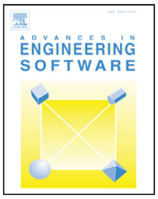
Advances in Engineering Software 000 (2017) 1–11

Contents lists available at ScienceDirect

Advances in Engineering Software

journal homepage: www.elsevier.com/locate/advengsoft



Numerical modeling of steel fillet welded joint

Martin Krejsa[a,*], Jiri Brozovsky[a], David Mikolasek[a], Premysl Parenica[b], Jakub Flodr[a], Alois Materna[a], Radim Halama[c], Jaroslav Kozak[d]

[a] *VSB - Technical University Ostrava, Faculty of Civil Engineering, Department of Structural Mechanics, Ludvika Podeste 1875/17, 708 33 Ostrava-Poruba, Czech Republic*
[b] *VSB - Technical University Ostrava, Faculty of Civil Engineering, Department of Building Structures, Ludvika Podeste 1875/17, 708 33 Ostrava-Poruba, Czech Republic*
[c] *VSB - Technical University Ostrava, Faculty of Mechanical Engineering, Department of Applied Mechanics, 17. listopadu 15/2172, 708 33 Ostrava-Poruba, Czech Republic*
[d] *Vitkovice Power Engineering a.s., Design department for steel construction, Ruska 1142/30, 706 02 Ostrava-Vitkovice, Czech Republic*

ARTICLE INFO

Article history:
Received 17 February 2017
Accepted 28 March 2017
Available online xxx

MSC:
68U07
74-05
74A10
74C05
74S05

Keywords:
Numerical modeling
Verification
Experiment
Steel structure
FEM
Fillet weld

ABSTRACT

The paper is focused on the numerical modeling of steel bearing elements and their verification using experiment. Currently, for the stress-strain analysis of the elements supporting structures it is possible to use many commercial software systems, based on the finite element method - FEM. It is important to 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 welds. The main aim was defining of tested samples numerical models for using FEM analysis and for the commercial software ANSYS. 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. The measurements were carried out for a more detailed analysis of stresses and deformations in welds samples using a strain-gauge and a Q100 laser device for measuring the 3D deformation and infrared thermographic non-destructive testing.

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.

1. 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 as well as to a qualitatively higher level in the design of structures [1–6]. Complex mathematical procedures take into account physical and geometric nonlinearities of the structure [7]. An important tool for mathematical modeling is particularly the Finite Element Method – FEM [8], whose principle is to discrete continuum to certain (finite) number of elements and the determination of calculated parameters in individual nodes [9]. In this area of the research it has already been published many high-quality papers and studies (e.g. [10–13]).

Numerical modeling finds its application in all sorts of areas of engineering. The paper [14] e.g. shows approaches to numerical modeling of shear wall joint experiments of large panel buildings. In [15] and [16] authors discuss results of numerical modeling of masonry columns and arches which have been reinforced by fibre-glass fabrics. There are publications that analyze problems of mathematical modeling of structures based on thin-walled cold-rolled cross-section [17,18], composite steel-concrete columns [19], fiber reinforced laminate plates [20] or round timber bolted joints with steel plates [21].

The results of numerical modeling have usually limited use without the experimental verification [22–24] or without the load test [25]. Test results may lead to calibration and validation of mathematical model [26,27], which should ensure compliance of the numerical model and the actual behavior of the investigated

---

* Corresponding author.
E-mail addresses: martin.krejsa@vsb.cz (M. Krejsa), jiri.brozovsky@vsb.cz (J. Brozovsky), david.mikolasek@vsb.cz (D. Mikolasek), premysl.parenica@vsb.cz (P. Parenica), jakub.flodr@vsb.cz (J. Flodr), alois.materna@vsb.cz (A. Materna), radim.halama@vsb.cz (R. Halama), jaroslav.kozak@vitkovice.cz (J. Kozak).
URL: http://www.fast.vsb.cz/ (M. Krejsa), http://www.fast.vsb.cz/ (J. Brozovsky), http://www.fast.vsb.cz/ (D. Mikolasek), http://www.fast.vsb.cz/ (P. Parenica), http://www.fast.vsb.cz/ (J. Flodr), http://www.fast.vsb.cz/ (A. Materna), http://www.fs.vsb.cz/ (R. Halama), http://www.vitkovice.cz/ (J. Kozak)

http://dx.doi.org/10.1016/j.advengsoft.2017.03.013
0965-9978/© 2017 Published by Elsevier Ltd.

Please cite this article as: M. Krejsa et al., Numerical modeling of steel fillet welded joint, Advances in Engineering Software (2017), http://dx.doi.org/10.1016/j.advengsoft.2017.03.013

structure [28]. Valuable results of mathematical modeling are also conditional on defining the material models [29–31], which are often associated with laboratory-obtained material properties [32–35]. Modeling of fibre reinforced concrete based materials under static and dynamic loading is shown e.g. in [36]. Some particular and selected problems of numerical modeling are aimed at ultimate limit state and probability-based studies pertaining e.g lateral-torsional buckling of steel structural elements [37] or fatigue crack progression [38–40].

