https://doi.org/10.51630/ijes.v6i2.177

FINITE ELEMENT NUMERICAL ANALYSIS FOR FATIGUE STRENGTH IN FSW WELDING OF ALUMINUM-5052 ALLOY

ARMANSYAH[1,2*], DJOKO SETYANTO[1]

[1]*Professional Engineer Certification; School of Biosciece, Technology, and Innovation, Atma Jaya Catholic University of Indonesia, Jl. Jenderal Sudirman 51, Jakarta (12930), Indonesia*
[2]*Study Program of Mechanical Engineering, Faculty of Engineering, Universitas Pembangunan Nasional Veteran Jakarta, Jl. Raya Limo Kecamatan Limo, Kota Depok, Jawa Barat, Indonesia*

**Corresponding author: armansyah@upnvj.ac.id*

(Received: 15 March 2025; Accepted: 15 May 2025; Published on-line: 18 May 2025)

ABSTRACT: This study applied observations on the fatigue loads of friction stir welding joints of 5052 Aluminum alloy using the finite element method approach. The 3-dimensional model of the test specimen for fatigue loading was designed using CAD software following the ASTM E466 standard. Fatigue loads were simulated numerically using Ansys Workbench software with loading sequences of 5000 N, 10,000 N, 15,000 N, and 20,000 N. The simulation showed a significant increase in equivalent stress from about 9.44e7 Pa to 3.82e8 Pa.. Deformation increased from 0.32 mm to 1.26 mm, while equivalent static strain rose from 0.001 m/m to 0.004 m/m. The S/N curve analysis showed that fatigue life was under 500,000 cycles at loads between 10,000 and 20,000 N, and between 500,000 and 6,000,000 cycles at 5000 to 10,000 N.

KEY WORDS: fatigue analysis; finite element method; friction stir welding; Aluminum 505.

1. INTRODUCTION

Friction stir welding (FSW) is a welding method that has developed rapidly lately because it can produce high-strength quality, especially in Aluminum alloys [1]. Many research references have been conducted on the strength characteristics of FSW through hardness testing and tensile testing. However, the fatigue strength characteristics of FSW joints on Aluminum alloys are still observed to increase the strength and durability of the joints, especially in repeated loading [2]. Therefore, the fatigue strength characteristics of FSW joints on aluminum alloys must be studied to increase knowledge, especially to determine the weld areas on FSW joints susceptible to failure due to repeated loads. In the stage of understanding the fatigue strength characteristics of FSW joints on Aluminum alloys, an appropriate and accurate analysis method is needed. The finite element method is one of the approaches through numerical analysis that can predict the fatigue strength of FSW joints on Aluminum alloys [3]. This approach separates FSW joints into discrete elements called finite elements. Each finite element has specific parameters such as strength, elasticity, and friction coefficient. The finite element model was then used to simulate the strength and fatigue characteristics of FSW joints in Aluminum alloys.

1.1. Friction Stir Welding Process

The FSW process is a method of welding or joining two metals without the need to melt the metals being joined in a pre-solid state and at a relatively low temperature [4]. The heat generated by friction between the workpiece and the rotating tools with high axial pressure is generated in the welding area. This method is usually used in applications where the base metals are joined without changing their characteristics. Melting metal materials through high heat allows damage to the microstructure and composition of the material. The FSW method carried out below the material's melting point will produce a narrow HAZ area, so that changes can be minimized, and the residual stress and distortion produced are also less. Material bonding occurs through tool displacement or shifting during movement and mechanical deformation of the workpiece [5]. The FSW welding process takes place in a solid state. Because the friction stir welding process occurs at the solvus temperature, no loss or decrease in strength is caused by over-aging or dissolution of stable deposits. The relatively low temperature minimizes the buildup of residual stress and distortion due to heat [6]. This method aims to ensure that the characteristics of the parent metal do not undergo many changes.

The FSW process uses a rotating cylindrical tool with an indenter to heat the material through friction. This tool will rotate, rub the material, and move along the weld line. The friction will cause the material to connect as shown in Figure 1. The operating principle in this welding process is to use a temperature of 70% to 90% of the melting point of Aluminum [7]. Using this lower temperature, the heat-affected zone (HAZ) can be minimized, and the level of material flexibility can be reduced. In addition, this FSW method also produces minimal distortion [8]. The cycle in the FSW process can be divided into several steps, each with a different heat flow and thermal profile. These cycles are shown in Fig. 1. The first step is preheating, which involves a dwell time where the tool rotates without translational movement on the workpiece [9]. The material under the tool is ready to move translationally along the joining line in this step. The next step is transient heating, where the heat generated and the temperature around the tool become unstable and fluctuate before reaching a steady state when the tool movement begins. Then, the pseudo steady state step occurs, where temperature fluctuates around the tool, but the microstructure is thermally stable. The final step is the post-steady state, where heat increases around the tool as the welding process ends.

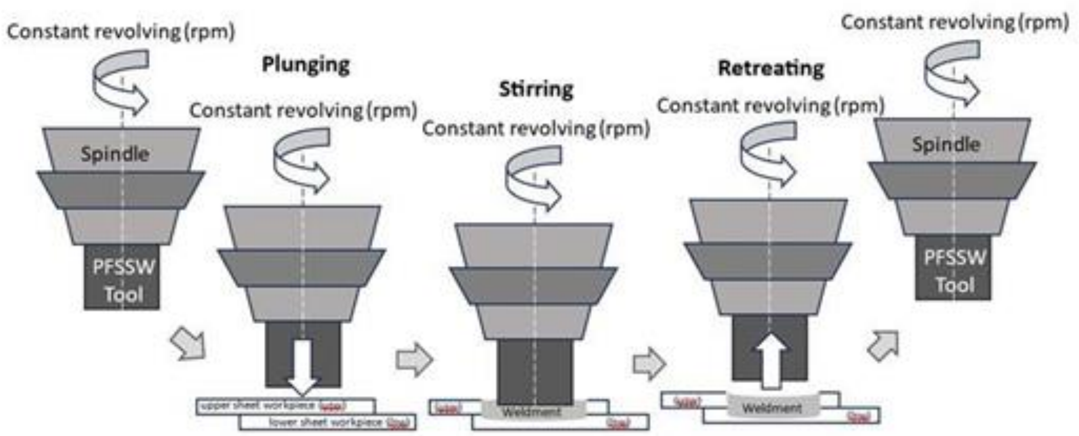


Fig. 1. Illustration of the principle of the friction stir welding process.

