DOI: 10.46793/41DAS2025.103J

ANALYTICAL AND FEM ASSESSMENT OF A DOUBLE-SIDED BUTT-WELDED S1000QL SPECIMEN FOR TENSILE TESTING PREPARATION

Snežana JOKSIĆ[1], Jovana ŽIVIĆ[2], Marija MATEJIĆ[3], Miloš MATEJIĆ[4], Živče ŠARKOĆEVIĆ[5], Ivica ČAMAGIĆ[6]

[1] 0009-0002-1909-2462, University of Pristina, Faculty of Technical Sciences with temporary settlement in Kosovska Mitrovica, Knjaza Miloša 7, Kosovska Mitrovica, Republic of Serbia, E-mail: snezana.joksic@pr.ac.rs;

[2] 0000-0003-1936-7921, University of Pristina, Faculty of Technical Sciences with temporary settlement in Kosovska Mitrovica, Knjaza Miloša 7, Kosovska Mitrovica, Republic of Serbia, E-mail: jovana.d.zivic@pr.ac.rs;

[3] 0000-0002-0580-1718, University of Pristina, Faculty of Technical Sciences with temporary settlement in Kosovska Mitrovica, Knjaza Miloša 7, Kosovska Mitrovica, Republic of Serbia, E-mail: marija.matejic@pr.ac.rs;

[4] 0000-0003-0488-0992, University of Kragujevac, Faculty of Engineering, Sestre Janjić 6, Kragujevac, Republic of Serbia, E-mail: mmatejic@kg.ac.rs,

[5] 0000-0001-5110-6620, University of Pristina, Faculty of Technical Sciences with temporary settlement in Kosovska Mitrovica, Knjaza Miloša 7, Kosovska Mitrovica, Republic of Serbia, E-mail: zivce.sarkocevic@pr.ac.rs;

[6] 0000-0003-4706-6333, University of Pristina, Faculty of Technical Sciences with temporary settlement in Kosovska Mitrovica, Knjaza Miloša 7, Kosovska Mitrovica, Republic of Serbia, E-mail: ivica.camagic@pr.ac.rs

1. Introduction

High-strength quenched and tempered steels, such as S1000QL, are increasingly employed in applications where a high strength-to-weight ratio is essential, including heavy-duty structural components, transport equipment, and machinery subject to demanding service conditions. The combination of high tensile strength and relatively low weight offers significant design advantages. However, these benefits can only be fully realized if welded joints maintain comparable structural integrity and if testing protocols are robust enough to capture the actual mechanical performance. Achieving this requires a comprehensive approach that combines analytical calculations and numerical simulations, particularly finite element method (FEM) analyses, to predict and validate stress distributions, deformations, and potential failure mechanisms.

