Survey on Design Procedures in Numerical Simulations of End-plate Moment Connections

Aleksandra Arsić
Research Assistant
University of Belgrade
Faculty of Mechanical Engineering

Vlada Gašić
Associate Professor
University of Belgrade
Faculty of Mechanical Engineering

Nenad Zrnić
Professor
University of Belgrade
Faculty of Mechanical Engineering

This paper gives survey on up-to-date studies which deal with numerical simulations of bolted end-plate moment connections. The selection is performed with the aim to cover different design concepts which reflect the problem of joint behaviour in general. Main characteristics of joint design and corresponding analysis features are presented in tabular form intended to serve as guidance for appropriate modelling of bolted end-plate connections. Consequently, the procedure for conducting numerical simulation in proper mode is proposed by authors. An illustrative example-case study of analysis of a extended stiffened end-plate bolted beam-to-column joint is shown. The design case is modelled with two finite element software packages which have different approaches and levels of complexity. Finally, the analytical calculation of this joint is done according to Eurocode 3 with the aim of validation. The moment-rotation curve, as a basis in this field, is obtained for each model. The results from numerical simulations with both the approaches are in good correlation. Basic characteristics of two mentioned design approaches are also explained as an orientation for engineers and reserchers in this field.

Keywords: *survey, end-plate joint, FEA, EC3, moment-rotation*

1. INTRODUCTION

Bolted connections with end-plate are widely used in steel buildings and framed structures. They present the main spots of calculation considering the overall structural safety. End-plate bolted (EPB) connections can be classified as flush or extended, with or without stiffeners of end-plate and column, and with a different number of bolts at the tension and compression zone. Hence, their design and mechanical behavior are not so easy to analyze due to various components of different properties and geometrical discontinuities.

In many publications, researchers agree that in frame analysis, the rotational behavior of joints should be considered [1, 2] and at least analyzed throughout with the moment-rotation (M-Ø) diagram.

The analytical approach of this type of joints is covered with EN 1993-Part 8 (EC3) [3] which gives design rules to predict flexural plastic resistance and initial stiffness of connection. This standard can be considered as basic background for the insight of joint behaviour but also has some deficiencies which were noted in [4, 5]. Generally, the most reliable way and direct method of obtaining the moment-rotation curve are experimental investigations. However, experimental tests are very expensive and require a huge amount of time. Hence, scientists and designers increasingly rely on finite element methods (FEM) of analysis through various software packages.

The development of computer capabilities enabled the parallel and extremely fast development of software for various simulations based on FEM. This is expressed to the extent that numerical studies from only twenty-thirty years ago can be viewed today as a guidance and inspiration, and much less extensive as present studies. Even today, computational efficiency is very important issue, considering that calculation of complex models can take days due to many finite element analysis (FEA), many nonlinearities involved, etc. Especially in the field of title topic, there are many factors during the modelling process that can affect the analysis of behaviour.

Within the numerical simulations of behaviour of EPB joints, the software packages can be generally separated in two groups: (I) well-known, often used and recognizable packages in engineering practice in last thirty years; (II) specific, custom-made packages which occured in recent time. The second group of software was developed with the aim to adopt design process in FEM to correspond to EC3 postulations. As example, one may find the software IDEA StatiCa [6] as easy-to-use package for joint calculation inteded for dayly engineering practice. In further text, the authors would try to point out the charactersics and advantages for each of the mentioned group.

This paper is organized in two main parts. The first part presents a brief overview of numerical analyses over the past ten years, followed by important guidelines when it comes to conducting numerical simulations of the behavior of an EPB joints. The second part of the paper deals with an example of conducting a finite element (FE) analysis of one extended EPB joint by means of two highly different software packages primarily in terms of level of complexity. The results of both approaches are presented in the form of moment vs. rotation curves.

Correspondence to: Aleksandra Arsić, research assistant
Faculty of Mechanical Engineering,
Kraljice Marije 16, 11120 Belgrade 35, Serbia
E-mail: aarsic@mas.bg.ac.rs

Results are compared with the calculations obtained by EC3 standard which can be considered as basic validation when numerical approach is adopted for the analysis of joint behaviour.

2. REVIEW: RECENT NUMERICAL ANALYSES OF END-PLATE JOINTS

High costs and long-time lasting of the experimental investigations have led that numerical simulations are increasingly popular in engineering in general. Finite element analysis are adequate for performing many parametric studies which are especially important in this field because of different components involved in a single joint. The more accurate finite element model for parametric analysis is particularly important.

Due to the rapid development of FEA software in the last decade, a review of numerical researches has been conducted for last ten years (since 2013). It is shown here for which purpose the researchers used numerical analysis to examine behaviour of EPB joint.

