INTERNATIONAL JOURNAL OF INNOVATION IN MECHANICAL CONSTRUCTION AND ENERGY
Vol. 2 No. 2 (2025), 67-73

IJIMCE
https://ijimce.ppj.unp.ac.id/index.php/ijimce
University of Negeri Padang

International Journal of Innovation in Mechanical Construction and Energy

ISSN: 3046-9112 e-ISSN: 3046-9104

SIMULATION OF WELDING STRENGTH TEST EQUIPMENT FRAME WITH A CAPACITY OF 20 TONS USING SOLIDWORK 2020 SOFTWARE

Torik Kuljanna [1], Ilham Alghani [2], ,Reyhan Stevano [3], Muchlisinalahuddin [4*]

[1]Departemen of Mechanical Engineering universitas Muhammadiyah Sumatera Barat, INDONESIA
JL. By Pass, Tarok Dipo, Kec. Guguk Panjang, Kota Bukittinggi, Sumatera Barat 26181
[2]Departemen of Mechanical Engineering universitas Muhammadiyah Sumatera Barat, INDONESIA
JL. By Pass, Tarok Dipo, Kec. Guguk Panjang, Kota Bukittinggi, Sumatera Barat 26181
[3]Departemen of Mechanical Engineering universitas Muhammadiyah Sumatera Barat, INDONESIA
JL. By Pass, Tarok Dipo, Kec. Guguk Panjang, Kota Bukittinggi, Sumatera Barat 26181
[4]Departemen of Mechanical Engineering universitas Muhammadiyah Sumatera Barat, INDONESIA
JL. By Pass, Tarok Dipo, Kec. Guguk Panjang, Kota Bukittinggi, Sumatera Barat 26181

*Corresponding Muchlisinalahuddin, Muchlisinalahuddin.umsumbar@gmail.com

Received 00 Month 2024; Accepted 01 Month 2024; Available online 02 Month 2024

Abstract

This study aims to analyze the strength of the 20-ton capacity welding test equipment frame using SolidWorks software-based simulation. The simulation evaluates the stress distribution, strain, and safety factor of the frame structure when receiving the maximum load. Analysis using the finite element method (FEA) is performed by modeling the frame along with boundary conditions and loads that match the actual test conditions. The simulation results show that the maximum stresses in the structure are still below the yield limit of the material, so the frame is considered safe to use. The highest stresses are distributed at the top of the frame especially around the load application area, while the bottom and legs of the frame experience lower stresses. The minimum factor of safety obtained in the simulation is 6.6 which indicates that the structure has high reliability against the applied load. Based on these results, the welding test equipment frame design can be said to be strong enough and safe to use for testing with a capacity of 20 tons. However, design optimization can still be done to improve the strength of the structure.

Keywords: Simulation, SolidWorks, Finite Element Analysis, Safety Factor, Stress

INTRODUCTION

In industry manufacturing and workshops, the welding process is often used to connect metal materials, but the welding results often experience distortion or imperfections, such as changes in shape due to uneven expansion and cooling. To overcome this, a press tool is needed that can return the shape of the material to the desired condition. One solution that can be used is a press tool based on a 20-ton hydraulic jack, which can provide sufficient pressure to correct deformations in the welded material [1].

The importance of bending testing to determine the mechanical properties of steel materials, especially in welding results to determine the strength and deformation that occurs. Many forms of technological development are aimed at the need for human work efficiency, so an effective technological development effort is needed. Along with the increasingly developing era, of course, it is very important to test the bending mechanism. One of them is a plate bending machine called a bending machine [2].

A bending test is one form of testing that is carried out to determine the quality of a material visually. In addition, the bending test was used to measure the strength of the material due to loading on the welded joint area[3].

