CONTACT FINITE ELEMENT ANALYSIS AS A TOOL FOR DIMENSIONING OF BOLTED CONNECTIONS

Todor Vacev[1]
Ivan Nešović[2]
Stepa Paunović[3]
Miloš Milić[4]

UDK: 624.014.2.078.4
DOI:10.14415/konferencijaGFS2017.033

Summary: *Contact finite element analysis (FEA) is a powerful tool for analysing and dimensioning of various connections. It is especially applicable in steel structures connected by bolts, where a contact between the bolt shaft and the bolt hole is a way of shear force transfer. Classic analysis of such connection simplifies the problem, thus reducing the complexity of the calculation. On the other hand, contact FEA gives much more detail of the connection behaviour. A simple example of thimbles connected by single bolt demonstrates advantages of the contact FEA solution performed by ANSYS software, but also poses questions regarding codification of the results of such calculation. Controlled and limited plasticization is one of the crucial problems which needs valid interpretation.*

Keywords: *steel structures, bolted connections, contact FEA, ANSYS software*

1. INTRODUCTION

Bolted connections take important place in steel structures. According to the classic calculations the force in shear connections is transferred by bearing pressure and by shear of the bolt shaft, assuming equal distribution of the bearing stress in the bolt hole.
The assumptions adopted for calculation are introduced in order to simplify the calculation, considering that it is almost impossible to encompass all real phenomena in the connection and to determine exact distribution of forces using manual calculation. For that purpose, much more advanced analysis methods are needed, like application of contemporary computer software based on the Finite Element Method (FEM), or some other numerical method. Aim of this paper is analysis of the stress and deformation of the specific elements in one bolted connection typical for steel structures. Special emphasis is

[1]Todor Vacev, PhD., associate professor, University of Niš, Faculty of Civil Engineering and Architecture, A. Medvedeva 14, Niš, Serbia, tel: +381 64 322 87, e – mail: todor.vacev@gaf.ni.ac.rs
[2]Ivan Nešović, Mast. Eng., student of doctoral studies, University of Niš, Faculty of Civil Engineering and Architecture, A. Medvedeva 14, Niš, Serbia, tel: +381 69 1734684, e – mail: ivan.m.nesovic@gmail.com
[3]Stepa Paunović, Mast. Eng., student of doctoral studies, University of Niš, Faculty of Civil Engineering and Architecture, A. Medvedeva 14, Niš, Serbia, tel: +381 63 1995661, e – mail: stepa.paunovic@gmail.com
[4]Miloš Milić, Mast. Eng., student of doctoral studies, University of Niš, Faculty of Civil Engineering and Architecture, A. Medvedeva 14, Niš, Serbia, tel: +381 69 4267126, e – mail: milos.cicevac @gmail.com

put on the phenomena related to contact, which encompasses the status of the elements, contact pressure, penetration, and gap. The connection is modelled and analysed using software ANSYS Workbench 14.5. The analysed connection represents joining of two parallel thimbles to the third one, placed between them. All thimbles are set vertically, where by the outer thimbles are fixed at its upper end, and the middle one is loaded at its bottom part by vertical force P=85 kN pointed downwards. The proposed bolt is M16, plain, with bolt hole diameter Ø17 mm.

2. MODELLING OF THE CONNECTION

Modelling of this connection is in fact creating of a numerical model which describes in the most approximate way its geometry, material, and its behaviour under load.
In this framework, the modelling process is developed through several phases:
- a) defining of the material and its characteristics,
- b) creating of the geometry of the model,
- c) defining of the type of the finite elements and meshing,
- d) defining of the contact surfaces,
- e) setting of the boundary conditions and load,
- f) setting of the analysis parameters,
- g) problem solving.

The steel material is adopted as materially nonlinear model of behaviour, namely bilinear kinematic with modulus of elasticity E=210 GPa, yield point f_y=235 MPa and tangent modulus of elasticity E_T=0.01E=2.1 GPa (*Fig. 1a*).
Geometry of the model is created using the *DesignModeler* module, and it consists of 4 independent geometric bodies: left thimble, right thimble, bolt, middle thimble, and „load" (fictive body used for applying of load only) – (*Fig. 1b*).

