Proceedings of 16th International Conference. Mechanika. 2011

Modeling of welded connections in SolidWorks Simulation

P. Griskevicius*, M. Urbas, V. Capas***, A. Kozlovas******

** Kaunas University of Technology, Kestucio 27, 44312, Kaunas, Lithuania, E-mail: paulius.griskevicius@ktu.lt*
*** Kaunas University of Technology, Kestucio 27, 44312, Kaunas, Lithuania, E-mail: m.urbas@stud.ktu.lt*
**** Kaunas University of Technology, Kestucio 27, 44312, Kaunas, Lithuania, E-mail: vytautas.capas@ktu.lt*
***** Kaunas University of Technology, Laisvės al. 55, 44309, Kaunas, Lithuania, E-mail: andriusvw@yahoo.com*

Abstract

Generally the design resistance of welded connections can be evaluated using clasical analysis or Finite Element Method (FEM). Design of steel structures welded joints in detail are described in Euro Code 3 part 8. The method basically is fitted for applying classical methods, nevertheless using of FE method also is possible. In presented work the Finite Element code SolidWorks Simulation 2010 has been used to simulate behavior of statically loaded welded specimens and to determinate weld loads for evaluation of welded connections strength. Comparing the experimental results of welded relatively simple geometries specimens the FE models of welded connections has been validated. The comparison of the design resistance evaluation of welded connections simulated using 3D solid models and shell models are presented. Also using the weld loads calculated by FE 3D solid models the way of the design resistance evaluation of welded connection according the requirements of European Standard EN 1993-1-8 is discussed.

KEY WORDS: Finite element analysis (FEA); welded joint, design resistance of welded connections, fillet weld

1. Introduction

There are many ways to simulate welded connection in the commercial Finite Element (FE) codes. Depending on task and complexity of welded structure it is possible to choose simply connected components using Bonded contact pair or deeply simulate the welding process including material nonlinearity and coupling between thermal and mechanical analysis with residual stresses [1]. The need to accommodate the effect of the evolving microstructure in the simulations was early recognized. Computational Welding Mechanics is therefore concerned with the thermo-mechanical response as well as the changes in material properties due to the thermal cycles in welding [2]. The pioneering work within simulation of welding by finite element method including material nonlinearity is presented at early 70's [3,4].

FEM are still by far the most commonly used numerical tool to obtain the forces in the complex structures [5], stress concentration factors for different types of welded joints [6], evaluate design resistance of welded joints [5], to simulate crashworthiness of weld-bonded steel joints in the vehicle structures [7]. Moreover, the mesh generation will become more difficult when the welds and especially cracks are to be included in the model, and the accuracy of the results depends on the use of element type, mesh refinement, integration scheme, boundary conditions and weld shape modeling and for it 3D solid elements [8, 9]. Predicting the strength of weld-bonded joints and discussing the simulation details of welded joints still are widely analyzed [10, 11].

Generally weld connections can be analysed using clasical analysis or Finite Element Method (FEM). Finite element analysis (FEA) quite accurately identifies the load path, which can be difficult using classical analysis in the case of complex structures. In the European Standard EN 1993-1-8 [12] there are in detail described the design of steel structures joints including main mechanical connections as bolting, riveting and welding. The procedure described in European standard is fitted for applying classical methods, nevertheless using of FE method also is possible. Just one disadvantage is that in some cases FE users have manually finishes the evaluation of welded connections strength taking the forces acting in welded joints. Bonded structures can be of two types based on either purely adhesive or an adhesive/mechanical connections. The bonded/mechanical types include bonded welded, bonded riveted and bonded-screwed connections [13].

FEA shell element models are effective for predicting loads in weldments fabricated from plate, sheet, structural shapes and tube. The formulation used for a FE shell model is that of full penetration welds at every joint. Although the loads carried through joints are calculated by FEA, they are not readily presentable.

Finite Element code *SolidWorks Simulation 2010* has been used to simulate behavior of statically loaded welded specimens and to determinate weld loads.

The aim of this work is to validate the FE models of welded joints comparing the results with experimental data of specimens with relatively simple geometries and compare the design resistance evaluation of welded connections simulated using 3D solid models and shell models. Also in the article using the calculated weld loads from the FEA 3D solid models simulation results the design resistance of welded connection according the requirements of European Standard EN 1993-1-8 has been evaluated.

2. Experimental research and results

The quasi static tension tests were carrying out on the universal hydraulic 5 t tension-compression testing machine. Flat square butt welding with small penetration depth (Fig. 1) has been tested in order to obtain the fracture in the welding instead of blank steel. The tension curves of welded material (Steel S275JR) and two types of welded samples (welding electrodes OK 46.30) single sided welding and double sided welding has been obtained (Fig. 2). The experimental data has been used to validate the numerical models.

