JOURNAL of
MATERIALS and MECHATRONICS:A

e-ISSN 2717-8811
JournalMM, 2023,4(1), 11-22
https://doi.org/10.55546/jmm.1116198

Araştırma Makalesi / Research Article

Comparison of Welded Joint Stress with Experimental and Finite Element Method Using of Hotspot Method

Osman Bahadır ÖZDEN[1*], Barış GÖKÇE[2], Abdullah ERDEMİR[3]

[1] MPG Machinery Production Group Co., Konya, Türkiye,
Orcid ID: https://orcid.org/0000-0003-1231-2936, bozden@mpg.com.tr
[2] Mechatronic Engineering Department, Necmettin Erbakan University, Konya, Türkiye,
Orcid ID: https://orcid.org/0000-0001-6141-7625, bgokce@erbakan.edu.tr
[3] MPG Machinery Production Group Co., Konya, Türkiye,
Orcid ID: https://orcid.org/0000-0002-7267-3111, aerdemir@mpg.com.tr

Geliş/ Received: 13.05.2022; **Kabul / Accepted:**14.12.2022

ABSTRACT: The purpose of the welded structures is to combine the two different structures defined as the workpiece and the main material in order to ensure that they remain in the elastic deformation zone by meeting the loading conditions safely. Welding parameters such as preheating, welding speed, shielding gas selection, filler wire selection, voltage, current values affect the mechanical properties of the HAZ zone and the general structure, especially when welding fine-grained structural steels are performed. Strain gauge sensors can give normal stress and shear stress values for structures forced by static loadings depending on the stable x, y, z axes. The hotspot stress method used with the finite element method gives closer results to experimental studies. In this study, two different S960QL steels were combined with workpiece and main material using MAG welding. Data were taken from the strain gauge sensor connected to the samples prepared by the hotspot stress method. Using the finite element method, different types of models were analyzed and experimental data were compared with analysis outputs. As a result of the comparison, the most accurate welded joint analysis modeling with the hotspot method has been determined by the results of the experiment and analysis and has been proven with an accuracy rate of 89%.

Keywords: Finite Element Method, Welded Joint Stress, Hotspot Stress, Strain Gauge Sensor.

*Sorumlu yazar / Corresponding author: bozden@mpg.com.tr

Bu makaleye atıf yapmak için /To cite this article

Özden O.B., Gökçe B., Erdemir A. (2023). Comparison of Welded Joint Stress with Experimental and Finite Element Method Using of Hotspot Method. Journal of Materials and Mechatronics: A (JournalMM), 4(1), 11-22.

Hotspot Yöntemi Kullanılarak Deneysel ve Sonlu Elemanlar Yöntemi ile Kaynaklı Bağlantı Gerilmelerinin Karşılaştırılması

ÖZET: Kaynaklı yapıların amacı, iş parçası ve ana malzeme olarak tanımlanan iki farklı yapıyı birleştirmek ve yükleme koşullarını güvenli bir şekilde karşılayarak elastik deformasyon bölgesinde kalmalarını sağlamaktır. Özellikle ince taneli yapı çeliklerinin kaynağı yapılırken ön ısıtma, kaynak hızı, koruyucu gaz seçimi, dolgu teli seçimi, gerilim, akım değerleri gibi kaynak parametreleri HAZ bölgesinin mekanik özelliklerini ve genel yapıyı etkilemektedir. Gerinim ölçer sensörler, x, y, z eksenine bağlı olarak statik yükler tarafından zorlanan yapılar için normal gerilme ve kayma gerilme değerlerini verebilmektedir. Kaynaklı bağlantılarda hotspot gerilme yöntemi, sonlu elemanlar yöntemi ve deneysel çalışmalarda daha doğru sonuçlar vermektedir. Bu çalışmada iki farklı numunede S960QL çeliği, MAG kaynağı kullanılarak iş parçası ve ana malzeme birleştirilmiştir. Hotspot gerilme yöntemi ile hazırlanan numunelere bağlı gerinim ölçer sensörlerinden veriler alınmıştır. Sonlu elemanlar yöntemi kullanılarak farklı parametrelerde modeller analiz edilmiş ve deneysel veriler ile karşılaştırılmıştır. Karşılaştırma sonucunda hotspot gerilme yöntemi ile en doğru kaynaklı bağlantı analizi modellemesi deney ve analiz sonuçları ile belirlenmiş ve %89 doğruluk oranı ile kanıtlanmıştır.

Anahtar Kelimeler: Sonlu Elemanlar Metodu, Kaynaklı Bağlantılarda Gerilme, Hotspot Gerilme, Gerinim Ölçer Sensör.

1. INTRODUCTION

Finite element analysis in welded joints is performed using many different modeling techniques. The results of the applied finite element modelling methods can cause great changes. The International Welding Institute proposes four methods including the hotspot stress method for the structural stresses and fatigue life values occurring at the welding toe and at the weld root (Niemi et al., 2018). Scientific studies conducted in recent years show that the hotspot method gives results closer to the tests performed (Iqbal et al., 2020). Calculation of analytical formulations is not effective due to the discontinuities and hotspot stresses occurring in the weld seam. If the structures are complex, and due to some basic formulation deficiency, finite element analysis reveals more practical and accurate results (Ali et al., 2020; Meyghani et al., 2019). Different finite element modeling techniques are used in hotspot stress calculations and the stresses are calculated in the most ideal way (Iqbal et al., 2020). If the discontinuities that will occur in the weld pool are included in the calculations, the stress value should be multiplied by the stress magnification factor (k_m) (Hobbacher, 2016).

