Budownictwo 23
DOI: 10.17512/znb.2017.1.22

David Mikolášek[1], Martin Krejsa[2], Jiří Brožovský[3], Přemysl Pařenica[4]
Petr Lehner[5]

NUMERICAL AND EXPERIMENTAL ANALYSIS OF WELDS IN STEEL STRUCTURAL ELEMENT

Introduction

Numerical modeling is increasingly promoting into design practice. Using powerful computers and efficient software systems can provide valuable results, which serve to increase the reliability of the proposed support systems and elements [1] as well as to a qualitatively higher level in the design of building structures [2-9]. An important tool for mathematical modeling is particularly the Finite Element Method - FEM, whose principle is to discrete continuum to certain (finite) number of elements and the determination of calculated parameters in individual nodes. Numerical modeling finds its application in all sorts of areas of engineering and many high-quality papers and studies have already been published in this area of the research [10-14]. The results of numerical modeling have usually limited use without the experimental verification or without the load test [15, 16]. Test results may lead to calibration and validation of mathematical model, which should ensure compliance of the numerical model and the actual behavior of the investigated structure [17]. Valuable results of mathematical modeling are also conditional on defining the material models, which are often associated with laboratory-obtained material properties [18, 19].

---

[1] VŠB - Technical University of Ostrava, Faculty of Civil Engineering, Czech Republic, e-mail: david.mikolasek@vsb.cz
[2] VŠB - Technical University of Ostrava, Faculty of Civil Engineering, Czech Republic, e-mail: martin.krejsa@vsb.cz
[3] VŠB - Technical University of Ostrava, Faculty of Civil Engineering, Czech Republic, e-mail: jiri.brozovsky@vsb.cz
[4] VŠB - Technical University of Ostrava, Faculty of Civil Engineering, Czech Republic, e-mail: premysl.parenica@vsb.cz
[5] VŠB - Technical University of Ostrava, Faculty of Civil Engineering, Czech Republic, e-mail: petr.lehner@vsb.cz

The stress analysis of welds has got also the considerable attention in the past. In [20] it was defined stress intensity factor of welded joint for typical structural details, among others for cruciform joint fillet weld. In [21] the authors discuss the derivation of the so called notch stress intensity factors for welded joints, using which can be accurately described stress distributions in the toe neighborhood of weld toes. An exact solution on the stress analysis of fillet welds was described e.g. in [22]. With the development of commercial computing systems there have been emerging works whose aim to describe the state of stress in the welds through FEM analysis. In [23] the approach for a determination of the notch stress intensity factor in welded joints using three-dimensional finite element models (SOLID 186) in software system ANSYS is described. Real behavior of welds in terms of stress analysis can be examined through physical tests. Publication [24] contains the results from twenty-four cruciform weld experiments and complementary finite element simulations to study the effect of the weld root notch on strength and ductility of fillet welds. Research of stresses in welds can also be performed e.g., using a digital camera, as in assessment of stress intensity factors for load-carrying fillet welded cruciform joints [25]. The publication [26] aims to investigate the stress-strain state in mechanically heterogeneous welded joints with a single-V butt weld by an analytical model along with a numerical simulation in ABAQUS finite element analysis software. From the above summary of publications aimed at mathematical modeling of structures, focusing on the problems of welded oriented problems is obvious that this is a very current topic. The further interpretation is dedicated to problems of mathematical modeling of welded bearing elements in steel structures - fillet welded lap joint and double V butt welded joints. Some works focused on numerical modelling of welded joints based on experimental verification have been published [27-29].

1. The proposal of the experiment

Three types of specimens were designed to investigate the state of stress in fillet of butt welds. Specimens were designed to reflect the fact that the stiffness of connected elements has to be higher than the stiffness of the welded joints (Figs. 1 and 2).

The tension test was performed during the experiment, wherein the totals load value and the corresponding deformation of the specimens under the load was monitored. Obtained data were used for the calibration of numerical models of test samples and they are necessary for further strain analysis of steel supporting elements. The results of the study suggest that the numerical modeling describes the stress-strain state in welded joints with sufficient precision. The proposed computation procedure could be used in design practice for calculations of the stress-strain state in welded joints.

It can therefore be assumed that the stress-strain diagram will reflect the behavior of strains and stresses in welds. It has to be noted also that for comparison of the behavior of the weld sample in Figure 2 was made special specimen with the same geometrical parameters, which is not welded, but is formed by only the base material.