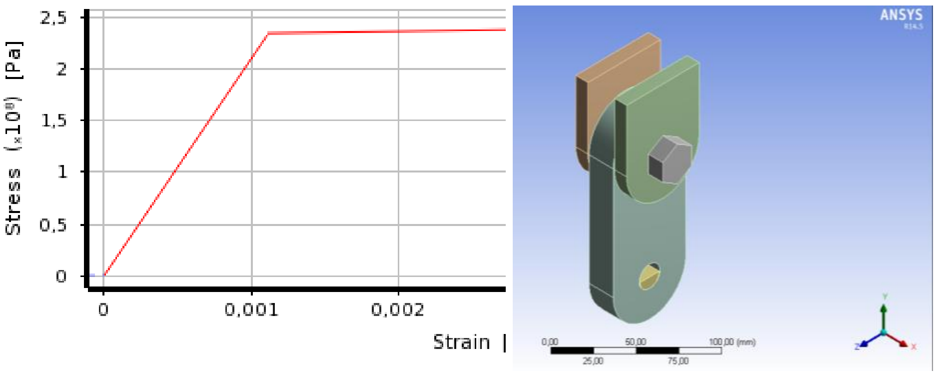

Fig. 1. a) working diagram for steel (left); b)geometry of the model (right)

For the modelng of the elements of the connection the finite element *SOLID186* from the software FE database is used. It is a solid parabolic finite element with 8 nodes and 12 midnodes, and three degrees of freedom per node (translations in X-, Y-, and Z-direction). During the meshing process elements with tetrahedron shape were used, with local increase of mesh density in contact regions. The analysis was performed for two different mesh densities with maximal element size of 10 mm, that is, 8 mm (*Fig. 2.)*

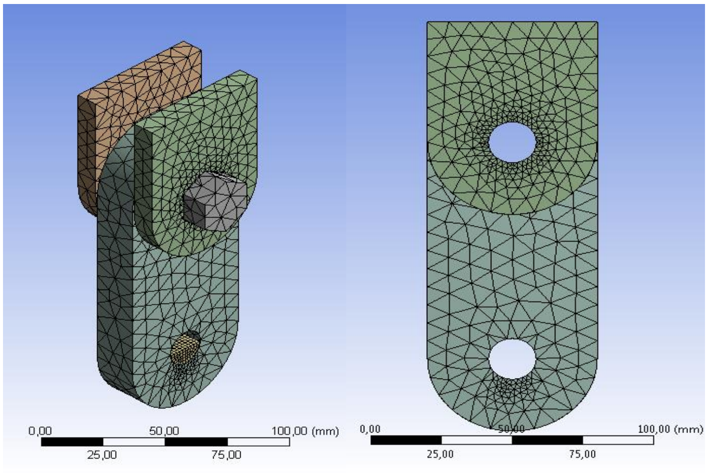

Fig. 2. FE mesh (denser mesh);isometric (left);side view (right)

Load transfer in the contact is realized by four contact pairs: bolt-right thimble, bolt-left thimble, bolt-middle thimble, and middle thimble-load. Every contact pair consists of the so called target or passive surface meshed by *TARGE170* finite elements, and contact or active surface meshed by *CONTA174* finite elements. All contacts are of *Frictional* type (with included friction of steel over steel) with coefficient of friction μ=0.2. Boundary conditions are set at two upper surfaces of the outer thimbles, as restrained displacements in all directions for the nodes on given surfaces. In this way, a fixed constraint of the two thimbles is realized (*Fig. 3a)*. The load is modelled as force with intensity of 85 kN, distributed across the upper surface of the semi-cylinder, which is not subject of the analysis, but its role is to transfer the load to the bolted connection (*Fig. 3b)*. Analysis of the proposed model covered both the geometrical and the material nonlinearity. The load was applied incrementally, with minimal number of 20 substeps, and maximal of 1500. The software uses direct method for the solving of systems of nonlinear equations.