Al-Fakih et al. [7] have shown that FEA can predict the behavior of the connection to an acceptable level of accuracy. Girao Coleho [8] presented a finite element analysis of partial strength steel joint of a previous experimental study (2013). Good agreement with the experimental data, regarding the general response of the moment-rotation curve, was used for further parametric analysis. The author investigated the affect of beam depth and end-plate thickness, which are considered to be influential factors on joint ductility.

Another parametric study was carried out in [9]. Two end-plate configurations were considered: four bolts and multiple row extended end plates. The studied parameters were: beam depth, end-plate thickness, bolts diameter, bolts pitch, bolts gage, and end-plate stiffener.

Taufik in [10] and Vahid et al. in [11] investigated flush end-plate steel connections by means of finite element analyses. Rui Bai et al. [12] used FEA to validate the prying force calculation with equations they presented in their work. Chen et al. have investigated in [13] the influence of end-plate stiffeners (angle and tickness of the stiffener) on the initial rotational stiffness of extended end-plate joints.

In [14] (Tartaglia et al.) it is investigated the behavior of extended stiffened beam-to-column end-plate joints under seismic loading. They made extensive parametric studies, based on finite element analysis, to observe the influence of the geometry of the stiffeners on the joint performance.

The study [15] is important because it is devoted to the comparison of the design rules for full strength extended stiffened end-plate bolted (ESEPB) joints in EC3 with the AISC 358-16 [16]. The aim of their work was to verify and to compare the effectiveness of both design procedures through the results of a comprehensive parametric study based on finite element simulations.

Extensive parametric study was done in [17] on extended end-plate bolted beam-to-column connections subjected to both monotonic and cyclic loading to study the effect of many parameters such as shear force, diameter of bolt, thickness of end-plate, and the effect of using end-plate stiffeners.

In [18], an experimental and numerical examination of the behavior of end-plate moment connections with the four bolts in a row was performed, as well as the characteristic component of this joints – T-element.

Paulina Krolo [19] introduced in her numerical analysis a steel damage model which allows the degradation of strength and stiffness and the appearance of a joint fracture after the accumulation od deformations. She worked on a hysteresis envelope model that defines the ratios between the monotonic properties of the joints and the properties of the joints during cyclic deformation. The proposed models are based on the hysteresis curves of the joints obtained by numerical simulations.

In order to examine the seismic behavior of high-strength steel extended end-plate connection in frame structures the three-dimensional finite element model was established in [20]. The accuracy of the finite element model was verified by four experimental tests. It has been shown that the proposed FE model give a quite accurate prediction for the pinched hysteresis loops and deformation modes of bolted end-plate connections.

In order to summarize the key points of mentioned papers in last decade, Table 1 provides an overview of main characteristics of the given numerical models. The tabelar representation covers the following parameters: type of EPB joint, number of bolts in tension region, presence of stiffener, type of finite element for the structural elements and bolts, analysis features such as contact problem/pre-loading of bolts/geometrical and material non-linearity and type of loading. Additionaly, it is given the software used for numerical simulations. At first, one may see that Ansys and Abaqus are most often used in studies due to the fact that they present well-known support in engineering calculations. Table 1 can serve as guidance for appropriate modelling of EPB joint behaviour, along with possibility to narrow the level of generality in such designs. Also, it can be used for prediction of extension or improvement of modelling assumptions and parameters.

In any case, it hase to be pointed out that every numerical simulation should be verified at least by analytical or experimental approach.

There is always some lack of the data in explanation of performing numerical simulations even in papers which are considered as comprehensive and lengthy. Hence, next chapter is devoted to propose the procedure for conductiong the appropriate simulation within the title topic.

3. PROCEDURE FOR CONDUCTING NUMERICAL SIMULATIONS OF EPB CONNECTIONS

In this chapter, all the necessary steps for an adequate simulation of the behavior of the end-plate beam-to-column joint will be proposed.

In order to reduce the computational run time and storage requirements for the analysis some of the software packages provides the ability to model only half or even a quarter of the model if there are one or two levels of symmetry. It should be noted that the use of this option implies that the model, in addition to symmetrical geometry, must also be symmetrically loaded and symmetrically restrained. This option should be considered in initial stages of design due to try-out of software/computer capabilites.

