Static behavior of the weld in the joint of the steel support element using experiment and numerical modeling

M Krejsa¹, J Brozovsky¹, D Mikolasek¹, P Parenica¹ and L Koubova¹

¹ Department of Structural Mechanics, Faculty of Civil Engineering, VSB-Technical University of Ostrava, Ludvika Podeste 1875/17, 708 33 Ostrava – Poruba, Czech Republic

E-mail: martin.krejsa@vsb.cz

Abstract. The paper is focused on the numerical modeling of welded steel bearing elements using commercial software system ANSYS, which is 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 test of steel supporting elements, whose main purpose is obtaining of material, geometry and strength characteristics of the fillet and butt welds including heat affected zone in the basic material of welded steel bearing element. The pressure test was performed during the experiment, wherein the total 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 stress and strain analysis of steel supporting elements.

1. Introduction

Welded details are indispensable in modern steel structures. With their use it can be achieved the homogeneous nature of the joint, while not weakening the original support element as is the case with mechanical fasteners. Welds and their use, of course, also have their limitations and disadvantages. Among the disadvantages can be stress in the vicinity of the weld, which results in residual stress. The material around the weld during cooling takes shape differently than the rest of thermally unaffected parts of the welded construction, which leads to deformations of the support element and the said residues. Welds and joints of welded parts also cause local geometric, stiffness and material non-linearity, which in real constructions creates local stress peaks.

Problems associated with numerical modeling of building structures and supporting elements have been published in the past in a number of valuable scientific papers (e.g. [2, 4, 6, 7, 14, 20]). For most of these purposes, the Finite Element Method – FEM is used (e.g. [19]), which is very suitable for these numerical studies. This method is applied in a number of commercial software that are more or less widely used in engineering practice [13, 16]. The ANSYS software system is probably the most widely used in academic and research sphere. ANSYS allows modeling of the entire construction or its supporting elements in detail (e.g. [8, 18]) and for the purposes of the research work of this article is very appropriate.

It is necessary to mention several publications that were inspiring for our research work. Based on ANSYS simulation software, the model of resistance welding drying plate - thin-wall pressure vessel
with multiple welds, large size and large deformation, was established to study the deformation and stress distribution under different temperatures and pressures in [11]. The interest on calculating welding residual stresses from the welding process, through finite element (FE) simulation, has increased in the last period (e.g. [5]). The effects of weld geometry size on bearing capacity of submerged arc welded pipe with double-side longitudinal seam under bearing internal pressure conditions were simulated and analyzed by finite element simulation based on the ANSYS software in [17]. The finite element software ANSYS was also used to set up the full-scale sectional model of the welded connection of the U-rib and the deck plate and the refined model for the local part of the welded connection was established with the focus on the stress in the weld toes and weld roots of the welded connections in [15]. Design optimization of spot welded structures to attain maximum strength by using the ANSYS, general purpose finite element analysis software was presented in [3]. A comparative study was conducted in [1] to determine the structural hot-spot stress at the weld toe of the welded joint in the orthotropic steel bridge deck by use the finite element models of the welded joint and the finite element software ANSYS with the solid and the shell elements. The software ANSYS was also used to establish the three-dimensional finite element model for the steel and concrete joint section of railway hybrid box girder cable-stayed bridge to investigate the force characteristics and force transfer mechanism in [21]. The welding process and in accordance with the occurrence mechanism and characteristics of the welding residual stress in the constructional details of the complex structure of the steel girder bridge were proposed in [22]. Finite element models built by ANSYS were used to simulate girth weld of marine steel tubular piles with focus on pipe splices in [12].

Basic problems of numerical modeling of fillet and butt welds in steel structural elements were published by authors in [9, 10]. The paper [9] presented basic research of modeling of fillet welds without taking account the influence of heat-affected zones. There were also published description of interesting methods for statical analysis of welded joint behavior, such as 3D strain measurement using laser sensor DantecDynamics Q100 or infrared thermographic non-destructive testing. The short international conference paper [10] dealt with the brief description of physical tests of steel supporting elements, whose main purpose is obtaining the material, geometry and strength characteristics of the fillet and butt welds. Both publications show that this is a topical theme and research in this topic is useful in engineering practice, which is complemented in this paper by research of investigating the chosen working diagrams for steel components, and influence of the heat-affected zones on resistances and stress strains in the analyzed types of joints. Furthermore, the effects of changes in stiffness of different types of materials, sharp edges and thickness of the bonded sheets were analyzed on the overall strain and strain of details. Numerical models were validated by physical models that were measured using multiple types of measuring instruments (extensometers, strain gauges and optical laser topography and temperature analysis in the infrared spectrum of light waves).

