CZASOPISMO INŻYNIERII LĄDOWEJ, ŚRODOWISKA I ARCHITEKTURY
JOURNAL OF CIVIL ENGINEERING, ENVIRONMENT AND ARCHITECTURE
JCEEA, t. XXXV, z. 65 (2/18), kwiecień-czerwiec 2018, s. 219-232, DOI:10.7862/rb.2018.37

Krzysztof OSTROWSKI
Aleksander KOZŁOWSKI

CREDIBILITY OF FEM ANALYSIS IN THE T-STUB MODELLING

The paper presents the results of a comparative analysis between numerical calculations of T-stubs of the 3rd stage of FEM models hierarchical validation and the results of laboratory tests. The procedure for the development of the material characteristics used in numerical calculations of FEM models is presented. The scope of this article allows determining the non-linear characteristics of the T-stub which maps the work of the end-plate joint of the beam to the column in the tensile zone. The results of laboratory tests of a series of T-stubs made of rolled profiles (HEB240, HEA240) and of welded profiles (thickness of end-plate: t_p = 12 mm and t_p = 20 mm) have been presented. The principles of shaping the geometrical features of the FEM model of end-plate joints of the T-stub type are given, with particular emphasis on the shaping of the bolt with a thread. The impact of the bolt thread on the accuracy of the obtained results was assessed. The criterion of reliability of the obtained results with respect to the maximum force in the bolt obtained on the basis of laboratory tests in the axial tensile test of the bolts in the configuration: bolt - washer - nut was formulated.

Keywords: T-stub, rotation capacity, material ductility, multistage hierarchical validation, FEM modelling

1. Introduction

For a relatively long time, traditional steel connections have been considered as a fully rigid or ideally hinged, regardless of their actual behavior. This assumption considerably simplified the calculation process and the expense of unoptimized projects and higher production costs borne to produce construction elements [1], [2]. Principles for the assessment of the structural elements behavior have well-established methodology, allowing to determine all the instability effects and to assess the safety of local systems as well as the whole structural system [3]. In the case of joints, a similar level of knowledge and the applied methodology is not available, in particular in the area of response surface prediction of rotation angle defined in the form of the joint rotation curve M-ϕ.

[1] Corresponding author: Krzysztof Ostrowski, MTA Engineering Ltd., ul. Poniatowskiego 14, 35-026 Rzeszów, krzysztof.ostrowski@mta-online.net
[2] Aleksander Kozłowski, Rzeszow University of Technology, ul. Poznańska 2, kozlowsk@prz.edu.pl

The behavior of steel structure joints is very complicated and requires consideration of many occurring phenomena including material nonlinearities, contact surface nonlinearities, local geometrical imperfections, as well as complex configurations of joints geometry. The difficulties we encounter in creating analytical models describing the joints' behavior in the full scope of their deformability is caused by the compilation of factors having their basis in the phase shift of the plastic deformations initiation of individual joint components. There is an equilibrium path for each element that is part of the joint, which has its own non-linear force-strain characteristics (F-Δ) [3], [4]. This relationship causes that the behavior of the joint subjected to load in the form of bending moment is also characterized by non-linear behavior.

This non-linearity occurs because a joint is a collection of several components which interact differently at different levels of applied loads. Each non-linearity regulates the behavior of the joint and is at the same time an obstacle to the systematic and theoretical solution to this problem [5]. Analysis of this complex behavior has a usually approximate character with the use of drastic simplifications. The tests (both laboratory and numerical) are often carried out in order to obtain an actual answer, which is then modeled by approximating the solution by means of mathematical formulas, having reference to the main properties of the joint structure.

During the last decades, different approaches have been applied in the area of assessment of the steel joints behavior. Extensive literature studies present a well-developed methodology of experimental research [6], [7], [8] and developed empirical [9], [10], analytical [11, 12], numerical [13], [14], [15], [16], [17] and mechanical models [18]. After the introduction of semi-rigid connections concept, many researchers have focused their efforts on accurately predicting parameters such as initial stiffness ($S_{j,ini}$), moment resistance ($M_{j,Rd}$) and rotation capacity of joints (ϕ_{cu}), to obtain the actual response surface of rotation angle of joint M-ϕ. Numerous research works dedicated to estimate the strength and stiffness of T-stubs connections (Zoetemeijer [19], Yee and Melchers [20]) contributed to the creation of a new trend in the analysis of the behavior of the bolted connections, where a particular example of this approach is the so-called component method, which was actually adopted as a calculation procedure in well-known regulatory standards, such as Eurocode 3 [5, 21]. Component method included in PN-EN 1993-1-8 [22] is used to determine the moment resistance and stiffness of joints. The wide application of the component method in the design of joints was possible due to the development of mechanical and spring models, supporting the development of analytical and empirical models. The basic principle of these models is to divide the connection into simpler components for which the moment-rotation relation (M-ϕ) of the whole bolted connection is determined by assembling all individual responses of its components in a spring system.

However, the procedures of the component method do not precisely define the rules for determining the rotation capacity of joints in the full range of deformability. In a relatively large number of laboratory tests conducted as part of the research work on the behavior of joints, the maximum rotation angle (ϕ_{cu}) was not focused on. In these tests, the initial zone of deformation of the joint was analyzed, which was associated with the determination of its initial stiffness $S_{j,ini}$. The second determined test parameter was the limit resistance of joint M_R.

