African Journal of Advanced Pure and Applied Sciences (AJAPAS)
Online ISSN: 2957-644X
Volume 4, Issue 3, 2025
Page No: 83-92
Website: https://aaasjournals.com/index.php/ajapas/index
ISI 2025: 1.126 SJIFactor 2024: 6.752 معامل التأثير العربي: 1.62

Friction Stir Welding Simulation of AA6061-T6 Aluminum Alloy: A SolidWorks-based investigation of lap joint configuration

Nedal Musbah Almeraash [1], Omar Ahmed Edbeib [2*]
[1] College of Technical Sciences, Bani Walid, Libya
[2] Mechanical and Industrial Engineering Department, Faculty of Engineering, Bani Waleed University, Bani Walid, Libya

محاكاة لحام التحريك الاحتكاكي (*FSW*) لسبائك *Aluminum alloy 6061- T6*: دراسة تطبيقية باستخدام *SOLIDWORKS* لوصلة تراكبية

نضال مصباح معتوق المرعاش [1]، عمر أحمد محمد الدبيب [2*]
[1] كلية العلوم التقنية، بني وليد، ليبيا
[2] قسم الهندسة الميكانيكية والصناعية، كلية الهندسة، جامعة بني وليد، بني وليد، ليبيا

*Corresponding author: aldbib220@yahoo.com

Received: May 23, 2025	Accepted: July 09, 2025	Published: July 19, 2025

Abstract
This study presents a detailed simulation-based investigation of Friction Stir Welding (FSW) applied to AA6061-T6 aluminum alloy in a lap joint configuration, utilizing SolidWorks Simulation. Two 150×50×6 mm plates, overlapped by 25 mm, were welded using typical FSW parameters. The simulation revealed a peak temperature of 470°C, significantly below the alloy's melting point, confirming solid-state welding. Mechanical analyses showed a safe von Mises stress of 137 MPa, an ultimate tensile strength of 240 MPa (77% of the base material), and minimal deformation of approximately 0.18 mm. These outcomes demonstrate FSW's ability to produce strong, defect-free joints without melting and validate SolidWorks as a reliable tool for simulating heat flow and stress distribution in the FSW process.

Keywords: Friction Stir Welding (FSW), AA6061-T6 Aluminum Alloy, SolidWorks Simulation, Lap Joint, Thermo-mechanical Analysis.

الملخص
تقدم هذه الدراسة تحقيقًا تفصيليًا قائمًا على المحاكاة لنمذجة عملية اللحام بالاحتكاك الدوراني (FSW) المطبقة على سبيكة الألومنيوم AA6061-T6في تكوين وصلة تراكبية، وذلك باستخدام برنامج SolidWorks للمحاكاة. تم لحام صفيحتين بأبعاد 150×50×6 مم، مع تراكب بمقدار 25 مم، باستخدام معلمات FSW النموذجية. كشفت المحاكاة أن درجة الحرارة القصوى التي تم الوصول إليها بلغت 470 درجة مئوية، وهي أقل بكثير من نقطة انصهار السبيكة، مما يؤكد حدوث اللحام في الحالة الصلبة. أظهرت التحليلات الميكانيكية إجهاد فون ميز آمن بلغ 137 ميجا باسكال، وقوة شد قصوى قدرها 240 ميجا باسكال (أي ما يعادل 77% من قوة المادة الأساسية)، بالإضافة إلى تشوه طفيف بلغ حوالي 0.18 مم. تثبت هذه النتائج قدرة FSW على إنتاج وصلات قوية وخالية من العيوب دون انصهار، وتؤكد موثوقية برنامج SolidWorks كأداة فعالة لمحاكاة تدفق الحرارة وتوزيع الإجهاد في عملية اللحام بالاحتكاك الدوراني.

الكلمات المفتاحية: اللحام بالاحتكاك الدوراني(FSW) ، سبيكة الألومنيوم AA6061-T6 ، محاكاة SolidWorks ، وصلة تراكبية، التحليل الحراري الميكانيكي.

1. Introduction
Friction Stir Welding (FSW), introduced in 1991 by The Welding Institute (TWI), is a solid-state welding technique that joins materials without melting them making it particularly suitable for aluminum alloys like AA6061-T6, which typically suffer from defects in conventional fusion welding. The FSW process employs a rotating non-consumable tool that generates localized heat through friction, softening the material and enabling

solid-state mixing and bonding. Despite the advantages of FSW, optimizing its parameters such as tool rotation speed, traverse speed, tilt angle, and axial force remains a challenge due to the complex interaction of thermal and mechanical behaviors during welding. Improper parameter selection may lead to defects like voids or weak mechanical properties. Traditional experimental approaches are often costly and time-intensive. Therefore, this study aims to develop a predictive, simulation-based model using SolidWorks to investigate the FSW of AA6061-T6 in a lap joint configuration. The objective is to analyze heat generation, temperature distribution, residual stresses, and mechanical strength, thus providing an efficient virtual method to improve weld quality and guide future experimental validation.