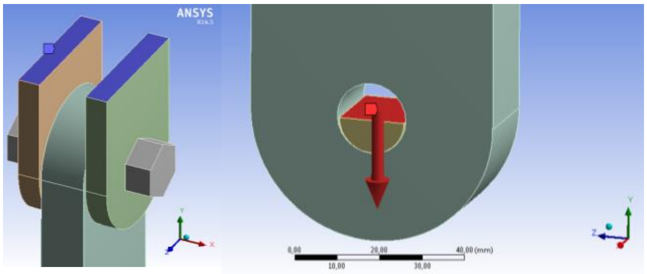

Fig. 3. Boundary conditions; a) fixed restraint (left);b) load application (right)

3. ANALYSIS OF RESULTS

3.1 Deformation

Based on the results of the analysis of the whole connection one may conclude that deformations cumulative increase from the point of fixation towards the point of load application, as expected, and that maximal value is under the the load (*Fig. 4a)*. Mechanism of deformation can be even better observed at upper thimbles (*Fig. 4b)*. The deformation of the bolt is maximal in the middle, where the middle thimble presses onto the bolt, and causes bending of the bolt shaft, also expected (*Fig. 4c)*.

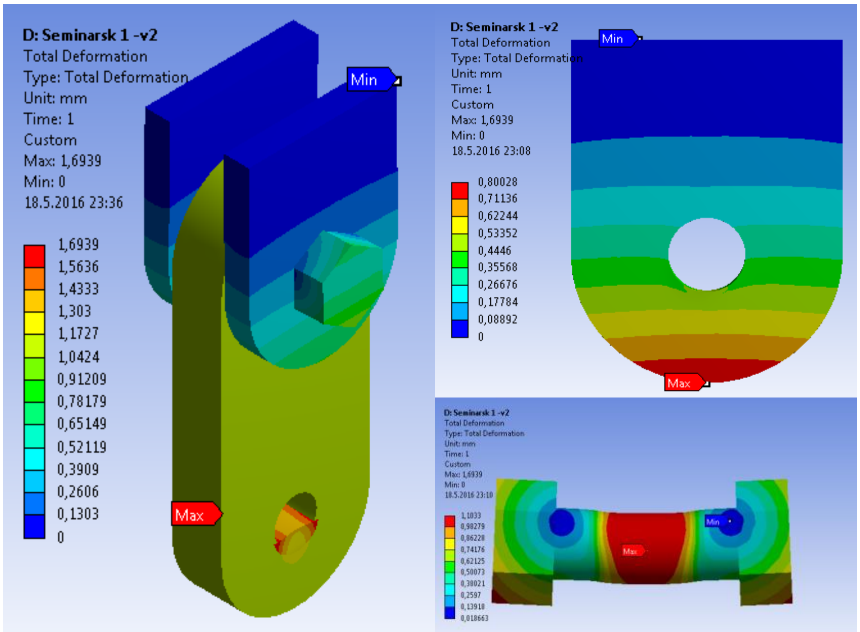

Fig. 4. Total deformation of the model; a) complete model, max. value 1.69 mm (left); b) upper thimble, max. value 0.8 mm (top right); c) bolt, max. value 1.1 mm (bottom right)

3.2 Stresses

In the *Fig. 5.* are presented von-Mises stresses across the vertical section of the whole connection (5*a*), as well as in the upper thimbles (5*b*) and in the middle thimble (5*c*). As one may see, stress distribution is symmetrical, and the values are far above the yield point for the steel used.

