Jurnal Teknologi dan Riset Terapan
Volume 7, Nomor 1 (Juni 2025), ISSN: 2685-4910
http://jurnal.polibatam.ac.id/index.php/JATRA

JATRA
Jurnal Teknologi & Riset Terapan
ISSN : 2685-4910

DESIGN AND BUILD PLATFORM BRACKET ON MANIPULATOR WELDING MACHINE

Benny Haddli Irawan[1]*, Danang Cahyagi[2], Abdullah Alwi[1], Rahman Hakim[1]

[1] Program Studi Teknik Mesin, Politeknik Negeri Batam
[2] Program Studi Teknologi Rekayasa Konstruksi Perkapalan, Politeknik Negeri Batam

*Corresponding author: benny@polibatam.ac.id

Article history

Received:
17-11-2024
Accepted:
20-06-2025
Published:
30-06-2025

Abstrak

Welding manipulator adalah jenis alat pengelasan yang digunakan untuk pengelasan pada plat pipa berukuran besar dan panjang pada bagian dalam dan luar dalam bentuk vertikal dan horizontal. Terdapat masalah ketika peletakan *platform* jauh dari *nozzle* selama proses *assembling* yang menyebabkan masalah bagi operator *weld manipulator*. Oleh karena itu *bracket* menawarkan solusi untuk masalah dan risiko yang muncul selama proses pemotongan sehingga memungkinkan proses pemotongan dilakukan dengan lebih aman dan efisien. Tujuan penelitian adalah untuk mendesain dan mengetahui hasil analisis keamanan *bracket* alat pengelasan *weld manipulator* bagian *platform*. Untuk menentukan nilai simulasi faktor keamanan dan distribusi tegangan, penelitian menggunakan *Finite Element Analysis* (FEA). Hasil penelitian menunjukkan bahwa desain *bracket* memenuhi syarat dari segi kekuatan dengan nilai *stress* maksimum 98.327.504,00 N/m^2; nilai maksimum *displacement* 1,094 mm dan nilai maksimum *strain* 0,00022, dan faktor keamanan 2,1 sehingga dapat disimpulkan kondisi berada di atas standar.

Kata Kunci: ***bracket, nozzle, stick wire, platform, weld manipulator***

Abstract

A welding manipulator is a specialist apparatus engineered for welding extensive and elongated pipe plates on both interior and exterior surfaces, as well as in vertical and horizontal positions. During the assembly process, a challenge occurs when the platform is excessively distanced from the nozzle, complicating the operator's task with the welding manipulator. Brackets are an effective option to mitigate hazards and address issues throughout the cutting process. The aim of this research is to design and evaluate the safety of the bracket utilized for the platform component of the welding manipulator. The research utilizes Finite Element Analysis (FEA) to assess the simulated values of the safety factor and stress distribution. The findings demonstrate that the bracket design satisfies strength criteria, producing a maximum stress value of 98,327,504.00 N/m². The recorded maximum displacement is 1.094 mm, and the maximum strain is 0.00022. The computed safety factor is 2.1, indicating that the design parameters surpass established safety criteria.

***Keywords*: bracket, nozzle, stick wire, platform, weld manipulator**

1.0 INTRODUCTION

A weld manipulator is a device utilized in the welding process to assist operators in controlling the position and movement of the welding apparatus. This apparatus may be mechanical or robotic, engineered to secure, orient, and maneuver the welding instrument by particular welding requirements. Weld manipulators improve accuracy and productivity in the welding procedure. Assembly work is essential in the fabrication of weld manipulators, as the components cannot be created in a single operation. Assembly entails integrating two or more components into a product designed for a particular function, including motor vehicles, airplanes, pumps, and combustion engines. The components can be interconnected by a system interface comprising assembly features such as holes, pins, and snap-fit mechanisms. A crucial assembly procedure for manufacturing weld manipulators is platform installation. In recent years, the swift progression of the Fourth Industrial Revolution and breakthroughs in information technology have redirected the emphasis of numerous industries towards intelligent systems, which possess the capacity to improve efficiency in building. This has resulted in the birth of a concept referred to as the Platform in the construction sector. The Platform is an essential element of the weld manipulator, serving as a foundation for the operator during machine operation. During operation, the operator utilizes a monitor positioned before the Platform to assess welding outcomes and sever the obstructed wire in the nozzle. The utilization of brackets is highly advantageous for the operator, as they assist in positioning the operator nearer to the nozzle, thus enhancing the work process [1][2].

