MODELING OF SINGLE-FILLET LAP JOINT IN SOLIDWORKS SIMULATION

Filip Zdraveski
Ss. Cyril and Methodius University, Faculty of Mechanical Engineering, Macedonia
Zoran Bogatinoski
Ss. Cyril and Methodius University, Faculty of Mechanical Engineering, Macedonia
Bojana Trajanoska
Ss. Cyril and Methodius University, Faculty of Mechanical Engineering, Macedonia

The aim of the work presented in this paper is to show the applicability of Solidworks Simulation module for calculation of welded structural joints. The presentation is performed through a simple model of axially loaded single-fillet lap joint. Solidworks Simulation offers several options for FEM analysis of welded joints. All reasonable options are exploited and their results are compared in this paper. The reference values are used to validate the results of FEM analysis. They are calculated based on formulas described in Eurocode 3 part 8 which refer to design resistance of fillet welds. FEM analysis is conducted in latest version of Solidworks Simulation 2015 and older version 2012. Results obtained in both versions differ from each other under same starting conditions. Finally, the paper provides recommendations for reliability of the Solidworks Simulation to be used as design tool for calculating welded joints.

Key words: Welded structure, Design resistance of welded connections, Fillet weld, Finite element method (FEM)

INTRODUCTION

Generally weld connections can be analysed using clasical analysis [01] or Finite Element Method (FEM). Finite element analysis (FEA) quite accurately iden-tifies the load path [02], which can be difficult using classical analysis in the case of complex structures.

FE package SolidWorks Simulation has been used to evaluate the strength of the welded joints [03, 04, 05]. The software allows simulat-ing welded connection using 3D solid or shell models. Choosing the type of model depends on structure complexity. For more complex struc-tures to avoid the convergence problems it is recommended to use shell model. The geometry of the tested specimen has been modeled using 3D solid and shell model. Shell model have been build from the middle surfaces of the solids.

Case study is conducted over the single-fillet lap joint shown on Figure 1. The joint is modeled using Weldments command in Solidworks. Weldment structure is designed as single multibody part.

Using Weldments command Fillet Bead, fillet weld bead feature is created between two dis-joint bodies. Connection between welded bod-ies and the fillet bead is established simply by choosing Bonded contact pair.

In SolidWorks Simulation the fillet bead body can be simulated using solid or beam elements. In case of solid elements Figure 2, the design re-sistance of weld can be evaluated using common strength criteria eg. Von Mises yield criterion.
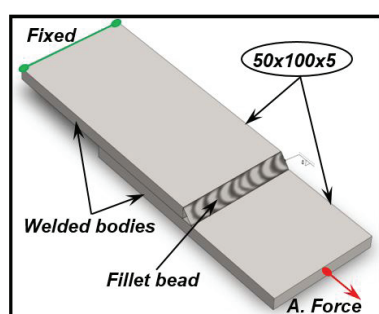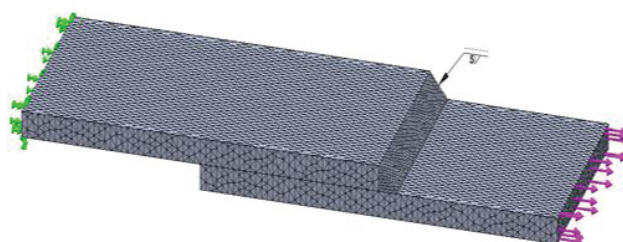

Figure 1: Single-fillet lap joint

Figure 2: Using solid element for fillet bead

* Ss. Cyril and Methodius University, Faculty of Mechanical Engineering, Karpos II bb, 1000 Skopje;
filip.zdraveski@mf.edu.mk

When fillet weld bead simulation is performed by using beam element, the program automatically according the fillet bead geometry calculates section properties of beam and converts fillet bead in to the beam elements Figure 3.
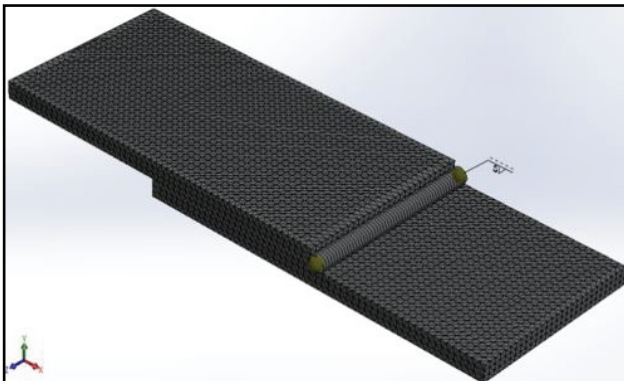

Figure 3: Using beam element for fillet bead

In this case the strength of weld can be evaluated using the same common strength criteria for beam elements.