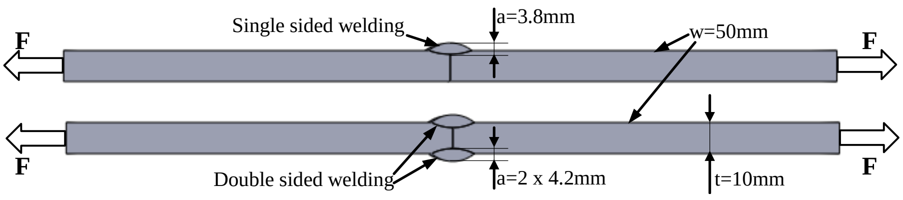

Fig. 1. Single and double sided flat square butt welded specimens with small penetration depth

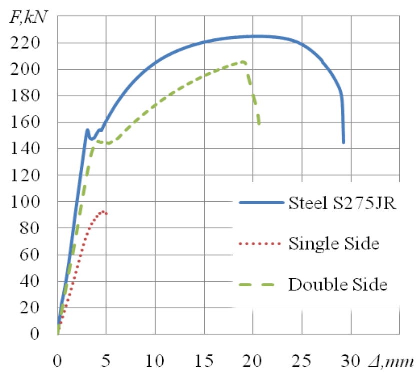

Fig. 2 Tension curves of welded material (Steel S275JR) and, single and double side flat square butt welded specimens

Mechanical properties obtained from the quasi static tension tests are presented in table.

Table

Mechanical properties of analyzed materials

Mechanical property	Yield strength, MPa (Requirement)	Tensile strength, MPa (Requirement)	Elongation, % (Requirement)	Impact energy at 0°, J/cm^2 (Requirement)
Steel S275JR	300 (>275)	450 (>430)	28 (>23)	(>54)
Electrode OK 46.30	430 (>380)	490 (470÷600)	(>22)	(>60)

The quasi static bending tests for fillet – single sided and fillet – double sided T shape welded specimens (Fig. 3) were performed according the conditions described in standard: Destructive tests on welds in metallic materials; fracture tests EN 1320 [14] in order to get experimental data for validation of FE models.

The tests were carrying out on the universal hydraulic 5 t tension-compression testing machine, applying the axial load through flat end platens without any additional fixing. The experimental results of the bending tests are presented in Figure 3 c.

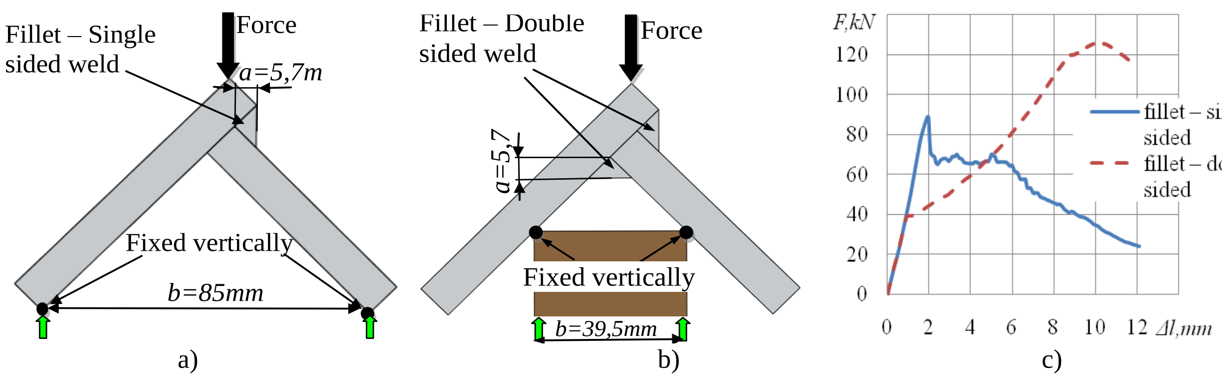

Fig. 3 Quasi-static bending tests for T shape welded specimens a) fillet – single sided and b) fillet – double sided c) force-deflection curves of bending tests

First has been tested fillet – single side welded specimens. Later applying the same testing conditions for the fillet – double side welded specimens the plastic hinge in the steel plates has been observed. Therefore to get data about welded connections, the fixing of the bended specimen has been changed.