Use the SolidWorks application to work on the initial bracket design in bracket design. SolidWorks is an engineering design software developed by Dassault Systemes, especially for 3D model design. Usually used to design 3D models, SolidWorks has three main views of parts to create model drawings, assemblies to unite the designed models into the desired construction, and drawings to create visual representations of part or assembly models that can be printed and submitted to industry[3]. In addition to design, there is testing; testing here only tests the load capacity loaded on the platform. The research method uses finite element analysis (FEA) to determine the simulation value of the safety factor and stress distribution. Testing uses SolidWorks simulation software to see the results obtained after being given a load. Here, the load in question is the human or operator operating the weld manipulator machine. SolidWorks Simulation is a component of SolidWorks software that is used to perform stress analysis on the design that has been made. The existence of this simulation is beneficial in reducing errors when designing. The level of design accuracy is also influenced by factors such as the material of the object, restraint (the part that is still on the part), and the load given. Through simulation expression, the step-by-step process will show how the design will behave under specific conditions[4]. Figure 1 is the modeling of a weld manipulator without a bracket, and Figure 2 is the modeling of a weld manipulator with a bracket as the part that the author will discuss:

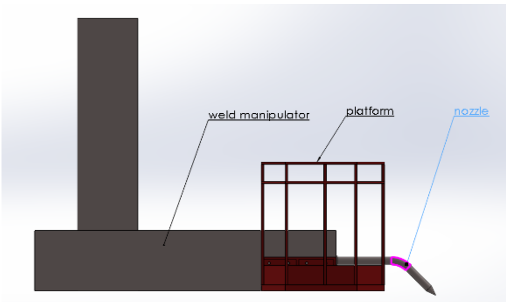


Figure 1: Front 3D Weld Manipulator View Without Bracket In SolidWorks

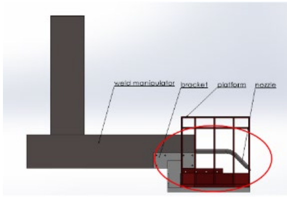


Figure 2: Front 3D Weld Manipulator View With Bracket In SolidWorks

The weld manipulator machine only has a platform that is not aligned with the nozzle, which makes it difficult for the operator to cut the stuck wire. This scientific paper aims to design a bracket tool on the platform for the stick wire cutting process and analyze it with simulations on the SolidWorks application to determine the results of stress, strain, shape changes, and safety factors. Overall, a bracket is a structure made of metal, aluminum, plastic, or wood that acts as a support or construction support to prevent bending or deformation of the structure [5]. The problem expected to be resolved after writing this final assignment is that the bracket tool for the platform is safe to use for the cutting process.

Moreover, the bracket presents drawbacks during removal and installation due to its construction from a substantial iron plate, resulting in increased weight. The removal or installation necessitates the assistance of an overhead crane and a forklift. Furthermore, the bracket addressed in this scientific document pertains solely to the automatic welding boom. In contrast, the column manipulator weld platform for booms and columns encompasses several structures, including stationary, rotating, and mobile versions. This platform can be categorized into light-duty, heavy-duty, and operator seat varieties based on its front boom lifting capability, telescoping features, and specifications. The platform configuration can be chosen based on user requirements and outfitted with diverse welding machines. Supplementary functionalities like seam tracking, oscillation, video monitoring, and flux recovery have been incorporated [6]. Figure 3 illustrates an example of a Weld Manipulator apparatus:


Figure 3: Weld Manipulator

2.0 RESEARCH METHODOLOGY

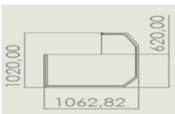
1. Literature Study

The weld manipulator machine, a crucial tool in the welding process, is composed of several parts, one of which is the platform. This platform serves as a dedicated space for employees or welding operators to carry out their tasks, including cutting stick wire. It's important to note that only one employee or operator can use the platform at a time, underscoring its significance in the welding process.