2. Description of numerical models

The accuracy of static calculations achieved by numerical methods depends on several basic factors. The first is the geometric description of the solved problem, which is well fulfilled with the current state of computation. The detail in the numerical model can be described on a very fine scale now. The second factor influencing the accuracy of the solution is the choice of the FEM element. The mathematical approximation of the modelled object is always a simplification of reality. With current numerical modelling options, the selection of suitable finite elements is sufficient to correctly describe nodal deformations. With proper FEM mesh selection and a correct definition of boundary conditions (load and external connections, together with appropriate load increments), solution of the numerical modelling converges to satisfactory results. The third factor is the choice of the stress-strain diagram of the behavior of the individual materials, which depends on the results under analysis and demands for the accuracy of the solution. Together with these requirements, computing time and hardware...
demands also come into play. With more complex material models, however, higher demands are placed on users, both on the correct interpretation of the results and on their evaluation.

Seven types of welded samples with different load effects on the weld were created and tested in the framework of numerical modeling of welding in the joint of the steel support element. The first three (#1, #2 – see Figs. 1 & 2, and #3 – see Figs. 3 & 4) were pressure tested, focusing on fillet welds behavior (details in [9]). The remaining four samples (#4 – see Figs. 5, 6 & 12, #5 – see Figs. 7, 8 & 14, #6 and #7) were stressed with a focus on the behavior in the fillet and butt welds (Figs. 9 & 10). The models were calculated for different FEM mesh densities and for different types of material stress-strain diagrams. The following is a more detailed description of these parameters.

![Figure 1. Test specimen #2, details of fillet weld damage.](image1)

![Figure 2. Test specimen #2, distribution of von Misses stresses at 100% limit load.](image2)

![Figure 3. Test specimen #3, details of fillet weld damage.](image3)

![Figure 4. Test specimen #3, distribution of von Misses stresses at 100% limit load.](image4)

2.1. Modeling of welds in ANSYS

Welds were modeled in the ANSYS using SOLID186 - homogeneous structural solid element. The SOLID186 is a higher order 3-D 20-node solid element that exhibits quadratic displacement behavior. The element is defined by 20 nodes having three degrees of freedom per node: translations in the x, y, and z directions. The element supports plasticity, hyperelasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials. This finite element is suitable for numerical models with large stress differences. Numerical
models also respect structural non-linearity through contact elements in areas that were not welded and bonded with weld metal.

2.2. FEM mesh density
Numerical models have been assembled for the different density of the FEM mesh. The FEM mesh density has been optimized from the welds to the external supports throughout the whole model volume. The aim of this parametric study was to determine the effect of the FEM mesh density on the resulting deformations and stresses in the area of welded joint. The FEM mesh density affects not only the accuracy of the obtained results but also the creation of areas with local numerical peaks of stress. Apart from these, however, the FEM mesh density is also tied to the computational demands of numerical models - with the speed of calculation, memory requirements and effectiveness of the results analysis. A detailed study of the effect of FEM mesh density on the resulting behavior of the numerical models can not be mentioned here due to the limited scope, and it is necessary to refer the readers to one of the other articles the authors prepare to publish, but the results show that the effect of the FEM density is not critical.

**Figure 5.** Test specimen #4 consists of two flat steel elements and fillet weld, during the tensile test.

**Figure 6.** Test specimen #4, detail of FEM in numerical model.

**Figure 7.** Test specimen #5 consists of two flat steel elements and butt weld, detail of damage.

**Figure 8.** Numerical model of test specimen #5, distribution of von Mises stress.
2.3. Heat-affected weld zone

The heat-affected zone (HAZ) can be determined on the basis of a standard EN ISO 9015-1:2011 (Destructive tests on welds in metallic materials - Hardness testing) using Vickers hardness test. This makes it possible to obtain hardness distributions around the fillet and butt welds (see Fig. 11).

![Figure 9. Fillet weld of test specimen #4 in detail during the Vickers hardness test.](image)

![Figure 10. Butt weld of test specimen #5 in detail during the Vickers hardness test.](image)

The heat-affected zone was included in the computational models of the tensioned specimens. The heat-affected zone was divided into several areas, which are different in the stress-strain diagrams (see Figs. 13 & 15). These zones have different strength, yield stress, and strain behavior during loading. Compared to the base material, the heat-affected zone has higher stiffness and strength but is more brittle. Local extremal stress and strain accumulate here, and there is an initiation of a material failure. It has been shown that the use of a heat-affected zone in numerical modeling of welds is necessary.