The shell models for the simulation of fillet weld bead have been build from the middle surfaces of the solids shown on Figure 4. Using Surface menu command Middle surface it is possible automatically to create the surface model and in Solid-Works 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 environment has to be choose Edge weld connection where as options can be groove, fillet or spot welds.

Fillet weld is created by selecting the terminated surface and the surface to which is welded. It can only be applied to perpendicular surfaces. Therefore, auxiliary surface is added to one of the middle surfaces which is shown on Figure 4. The auxiliary surface should be very short to avoid its influence (increasing the stiffness).

The axial force equal to 10 kN has been applied as shown on Figure 1.

STATIC CASE STUDY SIMULATION

As already explained, there are three main ways to simulate welded joint in Solidworks simulation: (1) all bodies to be solids, (2) the fillet weld bead to be beam while other bodies solids, and (3) instead of bodies to use surfaces and Edge

weld connection command.

When modeled in the first way, Mesh control is applied for the fillet weld bead. General mesh parametars are given on Figure 5, while mesh control is shown on Figure 6. It can be concluded that the size of the mesh elements should be at least half of the thickness of the welded bodies. Mesh control should reduce the size of the weld bead elements to half of the general mesh.
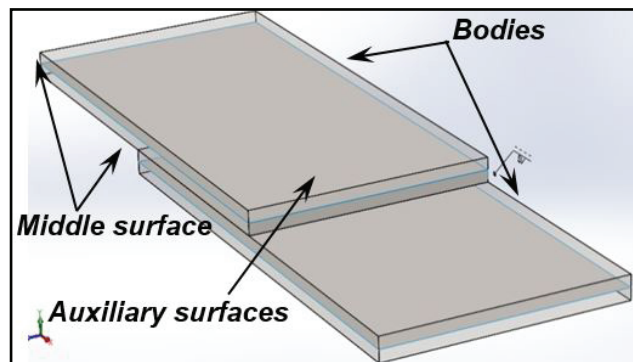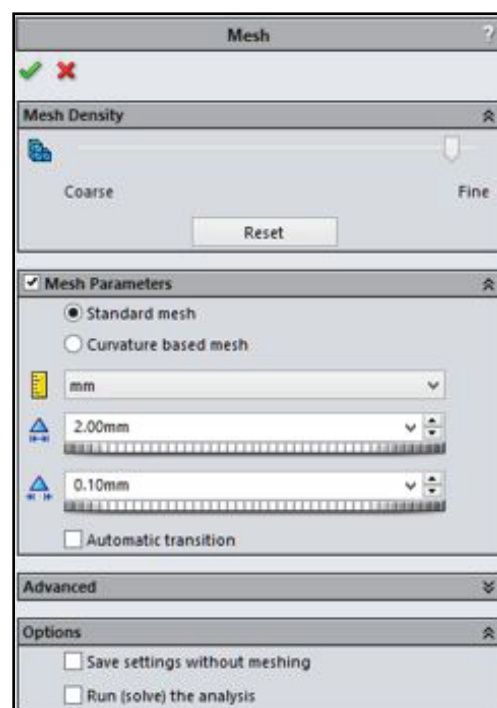

Figure 4: Middle surfaces from bodies

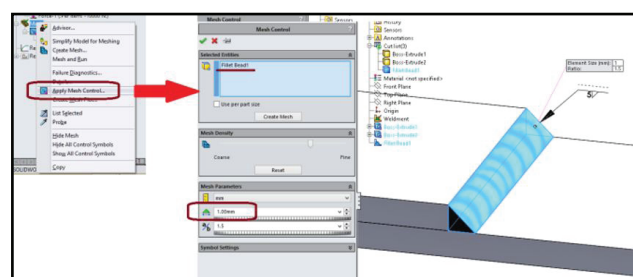*Figure 5: Creating mesh for fillet weld bead*