2. Methodology

This study employs a comprehensive numerical simulation approach to investigate the thermal and mechanical behavior of Friction Stir Welding (FSW) applied to AA6061-T6 aluminum alloy using SolidWorks Simulation. The methodology is structured around the following key components:

2.1 Friction Stir Welding Setup

The FSW process used in this research is modeled based on an industrial-grade CNC-controlled machine, which includes a non-consumable rotating tool and a secure clamping system (Figure 1). The rotating tool is plunged into the lap joint between two aluminum plates and traversed along the seam to generate frictional heat, causing the material to plastically deform and consolidate below the melting point [1].

Figure 1. FSW method.

2.2 Tool Design and Specifications

The FSW tool was modeled with a concave shoulder and a tapered cylindrical pin, optimized for heat concentration and material flow. The tool is made from H13 tool steel due to its high toughness, wear resistance, and ability to withstand elevated temperatures. Several design enhancements such as Whorl, Trivex, and Triflate profiles are considered to promote better stirring and reduce the axial load. These features ensure stable welding, reduced defects, and effective material mixing during the simulation [2].

2.3 Workpiece Dimensions and Configuration

The simulation modeled two AA6061-T6 aluminum plates, each measuring 150 mm × 50 mm × 6 mm, with a 25 mm overlapping lap joint (Figure 2). This configuration replicates common structural applications in aerospace and automotive components

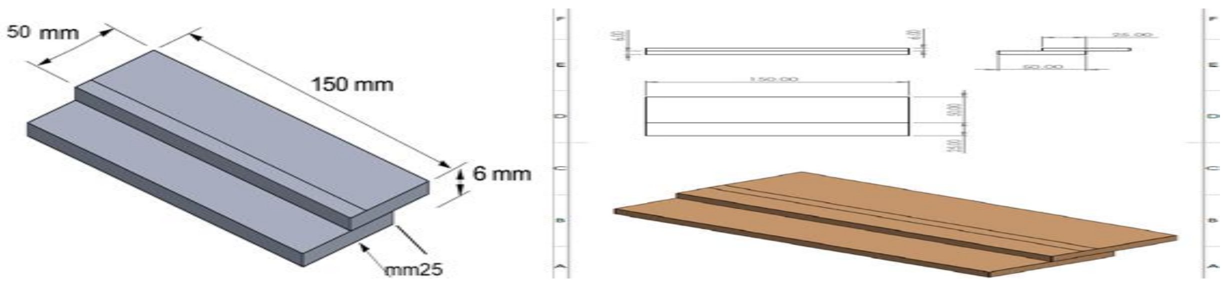


Figure 2. workpiece dimension.

2.4 Material Properties and Behavior

Temperature-dependent properties of AA6061-T6 were applied in the simulation, including thermal conductivity (~167 W/m.K), specific heat, density (2.7 g/cm³), and mechanical values such as yield strength (276 MPa) and ultimate tensile strength (310 MPa). The alloy's susceptibility to fusion welding defects makes it an ideal candidate for solid-state FSW.[4]

2.5 Finite Element Modeling in SolidWorks

A 3D finite element model was developed in SolidWorks Simulation. The following steps were followed:

- **Meshing:** A fine mesh was generated in the weld zone to accurately capture gradients of heat and stress.
- **Thermal Analysis:** A volumetric heat input of 5686 W was applied at the tool-pin interface based on analytical calculations. A transient thermal simulation was used to observe temperature distribution.
- **Mechanical Simulation:** Static structural analysis was performed using the same geometry. One end of the plate was fixed, and a tensile force of 10 kN was applied to the other end to replicate tensile testing. Von Mises stress and displacement were evaluated.[5]

2.6 Justification for FSW and AA6061-T6

FSW was selected for this study due to its low-heat, solid-state nature that avoids melting, preserving the T6 temper of AA6061 and minimizing heat-affected zone (HAZ) degradation. Compared to MIG/TIG, FSW produces stronger, defect-free welds without filler material or shielding gas. This is crucial for structural [5].

3. Design Data

3.1 FSW Process Parameters and Setup

The success of Friction Stir Welding largely depends on the accurate control of process parameters. These parameters directly influence the heat generation, material flow, and mechanical behavior of the welded joint. In this study, a standard lap joint configuration for AA6061-T6 aluminum alloy is considered Table 1. The chosen parameters are based on AWS recommendations and industrial practices.

