [8, 10, 11, 12, 13, 14, 15]. Generally, these studies have attempted to provide the response to the behavior of the joints such as interdependent relation bending moment-rotation curves, or hysteretic curves.
Due to the possibility of varying the number of the bolts, their diameter, the dimensions and thickness of the end plate, this type of joint show a wide variation of the characteristics. The purpose of this study is to develop the comparison of the result for a type joint, obtained according to the European designing rules [2], obtained from numerical modeling with FEM program specialized in the design and check of the steel joints IDEA StatiCa, and FEM multiphysics program Ansys Workbench, at the same time, to develop a FEM model for end plate joints, and obtaining of the interdependent relation bending moment – rotation curves, to study the influence of the thickness of the end plate.

2. Analysed models
By directing the formation of the plastic hinge in the beam end or the end plate, the ductile behaviour of the joint can be ensured [15]. Three configurations of the end plate joints were analyzed. The considered nodes have common elements: HEB300 column, IPE360 beam, and M22, 10.9 grade high strength bolts, with normal clamping. The steel grade in the beam, column, end plate and stiffeners is S235. Figure 4 shows the analyzed nodes.

3. EC3 analysis
The design standard for steel structures [2] (Eurocod 3. SR EN 1993-1, 2006) provide for element verification by the component method. Only the dimensions and characteristics of the node elements are required. Typically, the design of the joints according to EC3 is very labour intensive, manual computation requires a lot of time [16]. Following the calculation based on the component method according to (Eurocod 3. SR EN 1993-1, 2006), the following values were obtained for:

- \( M_{j,Rd} = 231,96\text{Knm} \)
- \( S_{j,ini} = 6,691*10^{10} \text{Nmm/\text{rad}} \)

With these values, the simplified, bi-linear interdependent bending moment-rotation curve were created, for the joint with 20mm thick end plate, figure 5.
4. IDEA StatiCa Analysis

The IDEA StatiCa Connections program was used to analyse the joint. IDEA StatiCa Connections is a program based on Component Based Finite Element Model (CBFEM). The CBFEM method keeps the key elements of the component method and combine them with the FEM calculation capabilities.

IDEA StatiCa Connections can perform two types of analysis:

- Static analysis with material and contact nonlinearities to analyse the state of stress and deformation.
- Second grade analysis for buckling analysis.

The materials of component for the FEM analysis are identical to the materials considered for the EC3 analysis.

Shel elements are used for modelling of the elements of the joint. Bi-dimensional elements with four nodes in the corners of the elements are used, each node have six degrees of freedom: three translations and three rotations.

Contacts between plates have a major impact in the redistribution of forces in the joint. Contacts between plates are modelled according to the “Penalty” method. The advantage of the “penalty” method is the automatically assemble of the model. This method is controlled by an iterative algorithm to achieve better numerical performance.

Figure 5. The interdependent relation bending moment-rotation bending moment-rotation according EC3.

Figure 6. Analyzed node configuration (IDEA StatiCa).

Figure 7. IDEA StatiCa mesh elements.
The Multi-Point Constraint (MPC) contact method is used for welding contact modelling. This method allows connecting elements with different meshing densities. Following the analysis in the IDEA StatiCa Connections program the following values were obtained for:

- \( M_{j,Rd} = 237.3\, \text{Knm} \)
- \( S_{j,ini} = 10.81 \times 10^{10} \, \text{Nmm/\text{rad}} \)

Also, was resulting the interdependent relation bending moment-rotation curve, figure 8.

5. **Ansys analysis**

Analyses were performed with the FEM program Ansys Workbench. Three-dimensional elements “Solid 185” were used to model the joint elements. The “Solid 185” element have 8 nodes, and each node have 3 degrees of freedom, orthogonal translation.