FSW is widely used in aerospace, automotive, and marine industries due to its ability to produce high-quality joints with low distortion. Because the process operates below the material's melting temperature, it results in a narrow heat-affected zone (HAZ), improved mechanical properties, and reduced residual stress and distortion [6][8]. The process is particularly suitable for aluminum alloys like 5052, which are known for their corrosion resistance, moderate strength, and excellent workability.

1.2. Fatigue

Fatigue is the progressive and localized structural damage that occurs when a material is subjected to cyclic loading. It typically initiates at stress concentrations, such as surface imperfections, and propagates until fracture [10]. The microstructure critically influences the fatigue behavior of FSW joints in the stir zone, the quality of surface finish, residual stresses, and defects like voids or tunnel defects formed during welding. Fatigue is a condition in which a material experiences damage due to fluctuating stress, where the magnitude of the stress is lower than the yield strength limit and the ultimate tensile strength limit of the material, which is given a constant load [10]. The treatment and operational temperature conditions influence the fatigue resistance of a material. Surface treatment can change the condition and residual stress on the surface of a material. For example, residual compressive stress can be generated using the shot peening method to increase fatigue resistance [11]. There is the highest concentration of compressive or tensile stress in a material. If tensile stress is applied to the material, residual compressive stress will cause greater compressive stress. This stress is the determining factor in the formation of initial cracks (crack initiation) or the rate of propagation of cracks [12]. As a result, fatigue resistance increases. Conversely, if there is residual tensile stress on the surface, fatigue resistance can decrease. In general, fatigue failure begins with the formation of cracks in the material or test object.

Fatigue strength testing applies a certain stress level to the sample or specimen until the sample fractures after several loading cycles [13]. Cracks in fatigue generally start at the surface of bending and torsion forces, creating high stress, or stress concentrations caused by uneven area positions. Therefore, the quality of a specimen or material significantly affects the fatigue endurance limit value [14]. Fatigue failure caused by the formation of cracks on the test object's surface proves that fatigue properties are susceptible to surface conditions. Several factors can influence surface conditions, such as changes in material properties, material roughness, and residual material stress. The resulting fatigue data is usually presented as a Wöhler curve, which maps the relationship between stress (S) and the number of failure cycles (N), Fig. 2.

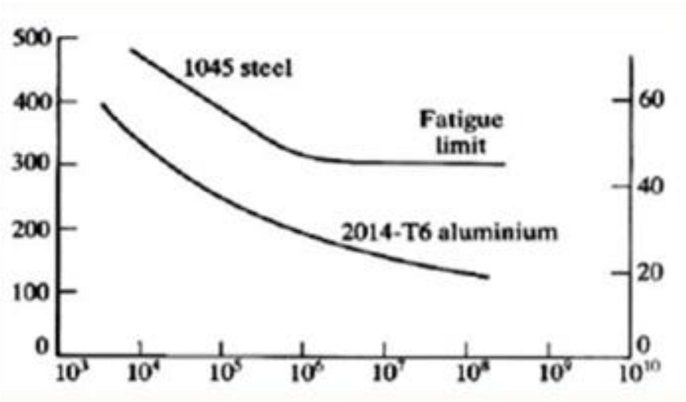


Fig. 2. S-N curve [15]

Studies have shown that residual compressive stresses, introduced through methods like shot peening, can significantly enhance fatigue resistance by impeding crack initiation and propagation [11][12]. Conversely, residual tensile stresses reduce fatigue life by facilitating crack growth. Fatigue testing is commonly represented by Wöhler curves (S–N curves), which plot the applied stress versus the number of cycles to failure. These curves are particularly sensitive to surface conditions, material roughness, and internal defects, emphasizing the importance of high-quality surface finishing in fatigue-critical applications [13][14]. While the fatigue behaviour of various aluminium alloys joined by FSW has been widely studied, particularly alloys like 2024, 6061, and 7075, there is a relative scarcity of detailed fatigue analysis for FSW joints in 5052 aluminium alloy. Previous studies have primarily focused on tensile strength, hardness, and microstructure characterization of FSW 5052 joints, but comprehensive fatigue performance data, especially under varying stress levels and thermal conditions, remains limited. This knowledge gap forms the basis of the present study, which aims to evaluate and characterize the fatigue behavior of 5052 aluminum alloy FSW joints under cyclic loading

1.3. Finite element method

The finite element method (FEM) is a numerical approach used to solve mathematical and engineering problems by dividing a complex structure or system into several simpler elements. Each element part is then analyzed separately by considering the relationships and interactions between the elements. In this way, the complexity of the problem can be overcome by breaking it down into parts that are easier to analyze, thus allowing for accurate and efficient numerical solutions [16]. The basic concept of FEM is to divide a structure into smaller or simpler parts with a certain number of limitations. After that, a combined analysis of the small elements is carried out. The purpose of FEM is to obtain a numerical solution that is close to the actual solution, so that it can be implemented with the help of a computer. Therefore, this method is often referred to as a computer-oriented method. The approximate values can be calculated quickly and accurately using a computer, allowing for more efficient problem solving in analysing structures and other phenomena [17].

The meshing process is important in an analysis using FEM. The mesh in a model refers to the network of elements and points forming a discrete structure. Mesh density refers to the number of finite elements contained in the model, and the greater the number of elements used, the higher the mesh density. When performing analysis on a model, changing the mesh can result in a more accurate and in-depth analysis. Generally, analysis results improve when mesh density increases in areas experiencing high stress [18]. The accuracy of FEM simulations heavily depends on the meshing process, where finer meshes are used in high-stress areas to enhance accuracy. Increasing mesh density can improve the predictive power of the model, but at the cost of higher computational resources [18][19]. However, it is important to remember that not all analysis results will always be the same, and optimal results may vary depending on the characteristics and nature of the analyzed model [19].

FEM has been extensively applied to study FSW joints, enabling prediction of stress distributions and identification of critical areas for fatigue crack initiation. In the case of 5052 aluminum alloy, FEM can be used to simulate the thermal and mechanical cycles during welding and subsequent loading, offering valuable insights into fatigue behavior.

2. METHODOLOGY

2.1. Research Stages

In this session, the research plan and method are explained. The research flowchart can be seen in Fig. 3. The first step begins with conducting a literature study using various references, both books and the Internet, to find guidelines for the completeness of research and knowledge related to FSW. Then, the preparation of research equipment and infrastructure that focuses on FEM simulation continues.

