**International Journal of Mechanics, Energy Engineering and Applied Science (IJMEAS)**
VOL. 1, NO. 1, October 2023, PP. 07-11
Print ISSN 3026-6815 | Online ISSN 3026-5673 | DOI prefix: 10.53893
https://journal.gpp.or.id/index.php/ijmeas/index

# Design Simulation of Upper Arm on Welding Table

**Fatahul Arifin[1,*], Azharuddin[1], Dwi Arnoldi[1], Sairul Effendi[1], Yusuf Dewantoro Herlambang[2], Yahya[1], Muhammad Arif Fadilah[1], Akhmad Radjiq[1]**

[1]Department Mechanical Engineering, Politeknik Negeri Sriwijaya, Palembang, Indonesia
[2]Department Mechanical Engineering, Politeknik Negeri Semarang, Semarang, Indonesia

## Email address:

farifinus@authorname.com

[*]Corresponding author

## To cite this article:

Arifin, F. ., Azharuddin, Arnoldi, D., Effendi, S. ., Herlambang, Y. D. ., Yahya, Fadilah, M.A., & Radjiq, A.. (2023). Design Simulation of Upper Arm on Welding Table. *International Journal of Mechanics, Energy Engineering and Applied Science (IJMEAS)*, 1(1), 7–11. *https://doi.org/10.53893/ijmeas.v1i1.215*

**Received**: 09 04, 2023; **Accepted**: 10 06, 2023; **Published**: 10 10, 2023

**Abstract:** Welding process is the joining of two metals by heating. There are two types of welding that are often applied in the field, namely gas welding (OAW) and electric welding (SMAW). Both of them have their advantages as metal connectors. This research originated from a problem that occurred in the upper arm on the welding table which experienced bending as a result of carrying the load when holding the welding disc. Therefore, CAD/CAE design is carried out. In this case, the strength and tension that occur in the upper arm are analyzed. This research aims to reveal the value of the stress that occurs in the upper arm and the safety value. Where in the design the planned force capacity is 300 newtons using the finite element method or also known as Finite Element Analysis (FEA). And the material used is ASTM A36. The results of the simulation and also the analysis that has been carried out show that the upper arm frame structure on the welding table has a stress, deformation, and safety factor of 311.987 MPa, 2.28286 mm, and 0.663488 ul.

**Keywords:** Welding, Table, Frame, Stress

## 1. Introduction

The welding machine is a machine that is often used and is very necessary in the scope of machining and mechanical processes. When operating a welding machine, you really need a welding table as a place to place the object to be welded. Electric welding machines need tools that can support someone who wants to weld so that with the operator's tools it is easier to carry out welding. Based on observations made in the welding workshop, several conditions were obtained which showed that the current electric welding machines were not equipped with auxiliary equipment. So, when carrying out welding, operators experience difficulties, especially for someone who has never welded before, so they need welding tools in the form of a welding table. This welding table has 4 supporting legs and angle iron to make the table stronger to withstand the load. This welding table aid must follow the welder in the field, namely with several welding variations because in welding there are several welding positions that must be present, several variations of welding positions include positions: 1G, 2G, 3G, and 4G (1). So, by having this position on the welding table welding tool, it is hoped that beginners who will learn to weld will also be able to master it, not just 1 position, because in the field the operator will encounter several difficult welding position conditions and must be able to weld in that position. With this welding aid, it is hoped that the quality of the welding results will be better because this welding table is equipped with a variety of positions that may be found when welding in the field and is also very helpful for someone who wants to learn with this aid. Apart from that, there is a need for a welding table that will be used for the welding teaching and learning process in the laboratory of the Mechanical Engineering Department of the Sriwijaya State Polytechnic. The calculations carried out in