After modeling and positioning of all elements of the assembly, it is necessary to define the material of the structure. Defining the properties of the material primarily depends on the possibilities provided by the software for finite element analysis. The representation of the material in the software packages actually implies the definition of the stress-strain curve. Depending on the accuracy and the allowable strain required for the analysis Eurocode [21] provides the following assumptions for the material behaviour that can be used in finite element analysis: (1) elastic-plastic without strain hardening, (2) elastic-plastic with a nominal plateau slope, (3) elastic-plastic with linear strain hardening – bilinear curve and (4) true stress-strain curve modified from the test results – multilinear curve. Undoubtedly, the best solution is to experimentally obtain an engineering stress-strain curve that must be translated into a true stress-strain curve and then implemented in software. However, this should be done only if really needed due to fact that structural materials are well-known and unmodified in long-time period.

The second important step is to define the mutual contacts of all elements in the joint. Depending on the loads, material, boundary conditions, and other factors, surfaces can come into and go out of contact with each other in a quite unpredictable and unexpected way. The contacts that need to be defined are: (a) contact between the end-plate and the column flange, (b) contact between bolt nuts and the surfaces of end-plate and column flange and (c) contact between bolt shanks and holes of both end-plate and column flange. For each contact area, it is necessary to define the properties that will enable the transfer of forces and stresses from one element to another. For example, for a contact area (a) both tangential and normal contact properties must be defined. In the normal direction, the transmission of force must be ensured when the surfaces are in contact and it is necessary to avoid the penetration between the elements in the contact pair. While the transmission of force in the tangential direction is provided by defining the friction between the surfaces.

Welds could be modeled as individual parts with defining the properties of the weld material. However, usually only appropriate constraints are set between the surfaces of welded elements.

Meshing of element has a noticeable influence on the accuracy of the results of the finite element analysis. Mesh sensitivity study should be conducted with different mesh densities in order to find optimum mesh density required for the FEA which can provide considerably accurate results with less run time. Depending on the geometry, each part in the model must have the appropriate type and size of the finite element. In contact regions, the mesh should be finer. Due to the holes in the end-plate and column flange, it is necessary to partition the elements in order to obtain a uniform and symmetrical mesh without any distorsion. The same goes for bolts. In numerical analyzes where bending is dominant, when choosing the type of finite element, one should pay attention to the phenomena of shear locking and hourglassing, [22].

The boundary conditions (BCs) and applied loads have to be applied as close as possible to the experimental test conditions (if known).

The representation of mentioned procedures will be shown in the next chapter for one specific model of end-plate moment connection. Delibaretely, numerical simulations will be performed on two software packages belonging to the different groups as mentioned in introductional chapter. First one is modelling with ABAQUS 6.13 [23]. Secondly, the same model will be done in the IDEA StatiCa 8.0 software [6], which combines FEA with the analytical method of joints based on the component method (CM) specified in the EC3. Finally, the stiffness and strength of considered joint will be calculated according to EC3 and the main result of all methods will be compared.

4. ILLUSTRATIVE EXAMPLE

Numerical simulation is performed for extended stiffened end-plate bolted beam-to-column joint which is exposed to monotonic loading by using two different software, ABAQUS 6.13 and IDEA StatiCa 8.0. Upon the results from the used software packages, the schematic moment-rotation response curves will be presented and compared.

4.1 Description of joint geometry

The joints consist of: a steel beam, a column, a double extended end-plate, rib stiffener of end-plate, a column web stiffeners (continuity plates) and high-strength preload bolts. Geometrical features of joint are shown in Figure 1 while the specific parameter values are given in Table 2. All parts of the joint, except bolts, are made of structural steel S355. A 120-cm-long beam made of IPE450 cross section is welded to the end-plate, which are then bolted to a column flange made of HEB340 cross section with 8 preload bolts M30 steel grade 10.9. The column stiffeners and rib of end-plate were subsequently welded to the coresponding joint surfaces.

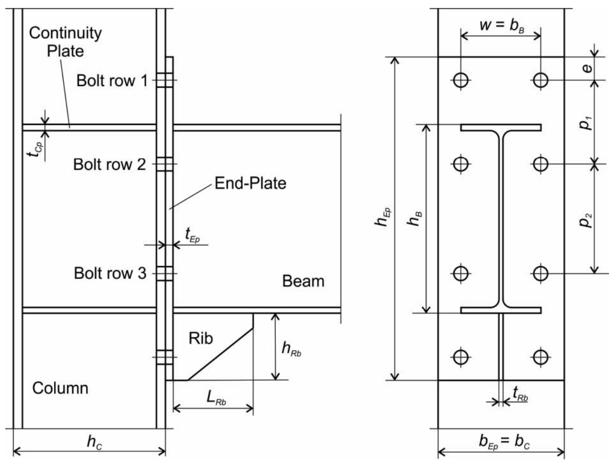

Figure 1. Features of the extended EPB stiffened joint

Table 2. Geometrical features of the reference joint

$b_B = w$ [mm]	$b_{Ep} = b_C$ [mm]	L_{Rb} [mm]	h_B [mm]	h_C [mm]	h_{Ep} [mm]	h_{Rb} [mm]	t_{Ep} [mm]	t_{Rb} [mm]	t_{Cp} [mm]	e [mm]	p_1 [mm]	p_2 [mm]
160	300	190	450	360	770	160	18	10	15	55	200	260