Table 1. Typical Process Parameter Ranges (based on AWS D17.3 and literature).

Parameter	Typical Range	Selected Value (Midpoint)
Rotational Speed (RPM)	500 – 2000	1250 RPM
Traverse Speed (mm/min)	50 – 200	125 mm/min
Tilt Angle (°)	0 – 3	1.5°
Plunge Depth (mm)	0.1 – 0.3	0.2 mm
Axial Force (kN)	5 – 15	10 kN

Although no fixed standard exists for FSW of AA6061-T6, this study used midpoint values from AWS D17.3 and literature to ensure balanced heat input and defect-free welding [7].

3.2 Specimen Preparation and Dimensions

For this study, the specimens were prepared using AA6061-T6 aluminum alloy plates. Two identical plates were used, each with a length of 150 mm, width of 50 mm, and thickness of 6 mm. The plates were arranged in a lap joint configuration with an overlapping area of 25 mm along the length of 150 mm Figure 3. The surfaces to be joined were cleaned and mechanically prepared to ensure proper contact and minimal contamination prior to welding. This setup ensures consistent load distribution and allows for effective evaluation of weld quality during mechanical testing.

3.4. Heat Input Calculations (Part of Design Data)

In Friction Stir Welding (FSW), heat is generated primarily due to two mechanisms: friction between the rotating tool (shoulder and pin) and the workpiece, and plastic deformation of the material. The total heat input (Q_total) is the sum of the heat from the tool shoulder and the pin. The efficiency of conversion from mechanical work to heat is assumed to be 0.9, a common approximation for aluminum FSW.

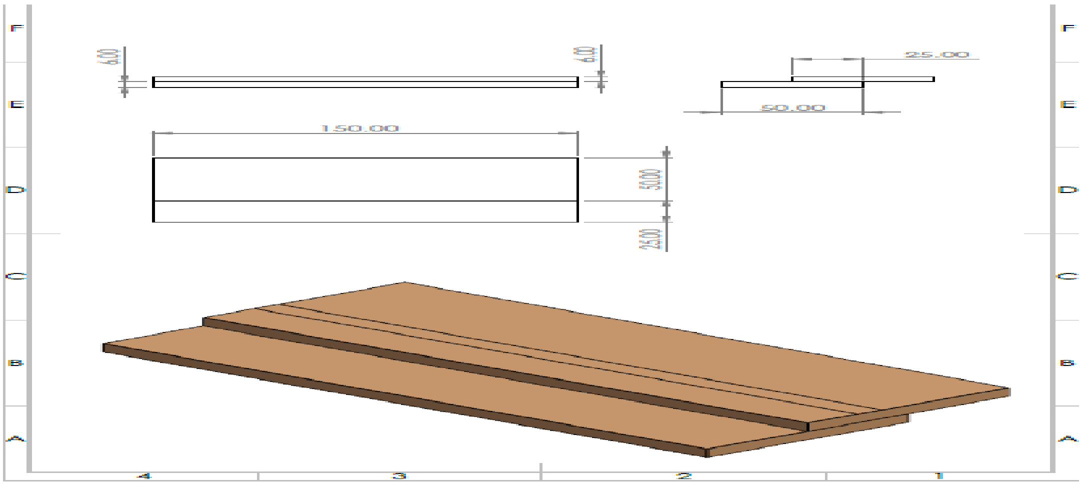


Figure 3. sample dimension (lap joint configuration).

3.5 Selected Parameters for Heat Input Calculation

The parameters selected for heat in put calculation are as shown in Table 2

Table 2. Selected Parameters for Heat Input Calculation.

Parameter	Value
Rotational Speed (N)	1250 RPM ≈ 130.9 rad/s
Shoulder Diameter	18 mm = 0.018 m
Pin Diameter	6 mm = 0.006 m
Pin Length	5.8 mm = 0.0058 m
Flow Shear Stress (τ)	30 MPa
Efficiency (k)	0.9

3.6 Heat Generation Equations

Heat from Shoulder:

$Q_{shoulder} = (3/2) \times \pi \times \omega \times R_{shoulder}^3 \times \tau \times k$

Heat from Pin:

$Q_{pin} = (1/2) \times \pi \times \omega \times R_{pin}^2 \times L_{pin} \times \tau \times k$

Where:

ω = Rotational Speed (rad/s), τ = Shear Stress, k = Efficiency

3.7 Final Heat Input Results

Table 3 final heat input.