Numerous studies have addressed various aspects of thermal cycles, residual stresses, welding distortions, and fracture behavior in welded structures. While these works often focus on different materials, joint configurations, or manufacturing processes, their findings contribute valuable insights into the modeling and experimental validation of welded joints. Attarha and Sattari-Far [1] investigated the temperature distribution in thin welded plates using a combination of experimental measurements and finite element simulations. Their study demonstrated a high level of agreement between numerical predictions and measured data, establishing a reliable foundation for analyzing thermal behavior in welded structures. This approach is directly relevant to the present study, where thermal effects and resulting stresses in high-strength steels are of key interest. Derakhshan et al. [2] conducted numerical simulations and experimental validation to assess residual stresses and welding distortions in laser-based welding of thin structural steel plates in a butt joint configuration. Their results confirmed that the finite element model accurately predicted both residual stress distributions and distortion patterns. These findings emphasize the importance of coupling simulation with experimental verification, a principle also adopted in the current work. Syahroni and Purbawanto Hidayat [3] performed a three-dimensional finite element simulation of T-joint fillet welds to evaluate the influence of welding sequences on residual stresses and distortions. They concluded that the welding sequence plays a critical role in determining the magnitude and spatial distribution of residual stresses, as well as the final distortion of the joint. This insight highlights the need for careful process planning in welding high-strength steels to minimize deformation and preserve dimensional accuracy. Dhage et al. [4] demonstrated that process parameters significantly influence final properties, a methodology applicable to welding, where such parameters determine joint integrity and service life. Kik and Górka [5] used simulations of laser and hybrid welding of S700MC steel T-joints to compare thermal cycles and stress distributions, offering insights useful for selecting optimal welding techniques for high-strength steels like S1000QL, where heat input control is critical for preserving microstructure. Stavropoulou et al. [6] carried out experimental and numerical studies on the mechanical cutting of Dionysos marble. Although the material and process differ significantly from steel welding, the study illustrates the broader applicability of combining experiments with numerical modeling to understand material removal and deformation mechanisms. Cui et al. [7] analyzed 2205 duplex stainless steel K-TIG welded joints using both simulations and experimental tests. Their results showed strong agreement between predicted and measured thermal and mechanical responses, reinforcing the necessity of cross-validation between numerical and experimental approaches—a methodology closely followed in the present work. Tanaka et al. [8] focused on crack propagation in welded joint structures, applying numerical simulation to study surface crack behavior. Their findings provided a deeper understanding of crack growth mechanisms, offering a foundation for improving the structural integrity and service life of welded components. This knowledge is particularly relevant for high-strength steels, where localized defects can rapidly compromise performance. Collectively, these studies demonstrate different numerical and experimental approaches available for analyzing welded joints.

In the present study, a double-sided butt-weld specimen representative of upcoming tensile tests was analyzed. The work began with an analytical weld calculation to estimate stress distributions and structural capacity. This was followed by a static finite element analysis to cross-validate stress and displacement predictions, thereby guiding specimen preparation and loading conditions. The combined analytical and numerical assessment aims to provide a reliable foundation for interpreting experimental test results, optimizing welding parameters, and ensuring that the mechanical performance of S1000QL welded joints meets the demands of high-strength structural applications.

2. Materials and Methods

2.1 *Material*

The base material considered in this study is 1000QL type high-strength steel. This high-strength steel is selected for applications requiring an exceptional strength-to-weight ratio combined with good toughness. For the purposes of the analytical weld calculation, a yield strength (R_{eH}) of 1050 MPa and an ultimate tensile strength (R_m) of 1100 MPa were adopted, values consistent with datasheet specifications for S1000QL.

2.2 *Specimen Geometry*

The tensile test specimen was designed with a rectangular cross-section measuring 15 × 20 mm. The joint configuration is a double-sided butt weld with a K-groove preparation, incorporating a 4 mm root gap between the flat surface on one side and the opposing K-prep tip on the other. The loaded length considered for the analytical calculation was 20 mm, corresponding to the net section under tensile loading. The model of the specimen is shown in Fig. 1.

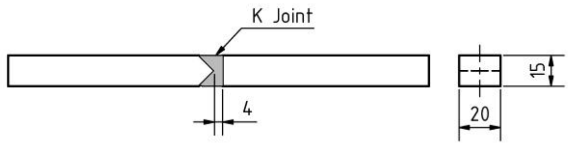

Fig. 1. Specimen dimensions

2.3 *Analytical Weld Calculation*

A comparative-stresses method was employed to assess the weld's capacity. The calculation assumed an axial force F_z = 128 kN. Under these loading conditions, the allowable stress was determined as σ_A = 505 MPa, the weld normal stress as σ = 426.667 MPa, and the reference stress as σ_S = 501.961 MPa. The verification check indicated that the weld design satisfies both static and fatigue loading criteria under the specified duty cycle (n_c = 2.622).

2.4 *Finite Element Analysis (FEA)*

A static finite element analysis was conducted using Autodesk Inventor Professional 2019 to evaluate the weld behavior. The weld interfaces were modeled with bonded contact conditions to ensure full load transfer across the joint. The mesh featured an average element size of 0.2 relative to the model dimensions, a minimum size of 0.3 of the average, and a grading factor of 2, without curved elements, providing an optimal balance between computational accuracy and efficiency. Boundary conditions included a fixed constraint at one end of the specimen and an applied axial force of 128 kN at the opposite end.