4.2 Development of model in ABAQUS software

Before meshing all parts of the model were partitioned in order to apply structured meshing tehnique to have a proper element shape, especially for round bolts and corresponding plates. All connected parts including the beam, column, end-plate, stiffeners and bolts were modeled using linear 8-node hexagonal (brick) FE type - C3D8I which is recommended in [24]. The mesh size change in function of the joint parts and in places of contacts the finer mesh was formed. Throughout the thickness, at least three elements were adopted. The bolt head, body and nut are modelled as one part while the bolt threads and the extended part of the bolt outside the nut are neglected. The finite element mesh for the beam-to-column joint and its parts is shown in Figure 2. The number of elements of the considered joint is 139 189.

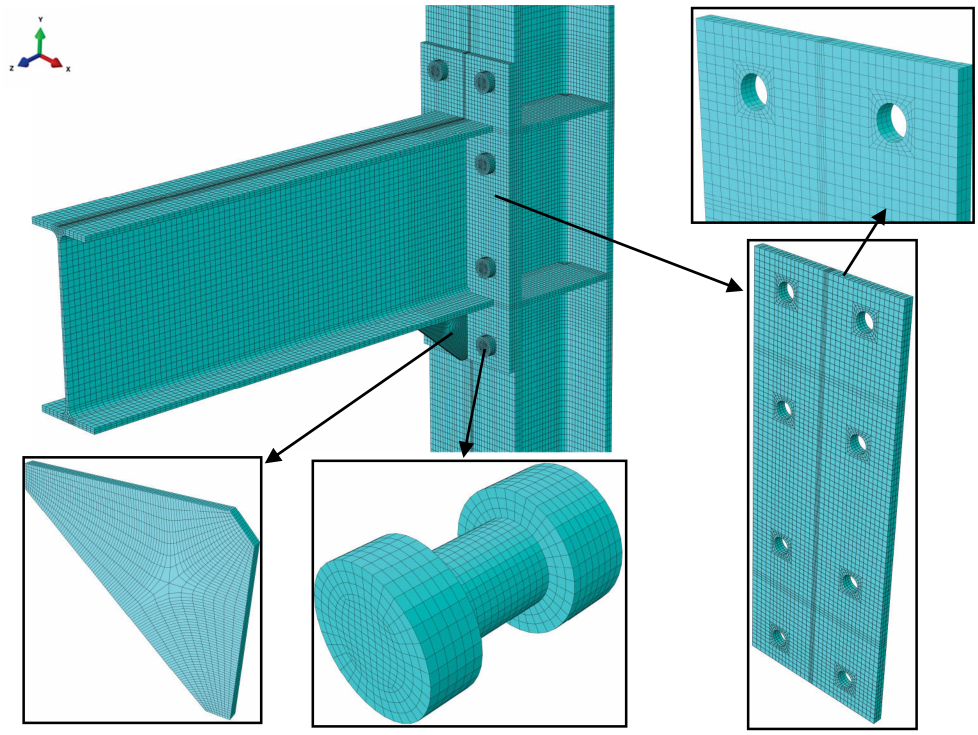

Figure 2. Meshing of the joint elements – ABAQUS

Geometric and material nonlinearity were adequately considered. An isotropic multilinear model of the material is applied, which includes the hardening between the yielding of the material and reaching the tensile strength. The plastic properties of steel are defined as the true stresses and strains (1) where the nominal values of stress σ_{nom} and strain ε_{nom} are obtained as mean values of laboratory data given in [19]. Young's modulus of elasticity is $E = 199{,}000$ MPa, while the Poisson ratio is equal to $\nu = 0.3$.

$$\sigma_t = \sigma_{nom}(1 + \varepsilon_{nom})\ ; \quad \varepsilon_t = \ln(1 + \varepsilon_{nom}) \qquad (1)$$

The stress–strain relationship for the high-strength preload bolts class 10.9 was taken as elastic–plastic trilinear curve, adopted by Wang et al. [24], and it is presented with Figure 3 and Table 3. Young's modulus of elasticity for high-strength bolts is $E = 206{,}000$ MPa, while the Poisson ratio is also $\nu = 0.3$.