Component	Heat (Watts)
Shoulder	5396.21 W
Pin	289.80 W
Total Heat Input	5686.01 W

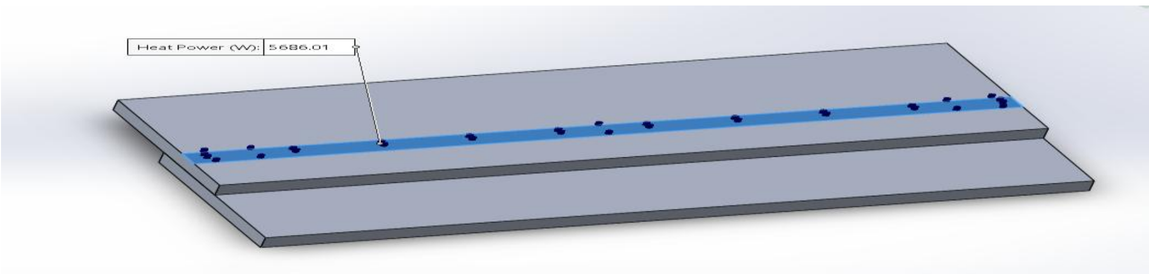


Figure 5: total heat input in SolidWorks.

4 Modeling and Simulations

SolidWorks is a solid modeling computer-aided design (CAD) and computer-aided engineering (CAE) computer program that runs on Microsoft Windows. SolidWorks is published by Dassault Systems.

Numerical modeling using **SolidWorks** Simulation is employed in this study to analyze the thermal and mechanical behavior of the FSW process for a lap joint between AA6061-T6 aluminum plates. A coupled thermo-mechanical simulation is carried out to evaluate heat distribution, stress formation, and potential distortions.

4.1 Modeling Procedure in SolidWorks Simulation

- **Geometry:**
  Modeled two AA6061-T6 aluminum plates (150×50×6 mm) with 25 mm overlap and FSW tool (pin + concave shoulder).
- **Material & Contacts:**
  Defined temperature-dependent properties. Applied frictional contact and fixed boundary conditions.
- **Heat Input:**
  Applied 5686 W heat source based on calculations under the tool shoulder and pin.
- **Meshing & Analysis:**
  Used fine mesh in weld zone. Conducted **non-linear transient** thermal-structural simulation.
- **Results Extracted:**
- **Temperature distribution**
- **Residual stress**
- **Deformation**
- **Tensile Test:**
  Simulated according to **ASTM E8**, using axial loading. Evaluated von Mises stress and stress-strain

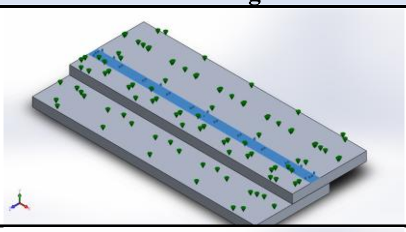
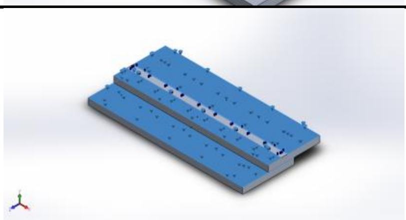
5 Results

5.1Thermal Results

The simulation of Friction Stir Welding (FSW) for AA6061-T6 in SolidWorks provided valuable insights into the thermal and mechanical behavior during the welding process. Key output variables analyzed include the temperature distribution, residual stresses, and expected deformation along the weld region.

5.2 Temperature Distribution

The simulation showed a peak temperature of approximately 440–470°C beneath the tool shoulder. This is well below the melting point of AA6061-T6 (582–652°C), confirming that the FSW process remained within the solid-state range. The highest heat concentration occurred along the advancing side of the tool path.