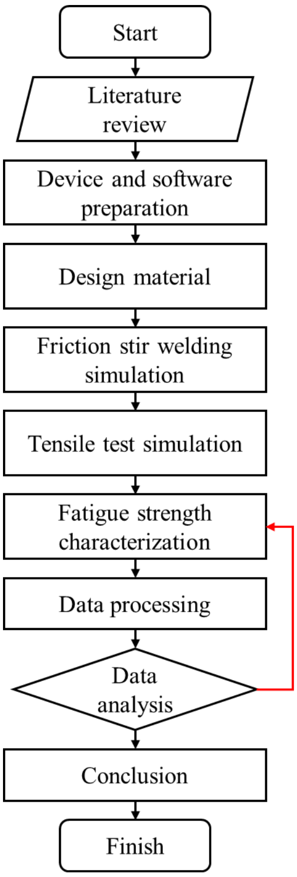


Fig. 3. Research flow chart.

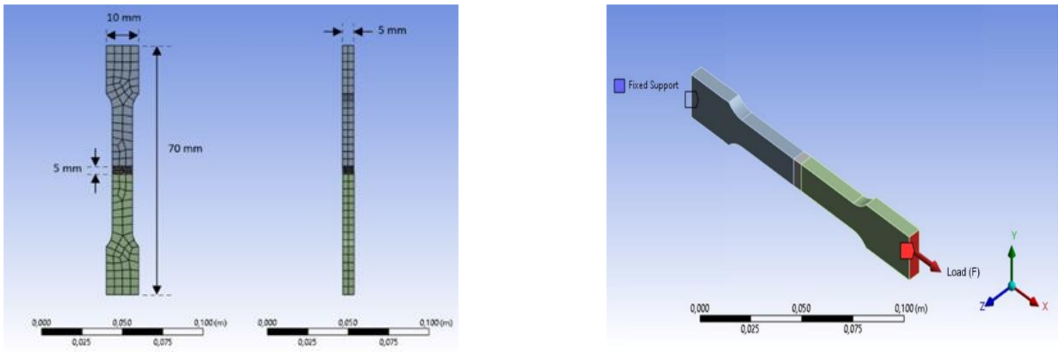
2.2. Specimen Design and Parameter Determination

The next step is testing the specimen's design for fatigue strength, namely the 5052 Aluminum alloy plate with ASTM E466 standards and a plate thickness of 5 mm. In this stage, the structural design of the 5052 Aluminum material uses the SolidWorks software. The parameters determined to perform the simulation are shown in Table 1.

Table 1: Specimen parameter

Parameter Description	Description
Aluminum thickness	5 mm
Element type	Quadrilateral
Maximum load (Newton)	20000 N
Gradual load (Newton)	5000 N, 10000 N, 15000 N, 20000 N
Gravitational acceleration	9,81 m/s2
Element size (default)	7,7982e-003m
Number of nodes	1938
Number of elements	441

Before the test simulation is carried out, it is necessary to determine the support point or fixed support located at one end of the specimen side. The meshing process divides the simulation domain into small elements that allow numerical calculations. At this stage, meshing is carried out on the test material with specimen specifications, as seen in Fig 4.



(a) (b)

Fig. 4. (a) Specimen dimensions, (b) Boundary conditions.

3. RESULT AND DISCUSSION

3.1. Results

In ANSYS 16.1, the workbench is applied to the test specimen sample until the part is deformed due to fatigue load, so its fatigue life can be estimated. A finite element simulation for a fatigue test is processed based on the stress applied to the stress ratio R-1. Figure 5 shows the numerical results of the Ansys software, which produces a maximum equivalent stress of 9.4423e7 Pa (Pascal) at a loading of 5000 N.

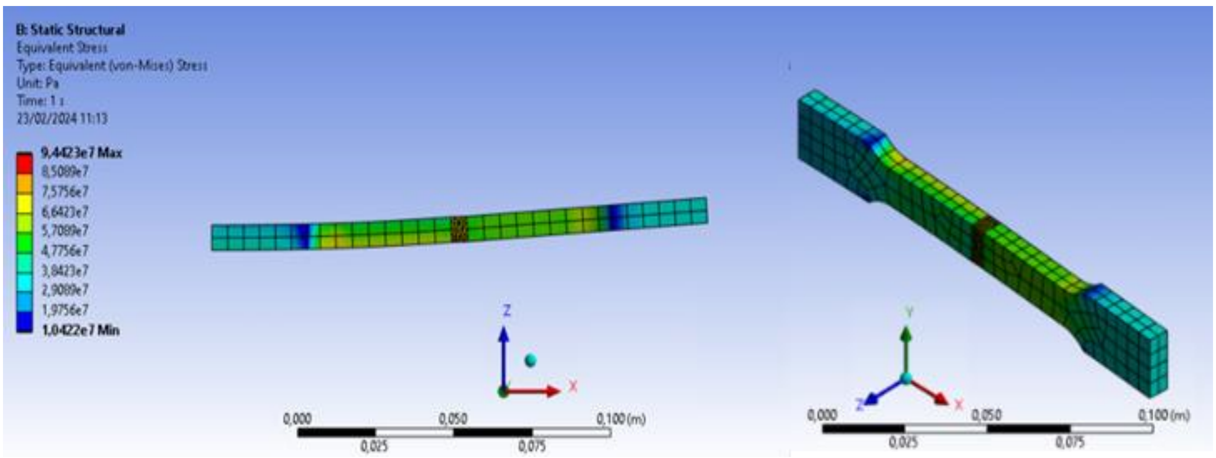
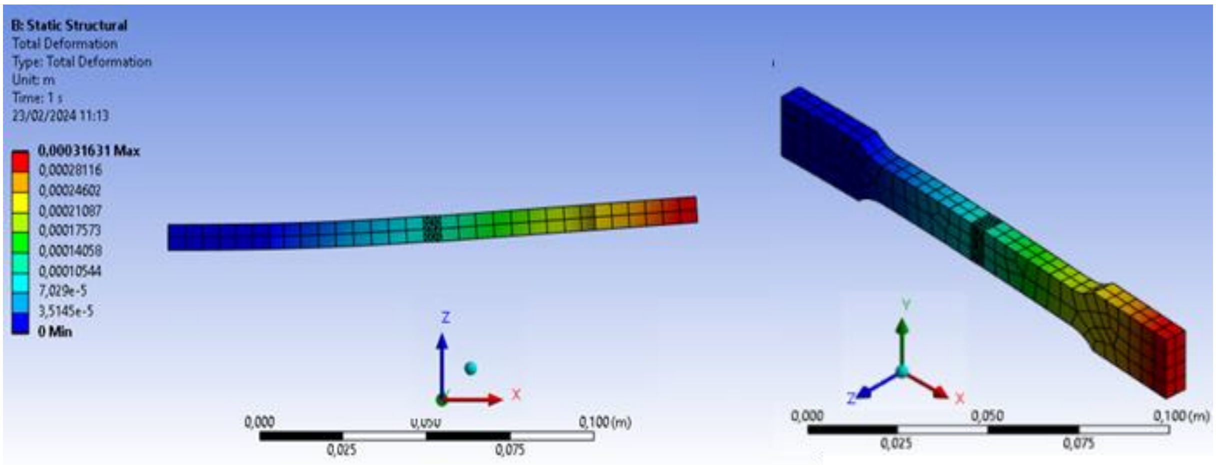
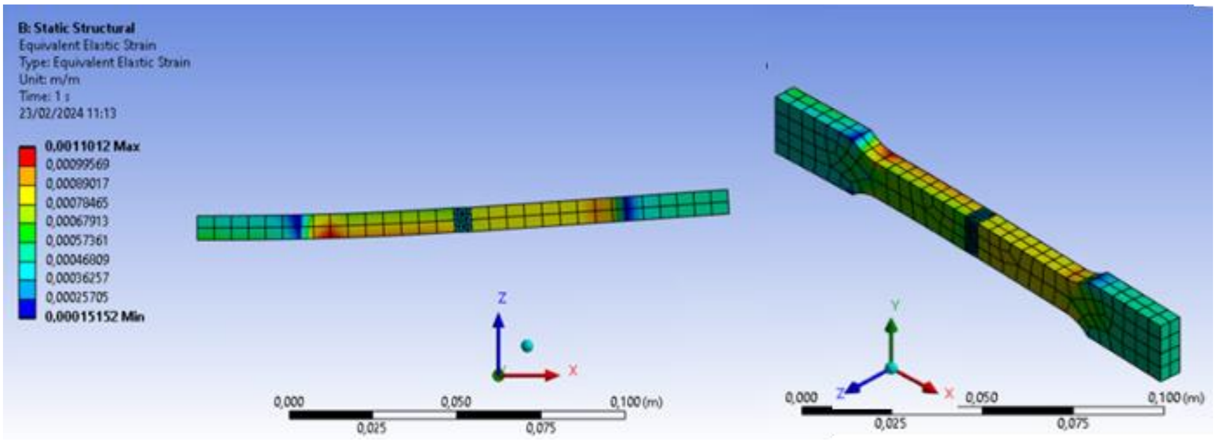