The contacts were modelled with elements “Conta185” and “Targe170”, to represent the contact and sliding of the elements. This element is located on the surfaces of solid 3D element and has the same geometric characteristics as the solid element with which it is connected. For modelling the contacts, the same methods were used as for modelling in IDEA StatiCa, namely “Pure penalty” for plate contact and “MPC” for welding contact.
The materials used in the modelling are described by a bilinear stress-strain curve, simplified according to Von-Mises criteria to define plasticity. Due to the fact that Ansys Workbench is not a program specifically designed for calculating joints, some manipulations were required. It is considered a column of 3m height and a 3m beam, according to the static scheme, figure 9. The loading type is displacement mode. For the evaluation of the rotation, the geometric transformation was used, according to figure 10. The meshed model is presented in figure 11.

Figure 11. Meshed model in Ansys Workbench.

Figure 12. The interdependent relation bending moment-rotation according Ansys Workbench (node 1, t=20mm).

Figure 12 shows the interdependent relation bending moment-rotation curve for node 1, with a 20mm end plate thickness resulting from FEM modelling in Ansys Workbench.
In Ansys the joints with end plate thickness \( t=15 \text{mm} \) and \( t=25 \text{mm} \) were analysed, to study the influence of the end plate on the node behaviour. For these joints the characteristic curves were obtained for interdependent relation bending moment-rotation (Figure 13, Figure 14).

6. Observations
Figure 15 shows the interdependent relation bending moment-rotation curves obtained by three different methods: calculated manually according to the component method in EC3; obtained from finite element modelling using the IDEA StatiCa program and obtained by finite element modelling with Ansys Workbench.
In figure 15 it can be noticed that the interdependent relation bending moment-rotation, obtained from EC3 manual calculation and following modelling in the IDEA StatiCa provide approximately similar values. These similarities are due to the fact that both calculations are based on the method of
components for the calculation and designing of the joints. At the same time differences between the slopes of the initial stiffness and the “consolidation” slopes are observed after the plastic hinges are formed. These differences result from the linear and simplified mode in manual computation and complex phenomena that are considered in the FEM computation.

The results from Ansys provide superior values for the moment, but the initial stiffness slope is similar to those in EC3. The differences of moment values are due to the fact that the bending moment is added to the beam stiffness and other elements of the joint and the plastic hinge is distributed to other element due to the isotropic hardening properties.

Figure 16 shows the interdependent relation bending moment-rotation for nodes 1, 2 and 3 obtained using the Ansys Workbench.

![Figure 15](image15.png)

**Figure 15.** The interdependent relation bending moment-rotation according EC3; IDEA StatiCa; Ansys Workbench (t=20mm).

![Figure 16](image16.png)

**Figure 16.** The interdependent relation bending moment-rotation according EC3; IDEA StatiCa; Ansys Workbench (t=20mm).
In figure 16 it is observed that the slope of the initial stiffness is similar in the elastic domain. It is noted that the bending moment capacity is dependent on the thickness of the end plate, with increasing thickness of the end plate increases the resistance to the bending moment. Figure 16 confirms that the behaviour of the joint is governed by the geometry of its components, especially the thickness of the end plate. Increasing the thickness of the end plate can lead to the failure of the column flange, which is inadmissible in the design of the joints, because that contradicts the “strong column – weak beam” principle. It was also confirmed that the thickness of the end plate influences the diameter of the bolts, and in order to obtain an efficient design, the ratio between the thickness of the end plate and the diameter of the bolts should be approximately equal to 1.2 (Venghiac, V.M., Stașcov, M., Budescu, M., 2017).