In addition to many advantages, the basic disadvantage of laboratory tests is the time and cost of their execution. For this reason, the use of numerical analyzes to simulate the behavior of joints, becomes a routine activity in research processes. The FEM analyzes reduce research costs, provide much more information about the state of strain and stress of the tested objects, which is not possible to achieve in such a wide range by performing traditional destructive laboratory tests. It should be noted that in case of using numerical analyzes, the results of such analysis may be subjected to a relatively large error in the absence of proper verification and validation of numerical models. In the article, the authors presented the extended results of 3rd stage of multistage hierarchical validation of FEM models, as a continuation of the validation process of FEM models, for the needs of rotation capacity prediction of joints [23].

2. The necessity of validation for the needs of the correct FEM modeling

The development of computer technology, whose dynamic growth has been recorded since the 80s of the last century, allowed to develop computational methods using the finite element method to the level where the complicated effects of the examined objects can be calculated on PCs. Previously, it was possible to do only in computing centers. It is assumed that the results obtained in numerical analyses using the finite element method can be considered reliable if they are comparable with the results of experimental research or other known precise solutions.

In each FEM analysis, the accuracy of the model is evaluated. Relevant regulations including the formalization of validation and verification procedures were developed by the American National Institute of Standards [24]. The evaluation of the accuracy of the FEM model should precede every more serious FEM analysis [25]. The verification process is an evaluation of the accuracy of the solution in the FEM calculation model compared to known solutions, e.g. analytical solutions. In the validation process, the computational accuracy of simulated solutions is evaluated by comparison with experimental results. The validation should be performed gradually, i.e. at the level of the material model, set of fasteners, subassemblies and structure fragments. In the literature it has been called as a hierarchical validation [24]. The validation is an iterative process, and the final result in the form of proper material characteristics and a calculation model is a set of requirements that should be met in a computational model that maps the analyzed real model (Fig. 1). The validation must evaluate the

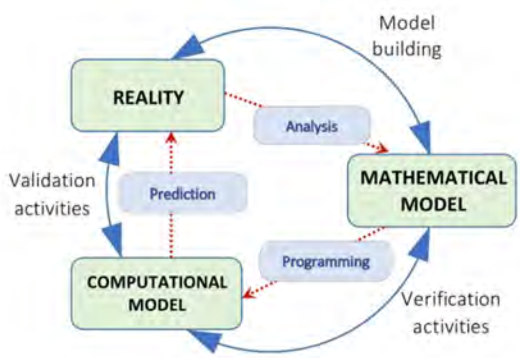

Fig. 1. Diagram of the verification and validation interaction circle in the numerical modeling process (on the basis of [24])

predictive ability of the FEM model in its physical aspect and should take into account any uncertainties that arise from both the results of the numerical simulation and the experimental data.

The hierarchical validation of FEM models used in the analysis of the joints behavior in the full range of deformations was carried out as part of the work [23], [26] in the following four stages:

I Stage – tensile test of steel and bolt specimens (Fig. 2a),
II Stage – bolt tensile in the configuration: bolt – washer – nut (Fig. 2b),
III Stage – tensile test of the T-stubs (Fig. 2c),
IV Stage – test beam to column connection in the configuration of frame (Fig. 2d).

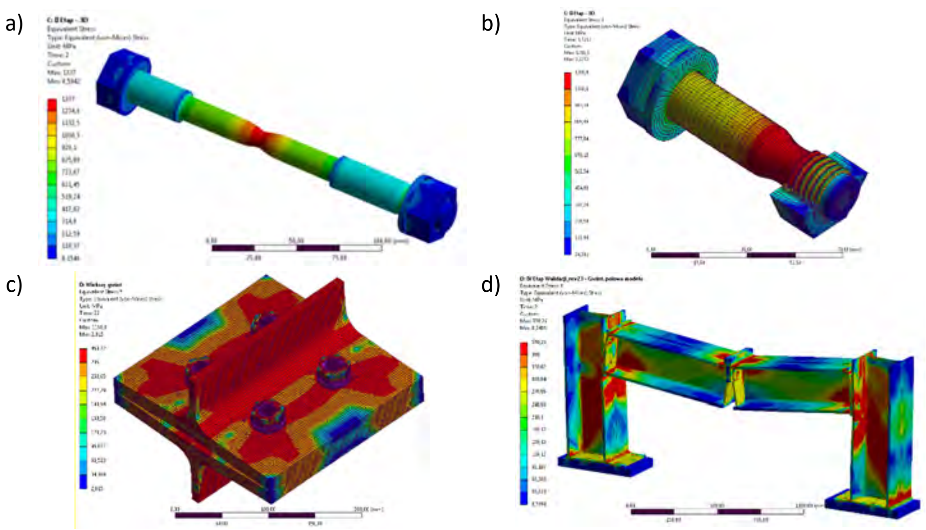

Fig. 2. Multistage Hierarchical Validation of FEM models: a) I stage: the tests of material of steel samples: b) II stage: the tests of fasteners in the configuration of bolt-washer-nut; c) III stage: the tests of T-stubs; d) IV stage: the joint tests in the configuration of frame