Fig. 5. The results of numerical simulations using Ansys software produce equivalent stress on virtual test objects at a load of 5000 N.

Fig. 6 shows the numerical results of the Ansys software for deformation and elastic strain at a loading of 5000 newtons, respectively. From the results of the numerical simulation, the maximum deformation of the test object was obtained at 0.00031631 m or 0.32 mm (Fig. 6a), and the maximum elastic strain, the maximum value was obtained at 0.0011012 m/m (Fig. 6 b). Fig 7 shows the numerical results of the Ansys software, which produces a maximum equivalent stress in the range of 1.9038e8 Pa (Pascal) at a loading of 10,000 N.



(a)



(b)

Fig. 6. The results of numerical simulations using Ansys with a loading of 5000 N obtained maximum results in (a) Total deformation, and (b) equivalent elastic strain.

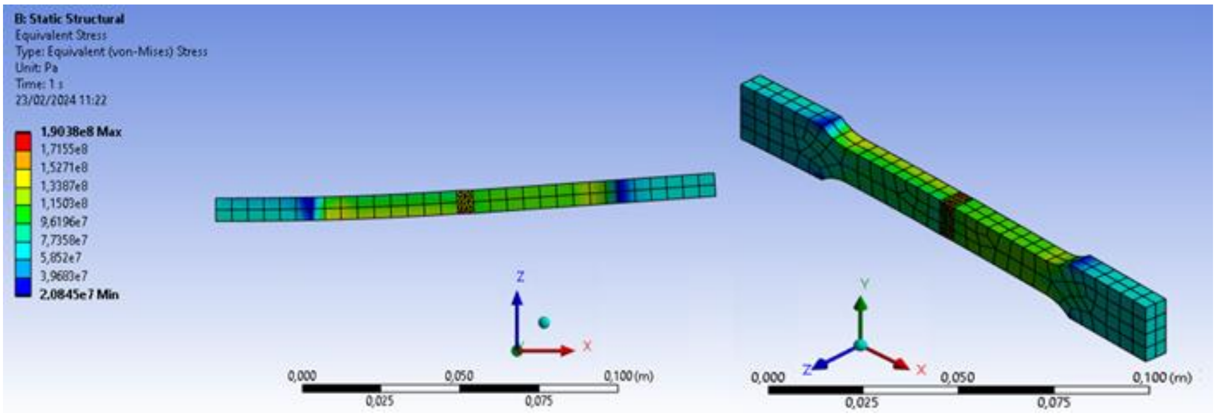
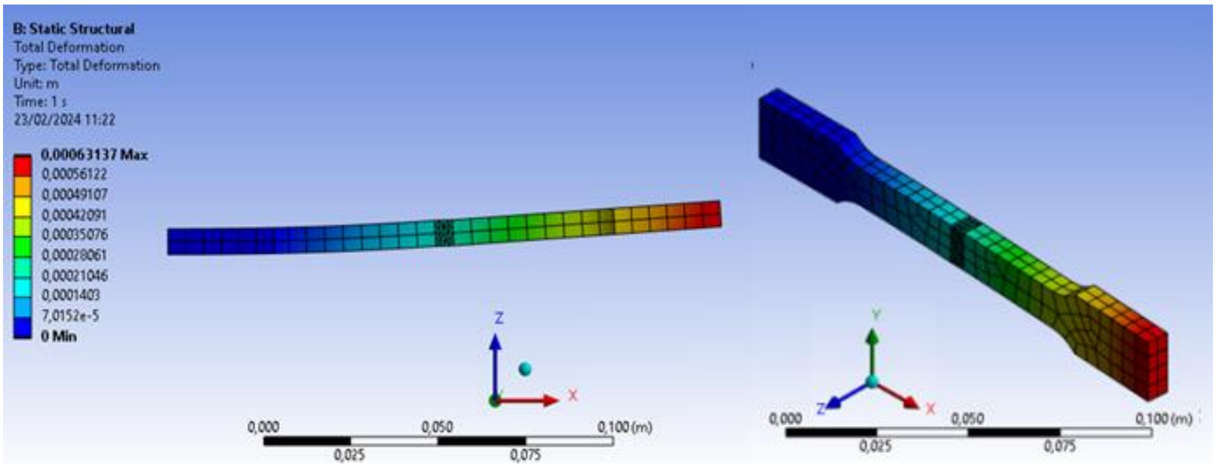
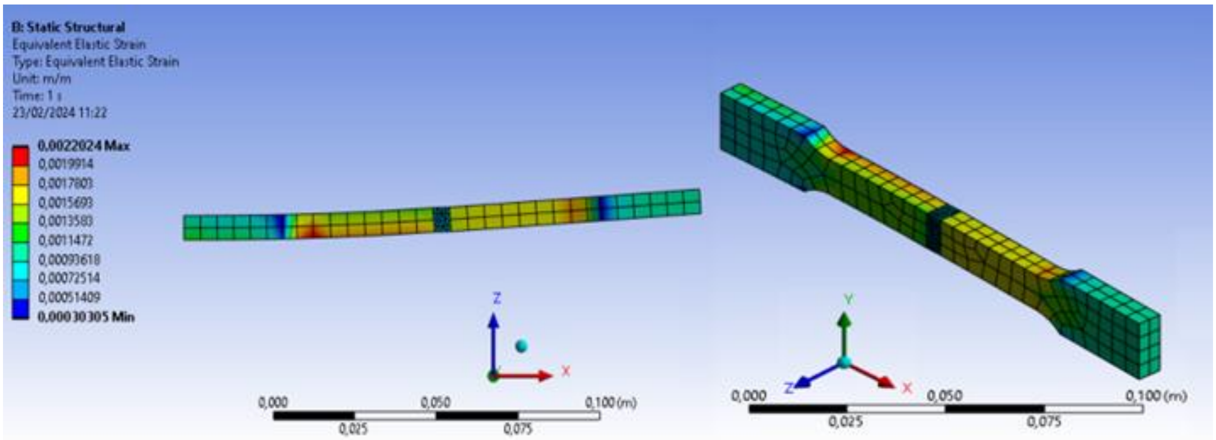


