COMAT

The 9th International Conference on Advanced Composite Materials Engineering

Transilvania University of Brasov
FACULTY OF MECHANICAL ENGINEERING

17-18October 2022

RESEARCH CONCERNING TO THE RIGIDITY OF THE SCREW JOINT BETWEEN BEAM AND COLUMN IN METAL STRUCTURES

Dumbrava F., Cerbu C*, Ursache S.

Department of Mechanical Engineering, Faculty of Mechanical Engineering, Transilvania University of Brasov, B-dul Eroilor, No. 29, Brasov 500036, Romania, florin.dumbrava@unitbv.ro, stefania.olareanu@unitbv.ro

*Corresponding author: cerbu@unitbv.ro

Abstract: *The research involved in this paper, refers to the connection with screws used to assemble the beam and the column in metal structures. In the past, the joints between beams and columns were considered either hinged or rigid. Subsequently, experimental research has shown that these types of joints have been classified as semi-rigid joints. The main purpose of this paper is to compare the experimental results and the results obtained by finite element analysis (FEA) for such a screw joint. Experimental research was made by testing in bending three identical assemblies. The beam-column assemblies tested consist in IPE-type column and IPE beam with flange assemblied with M16 screws of quality group 8.8. Comparison between the experimental results and theoretical results (FEA) in terms of moment-rotation curves, lead to the validation of the numerical model.*

Keywords: *steel structures, storage systems, connection rigidity, finite element analysis.*

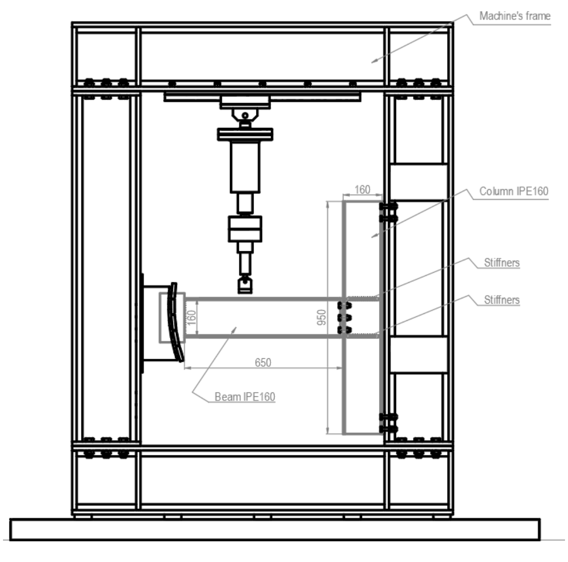
INTRODUCTION

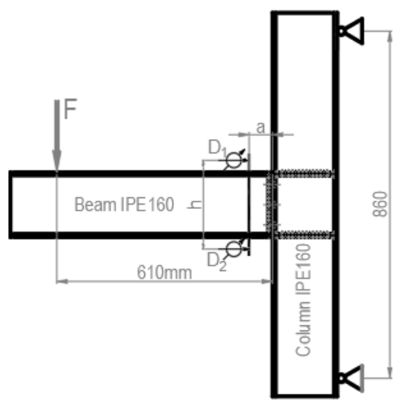
In metal structures for storage and those used in construction, joints between beams and columns are made either by screw assembly through a metal flange welded at the end of the beam, or by assembly with special

connectors. In this paper it is presented the investigations performed for a screw joint used to assemble the beam with the column in metal structures. The connection between the beam and column, investigated in this research, was made by assembly the column with the end plate welded at the end of the beam. The connection was made with screws and nuts.
The structural elements of the metal structures including the connections between columns and beams, are designed and tested according to Eurocode 3 [1-3] and the European standard EN 15512: 2021 [4].
In literature it was reported the results of numerous studies concerning the stability of the structural elements [5] of the steel structures and testing of the beam-to-upright connections [6, 7]. However, there are few works that numerically and experimentally investigate screw connections.
The main purpose of this study is to validate the numerical model of the screw joint used to assemble the beam and column, used in metal structures, by using the results obtained from testing carried-out on such connections.
This study presents two main parts: experimental investigation and finite element analysis. It is mentioned that the experimental analysis was made in the Testing Laboratory from Dexion Storage Solution SRL. For the finite element analysis two different softwares, RFEM and IDEA StatiCa, have been used. RFEM software is dedicated for the finite element analysis and the version 5.20 was used. The IDEA StatiCa 10 software is specialized for simulation of stresses and strains developed in the steel connections mechanically loaded. From experimental point of view, one has investigated the main characteristics of that joint: rotational stiffness, bending moment capacity and plastic deformations.