In Fig. 3 FEM models used in the process of hierarchical validation were presented. Introduced 4-stage process of model adjustment in the scope of material characteristics describing the properties of the used materials and geometrical features assumes the analysis of the models with different levels of complexity. Such an attitude enables to obtain the required level of detailing which is needed to obtain results convergent with the results of laboratory tests. Material characteristics which do not have logarithmic strain are a kind of reliability guarantor of optimized object due to the fact, that in such cases the analysis always proves earlier achievement of the tolerable stress and strain state. The only characteristic which complies with the requirements for accurate strain mapping of the tested object is the stress - strain characteristic referring to cross section momentary areas, which are real stresses in the deformed section. The adjustment process is obtained as a result of modification of the curve σ–ε to such a form by which the acceptable compatibility between results and laboratory tests is achieved.

$$\sigma_{true} = \sigma(1+\varepsilon) \tag{1}$$

$$\varepsilon_{true} = \ln(1+\varepsilon) \tag{2}$$

The area for which the characteristic stress – strain is known is determined by the formula (1) and (2). The modification of the curve can be made only in the unknown scope of material behavior, that is from the moment of creating the necking in the tested material sample, for which it is impossible to determine the stress - strain relation based on the analytical relations available in the literature. The value of the maximal stress σ_u is determined on the basis of the force value in the tensile test before failure referred to deformed cross-section area of the sample A after failure. The maximal value of the strain ε_u corresponding to maximal stress σ_u is determined in iterative manner by increasing deformation ε_u to such values at which the best adjustment of the actual response curve σ-ε is obtained.

3. Laboratory tests of T-stubs

Research program of the 3rd validation stage, included the tensile test of 12 T-stub connections. The study included performing the tests of 4 series T-stub connections with division into rolled and welded profiles.

T-stubs from rolled profiles:
– series H01 – T-stub of profile HEA 240, steel: S235 – 3 samples,
– series B01 – T-stub of profile HEB 240, steel: S355 – 3 samples.

Welded T-stubs:
– series SP01 – welded T-stub: end-plate 20 mm, steel S355 – 3 samples,
– series SP02 – welded T-stub: end-plate 12 mm, steel S235 – 3 samples.

The range of the tested T-stubs was constructed in such a way that, in the tested models, we obtain the 1st and 2nd failure mode according to the classification included in the standard [5]. In numerical models of tested objects, geometry projection has been made based on exact measurement of the elements subjected to tensile test. During the sample measurement, significant geometrical imperfections of the profiles have been found. Deviations dispersion in thickness of the flanges for HEB 240 (series B01) was in the range 16.35÷17.84 mm and it was the highest from all tested series. After measuring the fasteners sets, some dimension deviations with respect to nominal dimensions were also noted (ISO 4014). These deviations were introduced to the FEM model.

In sample A, series H01, a strain gauge system was used to measure strains at predefined characteristic points. The location of the strain gauges is shown in Fig. 3c. In order to measure the strains in the bolts, a system of strain gauges arranged on the periphery of the bolt shank was used in a radial system with a 120° offset (Fig. 4a). All tested samples were attached in an auxiliary holder,

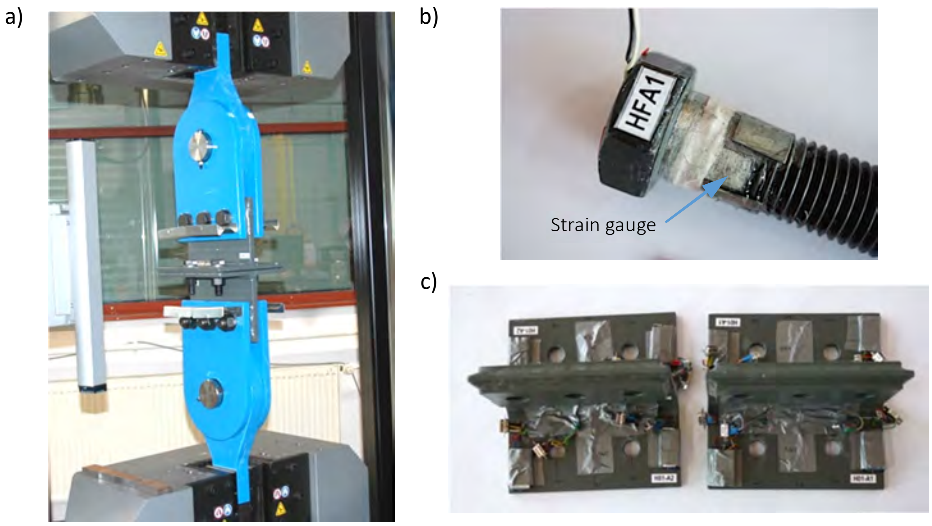

Fig. 3. Sample H01: a) sample of series H01 in a testing machine after damage; b) location of strain gauge in the bolt; c) location of strain gauge in the sample of series H01

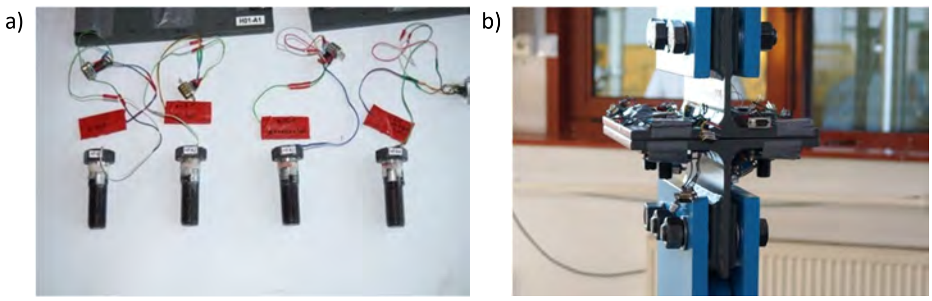

Fig. 4. Measuring system: a) set of fasteners with a strain gauge system; b) sample of series H01 before damage

which ensured the axial introduction of force in the test sample (Fig. 3a). Measurement of deformations of the examined joints was made by means of an optical extensometer, measuring the distance increment between 2 points applied to the side surface of the centre of the tested samples.