this research use a numerical method known as the finite element method or Finite Element Analysis (FEA) or can also be called the finite element method. The finite element method can also be said to be a numerical procedure for solving continuum mechanics problems with acceptable accuracy. accepted by engineers [1]. The finite element method is a typical procedure for obtaining estimates for boundary value problems. This strategy has been used with great success in dealing with various problems in almost all areas of design and numerical materials science. This method is used to find the value of deflection, stress, and safety factor that can be withstood by the upper arm structure on the welding table frame, as well as for the strength of the material used as the frame for the upper arm of the welding table, to obtain a safe, strong, frame combination. and able to withstand the load or force given to the welding table frame [2]. The material for making the frame structure of this welding table is ASTM A36 Profile L. ASTM A36 is low carbon steel which has good strength and also has the added properties of steel which can be changed using a machine and also carried out by welding. ASTM A36 steel plate can also be galvanized or coated to provide resistance to corrosion. The ASTM A36 material is low carbon steel which has good strength and also has the added property of being able to be deformed using a machine. This research focuses on the strength and safety of the Upper Arm on the upper welding table made from ASTM A36 Profil L Material. Calculations carried out in this research use the Finite Element Analysis (FEA) Method to find the values of deflection, stress and safety factors on the top of the frame up to the legs on the welding table, so that the strength of the upper arm on the welding table can be determined to withstand the pressure from the workpiece. This research focuses on analyzing the compressive strength of the upper arm of the welding table. This research was taken because of problems that occurred in the welding process which required a place to place the object to be welded, vertically. This welding table aid must follow the competence of the welder in the field, namely with several variations of welding because in welding there are several welding positions that must be present, several variations of welding positions include positions: 1G, 2G, 3G, and 4G. So by having this position on the welding table welding tool, it is hoped that beginners who will learn to weld will also be able to master it, not just 1 position, because in the field the operator will encounter several difficult welding position conditions and must be able to weld in that position.

# 2. Methods

### *2.1. Material*

The material chosen to conduct this research and design the frame structure of the welding machine table is ASTM A36 material. Because, the ASTM A36 is one of the important compounds of iron that has many modern applications. However, mild steel is susceptible to corrosion. The design and 3D drawing of the welding table are shown in Figure 1.

***Figure 1****. Design of 3D Welding Table*

### *2.2. Software*

The use of software to reduce errors and also speed up the calculation time of a work process or optimization has been commonly used by researchers such as designs wind turbines by changing the blade shape and lifting angle on horizontal type wind turbines, used of moldex software for the micro gear production process, and then using CAD software to design the fire fighter kit and water wheel with a floating system [3]-[5]. The design of optimize Savonius type wind turbines obtained optimal performance using CFD analysis [6]. In this research, Autodesk Inventor software was used to analyze the stress in the welding table arm structure. In this case we use a method in the form of finite element analysis. The finite element method is a method commonly used by structural analysis software. Finite element analysis is an estimation procedure that uses a numerical mathematical framework in calculating the strength and performance of a designed part structure by dividing the study into cross-sectional shapes. (mesh) [7].

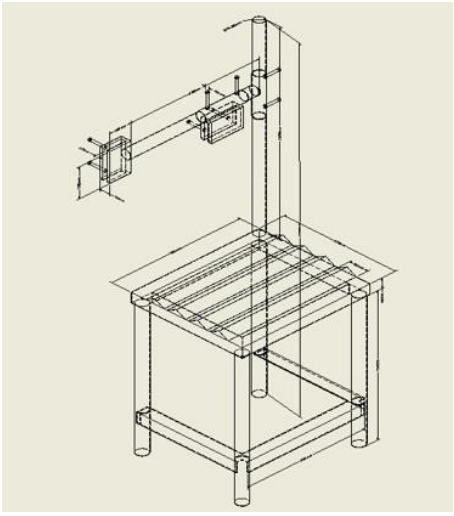

***Figure 2.*** *Welding Table Size (in mm)*

The procedure of running this simulation is obtained stress values through Autodesk Inventor Professional 2023 software analysis is as follows: First, the welding table frame structure was designed using CAD [8]. The design drawing is expressed in the dimensions and shape of the welding table frame structure. The detailed dimensions of the welding table frame are depicted in Figure 2.

The second stage is determining and looking for the type of material used. The material used is Steel Mild, which is a low carbon steel that complies with ASTM (American Society for Testing Materials) A36 standards. After that, proceed with creating constraints [9]-[11]. This limit is used in the fixed constraint at the top of the welding table and the beginning of the top arm. It can be seen in the Figure 3.

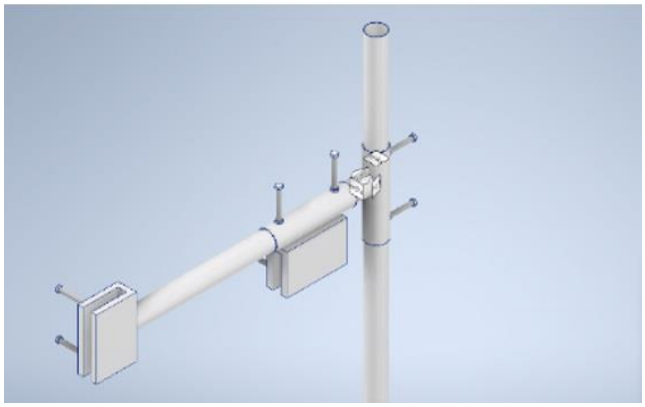

***Figure 3.*** *Position of Fixed Constraint Welding Table*

The next stage is to determine the amount of load or pressure. The amount of load given to the frame structure is 300 newtons (30kg). The location of the load is on the object gripper which is at the end and base of the upper arm pointing downwards (Figure 4)