1. EXPERIMENTAL PROGRAMME

1.1. Test assembly





Legend:
F – vertical force generated by the hydraulic jack;
D_1 and D_2 – displacement transducers;
a = 20 mm – dimension between the column and the location of the displacement transducers;
h = 200 mm – distance between the displacement transducers.

(a) (b)

Figure 1: Experimental set-up of the connection test in bending test: (a) sketch of the equipment used for bending tests of the beam-column connections; (b) loading scheme of the beam-column connection in bending test.

During the experimental program it has tested in bending the screw joints between the beam and column. The connections investigated were made by assemble the column with the plate welded to the beam end by using six M16 screws having the quality class 8.8. The screws were disposed on two columns and there were three screws per each column. Three identical assemblies were tested. Beam and column used in each assembly are made from the same hot rolled profile named IPE160 made of S275 steel. End plate whose thickness is 12 mm was made of S275 steel S235. The column presents also steel stiffeners made of S235 steel and thickness of 8 mm.
Each beam-column assembly was tested by using the experimental set-up shown in Figure 1. The force was applied statically in the bending test of the assembly consisting in beam, column and screw joint.
In Figure 2 it is shown the dimensions and shapes for the cross sections of the beam and column involved and also the dimensions for the end plate and for the stiffeners used.

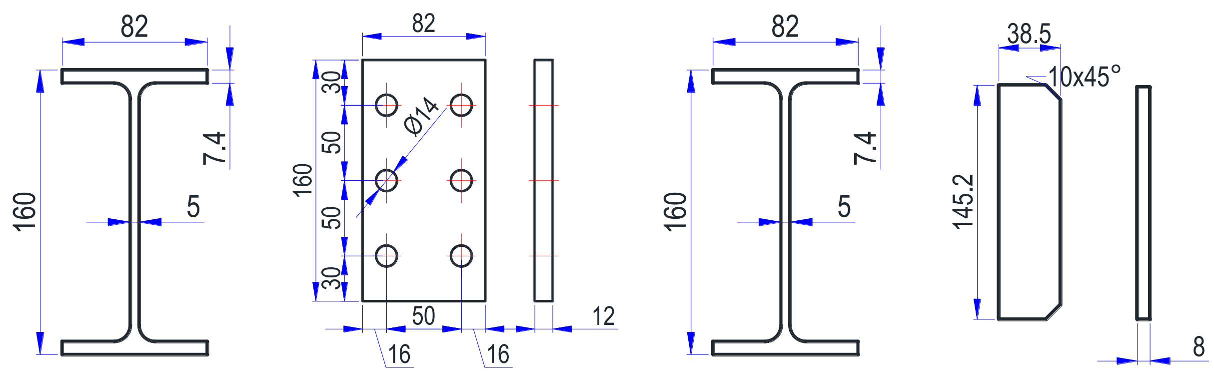


Figure 2: Dimensions of the elements of the assembly tested: (a) beam cross section; (b) end plate; (c) column cross section; (d) stiffeners.

1.2. Bending test and corrections

In bending test of the screw joint, one force transducer was used to evaluate the vertical force that was applied on the beam at distance of 600 mm with respect to the column. The force was applied in small steps till the failure had occured. In order to compute the rotation of the screw joint considering the experimental results, two displacement transducers were used (Fig. 1b). After each bending test the moment-rotation curve was plotted for each assembly tested.
All the results obtained from tests were corrected for thickness and yield strength. The corrections were made according to the European standard EN 15512: 2021 [4].