The distance between the points was about 100 mm. In order to eliminate the clearances in contact, the prestressing of the bolts with a force of F = 50 kN has been introduced (Fig. 6c). Due to geometrical imperfection in tested samples in each series the differences in the response curve F-Δ were noticed (Fig. 5a and Fig. 5b).

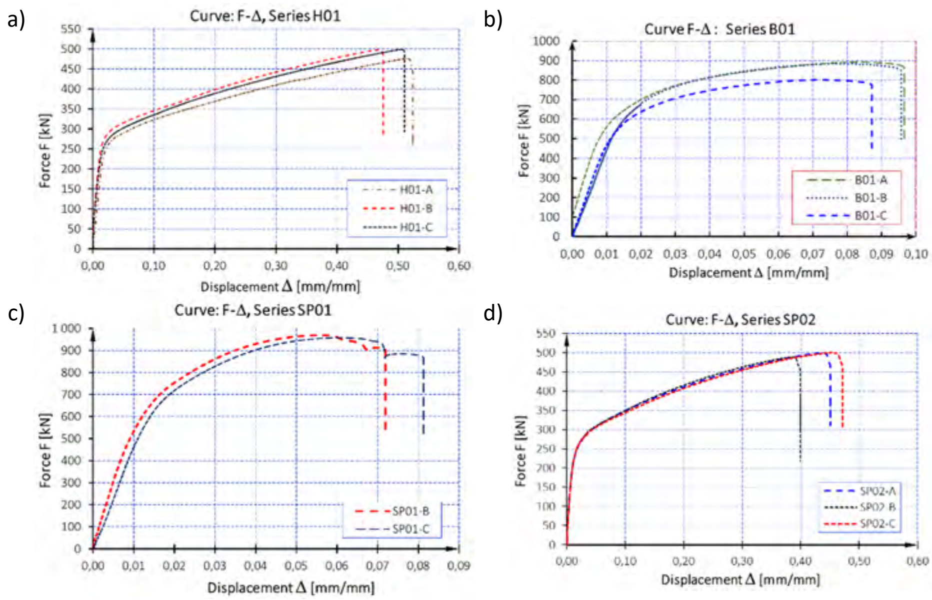

Fig. 5. The results of tensile test: a) series H01; b) series B01; c) series SP01; d) series SP02

4. Description of the numerical model

For the creation of the numerical model 3-dimensional finite elements type Tet10, Hex20, Pyr5, Hex8 and Wed6 were used (Fig 6a). The multi-linear material model defined on the basis of the 1st scope of hierarchical validation (Fig. 7a and Fig. 7b) was used. The contacts between particular elements of joint were created as nonlinear with the friction factor assumed for the surface in a natural condition with the value of $\mu=0.2$. A reduction in contact stiffness has been introduced with each subsequent iteration. For all contacts in the model, an augmented Lagrange contact formulation was applied [27]. Contact surfaces were introduced in the areas of contact between: end plates, washer – end-plate, washer – nut and washer – head of the bolt. Additionally, radial surface contacts between bolt hole and bolt shank and thread were introduced in the model (Fig. 9a).

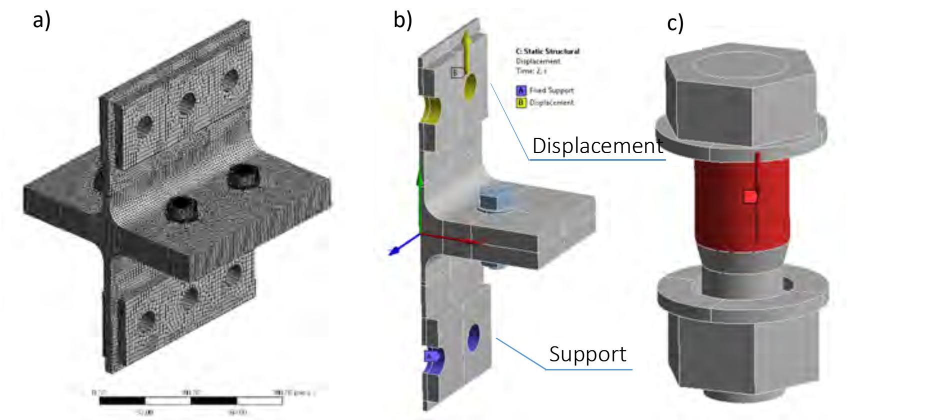

Fig. 6. FEM model: a) 3D view (meshing); b) model 3D of T-stub in double symmetry; c) location of a prestressing force application