***Figure 4****. Location of the Welding Table Arm Load*

The next procedure is meshing. Meshing is the process of dividing the shape of an object into several components, (Mesh) The number of cells is an area that has very sharp changes, generally cross-sectional measurements are done smoothly, while for different areas it is done a little rougher. [12], at this stage the components are divided into 856284 elements and 1853836 nodes, the details are shown in Figure 5.

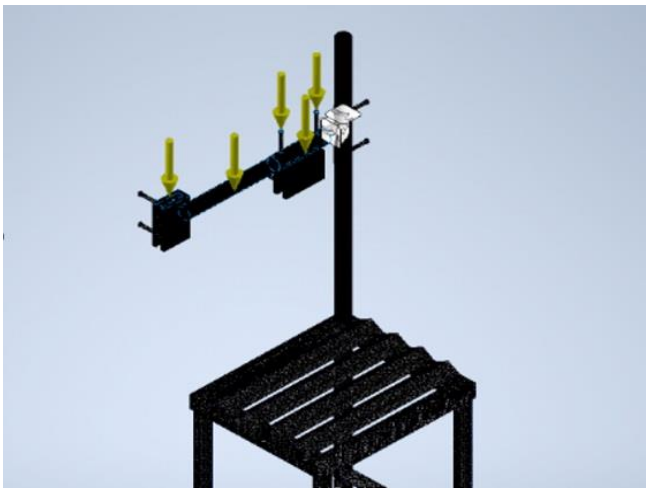

***Figure 5.*** *Meshing process of the upper arm of the welding table*

The next process is, running simulation or running a simulation program. The simulation program that is run will produce results in the form of mass, Von Mises stress, safety factor, and deformation (displacement). The simulation process will reveal parts that experience stress from the frame structure design that has been created. We can see the quantity values used as parameters in analyzing the stresses that occur in the frame using Autodesk Inventor Professional 2023 in detail from Table 1 below.

***Table 1****. Strength Value Parameters*

| **Parameters** | **Keterangan** |
|---|---|
| Simulation Type | Single Point |
| Load Capacity | 300 N (30 kg) |
| Gravity | 9,81 m/s$^2$ |
| Total Load | 300 N |
| *Average element size* | 0,1 mm |
| *Minimun element size* | 0,2 mm |
| *Safety factor* | Base on yield strength |
| Node Numbers | 1853836 |
| Elemen Numbers | 856284 |

# 3. Result and Discussions

## *3.1. Analysis Material Properties*

Analysis of material properties is very necessary to obtain values that will be included in structural analysis. The physical properties of the material stated in table 2 are the physical properties of the Steel Mild material. Mild Steel material has a density of 7.85 grams/cm3. This value affects the total mass of the welding table frame structure weighing 72.0165 kg.

***Table 2**. Physical properties of Steel Mild material*

| ***Material*** | ***Steel Mild*** |
|---|---|
| *Density* | 7,85 g/cm^3 |
| *Mass* | 15,8843 kg |
| *Area* | 3247910 $mm^2$ |
| *Volume* | 2023480 mm^3 |
| *Yield Strength* | 207 MPa |
| *Ultimate Tensile Strength* | 345 MPa |
| *Young's Modulus* | 220 GPa |
| *Poisson's Ratio* | 0.275 ul |
| *Shear Modulus* | 86.2745 GPa |

Three-dimensional loads that apply to an elastic object will result in complex stress, which states that at every point on the object there is pressure acting in various directions. Von Mises stress is an important factor in determining whether a structural design is safe or will fail [12]. The Von Mises barometer provides an indication that a ductile material will yield when the value is critical due to the second invariant value of deviatoric stress [13]. This event is included in the plasticity theory which is considered the best for ductile materials, especially metal materials. This voltage is often referred to as equivalent or equivalent voltage. The Figure 6 shown the simulation results of the upper arm analysis on the welding table against a load of 300 N. The maximum Von Mises stress is 311.987 MPa. The Von Mises stress is above the yield strength of Steel Mild material, which is 207 MPa. Changes in shape or deformation are one of the important markers for deciding whether the material used is extreme enough to withstand the ideal load.