Fig. 7. shows the results of numerical simulations using Ansys software, which produce equivalent stress on virtual test objects at a load of 10,000 N.

Fig. 8 shows the numerical results of the Ansys software for deformation and elastic strain at a loading of 10,000 newtons, respectively. From the results of the numerical simulation, the maximum deformation of the test object was obtained at 0.00063137 m or 0.63 mm (Fig. 8-a), and the maximum elastic strain, the maximum value was obtained at 0.0022024 m/m (Fig. 8-b).



(a)

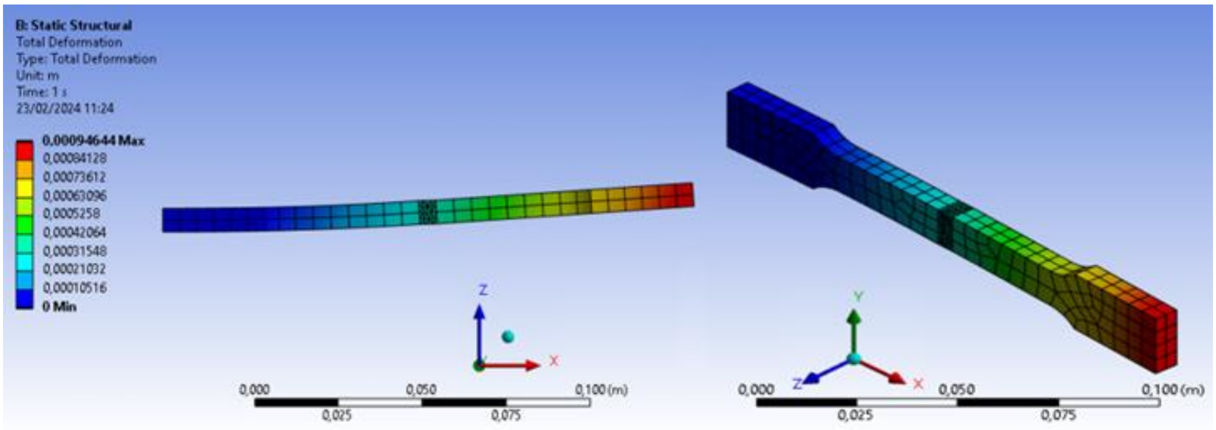


(b)

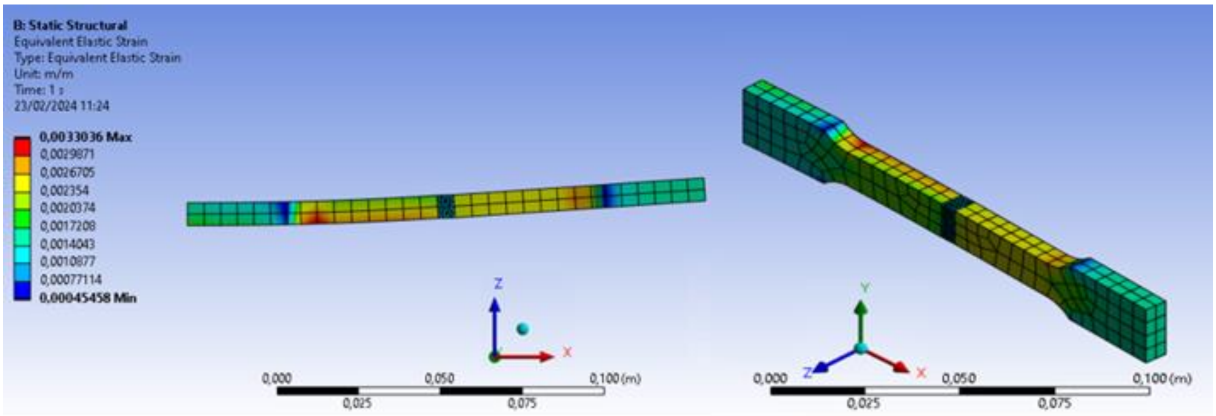
Fig. 8. shows the results of numerical simulations using Ansys software that produce equivalent stress on virtual test objects at a load of 15,000 N.

Fig. 9 shows the numerical results of the Ansys software for deformation and elastic strain at a loading of 15,000 newtons, respectively. From the results of the numerical simulation, the maximum deformation of the test object was obtained at 0.00094644 m or 0.95 mm (Fig. 9-a), and the maximum elastic strain, the maximum value was obtained at 0.0033036 m/m (Fig. 9-b). Fig 10 shows the numerical results of the Ansys software, which produces a maximum equivalent stress of 3.8231e^{8} Pa (Pascal) at a loading of 20,000 N.

https://doi.org/10.51630/ijes.v6i2.177



(a)



(b)

Fig 9. The results of numerical simulations using Ansys with a loading of 15,000 N obtained maximum results in (a) Total deformation, and (b) equivalent elastic strain.