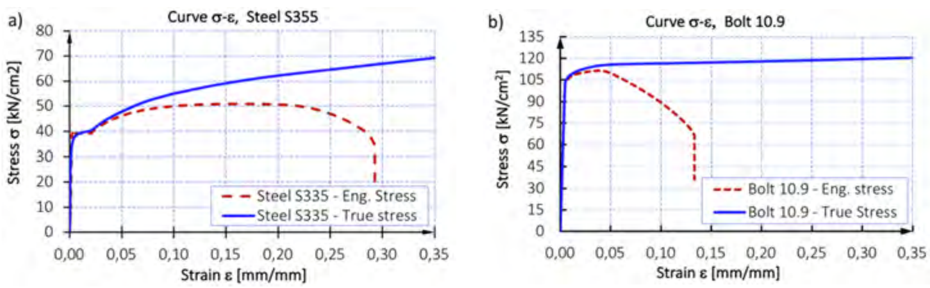

Fig. 7. Material characteristic: a) steel S355; b) bolt grade 10.9 (ISO 4014)

A specific type of contact surface which has been introduced into the FEM model is shown in Fig. 9c. The last from mentioned contacts create boundary conditions to support the bolt that rests on the inner surface of the plate's hole caused by a large joint gap. In the initial stage of joint deformation, the side surface of the washer and the upper surface of the plate have no contact, but with a large gap in the joint they interact between each other, and plate surface is the support for the lateral surface of washer. Introducing such contact surface is dictated by possible occurrence of non-coinciding the FEM model caused by penetrating objects.

In the model of washer, three layers of finite elements and the division into 48 elements were introduced. In the area where the thread connects to the nut, mesh density was increased to the size of 1 mm. In end–plate five layers of finite elements were introduced. The corresponding density of the mesh in this area greatly helps to achieve convergence of the FEM model. In summary, a well-elaborated meshing model is a necessary condition to obtain the correct deformation of particular elements of the analyzed joint.

In order to increase the calculation efficiency, a double symmetry was introduced to the computational model with respect to the center planes of the system, as illustrated in Fig. 6b. In order to evaluate the impact of the thread in the bolt 2 computational models were prepared. In the first model the bolts with modeled metric thread were used (Fig. 8a), in the second one the bolt without thread was used. In this model the nut was permanently connected with the bolt shank (Fig. 8b).

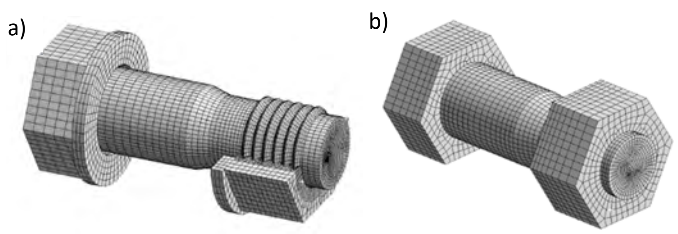

Fig. 8. FEM model of bolt: a) Bolt with thread; b) Bolt without thread

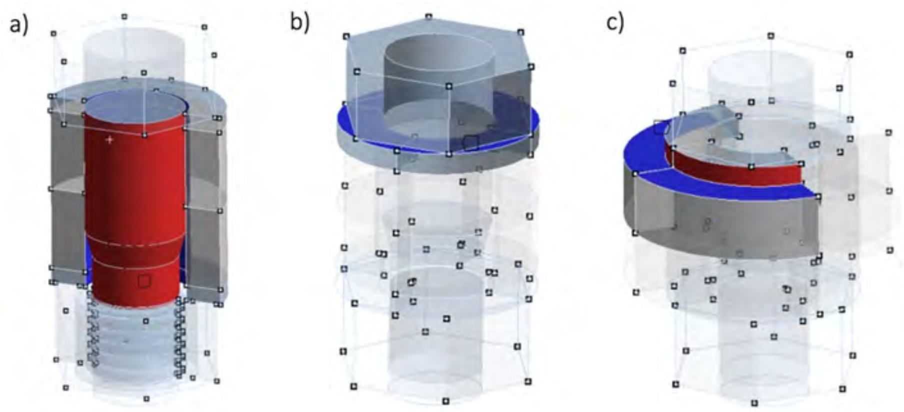

Fig. 9. Contacts: a) radial contact: bolt – end-plate b) anti-slip contact: bolt – washer; c) contact between end plate and external surface of washer

The FEM models were loaded in the same method, as was the case in laboratory tests, by introducing a displacement load. In models with thin end-plates (series: H01 and SP02) the load was applied in 150 steps. In the B01 and SP01 series, the load was applied in 50 steps. Locations of the applied loading are shown in Fig. 6b. The numerical calculations of the analysed objects were performed using the material characteristics shown in Fig. 7.

5. Results comparison and summary

The T-stubs research program presented in the paper, used for the needs of the 3rd stage of multi-stage hierarchical validation of FEM models, provides a wide range of information needed to geometric discretization of analysed objects in numerical analysis. Available tools in Ansys software, allow to obtain many interesting results that are only obtainable in the case of advanced numerical analysis performing. If during the creation of the numerical model in a sufficiently accurate manner the geometrical and material imperfections are mapped, then as a result we can obtain not only the deformation state of the tested object (Fig. 10), but also a reliable result of the force distribution (Fig. 11) and distribution of stresses (Fig. 12) in the analysed objects.