3. Simulation and results

FE package SolidWokrs Simulation has been used to evaluate the strength of the welded joints. The software allows simulating welded connection using 3D solid or shell models. Choosing the type of model depends on structure complexity. For more complex structures to avoid the convergence problems it is recommended to use shell model. The geometry of the tested specimens has been modeled using 3D solid and shell models. Shell models have been build from the middle surfaces of the solids.

For the modeling of the welded connections in 3D solid model, the welded faces have to belong to different bodies. With 3D solid in *SolidWorks Simulation 2010* can be modeled only fillet weld bead (Full length, intermittent and staggered). Then in *SolidWorks* using *Weldments* command *Fillet Bead* we have to add the fillet weld bead feature between two disjoint bodies. After it, in *SolidWorks Simulation* the fillet bead body can be simulated using solid or beam elements. In case of solid elements, the design resistance of weld can be evaluated using common strength criteria eg. Von Mises yield criterion. If we choose fillet weld bead simulation using the beam element the program automatically according the fillet bead geometry calculates section properties of beam and converts fillet bead in to the beam elements. In this case the strength of weld can be evaluated using the same common strength criteria for beam elements (Fig. 4) or by taking beam force from simulation results we can manually calculate the strength of welded connection according to requirements of Euro Code 3 part 8.

The adequate FE models of bending test specimens has been build using 3D Solid modeling. The fillet weld bead has been modeled using beam elements and simulated adequately test conditions (Fig. 4). The design resistance of welded connection has been evaluated using calculated beam forces and procedure based on requirements of Euro Code 3 part 8. The applied force for fillet single-side and fillet double-side was taken from experiments $F = 40\ \text{kN}$.

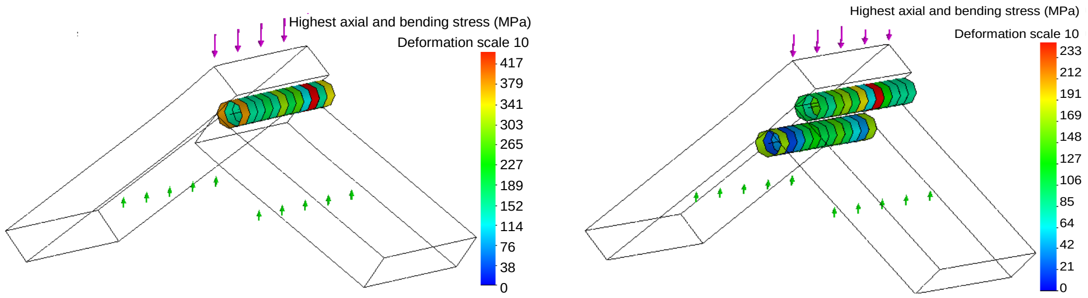

Fig. 4. Simulation results using 3D solid models with beam elements: a) Fillet single-sided, b) Fillet double-sided

The shell models for the simulation of fillet weld bead have been build from the middle surfaces of the solids (Fig. 5). In *SolidWorks* from the 3D Solid model using *Surface* menu command *Middle surface* it is possible automatically to create the surface model and in *SolidWorks Simulation* environment build shell model. The gaps obtained using *middle surface* command is normal phenomenon and should not be filled using *Extend surface* command. Removing the gaps can negatively influence to the results. To simulate weld in the *SolidWorks Simulation 2010* environment has to be choose *Edge weld connection* where as options can be groove, fillet or spot welds. Creating the fillet weld first we have to select the terminated surface and in second place select surface to which is welded (only perpendicular surfaces can be welded). To load adequately test conditions in the shell model the force $F = 40\ \text{kN}$ has been applied by *Remote loading*.

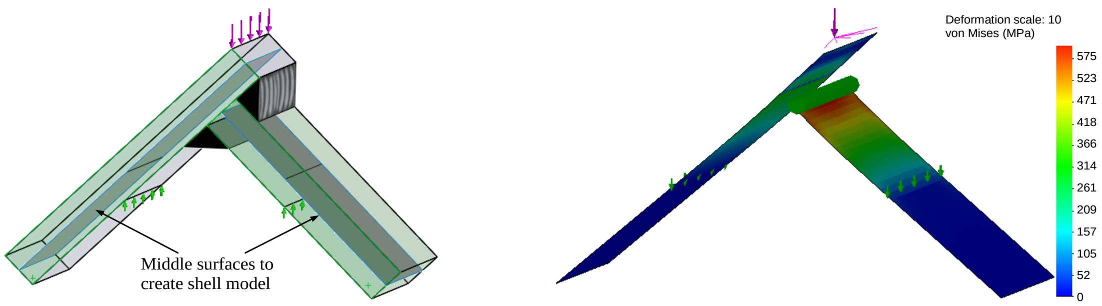

Fig. 5. Shell models for the simulation of fillet weld: a) Fillet single sided, b) Fillet double sided

The loads in welded joints calculated by FEA are not readily presentable but program automatically according the electrode and welded material strength properties estimates the needed weld size. Using 3D Solid modeling and beam elements to simulate fillet weld bead, the design resistance of welded connections according to requirements of Euro Code 3 part 8 can be evaluated by taking beam forces (Fig. 6) from FEA results.