By using the hotspot stress method, fatigue life calculations can also be made with the forces under static and dynamic loads in the weld seam. For the fatigue life, values are determined according to the FAT tables found in IIW documents according to the workpiece to be welded and the main material form (Dong et al., 2019; Shin et al., 2021). Finite element welding modeling can be done in 2 different types as shell modeling and solid modeling (Niemi et al., 2018). There are also different modeling variants for shell and solid modeling. The shape of the structure affects the results of the solid and shell modeling technique. (Kim et al., 2015; Yamamoto et al., 2020). However, some shell modeling techniques in studies gave results close to 88% accuracy to according to some experimental results (Büyükbayram et al., 2015).

Strain gauge sensors are used to measure the strain values in local areas and according to the Hooke’s law, the elasticity modulus and stress values of the material are found. It is used to measure the residual stresses in welded joints and to measure the local stresses as a result of static and dynamic loads (Feng et al., 2020; Li et al., 2017). It has different types of bridging and strain gauge strain measurement studies with Wheatstone bridging type are generally performed in studies (Güven & Rende, 2020).

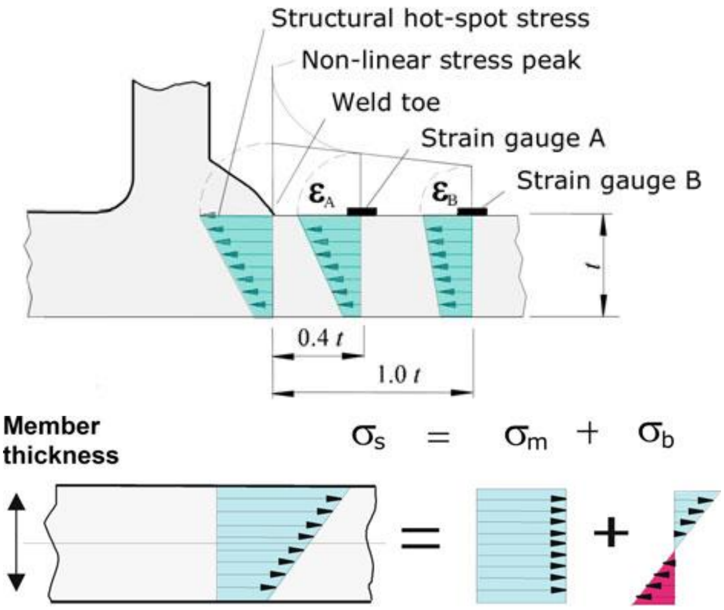


Figure 1. Hotspot stress linear extrapolation and linearized stress (Niemi et al., 2018)

In experimental studies, hotspot stress at the welding tip can be measured by sticking 2 strain gauges at 0.4t and 1t distances as strain gauge positions, where t is the thickness of the metal. In finite element analysis, the same stress value can be obtained by using fine mesh. In Ansys Mechanical, the sum of membrane and bending stress is determined with linearized stress, and linear extrapolation is made to the welding toe for 2 different locations. Figure 1 shows that hotspot stress linear extrapolation and linearized stress, and linear extrapolation is made to the welding toe for 2 different locations.

In test studies, the hotspot stress is calculated according to Equation (1) by taking linear extrapolation from A and B positions given in Figure 1 and the size of maximum element is 0.4t in finite element model (Niemi et al., 2018):

$$\sigma_{hs} = 1.67\sigma_{0.4t} - 0.67\sigma_{1t} \quad (1)$$

According to the quadratic extrapolation used in bending stresses in the finite element method, the hotspot stress at maximum 0.4t mesh element size is calculated according to Equation (2) (Niemi et al., 2018):

$$\sigma_{hs} = 2.52\sigma_{0.4t} - 2.24\sigma_{0.9t} + 0.72\sigma_{1.4t} \quad (2)$$

While shell modeling of the welded joint in the finite element method, it may vary depending on the shape of the weld seam in cases such as double sided, single sided, full penetration and half penetration. Double sided, half penetration and full penetration FEM (Finite Element Method) of welded joint models are shown in Figure 2:

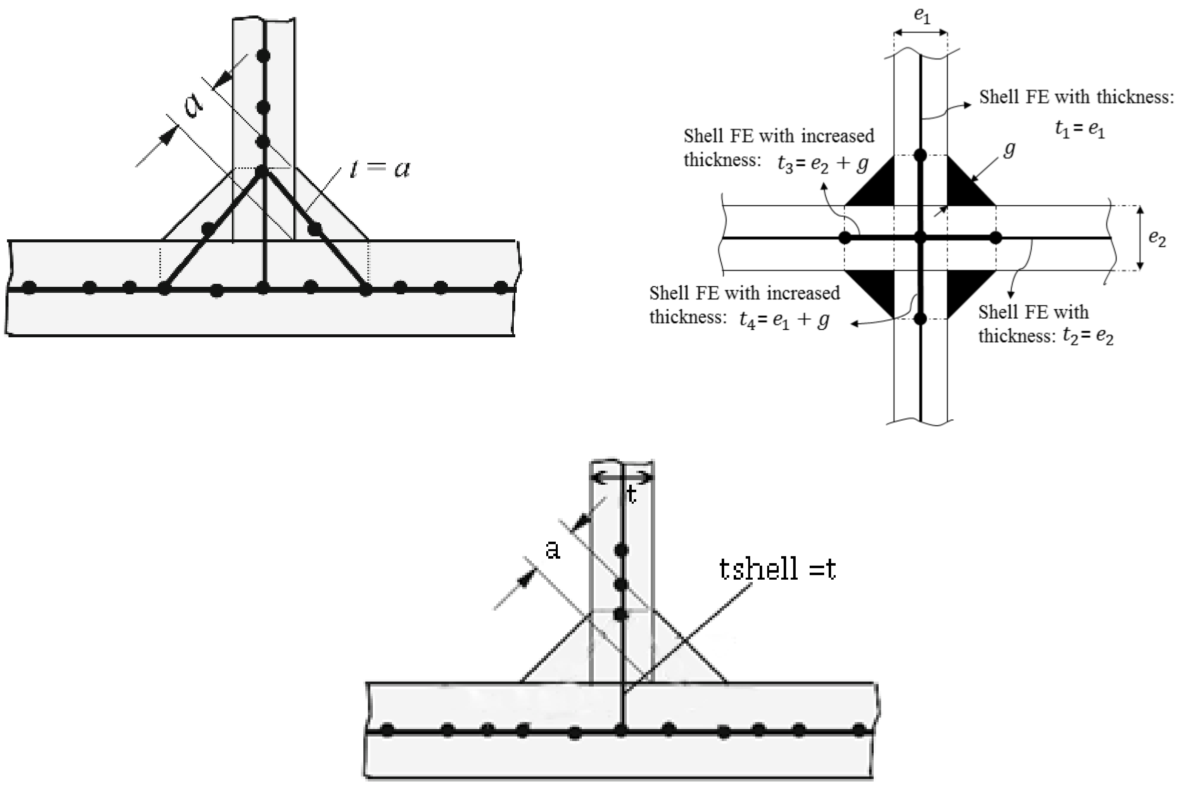


Figure 2. Double side, half penetration and full penetration FEM welded joint models (Eriksson et al., 2003; Niemi et al., 2018).

In the finite element method, singularity can affect the results while evaluating. While solid modeling the of welded joint, this situation should be considered while obtaining hotspot stress and results.

2. MATERIALS AND METHODS

2. 1. Experimental Setup and Materials

In the tests performed, the welding processes were applied to the S960QL steel for each material. In structures with high moment and axial load requirements such as mobile cranes, this type of ultra-high strength fine grain structural steel is preferred and therefore S960QL was chosen for this purpose. In order to compare the samples, parts with 2 different weld lengths were used. Both samples were made in one pass and the same welding parameters were used for comparison purposes. Table 1 shows the chemical compositions of S960QL.

Table 1. S960QL Chemical compositions

C %	Si %	Mn %	P %	S %	Cr %	Ni %	Mo %	V %	Ti %	Cu %
0.17	0.22	1.24	0.01	0.001	0.2	0.06	0.597	0.04	0.002	0.02

The filler material used for the welding process was low alloyed OK AristoRod 89 (ESAB) welding wire; its mechanical properties, which is given in Table 3, do not exceed the mechanical properties of S960QL. The filling material OK AristoRod 89 has a yield strength of 920 MPa and a tensile strength of 940 MPa. Table 2 shows the chemical compositions of OK AristoRod 89 welding filler metal.

Table 2. OK Aristorod 89 Chemical compositions

Element	**Percentage of Content (%)**
C	0.17
Si	0.80
Mn	1.75
Cr	0.41
Ni	2.22
Mo	0.53

Throat thickness in welded joints may vary depending on welding parameters. Welding parameters were determined according to the welding thickness specified in the study. Table 4 shows the welding parameters.

Table 3. Mechanical Properties of S960QL

Element	**Value**
Yield Strength - $Rp_{0.2}$ - MPa	1027
Elasticity Modulus - E - GPa	203.4
Tensile Strength - Rm - MPa	1066
Elongation – A5 - %	16
Poisson Ratio - ν	0.33

M21 (80% Ar, 20% CO_2) gas mixture was chosen as shielding gas. Due to the discontinuities that may occur in the weld seam after welding, the most ideal weld seam samples were prepared and made available for testing by performing visual inspection, penetration test and ultrasonic test.

Table 4. Welding Parameters

Parameter	**Sample 1**	**Sample 2**
Weld Length – mm	20	30
Welding Throat Thickness – mm	3	3
Pre heating - °C	100	100
Voltage - V	20 – 25	20 – 25
Current - I	200 – 210	200 – 210
Arc Length - mm	5	5
Wire Feeding Speed - m/min	8	8

Figure 3 shows the experimental setup and data acquisition software interface. Hydraulic cylinders are used to break the weld seam in the experimental setup, and force values are acquired with the pressure sensors connected. Dewesoft software and data acquisition card were used in order to collect data from strain gauge and pressure sensors.