RESEARCH METHODS

The scientific method used is FEA (finite element analysis). The method in the simulation of the welding strength test tool frame with a load of 20 tons aims to determine stress, displacement, and strain. And factor of safety on the frame of the welding strength tester. This process is carried out from a literature study by determining the source of the problems related to the field followed by designing the frame of the welding strength tester. The material used for the frame is ST37 carbon steel U profile. ST37 carbon steel has a carbon content of around 0.20% with additional manganese up to 1.40% and phosphorus and sulfur content limited to 0.045% with a tensile strength of between 370 to 510 Mpa and a minimum yield strength of 235 Mpa [4]. In this step, literature study starts, which is collecting information from various sources such as journals, books, and previous research. The purpose is to understand the basic concepts, methods that have been used, and get references related to the design of the welding strength tester frame. After understanding the relevant theory, the next step is to design the testing frame using SolidWorks design software. This software helps in making 3D models and simulating the structure to be tested. Next, determine the type of material used and the load to be applied to the structure. This information is very important to ensure that the design created matches the real conditions. After determining the loads and materials, a simulation programme is run to test the performance of the design. The simulation will give an insight into how the frame might behave under certain conditions. The programme will generate data on stress, displacement, and safety factor. This data is then analysed to determine if the design meets the expected criteria. For construction, if the simulation results satisfy the safety and durability criteria, the process can be continued to the construction phase.

If it is not eligible, then the design needs to be improved, and the process returns to the previous stage for revision and re-simulation. After the design has been approved, review of the analysis results is required. This includes evaluation of the structural performance based on the simulation results and validation against the literature used. The final step is to make conclusions from the entire process, including the effectiveness of the design as well as possible future improvements. The flow chart stage can be seen in **Figure 1**. For the design can be seen in **Figure 2**.

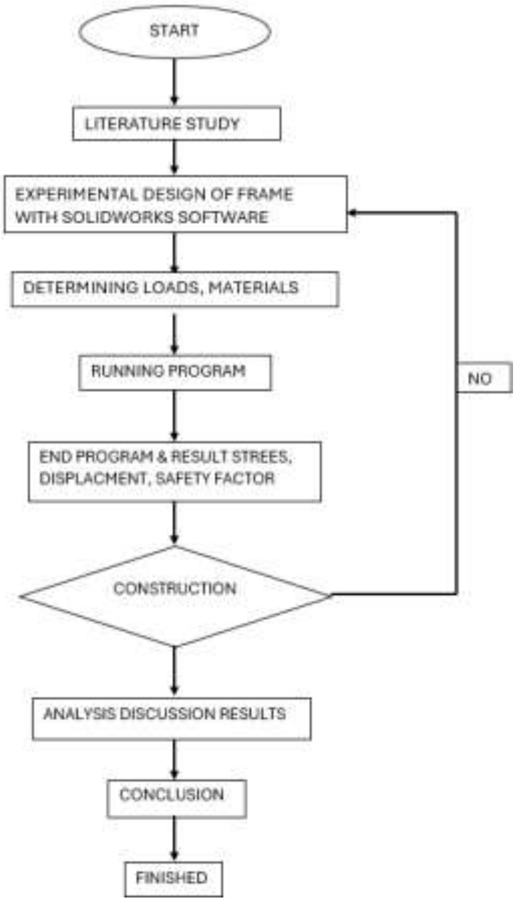

Figure. 1 Flow chart

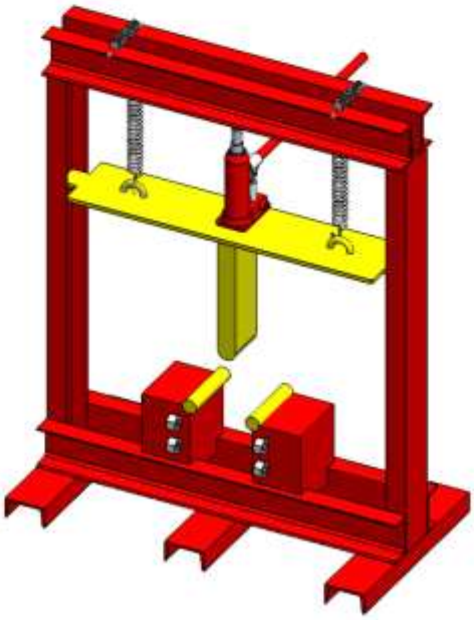

Figure 2. Welding strength tester frame

Table 1. Research tools used

Tools	Usage	Accuracy
Term Shove	To measure the frame diameter	0,02 mm
Welding Machine	To connect the parts to the frame	-
Laptop	To perform simulations on the frame	-