Table 3. Material parameters of high-strength bolts

Stress σ [MPa]	0	990	1160	1160
Strain ε	0	0.00483	0.138	0.15

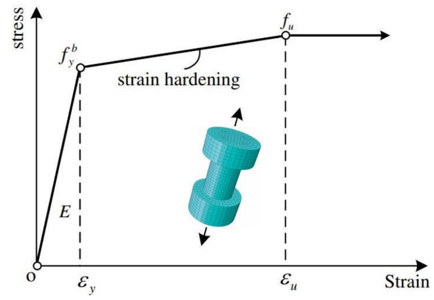

Figure 3. Stress–strain relationship for high-strength bolts

"Surface-to-surface" interactions were used to model the contacts between: end-plate and column flange, bolt head and nut with coresponding surfeces of end-plate and column flange, and between bolt body and holes. The surfaces of more rigid portions of the model were set as master surface. Contact properties consist of two parts: tangent and normal contact. The normal behavior is formulated by "Hard" contact law with allowing separation after contact while the penalty formulation with friction coefficient 0.3 is selected to model tangential behavior. "Tie" constraint was used to represent the full penetration weld behavior, specifically to model the connection between the beam and the end-plate and between the rib and end-plate, as well. The same constraint was adopted to model the interaction between the continuity plates and column.

The bolts are preloaded in the first step of the analysis using the “Bolt load” option. The pre-tension is simulated by splitting the bolt body into two equal parts and applying a magnitude of preload force to two parallel surfaces through an axis formed along the bolts. The value of clamping force was $F_p = 392700$ kN, recommended by EC3. After applying the force, the length of bolts is fixed at their current position. This technique helps to avoid the problem with extensive elongation of the bolts under the loading. The load is simulated by displacement control at the end of the beam with maximum value of 60 mm. Appropriate BCs were applied to the base and top of the column.

Modeling of the considered joint takes certainly a fiew hours and software completed the calculation in 50 minutes at work station with four processors and 24 GB of RAM memory.

4.3 Development of model in IDEA StatiCa software

IDEA StatiCa is a component-based FEA software package for steel connection design. It can be used for structural evaluation or design of a variety of welded and bolted structural steel connections and base plates. It is a semi-automated modeling of joints, where software independently creates a numerical model based on a given geometry discretized by shell finite elements. The modeling stages including FE type selection, mesh generation and analysis type selection are fully automated. Here, the intention is to perform the modelling with the usage of shell elements because this approach is simpler, less time-consuming and can provide good insight of the performance of each joint component [13]. The joint model is shown in Figure 4.

Modelling in this software considers an elastic-plastic material model for all the elements of the joint while the steel members are meshed with 4-node quadrilateral shell elements. Other predefined modeling features, which the program has by default, are: the bolts are modeled as massless points that act as a link between the holes of two plates; the weld is modeled as a special elastic-plastic element on the multi-node constraint; the approximation of 8 elements in the critical edge with mesh size between 50 to 10 mm and limit plastic strain of 5%.

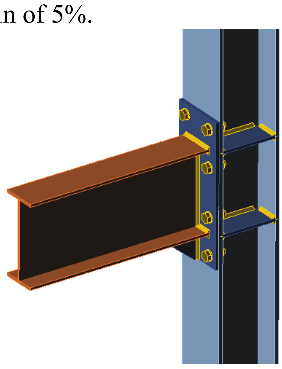

Figure 4. Considered joint model – IDEA StatiCa

Modelling of the considered joint takes approximately 10 minutes and software completed the calculation in 2 minutes so the chosen software has highly reasonable computational time.

4.4 Eurocode calculations

EN 1993-1–8 [3] with implemented component method provide analytical models to predict the mechanical behavior of the joints in terms of their strength and stiffness. The calculations for design moment resistance and initial stiffness of extended EPB joints have been provided on the basis of the T-stub yield line theory, but the influence of the rib stiffeners is not included.

As the considered model has the rib stiffener in the compression zone, the calculation given by EC3 is complemented with recommendations presented in [25], [26].

Precisely, the influence of the rib was taken into account in the calculation of the center of compression and lever arm, design compression forces acting on beam web component, and design strength and the initial stiffness of the joint, in general.

Design moment resistance of the considered joint is calculated with:

$$M_{j,Rd} = \sum_{r=1}^{3} h_r \cdot F_{tr,Rd} \tag{2}$$

where $F_{tr,Rd}$ is the effective design tension resistance of bolt-row r (Figure 1) and h_r is the distance from bolt-row r to the centre of compression. The adopted position of the center of compression is in the centroid of the stiffener and the bottom beam flange.