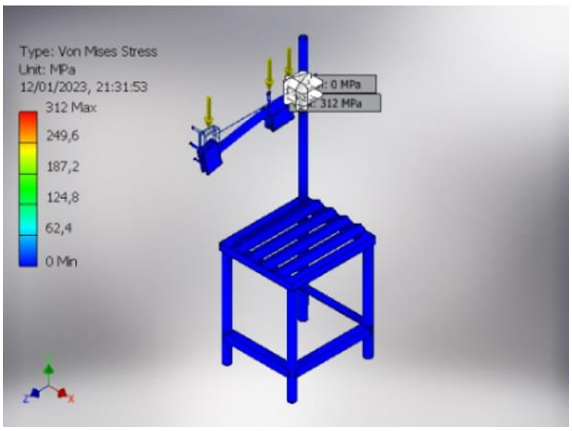


***Figure 6**. The Von Mises stress of the structure of the upper arm of a welding machine with a load of 300 N.*

Deformation occurs as a result of the material receiving force or load. The smaller the deformation value is the stronger the materials. The maximum deformation value in this simulation is relatively large, namely 2.28286 mm (Figure 7). The safety factor is a factor used to assess the safety of the arrangement of machine components, according to Mott's hypothesis, which is subjected to static loads with a level of certainty that cannot be denied. The value of the safety factor is 1.25 to 2.0 (1) (Arifin et al., 2020). Safety factors are factors used to assess so that the plan is guaranteed to be safe with the fewest measurements [14]. The safety factor can be determined either at the most extreme elastic pressure or the yield pressure of the material [15].

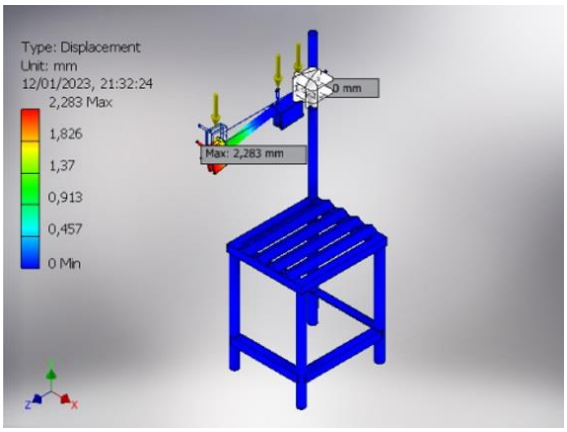


***Figure 7.** Deformation of the upper arm structure of the welding machine with a load of 300 N.*

The yield strength of a material using Inventor software can be calculated as a safety factor derived from the material which has been divided by the maximum Von Mises stress of the material. The general safety factor value in this research does not meet the requirements to be able to withstand static stress. Static stress is a stress that receives a load at a slow speed, without any shock loads and remains at a constant magnitude or value. Any stress produced in this situation is called static stress. So, this structure is declared safe because the minimum safety factor value is 15 (Figure 8). If the minimum value produced by the upper arm structure on the welding table.

This ranges from 1 or below, then the Upper Arm frame structure on this welding table is still unsafe to be used as a frame and needs to be remodeled, because the value of the safety factor given to a material is said to be safe or able to withstand static loads ranging from 1.25 -2 [16]. The high security value of this frame is still very safe, even to withstand shock loads.

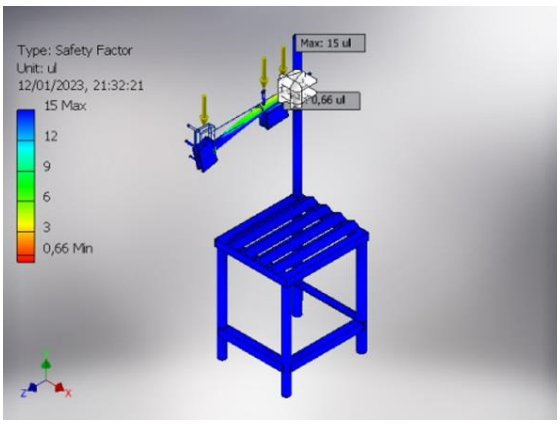


***Figure 8.** Safety factor of the upper arm structure of a welding machine with a load of 300 N*

# 4. Conclusion

The welding table frame structure design used Steel Mild material and has a mass is 72.0165 kg. At a load of 30 kg or 300 N, the frame structure on the upper arm of the welding table gets a maximum stress value in Von Mises of 311.987 MPa, and based on this value the yield strength, this stress value is no longer safe because the value is above the yield strength of the Mild Steel material namely 207 MPa. When applying a load is 30 kg or 300 N, the upper arm frame structure on the welding table receives force from the object load which has a maximum deformation value of 2.28286 mm. At a load of 30 kg or 300 N the welding table frame structure has a minimum safety factor value is 0.663488 ul. This value shown that the design of the upper arm frame structure on the welding table is no longer safe and needs to be remodeled so that it can withstand static and shock loads.