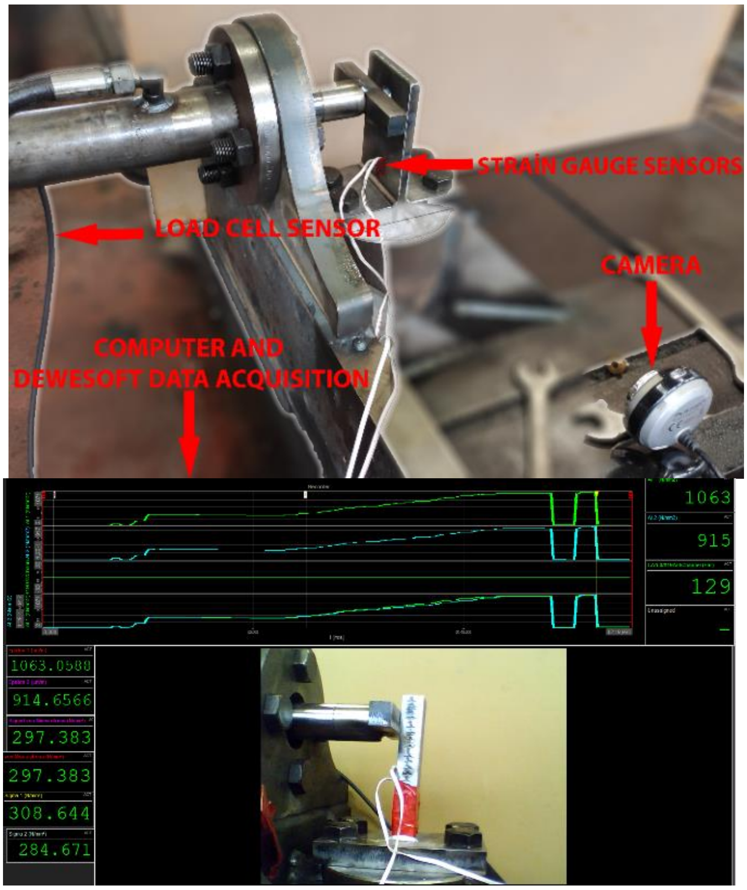


Figure 3. Experimental setup and software interface

The strain gauge locations on the sample are given in Figure 4. In the experiments, the sample thicknesses were chosen as 20 mm, and the 2 strain gauges were placed at 0.4t and 1t distances from the weld toe. In the finite element analysis, shell and solid modeling with maximum 0.4t element size were prepared and experimental data and analysis results were compared.

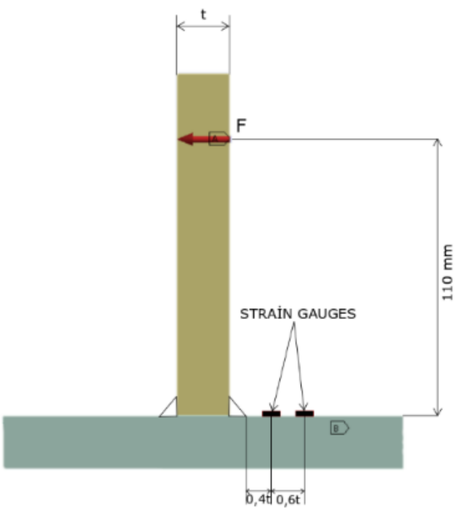
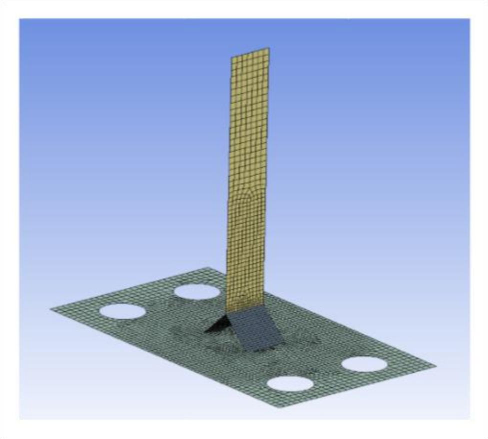


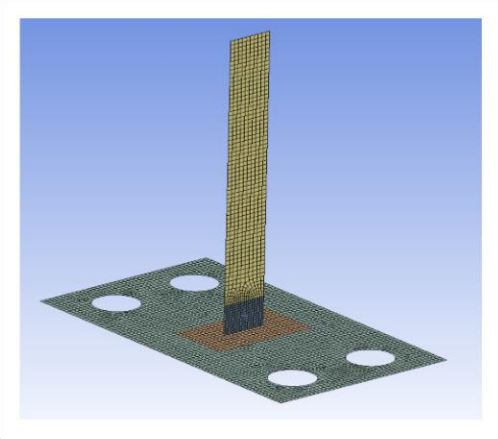
Figure 4. Strain gauge location and strained surface size in the simulation

2. 2. Welded Joint FEM Models

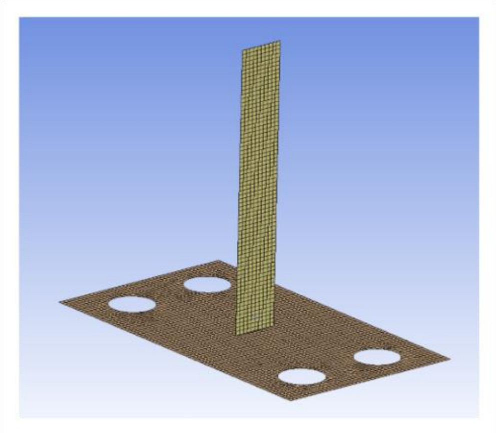
The contact areas are defined by share topology and frictional contact. Path definitions were made according to Equation (1) and Equation (2), and the stress values were taken from path points. Figure 5 shows the SHL and SLD FEM welding models prepared for the Finite Element Analysis.



