DOI:10.5937/IIZS25083R

STRESS ANALYSIS OF WELDED JOINT BEHAVIOR UNDER TENSILE LOADING USING FEM

Aleksandar Rajić[1], Dušan Jovanić[1], Ljubica Lazić Vulićević[1]
[1]Technical College of Applied Sciences in Zrenjanin, Serbia
e-mail: aleksandarrajic@yahoo.com

Abstract: The strength assessment of welded joints can be carried out through traditional analytical approaches or by employing the finite element method (FEM). This paper presents the modeling process of a butt-welded joint using SolidWorks Simulation software, along with an analysis of its behavior under load. Prior to the simulation, the tensile strength of a butt-welded aluminum specimen (Al 99.5) was experimentally determined. Since fractures occurred in the base material rather than in the weld, the simulation utilized material properties from the SolidWorks database corresponding to the base metal. The findings obtained through both experimental testing and numerical simulation offer practical insights and guidelines for evaluating the strength of welded joints.
Keywords: welded joints, structural strength, butt weld, finite element analysis, SolidWorks Simulation

INTRODUCTION

Welding is one of the main manufacturing processes used for assembling metallic and some non-metallic materials. In the welding process, high temperature differences result in significant thermal stresses, which affect the distribution of residual stresses in structural components.The significant time consumption and high costs of conducting experiments to determine the optimal parameters of the manufacturing process serve as a motivation to include the use of software simulations in component design, as well as in their production, which enables the optimization of the manufacturing process. The implementation of microstructural effects that develop during welding into simulation software represents a major advancement in such simulations. Computational welding mechanics is a modeling process used to predict the thermal, mechanical, and structural properties of materials affected by the welding process [1].
Welded joints can be analyzed using classical 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 [7].
This paper presents the modeling procedure of a butt-welded joint using SolidWorks Simulation software and its behavior under loading. The software allows for the simulation of welded joints using 3D solid or shell models. The choice of model type depends on the complexity of the structure. To avoid convergence issues, the use of shell models is recommended for more complex structures [8].
For the purpose of the simulation, the tensile strength of a butt-welded joint made from aluminum plates (Al 99.5) was previously determined from experimental testing. Tensile test specimens were prepared, and since fracture occurred in the base material, the simulation used material data from the SolidWorks Simulation material database for the base material. The research results and corresponding conclusions, obtained through experimental analysis and computer simulation, can be used as recommendations regarding the reliability of the SolidWorks Simulation software for the analysis of welded joints.

MATERIAL AND METHODS

Experimental testing

For the purpose of the simulation, the tensile strength of a butt-welded joint was previously determined using aluminum (Al 99.5) specimens welded by the MIG welding process. Tensile test specimens were prepared from the welded plate in accordance with the relevant standard [2]. The prepared specimens are shown in Figure 1, and in Figure 2, the fractured specimen after tensile testing in the testing machine.

Fig. 1. *The prepared specimens for testing*

Fig. 2. *The fractured specimen after tensile testing*

The tensile strength obtained from the experiment is R_{mN} = 81,165 N/mm², and since the fracture occurred in the base material, the simulation used material data from the SolidWorks Simulation software database for the base material. The data provided by the manufacturer includes a full range of values for tensile strength and yield strength for Al 99.5, depending on the treatment condition, and one of them is shown in Table 1, [10].

Table 1. *Values for tensile strength and yield strength for one case of treatment condition*

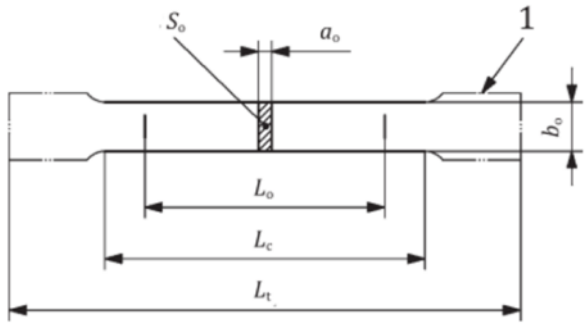
Designation	**Standard**	**Treatment condition**	**Tensile strength R_m (N/mm²)**	**Yield strength $R_{p0.2}$ (N/mm²)**
EN AW-1050A (Al 99.5)	EN 485	O/H111	65 – 95	20

The experimentally obtained R_m value fits within the specified interval.