Figure 6: Using mesh control for fillet weld bead

Figure 7 shows the definition of bodies as solids. This is particularly important for the fillet weld bead if the feature is created with the Weldment command. In this case, the program automatically defines the weld feature as beam element (Figure 3). Contact sets between individual bodies are given in Figure 8. Global Contact is selected to be No penetration.
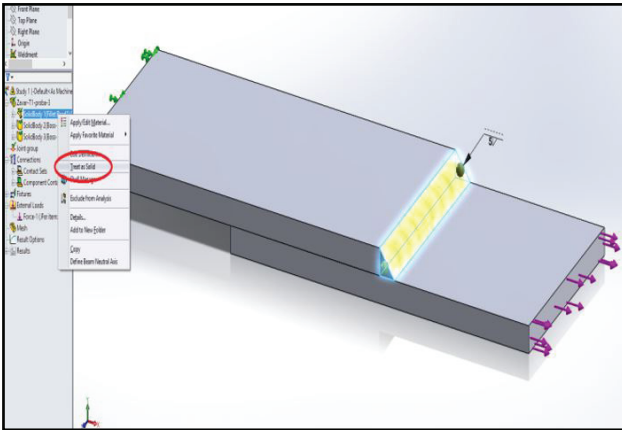

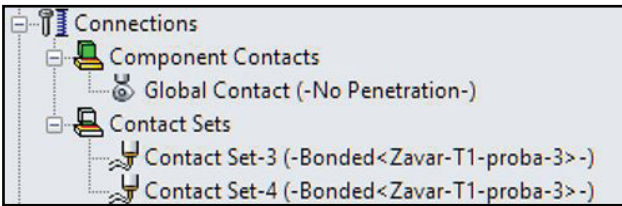*Figure 7: Defining all bodies as solids*

Figure 8: Defining contact sets between bodies

Second option to simulate weld joint is to use beam element.

Contact sets between beam and bodies are given in Figure 9. Global Contact is selected to be No penetration.
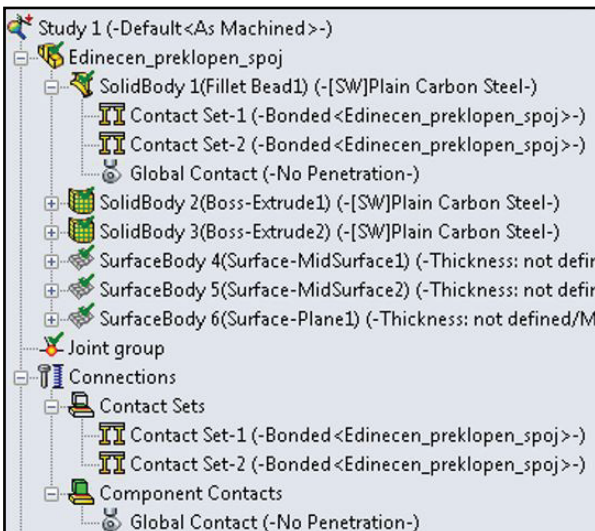

Figure 9: Contact sets between beam and bodies

In the simulation with beam element, mesh size is very important. As the beam is divided into smaller elements, more accurate results are obtained. Therefore, in the considered case, the beam is applied to finest mesh.

A third option that can be used for modeling of welded connection is Edge Weld Connector and its built-in tool of Solidworks Simulation specially designed for welds.

Figure 10 shows the option Edge weld connector and surfaces that are joined. Here the global contact set is selected Bonded because surfaces are separated with gap. This is shown on Figure 11.