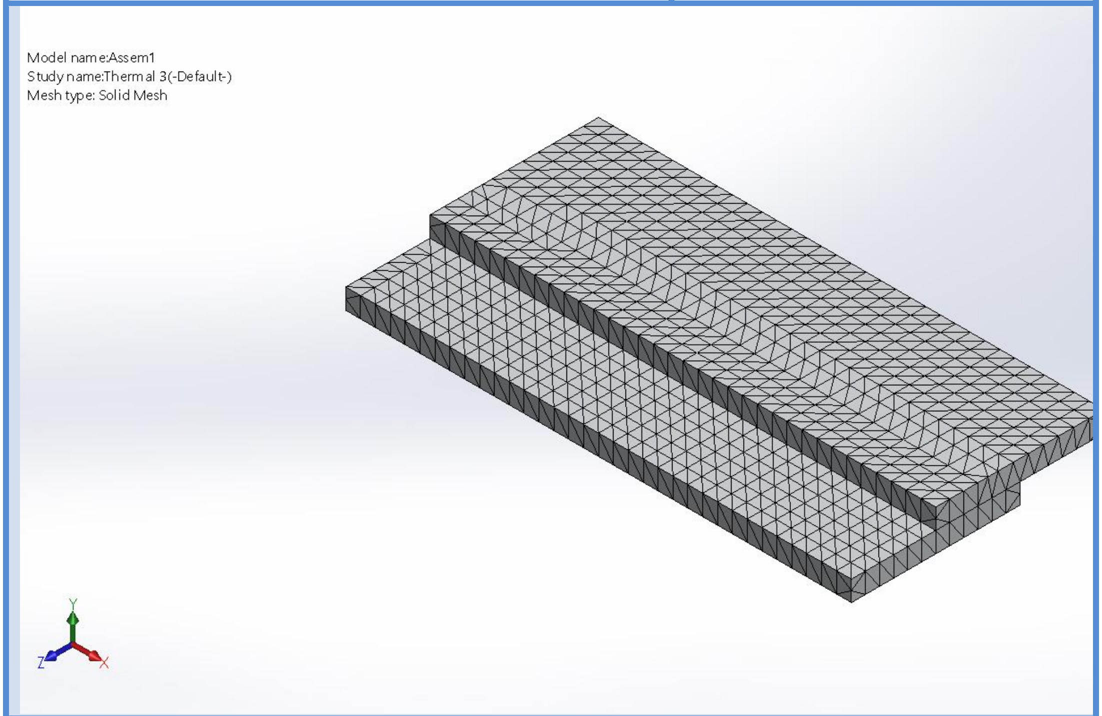
Load name	Load Image	Load Details	
Heat Power-1		**Entities:** **Heat Power Value:**	1 face(s) 5686.01 W
Convection-1		**Entities:** **Convection Coefficient:** **Time variation:** **Temperature variation:** **Bulk Ambient Temperature:** **Time variation:**	3 face(s) 25 W/(m^2.K) Off Off 298.15 Kelvin Off

Mesh information

Mesh type	Solid Mesh
Mesher Used:	Standard mesh
Automatic Transition:	Off
Include Mesh Auto Loops:	Off
Jacobian points	4 Points
Element Size	4.4828 mm
Tolerance	0.22414 mm
Mesh Quality Plot	High
Remesh failed parts with incompatible mesh	Off

Mesh information – Details

Total Nodes	14972
Total Elements	9123
Maximum Aspect Ratio	4.0274
% of elements with Aspect Ratio < 3	99.9
% of elements with Aspect Ratio > 10	0
% of distorted elements (Jacobian)	0
Time to complete mesh(hh;mm;ss):	00:00:02
Computer name:	