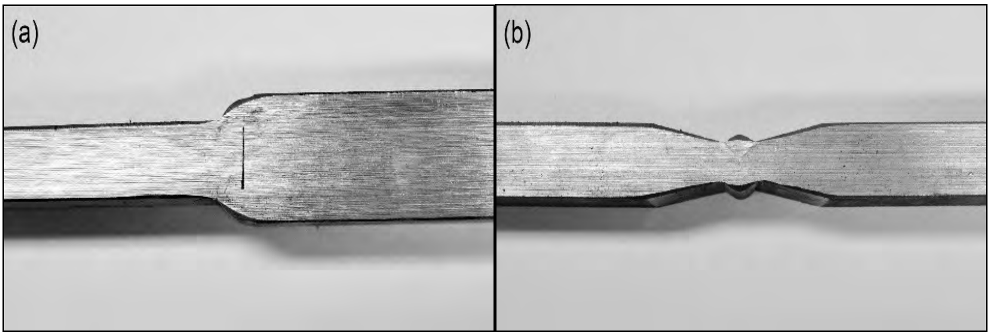


Fig. 1. a) Test specimen #1 - fillet welded lap joint, b) Test specimen #2 - double V butt welded joint

During the experiments it was monitored the displacement and the resulting force carried into the specimens. The elements were loaded with speed 2.5 kN/s and cyclically pressure unloaded for reasons relaxation due to imperfections. Additional tests were also carried out during the experiment using extensometer for measurements of the real shift of the press head.

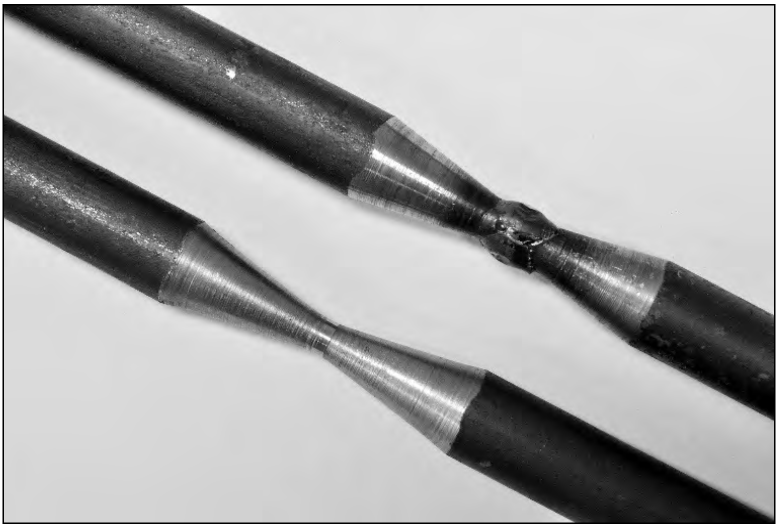


Fig. 2. Test specimens #3 and #4 - rotationally symmetric double U butt welded joint and massive element without welding

Laboratory measurements of the examined samples, focusing on the stress-strain analysis of the welds, were also supplemented by a test using a laser sensor Electronic Speckle-Pattern Interferometry - ESPI [30]. Infrared thermography non-destructive testing was also used for the determining the stress distribution in welds of tested samples (such as in [31]).

2. Numerical modeling of experiment

A welded joint consists of separate zones (such as the main material, the weld and the heat affected zone) with different mechanical properties, therefore it is mechanically heterogeneous. This fact was taken into account in numerical modeling.

The main aim was at defining the tested samples of numerical models for using FEM analysis in commercial software ANSYS. The SOLID186 20-node isoparametric finite elements were used for modeling of all elements of the models. The elastic--plastic computational analysis which is discussed in this article is based on using of the von Mises plasticity condition and isotropic hardening model.

Three variants of FEM mesh density (basic size of element side was 3, 2 and 1 mm, see Figs. 3-5) were performed for each of the model. Compare/sensitivity stress-strain analysis based on bilinear and multilinear stress-strain diagrams of basic steel material, heat-affected zone and weld material were done for each numerical model and FEM mesh density.

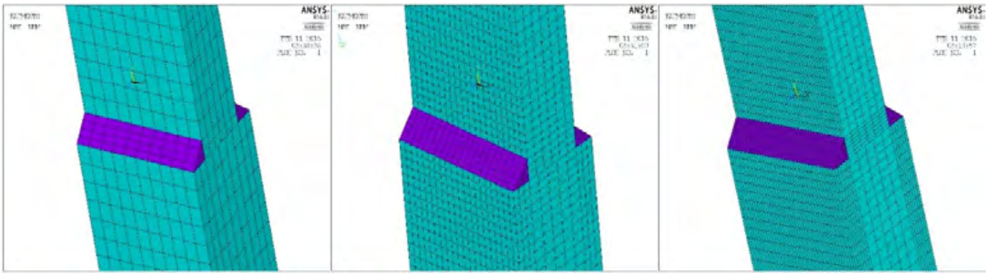


Fig. 3. Numerical model with various FEM mesh density of test specimen #1 - fillet welded lap joint