2. Data Collection

Data collection is a vital aspect of conducting research in welding technology and design. It involves gathering various data points, such as test loads that include the operator's weight and hand tools for cutting stick wire, providing materials used to make brackets, the distance between the platform and the nozzle, and the size of the base platform for adjusting the bracket design. The results of this data collection are then tabulated for further analysis and interpretation.

Table 1: Collection of Data

No.	Data Collection	Information
1	Material	Plate EH36
2	Platform Test Load	100 kg
3	Platform and Nozzle Distance	470 mm
4	Platform Base Dimension	

3. Bracket Design

The bracket design uses the SolidWorks application concerning previously collected data; the material used is an EH36 plate. Summary EH36 plate is a plate with excellent toughness and corrosion resistance. The chemical composition content for EH36 is 0.18% carbon, 1.60% manganese, 0.5% silicon, 0.035% phosphorus, and 0.35% copper[7].

4. Testing Method

At this stage, test the results of the bracket design using the SolidWorks application. With EH36 material, which has a yield strength of 350,000,000 N / m2, which is meant by testing here, the bracket is tested when given a load to test its feasibility when used. The load here is the human load or operator, the tool for cutting the weld manipulator machine. The load is 100 kg converted to newtons to 981 N, according to the data needed at PT. The research method uses finite element analysis (FEA) to determine the simulation value of the safety factor and stress distribution.

5. Simulation Results

There are four simulation results here, namely:

a) Stress or tension
b) Displacement or change in shape
c) Strain or strain
d) Fos (factor of safety) or safety factor

6. Handover To Maintenance

Data and drawing submission to the maintenance department for fabrication, finishing, and trials carried out directly in the workshop or field.

3.0 RESULTS AND DISCUSSION

Based on the research results, a support tool was produced in the form of a bracket for the platform installed on the boom part of the manipulator weld machine. The boom is a welding arm or welding extension arm that reaches a place away from the workpiece, helping the welders achieve complex welding easily. This bracket makes the platform for the operator closer and aligned with the nozzle, which makes the welding stick-cutting process safe.

Here are the steps to make a bracket:

1. Find the data on the size of the distance between the nozzle and the platform that has been measured in the field, then look for the data on the size of the base platform and adjust it to the data that we have measured in the field as shown in figure 4.

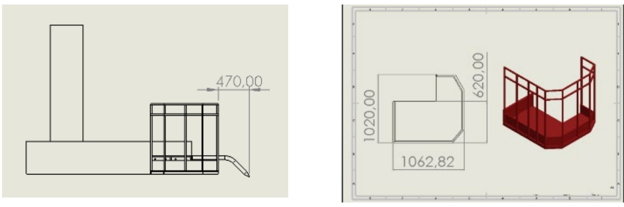


Figure 4: Platform distance to nozzle and platform base size

2. Making a 3D design modeling, the creation of modeling and drawing as shown in figure 6 is continued to testing or simulation using the features in the SolidWorks software, namely SolidWorks Simulation.