# References


[1] Rober D, C. (2007). Konsep Dan Aplikasi Metode Elemen Hingga.

[2] Bargess, M. F., Lesmana, C., Tallar, R. Y., Sipil, J. T., Maranatha, U. K., Prof, J., Suria, D., & Mph, S.(2000). *Dengan Metode Elemen Hingga*

[3] Arifin, F., Arnoldi, D., Sundari, E., Putri, F., Agassa, F., Susetyo, G., Herlambang, Y. D. 2020, Studi analisis simulasi kekuatan beban pada alat bantu pembuatan lubang dengan sudut kemiringan 45 derajat, jurnal Polimesin, 18 (2) pp., 116-123, http://dx.doi.org/10.30811/jpl.v18i2.1837

[4] Novarika, W., Arifin, F., Satria Martomi, E., Gunawan, I. ., & Perdana, F. R. . (2021). Design Manufacture of Fixture Fire Fighter Kit. *International Journal of Research in Vocational Studies (IJRVOCAS)*, *1*(3), 08–16. https://doi.org/10.53893/ijrvocas.v1i3.58

[5] Wang, M., W., Arifin, A., Huynh, T.T.N., (2019), Optimalization of Molding Parameter for a Micro gear with Taguchi Method, Journal of Physics: Conceferece Series 1167(1), doi:10.1088/1742-6596/1167/1/012001

[6] Herlambang, Y., D., Supriono, Prasetiyo, B., Alfauzi A., S., Marliyati, Purwanto, Arifin, F., Emzain F., Z., 2023, Optimization of Savonius Turbine Towards Different Innner Blade Position to Improve Turbine Performance, Jurnal Polimesin, 20 (1), pp 112-117, ttp://dx.doi.org/10.30811/jpl.v21i1.3248

[7] Ari, L., & Wibawa, N. (2021). Simulasi Kekuatan Komponen Sarana Pengujian Roket Menggunakan Autodesk Inventor Professional 2017.

[8] Seprianto, D., and Arifin, A., (2010), "Dasar-Dasar Solid Modeling Menggunakan Autodesk Inventor Professional 2010", Tunas Gemilang Press, ISBN: 978-602-8816-35-9

[9] Susanto, Dwi Nugroho, Setyaningrum, Ratih (2015) *Analisis dan Perancangan Meja Las dengan Pendekatan Ergonomi dan QFD*, https://publikasiilmiah.ums.ac.id/handle/11617/5741

[10] .Astanto, I., Arifin, F.., Bow, Y., & Sirajuddin. (2022). Study of Effect Changing the Blade Shape and Lift Angles on Horizontal Wind Turbine. *International Journal of Research in Vocational Studies (IJRVOCAS)*, *2*(1), 33–37. https://doi.org/10.53893/ijrvocas.v2i1.92

[11] Arifin, F., Sutanto, H. ., Iskandar, I. ., & Sukwadi, R. . (2022). The Design and Fabrication of Waterwheels with System Floating Pontoon. *International Journal of Research in Vocational Studies (IJRVOCAS)*, *2*(3), 01–06. https://doi.org/10.53893/ijrvocas.v2i3.143

[12] Danurji, Fawaid, Haryadi. (2015) Rancang Bangun Meja Las Untuk Variasi Posisipengelasan.6971-16281-1-SM%20(1).pdf

[13] Komarudin, Towip. (2022) Perancangan Meja Las Yang Ergonomis Berdasarkan Analisis Reba di Universitas Sebelas Maret 60068-184380-1-PB.

[14] Uji, B., Antariksa, P., Penerbangan, L., & Lapan, N. (2018). Pengaruh Kecepatan Landing Vertikal Terhadap Menggunakan Metode Elemen Hingga. 35–42.

[15] Wibawa, lasinta ari nendra., (2017). Simulasi Kekuatan Komponen Sarana Pengujian Roket Menggunakan Autodesk Inventor Provesional

[16] Lee, Seung-Heon, Si-Woo Song, Min-Jae Jeong, Won-Ho Kim, and Dong-Hoon Jung. (2023). "A Novel Sleeve Design to Reduce the Eddy Current Loss of High-Speed Electrical Machines" *Machines* 11, no. 7: 756. https://doi.org/10.3390/machines11070756