The initial stiffnes of the considered joint is evaluated by:

$$S_{j,ini} = \frac{E \cdot z^2}{\sum_i \frac{1}{k_i}} \tag{3}$$

where z is the lever arm taken as the distance from the center of compression to a midpoint of beam upper flange and k_i is the stiffness coefficient for basic joint component i. The stiffness coefficients that is taken into account are column web panel in shear - k_1, column web in compression - k_2, rib on the compression side - k_{Rib} and the equivalent stiffness coefficient related to the bolt rows in tension - k_{eq}, calculated using (4), (5) and (6).

$$k_{eq} = \frac{\sum_r k_{eff,r} \cdot h_r}{z_{eq}} \tag{4}$$

In (4) $k_{eff,r}$ is the effective coefficient for bolt-row r given by (5) and z_{eq} is the equivalent lever arm determined with (6).

$$k_{eff,r} = \frac{1}{\sum_i^{3,4,5,10} \frac{1}{k_{i,r}}} \tag{5}$$

$$z_{eq} = \frac{\sum_r k_{eff,r} \cdot h_r^2}{\sum_r k_{eff,r} \cdot h_r} \tag{6}$$

In (5) $k_{i,r}$ is the stiffness coefficient representing component i relative to bolt-row r, along with denotation from [3].

The summary of relevant results are given in Table 4 and Table 5.

Table 4. Joint flexural resistance by EC3

h_1	h_2	h_3	$F_{t1,Rd}$	$F_{t2,Rd}$	$F_{t3,Rd}$	$M_{j,Rd}$
[mm]	[mm]	[mm]	[kN]	[kN]	[kN]	[kNm]
579.635	379.635	119.635	187.3	580.1	533.6	**393**

Table 5. Stiffness of the joint by EC3

k_1	k_2	k_{eq}	$k_{eff,1}$	$k_{eff,2}$	$k_{eff,3}$	z_{eq}	z	$S_{j,ini}$
[mm]	[mm]	[mm]	[mm]	[mm]	[mm]	[mm]	[mm]	$\left[\frac{kNm}{rad}\right]$
4.56	∞	4.67	0.59	2.77	2.77	369.2	467.34	**87846**

4.5 Results and discussion

The results of the numerical analysis, from both software, are shown in Figure 5 through moment vs. joint rotation curves and compared with the joint behavior results obtained by EC3.

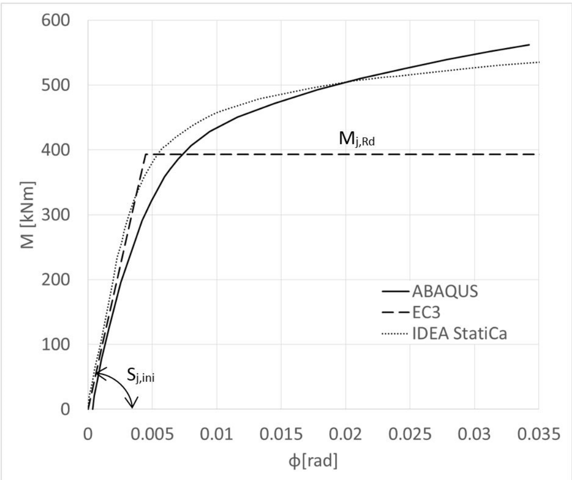

Figure 5. Moment vs. joint rotation curves

Even though the previously mentioned standard deficiencies are included in the calculation, the small value of design resistance of end-plate in bending (due to missing end-plate stiffener in the tension zone) has the significant impact on the overall design moment resistance of the considered joint. It can be seen from the diagram that approach of EC3 underestimates the moment of resistance of the connection, which was also mentioned in [4].

The largest joint plastic deformation in both software (not shown in this paper) occurred in the tension zone of the end-plate, thus confirming the Eurocode prediction that the end-plate is the weakest component in the joint.

It is obvious that the applied additions to the calculations of analytical approach regarding the shifting the compression center due to the presence of stiffeners give quite good results, when it comes to the initial stiffness of the joint, because the obtained value matches the values of the numerical analyses.

In general, there was good agreement between the IDEA StatiCa and the ABAQUS results regarding moment vs. rotation curve. In addition to the ease of use, the most important characteristic of IDEA StatiCa was found to be the computational time in which the results can be obtained in a fraction of time compared to the conventional FEA codes such as ABAQUS. Hence, ABAQUS software allows extracting the results values from the model during the post-processing stage (such as stresses, displacements, contact pressure, equivalent plastic strain, etc.) of every FE for every force increment during the simulation. This is needed for analysis where cyclic loading is included which is often necessary for scientific research in this field.

5. CONCLUSION

For advanced structural analysis of connections and frames it is necessary to consider the behaviour of the joint. However, their mechanical behavior are complex for analyzing because the EPB moment connections include various components of different properties and geometrical discontinuities.

Despite the fact that real and most relevant behavior of single joint is obtained throughout experimental tests their long-term and high costs led to numerical simulations become increasingly popular in a engineering practice.