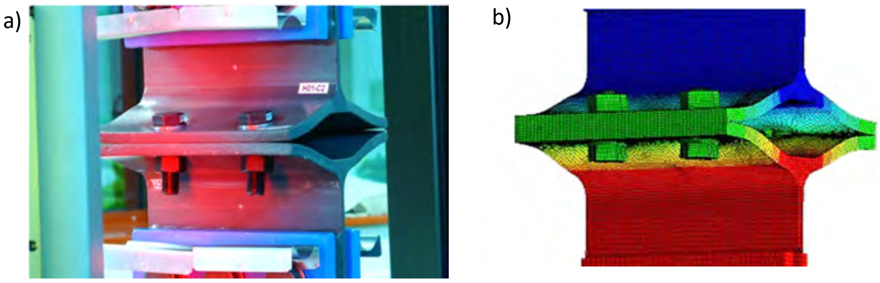

Fig. 10. Comparative analysis: a) laboratory test: sample H01 – type C; b) FEM analysis – sample series H01

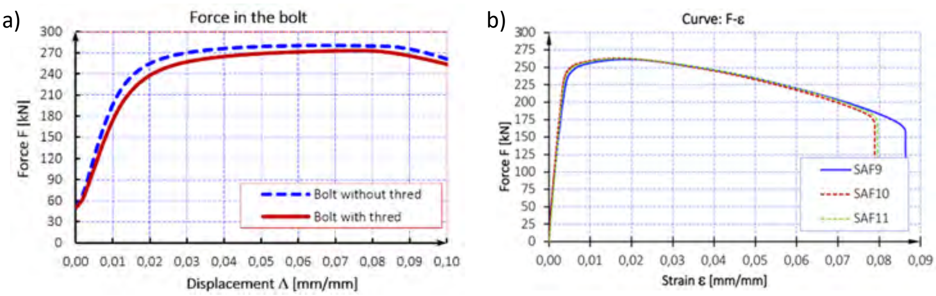

Fig. 11. Distribution of forces in the bolt: a) response curve F-Δ - T-stub connection: series B01, HEB 240, steel grade: S355, the forces in the bolt with thread and without thread; b) response curve F-ε – 2nd stage of hierarchical validation

One of the most important results obtained in the analysis is the value of the maximum force in the bolt, which is comparable to the value of force in the bolt obtained in the laboratory test in the 2nd stage of validation [23]. The maximum bolt force in the laboratory test was $F_{bolt,lab}$ = 263.18 kN (Fig. 10b – SAF10) and it is most approximate to the results obtained in numerical analysis in a bolt in set with inserted thread (Fig. 8a). The results presented in Fig. 8a and Fig. 8b indicate that bolt without thread achieves an earlier deformation achievement, at which the bolt achieves the maximum value of force. The value of the force in the bolt obtained in the set with the thread is smaller in relation to the value of the force in the bolt in the set without a thread and is F_{thread} = 273.11 kN.

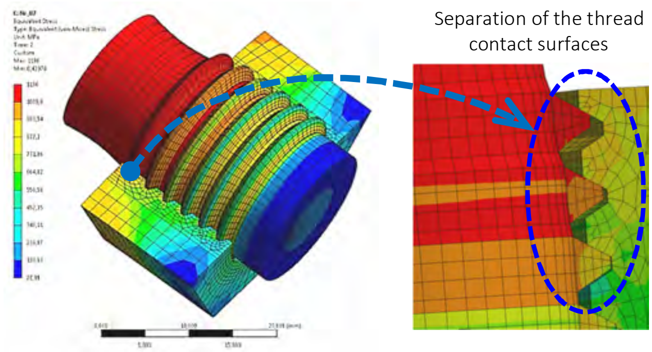

Fig. 12. Separation of the thread contact surfaces between the bolt and the nut

In the analyzed connection it was observed characteristic effect of the increase in the effective length of the thread. After reaching the plastic stress, a neck is formed in the area of the thread, which with its range reaches up to one-third the height of the nut (Fig. 12). This effect will not occur in the case of bolt modeling, in which the bolt is connected to the nut (Fig. 8b). Introduced to the FEM analysis model of the bolt with thread is dedicated to case studies where there is the need for accurate assessment of the forces distribution in the analyzed model.

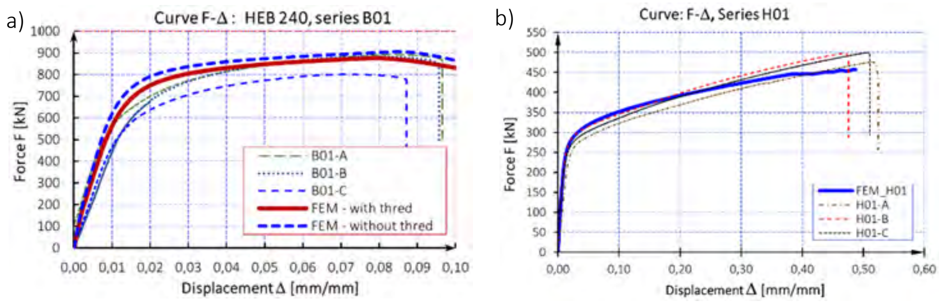

Fig. 13. The results comparison of the numerical analysis with the results of laboratory tests:
a) Sample of series B01; b) Sample of series H01

Relationship of loading F and relative change in the distance between reference points are shown in Fig. 13.

Determination of the force distribution in the bolt in laboratory conditions of the separate structure elements testing (joints, frames etc.) is quite troublesome and possible only in a certain range of stress and strain. The available research instruments are well suited for elastic stress and leave a free, unexplored space for analysis of the force distribution in the bolt in range of plastic deformation.

The analysis of end-plate bolted connections deformations defined in the T-stub form is usually performed assuming the possibility of occurrence of three plastic hinges (two in the front panel: the first in the bolt area, the second in the connection of and-plate with the web panel, the third plastic hinge in the bolt).