Numerical simulation of statically loaded welded specimens.

The SolidWorks Simulation module was used to assess the strength of welded joints. The geometry of the tested specimens was modeled using 3D solid and shell models. Since sheet metal components were considered, the Sheet Metal module was applied for creating sheet parts. For sheet components, the mesh is automatically generated on the mid-surface, with the shell thickness extracted from the actual sheet thickness. When creating welded joints in either 3D solid or shell models, the welded surfaces must belong to different bodies in the CAD model.

Figure 3 shows the shape and dimensions of the specimen, while Figure 4 presents the axonometric view. The specimen thickness is a_o = 3 mm, the width b_o = 20 mm, and the total length L_t = 300 mm. The butt-welded V-joint has a weld thickness equal to the sheet thickness s = a_o = 3 mm. The loading was performed for several values of the axial tensile force F = 3.750 [N], 4.000 [N], 4.250 [N], 4.445 [N], 4.750 [N], 4.870 [N], 5.285 [N].



Width b_o	Original gauge lenght L_o	Minimum parallel lenght L_c	Approximately total lenght L_t
40 ± 0,7	200	220	450
25 ± 0,7	200	212,5	450
20 ± 0,5	80	90	300

Fig. 3. *Technical drawing of the specimen with parameter values*

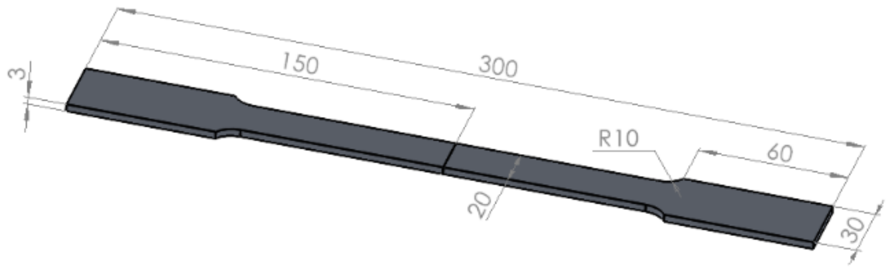


Fig. 4. *Axonometric view of the specimen with the specified dimensions*

For the purpose of the case study, a 3D model was created in the Assembly mode of SolidWorks 2023. In the SolidWorks Simulation module, supports (i.e., fixed surfaces) and external loads, namely the tensile force F, were defined (Figure 5).

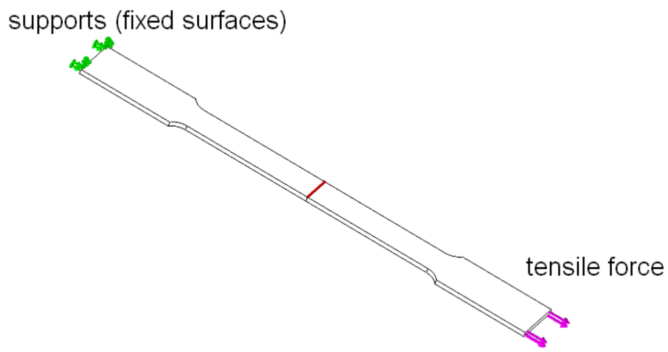


Fig. 5. *3D model of the specimen created in Assembly mode, showing the supports and the direction of the applied tensile force F*