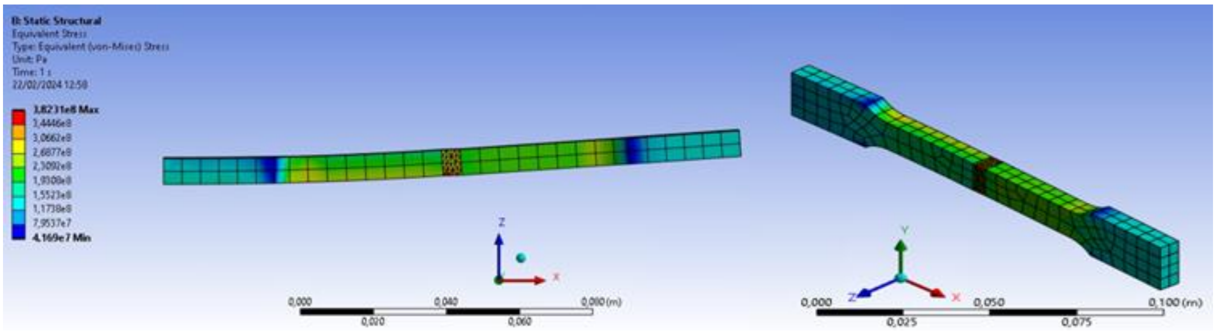
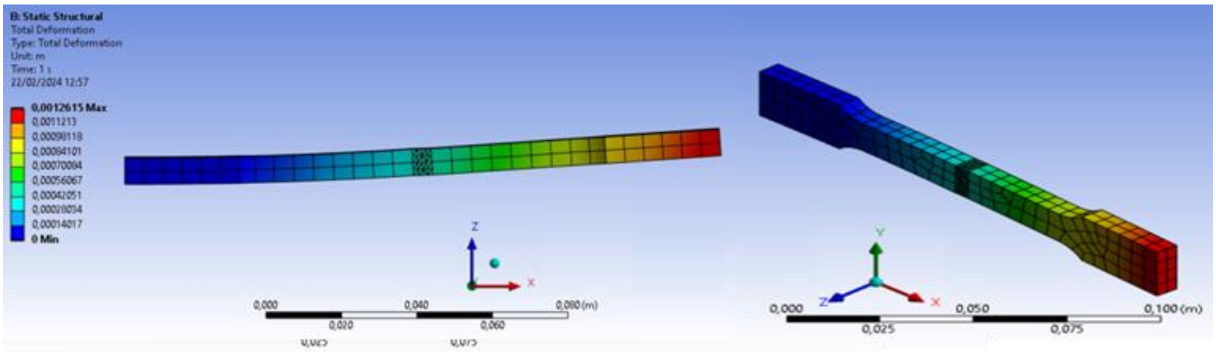
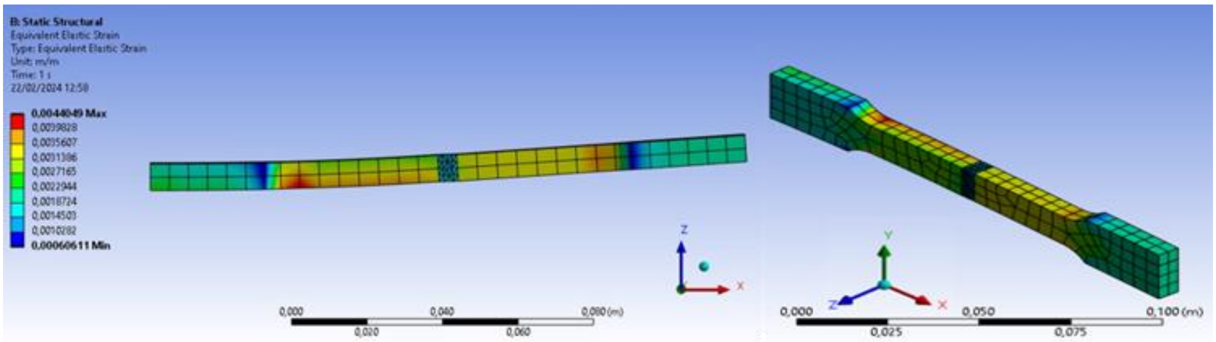


Fig. 10. The results of numerical simulations using Ansys software produce equivalent stress on virtual test objects at a load of 20,000 N.

Fig. 11 shows the numerical results of the Ansys software for deformation and elastic strain at a loading of 20,000 newtons, respectively. From the results of the numerical simulation, the maximum deformation of the test object was obtained at 0.0012615 m or 1.26 mm (Fig. 11-a), and the maximum elastic strain, the maximum value was obtained at 0.0044049 m/m (Fig. 11-b).



(a)



(b)

Fig. 11. The results of numerical simulations using Ansys with a loading of 20,000 N obtained maximum results in (a) total deformation, and (b) equivalent elastic strain.

Table 2 compares fatigue loading of 5000 N, 10,000 N, 15,000 N, and 20,000 N based on equivalent stress, deformation, and equivalent static strain in tabulation. It is clear that at each increase in fatigue loading, there is an increase in equivalent stress on the test object from about 9.44e7 Pa to about 3.82e8 Pa. In deformation, there is an increase from 0.32 mm to 1.26 mm; in equivalent static strain, there is an increase from 0.001 m/m to 0.004 m/m.

Table 2: Comparison of fatigue loading results on test specimens.

Fatigue load	Equivalent stress (max.)	Deformation (max.)	Equivalent elastic strain (max.)
5000 N	9,4423e7 Pa	0,32 mm	0,0011012 m/m
10,000 N	1,9038e8 Pa	0,63 mm	0,0022024 m/m
15,000 N	2,863e8 Pa	0,95 mm	0,0033036 m/m
20,000 N	3,8231e8 Pa	1,26 mm	0,0044049 m/m

Fig. 12 shows the numerical behavior of fatigue life (S-N) curves on each test specimen, compared through changes in fatigue loading with fatigue life behavior. The SN curve shows that the fatigue life at loading of 10,000 to 20,000 N is below 500,000 cycles, and at loading between 5000 N and 10,000 N, it shows fatigue life between 500,000 and 6,000,000 cycles.