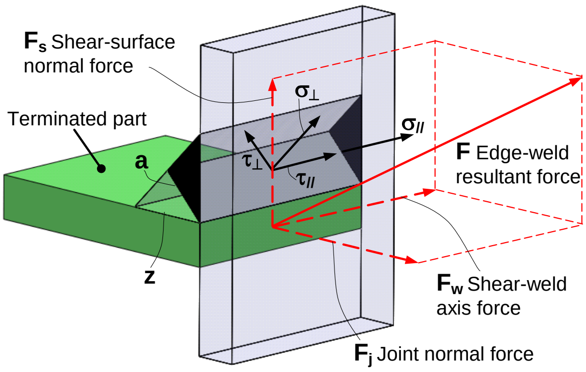

Fig. 6. Stresses on the throat section of a fillet weld and forces on the welded joint

Considering Euro Code 3 Part 1-8 [12] the design resistance of fillet welds will be sufficient if the following are both satisfied:

$$\sqrt{\sigma_{\perp}^{2} + 3\left(\tau_{\perp}^{2} + \tau_{//}^{2}\right)} \leq \frac{f_u}{\beta_v \gamma_{M2}} \quad \text{and} \quad \sigma_{\perp} \leq \frac{f_u}{\gamma_{M2}} \tag{1}$$

where f_u - is the nominal ultimate tensile strength of the weaker part joined $f_u = 430\ \text{MPa}$; β_v - is the appropriate correlation factor, for S275JR $\beta_v = 0.85$ [12]; γ_{M2} - is recommended resistance for connections $\gamma_{M2} = 1.25$ [12].

Design resistance of welded connections has been estimated using simplified method for design resistance of fillet weld. Independent of the orientation of the weld throat plane to the applied force, the design resistance per unit length $F_{w,Rd}$ at every point along the filled weld length should be bigger then the resultant of all the forces per unit length transmitted by the weld and satisfies the following criterion:

$$F_{w,Ed} \leq F_{w,Rd} = f_{vw,d} \cdot a \tag{2}$$

where: $F_{w,Ed}$ - is the design value of the weld force per unit length; $F_{w,Rd}$ - is the design weld resistance per unit length; $f_{vw,d}$ - is the design shear strength of the weld; a - effective throat thickness.

The design shear strength of the weld $f_{vw,d}$ should be determined from (1):

$$f_{vw,d} = \frac{f_u}{\sqrt{3}\beta_w \gamma_{M2}} = \frac{430}{\sqrt{3} \cdot 0.85 \cdot 1.25} = 233\ \text{MPa} \tag{3}$$

The software calculates the weld size and weld throat size for all mesh nodes on the intersecting edge of the terminated part. First calculates the resultant force components per unit length L_n for each node on the intersecting edge of the terminated part. The resultant forces per unit length on the weld, $F_{w,Ed}$ (Fig. 6):

$$F_{w,Ed} = \sqrt{F_s^2 + F_w^2 + F_j^2} \tag{4}$$

where: F_s - shear surface normal force; F_w - shear weld axis force; F_j - joint normal force.

The required weld throat thickness a and weld size z for each node on the intersecting edge of the terminated part was calculated by setting $F_{w,Ed} = F_{w,Rd}$. For double-sided welds:

$$a = \frac{F_{w,Ed}}{2 f_{vw,d}} \quad \text{and} \quad z = \frac{a}{\sin 45^{\circ}} \tag{5}$$

The results of calculated effective weld throat thickness for 3D Solid and shell models are presented in Figure 7.