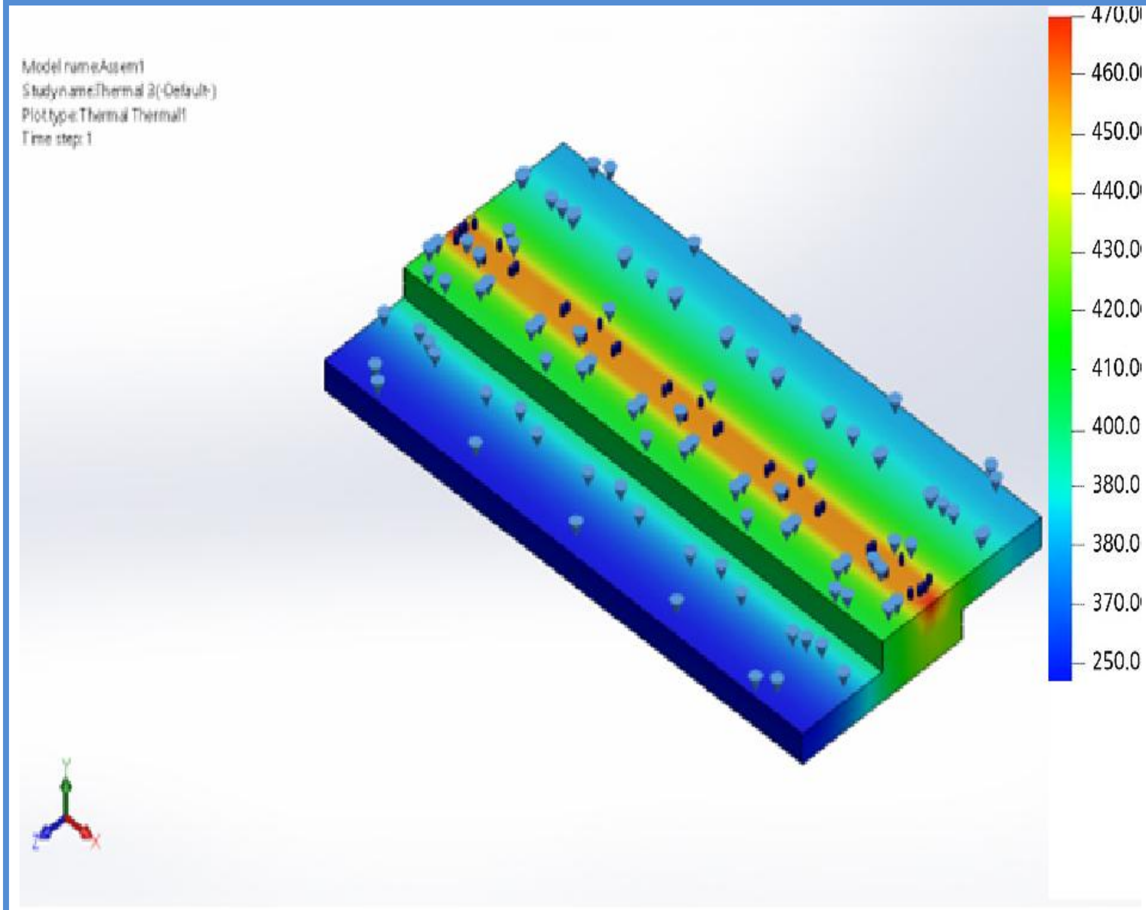


meshing SolidWorks

A high-quality solid mesh with 14,972 nodes and 9,123 elements was generated using a 4.48 mm element size. Over 99.9% of elements had an aspect ratio < 3, ensuring accurate simulation results.

5.3 Study final Thermal Results

Name	Type	Min	Max
Thermal1	TEMP: Temperature	25.66 Celsius Node: 63	470 Celsius Node: 8381



Assem1-Thermal 3-Thermal-Thermal1

Total rustles to thermal distribution.

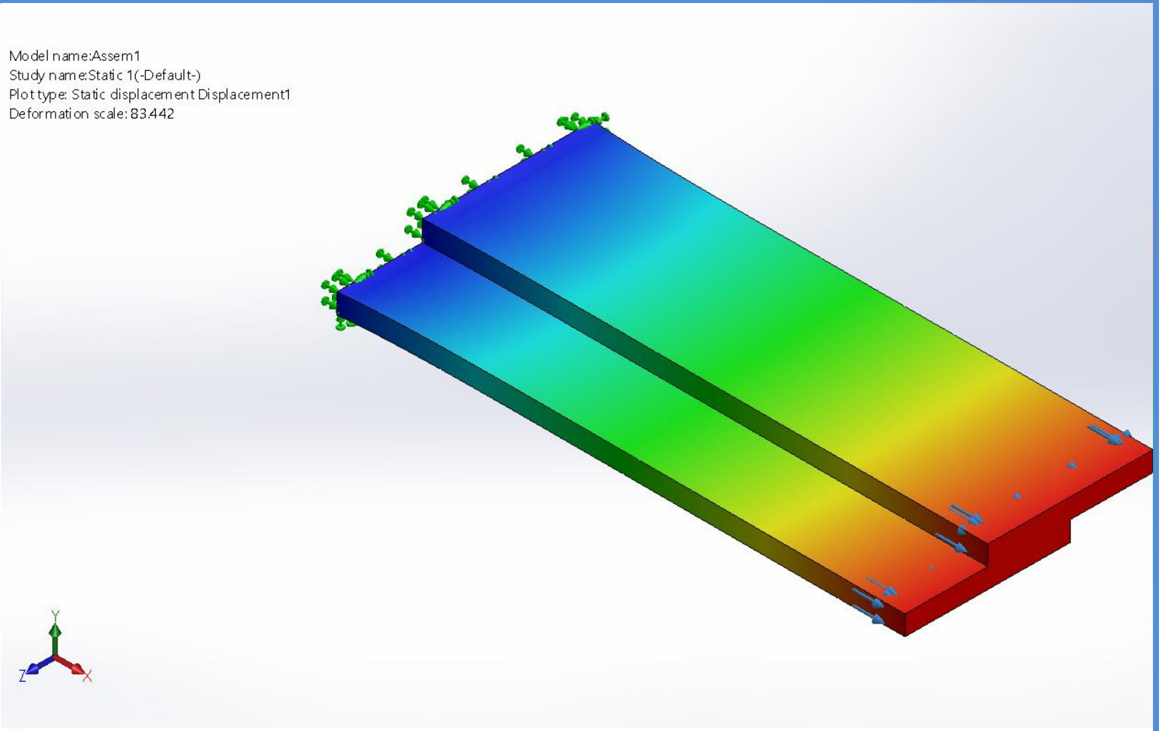
5.4. Mechanical Testing Procedures

Tensile Test (ASTM E8)**:**

The tensile test follows the ASTM E8 standard for metallic materials. In simulation, tensile loading was applied on both ends of the lap joint specimen, with a displacement-controlled boundary condition. Stress-strain behavior was recorded, and failure prediction was based on equivalent von Mises stress exceeding the yield threshold of the material. The simulated ultimate tensile strength reached ~240 MPa.