For the specimen material, a material was selected from the software’s material database, namely: Aluminium Alloys 1060-H16, with tensile strength R_{mN} = 110 N/mm^2, and yield stress R_{eN} = 105 N/mm^2. The electrode material is welding wire AlMg5, with tensile strength R_{mN} = 250 N/mm^2, and yield stress R_{eN} = 120 N/mm^2.
For the purpose of the numerical simulation, among several possible methods for static analysis of a welded joint using the finite element method, the Edge Weld Connector tool was selected, which is specifically designed for the calculation of welded joints in SolidWorks Simulation.
By applying this tool, the software automatically analyzes stresses (Von Mises stress), strains (Strain), displacements (Displacement), safety factors (Factor of Safety, FOS), as well as the required weld size (Weld Check Plot), based on the characteristics of the base material, electrode, and other relevant parameters.
The selection of the electrode is performed in the Edge Weld Connector dialog box with different electrode material definitions:
(a) European standard – electrode material equal to the weaker material of the welded joint,
(b) European standard – user-defined electrode material, and
(c) American standard – electrode material defined from the software material database.
In addition, the option for a single-sided butt V-groove weld was selected. The contact surfaces and weld edges were defined, as well as the default values for the correlation factor β_ω = 1 and the partial safety factor 1.25. The designed weld width is s = 3 mm.
Figure 6 shows a section of the generated finite element shell mesh with predefined element dimensions and density, see Table 2.

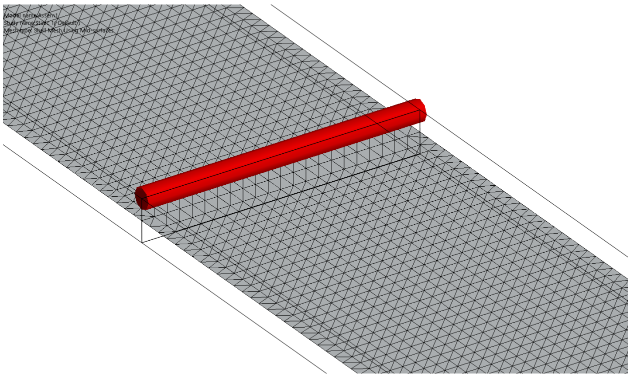


Fig. 6. *Shell mesh generated on the mid-surface of the specimen*

Table 2. *Mesh characteristics*

Mesh type	Shell mesh on the mid-surface
Mesh used	Standard mesh
Jacobian points for High quality mesh	4 points
Element size	3,647 mm
Tolerance	0,182 mm
Mesh quality	High
Number of nodes	35.631
Number of elements	17.446

After completing the finite element mesh generation process, the analysis was performed according to the specified parameters and loads, and the results of the static analysis of the welded joint were obtained. The results for numerical simulation no. 4 are presented in Figures 7–10.

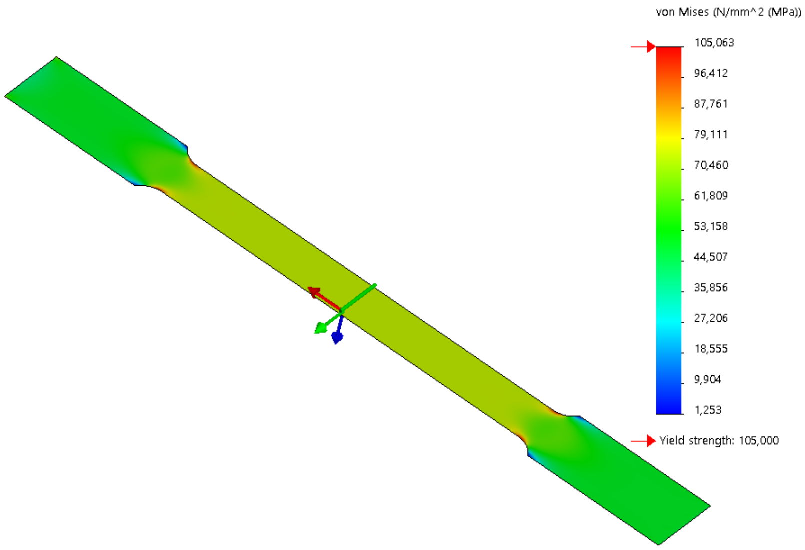


Fig. 7. *Von Mises stress distribution for the numerical simulation no. 4 (F = 4.445 N)*