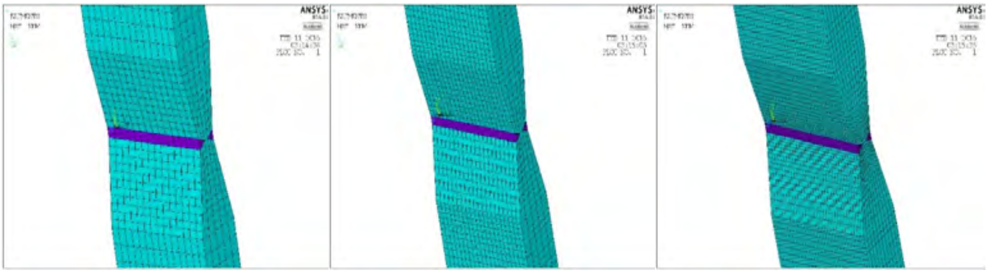


Fig. 4. Numerical model with various FEM mesh density of test specimen #2 - double V butt welded joint

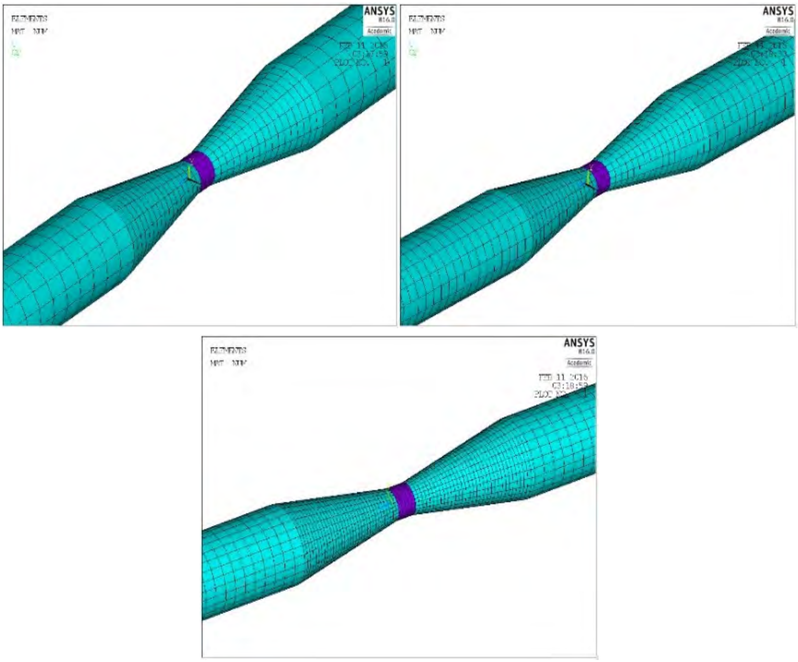


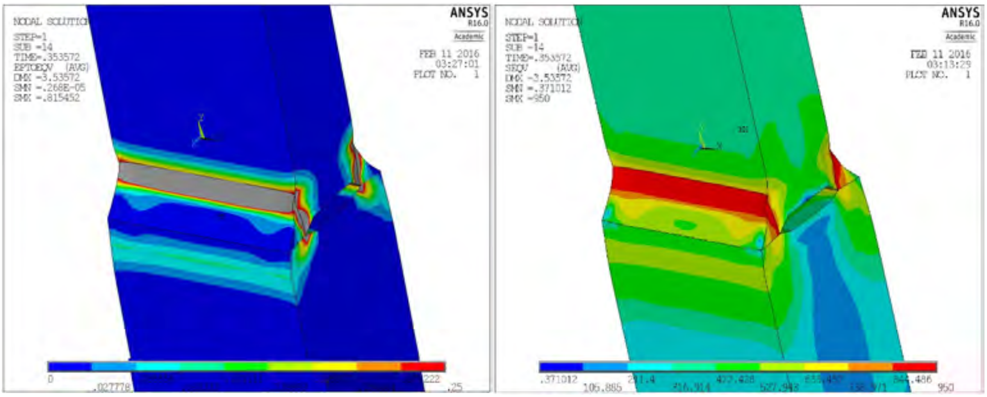
Fig. 5. Numerical model with various FEM mesh density of test specimen #3 - rotationally symmetric double U butt welded joint

These diagrams were verified using measurement data from experiments. Welded specimens was subjected to a detailed macroscopic and microscopic examination of using the Vickers hardness test according EN ISO 6507-1, EN ISO 17639 and EN ISO 9015-1 with the main focus on geometry and material properties of the heat-affected zone.