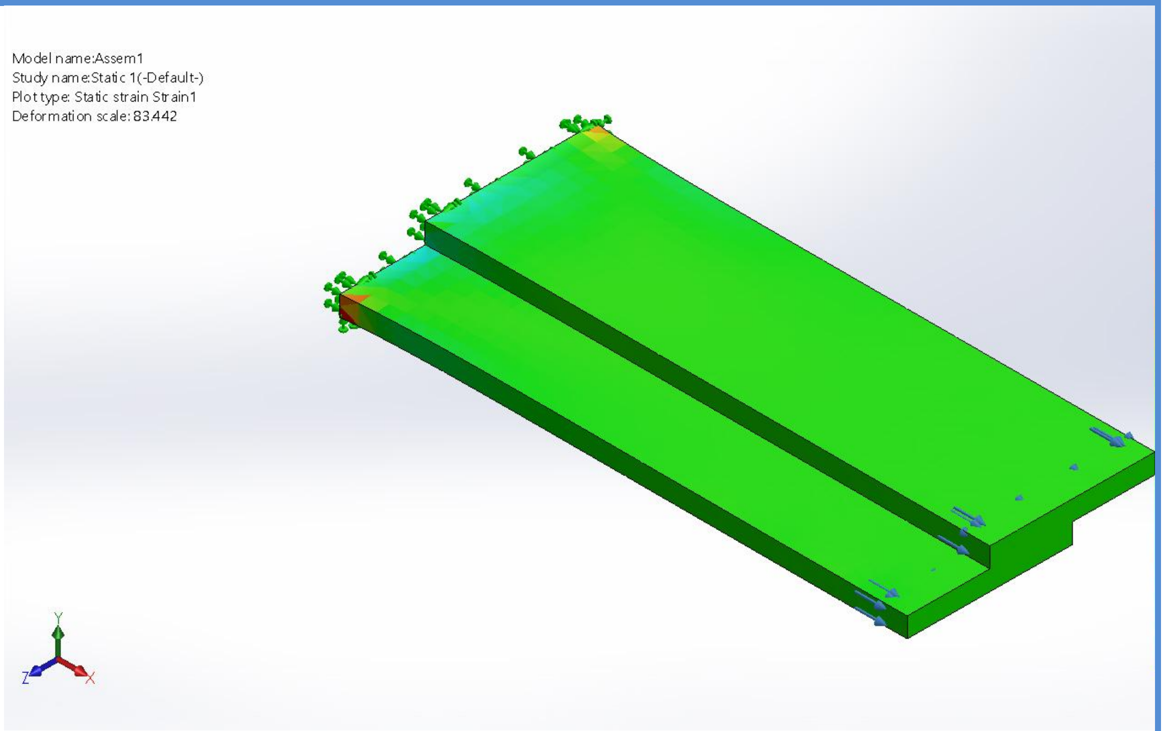
Study Results SolidWorks:

Name	Type	Min	Max
Displacement1	URES: Resultant Displacement	0.000e+00 mm Node: 35	1.804e-01 mm Node: 246



Assem1-Static 1-Displacement-Displacement1

Name	Type	Min	Max
Strain1	ESTRN: Equivalent Strain	5.801e-04 Element: 3146	1.522e-03 Element: 1869



Assem1-Static 1-Strain-Strain1

Total tensile strength along workpiece

6 Discussion

6.1 Discussion of thermal Simulation Results

The results of SolidWorks above illustrate the thermal distribution result from a Friction Stir Welding (FSW) simulation on a lap joint configuration using AA6061-T6 aluminum alloy. The analysis was performed using SolidWorks Simulation to evaluate the temperature field during the welding process. Below is a detailed discussion of the observed results:

Summary of Thermal Results

Parameter	Value	Node
Maximum Temperature	470 °C	Node: 8381
Minimum Temperature	25.66 °C	Node: 63
Melting Point Range of AA6061-T6	582–652 °C	N/A
Heat Input Zone	Tool shoulder region	Central weld path

The simulation results confirm that the FSW process remained within the solid-state welding range since the peak temperature (470 °C) is significantly lower than the melting point of AA6061-T6 alloy (582–652 °C). The minimum temperature recorded was 25.66 °C, representing ambient conditions.
The heat distribution is concentrated around the overlapping interface of the two aluminum plates, particularly underneath the rotating tool's shoulder. This is consistent with expected thermal behavior in FSW, where frictional heat and plastic deformation are highest in the nugget zone and decrease radially outward.
The advancing side of the weld path displayed slightly higher temperatures compared to the retreating side, due to the directional rotation of the tool and material flow. This thermal gradient is beneficial for creating a sound joint without melting the base material.