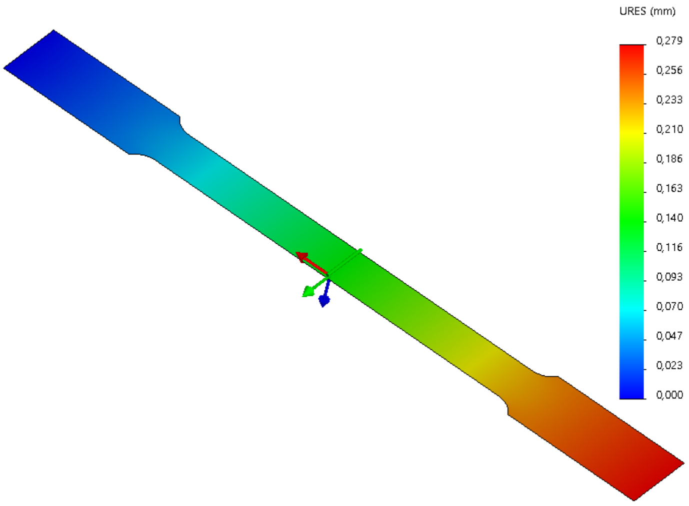


Fig. 8. *Displacement distribution for the numerical simulation no. 4 (F = 4.445 N)*

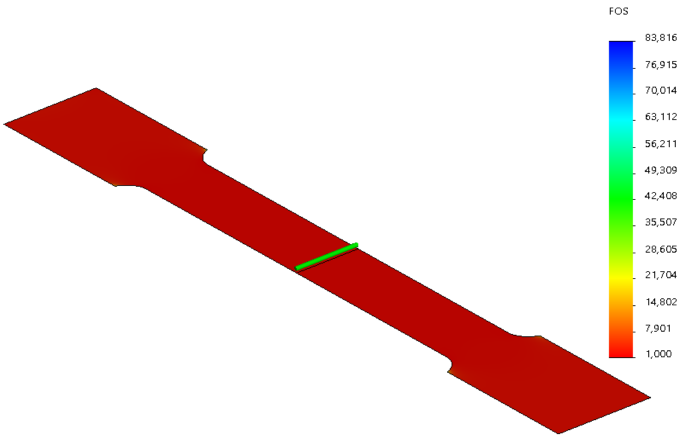


Fig. 9. *Factor of Safety for the numerical simulation no. 4 (F = 4.445 N)*

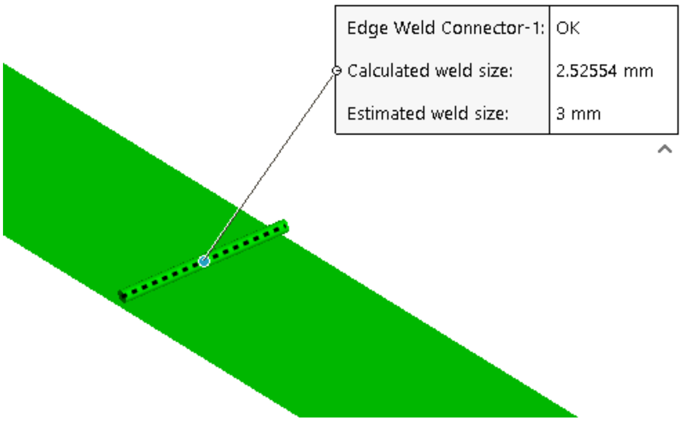


Fig. 10. *Weld size verification for the numerical simulation no. 4 (F = 4.445 N)*

RESULTS AND DISCUSSION

Based on numerical simulations with seven different values of the tensile force F = 3.750 [N], 4.000 [N], 4.250 [N], 4.445 [N], 4.750 [N] 4.870 [N], 5.285 [N] the results were obtained and are presented in Table 2.
The results of the numerical simulation indicate that the maximum allowable force is F = 4.445 N, with a safety factor of 1. The tensile stress at that point is 105,063 N/mm², and the displacement (i.e., elongation) is 0,279 mm. The weld size check (2,525 mm < 3 mm) indicates that failure would occur in the base material. A comparison of the results obtained from the experiment and the numerical simulation is presented in Table 3.