7. Conclusions
The results were focused on the study of the influence of variation of the thickness of the end plate on the behaviour of the joints as well as on the differences between the calculation methods. The FEM models developed with the Ansys program have shown that the thickness of the end plate is one of the most important parameters in terms of joint behaviour and moment capacity. According to the component method from EC3, resistance and rotation capacity of the connection is directly influenced by the resistance and ductility of weaker components in the joint. This component must act ductile and leave the possibility of redistribution of the internal forces to the other components. Bolts and welds are fragile, in those areas stress concentrations must be avoided, and these elements should be strengthened, to ensure the plastic behaviour and deformations of the ductile components.
It has been observed that the strength is proportional to the thickness of the end plate. At the same time, these results have demonstrated that FEM models are powerful tool for improving knowledge of joint design. The substantial differences between the results obtained by two different calculation programs indicate that the IDEA StatiCa Connection program is a program designed exclusively for the calculation of the joint, namely it has all safety and load reduction coefficients set according to the EC3 provisions, which substantially reduces the moment capacity of the joint.
The Ansys Workbench program is a multiphysics program that is not only designed for joint designing, and has no pre-set safety coefficients, it provides results that are provided only of element geometry and material properties. For this reason, the Ansys models/results must be adjusted to EC3 computing conditions.

8. References
[1] Stașcov M, Venghiac M V 2017 Beam-to-column connections with hysteretic dampers. Buletinul Institutului Politehnic din Iași, 63(67), pp. 75-88
[2] Eurocod 3. SR EN 1993-1 2006 Proiectarea structurilor din oțel, ASOCIAȚIA DE STANDARTIZARE DIN ROMÂNIA. București
[3] FEMA 350 2005 Recomended seismic design criteria for new steel moment-frame buildings.. Washington DC, USA: Federal Emergency Management Agency
[4] ANSI/AISC341 2016 Seismic Provisions for Structural Steel Buildings. Chicago (IL): American Institute of Steel Construction
[5] Engelhardt M D, Winnebeger T, Zekany A J, Ptyaraj T J 1996 The dog bone connections: part 2. Modern Steel Construction
[6] Engelhardt M D, Husain A S 1993 Cyclic loading performance of welded flange-bolted web connections. Journal of Structural Engineering, 119(12), pp. 3537-3550
[7] Tsavdaridis K D, Papadopoulos T 2016 A FE parametric study of RWS beam-to-column bolted connections with cellular beams. Journal of Constructional Steel Research, 116, pp. 92-113
[8] Maggi Y I, Gonçalves R M, Leon R T, Ribeiro L F L 2005 Parametric analysis of steel bolted end plate connections using finite element modelling. Journal of Constructional Research, 61, pp. 689-708
[9] Stankiewicz B 2013 Parametric analysis of stiffness of bolted end-plate connections of I beams using finite element method. *Journal of Civil Engineering, Environment, Vol. 30, No 60 (2/13)*, pp. 231-242

[10] Abolmaali A, Matthys J H, Farooqi M, Choi Y 2005. Development of moment – rotation model equations for flush end-plate connections. *Journal of Constructional Steel Research, Vol. 61*, pp. 1595-1612

[11] Chen S, Du G 2007 Influence of initial imperfection on the behaviour of extended bolted end-plate connections for portal frames. *Journal of Constructional Steel Research, Vol. 63*, pp. 211-220

[12] Brunesi E, Nascimbene R and Rassati G A 2014 Response of partially-restrained bolted beam-to-column connections under cyclic loads. *Journal of Constructional Steel Research, 97*, pp. 24-38

[13] Jablonska-Krysiewicz A 2015 Finite element modeling of the behaviour of steel end-plate connections. *Czasopismo Inżynierii Lądowej, Środowiska i Architektury*, pp. 173-178

[14] Moshaly E, El-Heweity M, Abou-Elfath H and Osman M 2011. Finite element analysis of beam-to-column joints in steel frames under cyclic loading. *Alexandria Engineering Journal, 50*, pp. 91-104

[15] Venghiac V M, Stașcov M, Budescu M 2017 Analize privind îmbinările grindă-stâlp cu șuruburi. *Proceedings of the 15th National Conference on Steel Structures with International Participation – 15 CONMET, ISSN 2559-0812*, pp. 227-237

[16] Dubina D, Grecea D, Ciutina A, Danku G, Vulcu C 2010 *Calculul și proiectarea îmbinărilor structurale din oțel în conformitate cu SR-EN 1993-1-8 Recomandări, comentarii și exemple de aplicare Redactarea II*. Timișoara