3. Analysis of results

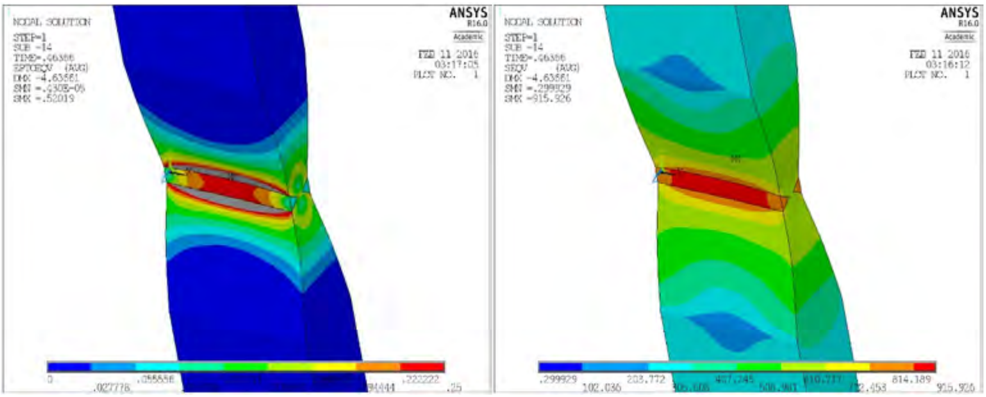
Damage in welds occurred in all samples according to the assumptions. Resulting total strain and von Misses stress for numerical models with the densest FEM mesh and for maximal achieved strength in tested samples is possible to see for all three types of tested and modeled welded specimens in Figures 6.

The resulting stress-strain diagrams of calibrated numerical models were subjected to detailed analysis and compared with the experimental results. Overall, the comparison between the results of numerical modeling and experimental data showed a good agreement (Figs. 7-10).



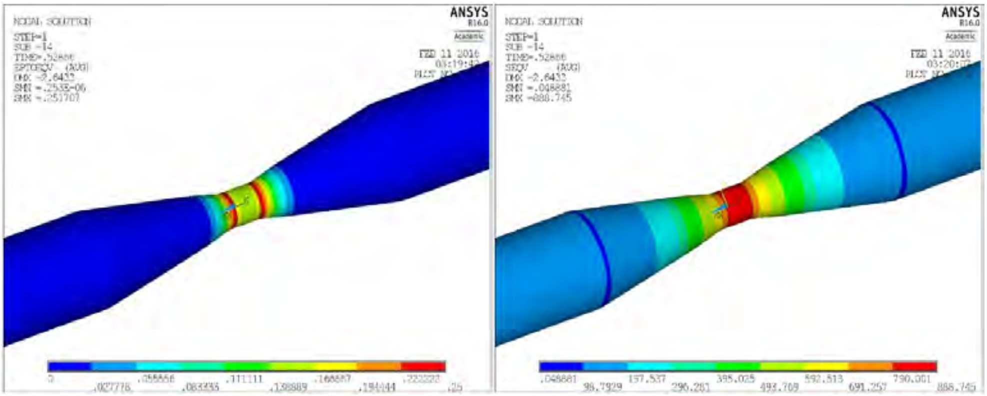
a) total strain - test specimen #1

b) von Mises stress - test specimen #1



c) total strain - test specimen #2

d) von Mises stress - test specimen #2



e) total strain - test specimen #3

f) von Mises stress - test specimen #3

Fig. 6. Resulting total strain and von Mises stress in numerical model of test specimen #1, #2 and #3

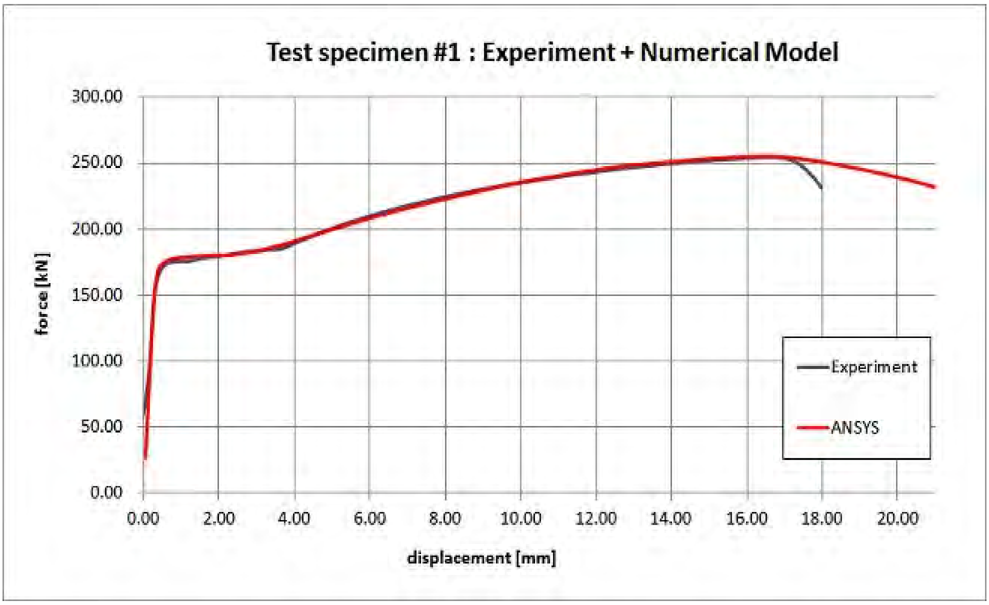


Fig. 7. Comparison of stress-strain diagrams - numerical model vs. experiment: test specimen #1