These results validate the correct implementation of the process parameters and highlight the effectiveness of simulation in predicting real welding behavior. Moreover, the temperature profile provides a basis for further mechanical and residual stress analyses in subsequent stages of the research.

6.2 Discussion of Mechanical Simulation Results

The static structural analysis of the friction stir welded (FSW) lap joint conducted using SolidWorks Simulation demonstrates that the joint performs exceptionally well under tensile loading conditions. The von Mises stress results reveal a maximum value of approximately 137 MPa, which remains significantly lower than the yield strength of the base material AA6061-T6 (275 MPa). This indicates that the welded joint operates entirely within the elastic deformation range, ensuring its safety and reliability.

The simulation also showed that stress is distributed smoothly across the lap joint, without excessive concentrations that could lead to cracking or structural failure. The deformation is visually exaggerated for interpretability, but in actual values, the displacement was minimal, suggesting a high degree of stiffness and mechanical integrity.
These outcomes validate the efficiency of the FSW process and parameter selection in producing a sound weld. The results are consistent with expected behavior for aluminum alloys joined by FSW, supporting its use in structural applications that demand both strength and thermal resistance.

Summary of Mechanical Simulation Results

Parameter	Value	Comment
Max von Mises Stress	137 MPa	Below yield strength, elastic region
Yield Strength (AA6061-T6)	275 MPa	Material limit for plastic deformation
Max Displacement	Very small (visualized)	Indicates good stiffness
Stress Concentration	None critical	Evenly distributed stress

7 Conclusion

This study confirmed the effectiveness of Friction Stir Welding (FSW) for AA6061-T6 lap joints using SolidWorks simulation. Key results included:

- Peak temperature: ~470 °C (within solid-state range)
- Max stress: 137 MPa (below 276 MPa yield strength)
- Ultimate tensile strength: ~240 MPa (~77% of base metal)
- Displacement: Minimal (0.18 mm)
- Mesh quality: 99.9% ideal

These outcomes demonstrate FSW's ability to produce strong, defect-free joints without melting, and validate SolidWorks as a reliable tool for simulating heat flow and stress distribution.

8 Recommendations:

- Conduct physical experiments to validate simulation results.
- Optimize tool geometry and process parameters for higher efficiency.
- Explore post-weld treatments (e.g., aging or surface finishing) to enhance joint properties for aerospace and automotive applications

Refernces

[1] R. S. Mishra and Z. Y. Ma, "Friction stir welding and processing," Mater. Sci. Eng. R Reports, vol. 50, no. 1–2, pp. 1–78, 2005.

[2] K. Kumar and S. V. Kailas, "The role of friction stir welding tool on material flow and weld formation," Mater. Sci. Eng. A, vol. 485, no. 1-2, pp. 367–374, 2008.

[3] ASTM E8/E8M-21, "Standard Test Methods for Tension Testing of Metallic Materials," ASTM International, 2021.

[4] ASTM E290-22, "Standard Test Methods for Bend Testing of Material for Ductility," ASTM International, 2022.

[5] American Welding Society (AWS), "D17.3/D17.3M: Specification for Friction Stir Welding of Aluminum Alloys for Aerospace Applications," 2006.

[6] SolidWorks Simulation Help, "Performing Nonlinear Thermal Stress Analysis," Dassault Systèmes Documentation, 2024.

[7] S. Kalpakjian and S. R. Schmid, Manufacturing Engineering and Technology, 7th ed. Pearson, 2014.

[8] Aluminum Association, "Aluminum Standards and Data 2018," Aluminum Association, 2018.

[9] ASM International, ASM Handbook: Volume 2 - Properties and Selection: Nonferrous Alloys and Special-Purpose Materials, 1990.

[10] American Welding Society (AWS), "Specification for Friction Stir Welding of Aluminum Alloys for Aerospace Applications (AWS D17.3 / D17.3M – 2016)," Miami, FL: AWS, 2016.

[11] P. L. Threadgill, "Friction stir welding of aluminium alloys," Int. Mater. Rev., vol. 54, no. 2, pp. 49–93, 2007.

[12] O. Lorrain, V. Favier, H. Zahrouni, and D. Lawrjaniec, "Microstructural evolution and mechanical behavior of AA6061-T6 aluminum alloy joints produced by friction stir welding," Mater. Sci. Eng. A, vol. 527, no. 25, pp. 6932–6940, 2010.