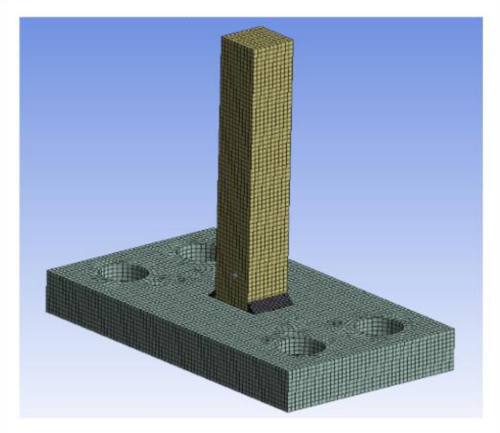
SHL-1



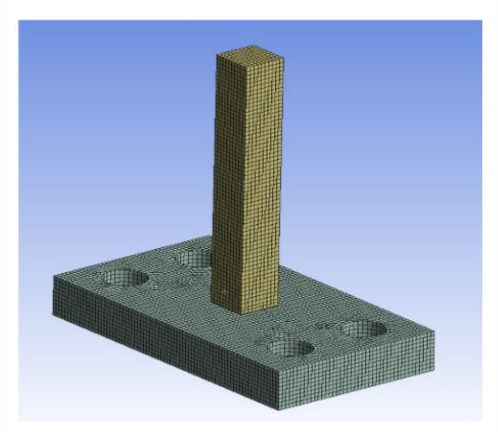
SHL-2



SHL-3



SLD-1



SLD-2

Figure 5. Prepared SHL (Shell) and SLD (Solid) FEM welding models

SHL1-3 models are defined as shell, while, SLD1-2 models are defined as solid. The mesh qualities for the shell and solid models are the same, and the mesh element quality is 0.85. Mesh structure consist of 5852 nodes and 4854 elements for shell models, and 18247 nodes and 15864 elements for solid models. The maximum element size of the mesh is 4 mm, and quadratic – hexagonal mesh types were used in the models. For the evaluation of FEM analysis results, Equation (1) and Equation (2) were applied and results were obtained only for the break/failure points. Hotspot stress method gives results closer to the experiments.

3. RESULTS AND DISCUSSION

For verification, 2 different samples with weld lengths of 20 mm and 30 mm were prepared and a test setup was prepared for testing of these samples, and strain gauge sensors were attached to these samples according to the weld tip distances specified in the hotspot stress. In the hotspot stress method, force was applied until the specimen failed and normal stress values were collected. In FEM analysis, the same force values were defined according to different weld models and 2 different equations, and the results were analyzed. In FEM software, welded fasteners can be defined with different models as shell and solid. In this study, results were obtained according to different weld modeling as shell and solid in 2 different equations in the hotspot stress method. In complex structures, analytical calculation methods and calculating stress values in the weld area are not preferred in terms of time and difficulty, therefore FEM analysis is preferred. In the FEM model, the analysis modeling of welded structures may differ in the results. In order for welded structures to combine different structures and meet the forces coming to this structure safely, analyzes should be evaluated with the most accurate FEM welding model.

3. 1. Results for Equation (1)

The results of Equation (1) applied for Sample 1 are given in Figure 6. As a result of the experiments, breaking of the weld occurred at 11138 N in Sample 1 and 16516 N in Sample 2. In Figure 6, for Equation (1), the closest results according to Sample 1 are SHL-3 and SHL-1 modeling. The accuracy rate was 68.34% for SHL-3 and 56.22% for SHL-1.