1.3. Testing of the materials

Table 1. Tensile test results

Element type	$f_{y,n}$ (MPa)	$f_{y,t}$ (MPa)	$f_{u,n}$ (MPa)	$f_{u,t}$ (MPa)	Elongation (%)	Young's Modulus (MPa)
Beam	275	336.8	430	475.2	39.7	210000
End plate	235	276.1	360	350.3	42.5	210000
Column	275	336.8	430	475.2	39.7	210000

Stiffeners	235	276.1	360	350.3	42.5	210000

Tensile tests were conducted for the materials of the elements (beam, end plate, column, stiffners) used in the experimental program in order to obtain the real yield strength, ultimate strength and elongation. In this manner, the material modeling is accuracy made for the numerical model used in FEA. In Table 1 it is shown the nominal values and the real values obtained by tensile tests.

2. FINITE ELEMENT ANALYSIS

The nonlinear analysis is used in numerical simulation by finite element method concerning the elasto-plastic behavior of the steel. A bilinear stress-strain curve is considered in FEA (Figure 3).

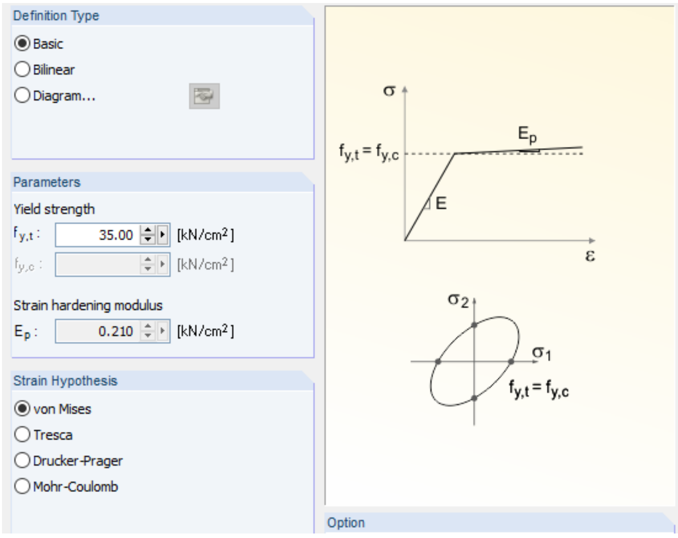


Figure 3: Strain – tensile curve used for modeling of the elasto-plastic behavior of steel

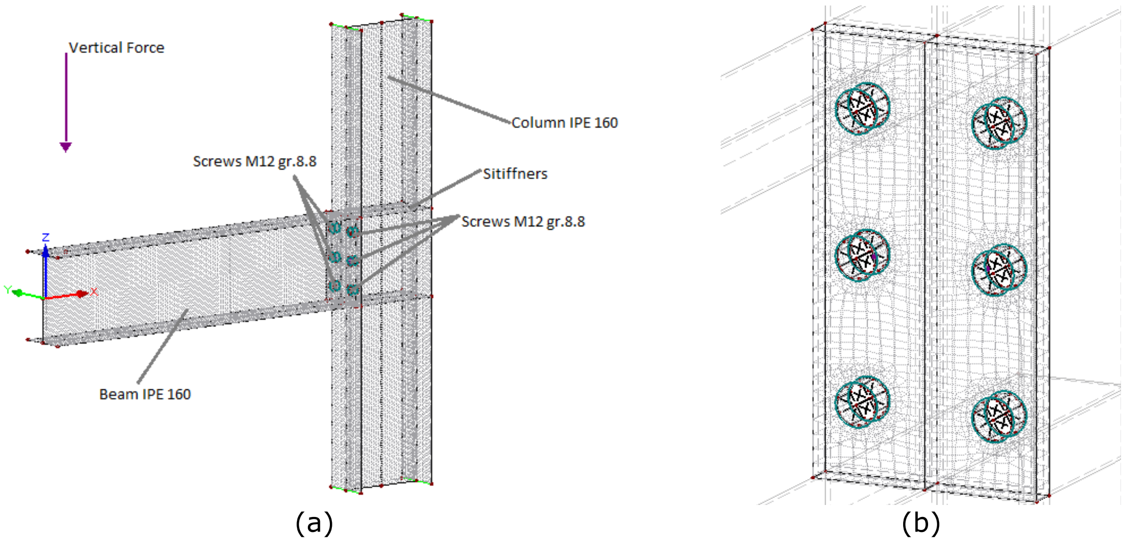
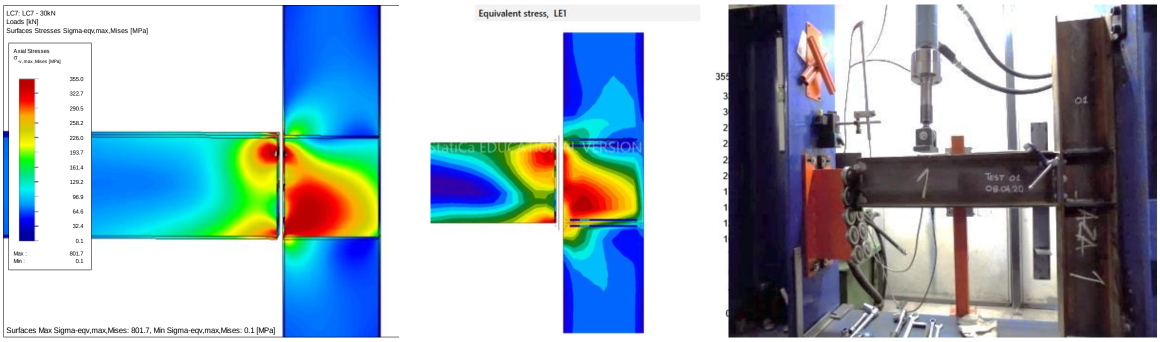