This paper gives comprehensive survey on the last decade research of end-plate moment connections from the aspect of numerical simulations. It is noticed that procedure of performing simulations is never fully defined in the available papers. Hence, in chapter 3 authors tried to explain all the necessary steps for an adequate simulation of the behavior of the EPB joint.

Finally, it can be concluded from results of an illustrative example of extended EPB joint with end-plate stiffener in the compression zone that:

- EC3 underestimates the moment of resistance of the connection (once more confirmed),
- the presence of end-plate stiffeners should be somehow included in analytical calculations of EC3 due to the extension of the lever arm which is obvious.

Regarding the usage of FE software packages that belongs to two different groups, one may found following:

- there are small deviations in results regarding the moment-rotation curve whish is basic for this field,
- custom-made, semi-automated and specialized software are easy-to-use and sufficient for dayly engineering practice. It can be also used in initial stages in scientific studies or for validation of experimental results,
- well-known, comprehensive software are complex for proper definition of FEM but provides better insight in local effects at EPB joints which is necesary for scientific research where numerical simulation is primary approach. Long computational time should be the cost for analysis of behaviour of these joints in detail.

Also, this work can serve as orientation for creating the custom-made software where level of automation will include additional parameters which can influence the joint behaviour.

ACKNOWLEDGMENT

This work is a result of research supported by the Ministry of Education, Science and Technological Development of the Republic of Serbia by Contract 451-03-68/2022-14/200105.

REFERENCES

[1] Lui, E.M. and Chen, W.F.: Steel frame analysis with flexible joints, Journal of Constructional Steel Research, Vol. 8, pp. 161-202, 1987.

[2] Faella, C., Piluso, V. and Rizzano, G.: Structural steel semirigid connections: theory, design and software, Boca Raton: CRC Press, 2000.

[3] EN 1993:1–8: Design of Steel Structures - Part 8: Design of Joints, European Committee for Standardization, Brussels, 2005.

[4] Abidelah, A., Bouchair, A. and Kerdal, D.E.: Experimental and analytical behavior of bolted end-plate connections with or without stiffeners, Jurnal of Constructional Steel Research, Vol. 76, pp. 13-27, 2012.

[5] Gašić, V., Arsić, A. and Zrnić, N.: Strength of extended stiffened end-plate bolted joints: Experimental and numerical analysis, Structures, Vol. 33, pp. 77-89, 2021.

[6] IDEAStatiCa 8.0.22.44178, 2017, https://www.ideastatica.com/.

[7] Al-Fakih, K., Chin, S. C. and Doh, S. I.: Behavior of Extended End-Plate Steel Beam to Column Connections, The Open Civil Engineering Journal, Vol. 12, pp. 250-262, 2018.

[8] Coelho, G. A. M.: Rotation capacity of partial strength steel joints with three-dimensional finite element approach, Computers and Structures, Vol. 116, pp. 88-97, 2013.

[9] Dessouki, A. K., Youssef, A. H. and Ibrahim M. M.: Behavior of I-beam bolted extended end-plate moment connections, Ain Shams Engineering Journal, Vol. 4, No. 4, pp. 685-699, 2013.

[10] Taufik, S.: Numerical Modelling of Semi-rigid Connection with High Strength Steel, Study of Civil Engineering and Architecture, Vol. 2, No. 2, pp.
38-47, 2013.

[11] Vahid, S. Z., Osman, S. A. and Khalim A. R.: Monotonic and cyclic loading simulation of structural steelwork beam to column bolted connections with castellated beam, Journal of Engineering Science and Technology, Vol. 8, No. 4, pp. 416-427, 2013.

[12] Bai, R., Chan, S. L. and Hao, J. P.: Improved design of extended end-plate connection allowing for prying effects, Journal of Constructional Steel Research, Vol. 113, pp. 13-27, 2015.

[13] Chen, S., Zhou, C. and Wang, Z.: Experimental Study and Comparative Numerical Analysis of the Mechanical Behavior of Extended End-Plate Connections with End-Plate Stiffeners, The Open Mechanical Engineering Journal, Vol. 9, pp. 653-665, 2015.

[14] Tartaglia, R., D'Aniello, M., and Landolfo, R.: The influence of rib stiffeners on the response of extended end-plate joints, Journal of Constructional Steel Research, Vol. 148, pp. 669–690, 2018.

[15] Tartaglia, R., D'Aniello, M., Rassati, G., Swanson, J. and Landolfo, R.: Full strength extended stiffened end-plate joints:AISC vs European design criteria, Engineering Structures, Vol. 159, pp. 155-171, 2018.