Steps for testing:

1. Make a 3D drawing of the engine frame Solidworks software.
2. Enter material data on solidwork in the form of ST 37 Carbon Steel
3. Do Fixtures.
4. Determine External load.
5. Doing Mesh.
6. Run the static simulation process.

RESULTS AND DISCUSSION.

In this chapter, we will analyze the welding strength tester frame by giving a load of 20 tons on the frame. This process uses the method of solidworks simulation to find out whether the frame is suitable for use or not.

1. Machine Design Specifications

Dimensional data in the design is as follows:

Spesifications	
Wideframe	1000 mm
Tallframe	1250 mm
U Steel profile	10×5×5 mm
Frame material	Carbon Steel ST 37 U Profile

2. StressAnalysis

The simulation results used involve analysis of static in solid work which is useful for evaluating responses. Structural frame made in solid work when given a load static. Analysis static is an important process in design to ensure that the designed frame or component functions safely and well under the applied load pressure[5].

3. Displacement

Displacement This is done to understand the movement or change in the shape of the frame under applied pressure. This analysis is important to be sure deformation remains within specified limits and does not cause functional problems in the design[6].

4. Safety Of Factor

Analysis safety factor This is used to see the feasibility of the frame against the design of the welding strength tester. In this analysis, it can be concluded that the frame design safety whether or not it is used by looking at the FOS value obtained in the analysis[7].

5. Strain

The analysis starts with a change in the shape of an object, consisting of changes in length and changes in angle. In this analysis, it can be seen at which part the heaviest point of the frame receives the load, marked with a red triangle[8].

Discussion

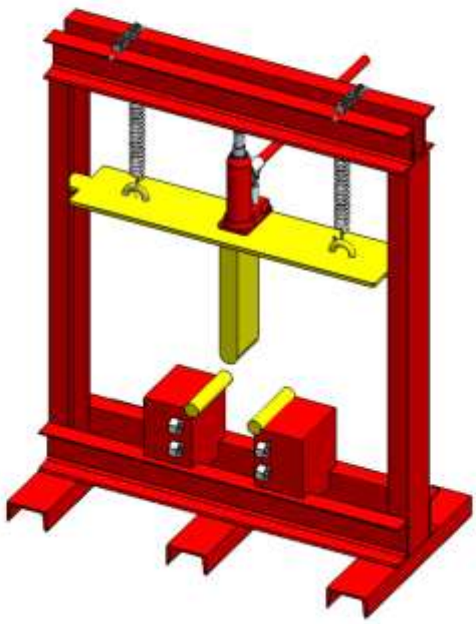

Figure 3. Frame Design

Based on the results of the analysis carried out on Solidworks software, four results were obtained. The following are the results of the analysis stress on the welding strength test tool frame using solidworks software2020. Analysis stress is the stress that occurs in the frame, in the analysis results it can be seen that the value yield strength on material worth 250 million Mpa with a load of 20,000 N. So it can be seen that the highest stress value is at 28,679,390,000 million N/m2The conclusion can be drawn that the frame of the welding strength test tool is still capable of withstanding a load of 20,000N at Level deformation the small one.