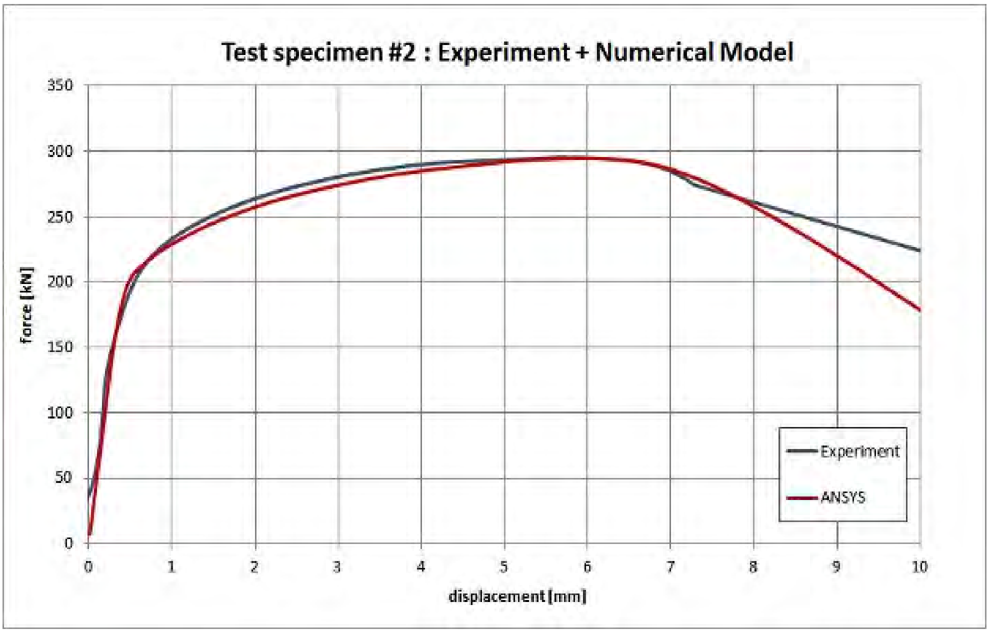


Fig. 8. Comparison of stress-strain diagrams - numerical model vs. experiment: test specimen #2

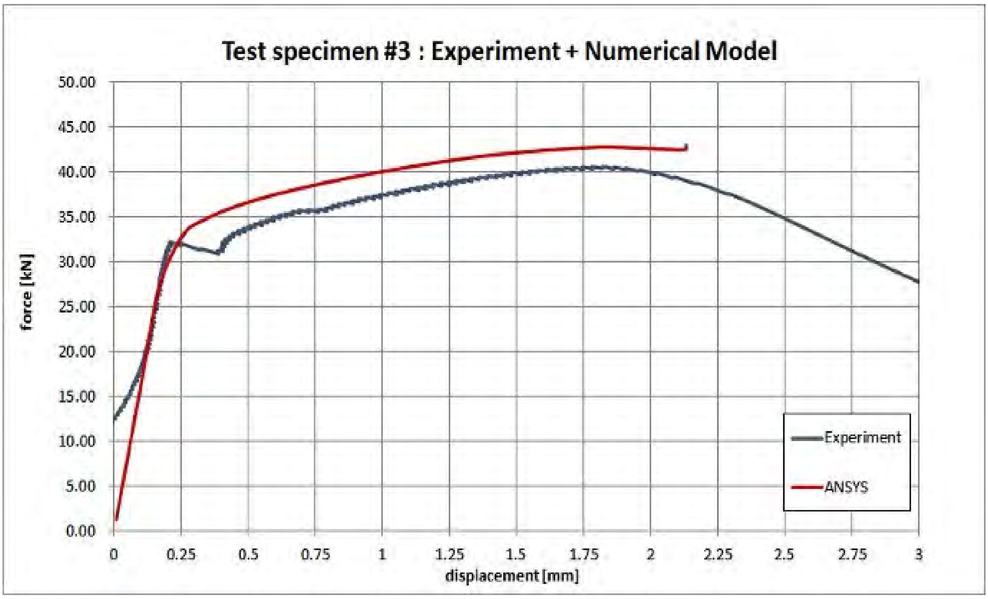


Fig. 9. Comparison of stress-strain diagrams - numerical model vs. experiment: test specimen #3