In the case under study (Fig. 13b), a fourth plastic hinge is formed in the web panel. Fig. 14 shows the progress development of plastic zones for the five load phases defined in the form of displacement. In the first phase, the plastic zones have developed in the face of end-plate (Fig. 14a), in the 2nd phase the plastic zones start to develop in the region of the web panel in the bolt axis (Fig. 14b), in the next phase the plastic zones (in the bolt axis) merge into one the compact zone and this state is maintained until the end of the test (Fig. 14c, Fig. 14d, Fig. 14e).

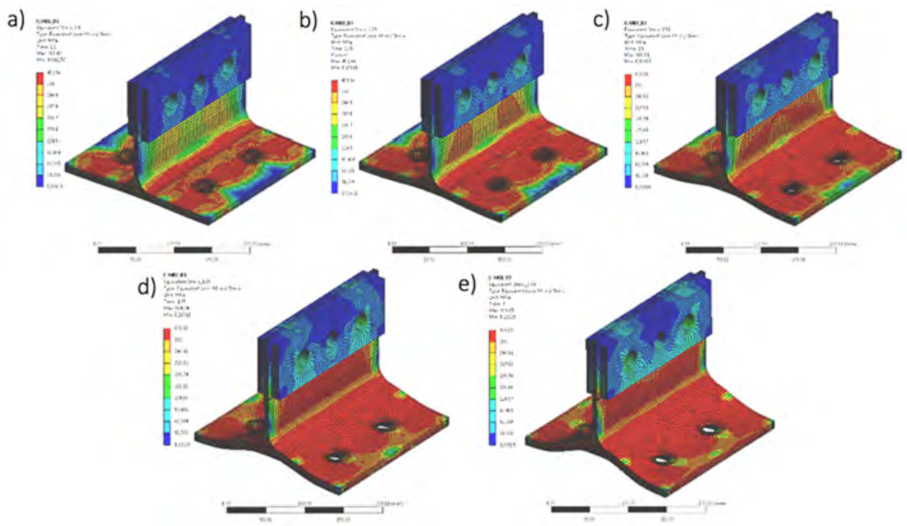

Fig. 14. Development of plastic zones: a) displacement Δ = 5 mm; b) displacement Δ = 12.5 mm; c) displacement Δ = 25 mm; d) displacement Δ = 37.5 mm; a) displacement Δ = 50 mm

This irregularity in the behaviour of the web panel is caused by the membrane effect which results from the low stiffness of the end-plate. The confirmation of this phenomenon is the distribution of von Mises stresses depicted in Fig. 15, along a path located in the bolt axis on the upper surface of the end-plate. There

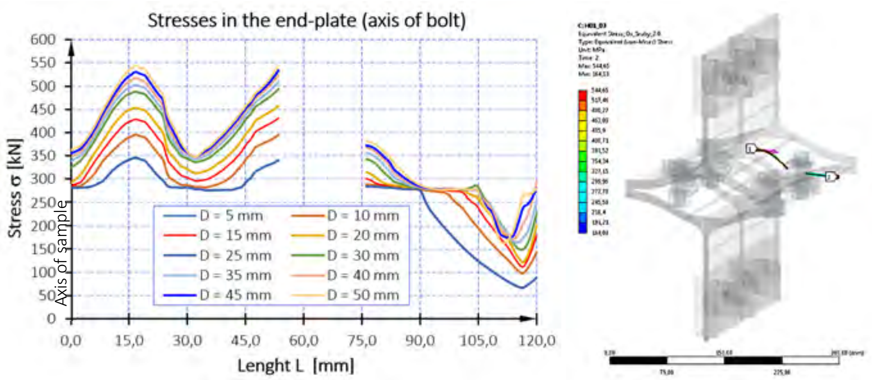

Fig. 15. Distribution of stresses in the end-plate along a path located in the axis of the bolt

was a significant decrease of stresses in relation to stress at the point of the bolt fixing and at the point where the end-plate connects to the web panel. This effect of the variable redistribution of deformations is a particular obstacle to the regular analytical description of the connection behaviour in the case where the web panel is the weakest link in the chain.

Conducting a multistage hierarchical validation is a prerequisite for obtaining reliable results of the FEM analysis of the examined objects. This is particularly important in relation to the structure and its parts subjected to large deformations. Introduction of material characteristics as a result of the process of tuning the FEM models implemented as part of the validation allows to analyze the behavior of joints subjected to significant deformations, close to the state of exhaustion of the structural capacity for load transfer.

Finite element method is an alternative approach in the predicting behavior of the T-stub type connections. Based on the results obtained in the numerical calculations, we encounter relatively large difficulties in formulating the analytical relationship describing the state of the joint's behavior, similarly in the case of laboratory tests. However, using numerical analyzes, it is much easier to obtain enough data for the behavioral assessment when statistical methods are used for this purpose.

References

[1] K. Ostrowski, "Experience in structures designing according to European standards," *Jurnal of Civil Engineering, Environment and Architecture,* vol. 58, no. 3/I, pp. 203–214, 2011.

[2] K. Weynand, J. P. Jaspart, and M. Steenhuis, "Economy studies of steel building frames with semirigid joints," *Journal of Constructional Steel Research,* vol. 46, no. 1, p. 85, 1998/04/01 1998.