3. Results

The results of the analytical calculation and the finite element analysis (FEA) are summarized in Table 1. The analytical method yielded a calculated weld normal stress of σ = 426.667 MPa, with a reference stress of σ_S = 501.961 MPa. The static FEA predicted a maximum von-Mises equivalent stress of 432.570 MPa in the weld region. The close agreement between the analytical and numerical values (difference < 1.5%) indicates that the simplified comparative-stresses approach provides a reliable estimation of the stress state under the given loading.

Table 1. Summary of analytical and numerical results

Name	Value	Unit
Stress (analytical σ)	426.667	MPa
Reference stress (σ_S)	501.961	MPa
Von Mises (FEA)	432.570	MPa
Max displacement (FEA)	0.434	mm

Fig. 2 presents a direct comparison between the analytical stress result and the FEA-derived von Mises stress, illustrating their close correlation.

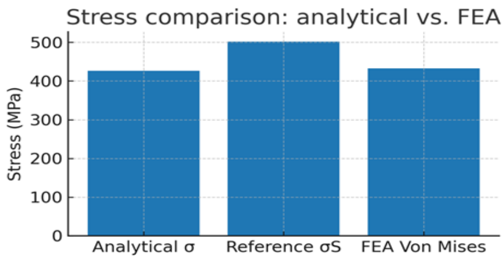

Fig. 2. Stress comparison between analytical results and FEA (von Mises)

The von Mises stress distribution obtained from the axial loading simulation is shown in Fig. 3.

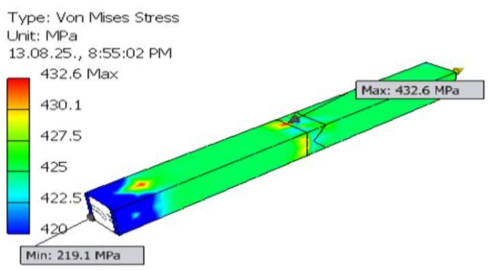

Fig. 3. Von Mises stress under axial loading

In addition to the stress distribution, the FEA predicted a maximum displacement of 0.434 mm at the free end of the specimen (Fig. 4). The corresponding minimum safety factor was also obtained from the simulation, indicating that the modeled weld joint remains within safe limits under the applied axial load of 128 kN.

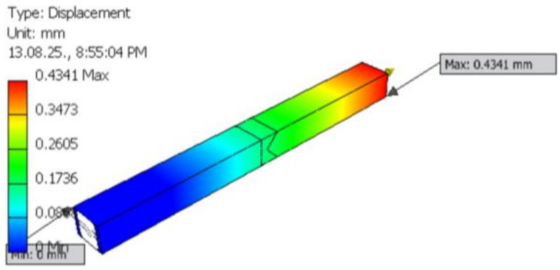

Fig. 4. Maximum displacement of 0.434 mm at the free end of the specimen

4. Results discussion

Analytical predictions and finite element (FEA) results demonstrate a high degree of agreement. The analytical weld normal stress (σ = 427 MPa) and reference stress (σ_S = 502 MPa) closely match the maximum von-Mises stress of 433 MPa obtained from the FEA simulation under an axial load of 128 kN. This close correlation confirms that the comparative-stresses method provides a reliable estimation of the stress distribution in the joint for the given geometry and loading conditions.

The FEA analysis predicts a maximum displacement of 0.434 mm, consistent with the expected stiffness of a double-sided butt weld specimen with a 15 × 20 mm cross-section. The safety factor distribution ranges from 0.64 to 1.26, with the default high-strength low-alloy (HSLA) steel material (R_{eH} = 276 MPa) applied in the simulation. Updating the FEA material parameters to reflect the actual yield strength of S1000QL (R_{eH} = 1050 MPa) is expected to proportionally increase the safety factor, maintaining values above unity throughout the joint under the applied static load.