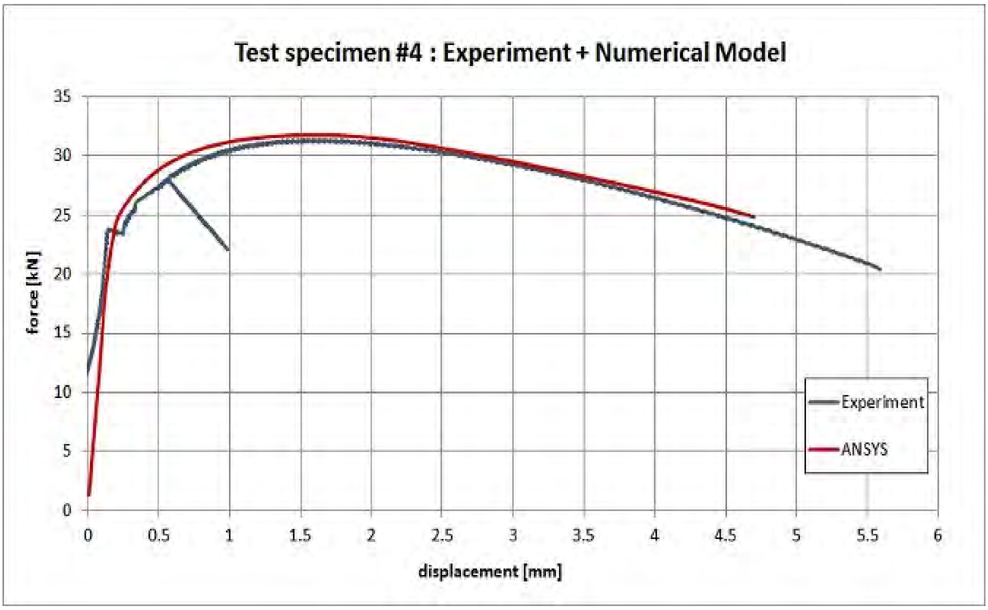


Fig. 10. Comparison of stress-strain diagrams - numerical model vs. experiment: test specimen #4

Conclusion

The article described numerical modeling of the welded joint in steel structural elements based on experimental tests and their numerical analysis in the program system ANSYS. The experiment was focused on stress analysis of fillet and butt welds in supporting elements of steel structures. On the basis of these experiments there were obtained parameters of a real stress-strain relations that will be used as input data for further works in the area of modeling of specific types of welded components.

The results of the study suggest that the numerical modeling describes the stress-strain state in welded joints with sufficient precision. The proposed computation procedure could be used in design practice for calculations of the stress-strain state in welded joints.

Acknowledgements

This contribution has been developed as a part of the research project GACR 17-01589S "Advanced computational and probabilistic modelling of steel structures taking account fatigue damage" supported by the Czech Grant Agency and also has been completed thanks to the financial support provided to VSB-Technical University of Ostrava by the Czech Ministry of Education, Youth and Sports from the budget for conceptual development of science, research and innovations for the 2017 year.

References

[1] Holicky M., Markova J., Sykora M., Target reliability levels in present standards, Transactions of the VSB - Technical University of Ostrava, Civil Engineering Series 2014, 14(2), 46-53. DOI: 10.2478/tvsb-2014-0018.

[2] Hofmeyer H., Rosmanit M., Bakker M.C.M., Prediction of sheeting failure by an ultimate failure model using the fictitious strain method, Thin-Walled Structures 2009, 47(2), 151-162. DOI: 10.1016/j.tws.2008.06.005.

[3] Kormanikova E., Kotrasova K., Sizing optimization of sandwich plate with laminate faces, International Journal of Mathematics and Computers in Simulation 2016, 10, 273-280.

[4] Kotrasova K., Sloshing of liquid in rectangular tank, Advanced Materials Research 2014, 969, 320-323. DOI: 10.4028/www.scientific.net/AMR.969.320.

[5] Kralik J., Safety of nuclear power plants against the aircraft attack, Applied Mechanics and Materials 2014, 617, 76-80. DOI: 10.4028/www.scientific.net/AMM.617.76.

[6] Krejsa M., Kala Z., Seitl S., Inspection based probabilistic modeling of fatigue crack progression, Procedia Engineering 2016, 142, 145-152. DOI: 10.1016/j.proeng.2016.02.025.

[7] Labudkova J., Cajka R., Comparison of measured displacement of the plate in interaction with the subsoil and the results of 3D numerical model, Advanced Materials Research 2014, 1020, 204-209. DOI: 10.4028/www.scientific.net/AMR.1020.204.