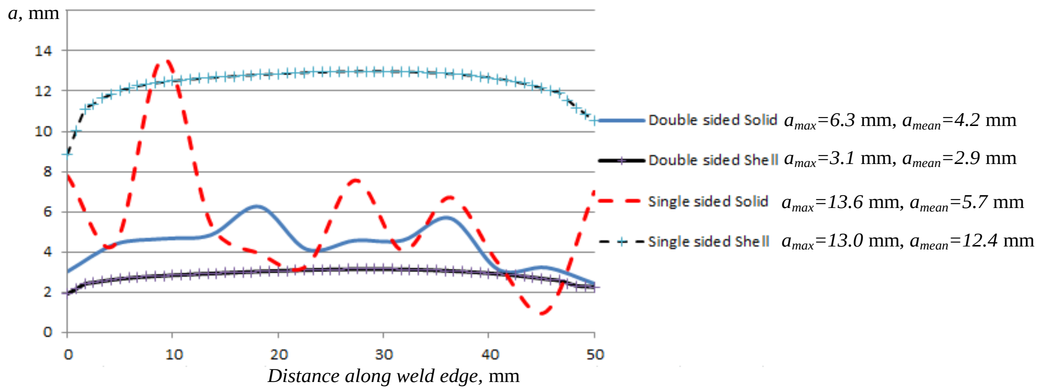

Fig. 7 Calculated effective weld throat thickness for 3D Solid and shell models

4. Conclusions

Accurate determination of weld loads and fast estimation of weld throat requirement or stress levels shows that the SolidWorks simulation package is useful for evaluation of welded joints resistance.

The effective weld throat thickness calculated using the 3D Solid models with beam elements are very variable and not symmetrical (the model was loaded symetrically). Also the simulation process has been performed not very fluently. The *no penetration contact* in 3D Solid models has not worked properly and therefore the *Global bonded contact* has been activated. Because of this fact the FE model of 3D Solid has not been identically adequate to the experimental one. The mean and maximum values of effective weld throat thickness calculated using shell models differs very little, while in 3D solid models it differs more then 50%.

From the experiments obtained force versus displacement curves and fracture forces unfortunately can not be simulated in SolidWorks Simulation software because of large plastic deformations and absence of fracture modeling possibilities. The comparison has been done taking in to account mode of deformation and fact that the 40kN force initiated the plastic deformation in welded plates and this can be simulation results when the stress exceeds the yield strength of welded plates.

References

1. **Mackwood, A.P., Crafer, R.C.,** Thermal modelling of laser welding and related processes: a literature review. Optics & Laser Technology, 2005, (37), p 99 – 115.
2. **Lindgren L.-E.,** Numerical modelling of welding. Comput. Methods Appl. Mech. Engrg., 2006, (195), p 6710–6736.
3. **Ueda Y., Yamakawa T.,** Analysis of thermal elastic-plastic stress and stain during welding by finite element method. JAWRI, 1971, 2 (2), p 90-100.
4. **Hibbit H.D., Marcal P.V.,** A numerical, Thermo-Mechanical Model of the Welding and Subsequent Loading of a Fabriculated Structure. Computers and Structutes, 1973, vol. 3, p.1145-1174.
5. **Koc M., Altan T.,** Application of two dimensional (2D) FEA for the tube hydroforming process. International Journal of Machine Tools & Manufacture, 2002, (42), p 1285–1295.
6. **Lie S.T., Lee C.K., Wong S.M.,** Modelling and mesh generation of weld profile in tubular Y-joint. Journal of Constructional Steel Research, 2001, (57), p. 547–567.
7. **YongXia, QingZhou, Wang P.C., Johnson N.L., Gayden X.Q., Fickes J.D.,** Development of a high-efficiency modeling technique for weld-bonded steel joints in vehicle structures, Part II: Dynamic experiments and simulations. International Journal of Adhesion & Adhesives, 2009, (29), p 427–433.
8. **Al-Samhan A., Darwish S.M.H.**, Finite element modeling of weld-bonded joints, Journal of Materials Processing Technology, 2003, (142), p 587–598.
9. **Scott D. Raymond, Peter M. Wild, Christopher J. Bayley,** On modeling of the weld line in finite element analyses of tailor-welded blank forming operations. Journal of Materials Processing Technology, 2004, (147), p 28–37.
10. **Klobčar D., Tušek J., Taljat B.,** Finite element modeling of GTA weld surfacing applied to hot-work tooling. Computational Materials Science, 2004, (31), p 368–378.
11. **Bražėnas, A., Daunys, M., Vaičiulis, D.,** Deformation and fracture resistance of butt-welded joints with mild interlayer under elastoplastic tension. // International Applied Mechanics. New York : Springer Science and Business Media Deutschland GmbH.. ISSN 1063-7095. 2006, Vol. 42, no. 4, p. 119-127.
12. European Sstandard, Eurocode 3: Design of steel structures - Part 1-8: Design of joints, EN 1993-1-8, 130 p.
13. **Weaver M. A.,** Determination of Weld Loads and Throat Requirements Using Finite Element Analysis with Shell Element Models – A Comprision with Classical Analysis.
14. Destructive tests on welds in metallic materials. Fracture tests. EN 1320:1996 18 p.