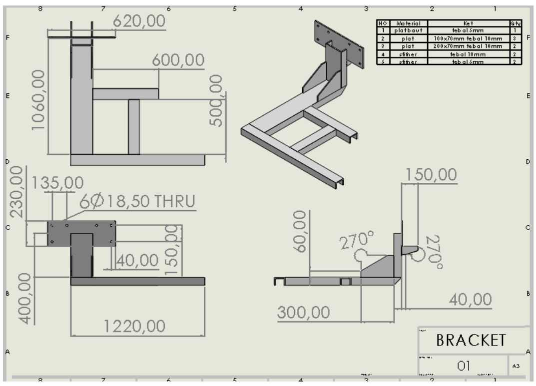


Figure 5: 3D Desing Modeling

Before the simulation of the first assembly between the bracket and the platform, the following process is carried out:

1) Provide materials
2) Give the connection, i.e., 20 mm bolt
3) Give fixed geometry
4) Give a load point of 100 kg, we convert to newtons, which is 981 N
5) Mesh all parts so that they can be simulated

6) From the simulation results, we know that brackets and platforms have a stress value (voltage)
7) From the simulation results, we know that the bracket and platform have a displacement value
8) From the simulation results, we know that the bracket and platform have strain values
9) From the simulation results, we know that the bracket and platform have a FOS (Factor of Safety) value

In Figure 6, it can be seen that the application of connecting elements, namely bolts on six holes marked with blue writing and the application of loads marked with purple arrows, then the application of fixed geometry is marked with brown arrows

Figure 6: Provision of bolt connections, fix geometry, and also loads

Based on the simulation results in Figure 7, the stress value on the red part of the bracket received tremendous pressure with a maximum value of 98,327,504.00 N/m2. The blue part describes that there is no effect on the specified pressure. The yield strength value of stress is when the material loses the ability to return to its original shape or elastic properties [8]. From the simulation results, the yield strength value obtained is 350,000,000 N/m2, indicating that the material is safe for use as it can still accept pressure.

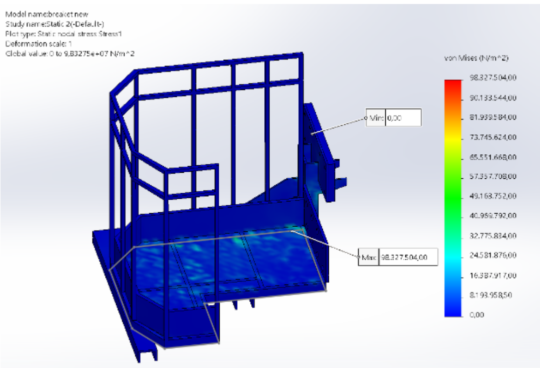


Figure 7: Stress Simulation

In Figure 8, a displacement simulation is carried out. The red part of the bracket is a platform that has undergone a deformation with a maximum value of 1,094 mm. In Figure 9, from the results of the strain simulation, the maximum strain was obtained, which was 0.00022

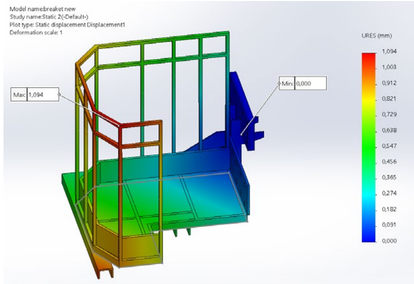


Figure 8: Displacement Simulation

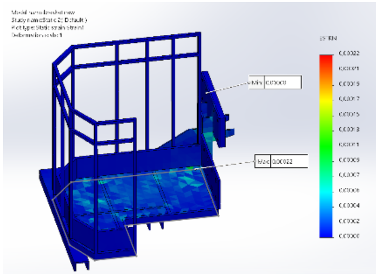


Figure 9: Strain Simulation

The safety factor is an indicator used to ascertain whether or not a construction design is safe enough to withstand the load placed on it [9]. According to Dobrovolsky's book "Machine Element," the recommended range of safety factors for dynamic loads is 2.0 to 3.0 [10]. In Figure 10, the results of the factor of safety (FoS) simulation are presented, and a safety value of around 2.1 is obtained. Because the FoS result is more than 2, the object is safe to use. The results of the analysis that the bracket meets the aspects of strength, design and the results of trials in the field have been tested by the maintenance unit that the bracket has been installed in line with the nozzle so that the process of cutting the stick wire can be carried out safely.