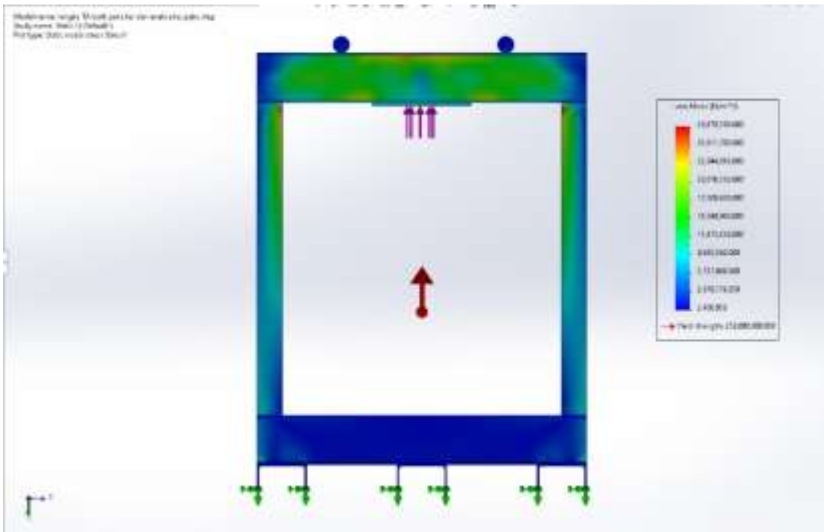

Figure 4. Stress Analysis carbon steel ST 37

Analysis results displacement(displacement) of a steel frame with welded joints using SolidWorks Simulation in study static. This analysis aims to evaluate the response of the structure to the given load. The colors in the image represent the level of displacement, where the blue color indicates the area with the smallest displacement or almost no movement marked with a value of 0.0 mm. In contrast, the green to red colors indicate areas with greater displacement, with the red color as the maximum displacement point of around 0.2 mm. In the picture, there is fixed support (marked with a green icon at the bottom) indicating that the bottom of the frame cannot move. The compressive load is applied at the top center (marked with a purple arrow pointing down), while the tensile force is applied at the bottom center (marked with a red arrow pointing up). The simulation results show that deformation The largest occurs at the top of the frame marked by a maximum displacement of 0.2 mm, while the bottom remains stable due to the given constraints. The transition color from blue to red indicates the load distribution along the structure. This analysis is important to understand whether the frame design is strong enough or needs modification to reduce excessive deformation. For deeper analysis, an evaluation of the voltage (stress) and safety factor (factor of safety, FOS) can be carried out to ensure the frame is safe against the applied load.

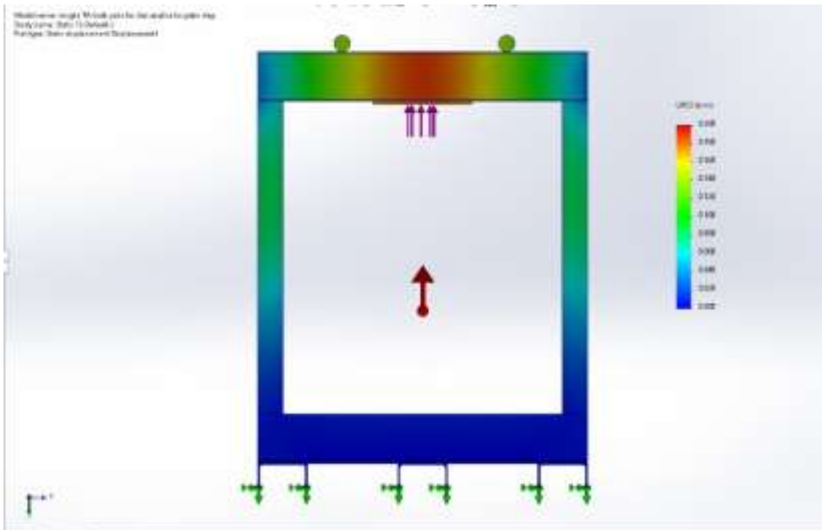

Figure 5. Displacement carbon steel ST 37

Analysis results strain (strain) of steel frames with welded joints using SolidWorks Simulation in study static. This analysis aims to evaluate how much strain occurs in the structure due to the given load. The colors in the image indicate the distribution of strain, where blue indicates the area with the smallest strain, while green to red indicates the area with the larger strain. There is fixed support at the bottom of the frame, meaning that the part is not moving, while the compressive load is applied to the top center and the tensile load is applied to the bottom center of the frame. The simulation results show that the largest strains occur at the top of the frame, especially around the area receiving the compressive load, while the bottom is relatively stable. High strains in an area can indicate a potential weak point in the design, so further evaluation of the stress (stress analysis) and safety factor (factor of safety, FOS) to ensure the frame remains safe and does not fail due to excessive deformation.