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. This is shown on Figure 12 and 13.
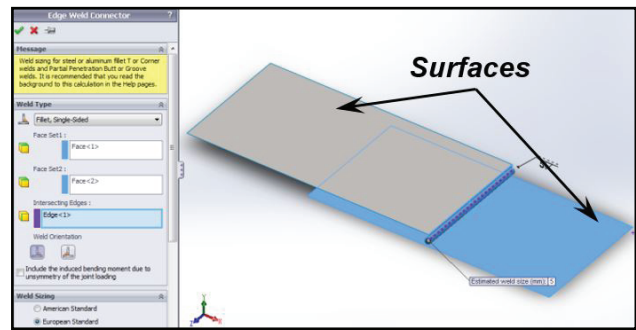

Figure 10: Edge weld connector for surfaces
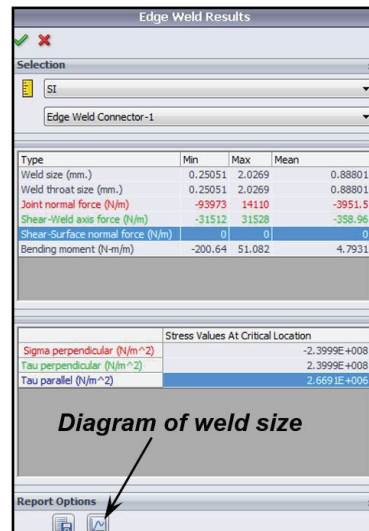

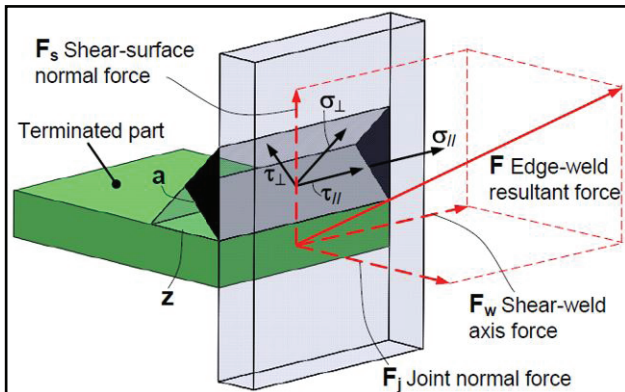*Figure 11: Contact sets between surfaces*

Figure 12: Edge weld results

Figure 13: Stresses and forces in a fillet weld

RESULTS FROM THE SIMULATION CASES

Taking into account what has been said so far about Solidworks Simulation, results of the first and second approach for FEA design resistance of fillet weld can be evaluated using common strength criteria eg. Von Mises yield criterion. This is presented on Figure 14 and 15. The results in Figure 14 refer to the solid weld bead, while the results in Figure 15 refer to the fillet weld modeled as beam.
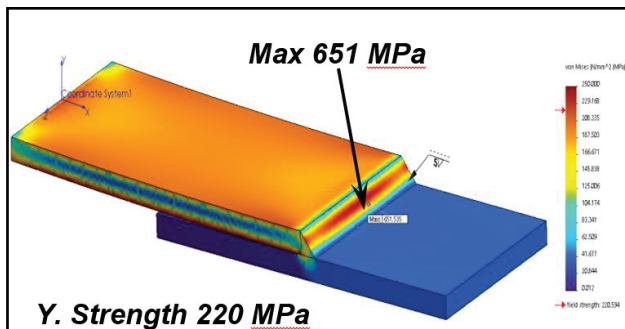

Figure 14: Von Mises criterion for 1st study case

When using Edge Weld Connector (EWC), results are given by calculated – estimated weld size. These are given on Figure 17.

Figure 16 shows the simplification that comes with using surfaces as way of modeling the FE structure. Its easy to separate the stresses (eg. axial, bending) which are caused by the load action pattern. This is done only by selecting the layer in which dominant stresses are included. Figure 16 shows only axial stress caused by the external acting force which is dominant in this case.

Calculated values for stresses are given on Figure 12 according to European Standard. Ultimate tensile strength of the welded joint is selected to be 600 MPa.