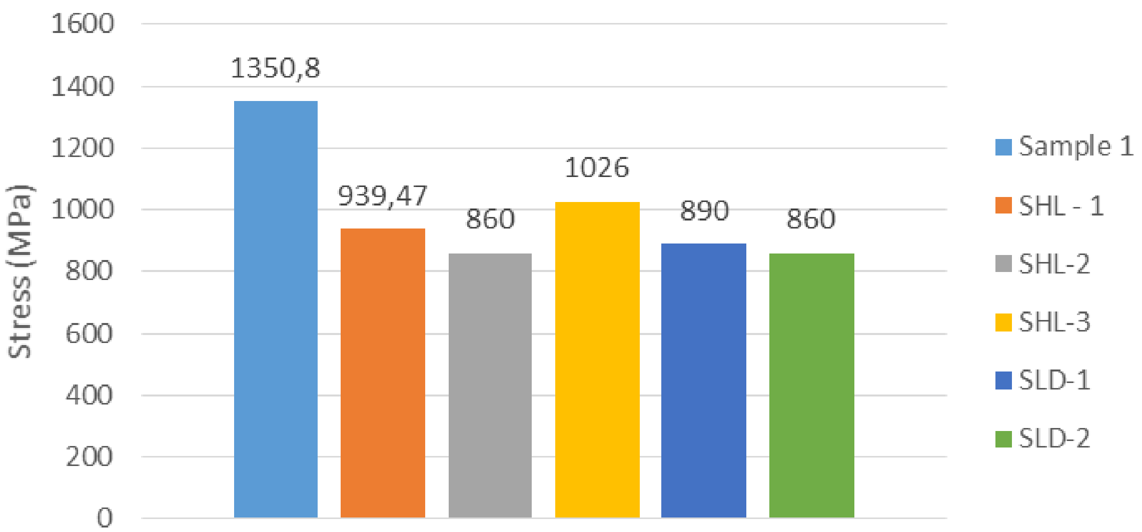


Figure 6. Results of Equation (1) for Sample 1.

In Figure 7, the analysis results of Sample 2 for Equation (1) are given. Accordingly, the closest results were SHL-3 and SHL-1 modeling. The accuracy rate was 86.20% for SHL-3 and 73.49%. for SHL-1.

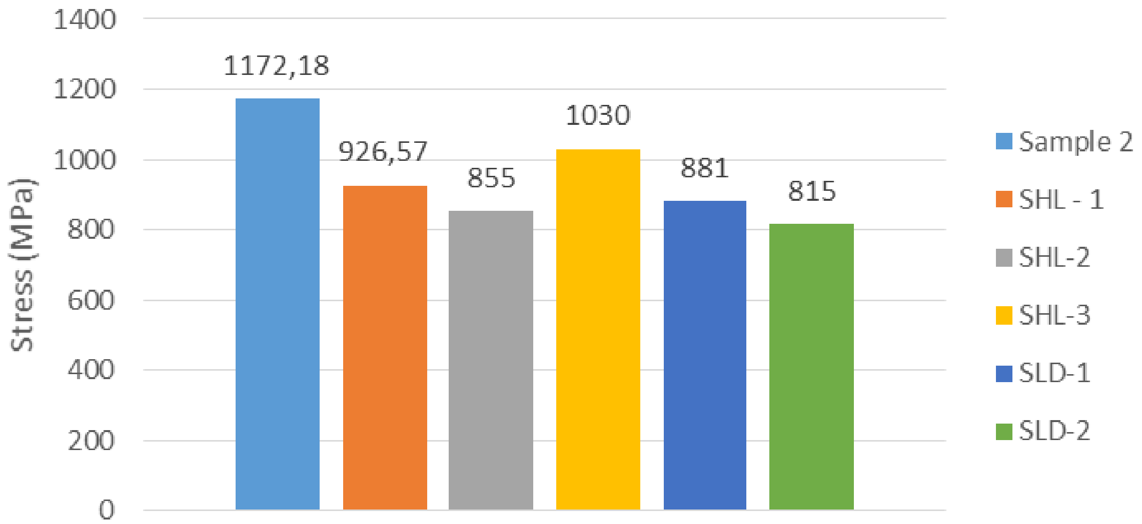


Figure 7. Results of Equation (1) for Sample 2.

3. 2. Results for Equation (2)

In Figure 8, results for Equation (2) are given according to Sample 1. Therefore, the closest results were SHL-3 and SHL-1 modeling. The accuracy rate was 71.21% for SHL-3 and 60.19% for SHL-1.

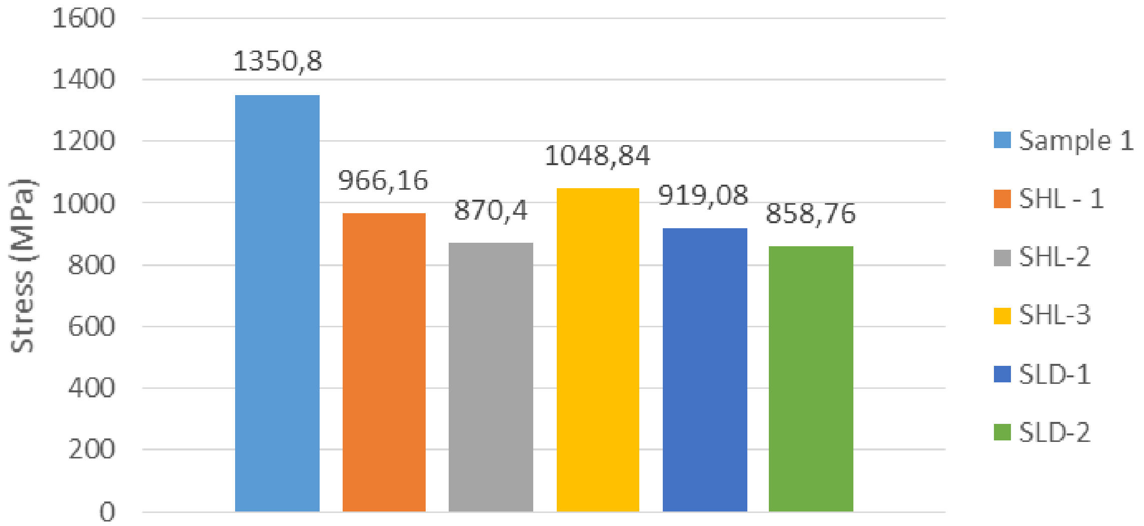


Figure 8. Results of Equation 2 for Sample 1

In Figure 9, the results for Equation (2) are given according to Sample 2. Hence, the closest results were SHL-3 and SHL-1 modeling. The accuracy rate was 89.47% for SHL-3 and 88.90% for SHL-1.