![Figure 11. Graphical dependence of hardness distribution in butt weld and its heat-affected zone for four measurements using Vickers hardness test.](image)

2.4. Stress-strain diagrams in numerical models

Stress-strain diagrams entering mathematical models play a significant role in achieving the correct results compared to physical tests. In case of requirements for the analysis of heavily plasticized materials, it is necessary to use more precise working diagrams. In principle, three basic material models are used for the calculations described here.

The first one is a linear material model that respects Hooke's law and has an informative value only to identify details where stress is to reaching the yield strength. This material model can advantageously be used for rapid numerical analysis before proceeding to use more advanced material models.

The second material model is an elastic-plastic material with bilinear hardening, which is very practical, relatively quick to apply and easily accessible. The advantage of this model is also its...
transparency and it can also be assessed and evaluated with regard to the basic knowledge of the materials used. This material model should be used in cases where knowledge of the stiffness of the behavior of the welded joint is not important. Its use can achieve the correct global deformations and also the maximum bearing capacity of the supporting element. However, the redistribution of the strain in the details of joints and welds is not correct because the stress does not correspond to the real values measured on the physically tested samples.

Figure 12. Numerical model of test specimen #4.

Figure 13. Used stress-strain diagrams for numerical modelling of test specimen #4.

Figure 14. Numerical model of test specimen #5.

Figure 15. Used stress-strain diagrams for numerical modelling of test specimen #5.

The third material model has a general curve for hardening that characterizes the actual behavior of the material under load. This curve can have a continuous character, and can be approximated by a polyline. This material model most closely matches the actual behavior of the welded joint under analysis or detail under loading. This claim is based on the fact that the resulting stress-strain diagrams of numerical models were subjected to detailed analysis and compared with the experimental results. Overall, the comparison between the results of numerical modeling using the third material model and experimental data actually showed the best agreement – see Figs. 16 and 17. However, obtaining this curve is usually complex and time-consuming and costly.

3. Analysis of numerical modeling results
Analysis of numerical models was done in two ways. The first and basic is the physical test. Real measurement of deformations and strains together with input load is the most important benchmark for numerical modeling. A comparison showed very good agreement (see Figs. 16 & 17). The second way of evaluating is the experience with the material and the loadings, the standard assessments and the provisions that have an implemented estimate of the behavior of the phenomena studied. The experience and the feelings of the model maker that the numerical model under analysis creates and
harmonizes is also essential. Even the exact numerical model may exhibit deviations from the physically measured values.

![Figure 16](image1.png)  
**Figure 16.** Comparison of stress-strain diagrams - numerical model vs. experiment, specimen #4.

![Figure 17](image2.png)  
**Figure 17.** Comparison of stress-strain diagrams - numerical model vs. experiment, specimen #5.

In evaluating the numerical tests carried out, the following facts were found:

- It has been found that the analysed numerical models are not too sensitive to the increased density of the FEM network.
- Numerical models had to be modelled with taking account to the heat-affected zone for reasons of consistency with physical tests.
- The actual geometry of the entire welded joint and the heat-affected zone was modelled for a more precise match of the stress-strain curve behaviour of the numerical model with the physical tests.
- Numerical models included advanced material models for compliance with physical tests until sample failure.

4. Conclusion
In conclusion, we can summarize the newly acquired experience useful in the numerical modeling of the welds of supporting elements in steel structures. When creating numerical models in common cases, it is sufficient to use a FEM mesh with standard density, define description of geometry of the structure without considering initial imperfections and use the bilinear stress-strain diagram for the description of the used base material and the heat-affected area behavior. The reason is that the use of building structures in terms of stress occurs at the level of stress on the yield point and the effects of the load are below the level of plastic utilization of the supporting elements. The stress peaks are redistributed in the design using a bilinear work diagram, and the static behavior of the structure and its supporting elements can be evaluated at the strain according to standards.

If a more accurate analysis is needed, it is appropriate to use the numerical modeling procedure described in the article. This analysis is suitable for complex details especially for welded structures where there is multi-axis strain, notches and heat-affected zones with modified material properties. The residual stress produced by steel structures also plays its role here. In this regard, numerical models give valuable insight into the behavior of the structure and redistribution of stresses in it. Numerical models, however, need to be supplemented by physical tests.