Another important note from the simulations is the difference in results that occurs between different versions of Solidworks Simulation.
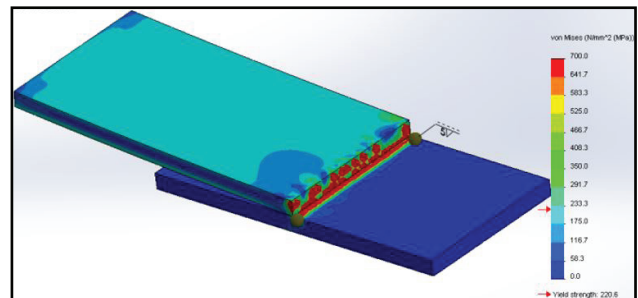

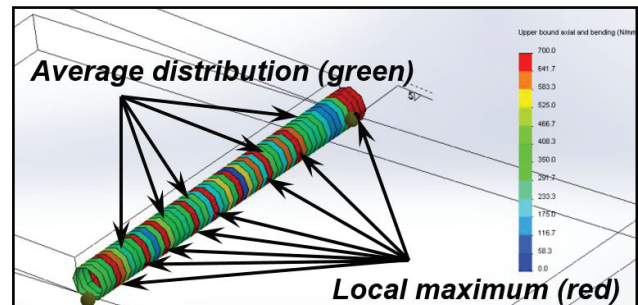Figure 15a: Von Mises for 2st study case (solids)

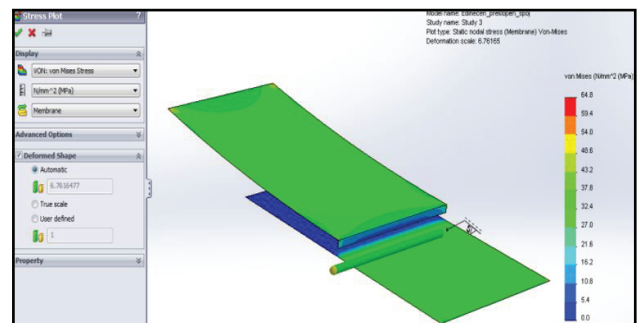Figure 15b: Von Mises for fillet weld bead (beam)

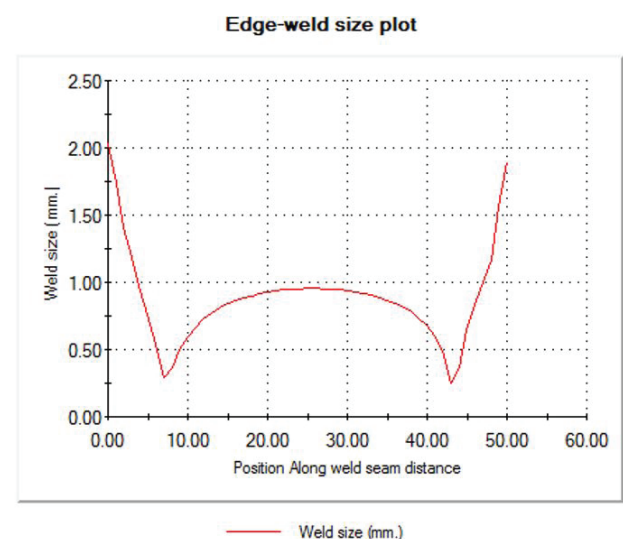Figure 16: Simplified stress plot for surfaces

Figure 17: Estimated weld size according EWC

The difference in results shows under same conditions of meshing, restraining and loading. Stress plot follows same pattern, but values are

lower in case of older version. This is presented on Figure 14 and 18.

The location of maximal value is on different place. In general, values in 2012 are lower than values received in Solidworks Simulation 2015. Also, older version gives only automatic transition of mesh between different bodies/parts bonded together.

Last remark can be seen when comparing Figure 2 (version 2015) with Figure 19. Figure 19 shows created mesh transition in version 2012 compared to version 2015.
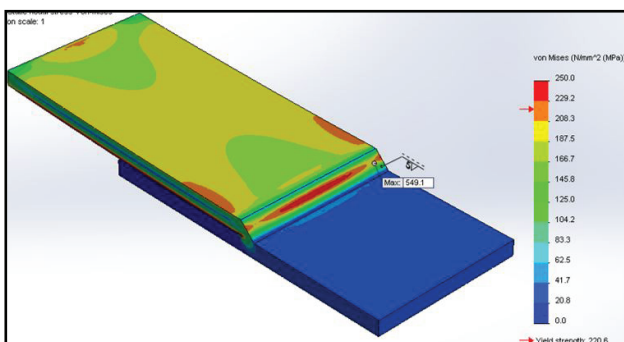