Fatigue assessment further confirms the joint's adequacy, with a calculated cycle ratio of n_c = 2.622 for the specified pulsating load range (8–15 kN), which is significantly below the static test load. This indicates that the joint configuration satisfies the requirements for both static and cyclic loading within the designed operating range.

These results support proceeding with the planned tensile testing of straight specimens. The selected geometry and weld length are considered sufficient to represent the mechanical behavior, while the close agreement between analytical and FEA results enhances confidence in the chosen loading protocol. Prior to experimental validation, it is recommended to:

- update FEA material properties to those of S1000QL,
- perform a mesh-sensitivity study at the weld toe and root to refine local stress predictions, and
- extend the FEA model to include elastoplastic material behavior for a more comprehensive structural assessment.

5. Conclusions

A combined analytical and finite element analysis (FEA) of a double-sided butt-welded S1000QL tensile specimen under a 128 kN axial load showed close agreement between analytical and numerical results, confirming the reliability of the simplified comparative-stresses method for preliminary strength evaluation in high-strength steels. The specimen geometry and weld configuration were found suitable for the planned tensile testing.

For improved simulation accuracy, it is recommended to update FEA material properties to S1000QL values, perform mesh-sensitivity studies at the weld toe and root, and incorporate elastoplastic material behavior. These refinements will enhance correlation between simulations and experimental results, enabling precise characterization of S1000QL welded joints and ensuring that the findings are applicable to real-world engineering applications.

Further research should focus on expanding the numerical model to include the effects of different welding technologies, variations in groove geometry, and actual heat input parameters. Special attention should be given to modeling the fatigue behavior of the joint under variable loading, as well as comparing the results with experimental data obtained from a larger number of specimens.

References

[1] Attarha, M.J.; Sattari-Far, I. Study on welding temperature distribution in thin welded plates through experimental measurements and finite element simulation. *J. Mater. Process. Technol.* 2011, 211(4), 688–694. doi:10.1016/j.jmatprotec.2010.12.003

[2] Derakhshan, E.D.; Yazdian, N.; Craft, B.; Smith, S.; Kovacevic, R. Numerical simulation and experimental validation of residual stress and welding distortion induced by laser-based welding processes of thin structural steel plates in butt joint configuration. *Opt. Laser Technol*. 2018, 104, 170–182. doi:10.1016/j.optlastec.2018.02.026

[3] Syahroni, N.; Purbawanto Hidayat, M.I. 3D Finite Element Simulation of T-Joint Fillet Weld: Effect of Various Welding Sequences on the Residual Stresses and Distortions. Numerical Simulation—*From Theory to Industry*, 2012. doi:10.5772/50015

[4] Dhage, G.S.; Pawar, R.; Patil, J. Experimental Studies On Surface Roughness of Spur Gear. REST *Journal of Emerging trends in Modelling and Manufacturing*. 2024, 10(4), 1–9. doi:10.46632/jemm/10/4/1

[5] Kik, T.; Górka, J. Numerical simulations of laser and hybrid S700MC T-joint welding. *Materials*. 2019, 12(3), 516. doi:10.3390/ma12030516

[6] Stavropoulou, M.; Giannakopoulos, K.; Exadaktylos, G. Experimental and numerical study of mechanical cutting of Dionysos marble. *7th National Congress on Mechanics,* 2004, 236–245.

[7] Cui, S.; Pang, S.; Pang, D.; Zhang, Q.; Zhang, Z. Numerical simulation and experimental investigation on 2205 duplex stainless steel K-TIG welded joint. *Metals*. 2021, 11(8), 1328. doi:10.3390/met11081323

[8] Tanaka, S.; Kawahara, T.; Okada, H. Study on crack propagation simulation of surface crack in welded joint structure. *Marine Structures.* 2014, 39, 315–334. doi:10.1016/j.marstruc.2014.08.001.