Figure 4: Finite element model analyzed: (a) general meshing of the model; (b) detail on meshing of the hole areas.

To simplify the numerical model shell elements were used to model the structural elements assembled (Figure 4). Shell elements with three and four nodes were used for meshing, having six degrees of freedom (DOF). The connection between the beam end plate and the top flange of the column was made with a solid contact element which is able to work just on compression in perpendicular direction. In parallel direction a friction coefficient μ=0.1 was considered to model the interaction. The numerical model of the beam-column assembly investigated is presented in Figure 4(a). A member element with a round bar section of 12 mm is considered in order to simulate the mechanically behavior of the connection with screws. To obtain a smooth stress distribution on the edges of the holes, eight rigid elements were modeled for every hole as shown in Figure 4(b).

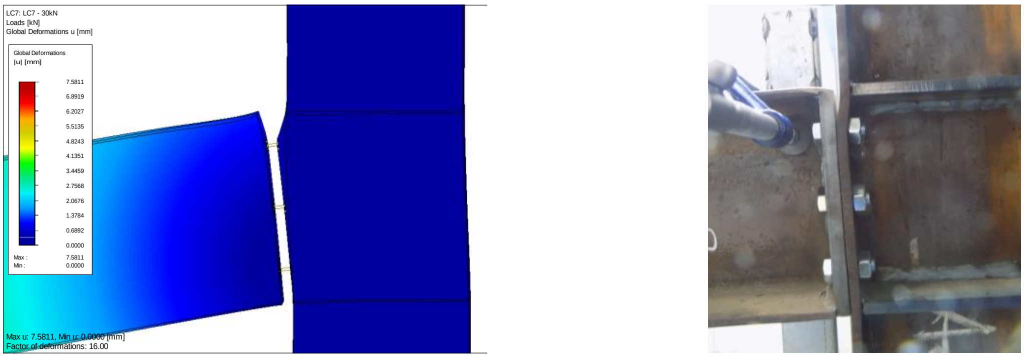
3. RESULTS

From numerical analysis it was noticed that the failure mode occurred was bending of the end plate together with the top flange of the column, but also the tension stresses develop in the screws located on the first row. In Figure 5 it is plotted the stress fields obtained by FEA analysis conducted by using RFEM and IDEA StatiCa software, respectively. In the same manner the distribution of the global displacement is shown in Figure 6(a). Looking also regarding the failure modes from experiments (Figure 5c and Figure 6b), it may observe that the numerical model shows similar failure mode.