References
[1] Di S-K, Wen C and Ye X-W 2015 Finite element analysis of structural hot-spot stress for orthotropic steel bridge deck *Journal of Zhejiang University (Engineering Science)* 49(2) 225-231
[2] Drienovska J and Tvrdá K 2017 Deflection of a Beam Considering the Creep *Procedia Engineering* 190 459-463
[3] Ertas, A H 2015 Design optimization of spot welded structures to attain maximum strength *Steel and Composite Structures* 19(4) 995-1009

[4] Kala Z 2015 Reliability analysis of the lateral torsional buckling resistance and the ultimate limit state of steel beams with random imperfections *Journal of Civil Engineering and Management* 21(7) 902-911

[5] Knoedel P, Gkatzogiannis S and Ummenhofer T 2017 Practical aspects of welding residual stress simulation *Journal of Constructional Steel Research* 132 83-96

[6] Kormanikova E, Riecky D and Zmdinak M 2011 Strength of composites with fibres *Computational Methods in Applied Sciences* 24 167-183

[7] Kotrasova K and Grajciar I 2014 Dynamic analysis of liquid storage cylindrical tanks due to earthquake *Advanced Materials Research* 969 119-124

[8] Krašlik J 2017 Actual problems of the safety and reliability of the NPP Structures in Slovakia *Key Engineering Materials* 738 261-272

[9] Krejsa M, Brozovsky J, Mikolasek D, Parenica P, Flodr J, Materna A, Halama R and Kozak J 2017 Numerical modeling of steel fillet welded joint *Advances in Engineering Software* (in press) DOI: 10.1016/j.advengsoft.2017.03.013

[10] Krejsa M, Brozovsky J, Mikolasek D, Parenica P, Koubova L and Materna A 2017 Numerical Modeling of Fillet and Butt Welds in Steel Structural Elements with Verification Using Experiment *Procedia Engineering* 190 318-325 DOI: 10.1016/j.proeng.2017.05.344

[11] Li H, Duan D, Huang C and Qi H 2017 Finite Element Analysis and Design of the Resistance Welding Structure of the Drying Plate *Journal of Tianjin University Science and Technology* 50(10) 1093-1098

[12] Li Y, Zhou X-P, Qi Z-M and Zhang Y-B 2014 Numerical study on girth weld of marine steel tubular piles *Applied Ocean Research* 44 112-118

[13] Major M and Major I 2017 Modelling of wave phenomena in the Zahorski material based on modified library for ADINA software *Applied Mathematical Modelling* 46 727-735

[14] Malikova L and Seitl S 2017 Application of the Williams expansion near a bi-material interface *Key Engineering Materials* 754 206-209

[15] Mei D-P and Liao G-X 2017 Notch stress analysis of welded connection of new type of edge upset U-rib and deck plate *Bridge Construction* 47(1) 65-70

[16] Nemec I, Stekbauer H, Vaneckova A and Vlk Z 2017 Explicit and implicit method in nonlinear seismic analysis *MATEC Web of Conferences* 107 00066

[17] Qiao G, Liu Y, Han X, Wang X and Xiao F 2017 Simulation study on effects of geometry size of weld joint on bearing capacity of steel pipe *Transactions of the China Welding Institution* 38(3) 33-36

[18] Salažka V, Hradil P and Kala J 2013 Assess of the nuclar power plant structures residual life and earthquake resistance *Applied Mechanics and Materials* 284-287 1247-1250

[19] Vales J, Kala Z, Martinasek J and Omishore A 2016 FE nonlinear analysis of lateral-torsional buckling resistance *International Journal of Mechanics* 10 235-241

[20] Vican J, Gocal J, Melis B, Kotes P and Kotula P 2008 Real behaviour and remaining lifetime of bridge structures *Komunikacie* 10(2) 30-37

[21] Yao Y-D, Yang Y-Q, Liu Z-B, Pu Q-H and Shi Z 2015 Finite element analysis of steel and concrete joint section of railway hybrid box girder cable-stayed bridge *Bridge Construction* 45(1) 45-50

[22] Wei J, Wang D-M, Liu X-C, Wu Z-Q and Gao Z-Y 2015 Computer simulation of welding residual stress in steel girder bridge *Bridge Construction* 45(4) 94-99

**Acknowledgments**

This contribution 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 2018 year.