[16] ANSI/AISC 358-16: Prequalified Connections for Special and Intermediate Steel Moment Frames for Seismic Applications, AISC (American Institute of Steel Construction), Chicago, IL, 2016.

[17] ElSabbagh, A., Sharaf, T., Nagy, S. and ElGhandour, M.: Behavior of extended end-plate bolted connections subjected to monotonic and cyclic loads, Engineering Structures, Vol. 190, pp. 142–159, 2019.

[18] Jovanović, Đ.: *Behaviour of end-plate moment connections with four bolts per row*, Ph.D. dissertation, University of Novi Sad, Faculty of technical sciences, Novi Sad, 2019.

[19] Krolo, P.: *Influence of bolted joints behaviour on seismic response of steel frames*, Ph.D. dissertation, University of Rijeka, Faculty of civil engineering, Rijeka, 2017.

[20] Lin, T., Wang, Z., Hu, F. and Wang, P.: Finite-Element Analysis of High-Strength Steel Extended End-Plate Connections under Cyclic Loading, Materials, Vol. 15, No. 8, 2022.

[21] EN 1993:1–5: Design of Steel Structures - Part 5: Plated structural elements, European Committee for Standardization, Brussels, 2006.

[22] Sun, E. Q.: Shear Locking and Hourglassing in MSC Nastran, ABAQUS, and ANSYS, 2006.

[23] ABAQUS 6.13, Analysis User`s Manual, Dassault Systemes, USA, 2012.

[24] Wang, M., Shi, Y., Wang, Y., Shi, G.: Numerical study on seismic behaviors of steel frame end-plate connections, Jurnal of Constructional Steel Research, Vol. 90, pp. 140-152, 2013.

[25] Landolfo, R., D'Aniello M., Costanzo S., et al. European pre-QUALified steel JOINTS (EQUALJOINTS), Final Report, Directorate-General for Research and Innovation (European Commission). https://publications.europa.eu/s/j7q0, 2018.

[26] D'Aniello, M., Tartaglia, R., Costanzo, S. and Landolfo, R.: Seismic design of extended stiffened end-plate joints in the framework of Eurocodes, Journal of Constructional Steel Research, Vol. 128, pp. 512–527, 2017.

Table 1. Review of numerical models of beam-to-column end-plate joints in last ten years

Autor/s, year	Type of EP	Bolts tension region	EP stiffener	FE type			Analysis features					Load	Software
							Contact sliding	Bolt		Non-linearity			
				Beam, EP	Column	Bolt		Contact	Pre-loading	Geometric	Material		
Coleho, 2013	Extended	4	–	HX8M	HX8M	HX8M	+	+	–	+	True stress-strain curve	M	Lusas
Dessouki et al., 2013	Extended	4/6	+	SHELL181	–	SOLID95	–	Unknown	+	+	Trilinear curve	M	Ansys
Vahid et al., 2014	Flush	2	–	–	–	–	+	+	Unknown	Unknown	Trilinear curve	M, C	Ansys
Rui Bai et al., 2015	Extended	4	+	SOLID185	SOLID185	SOLID185	+	+	+	+	Bilinear curve	M	Ansys
Chen et al., 2015	Extended	4	+	C3D8R	C3D8R	C3D8R	Unknown	Unknown	+	Unknown	Multilinear/ Bilinear curve (bolts)	M	Abaqus
Tartaglia, 2017	Extended	4	+	C3D8I	C3D8I	C3D8I	+	+	+	+	True stress-strain curve	M, C	Abaqus
Ashraf et al., 2019	Extended	4	+	SOLID185	SOLID185	SOLID185	+	+	+	+	Bilinear curve	M, C	Ansys
Jovanović, 2020	Flush	4	-	C3D8R	C3D8R	C3D8R	+	+	Unknown	+	True stress-strain curve	M	Abaqus
Krolo, 2020	Extended	4	-	C3D8I	C3D8I	C3D6	+	+	+	+	Chaboche model	M, C	Abaqus
T Lin et al., 2022	Extended	6	-	C3D8R	C3D8R	C3D8R	+	+	+	+	Cyclic plasticity model	C	Abaqus

Symbols: + Included – Not included M Monotonic load C Cyclic load

HX8M (Lusas) – 8-node isoparametric brick element
SHELL181 (Ansys) – 4-node quadrilateral finite strain shell element
SOLID95 (Ansys) – 20-node 3D structural solid element
SOLID185 (Ansys) – 8-node 3D structural solid element
C3D8R (Abaqus) – 8-node linear hexahedral element with reduced integration
C3D8I (Abaqus) – 8-node linear hexahedral element, incompatible modes
C3D6 (Abaqus) – 6-node linear triangular prism element