(a) (b) (c)

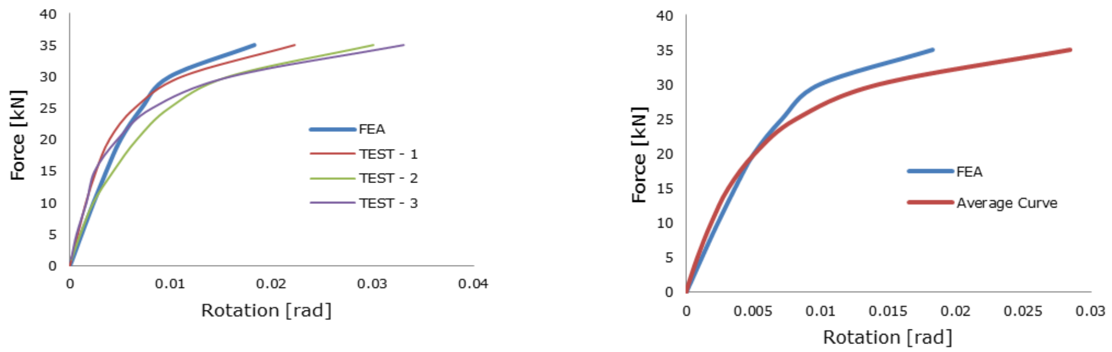
Figure 5: Stress fields obtained by numerical simulation versus failure mode in bending test: (a) stress field by RFEM analysis; (b) stress field by IDEA StatiCa analysis; (c) failure mode.



(a) (b)

Figure 6: Displacements obtained by numerical simulation versus failure mode in bending: (a) displacement obtained by RFEM; (b) detail of the failure area.

In Figure 7, it is comparatively analyzed the force-rotation curves recorded in bending tests with the force-rotation curve obtained by FEA. The matching of those force-rotation curves proves that the finite element model is validated by the experimental results and the numerical model may be used in further investigations of this type of screw joint.



(a) (b)

Figure 7: Comparison between the experimental and FEA results: (a) comparison between all experimental force-rotation curves and the curve obtained by FEA; (b) comparison between force-displacement curve obtained by FEA and average experimental curve.

The difference between experiment and numerical model can be explained because the numerical model used has been simplified and some effects of the assembly with screws (looseness, torque moment) were not considered in FEA. Also, the numerical model considered does not contain any geometrical imperfection.

4.CONCLUSIONS

The rotational stiffness obtained by using the first portion of the force-rotation curves obtained by bending test matches with the one obtained by FEA for all assemblies tested. The differences between the experimental and numerical analysis are recorded in the plastic field of the moment-rotation curve. Nevertheless, a nonlinear analysis is mandatory for a reliable finite element analysis.
For all type of analysis, the failure mode is the same in the areas subjected to tensile stresses of the elements (end plate and top flange of the column).
The numerical simulation shows that plastic deformations have occured.

BIBLIOGRAFIE

[1] BS EN 1993-1-3: 2006, Eurocode 3: Design of steel structures: Part 1-3: General rules. Supplementary rules for cold-formed members and sheeting. In Design of joints, European Committe for Standardization: Brussels, 2006; p 12.

[2] SR EN 1993-1-8:2006 - Eurocod 3: Proiectarea structurilor de oţel. Partea 1-8: Proiectarea îmbinărilor.

[3] SR EN 1993-1-8:2006/NB:2008 - Eurocod 3: Proiectarea structurilor de oţel. Partea 1-8: Proiectarea îmbinărilor. Anexă naţională.

[4] EN 15512: 2021. Steel static storage systems - Adjustable pallet racking systems - Principles for structural design. European Committe for Standardization.

[5] Dubină D., Ungureanu V., Zaharia R, Dogariu A., Crisan A, Tuca I., Neagu C., Verificarea la stabilitate a elementelor din oțel în conformitate cu SR EN 1993-1.1. Recomandări de calcul, comentarii și exemple de aplicare. Timișoara 2010.

[6] Dubină D., Contract nr. 426/08.12.2009. Calculul si proiectarea îmbinărilor structurale din oțel în conformitate cu SR-EN 1993-1-8. Recomandări, Comentarii si Exemple de Aplicare, no. 426. 2010.
[7] Dumbrava F., Cerbu C., Experimental Study on the Stiffness of Steel Beam-to-Upright Connections for Storage Racking Systems. Materials, Vol. 13(13), Article 2949. 2020.