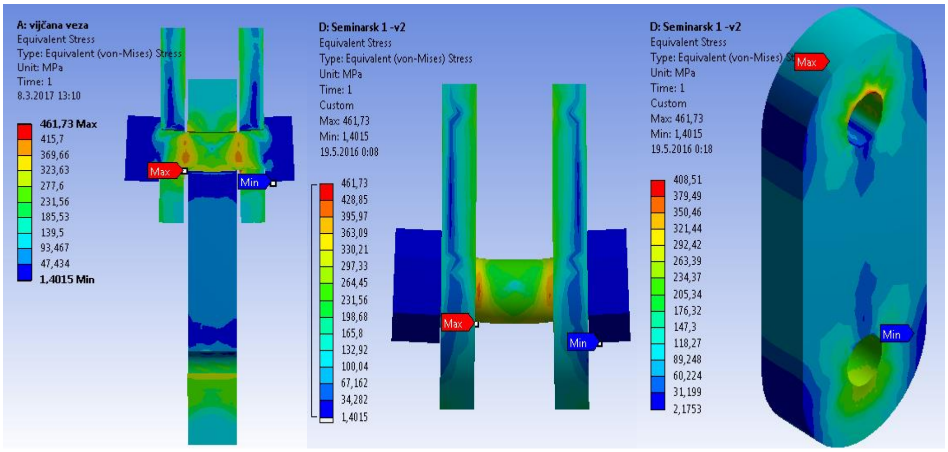

Fig. 5. von-Mises stress: a) whole connection, vertical section, max. value 461.7 MPa (left); b) upper thimble and bolt, max. value 461.7 MPa (middle); c) middle thimble, max. value 408.5 MPa (right)

3.3 Contact analysis

Basic principle of the contact analysis is that two touching bodies under load withstand pressure stresses which are transferred across the contact surface which may take different shapes, depending of the shapes of the bodies in contact. In case of separation of the bodies, the contact disappears, and so do the pressure stresses, while alternatively tension stresses do not occur.

Analysing this connection, one may notice that due to the bending of the bolt asymmetric pressures emerge in the bolt shaft, as well as in the bolt hole edges (*Fig. 6*).

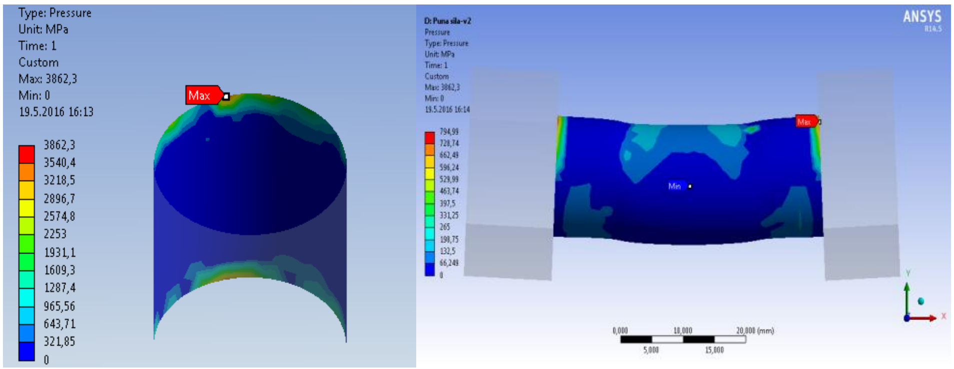

Fig 6. Contact pressure: a) middle thimble, max. value 3862MPa (left); b) bolt, max. pressure value 795MPa (right)

Beside the contact pressure, an important parameter for the insight into the behaviour of the connection provide penetration values. Penetration is the effect of deformation of the bodies in contact, where active body stamps into the passive body. In the analysed connection, penetration is the highest at the contact of the bolt and the outer thimbles, because of the bolt shaft bending and its uneven contact with the thimbles (*Fig. 7.*)