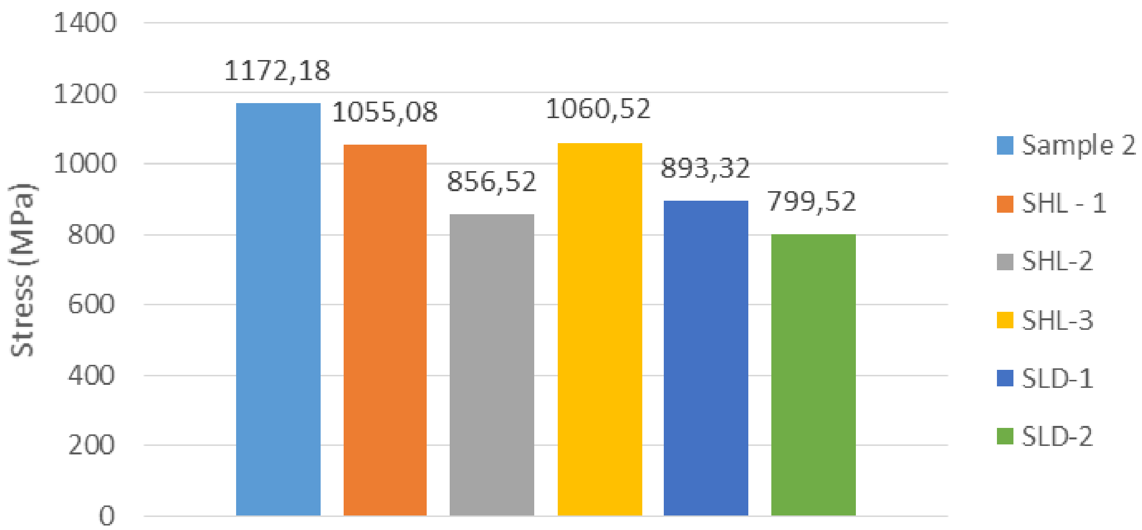
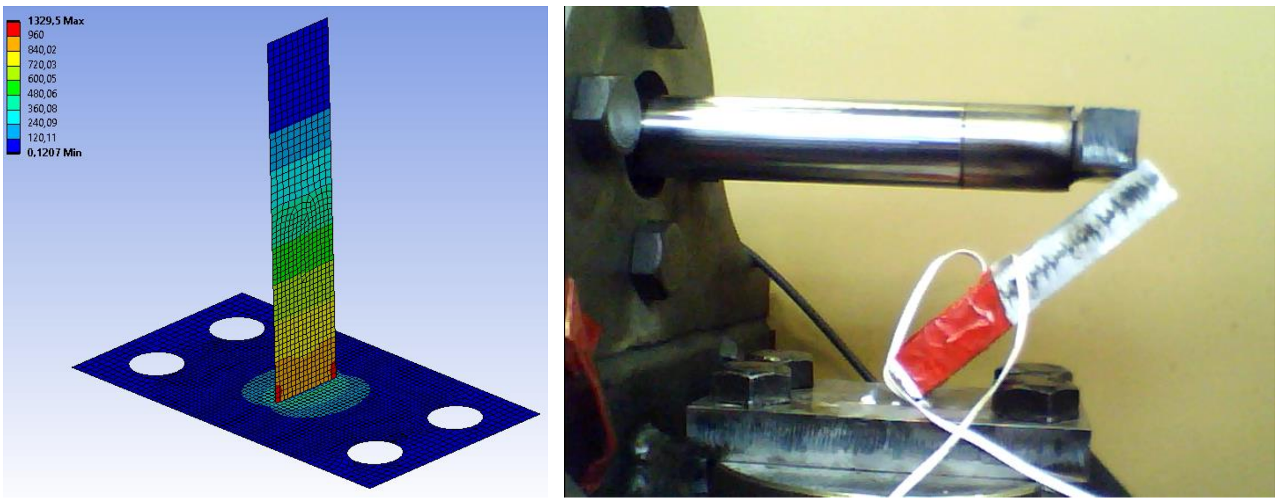


Figure 9. Results of Equation (2) for Sample 2

According to the destructive test results applied, their tensile strength was seen to be over 1172 MPa. According to the test results and a study by Schroepfer et al. (Schroepfer et al., 2015), it was determined that the HAZ region is suitable in terms of mechanical properties. Thus, welding parameters were evaluated appropriately. The fracture instant and test sample for the 2nd sample are shown in Figure 10, with an accuracy of 89.47% according to (a) SHL-3 FEM welded joint modeling approach and (b) test results. In a study performed as a dynamic analysis in the literature (Büyükbayram et al., 2015), it is seen that the SHL-2 model has an accuracy rate of approximately 88%. In this study, SHL-3 was combined from the edge of the workpiece to the surface of the main material with a bond called share topology in welding modeling. According to the other study in the literature, the difference is thought to be due to dynamic loading and bond type.



a **b**

Figure 10. Instant of fracture (a) SHL-3 (b) Sample 2.