[8] Marschalko M., Yilmaz I., Kubecka K., Bouchal T., Bednarik M., Drusa M., Bendova M., Utilization of ground subsidence caused by underground mining to produce a map of possible land-use areas for urban planning purposes, Arabian Journal of Geosciences 2014, 1-10. DOI: 10.1007/s12517-013-1250-5.

[9] Vican J., Gocal J., Odrobinak J., Moravcik M., Kotes P., Determination of railway bridges loading capacity, Procedia Engineering 2015, 111, 839-844. DOI: 10.1016/j.proeng.2015.07.155.

[10] Flodr J., Krejsa M., Mikolasek D., Brozovsky J., Parenica P., Numerical modeling of a thin-walled profile with respect to the redistribution of bending moments, [In:] J. Kruis, Y. Tsompanakis, B.H.V. Topping (Eds.), Proceedings of the Fifteenth International Conference on Civil, Structural and Environmental Engineering Computing, Civil-Comp Proceedings, Vol. 108, Civil-Comp Press, Stirlingshire, 2015, 1-15. DOI: 10.4203/ccp.108.37.

[11] Ilcik J., Arora V., Dolejs J., Design of new scaffold anchor based on the updated finite element model, Engineering Structures 2016, 118, 334-343. DOI: 10.1016/j.engstruct.2016.03.064.

[12] Jendzelovsky N., Balaz L., Numerical modeling of cylindrical tank and compare with experiment, Applied Mechanics and Materials 2014, 617, 148-151. DOI: 10.4028/www.scientific.net/AMM.617.148.

[13] Melcer J., Lajcakova G., Comparison of finite element and classical computing models of reinforcement pavement, Advanced Materials Research 2014, 969, 85-88. DOI: 10.4028/www.scien tific.net/AMR.969.85.

[14] Salajka V., Hradil P., Kala J., Assess of the nuclear power plant structures residual life and earthquake resistance, Applied Mechanics and Materials 2013, 284-287, 1247-1250. DOI: 10.4028/www.scientific.net/AMM.284-287.1247.

[15] Lokaj A., Klajmonova K., Round timber bolted joints exposed to static and dynamic loading, Wood Research 2014, 59(3), 439-448.

[16] Urban V., Krivy V., Fabian L., Experimental testing of the weathering steel road bridge in Ostrava, Advanced Materials Research 2014, 849, 228-233. DOI: 10.4028/www.scientific.net/AMR.849.228.

[17] Kmet S., Stanova E., Fedorko G., Fabian M., Brodniansky J., Experimental investigation and finite element analysis of a four-layered spiral strand bent over a curved support, Engineering Structures 2013, 57, 475-483. DOI: 10.1016/j.engstruct.2013.09.019.

[18] Major I., Major M., Modeling of wave propagation in the ADINA software for simple elastic structures, Advanced Materials Research 2014, 1020, 171-176. DOI: 10.4028/www.scientific.net/AMR.1020.171.

[19] Strauss A., Kala Z., Bergmeister K., Hoffmann S., Novak D., The object of this contribution is the comparison of the statistical characteristics of yield strength, ultimate strength and ductility of Austrian and Czech steels, Stahlbau 2006, 75(1), 55-60. DOI: 10.1002/stab.200610007.

[20] Hobbacher A., Stress intensity factors of welded joints, Engineering Fracture Mechanics 1993, 46(2), 173-182. DOI: 10.1016/0013-7944(93)90278-Z.

[21] Lazzarin P., Tovo R., A notch intensity factor approach to the stress analysis of welds, Fatigue & Fracture of Engineering Materials & Structures 1998, 21(9), 1089-1103. DOI: 10.1046/j.1460-2695.1998.00097.x.

[22] Dawei X., An exact solution on the stress analysis of fillet welds, Applied Mathematics and Mechanics 1995, 16(11), 1019-1024.

[23] Meneghetti G., Guzzella C., The peak stress method to estimate the mode I notch stress intensity factor in welded joints using three-dimensional finite element models, Engineering Fracture Mechanics 2014, 115, 154-171. DOI: 10.1016/j.engfracmech.2013.11.002.

[24] Kanvindea A.M., Gomeza I.R., Robertsa M., Fella B.V., Grondinb G.Y., Strength and ductility of fillet welds with transverse root notch, Journal of Constructional Steel Research 2009, 65(4), 948-958. DOI: 10.1016/j.jcsr.2008.05.001.

[25] Chung H.Y., Liu S.H., Lin R.S., Ju S.H., Assessment of stress intensity factors for load-carrying fillet welded cruciform joints using a digital camera, International Journal of Fatigue 2008, 30(10-11), 1861-1872. DOI: 10.1016/j.ijfatigue.2008.01.017.