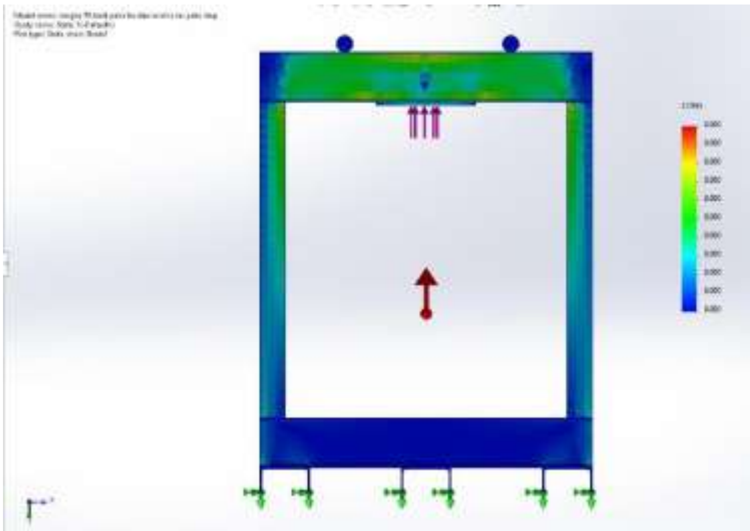

Figure 6. Strains Analysis of carbon steel ST 37

Analysis results in Factor of Safety (FOS) of steel frame with welded joints using SolidWorks Simulation in study static. FOS is a parameter that indicates the extent to which a structure is safe from failure due to a given load. From the figure, it can be seen that the minimum FOS value is 6.6, which means that the structure has a fairly high level of safety, because generally, the recommended FOS value for mechanical structures is above 1.5 to 3, depending on the application.

Almost all parts of the frame are red, indicating that the FOS is in the high range, so the structure is not in a critical condition or near failure. This indicates that the frame is still far from its material limits and is not experiencing excessive stress. However, if the design needs to be optimized, the dimensions or thickness of the material can be reduced to save material without significantly reducing safety. This analysis ensures that the frame can withstand the given load with a margin high enough security, so as not to require major changes in design unless there is a need for material efficiency.

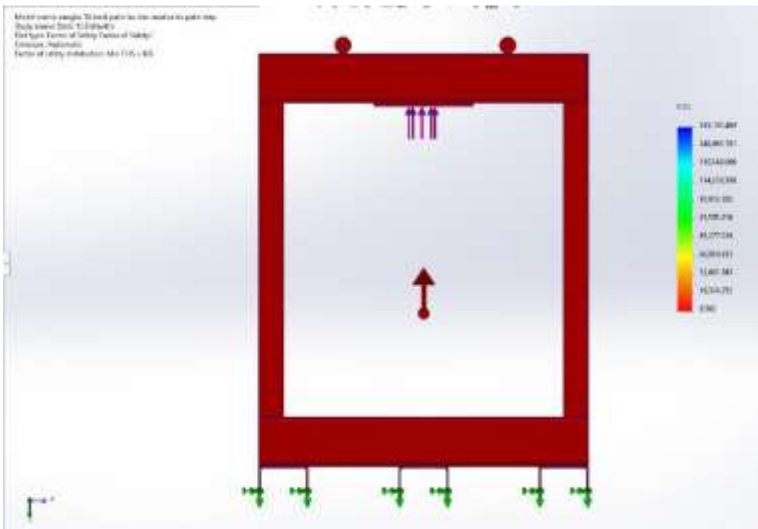

Figure 7.Factor Of Safety Analysis of carbon steel ST 37

CONCLUSIONS.