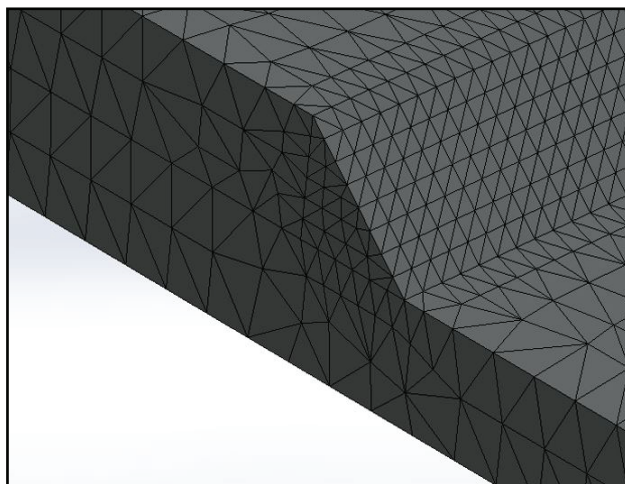*Figure 18: Von Mises for 1st case – 2012 version*

Figure 19: Automatic transition (version 2012)

COMMENTING ON RESULTS

In all three cases, simulations show the same pattern of distribution of load through bodies and fillet weld. However results vary depending on the approach that was used for modeling.

Most evident difference is seen with the second case when beam element is used to simulate the weld feature. Local maximums have very high values (Figure 15b) compared to other two cases (1 and 3). This values are way above the material strenght.

Another concern about the accuracy and therefore usability of the program is the difference in results in that come from different versions of Solildwork Simulation.

Based on the simulations, the second method is not recommended for modeling welded connection. The first and third approach gives acceptable results can be simply verified.

VERIFICATION OF RESULTS

Here, the first following section presents the procedure concerning the determination of resistance welded connection according Euro Code 3 Part 1-8 [6] and in another section calculated results are presented.

Design resistance of fillet weld EC 3, 1-8

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

$$\sqrt{\sigma_\perp^2 + 3(\tau_\perp^2 + \tau_{II}^2)} \le \frac{f_u}{\beta_v \, \gamma_{M2}} \quad and \quad \sigma_\perp \le \frac{f_u}{\gamma_{M2}} (1)$$

where u f_u - is the nominal ultimate tensile strength of the weaker part joined f_u=430 MPa; β_v – is the appropriate correlation factor, for S275JR β_v=0.85; γ_{M2} – is recommended resistance for connections γ_{M2}=1.25.

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:

$$F_{w,Ed} \le F_{w,Rd} = f_{vw,d} \cdot a \qquad (2)$$

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):

$$f_{vw,d} = \frac{f_u}{\sqrt{3} \, \beta_v \, \gamma_{M2}} = 233 \, MPa \qquad (3)$$

The resultant forces per unit length L_n on the weld, $F_{(w,Ed)}$ (Figure 13):

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

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 can be calculated by setting $F_{(w,Ed)}=F_{(w,Rd)}$. For double-sided welds:

$$a = \frac{F_{w,Ed}}{2 f_{vw,d}} \; and \; z = \frac{a}{\sin 45°} \qquad (5)$$

Verification results

Instead of calculating weld throat thickness, comparison is made only with stress caused by dominant axial force (Von Mises stresses – equation 1).

$$\sigma_{von\,Mises} = \sqrt{\sigma_\perp^2 + 3(\tau_\perp^2 + \tau_{II}^2)} = \sqrt{\sigma_\perp^2 + 3\,\tau_\perp^2} \qquad (6)$$

$$\sigma_\perp = \tau_\perp = \frac{F_{axial}}{\sqrt{2}\,a_z\,l_z} = \frac{10000}{\sqrt{2}\,3.5\,50} = 40.4 MPa \qquad (7)$$

$$\sigma_{von\,Mises} = \sqrt{\sigma_\perp^2 + 3\,\tau_\perp^2} = 80.4\,MPa \qquad (8)$$

It may be noted that the calculated value for stress in the welded joint is significantly different from the results obtained with FE analysis.

This can be explained by the fact that the standard does not predict turning moment that occurs in this asymmetrical welded connection [07, 08].