[26] Daunys M., Dundulis R., Kilikevicius S., Cesnavicius R., Analytical investigation and numerical simulation of the stress-strain state in mechanically heterogeneous welded joints with a single-V

butt weld, Engineering Failure Analysis 2016, 62, 232-241. DOI: 10.1016/j.engfailanal.2016.01.016.

[27] Krejsa M., Brozovsky J., Mikolasek D., Parenica P., Zidek L., Kozak J., An experimental testing of fillet welded specimens, Applied Mechanics and Materials 2015, 752-753, 412-417. DOI: 10.4028/www.scientific.net/AMM.752-753.412.

[28] Krejsa M., Brozovsky J., Mikolasek D., Parenica P., Halama R., Experimental verification of a steel fillet welded joint model, [In:] J. Kruis, Y. Tsompanakis, B.H.V. Topping (Eds.), Proceedings of the Fifteenth International Conference on Civil, Structural and Environmental Engineering Computing, Civil-Comp Proceedings, Vol. 108, Civil-Comp Press, Stirlingshire 2015, 1-18. DOI: 10.4203/ccp.108.34.

[29] Krejsa M., Brozovsky J., Mikolasek D., Parenica P., Halama R., Numerical modeling of steel welded supporting elements, [In:] I. Zolotarev, V. Radolf (Eds.), Proceedings of 22nd International Conference Engineering Mechanics 2016, Institute of Thermomechanics, Academy of Sciences of the Czech Republic, Prague 2016, 322-325.

[30] Halama R., Pecenka L., Hornacek L., Smach J., Krejsa M., Selected engineering applications of 3-D strain measurements using ESPI, [In:] P. Padevet, P. Bittnar (Eds.), Proceedings of EAN 2015 - 53rd Conference on Experimental Stress Analysis, Czech Technical University in Prague, 2015, 108-112.

[31] Rodriguez-Martin M., Laguela S., Gonzalez-Aguilera D., Martinez J., Thermographic test for the geometric characterization of cracks in welding using IR image rectification, Automation in Construction 2016, 61, 58-65. DOI: 10.1016/j.autcon.2015.10.012.

Abstract

The paper is focused on the numerical models of steel welded supporting elements and their verification using experiment. Currently, for the stress-strain analysis of the elements in supporting structures it is possible to use many commercial software systems, based on the finite element method - FEM. It is important to check and compare the results of FEM analysis with the results of physical verification test, in which the real behavior of the bearing element can be observed. The results of the comparison can be used for calibration of the computational model.

The article deals with the physical tests of steel supporting elements, whose main purpose is obtaining the material, geometry and strength characteristics of the fillet and butt welds. Three types of specimens were designed to investigate the state of stress in fillet of butt welds. Specimens were designed to reflect the fact that the stiffness of connected elements has to be higher than the stiffness of the welded joints. It can therefore be assumed that the stress-strain diagram will reflect the behavior of strains and stresses in welds. During the experiments it was monitored the displacement and the resulting force carried into the specimens.

Keywords: numerical modeling, experiment, steel structure, FEM, fillet weld, butt weld

Analiza numeryczna i eksperymentalna spoin w stalowym elemencie konstrukcyjnym

Artykuł koncentruje się na modelach numerycznych stalowych spawanych elementów nośnych i ich weryfikacji za pomocą badań eksperymentalnych. Obecnie do analizy naprężeń i odkształceń elementów w konstrukcjach nośnych można wykorzystać wiele komercyjnych systemów oprogramowania, opartych na metodzie elementów skończonych (MES). Ważne jest, aby sprawdzić i porównać wyniki analizy MES z wynikami badań eksperymentalnych, w których można zaobserwować rzeczywiste zachowanie elementu konstrukcyjnego. Wyniki porównania można wykorzystać do kalibracji modelu obliczeniowego. W artykule omówiono badania laboratoryjne stalowych elementów nośnych, których głównym celem było uzyskanie informacji na temat materiału, geometrii i charakterystyki wytrzymałościowej spoin pachwinowych i czołowych. Zaprojektowano trzy rodzaje próbek w celu zbadania stanu naprężeń w spoinach czołowych. Próbki zostały zaprojektowane tak, aby odzwierciedlały fakt, że sztywność połączonych elementów musi być większa niż sztywność połączeń spawanych. Można zatem założyć, że wykres naprężenie-odkształcenie odzwierciedla zachowanie odkształceń i naprężeń w spoinach. Podczas eksperymentów monitorowano przemieszczenie i wynikową siłę przenoszoną przez próbki.

Słowa kluczowe: modelowanie numeryczne, eksperyment, konstrukcja stalowa, MES, spoina pachwinowa, spoina czołowa