The results of the strength simulation of the welding strength test tool frame using steel material. Carbon ST37 uses SolidWorks software the following conclusions were obtained:

1. Stress analysis stress with a load of 20,000 N shows that the stress maximum that occurs in the structure is 28,679,390,000 MPa, far below the material yield limit of 250 million MPa.
2. In the analysis displacement, it was found that the movement that occurred in the upper part of the frame that was subjected to the load was worth 0.2 mm, while the frame that did not experience. Movement in the lower part was marked with a value of 0.0 mm.
3. Based on the simulation results using solid work, the distribution of strains on the test equipment frame shows that the upper part is experiencing concentration higher stretch compared to other parts.
4. The highest strains are at the green to red points that receive direct. loads, while the bottom and legs of the frame show smaller strains marked in blue on the frame.
5. The red color in the FOS Analysis indicates the area with the highest safety value, so that the design can be categorized as safe and does not experience the risk of structural failure under load test conditions of 20,000 N.

REFERENCES

[1] F. M. Fais and T. H. Ningsih, "Rancang Bangun Alat Uji Bending dengan Sistem Hidrolik," *J. Rekayasa Mesin*, vol. 7, no. 1, pp. 47–53, 2022.

[2] M. Syaukani, F. Paundra, F. Qalbina, I. Dwi Arirohman, P. Yunesti, and Sabar, "Desain dan Analisis Mesin Press Komposit Kapasitas 20 Ton," *J. Sci. Technol. Soc. Cult.* , vol. 1, no. 1, pp. 29–34, 2021.

[3] N. P. Putra, D. Y. Sari, and W. Afnison, "Analisis Statik pada Alat Press Hidrolik Menggunakan Solidwork," vol. 8, pp. 42747–42751, 2024.

[4] N. A. Sutisna, "Rancang Bangun Mesin Uji Universal Untuk Pengujian Tarik dan Tekuk Bertenaga Hidrolik," *J. Mech. Eng. Mechatronics*, vol. 6, no. 1, p. 32, 2021, doi: 10.33021/jmem.v6i1.1481.

[5] F. Restu, R. Hakim, and F. S. Anwar, "Analisa Kekuatan Material ASTM A36 pada Konstruksi Ragum terhadap Variasi Gaya Cekam dengan Menggunakan Software SolidWorks 2013," *J. Integr.*, vol. 9, no. 2, p. 113, 2017, doi: 10.30871/ji.v9i2.444.

[6] Y. E. Cumhur *et al.*, *J. Wind Eng. Ind. Aerodyn.*, vol. 26, no. 1, pp. 1–4, 2019, [Online]. Available: https://doi.org/10.1007/s11273-020-09706-3%0Ahttp://dx.doi.org/10.1016/j.jweia.2017.09.008%0Ahttps://doi.org/10.1016/j.energy.2020.117919%0Ahttps://doi.org/10.1016/j.coldregions.2020.103116%0Ahttp://dx.doi.org/10.1016/j.jweia.2010.12.004%0Ahttp://dx.doi.o

[7] R. Chavan and N. Patil, "Design , Development and Analysis of Press Tool for an Industrial Part," *Res. publik*, vol. 4, no. 1, pp. 112–123, 2016.

[8] L. T. Kusuma and H. Mahmudi, "Analisa Kekuatan Rangka Mesin Pengupas Kacang Tanah Menggunakan Software Solidworks," *Agustus*, vol. 7, pp. 384–392, 3AD.

[9] M. Balaji, N. Kumar, and T. Rao, "Structural Design and Analysis of 100 Ton Hydraulic press," *Int. Res. J. Eng. Technol.*, vol. 11, no. 4, pp. 614–617, 2024.

[10] A. A. N. Santoso, D. T. Santoso, and Aripin, "Analisis Kekuatan Rangka pada Perangkat Grading Fish menggunakan Software Solidworks," *J. Kaji. Tek. Mesin*, vol. 8, no. 1, pp. 111–120, 2023, [Online]. Available: http://journal.uta45jakarta.ac.id/index.php/jktm/article/view/6985