The calculation according EC standards neglects the existence of τ_{II}. On the other hand, FE analysis takes into account the distribution of force through the solid and this can be seen on Figure 14 – 18.

Given that Solidworks Simulation is a tool for linear analysis (up to yield point), and EC standard applies to linear behavior of welded joint, the decision about which method of modeling within Solidworks Simulation is the most accurate should be checked through experiment.

As an option for experimental analysis is the use of strain gauges in the form of rosettes given in Figure 20, which can determine the direction and intensity of deformation at a certain point of the test piece. Thus the results should be compared with FE analysis.
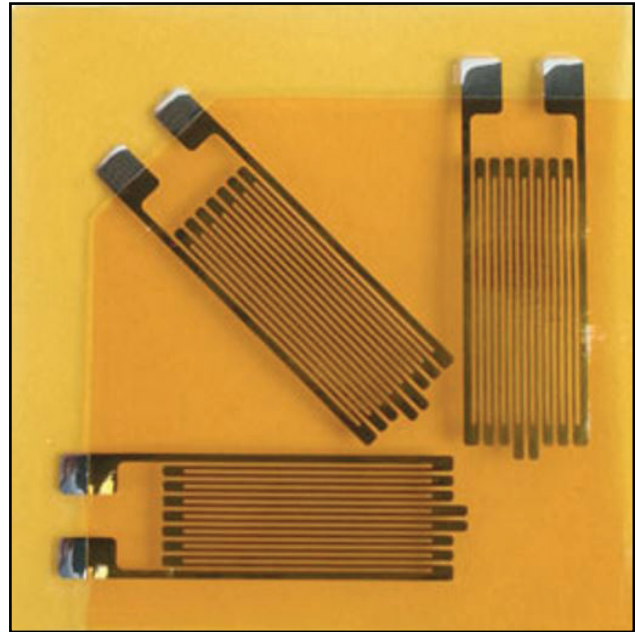

Figure 20: Rosettes strain gauges

CONCLUSION

The single–fillet lap joint treated in this paper is specific case because it is asymmetrically loaded welded joint. Its taken as an example on which it was conducted FE analysis as well as verification check according to Eurocode 3 part 8.

General conclusions can be summarized in the following list:

- Solidworks Simulation is an easy tool for fast calculation;
- There are three methods by which it can be modeled welded joint;
- There is a difference in the results depending on which version of the program used;
- Most anticipated results were obtained under the first method shown in Figure 14;
- The existence of local maxima with a value above the yield poin clearly shows that certain points in the welded joint are above the linear limit;
- The difference that appears in results between different versions gives doubt about the usefulness of the program;
- EC 3 part 3 do not describe the case of unsymmetrical load, so get significantly lower values of stresses in the weld which can't be use for comparison;

Finally, based on the research of this paper, its recommended conducting experimental test for

verification of the accuracy of one of the 3 methods provided within Solidworks Simulation.

REFERENCES

1) Al-Samhan A., Darwish S.M.H., Finite element modeling of weld-bonded joints, Journal of Materials Processing Technology, 2003, (142), p 587–598.

2) EN 1993-1-8 (2005) : Eurocode 3: Design of steel structures - Part 1-8: Design of joints

3) J.M Mínguez, J. Vogwell, The stress distribution over open and closed welded joints, Engineering Failure Analysis, 10 (1), 2003, p 103-111

4) K.S. Virdi, F.K. Garas, at al., Structural Assessment: The Role of Large and Full-Scale Testing, CRC Press, 1999

5) Michael A. Weaver, P.E., Determination of weld loads and throat requirments using finite elemt analysis with shell element models – a comparison with classical analysis, Welding Journal, 1999, 78(4)

6) O. Doerk, W. Fricke , C. Weissenborn, Comparison of different calculation methods for structural stresses at welded joints, International Journal of Fatigue, 25 (2003), pp 359–369

7) P.Griskevicius, M.Urbas, V.Capas, A. Kozlovas, Modeling of welded connections in SolidWorks Simulation, Proceedings of 16th International Conference. Mechanika, 2011

8) R.N. Mhetre, S.G. Jadhav, Finite element analysis of welded joints, International Journal of Instrumentation, Control and Automation (IJICA), 1(3,4), 2012