Table 3. *Presentation of results obtained from numerical simulations with different values of the axial tensile force F*

Simulation no.	**F** [N]	**Von Mises Stress** [MPa]	**Displacement** [mm]	**Factor of Safety, FOS**	**Weld Check Plot** [mm]
1	3.750	88,636	0,236	1,185	2,130 < 3
2	4.000	94,545	0,251	1,111	2,272 < 3
3	4.250	100,454	0,267	1,045	2,414 < 3
4	4.445	105,063	0,279	1	2,525 < 3
5	4.750	112,272	0,299	0,935	2,698 < 3
6	4.870	115,108	0,306	0,912	2,767 < 3
7	5.285	124,917	0,332	0,841	3,002 > 3

By observing the results in Table 3, it can be seen that failure occurs in the base material during the 5th simulation, at a tensile force of 4.750 N, which is very likely realistic, as fracture occurred experimentally at 4.869,9 N (factor of safety is 0,935). It can also be concluded that if the base material had not failed in simulations 5 or 6, failure would have occurred in the weld metal during the 7th simulation, at a force of 5.285 N (weld check plot is 3,002 > 3).

Table 4. *Comparison of results obtained from experiment and numerical simulation*

Parameters	**Values**
Tensile force F obtained from the experiment [N]	4.869,9
Tensile force F obtained from the numerical simulation [N]	4.445
Tensile strength R_m obtained from the experiment [N/mm^2]	81,165
Von Mises stress obtained from the numerical simulation [N/mm^2]	105,063

By analyzing the regions where the maximum Von Mises stress values obtained from the simulation occur, it can be observed that these are the areas of the specimen with a radius

R=10 mm. However, the fracture region of the specimen in the performed experiment is located in the central part of the specimen next to the welded joint. For this reason, an analysis was carried out using the Probe tool, which allows reading values for different regions of the specimen.
Figure 11 shows the Von Mises stress value obtained from an analysis using the Probe tool. In this case, the numerical simulation was performed with a tensile force of F = 4.869.9 N, which is precisely the force at which the specimen fractured in the experiment. The Von Mises stress value of 81,165 N/mm² obtained experimentally corresponds to the average stress value in the fracture region of the specimen obtained from this numerical simulation.