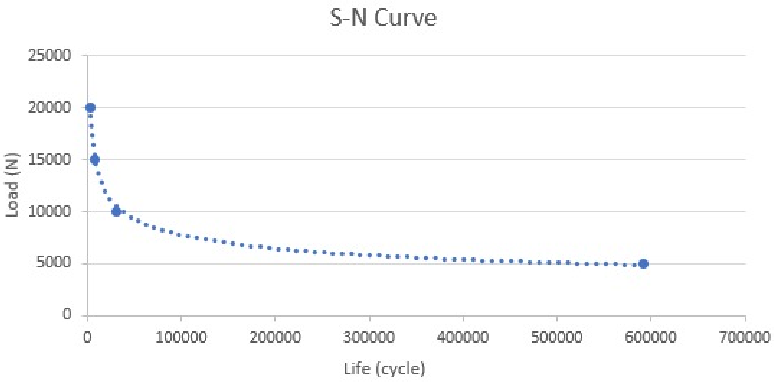


Fig 12. S-N curve based on weld strength against loading cycles.

3.1. Discussion

The numerical results obtained from the ANSYS Workbench simulations demonstrate a clear and consistent trend of increasing equivalent stress, elastic strain, and deformation with increasing cyclic loading. This behavior agrees with the basic principles of fatigue mechanics, where higher applied stress amplitudes result in accelerated material degradation and reduced fatigue life.

Compared with existing literature, the findings align well with previously reported trends in fatigue behavior of aluminum alloys welded using the Friction Stir Welding (FSW) method. For instance, Elangovan and Balasubramanian (2008) studied the fatigue properties of FSW joints of AA6061 and noted a significant influence of tool design and process parameters on fatigue performance. While their focus was on a different alloy, their conclusions regarding the role of residual stress and microstructural uniformity are consistent with the present findings. Similarly, Mishra and Ma (2005) highlighted that the fine-grained microstructure and residual compressive stress within the stir zone of FSW joints contribute to improved fatigue resistance—an observation that supports the relatively long fatigue life observed under moderate loads (5000 N–10,000 N) in this study.

In the current analysis, fatigue life under a 5000 N load reached 6,000,000 cycles, demonstrating the material's high endurance limit at lower stress amplitudes. This endurance is likely attributable to the intrinsic properties of 5052 aluminum alloy—particularly its excellent ductility and resistance to crack initiation—and the advantages of FSW, which include minimized porosity, fine grain structure, and reduced residual tensile stress in the weld zone.

As the applied load increases, however, the fatigue life declines sharply. At 10,000 N, fatigue life begins to drop below 500,000 cycles, and by 20,000 N, the maximum equivalent stress nearly quadruples compared to the 5000 N case, reaching approximately 3.82×10^8 Pa. The deformation and strain levels also rise accordingly, indicating the material approaches its elastic-plastic threshold, potentially entering a region of microstructural instability and fatigue crack propagation.

Comparing these results with the work of Sato et al. (1999), who investigated the fatigue crack initiation and growth in aluminum FSW joints, the current data further supports the conclusion that the heat-affected zone (HAZ) and thermo-mechanically affected zone (TMAZ) are typically the most vulnerable regions for crack initiation, particularly under higher loads. Although a detailed microstructural analysis was not included in this numerical simulation, the observed increase in deformation and strain at higher loads suggests that fatigue failure in the tested specimens may also originate from these transitional zones.

Additionally, the results are consistent with the fatigue model proposed by Schijve (2009), which predicts that surface roughness, micro-defects, and residual tensile stresses accelerate fatigue crack initiation and reduce fatigue life. While surface roughness was not a variable in this simulation, the sharp decline in fatigue life at higher loads implies the onset of crack initiation due to local stress concentrations, which may be exacerbated by geometric discontinuities or imperfections introduced during the FSW process.

Overall, the simulation confirms that:

- 5052 aluminum alloy FSW joints exhibit excellent fatigue resistance at lower cyclic loads, supporting their use in structural applications where moderate dynamic loading is expected.
- A rapid decrease in fatigue life at higher load points indicates the need for additional reinforcement, optimised welding parameters, or post-weld treatments (e.g., surface polishing, shot peening) when used in high-load applications.
- The finite element results correspond well with established S–N curve behavior, validating the model's accuracy and demonstrating its usefulness in predicting fatigue life under various loading conditions.

Future work could expand upon these findings by incorporating experimental fatigue tests, microstructural characterization of the weld zones, and a broader range of stress ratios (R-values) to evaluate mean stress effects and multiaxial fatigue behavior. Additionally, validating the FEM model against actual experimental data would further improve the reliability of predictions for industrial applications.

4. CONCLUSION

From this study, the mechanical properties of FSW weld material under fatigue loading on Aluminum alloy 5052 were tested through numerical simulation with the finite element method using Ansys software at 5000 N, 10,000 N, 15,000 N, and 20,000 N loads imposed on the test specimen in the form of a 3-dimensional model. The results of the numerical simulation study showed a significant increase in equivalent stress from around 9.44e7 Pa to around 3.82e8 Pa. The deformation changed from 0.32 mm to 1.26 mm, and the equivalent static strain increased from 0.001 m/m to 0.004 m/m. Then, the presentation of the S/N curve shows that the fatigue life at a loading of 10,000 to 20,000 N is at a cycle below 500,000 cycles, and a loading between 5000 N and 10,000 N, the fatigue life is between 500,000 and 6,000,000 cycles.

This study presents a detailed numerical investigation into the fatigue performance of Friction Stir Welded (FSW) 5052 aluminum alloy joints under various cyclic loading conditions using the Finite Element Method (FEM) in ANSYS Workbench. By simulating fatigue loads of 5000 N, 10,000 N, 15,000 N, and 20,000 N on a 3D model, this research quantifies the mechanical response in terms of equivalent stress, elastic strain, and deformation while assessing fatigue life behavior through S–N curve analysis.

Providing quantitative fatigue data for FSW joints of 5052 aluminum alloy, a material widely used in aerospace and marine industries, yet underrepresented in detailed fatigue simulations. Demonstrating a clear correlation between increasing cyclic load and reductions in fatigue life, with stress and strain rising significantly, from 9.44×10^7 Pa to 3.82×10^8 Pa, and strain from 0.001 m/m to 0.004 m/m. Showing that fatigue life drops below 500,000 cycles at high loads (10,000–20,000 N) while maintaining up to 6 million cycles at lower loads, highlighting critical load thresholds for fatigue performance.