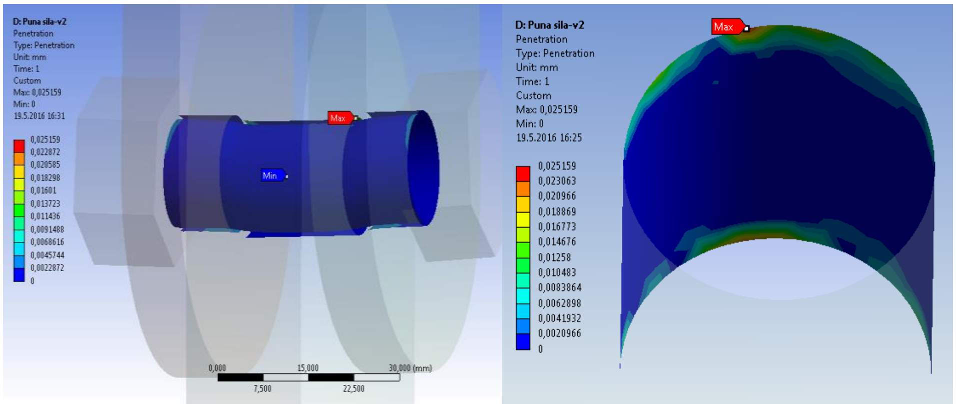

*Fig. 7. Penetration a) whole connection, max. value 0.025 mm (left);
b) detail of the middle thimble, max. value 0.025 mm (right)*

Result of the deformation and penetration in the analysed connection is emerging of the gap between the connected elements. Maximal gap value is in the contact of the outer thimble with the bolt, at the opposite side from the penetration location, as expected (*Fig. 8a*).

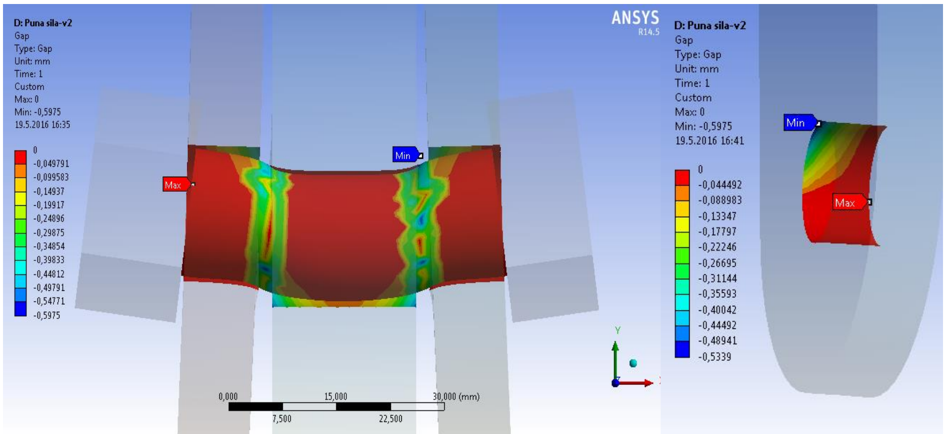

*Fig. 8. Gap a) outer thimbles and bolt, max. value -0.59 mm, scale 8x (left);
b) right thimble, max. value -0.53 mm. (right)*

4. CONCLUSIONS

In the *Table 1* are presented extreme values of the specific results of the FE analysis for two analysed models with different mesh density, which have to serve for verification of the results of the analysis.

Table 1. Characteristic results of the FE analysis vs. mesh density

Result	*Model 1 FE number: 38811*	*Model 2 FE number: 45860*	Δ [%]	*Element*
Total deformation [mm]	1.19	1.23	3.36	Middle thimble
von Mises stress [MPa]	455	461.7	1.47	Outer thimbles
Contact pressure [MPa]	3995	3862	-3.44	Middle thimble
Penetration [mm]	0,024	0.025	4.16	Middle thimble
Gap [mm]	-0.57	-0.59	3.5	Bolt

Based on the results presented one may draw the following conclusions:

- Difference in mesh density between the Model 1 and Model 2 does not give significant differences regarding all analysed results in this case. All differences are under the limit of 5%, which is acceptable for this method of analysis.
- Most part of the connection is exposed to the stress under the elasticity limit, i.e., less than 235 MPa. The exceptions are bolt hole edges and the bolt where local stress concentrations (σ_{max}=461 MPa) overcome the yield point for the steel used, so that plasticization occurs at occasional spots.
- The same bolted connection calculated analytically according to the allowable stress method [3], fulfils all requirements of the relevant standards [SRPS U.E7.145], with remark that a bolt with grade 8.8 was used. Besides that, the classical calculation assumes constant stress distribution in the connection, whereby the local stress concentrations, which are certainly present, are neglected, and they were documented by the numerical model analysis.
- Numerical analysis gives us also an excellent opportunity of considering of other phenomena that emerge in such connections. Maximal contact pressure of 3862 MPa far exceeds even the allowed contact pressure obtained by Herz formulas for supports and bearings (815 MPa, for the steel used). However, such high value is noticed on a very small area (1-2 mm^2), so its influence may be neglected if one takes into account that contact pressure on more than 95% of the total area is well bellow the permitted values.
- Penetration of the elements gives useful information about the real behaviour of the connection, and can help in determination of the critical position of the bolt holes regarding the edge of the base element, more precisely than the traditional recommendations (e.g. e=2d_0). Such information can serve as a tool for potential compacting of the whole connection, especially in case of limited space for connection, e.g., at space truss node joints, etc.
- Gap between the elements is undesirable in a well designed bolted connection, but obviously it is present. Contact analysis can be useful in the way that it may

be used to reduce gaps to a minimum, by proper setting of the difference between the bolt diameter and the bolt hole diameter.

Besides the conclusions given above, conducting of such advanced analysis unavoidably poses some questions about interpreting of the contact analysis results and their codification. To our knowledge, none of the modern codes (including Eurocode) yet treats connections in steel structures in this way. Software skills should not represent an obstacle for a more broad implementation of such methods in structural analysis, which would be our goal in future investigations.

REFERENCES

[1] Ansys 14.5 Help documentation

[2] Buđevac, D., Marković, Z., Bogavac, D., Tošić, D.: *Metalne konstrukcije - osnove proračuna i konstruisanja;* Građevinski fakultet Univerziteta u Beogradu, Beograd **1999**.

[3] Vacev, T., Živković, S., Petrović, M.: *Zbirka rešenih zadataka iz metalnih ,* Građevinsko-arhitektonski fakultet Univerziteta u Nišu, Niš **2015**.

[4] SRPS U.E7.131 – *Ležišta i zglobovi nosećih čeličnih konstrukcija*

[5] Vacev, T., *Optimal solution of an steel space truss joint using nonlinear analysis*, Doctoral thesis, Novi Sad **2009**., pp. 150

[6] Vacev, T., Nešović, I., Živković, S., Ranković, S.: Bolted connection of steel elements: classic solution and contact fem analysis. *Društvo gradjevinskih konstruktera Srbije*, **2016**., ISBN 978-86-7892-839-0, pp. 336-345

МЕТОДА КОНАЧНИХ ЕЛЕМЕНАТА КАО СРЕДСТВО ЗА ДИМЕНЗИОНИСАЊЕ ВИЈАЧНИХ ВЕЗА

Резиме: *Контакт остварен методом коначних елемената (FEA) је моћан алат за анализу и димензионисање различитих спојева. То се посебно односи на челичне конструкције повезане завртњима, где се контактом између омотача вијка и омотача рупе преноси сила смицања. Класична анализа такве везе поједностављује проблем, чиме се смањује сложеност прорачуна. С друге стране, контакт FEA даје много више детаља о понашању везе. Једноставан пример ушица повезаних једним вијком показује предности решења методе FEA моделиране у ANSYS софтверу, али и поставља питања у вези кодификације таквих резултата прорачуна. Контролисана и ограничена пластификација је један од кључних проблема који треба исправно интерпретирати.*

Кључне речи: *челичне конструкције, вијачни спојеви, контакт FEA, ANSYS софтвер*