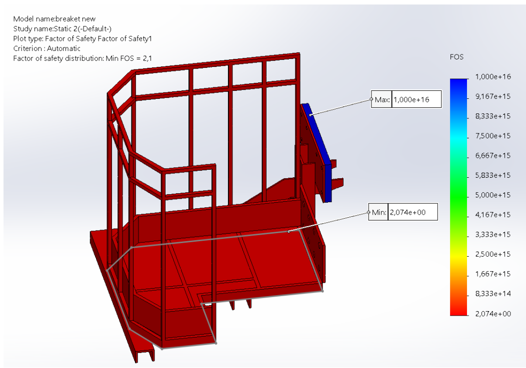


Figure 10: Result Simulation for Safety Factor

4.0 CONCLUSION

The conclusion obtained based on this study is that the design of the bracket aid using EH36 plate material and applying a load of 100 kg produces a stress or stress simulation value with a maximum value of 98,327,504.00 N/m2 because the maximum value is still below the yield strength value of 350,000,000 N/m^2 so that it can be concluded that the bracket can accept pressure, displacement or deformation with a maximum deformation distance value of 1,094 mm which is still small enough that there is no significant change in shape. In terms of the factor of safety, a value of around 2.1 was obtained, so it can be concluded that the bracket is still feasible to use because the safety standard is 2. It can be concluded from the analysis results that the brackets met both the strength and design aspects.

BIBLIOGRAPHY

[1] D. W. P. Syam, “Analisis rantai variasi dalam proses perakitan produk,” *Toleransi Dimensi dan Geometri,* pp. (3),204, 2019.

[2] D. K. S. Maria Kozlovska, “Impact of Industry 4.0 *Platform* on the Formation of Construction 4.0 Concept: A Literature Review,” *Sustainability,* pp. 13(5), 2683, 2021.

[3] A. D. Y. d. M. Djuhana, “Plate Mold dengan Software Simulasi (SolidWorks 3D),” *Piston,* pp. Vol. 3, No. 1,6-16, 2020.

[4] Helmiyansah, “Analisa Tegangan Pada Crankshaft Sepeda Motor Suzuki Smash Menggunakan Software SolidWorks,” *Jurnal Teknik Mesin (JTM),* vol. VOL.5 NO.3, p. 107, 2016.

[5] J. A. P. d. M. N. Misbah, “Studi Pengaruh Ukuran Bracket Pondasi Mesin terhadap teganagan dengan menggunakan finite elemen method,” *jurnal Teknik ITS,* pp. Vol. 11, No. 1, 2022.

[6] Bota, “welding column boom manipulator,” *welding column boom ,* p. 1, 2023.

[7] A. G.S, “Studi Eksperimen Dan Numerik Pengaruh Variasi Urutan Pengelasan Terhadap Distorsi Sudut Pada Struktur Panel Baja A36,” *Teknik Kelautan,* vol. N0 1, p. 12, 2018.

[8] W. L. A. Nendra, “Desain Dan Analisis Tegangan Crane Hook Model Circular Section Kapasitas 5 Ton Menggunakan Autodesk Inventor,” *SIMETRIS,* vol. VOL 10. NO 01, p. 29, 2019.

[9] L. A. N. Wibawa, “Desain Dan Analisis Tegangan Crane Hook Model Circular Section Kapasitas 5 Ton Menggunakan Autodesk Inventor,” *SIMETRIS,* vol. 10. No. 01, p. 30, 2019.

[10] K. R. H. Muhammad Rifqi Firdaus, “Simulasi Faktor Keamanan Dan Pembebanan Frame Pada Turbin Angin Type Darrieus,” *kajian teknik mesin,* vol. vol 8. no 2, p. 4, 2022.