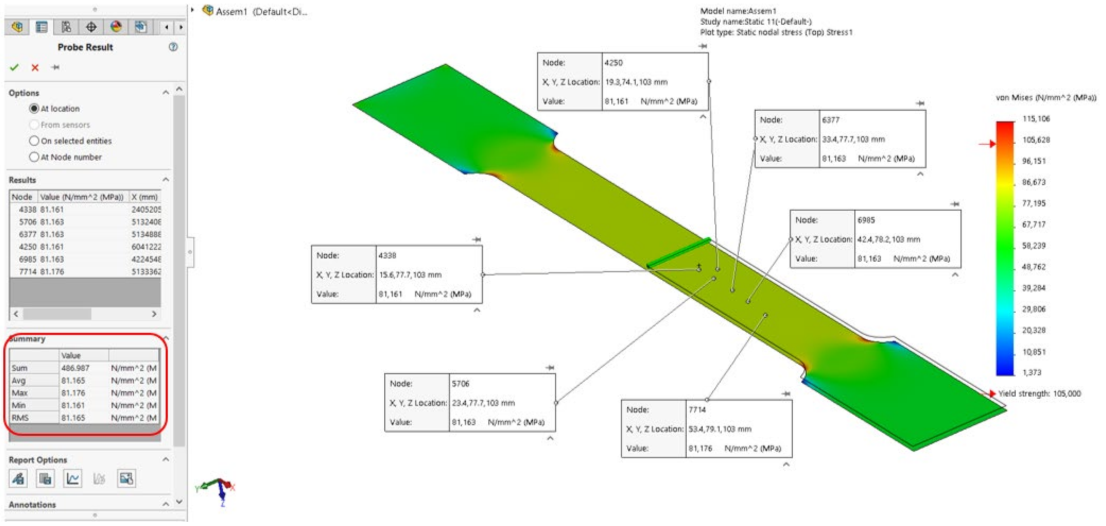


Fig. 11. *Results obtained from an analysis using the Probe tool (F = 4.869,9 N, Von Mises stress value = 81,165 N/mm²)*

The comparison between the experimental results and the average Von Mises stress values obtained using the Probe tool in the numerical simulation was carried out (Table 5).

Table 5. *Comparison of Experimental Results and Average Von Mises Stress (Probe Tool)*

Parameters	Values
Tensile force F obtained from the experiment [N]	4.869,9
Tensile strength R_m obtained from the experiment [N/mm^2]	81,165
Average Von Mises stress value obtained from the numerical simulation [N/mm^2]	81,165

CONCLUSION

The SolidWorks Simulation software has proven to be an efficient tool for the analysis of welded joints. Three different methods for analyzing welded joints in SolidWorks Simulation can be distinguished:

- All components can be modeled as solid bodies (Solid Mesh),
- The weld can be treated as a beam (Beam Mesh) while the other components remain solid bodies, and
- Instead of solid bodies, shell meshes (Shell Mesh) can be used along with the Edge Weld Connector tool.

The Weld Check Plot tool has proven to be effective in verifying the parameter values of the welded joint.
The results obtained through computer simulation differ to some extent from those obtained experimentally. This may be due to the different mechanical properties of the tested material compared to the values defined in the software, as it has been shown that the tensile strength of this material falls within a range of possible values.

Furthermore, the results indicate the need for continued research to assess the reliability of the SolidWorks Simulation software in the analysis of various welded joints, including both butt and fillet welds.

REFERENCES

[1] Lindgren, L.E., Numerical modelling of welding, Comput. Methods Appl. Mech. Engrg., 2006, (195), p 6710–6736.
[2] EN ISO 4136:2013: Ispitivanje sa razaranjem zavarenih spojeva metalnih materijala - Poprečno ispitivanje zatezanjem, 2013.
[3] Majstorović, A., Đukić, V., Ispitivanje mašinskih materijala - uputstvo, Naučna knjiga, Beograd 1989.
[4] Seferijan, D., Metalurgija zavarivanja, Građevinska knjiga, Beograd, 1969.
[5] Miltenović, V., Mašinski elementi, Mašinski fakultet u Nišu, Niš 2009.
[6] Al-Samhan, A., Darwish, S.M.H., Finite element modeling of weld-bonded joints, Journal of Materials Processing Technology, 2003, (142), p 587.598.
[7] Griškevičius, P., Urbas, M. Capas, V. Kozlovas, A., Modeling of Welded Connections in SolidWorks Simulation, Mechanika 2011: Proceedings of the 16th International Conference, 2011., p 104-108.
[8] Zdraveski, F., Bogatinoski, Z., Trajanoska, B., Modeliranje single-fillet preklopnog spoja u Solidworks simulaciji, Journal of Applied Engineering Science. 2016;14(1):21-7. doi: 10.5937/jaes14-8900
[9] Mohammadi S., Extended finite element method for fracture analysis of structure, Blackwell Publishing Ltd., Oxford, UK, 2008.
[10] https://www.thyssenkrupp-materials-processing-europe.com/en/aluminum/alloy-group-1000-or-en-aw-1050-a-or-en-aw-1200a/en-aw-1050a