[3] U. Kuhlmann and A. Fürch, "Rotation capacity of steel joints (second draft)," in *COST C1 – Working Group Meeting*, Helsinki, 1997.

[4] U. Kuhlmann and A. Fürch, "Rotation capacity of steel joints," in *COST Project C1 – Working Group Meeting*, Clermont-Ferrand, 1996.

[5] M. E. Lemonis and C. J. Gantes, "Incremental modeling of T-stub connections," *Journal of Mechanics of Materials and Structures,* vol. 1, no. 7, pp. 1135–1159, 2006. Journal of Mechanics of Materials and Structures.

[6] L. Massimo, R. Gianvittorio, S. Aldina, and S. da Silva Luis, "Experimental analysis and mechanical modeling of T-stubs with four bolts per row," *Journal of Constructional Steel Research,* vol. 101, pp. 158–174, 2014.

[7] A. M. Girão Coelho, F. S. K. Bijlaard, and L. Simões da Silva, "Experimental assessment of the ductility of extended end plate connections," *Engineering Structures,* vol. 26, pp. 1185–1206, 2004.

[8] Y. Shi, G. Shi, and Y. Wang, "Experimental and theoretical analysis of the moment–rotation behaviour of stiffened extended end-plate connections," *Journal of Constructional Steel Research,* vol. 63, no. 9, pp. 1279–1293, 9// 2007.

[9] G. L. Kulak, J. W. Fisher, and J. H. Struik, "Guide to Design Criteria for Bolted and Riveted Joints Second Edition," 2001.

[10] H. Agerskov, "Analysis of bolted connections subject to prying," *Journal of the Structural Division,* pp. 2145–2163, November January 1977.

[11] V. Piluso, C. Faella, and G. Rizzano, "Ultimate behavior of bolted T-stubs. I: Theoretical model," *Journal of Structural Engineering,* vol. 127, no. 6, pp. 686–693, 2001.

[12] V. Piluso, C. Faella, and G. Rizzano, "Structural Semi-Rigid Connections: Theory, Design and Software," ed: Boca Raton (USA): CRC Press, 2000.

[13] A. M. Girão Coelho and L. Simões da Silva, "Numerical evaluation of the ductility of a bolted T-stub connection," in *Advances in Steel Structures (ICASS '02)*Oxford: Elsevier, 2002, pp. 277–284.

[14] A. Girão Coelho, "Rotation capacity of partial strength steel joints with three-dimensional finite element approach," *Computers and Structures,* vol. 116, pp. 88–97, 2013.

[15] K. Ostrowski and A. Kozłowski, "FEM based assessment of the rotation capacity of bolted joints," in *Recent Progress in Steel and Composite Structures: Proceedings of the XIII International Conference on Metal Structures (ICMS2016, Zielona Góra, Poland, 15-17 June 2016)*, 2016, p. 479: CRC Press.

[16] K. Ostrowski and A. Kozłowski, "The influence of end–plate joints stiffening on the rotation capacity," *ce/papers,* vol. 1, no. 2–3, pp. 381–388, 2017. ce/papers.

[17] K. Ostrowski, "Finite element analysis of the rotation capacity of beam-to-column end-plate bolted joints," ed: Eurosteel, 2014.

[18] J.-P. Jaspart, "Etude de la semi-rigidité des noeuds poutre-colonne et son influence sur la résistance et la stabilité des ossatures en acier," Université de Liège, Belgique, 1991.

[19] P. Zoetemeijer, "A Design Method for the Tension Side of Statically Loaded, Bolted Beam-to-Column Connections," (in en), 1974.

[20] B. Yoke, L. Yee, and R. E. Melchers, "Moment-rotation curves for bolted connections," *Journal of Structural Engineering,* vol. 112, pp. 615–635, 1986.

[21] *DD ENV 1993-1-1:1992, Eurocode 3: Design of steel structures – Part 1.1: General rules and rules for buildings*, 1992.

[22] *EN 1993-1-8:2005 (E), Eurocode 3: Design of steel structures – Part 1-8: Design of joints*, 2005.

[23] K. Ostrowski and A. Kozłowski, "Hierarchical validation of FEM models," in *XIV International Scientific Conference Rzeszów–Lwow–Koszyce. Rzeszów*, 2015.

[24] L. E. Schwer, "Verification and validation in computational solid mechanics and the ASME Standards Committee," *WIT Transactions on the Built Environment,* vol. 84, 2005.

[25] F. Wald, L. Gödrich, M. Kurejková, L. Šabatka, J. Kabeláč, and D. Kojala, "Validation and verification in design of structural steel connections," *ce/papers,* vol. 1, no. 2–3, pp. 143–152, 2017.

[26] K. Ostrowski and A. Kozłowski, "Experimental study of the portal frame for the validation of FEM model of beam to column joint (in Polish)" Czasopismo Inżynierii Lądowej, Środowiska i Architektury – Journal of Civil Engineering, Environment and Architecture, JCEEA. T. XXXIV, Z. 64 (3/I/17), pp. 437–446, 2017, DOI:10.7862/rb.2017.136.

[27] A. Mijar and J. Arora, "An augmented Lagrangian optimization method for contact analysis problems, 1: formulation and algorithm," *Structural and Multidisciplinary Optimization,* vol. 28, no. 2–3, pp. 99–112, 2004.

Przesłano do redakcji: 10.05.2018 r.
Przyjęto do druku: 15.06.2018 r.