The stress analysis of welds has got also the considerable attention in the past. In [41] it was defined stress intensity factor of welded joint for typical structural details, among others for cruciform joint fillet weld. In [42] 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 [43].

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 [44] it is described the approach for a determination of the notch stress intensity factor in welded joints using three-dimensional finite element models (SOLID 45) in software system ANSYS. The paper [45] reports a parametric stress analysis of various configurations of rack plate stiffened multi-planar welded joints using FEM in ABAQUS software. In [46] authors focused on modeling of fillet welded cylindrical joints under tension and torsion loading with regard to the fatigue resistance. Numerical implementation for fatigue analysis of welded joints is the subject of the research in e.g. [47]. The work [48] deals the mechanical analysis with the simulation of fusion welding by the Finite Element Method, where the implemented models include a moving heat source, temperature dependence of thermophysical properties, elastoplasticity and non-steady state heat transfer. By using software systems MSC Marc and ANSYS it is possible to simulate and modeling the welding process and the rise of residual stresses [49].

Application of the finite element method to predict thermal, material and mechanical effects of welding are comprehensively described in [50]. The actual shape of the weld also appears to be a very important factor in numerical modeling of the welds [51]. Real behavior of welds in terms of stress analysis can be examined through physical tests. Publication [52] 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 [53]. The publication [54] is focused on numerical modeling of residual stress in under-matched welded joint based on equal-load theory by 3D finite element method. Article [55] introduces a welding process design tool to determine optimal arc fillet welding process parameters based on FEM, Response Surface Method (RSM) and Genetic Algorithms (GA). 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 discussion is dedicated to problems of mathematical modeling of welded bearing elements in steel structures.

2. Stress analysis of welded joints according Eurocodes

The basic document for the design of steel structural joints subjected to predominantly static loadings is the standard EN 1993-1-8 (Eurocode 3: Design of steel structures - Part 1–8: Design of joints), where it is possible to find among the other procedures specifications for designing and reliability assessment of welded joints, e.g. for fillet welds, fillet welds all round, butt welds, plug welds and flare groove welds. For the calculation of design resistance of welds it is particularly necessary to define the length and the effective thickness of the welds and their design tensile and shear strength.

Hereafter, this paper is focused only on the fillet welds, which may be used for connecting parts where the fusion faces form an angle of between 60° and 120°. The connected parts of the welded joints does not require further modification, such as preheating or bevel the edges of the edges. The effective throat thickness of a fillet weld should be taken as the height of the largest triangle (with equal or unequal legs) that can be inscribed within the fusion faces and the weld surface, measured perpendicular to the outer side of this triangle. The effective throat thickness of a fillet weld should not be less than 3 mm.

When calculating the design resistance of fillet welds in particular the direction of stresses is respected. The forces transmitted by a unit length of weld are resolved into components parallel and transverse to the longitudinal axis of the weld and normal and transverse to the plane of its throat. A uniform stress distribution is assumed on the throat section of the weld, which leads to the normal stresses ($\sigma_\perp$ and $\sigma_\parallel$) and shear stresses ($\tau_\perp$ and $\tau_\parallel$). The design resistance of the fillet weld will be sufficient if the following are both satisfied:

$$\sqrt{\sigma_\perp^2 + 3 \cdot (\tau_\perp^2 + \tau_\parallel^2)} \le \frac{f_u}{\beta_w \cdot \gamma_{M2}} \tag{1}$$

and

$$\sigma_\perp \le 0.9 \cdot \frac{f_u}{\gamma_{M2}} \; , \tag{2}$$

where f_u is the nominal ultimate tensile strength of the weaker part joined, β_w is the appropriate correlation factor, which is related to the strength class of the used steel, and γ_{M2} is the partial safety factor for joints. It may be defined in the National Annex of appropriate Eurocodes.

3. Experimentally obtained material properties of welds

Numerical modeling of details of steel structures connections among others need to define exactly the constitutive behavior of the material used in the computational model. In the case of weld materials the manufacturers provide nominal materials characteristics, which are changed during the technological process of welding. There are changes in the structure of the weld metal and also in the material of connected parts around the weld. Therefore the authors are focused on the preparation of laboratory experimental tests, results of which can be used to obtain necessary data for the numerical modeling, e.g. parameters of stress-strain relation [56].

3.1. Experimental test in laboratory

Three types of specimens were designed to investigate the state of stress in fillet welds. For each of them it was designed a different orientation in relation to the axe of weld. Therefore it can be expected to acquire valuable data of stresses in direction perpendicular to the weld and in the direction parallel to the weld. Specimens were designed to having regards to 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 particularly at the location of the welds. Load pressure were considered in all tests due to the limited possibilities of laboratory device.

Test specimen #1 (Fig. 1) is a biaxially symmetrical. This fact causes that the four fillet welds are loaded equally. The load is applied to the specimen in a direction parallel to the axes of the

Please cite this article as: M. Krejsa et al., Numerical modeling of steel fillet welded joint, Advances in Engineering Software (2017), http://dx.doi.org/10.1016/j.advengsoft.2017.03.013