This work provides valuable insights for design engineers and manufacturers working with FSW joints in structural applications. It supports optimizing operational load ranges to extend component lifespan and reduce maintenance frequency. To build upon these findings, future research should consider the following:

- Experimental validation of the numerical simulations to confirm model accuracy and assess real-world behavior under complex fatigue conditions.
- Microstructural analysis of the weld zones to identify crack initiation sites and better understand the role of FSW-induced grain refinement and residual stresses.
- Multiaxial and variable amplitude loading conditions, which more accurately reflect service environments.
- Investigation of surface treatments (e.g., shot peening, polishing) and post-weld heat treatments to enhance fatigue performance.
- Comparison with other aluminum alloys or joining techniques to benchmark the fatigue resistance of FSW joints.

By expanding the scope to include these aspects, future studies can further optimize FSW parameters and improve the fatigue resilience of aluminum structures in critical applications.

ACKNOWLEDGEMENT

The author would like to express special gratitude to the Atma Jaya Catholic University of Indonesia, DKI Jakarta, Indonesia, and the Faculty of Engineering, National Development University "Veteran" Jakarta, Indonesia, for participating in the 2025 Engineering Profession Study Program (RPL) and for all the support provided.

REFERENCES

[1] X. Ge, D. Jiang, W. Song, and H. Wang, "Effects of Tool Plunging Path on the Welded Joint Properties of Pinless Friction Stir Spot Welding," Lubricants, vol. 11, no. 3, p. 150, Mar. 2023, doi: 10.3390/lubricants11030150.

[2] Z. Wang, R. Ye, and J. Xiang, "The performance of textured surface in friction reducing: A review," Tribol Int, vol. 177, p. 108010, Jan. 2023, doi: 10.1016/j.triboint.2022.108010.

[3] Armansyah, H. H. Chie, J. Saedon, and S. Adenan, "Temperature distribution in friction stir spot welding of aluminium alloy based on finite element analysis," IOP Conf Ser Earth Environ Sci, vol. 426, no. 1, p. 012127, Feb. 2020, doi: 10.1088/1755-1315/426/1/012127.

[4] Mohd Saiful Bahari, M. Shamil Jaffarullah, Zulkifli Mohamed, and Armansyah, "Heat analysis in friction stir welding using finite element method," Journal of Mechanical Engineering, vol. SI 5, no. 4, pp. 174–188, Mar. 2018.

[5] B. H. Abed, O. S. Salih, and K. M. Sowoud, "Pinless friction stir spot welding of aluminium alloy with copper interlayer," Open Engineering, vol. 10, no. 1, pp. 804–813, Sep. 2020, doi: 10.1515/eng-2020-0090.

[6] R. Z. Xu, D. R. Ni, Q. Yang, C. Z. Liu, and Z. Y. Ma, "Pinless Friction Stir Spot Welding of Mg–3Al–1Zn Alloy with Zn Interlayer," J Mater Sci Technol, vol. 32, no. 1, pp. 76–88, Jan. 2016, doi: 10.1016/j.jmst.2015.08.012.

[7] Q. Chu, X. W. Yang, W. Y. Li, and Y. B. Li, "Microstructure and mechanical behaviour of pinless friction stir spot welded AA2198 joints," Science and Technology of Welding and Joining, vol. 21, no. 3, pp. 164–170, Apr. 2016, doi: 10.1179/1362171815Y.0000000078.

[8] S. S. Parkhe and R. J. Patil, "A Review of Friction Stir Welding and Processing on Aluminium Alloys," Journal of Mines, Metals and Fuels, pp. 2473–2492, Dec. 2023, doi: 10.18311/jmmf/2023/34919.

[9] Jeroen De Backer, Sam Wei, and Jonathan Martin, "Robotic FSW for three-dimensional components," in SEEIIW2015 Romania, 2015.
[10] C. Niu, S. Xie, and T. Zhang, "Research on anti-fatigue design method of welded structure oriented to stiffness coordination strategy," International Journal of Structural Integrity, vol. 13, no. 2, pp. 196–211, Mar. 2022, doi: 10.1108/IJSI-10-2021-0115.
[11] C. Niu, S. Xie, and T. Zhang, "A New Anti-fatigue Design Method for Welded Structures Based on Stiffness Coordination Strategy and Its Application," 2022, pp. 93–100. doi: 10.1007/978-3-030-97822-8_10.
[12] Peng, Chen, and Dong, "Experimental Data Assessment and Fatigue Design Recommendation for Stainless-Steel Welded Joints," Metals (Basel), vol. 9, no. 7, p. 723, Jun. 2019, doi: 10.3390/met9070723.
[13] L. Long, X. Zhang, S. Gu, X. Li, X. Cheng, and G. Chen, "Experimental and Simulation Investigation on Fatigue Performance of H13 Steel Tools in Friction Stir Welding of Aluminum Alloys," Materials, vol. 17, no. 7, p. 1535, Mar. 2024, doi: 10.3390/ma17071535.
[14] V. P. Pervadchuk and A. R. Davydov, "Fatigue tests simulation of materials with a random endurance limit," J Phys Conf Ser, vol. 1730, no. 1, p. 012006, Jan. 2021, doi: 10.1088/1742-6596/1730/1/012006.
[15] G. Li, H. Zhang, and W. Zang, "Fatigue Crack Growth Performance in Refilled Friction Stir Spot Welding and Friction Stir Welding of Aluminum Alloy Joints," 2024, pp. 511–524. doi: 10.1007/978-981-99-8861-7_51.
[16] E. M. M. Mubarak, "Experimental and Numerical Study of 6061-T6 AL- Alloy Fatigue Life using Friction Stir Welding," The Iraqi Journal for Mechanical and Materials Engineering, vol. 18, no. 2, pp. 277–295, Aug. 2018, doi: 10.32852/iqjfmme.Vol18.Iss2.93.
[17] J. Langari, K. Aliakbari, and F. Kolahan, "Fatigue life simulation of AA7075-T651 FSW joints using experimental data," Eng Fail Anal, vol. 154, p. 107690, Dec. 2023, doi: 10.1016/j.engfailanal.2023.107690.
[18] N. J. Aghdam, S. Hassanifard, M. M. Ettefagh, and A. Nanvayesavojblaghi, "Investigating Fatigue Life Effects on the Vibration Properties in Friction Stir Spot Welding Using Experimental and Finite Element Modal Analysis," Strojniški vestnik – Journal of Mechanical Engineering, vol. 60, no. 11, pp. 735–741, Nov. 2014, doi: 10.5545/sv-jme.2013.1324.
[19] Y. Jin, Y. Ma, Y. Chen, X. Wang, and T. Guo, "Residual stress effects on fatigue crack growth in different regions of aluminum alloy 7050 friction stir weld," Fatigue Fract Eng Mater Struct, vol. 46, no. 3, pp. 1093–1106, Mar. 2023, doi: 10.1111/ffe.13923.