In FEM analyzes of structures with high moment and axial loads, especially attention should be paid to the selection of welded joint modeling type to ensure structural safety. As a result of this study, SHL-3 modeling with quadratic extrapolation is recommended for static analyzes.

4. CONSLUSION

In this study, results were evaluated with 2 different equations with the stress values taken from different distances from the weld toe according to IIW documents in FEM software. According to the results obtained, it has been predicted that the same modeling can be used in complex structures. The results obtained as a result of experimental measurements with strain gauge according to prepared Sample 1 and Sample 2 were compared with the FEM analyze. According to the results:

1. Indicated in Figure 6, 7, 8 and 9, it was seen that SHL-3 and SHL-1 weld modeling gave more accurate with an accuracy rate of 89%.

2. According to the experiment forced by static loading, more accurate results were obtained according to the quadratic extrapolation equation given in Equation (2).

3. In the literature (Büyükbayram et al., 2015), it has been observed that some shell modeling techniques give results close to 88%, according to dynamically forced experimental results. In this study, which was performed statically, it is seen that SHL-3 and SHL-1 models made with share topology give more accurate results.

4. According to the welding parameters selected for S960QL ultra high strength steels, this design has been found to have sufficient tensile strength (1172 MPa).

5. ACKNOWLEDGEMENTS

The authors gratefully acknowledge the support of “MPG Machinery Production Group Inc. Co.” in this study.

6. CONFLICT OF INTEREST

Author(s) approve that to the best of their knowledge, there is not any conflict of interest or common interest with an institution/organization or a person that may affect the review process of the paper.

7. AUTHOR CONSTRIBUTION

Osman Bahadır ÖZDEN contributed to the preparation of the experimental setup of the study, the realization of the FEM analysis, and the drafting of the article. Barış GÖKÇE contributed to the drafting of the article and to the criticism of the content. Abdullah ERDEMİR contributed to the drafting of the article and to the FEM analysis.

8. REFERENCES

Ali D., Abbas H., Abdullah T., Numerical analysis of stress distribution during tunneling in clay stone rock. International Journal of Engineering 33(8), 1472-1478i, 2020.

Büyükbayram C., Koç B., Tekin G., Salihoğlu B., Fatigue strength evaluation approach of welded joints for more reliable and lighter military ground vehicles. Procedia Engineering 133, 485-500, 2015.

Dong Y., Teixeira A., Soares C.G., Fatigue reliability analysis of butt welded joints with misalignments based on hotspot stress approach. Marine Structures 65, 215-228, 2019.

Eriksson Å., Lignell A.-M., Olsson, C., Spennare, H., Weld evaluation using FEM: a guide to fatigue-loaded structures. Industrilitteratur 2003.

Feng R., Tang C., Roy K., Chen Z., Chen B, Lim, J. B., An experimental study on stress concentration factors of stainless steel hybrid tubular K-joints. Thin-Walled Structures 157, 107064, 2020.

Güven F., Rende H., An analysis of endurance limit-modifying factors depending on bead shape and thickness in load-carrying welded T-joints. Journal of the Brazilian Society of Mechanical Sciences and Engineering 42(2), 1-8, 2020.

Hobbacher A., Recommendations for fatigue design of welded joints and components (Vol. 47). Springer, 2016.

Iqbal N., Fang H., Naseem A., Kashif M., De Backer H., A numerical evaluation of structural hot-spot stress methods in rib-to-deck joint of orthotropic steel deck. Applied Sciences 10(19), 6924, 2020.

Kim Y., Oh J.-S., Jeon S.-H., Novel hot spot stress calculations for welded joints using 3D solid finite elements. Marine Structures 44, 1-18, 2015.

Li S., Ren S., Zhang Y., Deng D., Murakawa H., Numerical investigation of formation mechanism of welding residual stress in P92 steel multi-pass joints. Journal of Materials Processing Technology 244, 240-252, 2017.

Meyghani B., Awang M., Wu C., Thermal analysis of friction stir welding with a complex curved welding seam. International Journal of Engineering 32(10), 1480-1484, 2019.

Niemi E., Fricke W., Maddox S. J., The structural hot-spot stress approach to fatigue analysis. In Structural Hot-Spot Stress Approach to Fatigue Analysis of Welded Components (pp. 5-12). Springer, 2018.

Schroepfer D., Kromm A., Kannengiesser T., Improving welding stresses by filler metal and heat control selection in component-related butt joints of high-strength steel. Welding in the World 59(3), 455-464, 2015.

Shin W., Chang K.-H., Muzaffer S., Fatigue analysis of cruciform welded joint with weld penetration defects. Engineering Failure Analysis 120, 105111, 2021.

Yamamoto N., Sugimoto T., Ishibashi, K., Tanaka, S., A study on the type-b hot spot stress. Welding in the World 64(11), 1833-1842, 2020.